

LightBurn User Guide

Generated November 05, 2026 from https://docs.lightburnsoftware.com/2.2/

LightBurn Software Inc.

Contents

Home **8**

Safety **10**
- Fire Safety 12

Get Started **22**
- Identifying Your Laser 23
- Installing LightBurn 26
- Find My Laser 33
- Create Manually 40
- Add a Galvo Laser 48
- Connecting to Your Laser 54
- User Interface Tour 55
- Creating and Importing Artwork 63
- Moving Around the Workspace 68
- Selecting, Moving, and Sizing Artwork 70
- Colors and Layers 77
- Cut Settings 86
- Previewing 91
- Coordinates and Job Origin 95
- Framing 103
- Galvo Framing 112
- Job Control 117
- First Material Test 121
- First LightBurn Project 139
- Beginner Resources 168

Guides **170**
- Guides 171
  - How to Use These Docs 177
- Advanced Hardware Setup 182
  - Laser 2 Offset Setup 183
  - Adjust Red Dot Framing Offset 186
  - Scanning Offset Adjustment 189
  - WeCreate Camera Calibration & Alignment 198
- Firmware Configuration 201
  - GRBL Configuration 202
  - Configuring a Ruida Controller 208
- Galvo Laser Guides 215
  - Add a Galvo Laser 48
  - Changing a Galvo Laser Lens 216
  - Installing Galvo Drivers 218
  - Focusing Galvo Lasers 230
- LightBurn Configuration 233
  - Customizing the LightBurn Window 234

Updating LightBurn 241
Downgrading to Older Versions 243
Move LightBurn to a New Computer 245
Opening Multiple LightBurn Instances 249
Setting up the CorelDRAW Macro 254
Automation with UDP 258
Shared Settings for Multiple Users 261
Network and Wireless Connections 265
Connect to a Ruida or Trocen Laser with Ethernet 266
GRBL Network Connection Setup 270
LightBurn Bridge 277
Projects and Techniques 292
5 Steps to Perfect Image Engraving 293
Kerf - What it is and How to Compensate 298
Engraving Full Wrap Tumblers 305
3D Sliced Engravings 345

License & FAQs 353
License FAQ 356
License Management Guide 359
Educational and Volume Licensing 366
System Locked and Floating License Setup 367
Enterprise Security FAQ 370
LightBurn End User License Agreement (EULA) 373
LightBurn Accessibility Statement 388

Resources 389
Downloadable Art Libraries 393
External Resources 406

Troubleshooting 412
Device Specific Troubleshooting 417
Job Quality 421
Wrong Orientation 440
GRBL: Low Power or No Output 442
Wrong Start Location 444
Warped or Disconnected Lines 445
Laser Fires During Travel Moves 446
Incorrect Size 447
Fuzzy, Blurry, or Overblown Edges 448
Dark & Burned Edges 449
Wobbly Engraving 450
Poor Image Quality 451
Inconsistent Engraving Tone or Color 452
Periodic (Repeated) defects in Engravings 453
Jagged Lines 454
Corners are Too Dark, Too Light, or Missing 455
Laser Lowing Power at Low Speeds or Not Reaching Full Power 457
Slanted or Skewed Results 459

Engraving Shifted Relative to Cut Lines ... 461
Layers Shift When Cutting Multiple Objects ... 462
Wiggly Lines at Corners ... 463
Preview Window ... 464
Added Lines ... 476
Ballooned or Incorrectly Sized Shapes ... 478
Blurry Edges ... 480
Double-Engraved Areas ... 482
Engraved Areas Opposite of Expectation ... 487
Excessive Overscanning ... 490
Gaps in Lines ... 492
Gray Shapes ... 495
Inaccurate Time Estimates ... 497
Missing Shapes ... 498
Solid Black Images ... 501
Strange Banding / Scan Lines Not Visible ... 505
Connection Troubleshooting ... 507
LightBurn Editor Help ... 511
GRBL Errors ... 515
GRBL Communication ... 519
License Troubleshooting ... 521
Camera Troubleshooting ... 528
CNC-Based Lasers ... 537
Drivers ... 541
Electrical Problems ... 542
Galvo Laser Troubleshooting ... 543
How to Ask For Help ... 545
Vector Artwork Imports With Extra Lines ... 555
Machine Size Mismatch ... 557
macOS-Specific Problems ... 559
Windows-Specific Problems ... 560
Mechanical Issues ... 562
Reset to Default Settings ... 563
Restoring Lost or Changed Settings ... 564
Serial Port Problems ... 567
USB Cables ... 569

Tools & Features ... 571
UI ... 572
Arrange Menu ... 577
Arrange Toolbars ... 582
Color Palette ... 585
Creation Toolbar ... 590
Edit Menu ... 593
File Menu ... 599
Help Menu ... 604
Language Menu ... 608
Laser Tools Menu ... 609

Main Toolbar ... 613
Menu Toolbar ... 616
Modes Toolbar ... 618
Modifiers Toolbar ... 620
Status Bar ... 623
Tools Menu ... 627
Tooltips and Topic-Aware Help ... 633
Web Tools Menu ... 635
Window Menu ... 640
Workspace / Edit Window ... 645
Essential Functions ... 648
File Management ... 649
Selection Tools ... 658
Zooming and Panning ... 666
Undo & Redo ... 670
Clipboard Tools ... 672
Transform Controls ... 675
Grouping & Ungrouping ... 683
Preview ... 691
New Window ... 696
View Style ... 697
Show Notes ... 699
Print ... 700
Layout and Design ... 701
Creation Tools ... 702
Editing ... 775
Modifying and Combining ... 816
Arrangement ... 862
Image Tools ... 913
Laser Control ... 944
Cut Settings ... 945
Output and Positioning ... 1043
Quality Optimization ... 1092
Modes and Advanced Control ... 1111
Machine Management ... 1203
Cameras ... 1323
Cameras / Overlays Window ... 1326
Camera Installation & Focusing ... 1333
Camera Selection ... 1336
Add a Camera ... 1342
Edit Camera ... 1351
Camera Lens Calibration ... 1355
Camera Alignment ... 1366
Head Mounted Camera ... 1379
Lidar Cameras ... 1396
Save Background Capture ... 1399
Settings and Preferences ... 1401
Beginner Mode ... 1402
Settings / Preferences ... 1408

Managing Preferences ... 1430
User Bundles ... 1432
Edit Hotkeys ... 1435
Reset to Default Layout ... 1437
Help and Software Features ... 1438
License Management ... 1439
Check for Updates ... 1442
Help and Notes ... 1444
REST API ... 1445
View Usage Data ... 1459
Enable Debug Log ... 1462
Debug Drawing ... 1465

Explainers ... 1466
Laser Types ... 1469
Layer Modes ... 1474
Speed vs. Power ... 1481
Images vs. Vectors ... 1484
Open vs. Closed Shapes ... 1489
How to Invert a Vector Engraving ... 1497
Template Files ... 1500
Ovescanning ... 1502
Steps per MM ... 1511
About Galvos ... 1516
Galvo Glossary ... 1520
LightBurn Cut Settings and EZCAD Hatches ... 1523

Collections ... 1525
Tips and Tricks ... 1531
Job Optimization ... 1543
Material Utilization ... 1549
Batch Production ... 1557
Design Tools ... 1561
Getting a Design Laser-Ready ... 1568
Working With Images ... 1576
LightBurn for Galvos ... 1582

Learn ... 1589
Learn Volume 1 ... 1590
Maker Spotlight 1 ... 1596
Staff Spotlight 1 ... 1610
Classroom Safety ... 1630
Projects ... 1635
Graham Cracker Houses ... 1641
Image Engraving on Coasters ... 1684
Laser Cut Dinosaur ... 1698
Laser Cut Fridge Magnets ... 1705
Laser Engraved Pencils ... 1722

Laser Etched Clock 1740
Purrfect Present: Pet Puzzle Postcards 1751

EDU 1760

What's New 1765

Home

Get Started

Our getting started guides will help you get started using LightBurn with your laser.

Get Started

What's New?

See what's new and updated in LightBurn 2.2.

New in 2.2

License & FAQs

Information about how the LightBurn license works, and other frequently asked questions.

Licensing & FAQs

Resources

Downloadable art libraries and other resources.

Resources

Troubleshooting

Get help when something isn't working right.

Troubleshooting

Education

Educational resources for teachers and schools.

Education

Reference

Get information about all of LightBurn's tools and features.

Reference

For automated translations with Google Translate, use the language menu to pick from one of our most commonly used languages. If you want to use a different language, you can choose your preferred language from the dropdown.

Popular Topics

See some of our most commonly used pages.

Camera Rotary Kerf Material Test Snapping Text Preview

Safety and Warnings

Safety and Warnings



WARNING

LASERS USE INTENSE BEAMS OF LIGHT, INCLUDING LIGHT THAT MAY NOT BE VISIBLE.

IMPROPER USE OF LIGHTBURN SOFTWARE WITH ANY DEVICE MAY BE DANGEROUS AND COULD RESULT IN PERSONAL INJURY, DEATH, OR PROPERTY DAMAGE. ALWAYS FOLLOW THE DEVICE MANUFACTURER'S INSTRUCTIONS AND GUIDELINES.

Safety and Warnings

ALWAYS WEAR PROPER EYE PROTECTION WHEN USING YOUR LASER.



WARNING

DO NOT LEAVE A LASER UNATTENDED WHILE IN OPERATION.

Fire Safety

Here at LightBurn, we've always asked our users to supervise their lasers whenever they're running, and discouraged the remote use of lasers. We've also reminded users to make proper preparations in case a fire does occur.

It can be tempting to leave your laser running and walk away during a long job, especially when you're busy and have other things to do. You've made sure your settings are good, you've never had a fire before, you've got a camera set up so you can watch your laser...



You still need to stay with your laser and supervise it.

Jump to Fire Safety Preparations

These two pictures are from two different incidents. In the first photo, close supervision of the laser and the quick reaction of the LightBurn staff member who was running it prevented a fire and further damage. In the second, proper preparations as well as quick reactions once the fire was noticed prevented a fire from becoming a tragedy.



A covering inside this laser got in the way of the laser beam, but the damage was spotted immediately and the laser was stopped before anything worse could happen.



A minor flare-up happened while the laser was unsupervised. Within a few minutes, this was the result.

Whenever you laser, you're deliberately burning away material, and that also creates smoke full of flammable particulates. Fires can happen unexpectedly and grow shockingly quickly. Right when they start, they can usually be put out without serious damage, but only if you're prepared and can act right away. Stopping the laser from firing and either moving the head away or turning off the air assist is a good start, and may even be enough on its own. By stopping the laser and the air assist, you're not putting more energy into a growing fire or blowing oxygen-rich air right into it. If that's not enough to stop the fire and it's still small, a spray bottle of water may be enough.

If smaller interventions aren't working, or if you didn't respond quite as quickly, you're going to need to take more serious action.

In case a fire happens, you should always have the following easily accessible near your laser, placed so you can reach them without reaching over or around the machine:

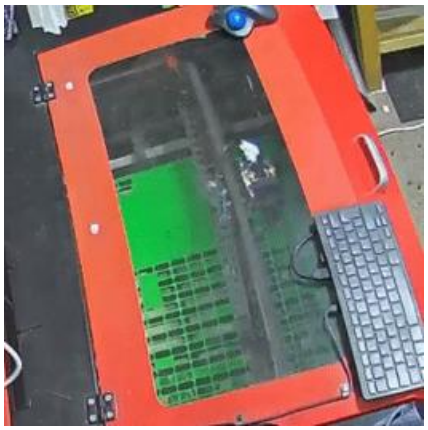
- A spray bottle of water. For small flare-ups on the material you're cutting, this may be adequate.
- A suitable fire extinguisher. We use CO_2 fire extinguishers, which don't damage the laser, reducing reluctance to use them. Read more here.
- A fire blanket.

Check the manufacturer's information on how to use the fire extinguisher and fire blanket before you need them, and don't hesitate to use any of these if there's even a chance the situation calls for it.

The Laser Fire

The burnt laser above belongs to one of our staff here at LightBurn. He had a camera set up to watch the laser while it ran a job. He's typically very cautious, but relaxed a bit this time and let it run in the background. It only took a few minutes to go from the first hints of sustained flame to what you'll see in these pictures he's shared.

This GIF is sped up, but shows the material catching fire, which then spreads through the machine and into the crumb tray below the bed. In reality, this was about 4 minutes. At the start, when there were just a few small flames, this would have been easy to put out.



This laser had a metal frame, which helped contain the fire and protect the surroundings. For an open-frame or primarily plastic laser, the fire likely would have gotten out of control faster, with more damage. By the time our staff member got to the laser to put it out, the workshop was full of smoke. Fortunately, he was well-prepared with an easily accessible fire extinguisher, and was able to put out the fire before it spread further. Even so, the heat from the fire destroyed the chiller on the shelf above the laser in addition to the laser itself.



After the fire was extinguished.



An emergency stop button like this is an important safety feature for being able to cut power to the laser immediately in an emergency situation.



The panel holding the machine's Ruida control panel melted and dropped down into the body of the laser cutter.



More damage inside the laser.

Fire Safety Preparations

1. Never run your laser unsupervised. A camera feed does not count as supervision.
2. Have a smoke detector near the laser cutter.
3. Have a suitable fire extinguisher, fire blanket, and spray bottle of water near the laser. Make sure you'll be able to reach them if there is a fire. Ensure that you and everybody who might use your laser knows how to use these tools to put out a fire.
4. Know how to shut off your laser quickly in an emergency.
5. Make sure that anybody who might operate or be asked to supervise your laser knows what to do in the event of a fire.
6. Avoid having easily flammable objects near the laser.
7. Keep the interior of your laser clean. Small bits of material in the crumb tray under the laser bed can act like kindling and worsen a fire.
8. Be aware of the materials you're using. Materials with air pockets (like corrugated cardboard or some plywood) are more prone to catching fire.

9. Refer to your manufacturer's recommendations for proper machine operation, including how to reduce the chances for accidents and prevent fires.
10. Do not rely on hardware-based fire detection or fire suppression accessories — they're not an adequate substitute for your attention and preparedness.

What to Do If A Fire Happens

1. Stop firing the laser and turn off the air assist. Most devices will have a way to pause or stop the job right from the machine. If you catch the fire fast enough that it's still just a small flare up, this may be enough to extinguish the fire.
2. If the fire isn't going out but is still small, a spray bottle of water may be enough to put it out. Keep this near your laser just in case.
3. If the fire still isn't going out, or if you didn't catch it early enough and it's more than a small flame, it's time for a fire blanket and/or fire extinguisher. Make sure you know how to use these before a fire happens. There won't be enough time to figure it out when you need it. After the fire is extinguished, make sure that if any material is still smoldering, you fully extinguish it with water.
4. If you've still got a fire, call your fire department.
5. After the fire is out, ventilate the room to bring in fresh air and avoid breathing in smoke for longer than you need to. This is especially important if you've used a CO_2 extinguisher, which displaces oxygen to put out the fire.

Fire Extinguishers

Please make sure you check your local standards for fire extinguishers.

Read more about fire extinguishers from:

- U.S. Fire Administration
- National Fire Protection Association

Fire extinguishers are rated according to capacity and the types of fires they can be used with, and many are multi-purpose, suitable for multiple types of fires. For a laser cutter, make sure your fire extinguisher includes a **Class C** rating, meaning it uses a non-conductive extinguishing agent and can be used on appliances that are plugged into an outlet.

There are several types of fire extinguishers, but the most relevant for a laser cutter are **CO_2** and **Dry Chemical** extinguishers.

CO_2 extinguishers smother a fire by replacing oxygen with CO_2 gas.

- They are generally preferred for use with laser cutters and other electronics, because they don't damage the laser.
- They pose a risk of asphyxiation in small, poorly ventilated rooms. It's important to air out the room well after use to bring in breathable air.

- They are somewhat less effective than **Dry Chemical** extinguishers on combustible materials, but because they don't damage equipment, users are likely to use them earlier, while a fire is still small.

Dry Chemical extinguishers are multipurpose extinguishers, which use monoammonium phosphate powder to extinguish a fire.

- They are suitable for electrical fires and also highly effective on combustible materials.
- They do not pose a risk of asphyxiation, but contacting or breathing in the powder can cause throat and skin irritation.
- The powder used is corrosive, and will destroy your laser.

Getting Started

Welcome! If you're here, it probably means you're new to LightBurn or the wider world of laser cutting and engraving. If you're not sure where to begin, you're not alone — there's a lot to take in.

We've created this guide to get you going with the most important concepts you need to know in order to use LightBurn and your laser.

We'll cover:

- Identifying your type of laser
- Installing LightBurn
- Creating a device profile
- Connecting to your laser
- Getting to know the LightBurn interface
- Creating and importing artwork
- Assigning colors/layers to artwork, and different layer modes
- Establishing cut or engrave settings
- Previewing a job
- Determining where your project will output in your laser's work area
- Commanding your laser to Frame the area your project will output
- Telling your laser to start a job

We'll finish up with a step-by-step first project that puts the concepts we covered into action.

> **Info**
>
> Laser cutting and engraving performance, quality, and speed is very machine and user dependent. Beginners should familiarize themselves with the user guide provided by their laser's manufacturer prior to using it.

At the bottom of each page you'll see a button to direct you to the next step in the process, or return you to the previous page if you need a refresher. You can also jump ahead, or back, using the navigation to the left, but we don't recommend skipping steps unless you're really sure you know what you're doing.

Ready? Click below to get started!

Identifying Your Laser

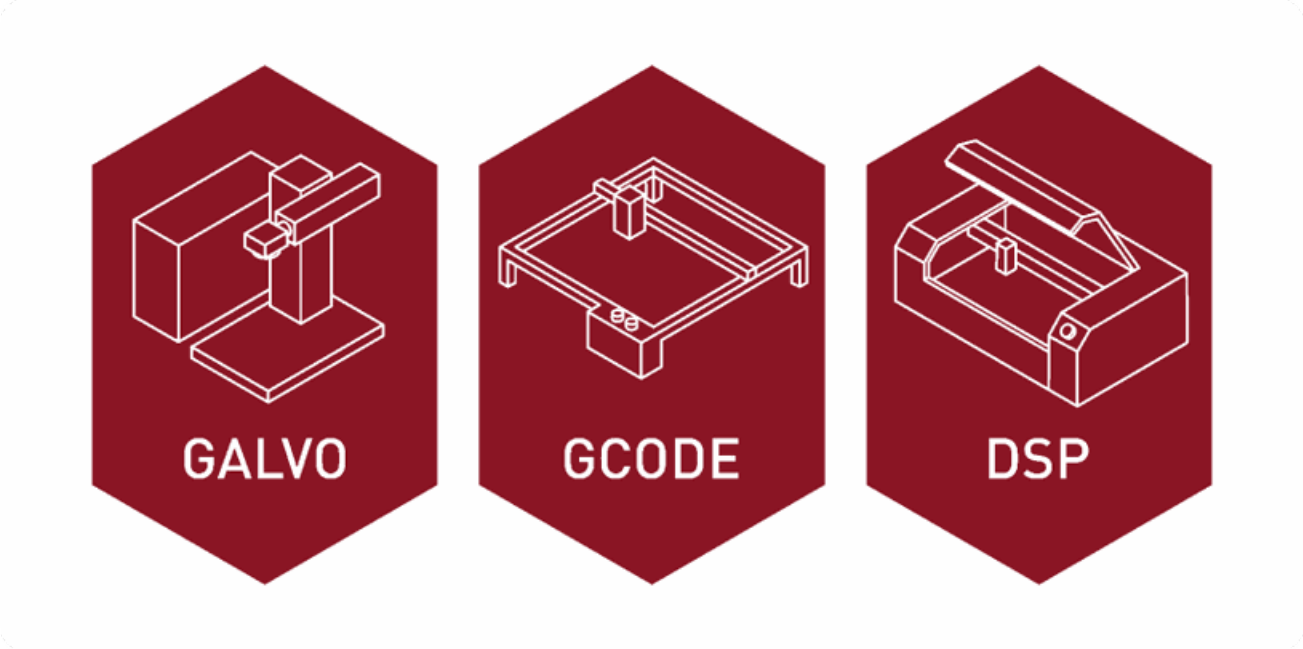
You'll need to know what type of laser you're using in order to know which LightBurn license you need, and to follow along with some of the terms used throughout our documentation.

If you're looking for a new laser, see our Hardware Compatibility matrix for information on finding LightBurn-compatible hardware.

LightBurn Compatible Lasers

Your laser type also affects what your laser is capable of, what the LightBurn UI looks like, and how you interact with your laser.

While there are many attributes that distinguish the different types of lasers, you'll see us refer most often to three general categories. Each category is depicted by an example of a typical model below:



To get started with LightBurn, you'll need to know two things about your laser:

- What **controller** does your laser use?
- What **motion system** does your laser use?

Controllers and Firmware

The controller — and the firmware it runs — determines whether or not a machine is compatible with LightBurn, and what type of license you need.

A controller is essentially a computer within your laser, and the firmware it runs determines the type of instructions it can translate into signals it delivers to the motion system.

LightBurn needs to know what type of firmware your laser's controller is running to know what "language" to speak to the machine. In most cases, LightBurn can automatically identify the firmware if you set up your machine using Find My Laser, but you'll need to be able to identify the controller yourself in order to complete a manual setup.

LightBurn is currently compatible with three main categories of controllers: GCode, DSP, and Galvo.

- To connect to a GCode-based laser, any LightBurn license will work. If you only have a GCode-based laser, you'll probably want a LightBurn Core license.
- To connect to a DSP or Galvo laser, you'll need a LightBurn Pro license. LightBurn Pro licenses also support GCode-based lasers.

Below is a brief description of each type, as well as a list of common controllers/firmware used by each. If you're not sure what controller or firmware your laser uses, consult your machine's manufacturer, or contact us at support@lightburnsoftware.com.

GCode

Most entry-level diode lasers use GCode-based controllers.

Supported controllers/firmware: GRBL, Smoothieware, Marlin, FluidNC, grblHAL, xTool

DSP

DSP controllers are common in more industrial-grade machines. If your machine is a CO_2 laser in a metal case and has an LCD display, it's probably a DSP model.

Supported controllers: Ruida, Trocen, TopWisdom

Galvo

Galvo lasers use a fixed scanning head mounted to an arm, and project the beam from above. If your laser uses EZCAD2, EZCAD3, or SeaCAD as its default software, it is this type of laser.

Supported controllers: EZCAD2, EZCAD2 Lite, EZCAD3, BSL

> **Does LightBurn support EZCAD3 lasers?**
>
> LightBurn 2.1 adds support for EZCAD3 lasers.

Motion System

The motion system consists of the mechanical components that direct the beam from the laser source to the material. These include parts such as motors, mirrors, lenses, and axes.

The choice of motion system determines which LightBurn tools are available for your machine, as well as the maximum job size, and the speed of the laser.

> **Note**
>
> Many tools in LightBurn are only available for machines with a particular type of motion system.

There are two common motion systems:

- **Gantry motion systems** have frames, wheels, and motors that move a laser head around a work area. Some Gantry lasers use a system of mirrors to reflect a beam from its source, while others hold the laser source and move it around directly. Gantry systems are relatively slow due to the mass of the components moving around, but can have very large working areas and fire the beam straight down at the material.
- **Galvo motion systems** use tiny moving mirrors to bounce the beam to different locations on a large lense, which focuses the beam and points it back down at the work. Because the mirrors are so lightweight, they can bounce the light around at very high speeds, but Galvos are limited by the size of the lense to relatively small jobs.

> **Note**
>
> All Gantry lasers supported by LightBurn have GCode-based or DSP controllers.
>
> Almost all Galvo lasers supported by LightBurn use EZCAD2, EZCAD3, or BSL controllers and require a LightBurn Pro license. However, there are a limited number of GCode-based Galvo lasers that are compatible with LightBurn Core, and require special setup — consult your laser's manufacturer for more information.

Additional Information

Knowing your laser's controller/firmware and motion system is enough to get you started, but there's a lot more to know about laser cutting and engraving. For additional information on the ins-and-outs of different types of lasers and accessories, see Laser Types.

Installing LightBurn

The first step to using LightBurn is downloading it and getting it installed on your computer.

See below for information on system requirements, where to download LightBurn, and how to install LightBurn on your preferred operating system. If you've already downloaded and installed LightBurn, skip ahead to Adding a Laser.

System Requirements

If you want to know whether your computer will run LightBurn effectively, or are considering a computer to buy, the chart below shows recommendations as well as minimum hardware requirements for running LightBurn 2.2.

	Recommended Setup (Heavy Designing & Multitasking)	**Practical Minimum (Smooth Workflow)**	**Absolute Minimum (Will Run)**
Operating System	Windows 10 or 11, macOS 13 or higher. 64-bit version of Windows is required. For Windows N edition, you'll need to install the Media Feature Pack		
Processor (CPU)	Intel Core i5 / AMD Ryzen 5 or newer, Apple M-Series	Intel Core i3 / AMD Ryzen 3	64-bit Dual-Core (Intel Celeron or similar)
System Memory (RAM)	16 GB Smooth multi-tasking with CAD, image editors, or other software	8 GB Works for typical usage	4 GB Minimum required for operating system, struggles with large files
Storage Type	≥256 GB NVMe Solid State Drive (SSD) Fastest option for opening/importing/saving files	Solid State Drive (SSD)	Traditional mechanical hard drive LightBurn will run, but opening/importing/saving may be slow
Screen Resolution	1920 x 1080 (1080P/FHD) or higher Larger workspace, with room for expanded toolbars and larger text/icons if desired	1920 x 1080 (1080P/FHD)	1280 x 1024 Works, but limited workspace and condensed/smaller UI can be frustrating
Free Storage Space	At least 450 mb free to install the software, plus additional space for files		

These system requirements are for LightBurn 2.2. If your computer doesn't meet these requirements, we have information on compatibility for older versions.

You will always be able to use the most recent version of LightBurn that is compatible with your operating system and was released within your license's valid update period. You can download earlier versions of LightBurn from our release repository, or see Operating System Compatibility for compatible versions and download links.

Support for other operating systems

- LightBurn version 1.7 is the last version compatible with Linux, versions of Windows prior to 10, and versions of macOS prior to 11. Download LightBurn 1.7.08 and see our 1.7 documentation for information and installation instructions.
- Users with macOS 12 should download LightBurn 2.1.04.
- Users with macOS 11 should download LightBurn 2.0.05.
- macOS users running versions older than 10.13 should download LightBurn 1.6.03, which supports macOS 10.11 or newer. See our legacy documentation for more information and installation instructions.
- Users limited to 32-bit versions of Windows should download LightBurn 1.5.06, which is the final 32-bit release of LightBurn. See our legacy documentation for more information and installation instructions.

ARM Device Support

Apple Silicon

LightBurn 2.2 is compatible with computers using Apple Silicon processors (2020's M1 chip and anything newer), and runs natively on macOS without requiring Rosetta 2.

Upgrading to LightBurn 2.2 will eliminate the compatibility warning in macOS 26.4 and newer, which warned that LightBurn was an Intel App and would not be compatible with macOS 28, expected in Fall of 2027.

If you're using LightBurn on an Apple Silicon-based Mac and plan to update to macOS 28, make sure you update LightBurn to 2.2 before installing macOS 28.

More Information

Windows on Arm

Windows devices running on Arm processors are compatible with LightBurn, and starting with LightBurn 2.2, no longer need a special driver installation process. The drivers automatically installed by LightBurn are now compatible with Arm processors. If you've had issues with the drivers before, running the LightBurn 2.2 installer and selecting the driver to install should fix your problem.

LightBurn Operating System Compatibility

Click the link for your operating system in the colored bar to go directly to the download page for the most recent compatible LightBurn version.[1]

Operating System	Final Supported Release	Notes
Windows 11	Currently Supported (2.2)	
Windows 10	Currently Supported (2.2)	
Windows 8	Supported up to 1.7	Legacy 2.0 release
Windows 7	Supported up to 1.7	Legacy 2.0 release
Windows 7 (32-Bit)	Supported up to 1.5	
macOS 27 - Golden Gate	Currently Supported (2.2)	Compatible with macOS 28
macOS 26 - Tahoe	Currently Supported (2.2)	Compatibility info for macOS 28
macOS 15 - Sequoia	Currently Supported (2.2)	
macOS 14 - Sonoma	Currently Supported (2.2)	
macOS 13 - Ventura	Currently Supported (2.2)	
macOS 12 - Monterey	Supported up to 2.1	
macOS 11 - Big Sur	Supported up to 2.0	
macOS 10.15 - Catalina	Supported up to 1.7	Legacy 2.0 release
macOS 10.14 - Mojave	Supported up to 1.7	Legacy 2.0 release
macOS 10.13 - High Sierra	Supported up to 1.7	Legacy 2.0 release
macOS 10.12 - Sierra	Supported up to 1.5	
macOS 10.11 - El Capitan	Supported up to 1.5	
macOS 10.10 - Yosemite	Supported up to 0.8	
Ubuntu 24.04	Supported up to 1.7	
Ubuntu 22.04	Supported up to 1.7	
Ubuntu 20.04	Supported up to 1.7	
Ubuntu 18.04	Supported up to 1.7	

Download LightBurn

To ensure you get the most recent version of LightBurn, download LightBurn directly from the Download / Trial page at LightBurnSoftware.com. On this page, you'll find links to download the current version of LightBurn, as well as links to previous versions of the software and public beta versions. You can also find links to the appropriate version for older operating systems in our compatibility table.

When you've completed the download, you should see the file in your "Downloads" folder or a manually selected folder.

Installation Directions

Choose your operating system below for additional installation help.

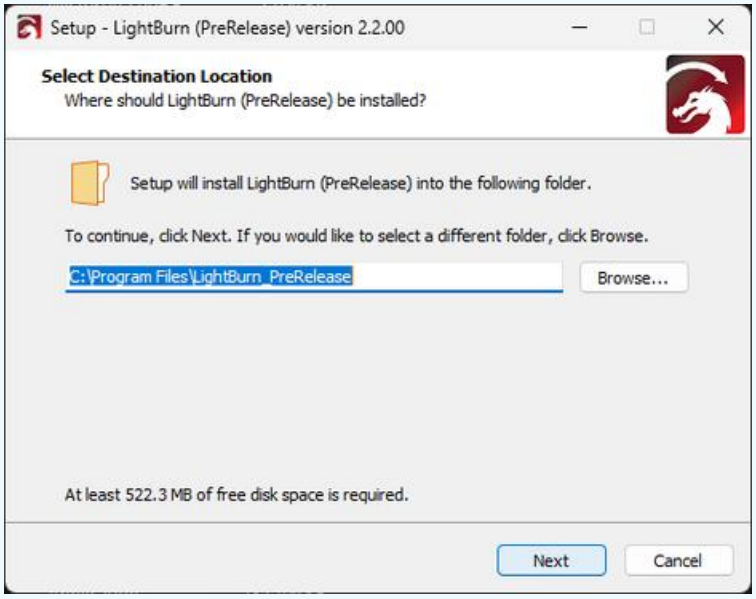
Windows Installation

Windows Installation

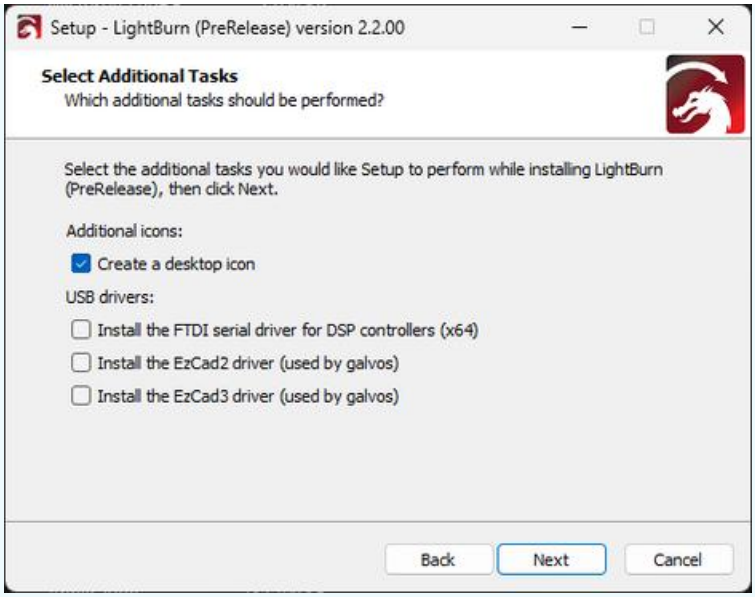
> **Info**
>
> Some of these screenshots show the installer for the pre-release version of LightBurn. The full release version will default to installing in the `LightBurn` directory/folder.

Installation works the same way as most Windows software. Start the installer by double-clicking the file you downloaded from the LightBurn website. Windows may ask if you trust us first.

The installer will ask where you want to install the software. Most users should use the default location, but if you want to have multiple versions installed, you can make version-specific install directories here.



Click **Next**, then click **Install** to begin the installation process. When it completes, you'll see the following screen with options for installing drivers, viewing the change log, and launching LightBurn.



Installing Drivers

If you have never installed LightBurn on this computer before, you might need to install drivers. This is not necessary when updating an existing installation.

- Check **Install FTDI Driver** to install the driver used by DSP controllers, like Ruida and TopWisdom.
- The EZCAD2 and EZCAD3 drivers are used by supported Galvo devices using EZCAD2 and EZCAD3 boards. Read more on our Galvo Driver Installation page.

Installing the driver for BSL Galvo Lasers

If you have a BSL Galvo laser, you may need to separately install a driver supplied by your machine's manufacturer, or install the stock software that came with your machine to get the driver.

Start LightBurn however you usually do to begin using the software.

macOS Installation

macOS Installation

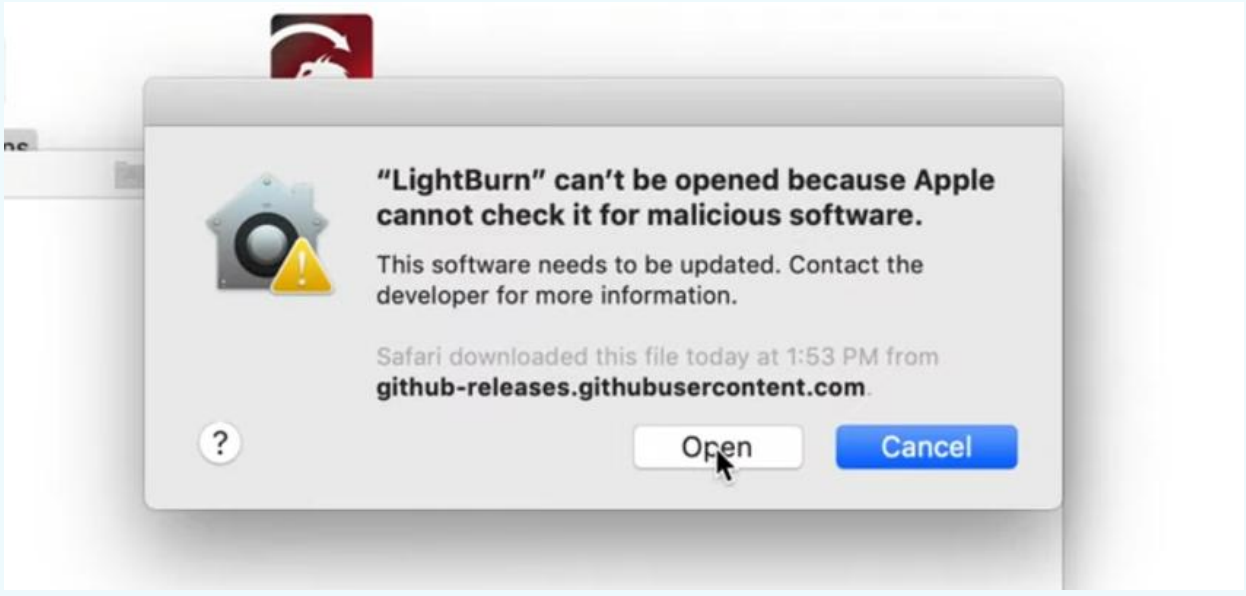
Installing on M1 or newer Macs

Versions of LightBurn older than 2.2 required Rosetta. This is no longer required for LightBurn 2.2.

Installation works the same way as most macOS software. After downloading the DMG file, open the file and drag LightBurn into your Applications folder.



Notarization bypass for LightBurn 1.1 and earlier



Versions before 1.2 were not notarized to be recognized by your computer as coming from a trusted developer. If you installed an older version, you'll need to follow the steps below the first time you run LightBurn:

1. Open a Finder window.
2. Browse to the **Applications** folder.
3. Right-click or two-finger tap the LightBurn icon.
4. Choose **Open** from the menu.
5. When macOS asks if it should open the program, say yes to add an exception allowing your computer to run LightBurn normally. Once this is done, you will not need to repeat the process unless you reinstall LightBurn.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=nH3rSbmn-H4

Video walkthrough

---

1. **2.0.05 Legacy Releases:** Official LightBurn support for macOS 10.13 - 10.15 and 64-bit versions of Windows 7 and 8 ended with LightBurn 1.7.08, but we do have a legacy release of 2.0.05 available if you want to upgrade to 2.0 but are limited to one of these older operating systems. Our support team is available to help, but if your computer is unable to run these versions reliably, you may need to either use LightBurn 1.7.08 or upgrade your computer/operating system.

   LightBurn 1.7.08 for macOS 10.13 - 10.15

   LightBurn 1.7.08 for Window 7 and 8

Find My Laser

> **Consult Manufacturer Instructions**
>
> Always consult the documentation that came with your laser, or the manufacturer's online resources, before beginning the device setup process. Some manufacturers provide settings bundles, importable device profiles, or special setup instructions that are necessary to use their machines effectively with LightBurn.

The first time you open LightBurn, you'll be greeted by the Devices window, where you'll see an empty list, along with several options for creating a device profile at the bottom. You need to have at least one device profile created in order to use LightBurn, so if you close out of this window without creating one, you'll see a warning message with the option to go back and set up a device, have LightBurn automatically create a dummy profile for you, or close out of the software.

If your laser's manufacturer hasn't provided any special instructions or importable files, the quickest way to add a new laser to your list of devices is with the **Find My Laser** wizard. Follow the instructions below to use the wizard to set up your device.

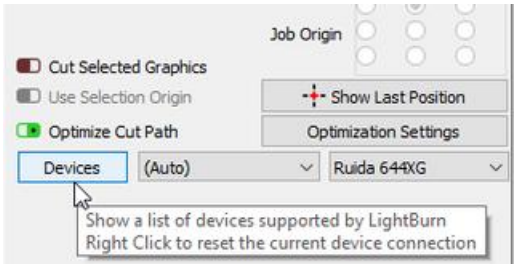
Starting in LightBurn 2.2, Find My Laser can detect some network devices.

> **Info**
>
> If you don't have a laser yet, go to Create Manually to create a dummy profile as a placeholder until you get a laser.

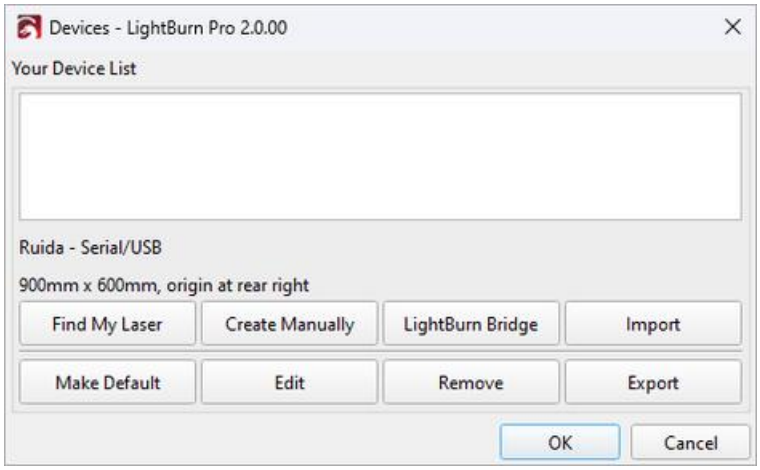
Automatically Add a Laser to LightBurn

1. Make sure that your laser is powered on and connected by USB cable to a computer running LightBurn. Wait for the laser to complete any power-on moves, like homing, before continuing. Some devices, such as certain xTool machines, the Creality Falcon A1C, and supported Epilog machines can be detected over a network connection - for these machines, connect with USB or connect the laser to the network as needed.
2. Click on the **Devices** button in the Laser Window, found in the lower right-hand corner of the program window by default, to open the Devices window.

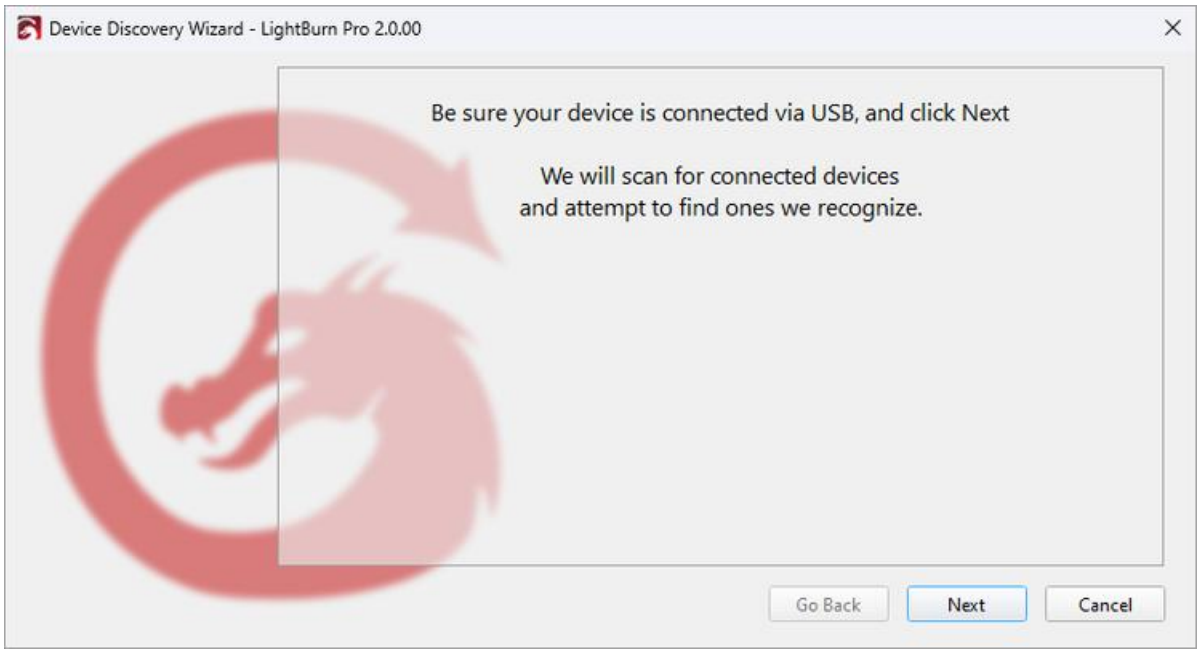
   > **Note**
   >
   > If this is your first time running LightBurn, the Devices window will open automatically.



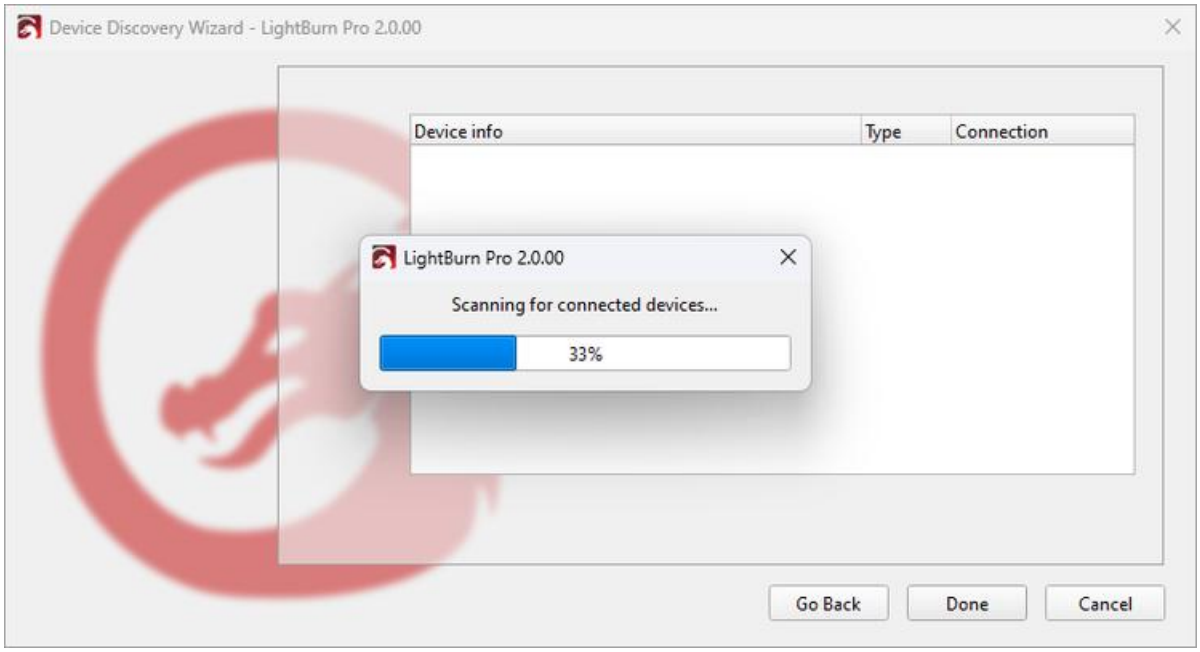
3. Click the **Find My Laser** button to launch the **Device Discovery Wizard**.



4. Click the **Next** button to begin scanning for connected devices, or the **Search Network** button to scan for networked devices.

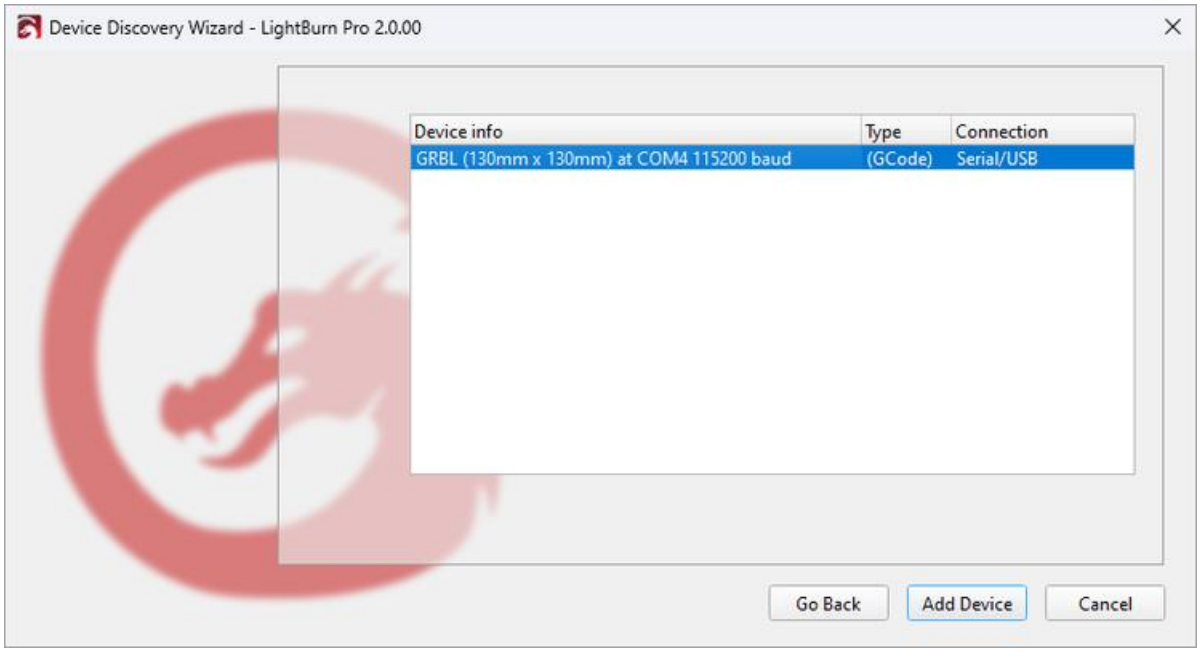


5. A new window will open showing the progress of scanning for connected devices.



6. The Device Discovery Wizard will display a list of recognized devices. Select your laser from the list and click **Add Device**.

> **Ruida: Packet vs. Serial**
>
> Some laser models will appear on the list twice, with two connection types, **Packet/USB** or **Serial/USB**. Serial/USB is the recommended option, while Packet/USB is an alternative option that does not work on all systems, but can sometimes improve file transfer problems.

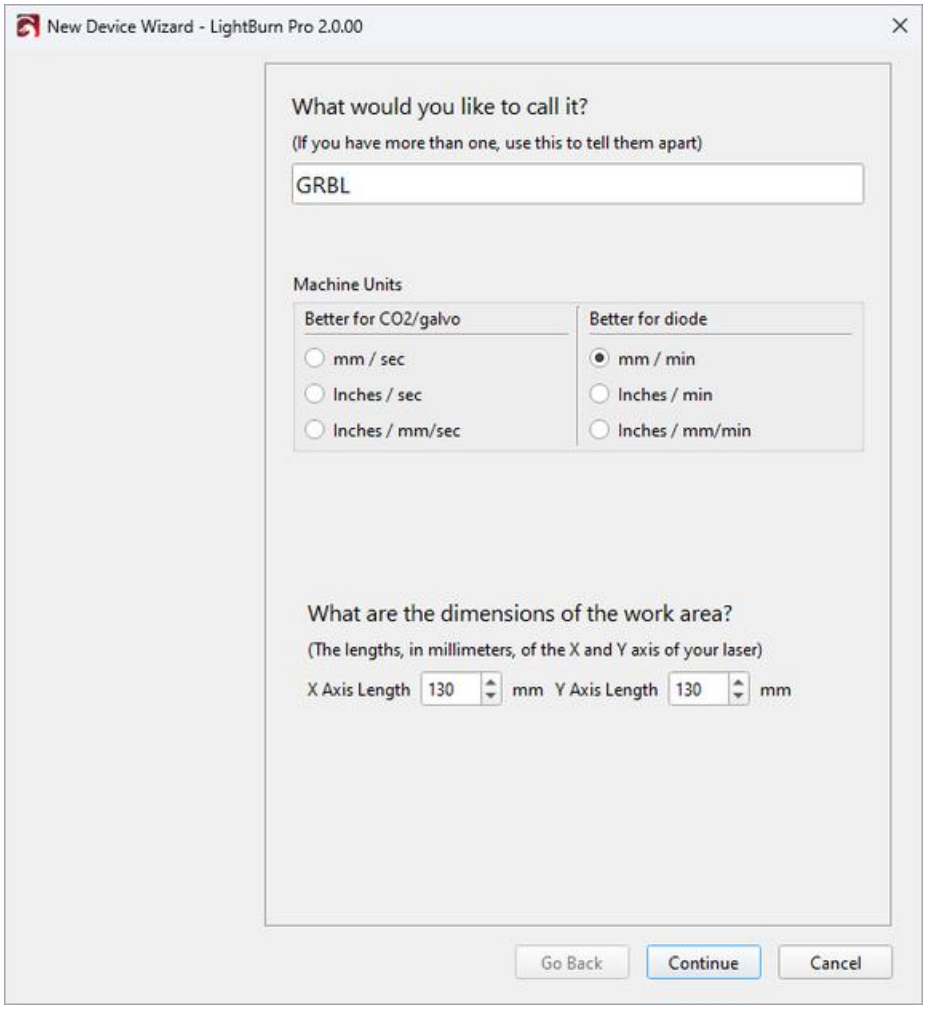


> **Warning**
>
> If your laser wasn't found automatically, see Troubleshooting.

If you don't have a laser available to connect, you'll need to manually create a device profile.

Manually Create Profile

7. Name your device, choose the units you want to use for this device, and confirm that the dimensions of the work area are correct.



Note

Some lasers require additional information in order to complete the setup process. Continue below.

Additional Hardware-Specific Instructions

Click one of the options below to see instructions specific to your type of laser.

Not sure what type of laser you have? Head back to Identify Your Laser.

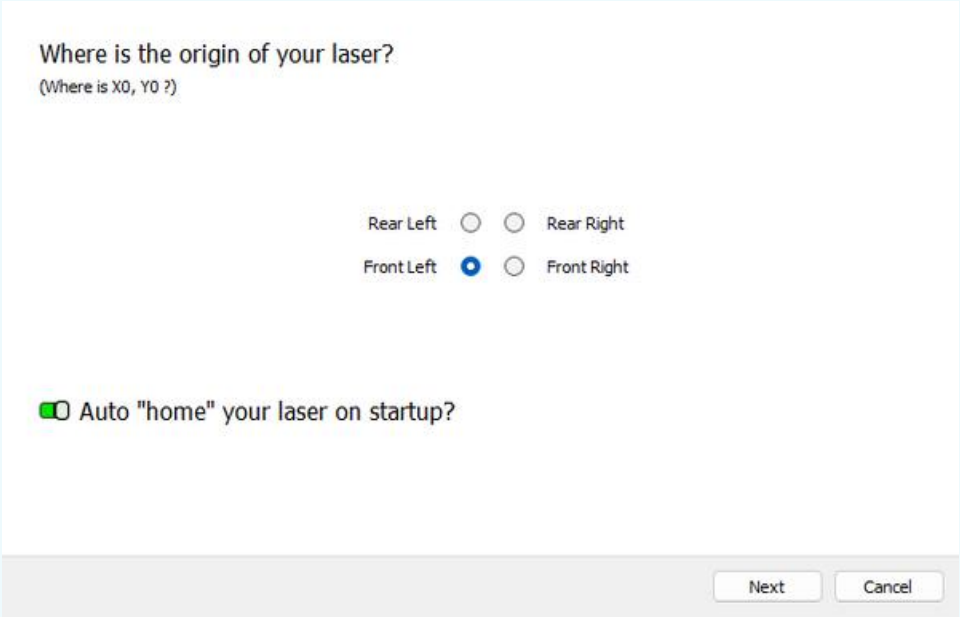
GCode Devices

GCode Devices

1. Set the machine origin. Nearly all GCode systems use the **Front Left** corner as the origin, regardless of where they home to. If you select the incorrect origin, your output will be flipped or mirrored — you can correct the origin later in Device Settings.
2. Enable **Auto "home" your laser on startup** if your machine has homing or limit switches, to automatically send your laser a homing command each time the device has been connected.

   > **Warning**
   >
   > If your laser does not have homing switches, leave **Auto-home** off.
   >
   > If you see **Error: 9** in the Console Window later, it means you've enabled this feature but your machine is not equipped or configured for homing. You can disable Auto-Home later in Device Settings.

3. Select **Next** to continue.



4. Click **OK** in the Devices window to exit and save the new device profile.

   > **Note for X-Carve and Shapeoko Users**
   >
   > Some GCode controllers used on X-Carve or Shapeoko devices may need specific configuration settings. If your device is not listed, click **Other**, and refer to manufacturer documentation for machine-specific parameters. You may also want to check out MillMage for use with CNC routers.

Next: Connecting to the Laser

DSP Devices

DSP Devices

1. Set the machine origin. For DSP lasers, the machine's origin is the same as the location it homes to when it first starts up. If unsure, select the **Rear Right** corner, as most DSP controllers follow this convention. If you select the incorrect origin, your output will be flipped or mirrored — you can correct the origin later in Device Settings.
2. Select **Next** to continue.
3. Click **OK** in the Devices window to exit and save the new device profile.

Next: Connecting to the Laser

Galvo Devices

Galvo Devices

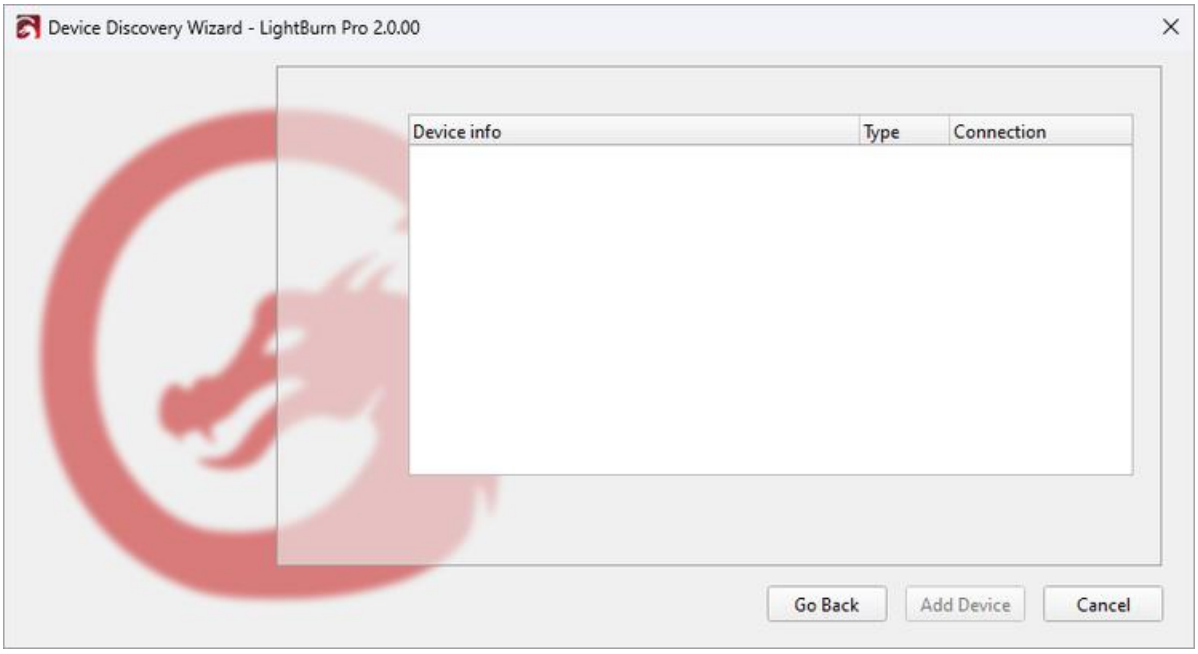
When adding a Galvo device, LightBurn will ask you to import the machine-specific configuration file from your laser's setup or installation drive. Navigate to the location it's saved and select the file to import crucial device settings. See Add a Galvo Laser for more information, including what to do if the configuration file is not available.

Video: Galvo Setup Guide

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=YbzSO20bSuA

Next: Add a Galvo Laser

Troubleshooting



If LightBurn can't find your laser, it could be for a number of reasons:

- **Missing drivers**: Install your laser-specific drivers and software. Even if you don't plan to use their software, many device manufacturers include the necessary drivers that are not included with LightBurn.
  - Ruida controllers use an FTDI driver available as part of the LightBurn installation process. Re-install LightBurn and select the box for the FTDI driver at the end of the process to install the driver — no LightBurn settings will be lost in the re-installation process.
  - EZCad2-based and EZCad3-based Galvo devices require a LightBurn-specific driver. Re-install LightBurn and select the box for either the EZCad2 or EZCad3 driver at the end of the process to install the driver — no LightBurn settings will be lost in the re-installation process. See Galvo Driver Installation for more information.
  - If you have a BSL Galvo laser, you may need to separately install a driver supplied by your machine's manufacturer, or install the stock software that came with your machine.
  - Most diode lasers require the CH340 driver, available here.
  - See Troubleshooting: Drivers for more information.
- **Can't connect**: Only one application can talk to your laser at a time. If you run other software, like RDWorks, Easel, Carbide Create, AutoLaser, or LaserCAD, make sure that software is not running when running LightBurn.
- **Networked device**: Not all network-connected lasers cannot be automatically connected. Follow the steps in Create Manually to set up your device.
- **Marlin controller**: Marlin controllers have a variety of baud rates and configuration options. They take significantly longer to reset than most other controllers, making them incompatible with **Find My Laser**. Follow the steps in Create Manually to set up your device.

Create Manually

Consult Manufacturer Instructions

Always consult the documentation that came with your laser, or the manufacturer's online resources, before beginning the device setup process. Some manufacturers provide settings bundles, importable device profiles, or special setup instructions that are necessary to use their machines effectively with LightBurn.

If you need to add a laser that isn't connected to your computer or is connected over a network, you can create a device profile manually.

What if I don't have a laser yet?

If you don't have a laser, use the **No Machine** device type to create a placeholder device of any size you want.

LightBurn Without a Laser

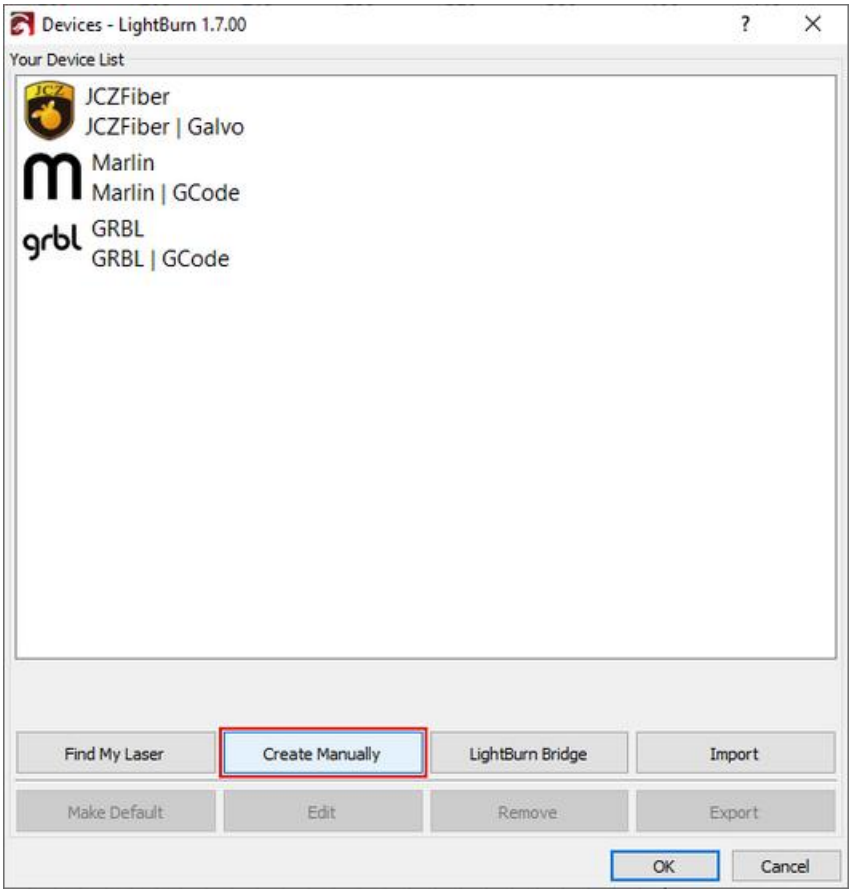
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=icWK5v8E6rc

First Steps

Start by clicking the **Devices** button in the Laser window to open the Devices window.



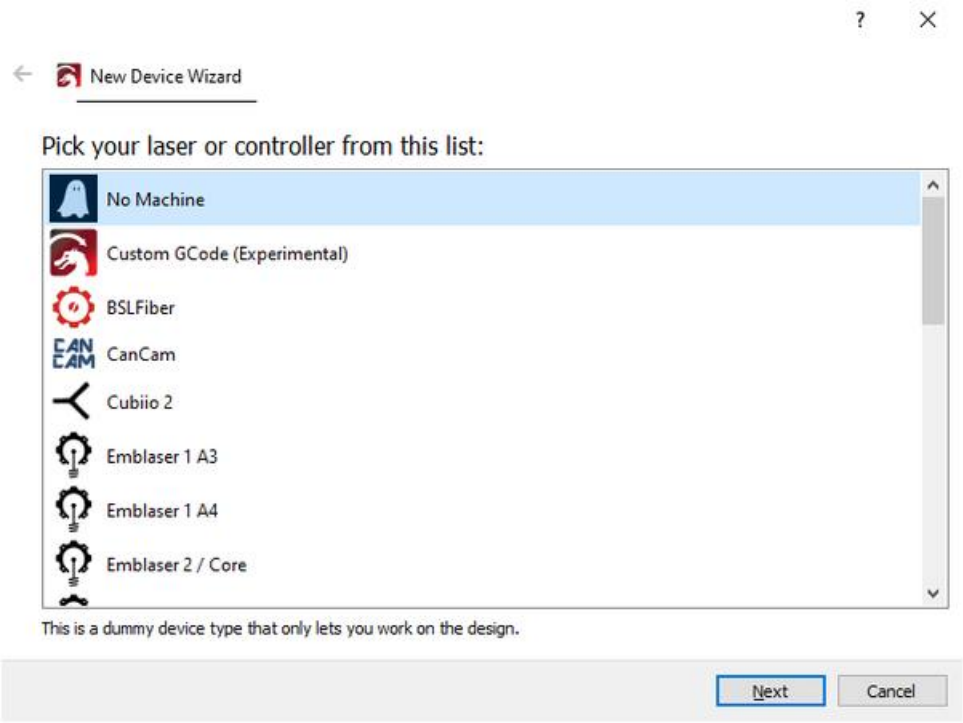
Click the **Create Manually** button in the Devices window to open the **New Device Wizard**.



New Device Wizard

Device Type

On the first page of the **New Device Wizard**, you'll see a list of controllers and types of firmware supported by your version of LightBurn.

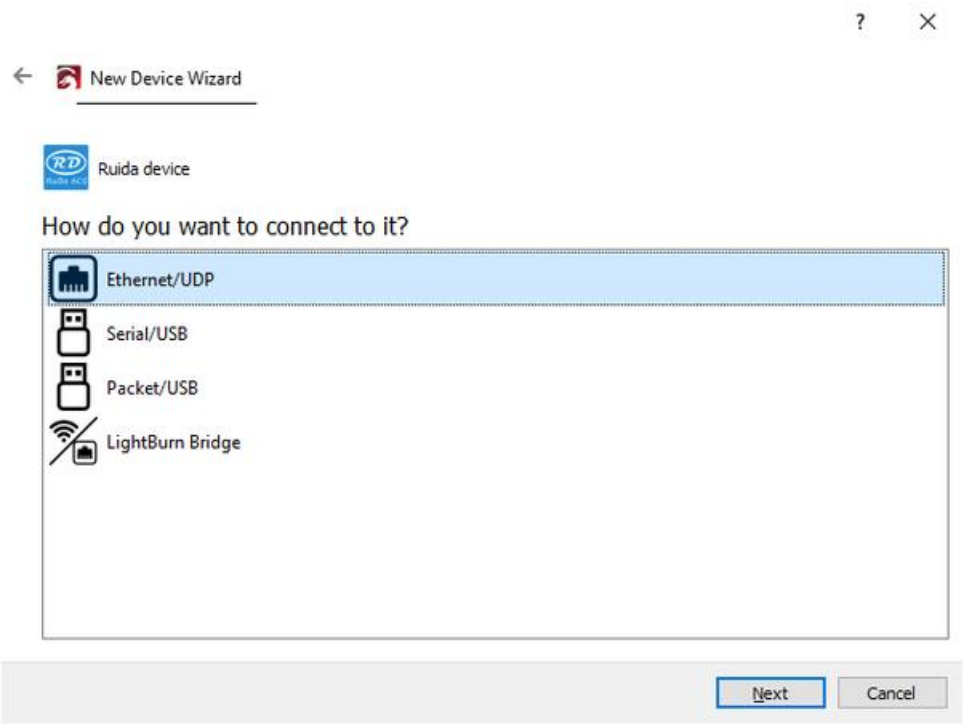


- If you don't have a device to connect yet, choose the **No Machine** type to move ahead with using the software.
- The **Custom GCode** device type allows you to create custom templates for generating GCode. This won't be necessary for most users, but if you have a custom machine or purchased a machine from a vendor using non-standard GCode, go to Custom GCode to learn more.
- All other options refer to specific controllers or types of firmware. If you are unsure what type of controller or firmware your laser is using, consult your laser's manufacturer.

After choosing the correct device type for your laser, click **Next**.

Connection Type

The next step is choosing how you connect to your laser. The choices you see will depend on the type of connection methods supported by your controller and LightBurn.



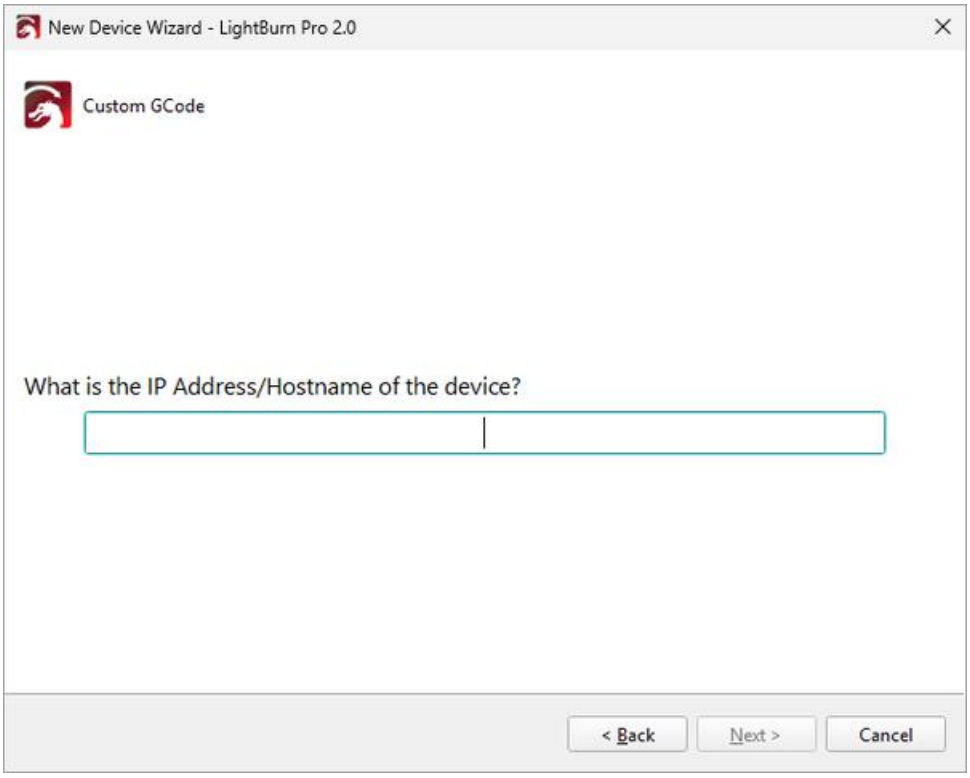
- **Ethernet or Wifi/TCP** is used for both wired and wireless network connections.
  - See Connecting via Ethernet to Ruida Controllers or GRBL Network Connection for more information.
- **Serial/USB** is the most common connection type, used for direct USB connections.
- **Packet/USB** is an alternative option for Ruida controllers only, which uses a built-in driver, similar to RDWorks. This option can be inconsistent on some systems, and is best not to use unless you're having issues with a Serial/USB connection.
- **LightBurn Bridge** is used to connect to a LightBurn Bridge. This option is available for Ruida controllers only, and if this is your first time setting up a Bridge, it's best to use the **Bridge Discovery Wizard** instead of **Create Manually**.

After selecting your connection method, click **Next**.

Ethernet and LightBurn Bridge — Set IP Address

> **Ethernet on macOS 15+**
>
> If you're using macOS 15 or higher, you'll need to allow LightBurn access to the local network.

If you selected **Ethernet or Wifi/TCP**, or **LightBurn Bridge** as your connection type, you'll need to enter the device's IP address or hostname on your local network.

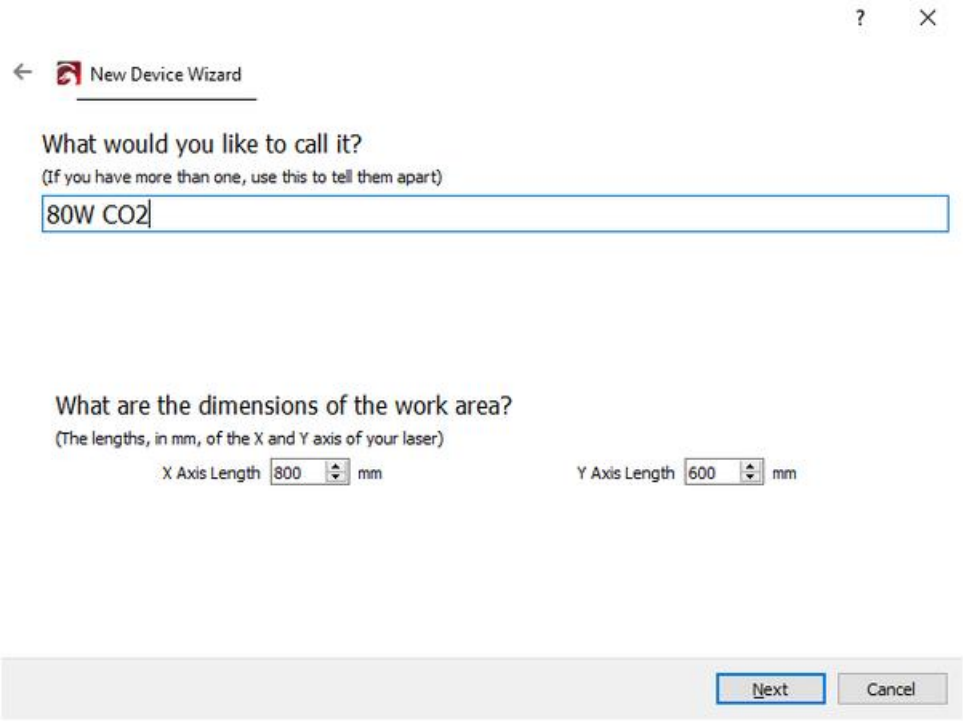


After setting the IP address, click **Next**.

Name and Work Area Size

You can give your laser a custom name, which is often useful if you have more than one, or just leave it as is.

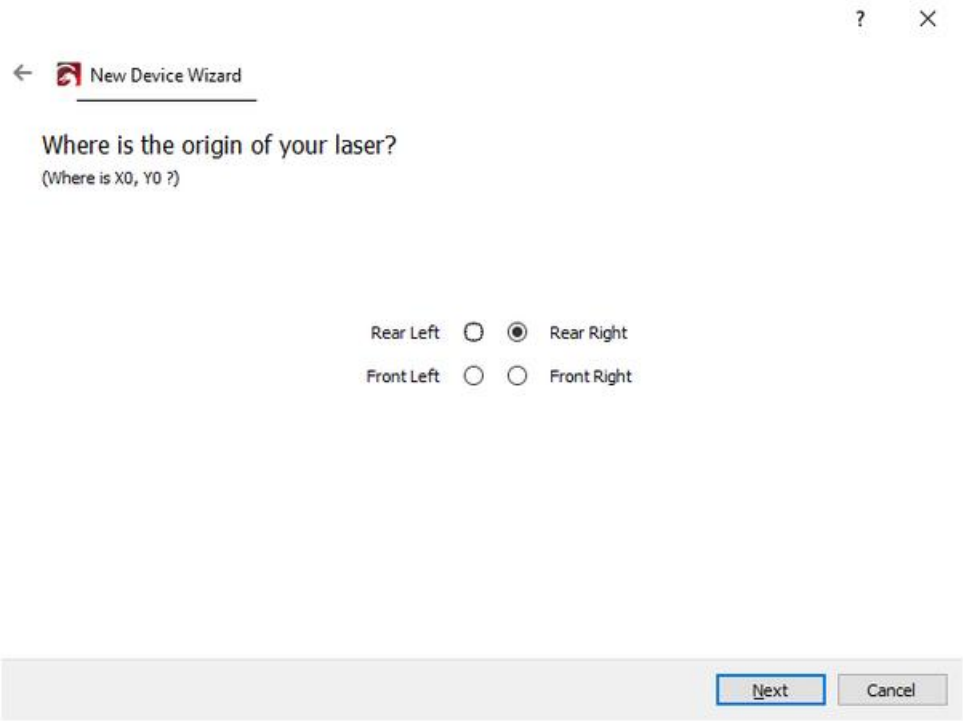
You must set the size of the work area for your laser so that LightBurn can customize the Workspace to match the laser's working dimensions. If you don't know the exact size, you can change this later in the Device Settings window.



After naming and entering your machine's dimensions, click **Next**.

Laser Origin And Homing

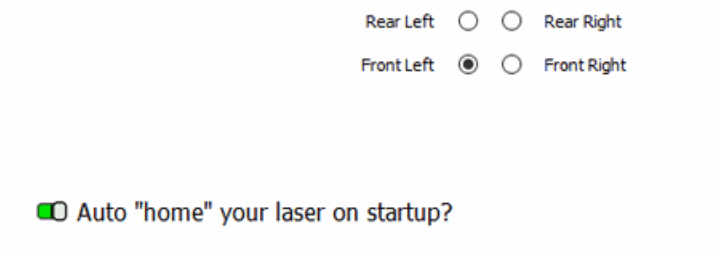
The **Origin** setting is your laser's 0,0 point — where the X & Y Axis both meet and begin. If you get this wrong, you can change it later in the Device Settings window.



- With a few exceptions, the Origin for GCode lasers is almost always the bottom left.

- If you have a DSP laser, set the Origin to the location the laser homes to when it first powers on.
- If you are unsure about your laser's Origin point, consult your laser's manufacturer.
- The Origin also controls the orientation of output to your laser — if it's wrong, output to your laser may be mirrored or upside down.

If you have a GCode based system, you'll see the option to send the homing command each time LightBurn first connects to your laser. You can change this setting later in the Device Settings Window.



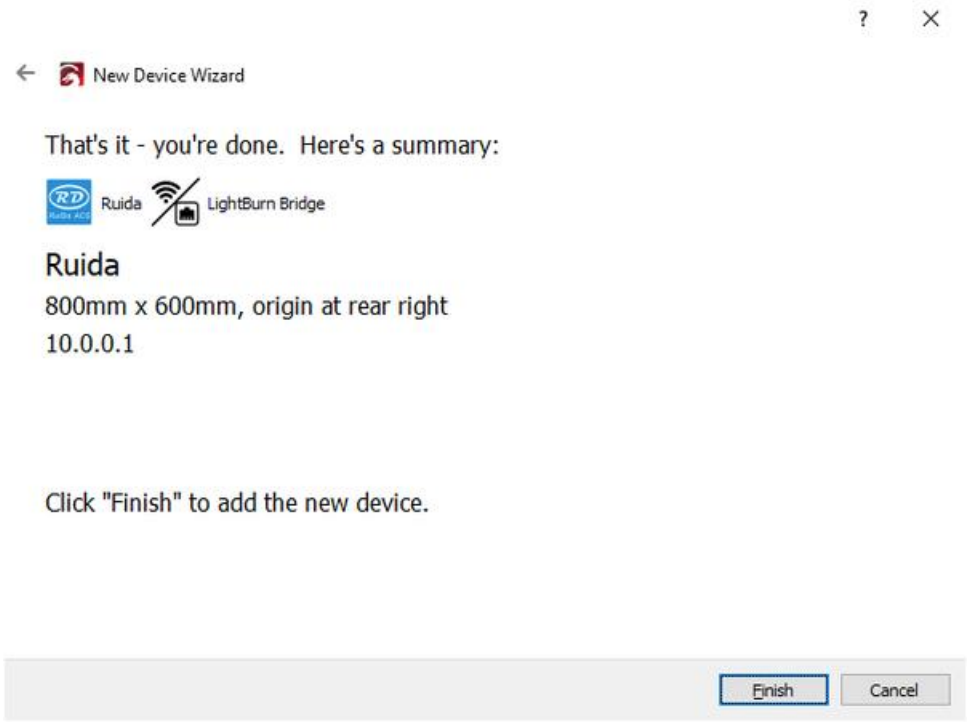
Warning

If your laser does not have homing switches, leave this off.

After setting the Origin, click **Next**.

Review Settings

The final page will show you a summary of your choices. You can click **Back** to return to a previous step and fix anything if necessary, or click **Finish** to create the new device entry.



Confirming Device Creation

Click **OK** to save and confirm the creation of the new device. You'll return to the **Devices** window once you've exited the **New Device Wizard**.

If you click **Cancel** or click the window close button, all the changes you have made in the Devices window will be discarded — including creating the new device.

Add a Galvo Laser

Consult Manufacturer Instructions

Always consult the documentation that came with your laser, or the manufacturer's online resources, before beginning the device setup process. Some manufacturers provide settings bundles, importable device profiles, or special setup instructions that are necessary to use their machines effectively with LightBurn.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=YbzSO20bSuA

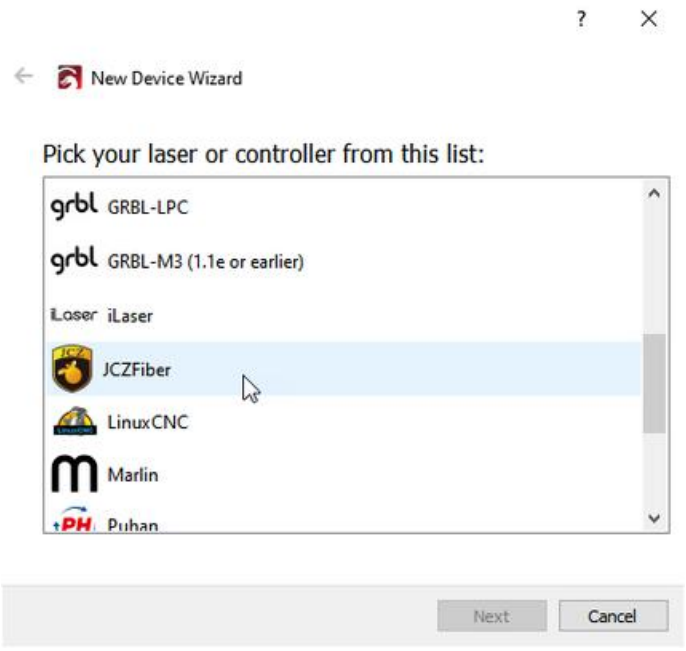
When adding a Galvo device, LightBurn will ask you to import the machine-specific configuration file from your EZCAD2, EZCAD3, or SEACAD installation drive or from an existing installation. If you don't have this file, you can also use manual entry to configure your laser with settings provided by the manufacturer.

It's a good idea to set up a new device profile for each lens you have, because changing lenses will affect many parameters. You can duplicate a device after setting it up by right-clicking on it in the **Devices** window and choosing **Duplicate**. This is the easiest way to add a new profile while keeping the settings, so you only have to update parameters that change with the lens.

See Changing a Galvo Laser Lens for more information.

Import Settings

Follow the directions in Find My Laser to begin the process of adding your Galvo laser. Your laser should be detected as **JCZFiber** (for EZCAD2 devices), **BSLFiber** (for SeaCAD devices), or **EZCad3**. Select the appropriate device and connection type – USB is currently the only supported type for Galvo devices – and continue with the process.



The **New Device Wizard** will ask you to import one or more configuration files, which will configure your specific laser appropriately. The name and expected location will vary depending on the type of Galvo laser you've selected:

JCZFiber / EZCad2

- The EZCad2 configuration file is typically `markcfg7`, but can occasionally be `markcfg0`.
- You can usually find this on the supplied drive by going to the `EZCad` application directory/folder, opening the `plug` folder, and selecting the `MarkCfg7` file.

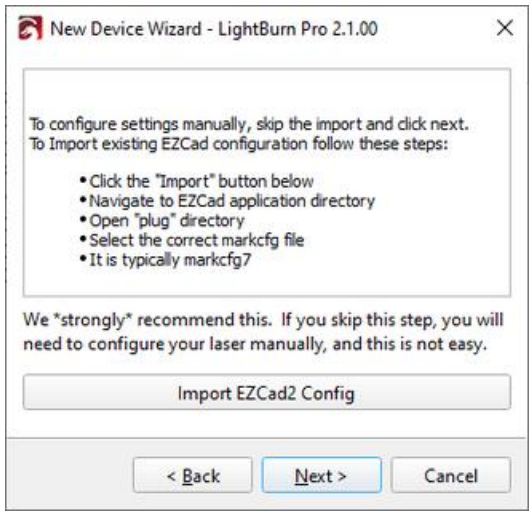
BSLFiber / SeaCAD

- For BSLFiber devices based on SeaCAD, the configuration file is usually `BslCAD.cfg` or `LmcPar.cfg`.
- You can usually find this on the supplied drive by going to the `SEACad` application directory/folder, openining the `config` folder if it's present, and selecting the appropriate configuration file.

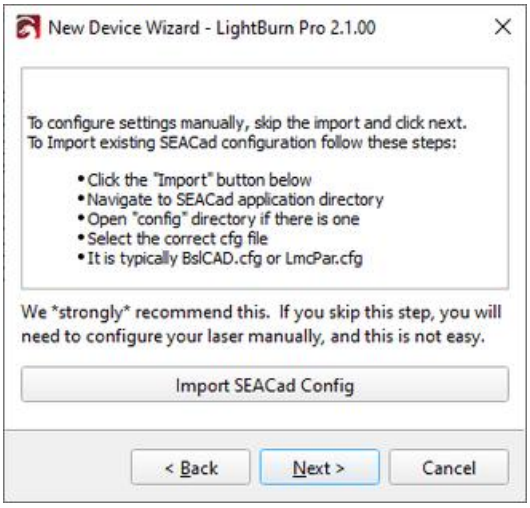
EZCad3

- For EZCAD3 devices, the configuration files are typically called `markcfg0`, `Motors.ini`, and `COR`.
- You can usually find `MarkCfg0` by going to the `EZCad3` application directory/folder, opening the `PARAM` folder, and selecting the `MarkCfg0` file.
- If `Motors.ini` exists, it's generally in the same location as `MarkCfg0`.
- If your laser came with a COR file, it's usually in a separate folder called `COR File`.

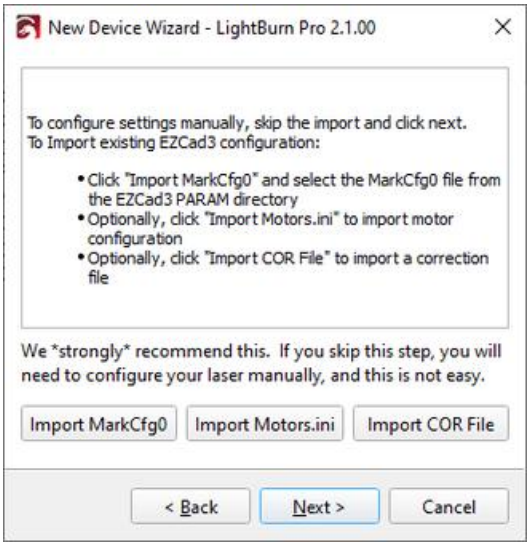
See the window in LightBurn for additional details on where to look for the file.



Importing an EZCAD config file



Importing a SeaCAD config file

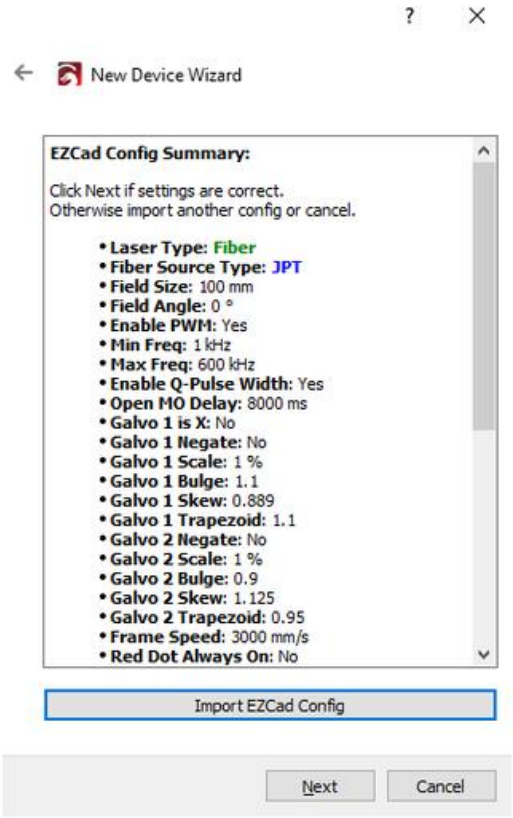


Importing EZCAD3 config files

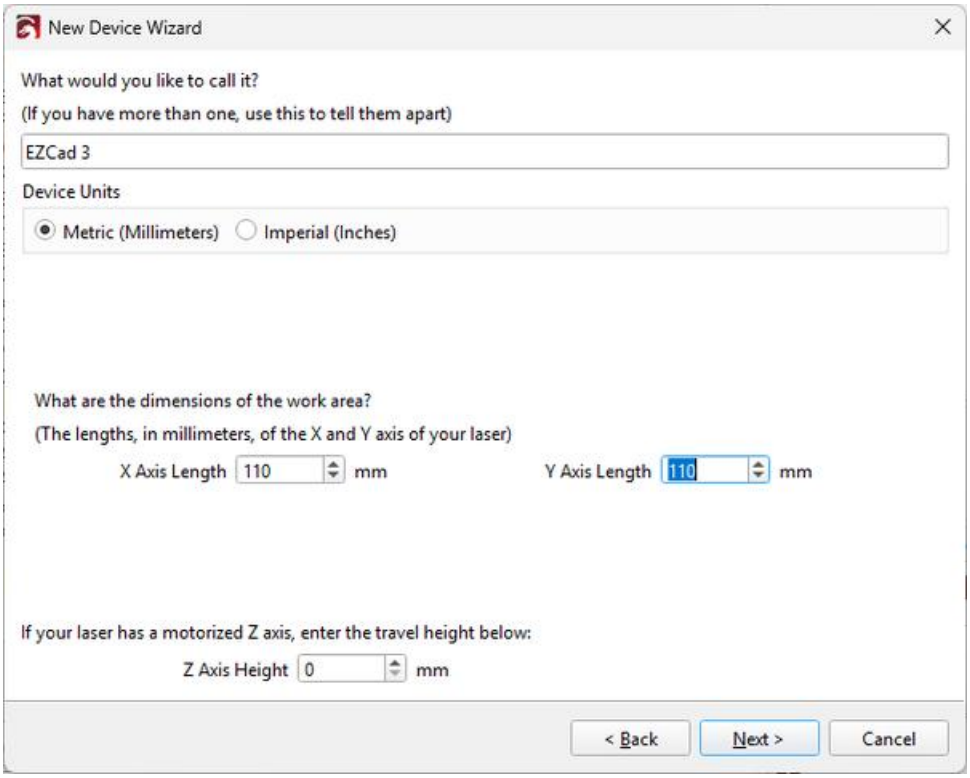
Select your configuration file – typically found on the USB drive you received from the manufacturer – and click **Next**. LightBurn will show you a preview of the settings to be imported.

If you don't have a configuration file(s), click **Next** without selecting a file and you'll be able to enter settings manually. If your laser's manufacturer didn't provide the necessary settings, please contact them for assistance.

> **Info**
>
> If your markcfg7 file includes information about Lens Correction, LightBurn will attempt to automatically import this for you. If your configuration file doesn't include this info, you'll need to manually enter it in Device Settings after finishing the New Device Wizard.



After entering the size of the work area, setting the name of the laser, and setting the travel distance of your laser's Z axis (if it's motorized), you're ready to connect to your laser.



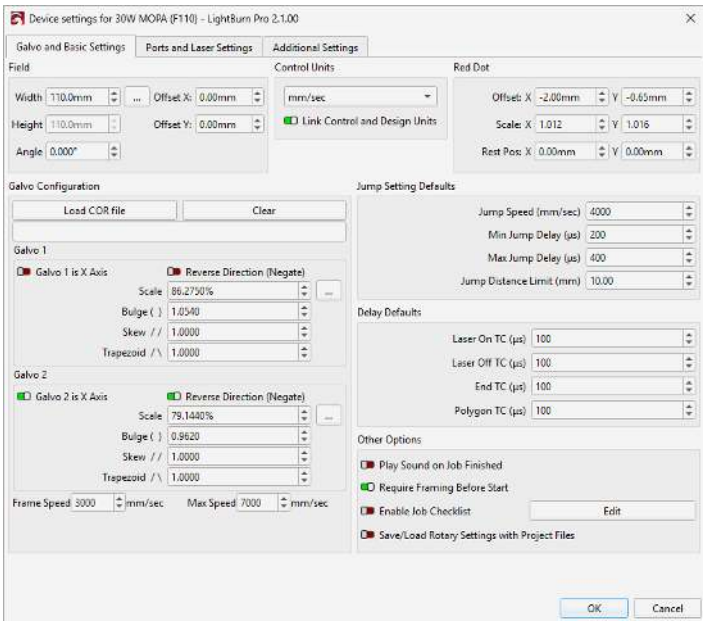
Next: Connect to the Laser

Manually Add a Galvo

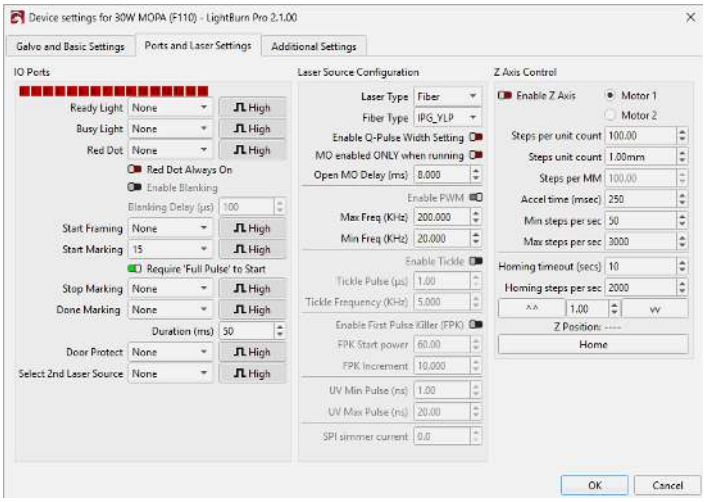
If your laser did not come with the settings on a USB drive or if you don't have the USB drive available, you may need to contact the manufacturer for settings. Galvo settings can't be found through trial and error, due to the complexity and variety of machine configuration. Settings that work on one laser may destroy another.

If your settings were provided on a card, in an email, or in any other way that doesn't work with the automatic importer, click **Next** when asked to import the EZCAD config to skip this step, and enter your settings manually after adding the laser.

Go to Edit → Device Settings to open the **Device Settings** window. From here, you can manually enter the settings provided by the manufacturer in the Galvo and Basic Settings tab and the Ports and Laser Settings tab.

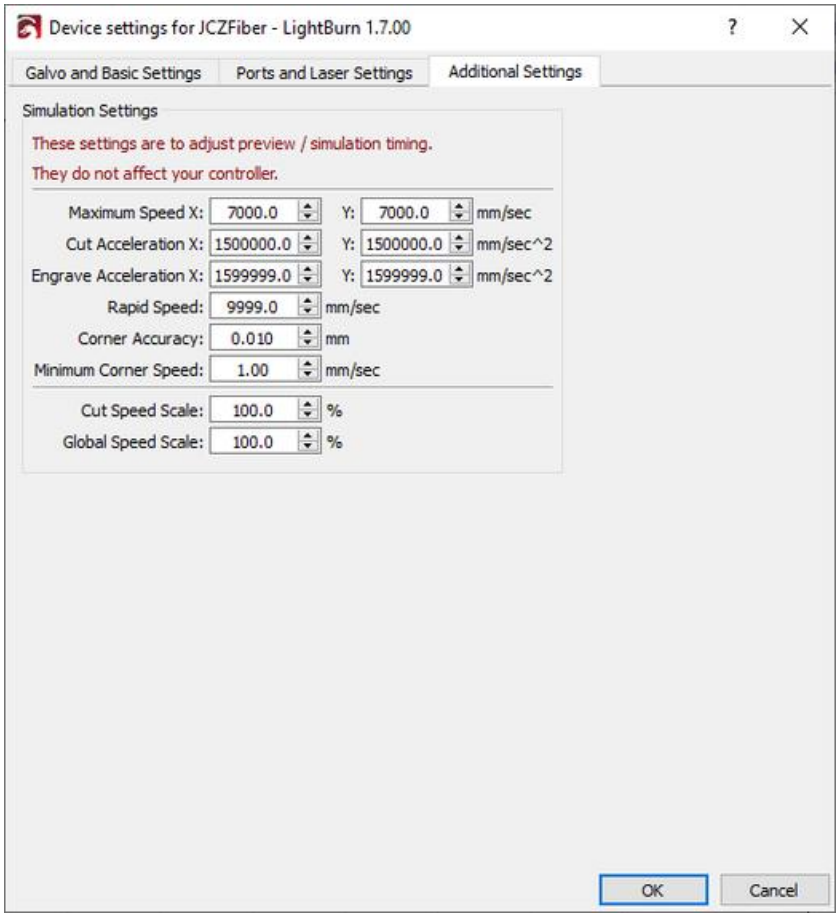


The Galvo and Basic Settings tab



The Ports and Laser Settings tab

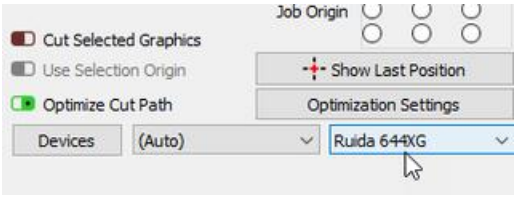
Please note that when entering these settings manually, the Additional Settings tab needs to be updated with the correct information for the simulation in Preview to function correctly.



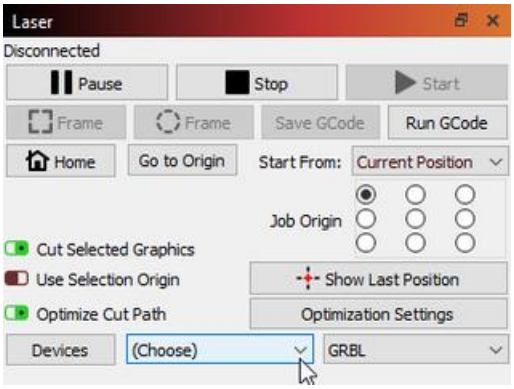
Connecting to Your Laser

Once a laser has been added to LightBurn, it should appear in the list of devices to the right of the **Devices** button in the Laser Window. If you only have a single laser, it will be automatically chosen for you.

If you have more than one laser set up, you might need to select the one to connect to by clicking here:



Depending on the type of controller you have, you might have to manually choose the port that the laser is connected to, by clicking where you see **Choose** in the Laser Window:



As long as your laser is connected to the same communication port, LightBurn will reconnect when you re-start. If you reboot your computer, or plug the controller into a different USB port, you might need to re-select it.

If you see (Choose) as shown above, you need to select the port. If no ports are listed in the drop-down, it means that no devices were found, which could mean that it is not plugged in correctly, isn't powered, or you're missing a driver.

If your laser disconnects for some reason, or enters an alarm state and needs to be reset, you can quickly re-connect by right-clicking the **Devices** button in the Laser Window.

Troubleshooting

If you're having trouble connecting to your laser, see Connection Troubleshooting for help.

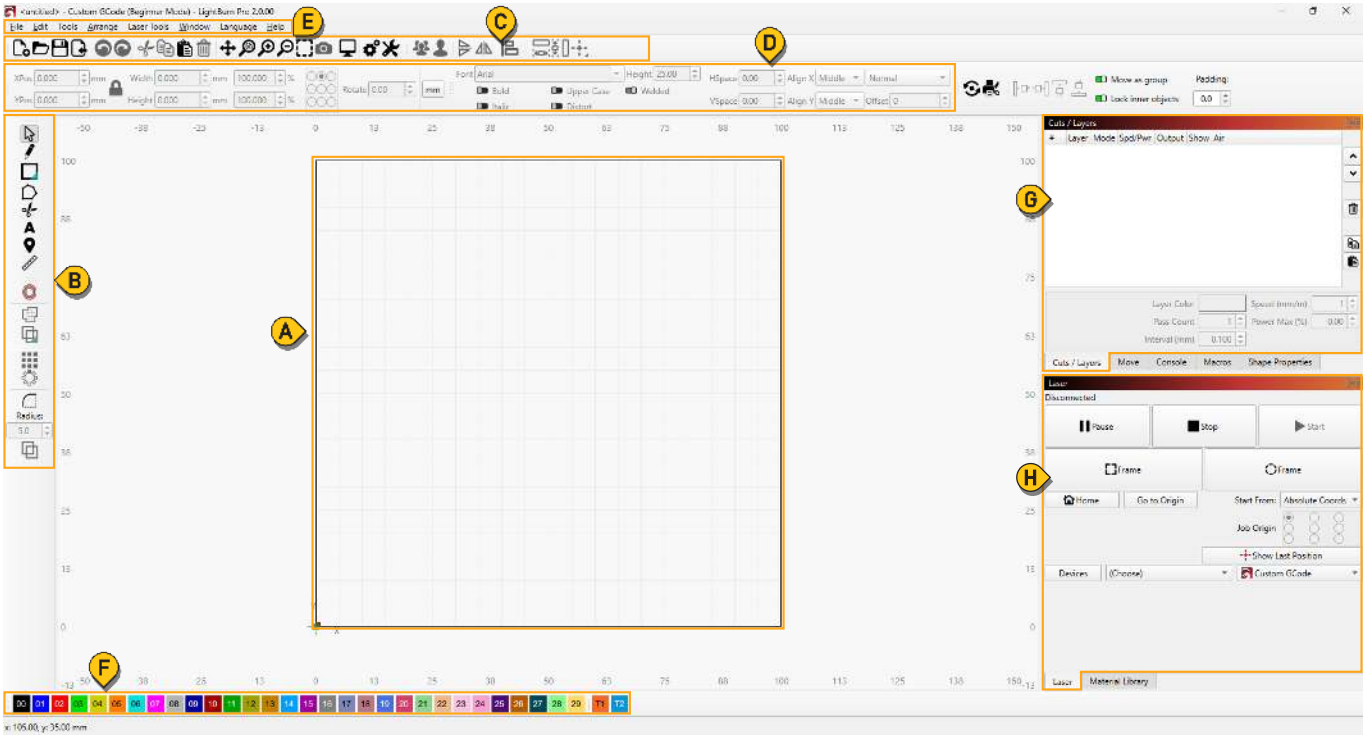
User Interface Tour

Once you've installed LightBurn and set up your first device profile, you'll see LightBurn's Main Window. If you've never used LightBurn before, the Main Window might seem a little intimidating. Try not to let it scare you — we'll break down the most important sections to get familiar with below.

> **Finding Help**
>
> LightBurn has some built-in features to make it easier to learn more:
>
> - **Easy Access Help Pages**: Press `F1` while hovering to launch the documentation page for that feature in your default web browser.
> - **Tools Tips**: Hover your mouse over a control to see helpful text pop up, with a description of that button or feature.

Below, we've divided the Main Window into labeled sections. The labeled sections contain the tools and features you'll use most frequently in LightBurn, and we'll give a brief explanation of each further down. You can click in the image beneath it to jump directly to the information on that section.

You may notice that some sections are unlabeled. To keep from getting overwhelmed when starting out, it's best to focus on the basics, and leave those sections for later. At the end of this walkthrough, we'll give you some advice on finding and learning more about tools that could work for your specific needs.



> **Info**

> Your LightBurn window may not match the screenshot exactly. LightBurn's user interface changes depending on what type of laser is selected and whether Beginner Mode is active.

A: Workspace / Edit Window

B: Creation and Modifier Toolbars

C: Main and Arrange Toolbars

D: Numeric Edits and Text Options Toolbar

E: Menus

F: Color Palette

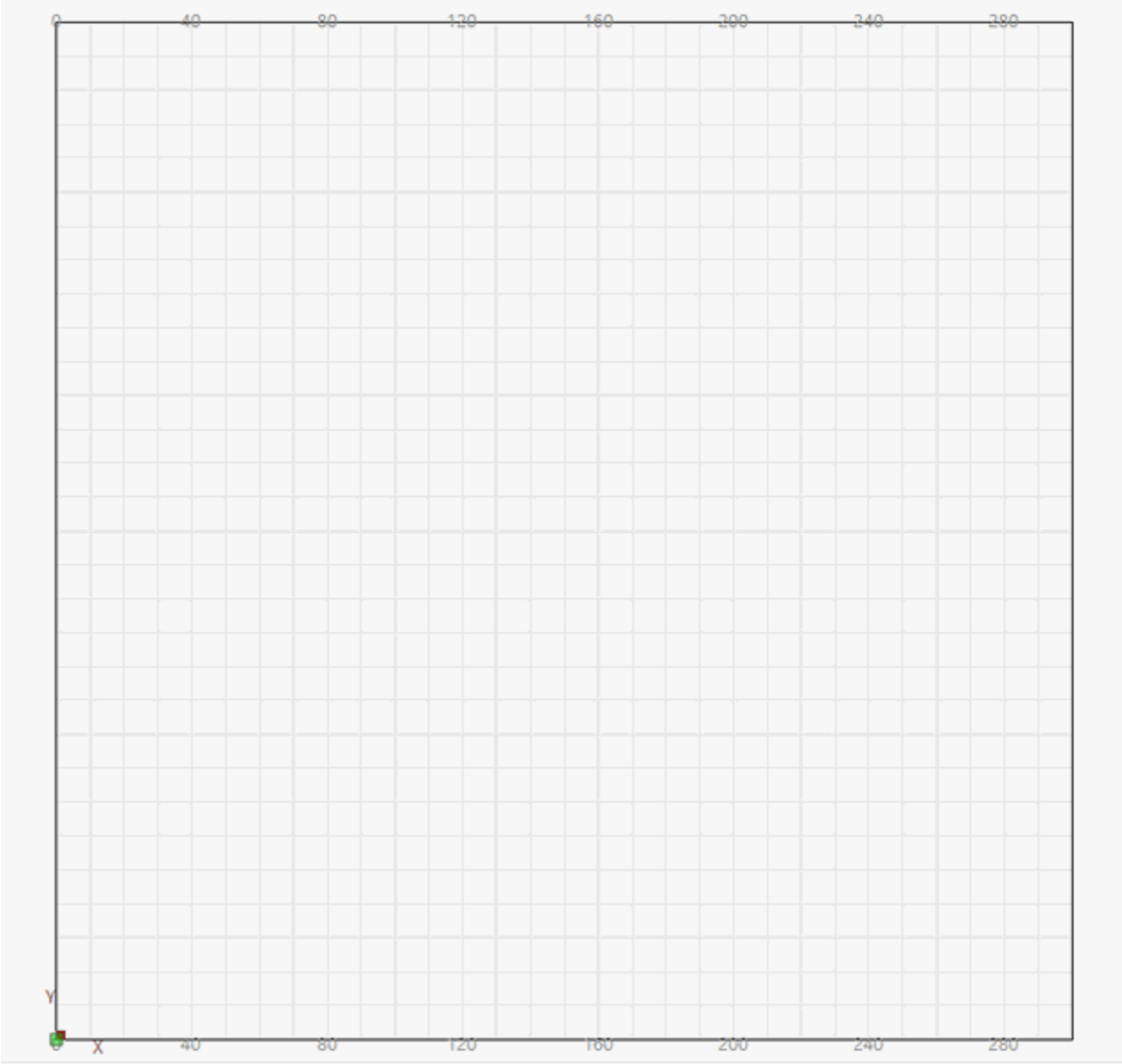
G: Cuts / Layers Window

H: Laser Window

Workspace / Edit Window

The Workspace, or **Edit Window**, is the area where you create, edit, and lay out graphics in LightBurn. The grid is a representation of your machine's available work area, determined by the settings LightBurn detected or you entered manually when you set up your device profile.

The artwork arranged and displayed within the Workspace will be processed and sent to your laser — we'll explain how that works in greater detail later.



Main, Arrange, Creation, and Modifier Toolbars

These toolbars contain buttons that, when clicked, perform certain actions.

Above the **Workspace**, between the Menus and Numeric Edits and Text Options Toolbars, are the Main and Arrange Toolbars.

Along the left side of your Workspace are the Creation and Modifier Toolbars. They're usually oriented vertically, but we show them horizontally below.

There's a lot to say about all these tools, but here's a brief rundown:

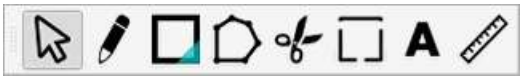
- The **Main Toolbar** contains functions for opening or importing files, saving, using the clipboard (copy and paste), and moving or zooming your view.



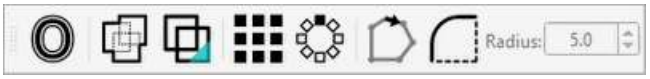
- The **Arrange Toolbar** contains tools for positioning objects, both on an individual basis and in relation to each other.



- The **Creation Toolbar** contains tools for selecting objects in your Workspace, or creating Primary Shapes, text, lines, and more.



- The **Modifiers Toolbar** contains tools for combining, altering, and producing arrayed duplicates of graphics.



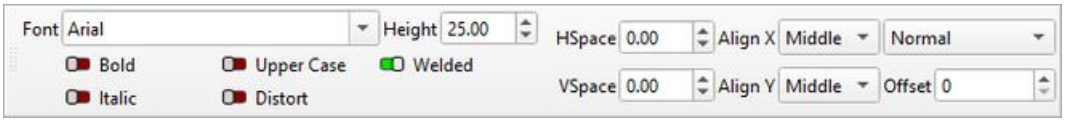
Numeric Edits and Text Options Toolbars

Located just above the **Workspace**, the Numeric Edits and Text Options Toolbars are special toolbars that allow you to directly edit specific properties of shapes.

- Use the **Numeric Edits Toolbar** to adjust objects' position, size, or orientation.



- Use the **Text Options Toolbar** to control the font, size, style, and alignment of selected text.



The options in the Numeric Edits Toolbar are only available when you have objects in your selection, and the options in the Text Edit Toolbar are only available when you have a specific type of object in your selection: text made with the Create Text tool.

Menus

The Menus at the top of LightBurn contain almost every tool that's available as a button in one of the toolbars, along with a handful more. Most have icons that correspond with a toolbar button (if applicable), and the keyboard shortcuts for their actions (if available) are listed next to each menu option.

Some tools are only available in the Menus or by keyboard shortcut — for the most part these are advanced options that you don't need to worry about when you're starting out, but we'll cover a couple Menu-only options as we go along.



Color Palette

Along the bottom of the Main Window, you'll see the Color Palette.

Lasers don't print in color, so these colors are only used to assign different kinds of operations to the shapes in your design.

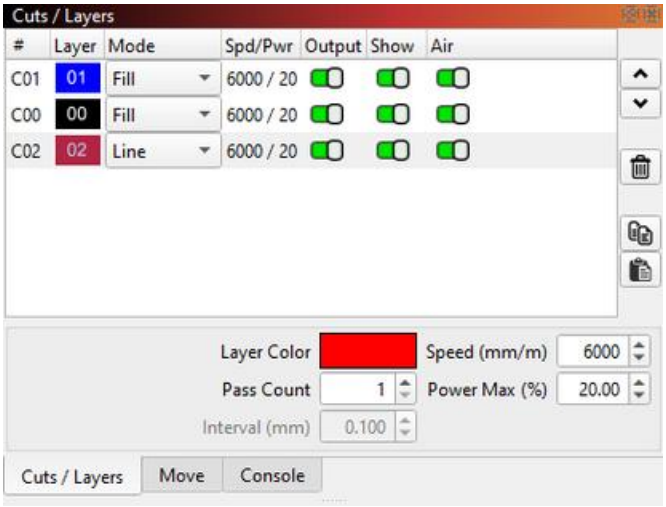
With nothing selected in the Workspace, click a color entry and subsequent new shapes will be created in that color. If you have something selected, clicking a color entry will apply that color to the shapes in your selection. The colors currently in use in your design will also appear as entries in the **Cuts / Layers Window**, where you can choose the operations that each color will represent.



Cuts / Layers Window

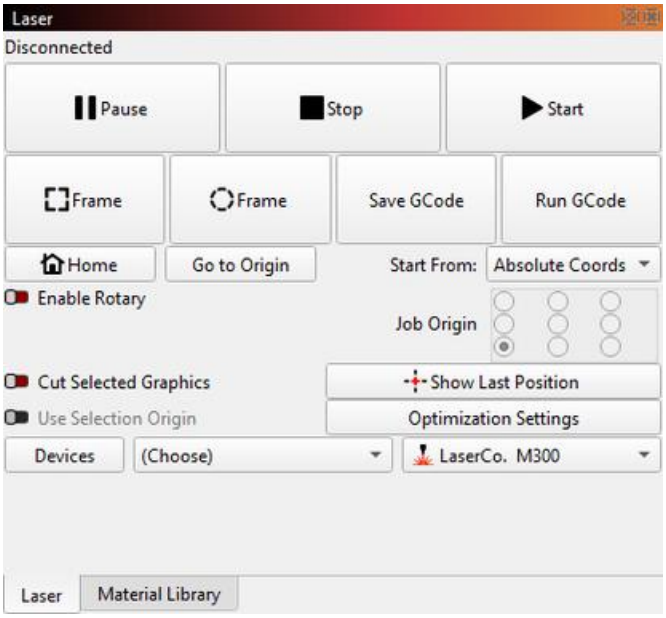
Located in the upper right of LightBurn by default, the Cut / Layers Window shows the layers currently in your design, along with a a handful of essential cut settings like **Speed** and **Power**.

You can double-click a layer in the Cuts / Layers Window to open the Cut Settings Editor, where you can access more detailed settings that change depending on the type of operation you have a given layer set to.



Laser Window

The Laser Window is located in the bottom right corner of LightBurn by default. It's the hub for setting up and selecting a laser, Framing and Starting jobs, determining job positioning, and more. In short — it's an important window!

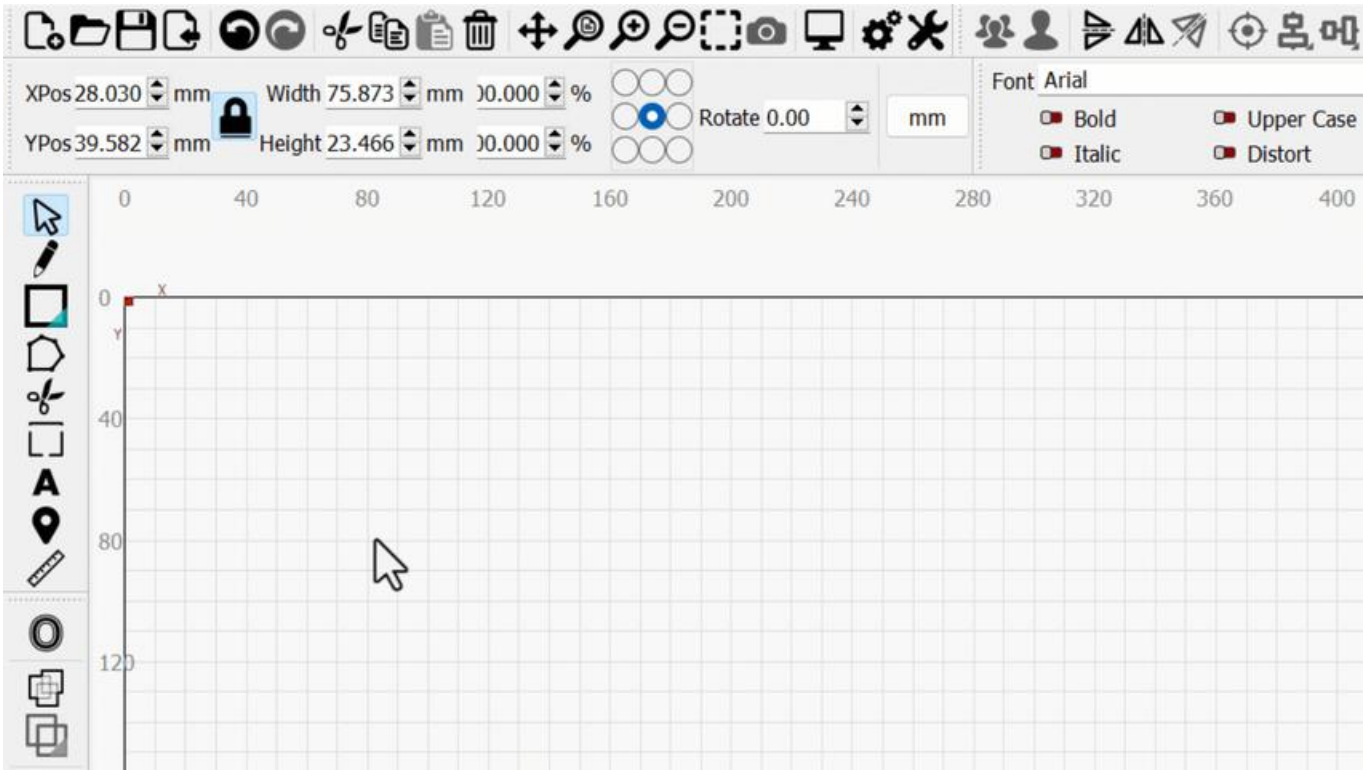


Note

This window may look a little different depending on the type of device you have selected, whether or not you have Beginner Mode enabled, and a few other options available in your Settings / Preferences.

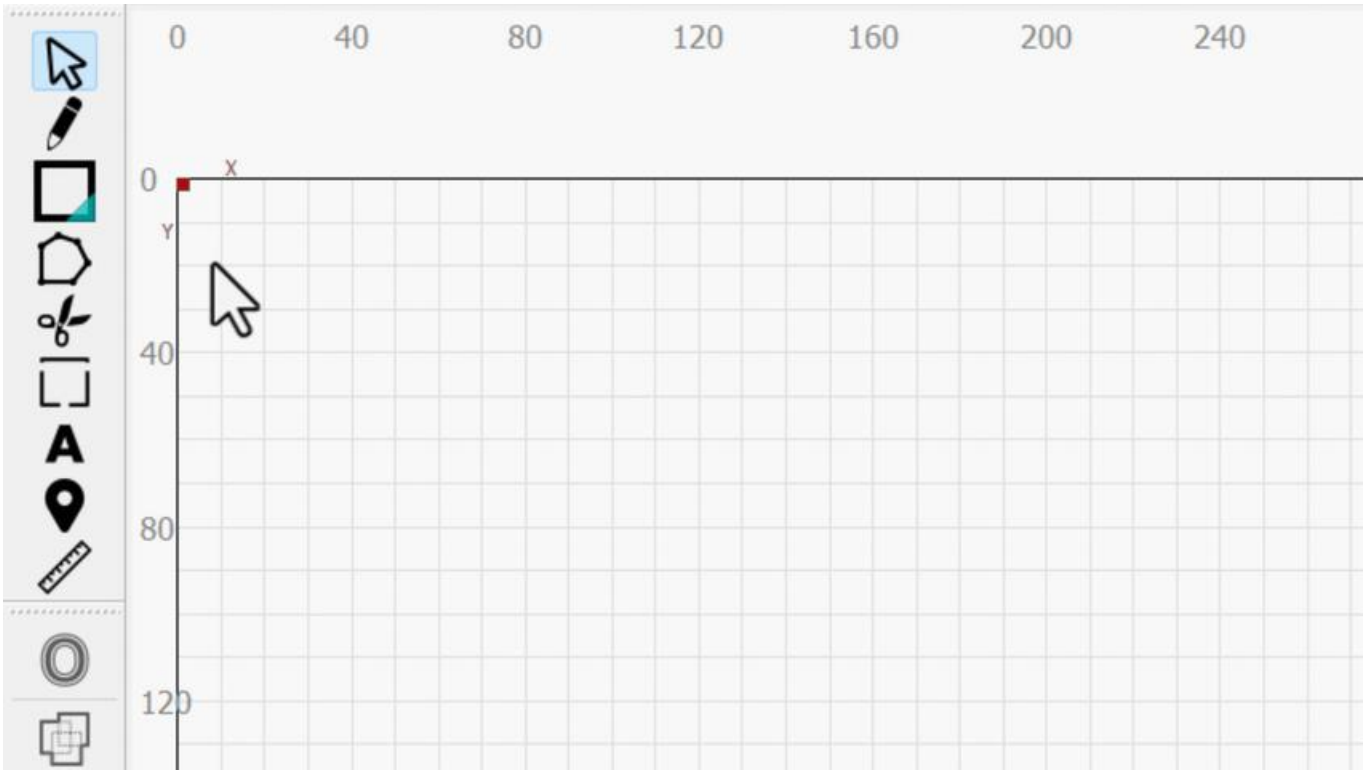
Getting Familiar and Learning More

We'll have more to say about some of these toolbars and windows as we go along, but, for now, spend some time in LightBurn hovering over buttons to see what they do. As you hover, a tooltip will appear and give a brief description of the action the button performs. In many cases, you'll also see a keyboard shortcut — pressing that combination of keys will perform the button's action, or activate that tool.

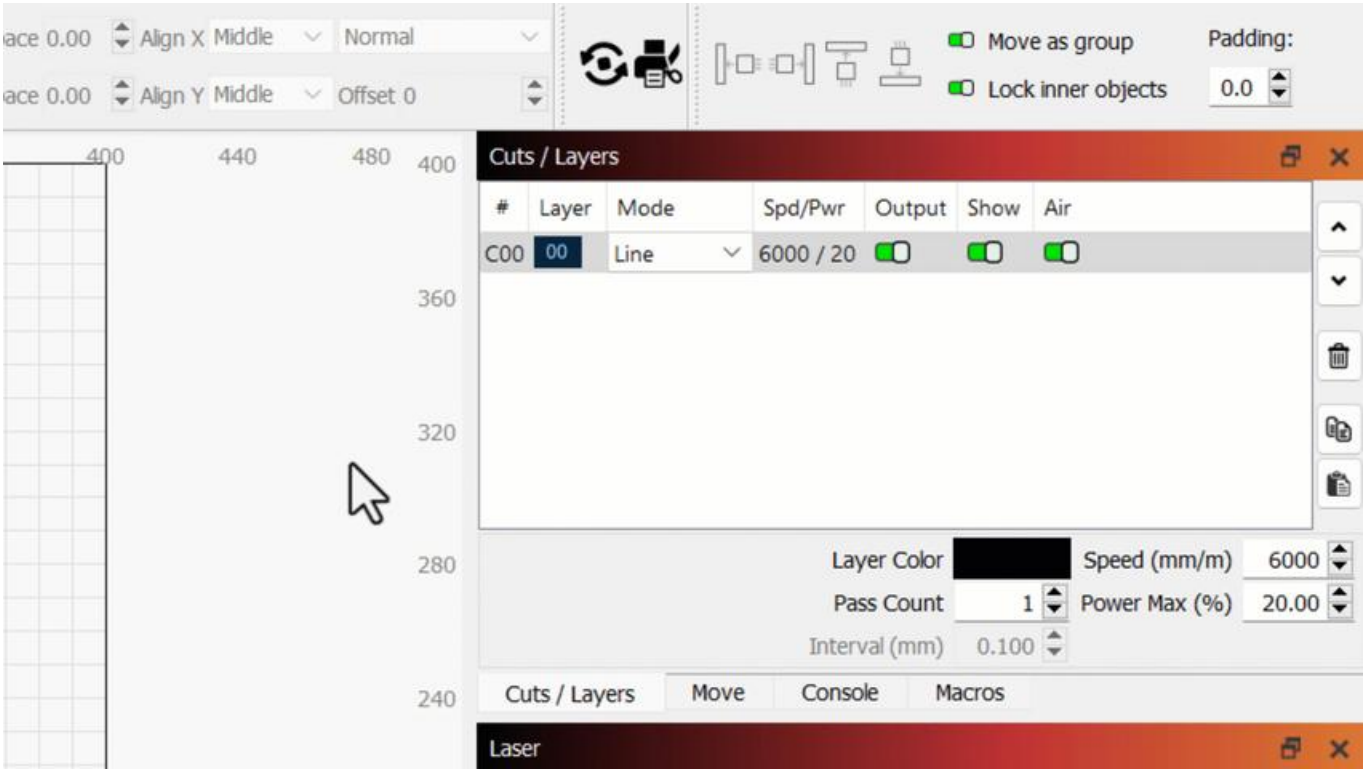


Some buttons will be grayed out and unclickable — that means you can't perform that button's action unless you have valid shapes in your selection. We'll explain how to select objects later, but if you ever run into a case where you do have objects selected and aren't sure why a button is still grayed out, try hovering over the button and pressing `F1` to go to the docs page for that tool, and learn what types of shapes are required to use it.

You'll also see a few buttons with small arrows in their bottom left corners. That means several tools are collapsed within a single button — click the arrow to expand the list of tools.



Finally, you'll notice labeled tabs beneath the Cuts / Layers and Laser Windows. Clicking on one of these tabs will bring a new window to the front, and hide the previously exposed window behind it. We haven't covered any of the windows that are hidden by default in our brief tour because you won't use them as often, but try clicking tabs to show and hide different windows, just to get the hang of it.



Creating and Importing Artwork

If you've followed along so far, you've installed LightBurn, set up your first device profile, connected to your laser, and gotten familiar with the LightBurn interface. If you need a breather, we wouldn't blame you — we'll be waiting when you get back.

When you're ready, it's time to dig into the reason you're here: using LightBurn to make stuff with your laser. Every laser project requires some sort of artwork to send to the laser, and that artwork has to come from somewhere. You can create original graphics directly in LightBurn, or import graphics created elsewhere and modify them (or just leave them be).

Creating

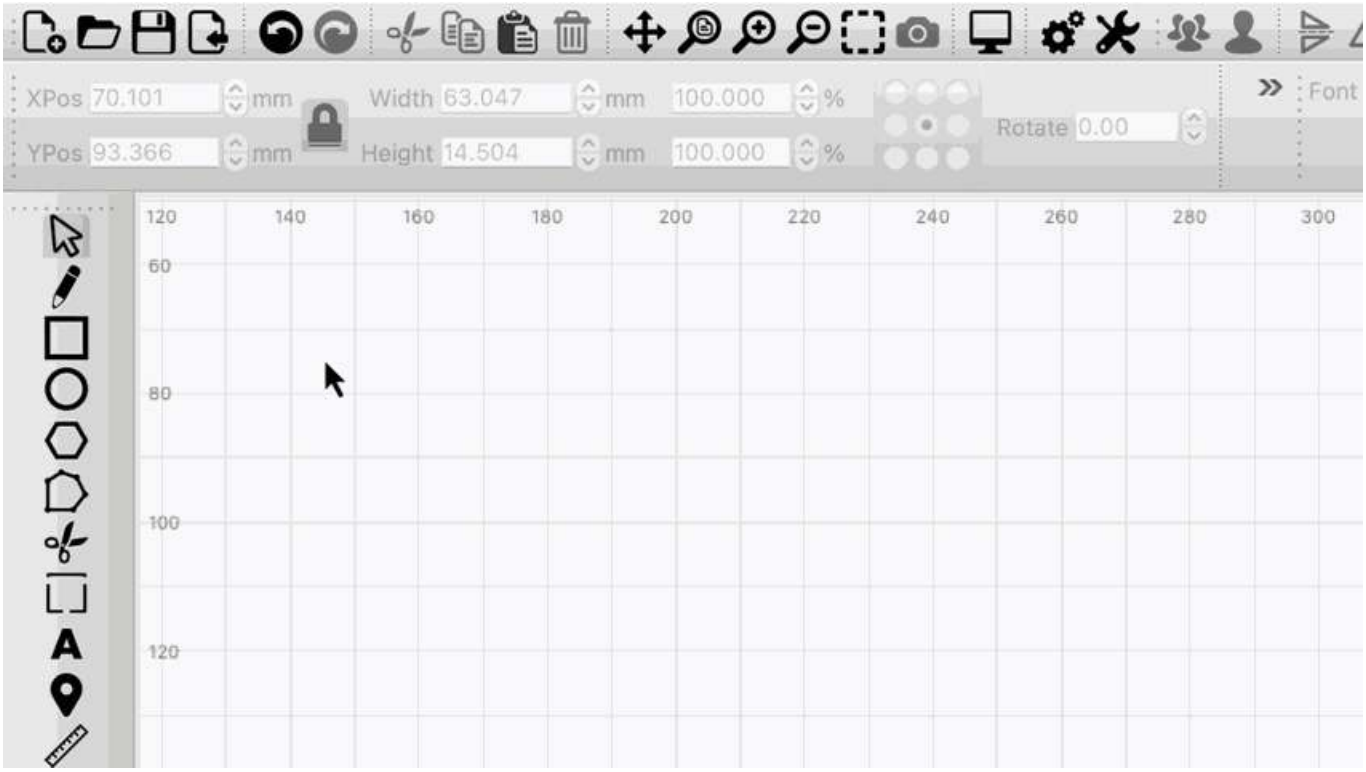
There are three tools in the Creation Toolbar that you can use to make original graphics in your Workspace. All three work similarly, with minor differences. As with most LightBurn tools, there's a lot we could say about each of them, but we'll stick to brief explanations for now.

Draw Lines

You can use the Draw Lines tool to create custom lines and curves. Select the tool, click in your **Workspace** to place one end of a line, then drag and click again to place another point, and a line between. Press `S` to toggle between creating straight lines and curves between points.

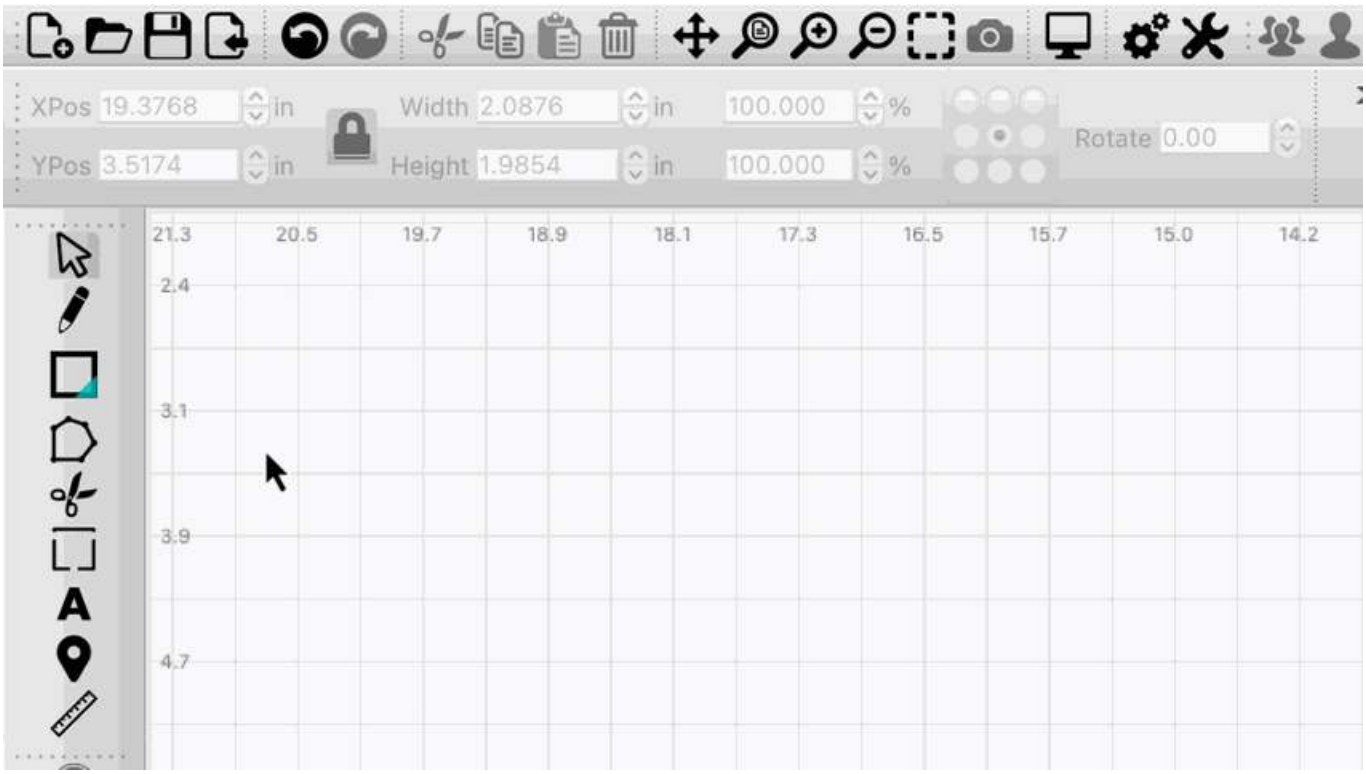
Click here to see a full video guide on using the Draw Lines tool

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=RURJuZhn_IA



Draw Shapes

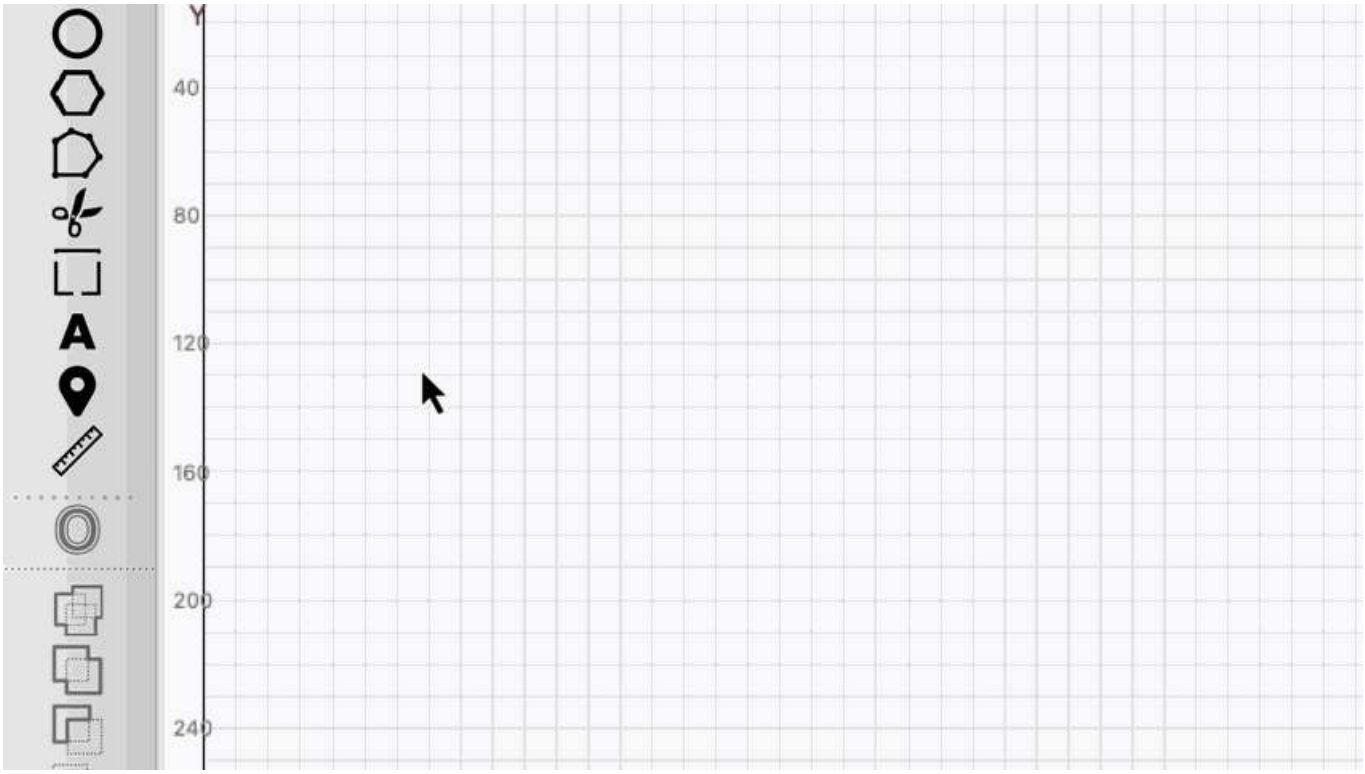
Click the arrow in the bottom right corner of the Draw Shapes icon to select from a menu of eight different types of shapes. Then click in your **Workspace**, hold, and drag to set the size of the shape. Release to finish creating it. If you hold `Shift` while dragging, the shape's width and height will be the same, so you can create perfect circles and squares using the **Ellipse** and **Rectangle** options.



Create Text

Click the Create Text icon to enable the tool, then click anywhere in your **Workspace** and type in the text you'd like to add.

Press `Enter` while typing your text to jump to a new line and continue typing. Press `Esc` to finish creating the text.



Importing

We know that many people prefer to design in other software they're already familiar with, or to work from stock artwork.

For that reason, LightBurn can Import a variety of file formats:

Vector / mixed formats:

- `.ai` — Adobe Illustrator
- `.svg` — Scalable Vector Graphics
- `.dxf` — AutoCAD Drawing Exchange Format
- `.pdf` — Adobe Portable Document Format
- `.plt` / `.hpgl` — Plotter / Hewlett-Packard Graphics Language

Importing Text

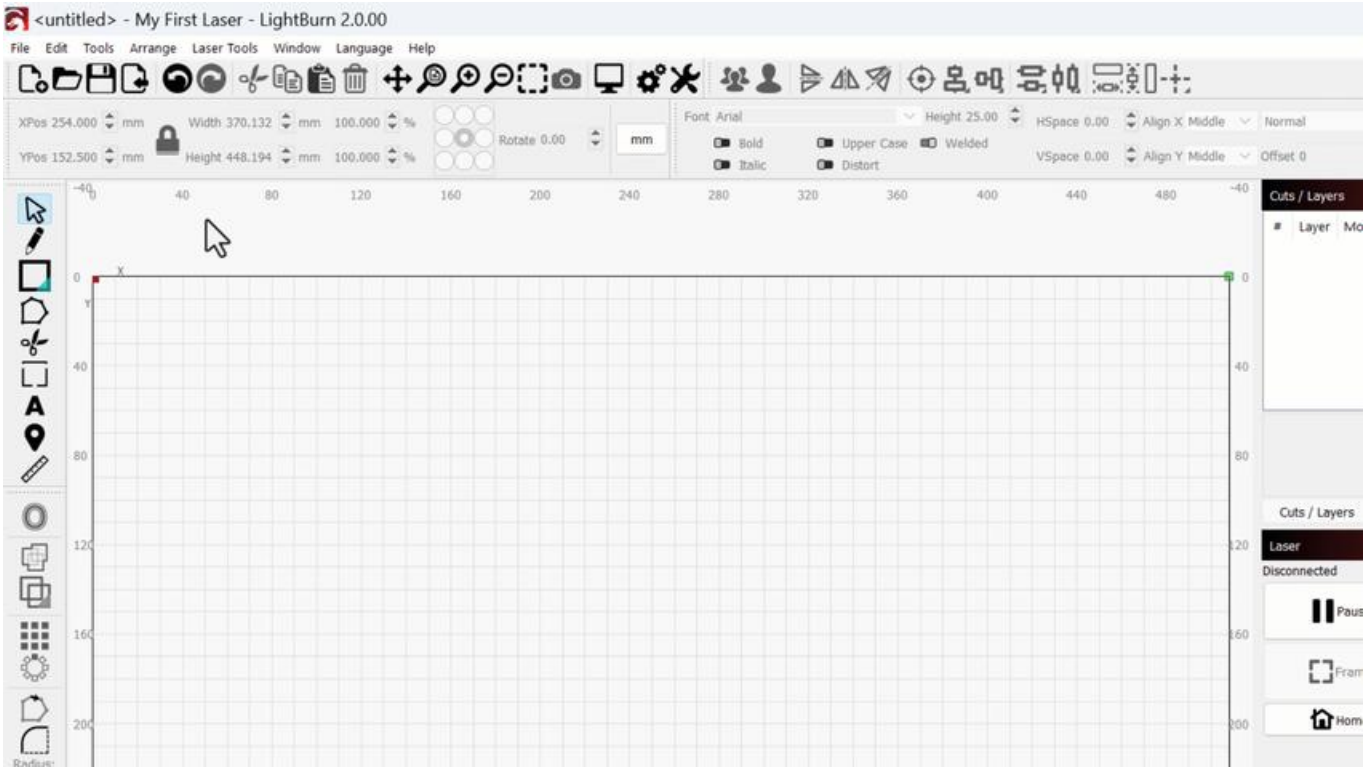
If you import vector files not made in LightBurn that contain text, make sure that any fonts used in the file are installed on your computer, or that the text has been converted to paths using whatever tools are available in the other program. LightBurn is not able to import embedded fonts. If LightBurn is unable to import anything from your file, the status bar will display **Nothing Imported**.

Different programs use different terms for this process, but terms such as welding, convert to outline, and convert to path are common.

Image formats:

- `.png` — Portable Network Graphics
- `.jpg` / `.jpeg` — Joint Photographic Experts Group format
- `.bmp` — Windows Bitmap
- `.gif` — Graphics Interchange Format
- `.tif` / `.tiff` — Tagged Image File Format

To import artwork saved in any of the above formats, click the **Import** button in the Main Toolbar, go to File → **Import**, or press `Ctrl`/`Cmd`+`I`, then navigate to the location the file is saved on your computer and select it.



Use caution with AI-generated files

If you use AI/LLM tools to generate files, please be aware that the files generated may not conform to standards for the file type you're importing and you may experience failed imports or unexpected results.

Quick Tip: Drag and drop directly into LightBurn

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=oQQK_ES4Xgo

Quick Tip: Importation Tricks

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=WGScA_9-RbE

Moving Around the Workspace

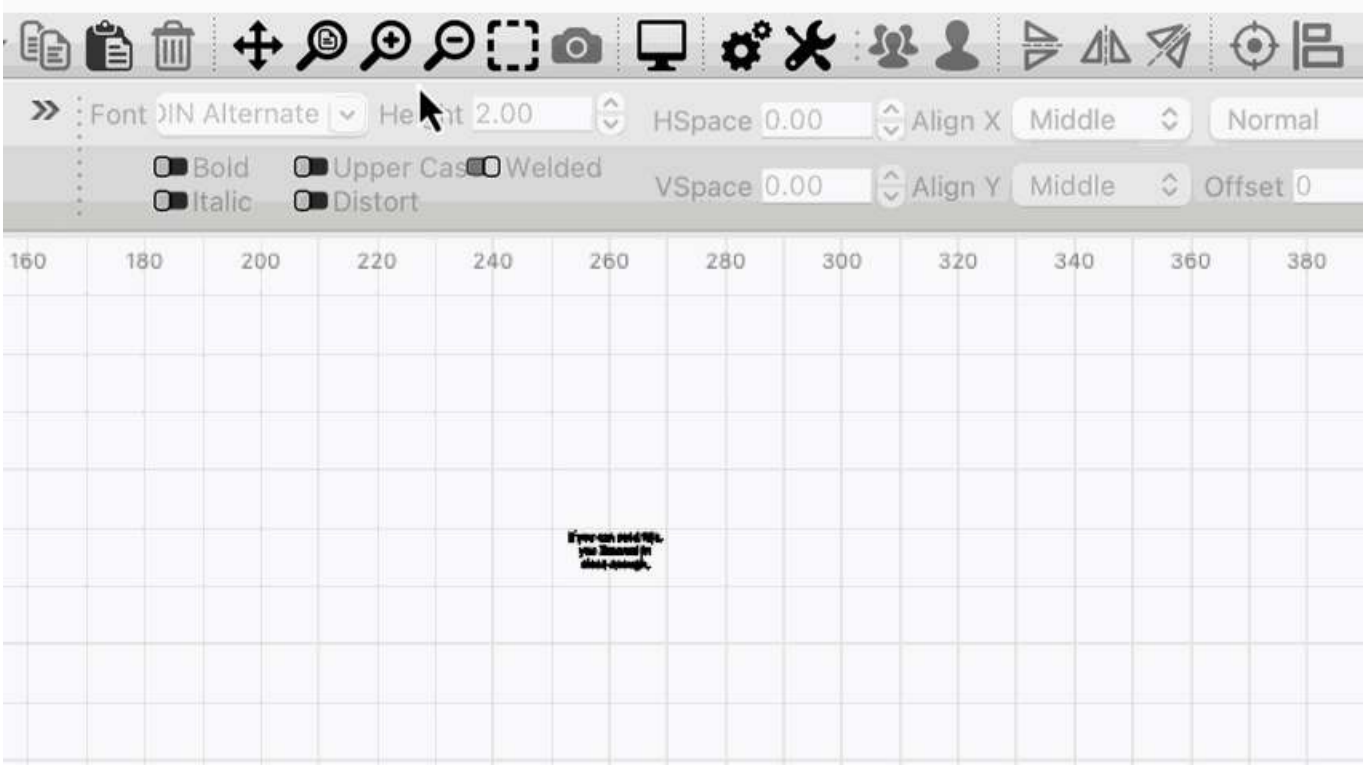
Once you've created or imported artwork into your Workspace, you may want to make modifications, and in many cases the easiest way to do that is to adjust your focus using the **Zooming** and **Panning** tools.

Zooming

Use **Zooming** to get a close-up look at your graphics, or to pull back out and see the bigger picture.

There are a few ways to Zoom In and Out:

- Press the **Zoom In** or **Zoom Out** buttons in the Main Toolbar.
- Press the `+` or `-` keys on your keyboard.
- Use your mouse's middle scroll wheel, or two-finger dragging if you're using a trackpad.



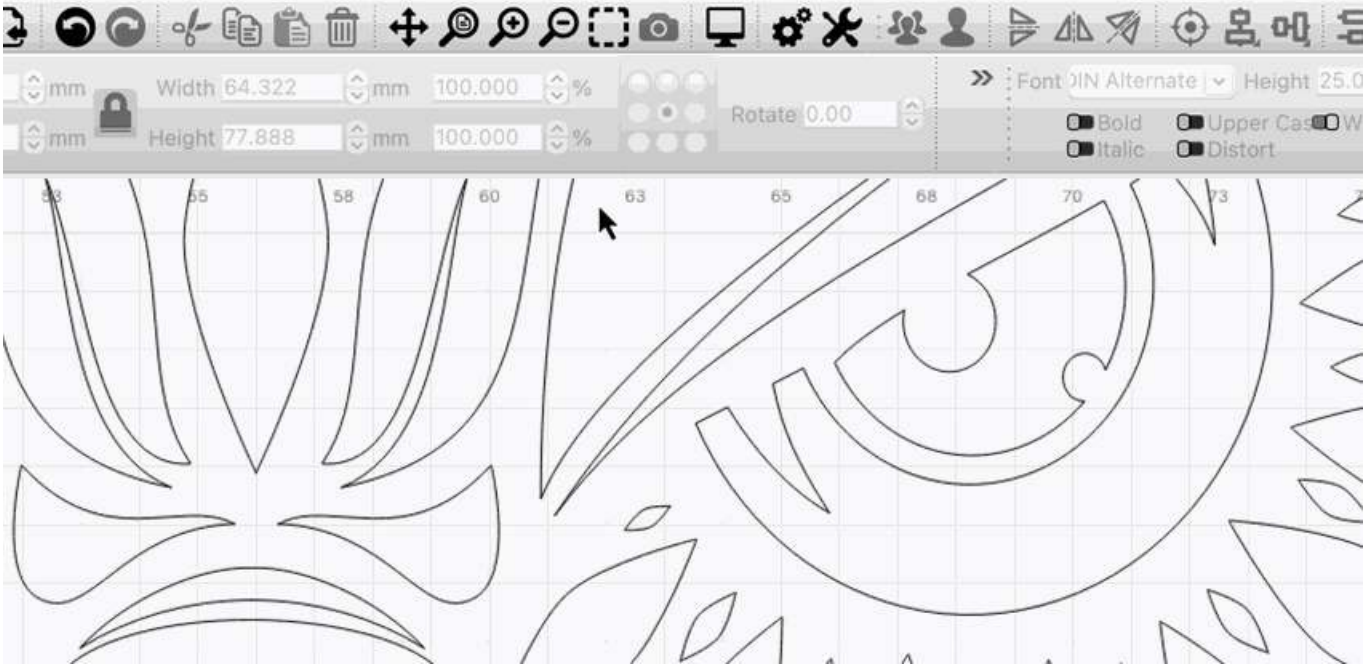
Panning

Panning allows you to move the area your screen is focused over, sliding the field of view between locations in your **Workspace**.

Just as with **Zooming**, there are a few ways to Pan:

- Click the Pan icon in the **Main Toolbar** to enable panning, then click and drag in your Workspace.
- Press and hold your mouse's middle scroll wheel, then move the mouse.
- If you're using a track pad, hold the `Space` key, then click and drag.

When you've enabled Panning mode using any of the above methods, you'll see the mouse cursor change to a hand . Select another tool, or press Esc to exit Panning mode.



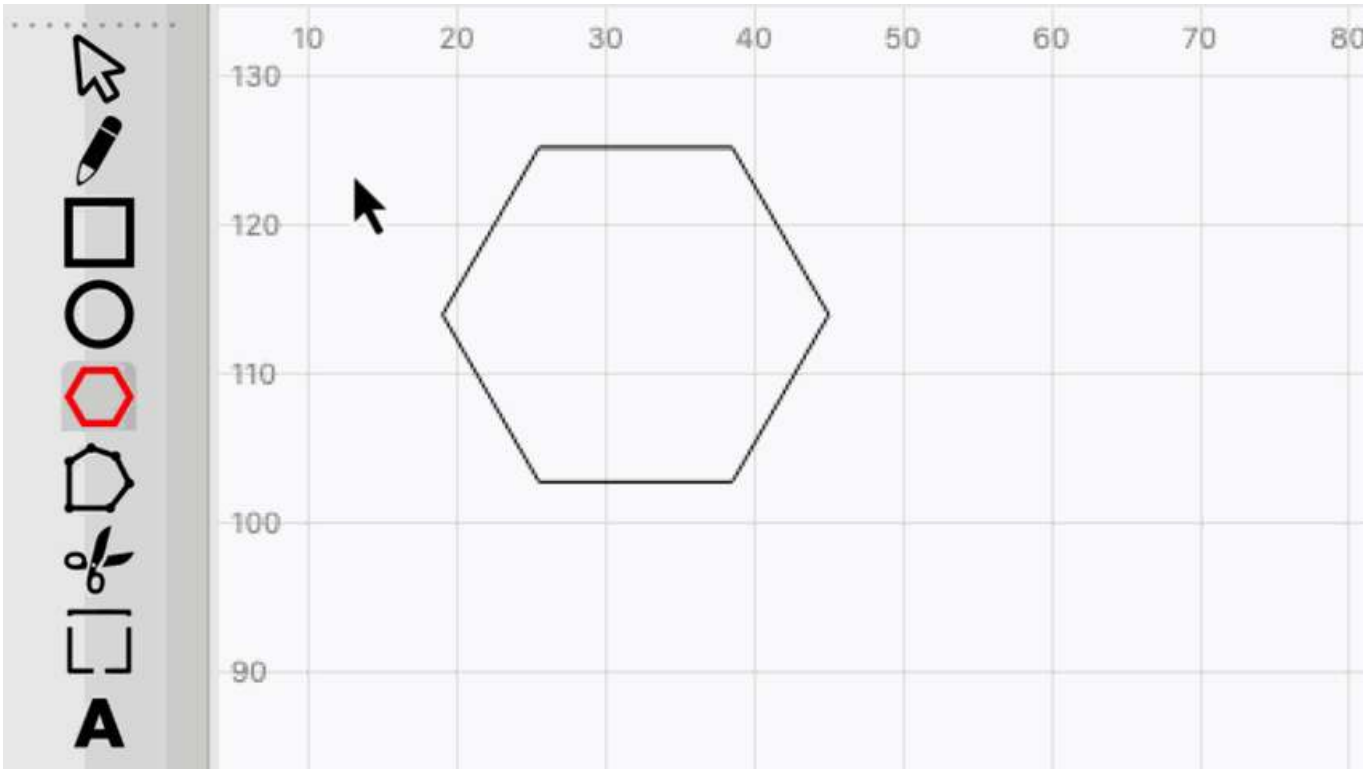
Selecting, Moving, and Sizing Artwork

Now that you know how to get around the Workspace, you're ready to start selecting graphics and making modifications. LightBurn has lots of tools to do that, but we'll focus on a handful of options that you can't get by without.

Selecting

The Selection tool is active by default, but if you've been working with another tool, it will be inactive. To activate it, you can click on the cursor icon in the Creation Toolbar, or press the `Esc` key to close the other tool you were working with.

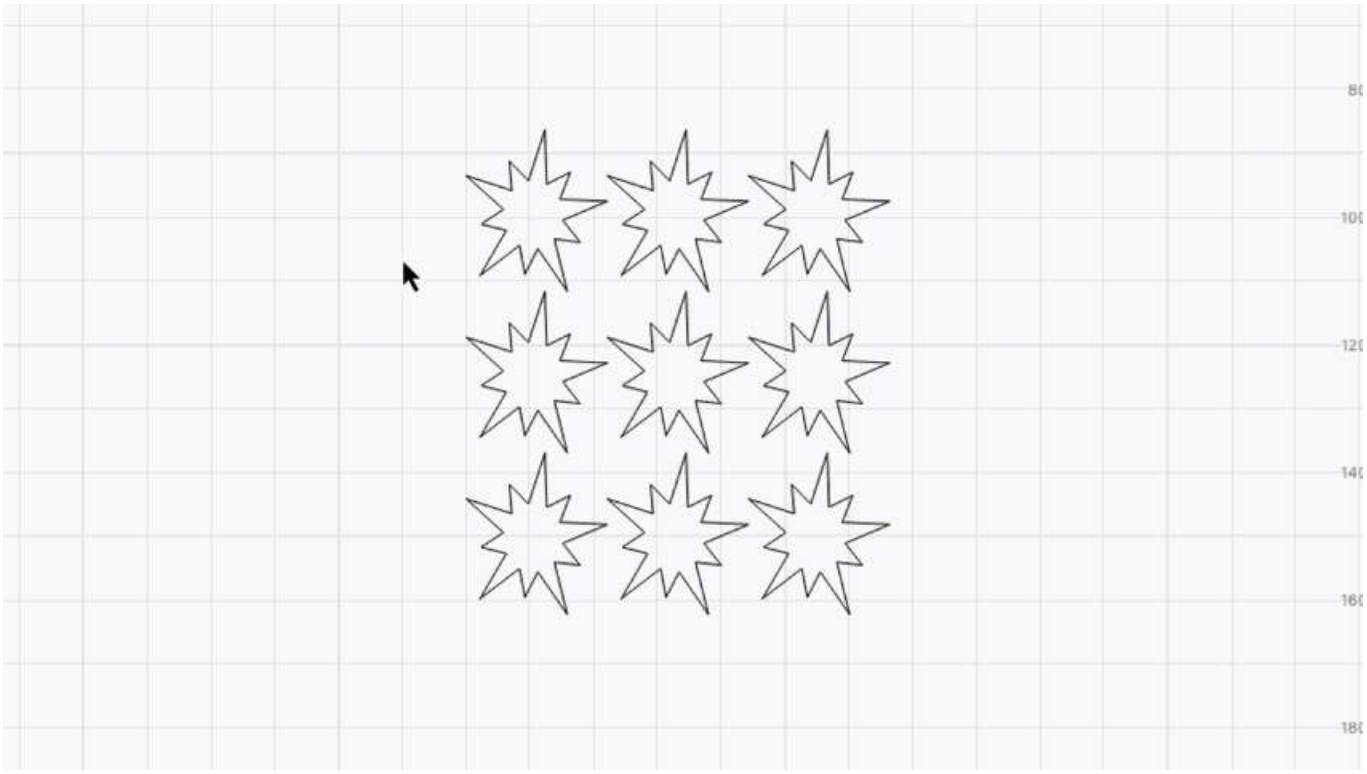
Once it's active, click any object in your **Workspace** to select it.



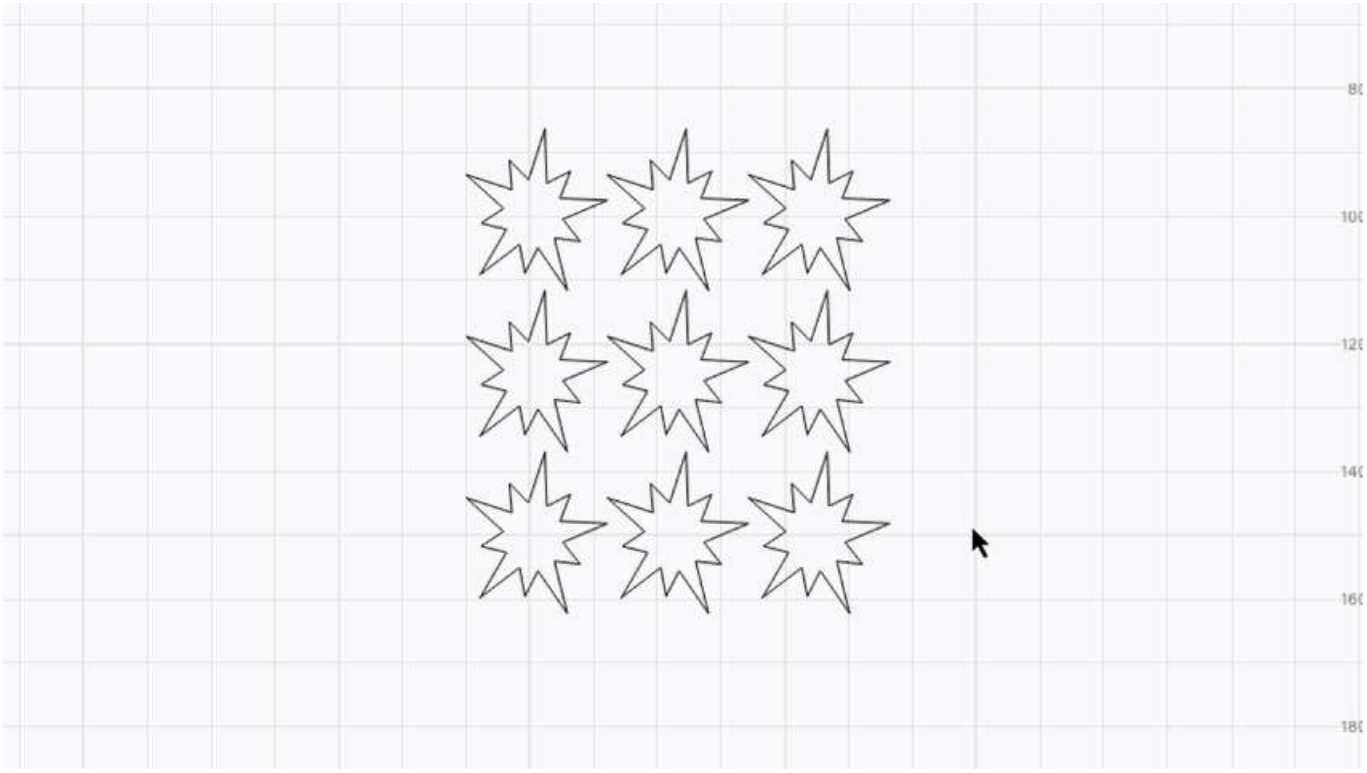
- If you already have an object selected, and want to add another, hold `Shift` while clicking the next object.
- Holding `Ctrl`/`Cmd` will toggle the selection status of any object, meaning that if it's currently selected it will be removed from your selection, and if it's currently not selected it will be added.

You can also click and drag to create boxes that will select objects, or add or remove objects from your selection, when combined with the above modifier keys. There are two types of selection boxes that work slightly differently.

Click, hold, and drag to the right to create a red **Enclosing Selection** box. Release to select all objects that are fully contained within the box.



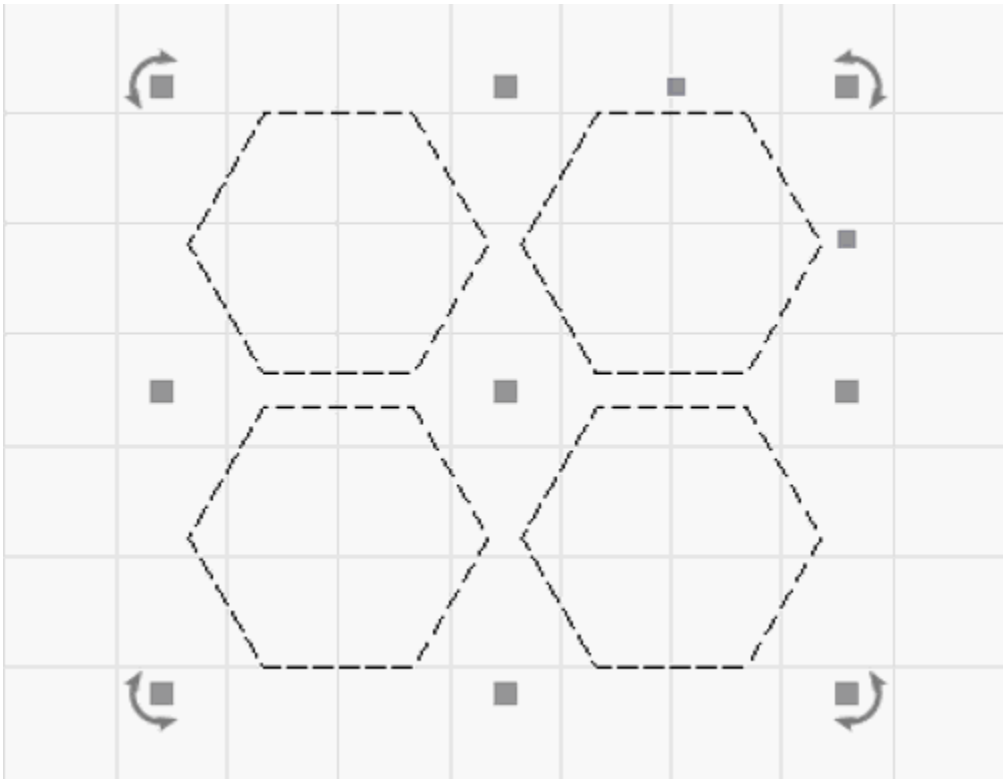
Click, hold, and drag to the left to create a green **Crossing Selection** box. Release to select all objects that are fully contained or crossed by the lines of the box.



Moving and Sizing

Once you've selected objects, you'll notice a few changes to their appearance.

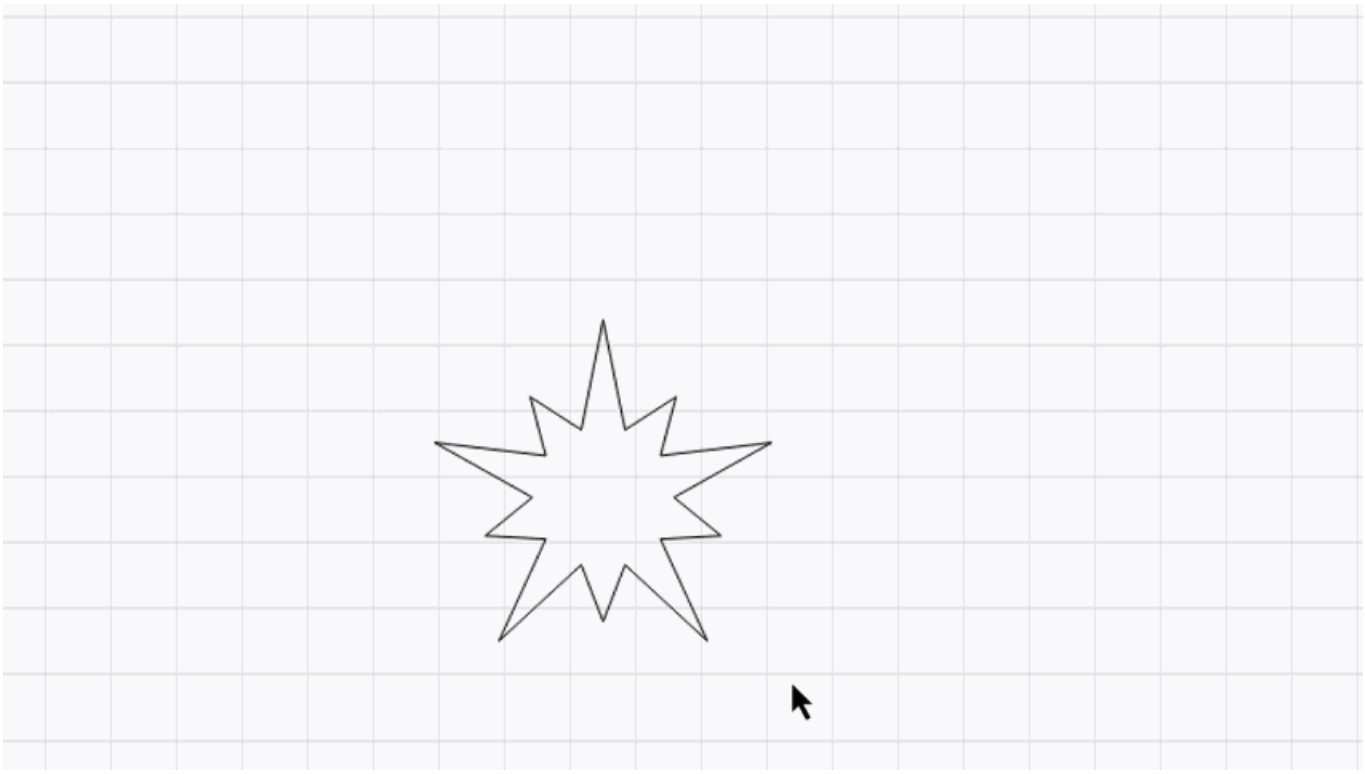
Selected objects are drawn with an animated dash pattern instead of solid lines, and you'll see several gray handles appear on and around the perimeter of the selection area.



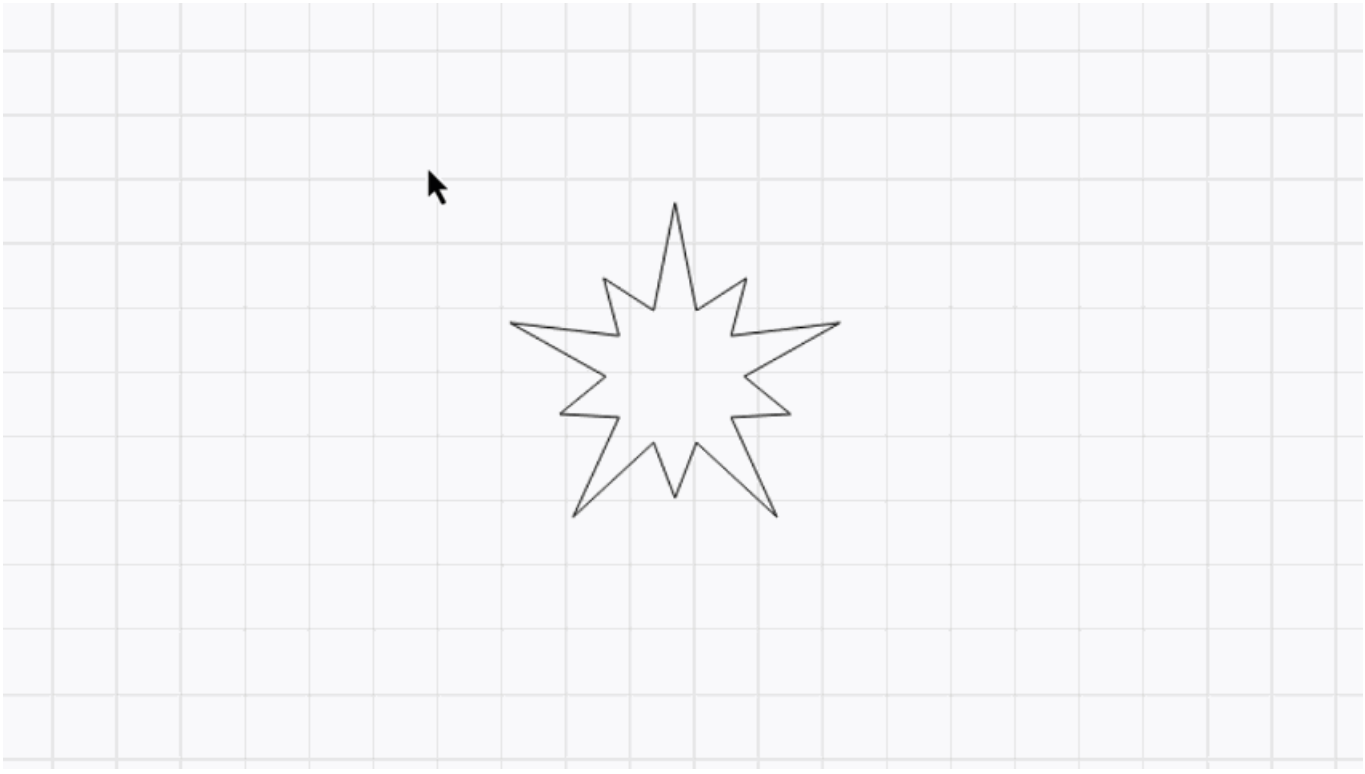
You can use these handles to move, rotate, and resize the selection.

Click, hold, and drag the gray handle in the middle to move an object around.

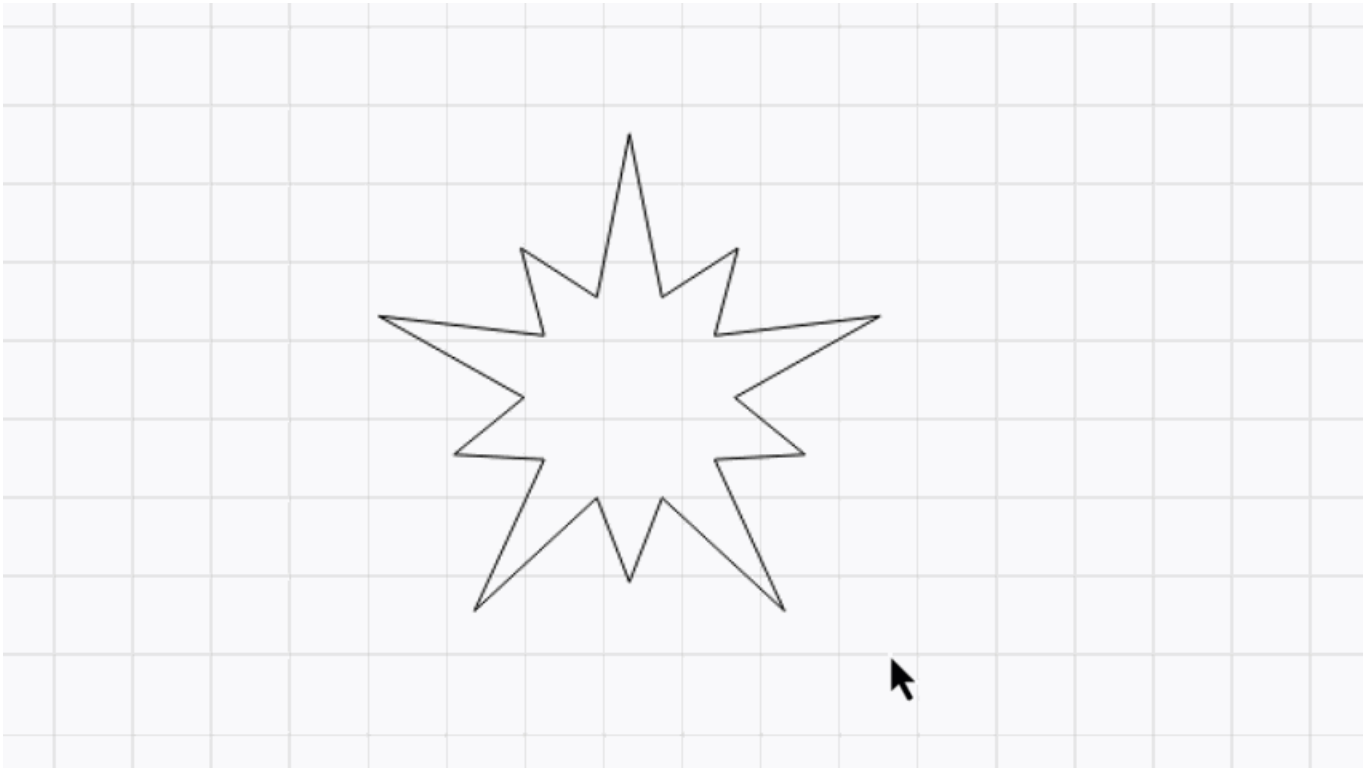
> **Snapping**
>
> You can also click, hold, and drag over any line in your selection to move the selected objects from that location.



Click, hold, and drag any of the handles around the selection's perimeter to resize the selection. Drag outward to make it bigger, or inward to make it smaller — the handle on the opposite side of the selection from the one you clicked will be anchored in place, meaning the selection will be resized in toward, or out and away from, that point.



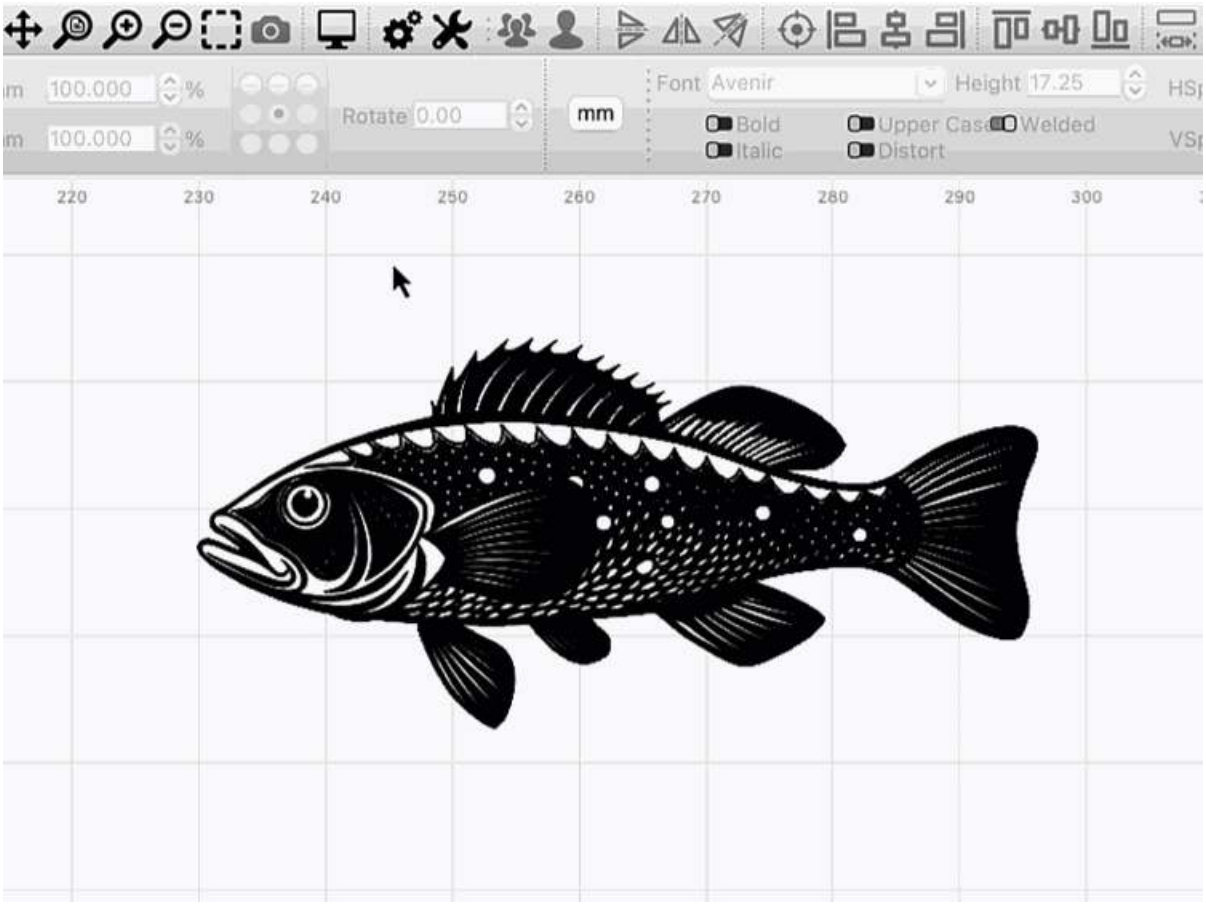
Click, hold, and drag any of the curved double arrow icons around the selection area's corners to rotate the selection.



Grouping

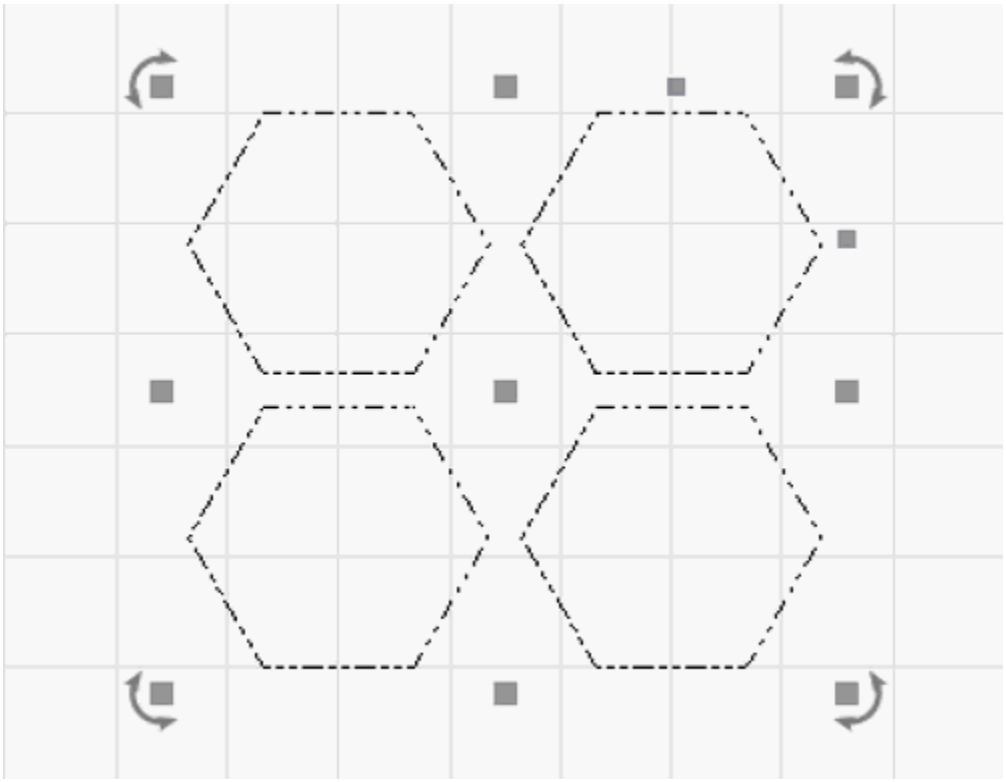
When you move and resize graphics, you'll sometimes want to make sure that certain collections of objects maintain their size and position relative to one another.

You can accomplish that using Grouping, which tells LightBurn to treat several objects as a single unit. To Group objects, first select them, then press the Grouping icon in the Main Toolbar, or `Ctrl`/`Cmd`+`G` on your keyboard.



To treat the objects as individual units again, press the **Ungrouping** icon or `Ctrl`/`Cmd`+`U`.

Grouping objects interferes with the ability to use some of the tools in LightBurn, so if you can't use a given a tool, it could be because you have Grouped objects in your selection. You can tell an object is Grouped when you select it by the animated dash pattern — instead of the normal dash pattern, you'll see a dot-dot-dash pattern.



Colors and Layers

If you've worked with other design software in the past, a lot of what we've covered so far may have been pretty familiar to you — you might even have skipped a few pages. That's ok, we forgive you.

From here we get into the areas where LightBurn functions a bit differently from other design software, due to the nature of the end goal: getting graphics ready to be cut or engraved by a laser.

Colors and Layers

In LightBurn, different colors indicate each layer. Lasers don't print in color, so these colors don't represent the final product's colors but instead differentiate each operation needed to complete the job. Each layer defines a process and a collection of settings that are assigned to objects within the project.

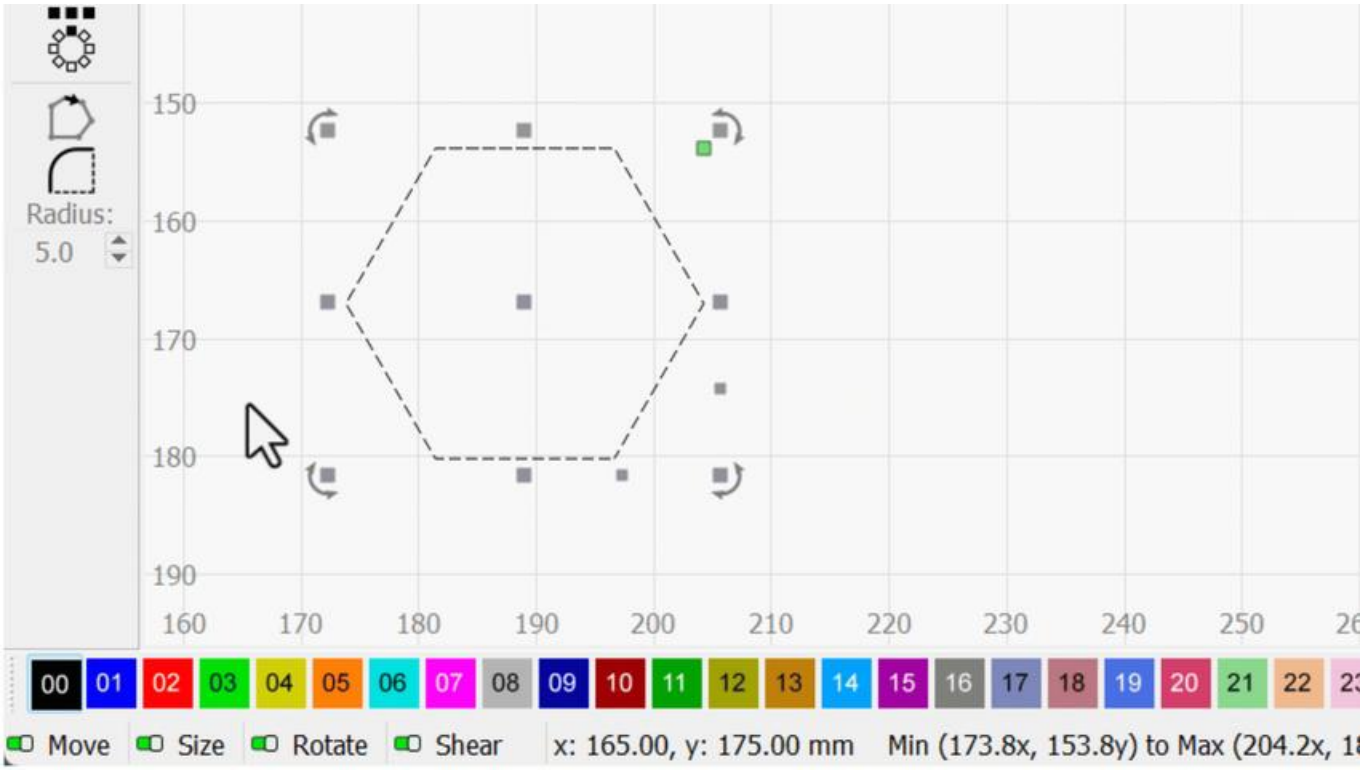
In other words, layers let you tell the machine what it needs to do to produce the final work.

> **Note**
>
> See Images vs. Vectors for more information on the differences between these types of graphics.

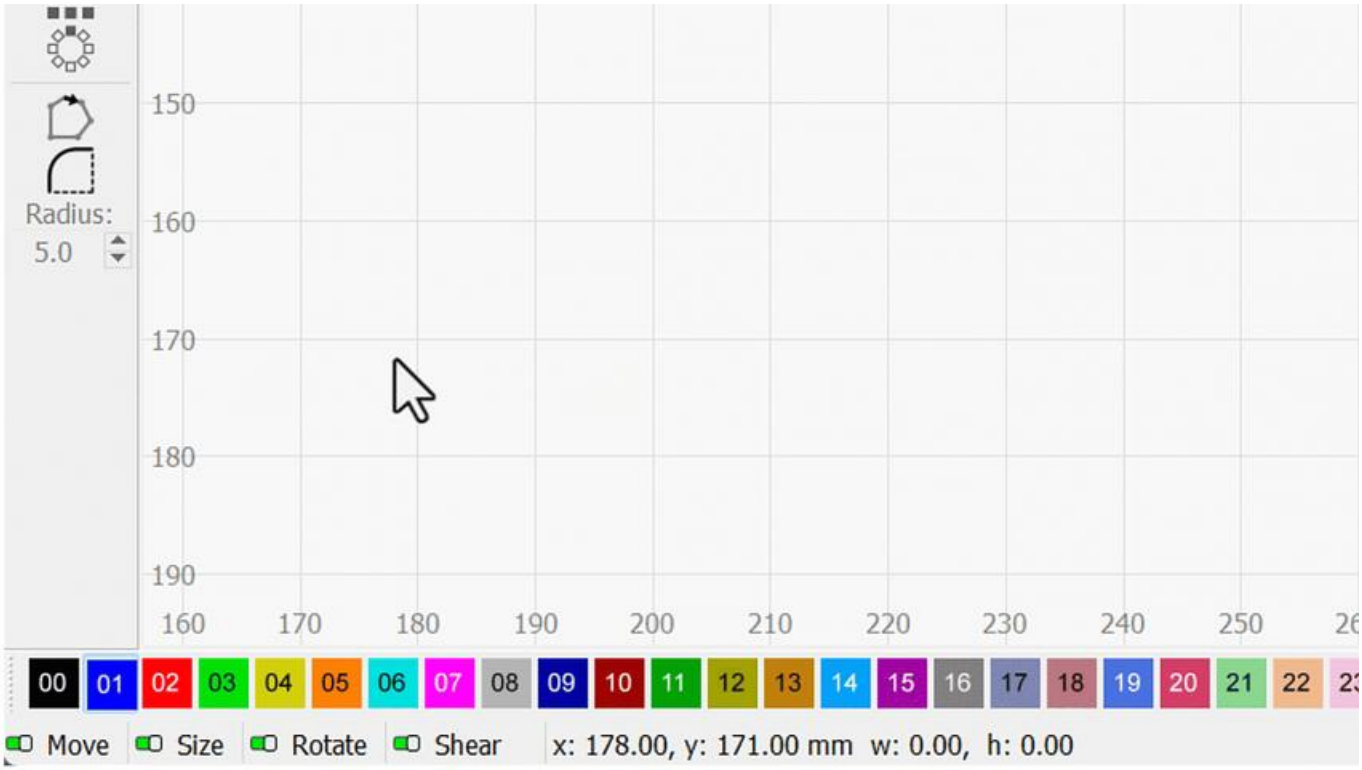
Generally, each color refers to a different layer, with one exception — if you add an image to a vector layer (or vice versa), then the layers will split to show two entries in the Cuts / Layers Window.

In all other cases, all graphics that share the same color/layer will be cut or engraved by your laser with the exact same settings.

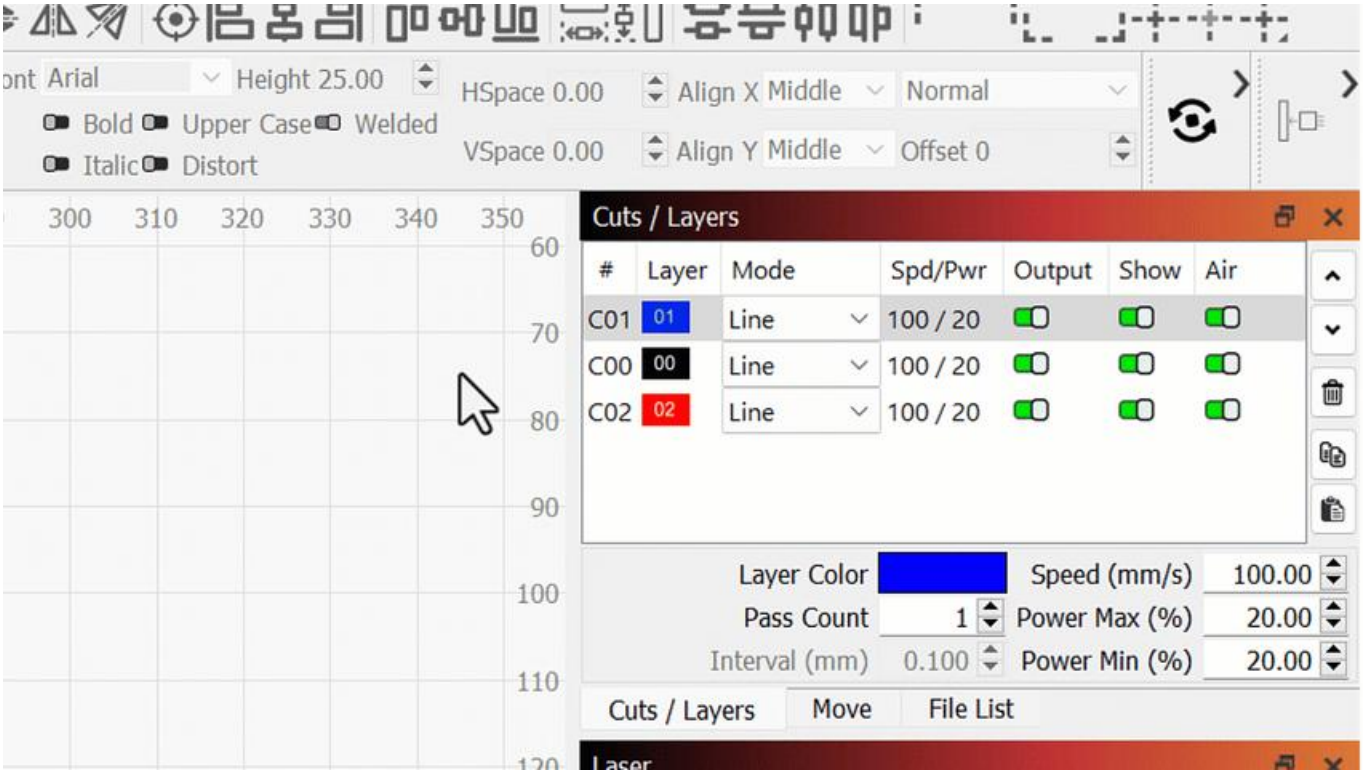
You can use the Color Palette at the bottom of the Main Window to assign colors/layers to graphics in your project. If you already have objects selected, clicking a color will set the selected objects to that color/layer:



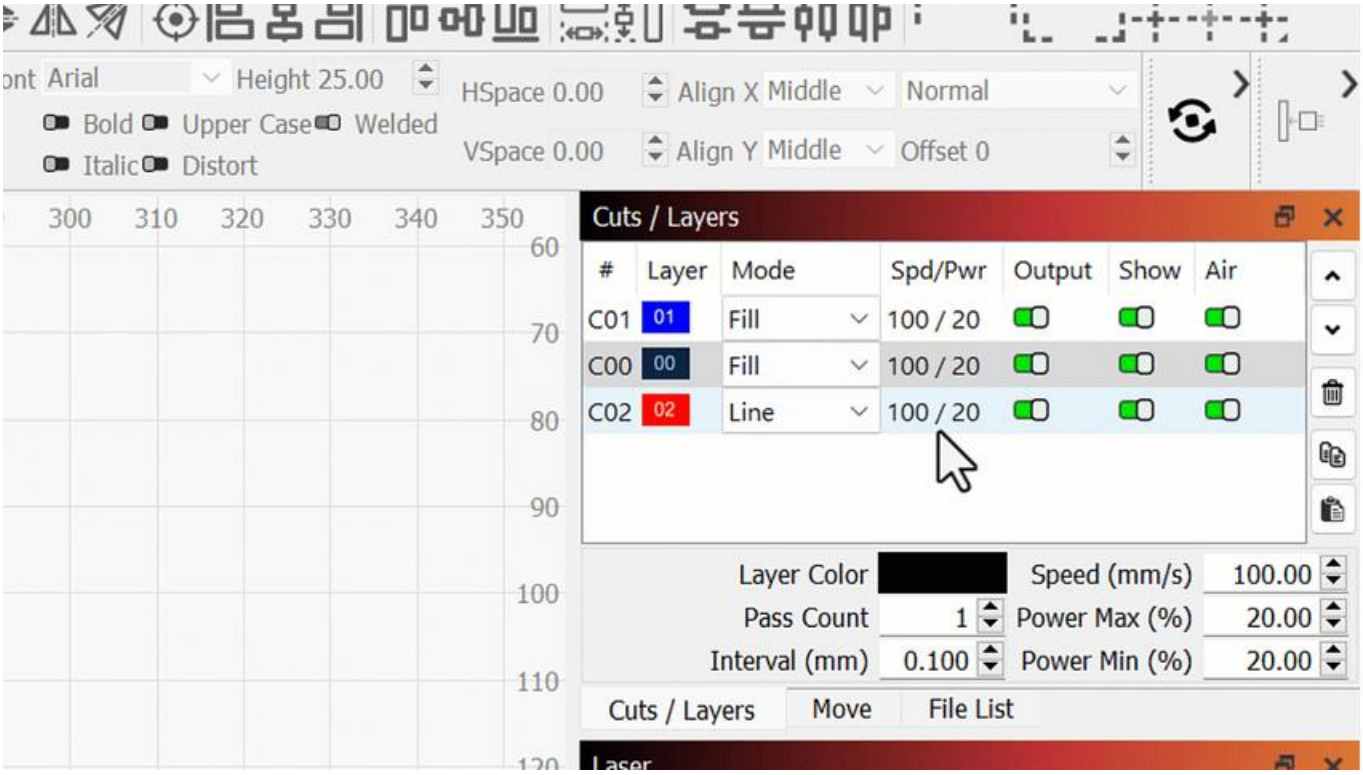
If you don't have any objects selected, clicking a color will determine the color/layer that subsequent new objects you create will be set to:



Once any graphic in your project has been assigned to a color/layer, you'll see that layer appear in the **Cuts / Layers Window**. You can use the drop down menu in the Cut / Layers Window to set the mode of given layer:



By default, operations are performed by your laser according to the order of the layers in the Cuts / Layers Windows. Click and drag a layer or use the arrow button on the side to change the layers' order:



Layer Modes

> **Note**
>
> By default, all vector graphics in LightBurn appear as outlines only, regardless of the type of layer mode they're assigned to. You can change the appearance of graphics set to Fill and Offset Mode by turning on a Filled View Style — we have more information on that below.

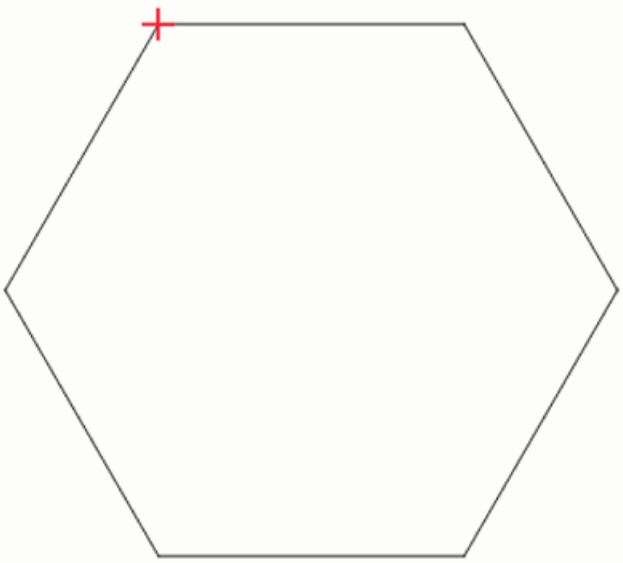
There are four layer modes in LightBurn. You can assign vector graphics — shapes made of lines, curves, and points — to layers set to Line Mode, Fill Mode, or Offset Fill Mode. Image graphics — pictures made up of pixels, or tiny blocks of color in a grid — can only be assigned to Image Mode layers.

Offset Fill Mode is meant for very specific and relatively uncommon use-cases, and Image Mode is a complex topic, best left for after you've gotten the hang of lasering-basics.

We'll focus on the two most commonly-used types of layer modes for the rest of this walkthrough: Line Mode and Fill Mode.

Line Mode

Line Mode tells your laser to trace a path around the contours of vector graphics.

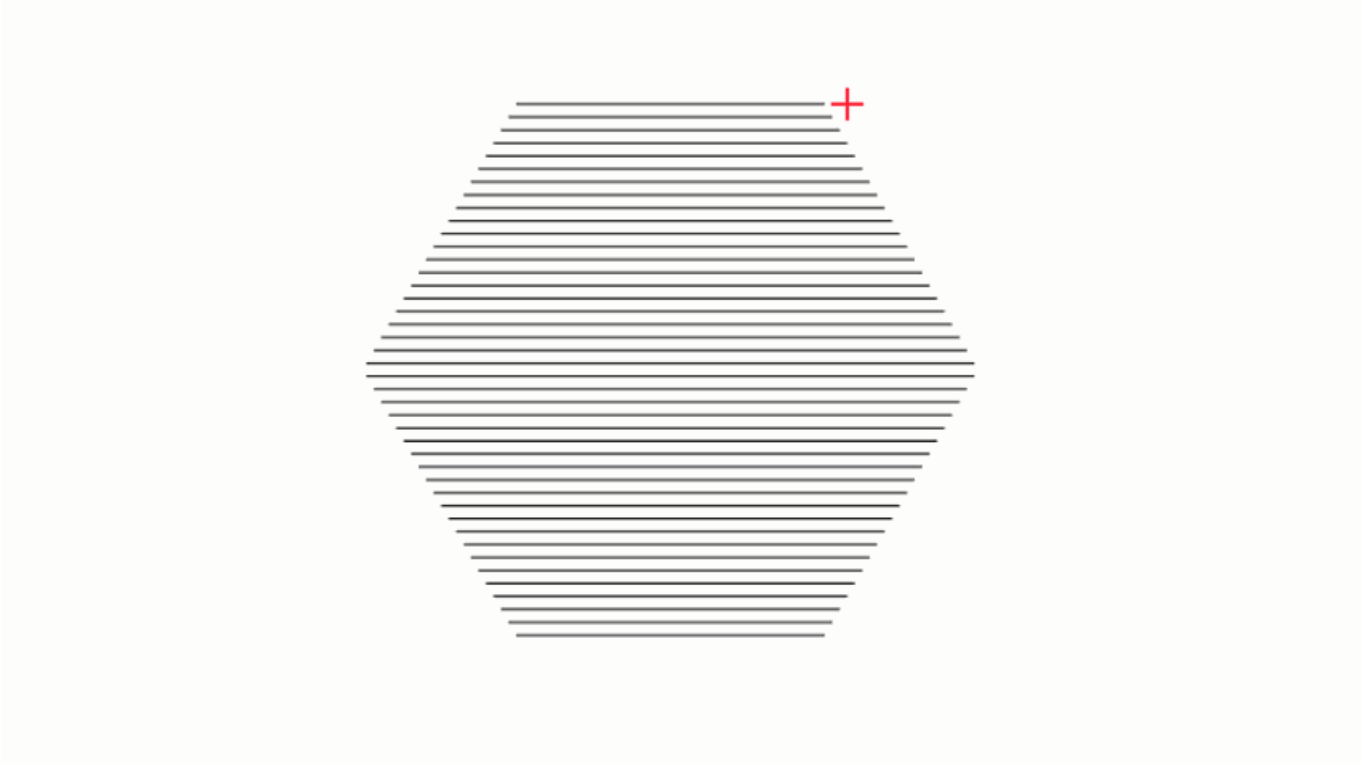


Line Mode is the most straightforward mode for new users to understand, because the output from your laser is the same as the outlines you see in your LightBurn Workspace.

The darkness of a Line Mode engraving or the depth of the cut it achieves depends on your cut settings, which we'll return to on the next page.

Fill Mode

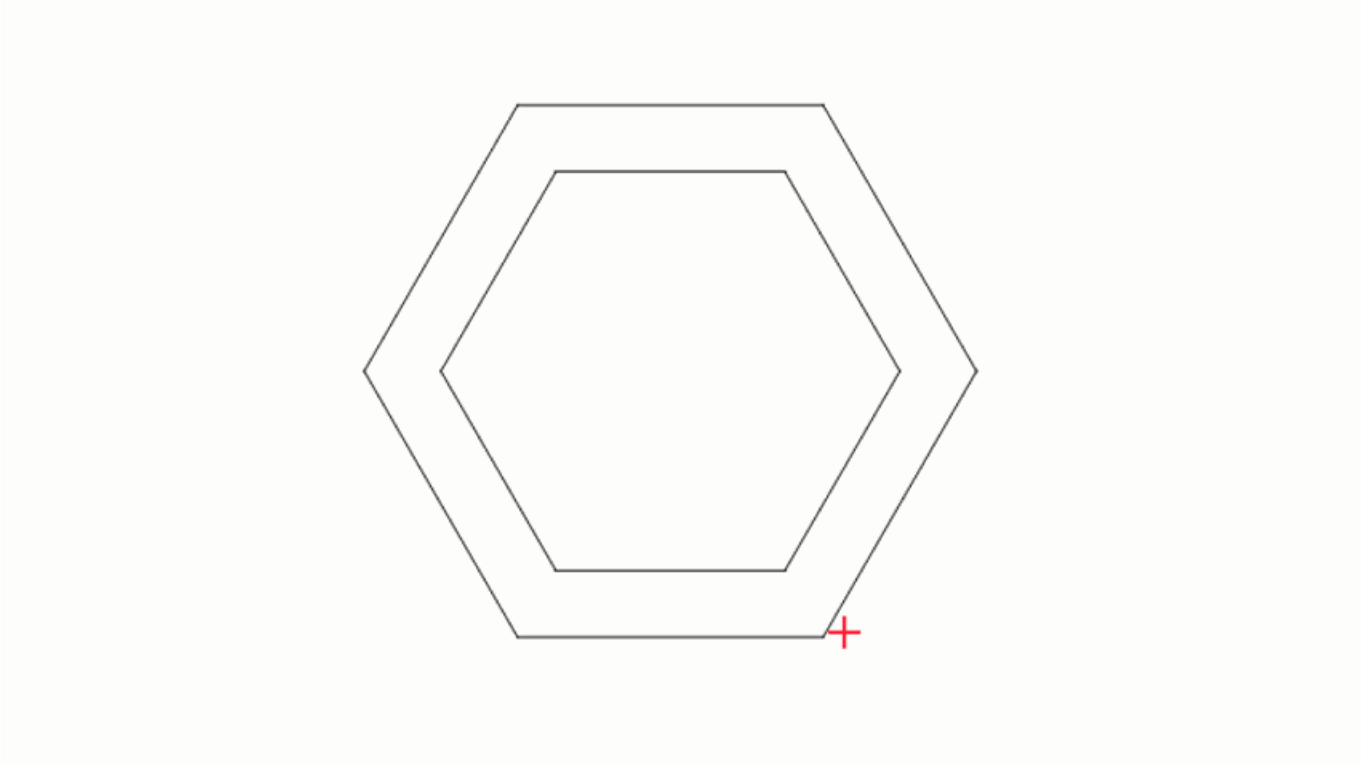
Fill Mode tells your laser to etch parallel lines within the boundaries of vector graphics.



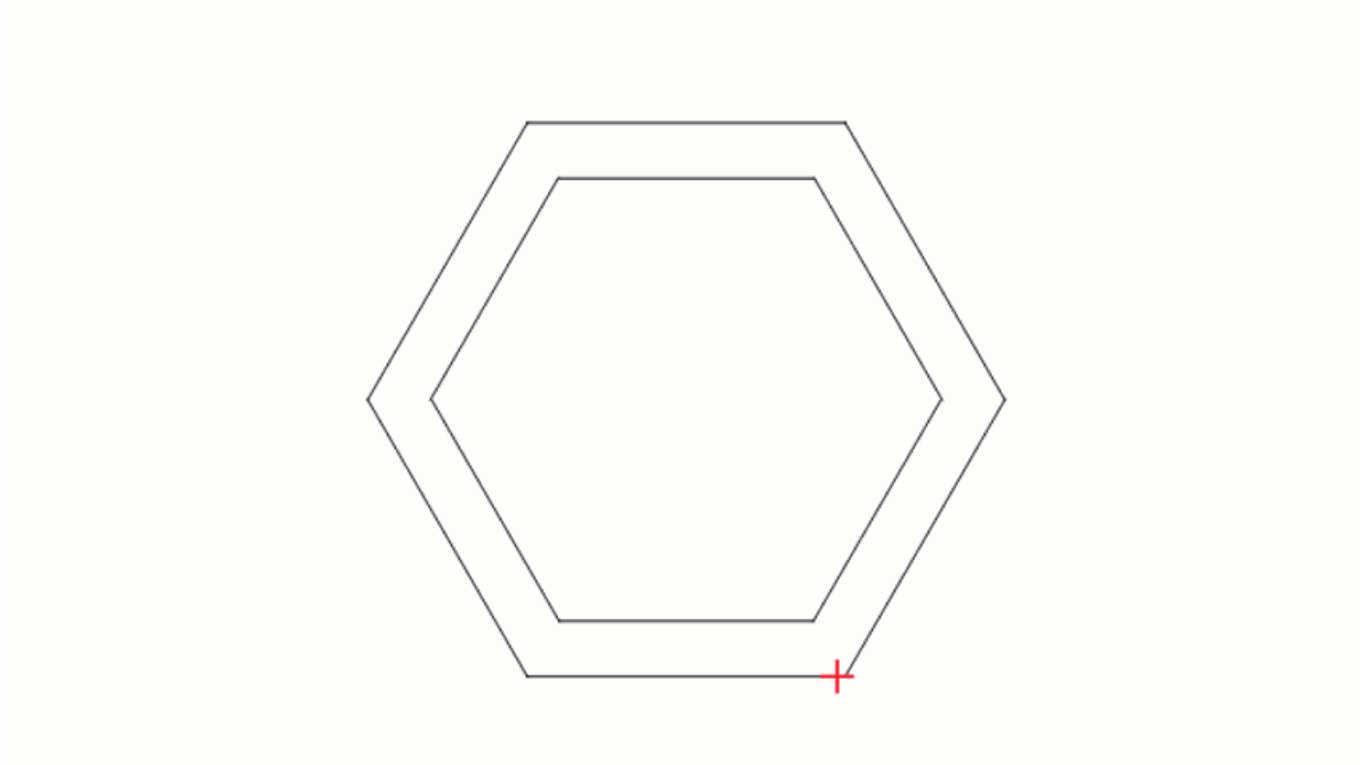
The darkness and depth of a Fill Mode engraving also depend on your cut settings, though not in exactly the same way as **Line Mode**. More on that later.

Fill Mode is a bit more complicated in other ways as well — graphics interact with one another differently depending on whether they are set to the same or different layers.

For instance, if you have two shapes set to the same layer, and one is inside of the other, the area between the two outlines will be filled.

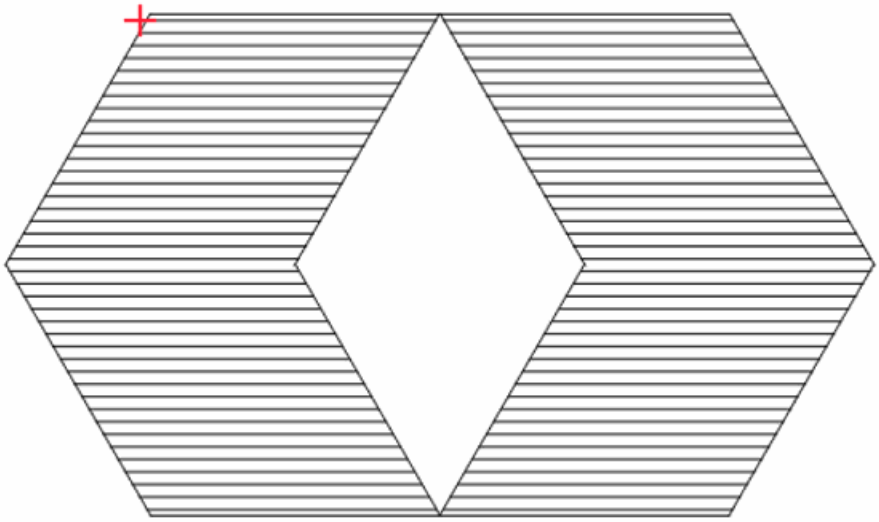


If those same two shapes are set to different layers, the entirety of the outer shape and the entirety of the inner shape will both be engraved, meaning the area within the inner shape will be engraved twice.

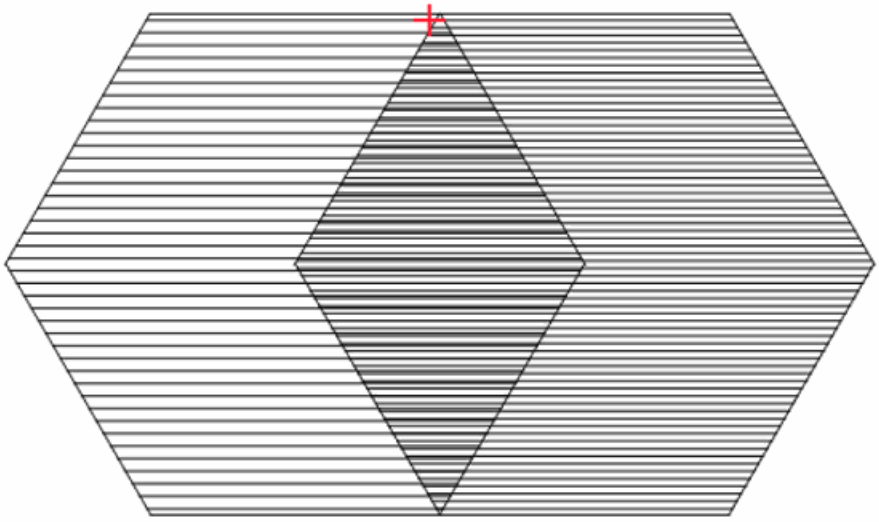


This difference in logic may not be what you're expecting if you come from other design software, where parts of layers can be hidden behind others, and not appear in the final outcome.

The same logic holds true when shapes only partially overlap — if the shapes are assigned to the same Fill Mode layer, the area where they overlap will not be engraved. If they are set to different Fill Mode layers, the area where they overlap will be engraved twice.



Partially overlapping shapes set to the same layer



Partially overlapping shapes set to different layers

In most cases (but not all!) you probably do not want these overlapping areas engraved twice, so always keep this in mind as you're creating and modifying designs.

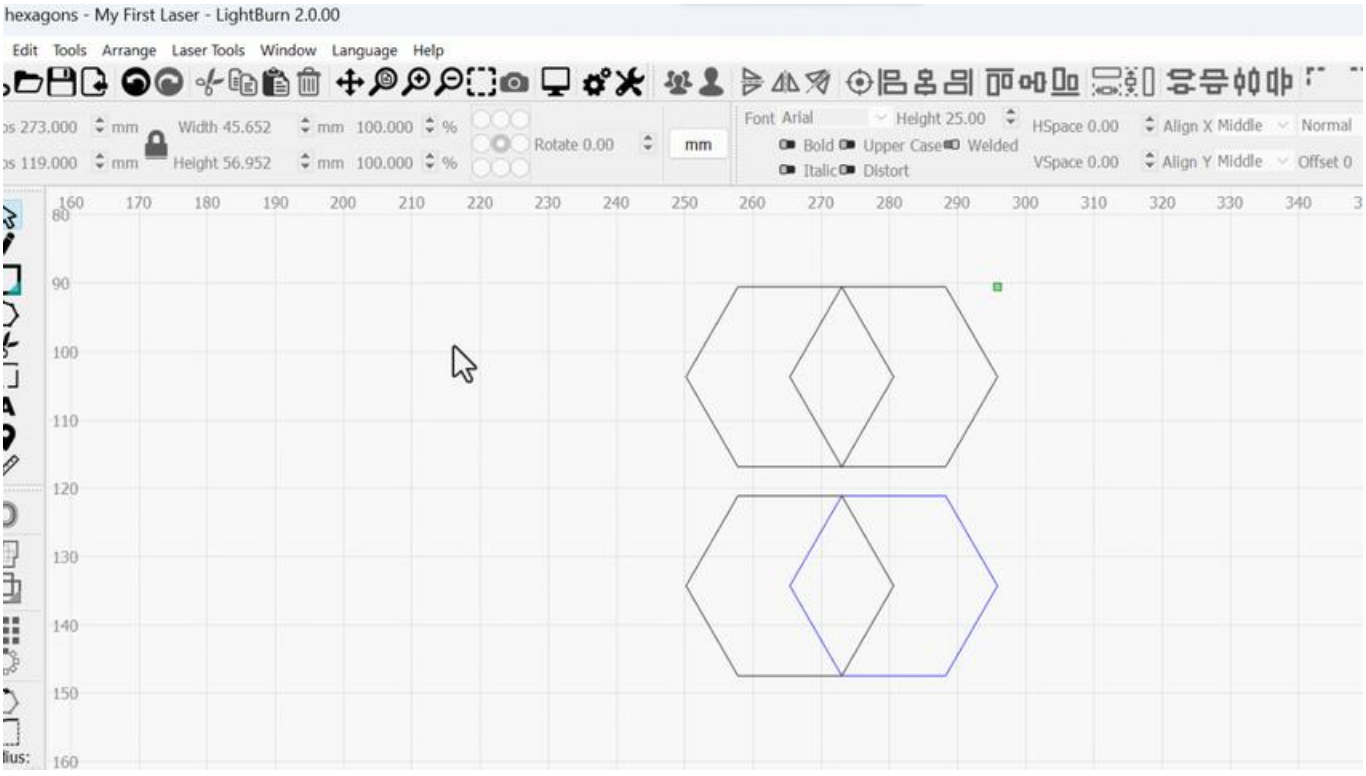
> **Open vs. Closed Shapes**
>
> LightBurn must determine an inside and outside of all shapes in order to know what to tell your laser to fill. That means that all shapes you assign to Fill Mode must be closed, meaning they're made up of complete, continuous loops whose start and end points are the same. When a shape's start and end points are different, it is open.
>
> Several other features in LightBurn require closed shapes in order to work. See Open vs. Closed Shapes for more information.

View Style

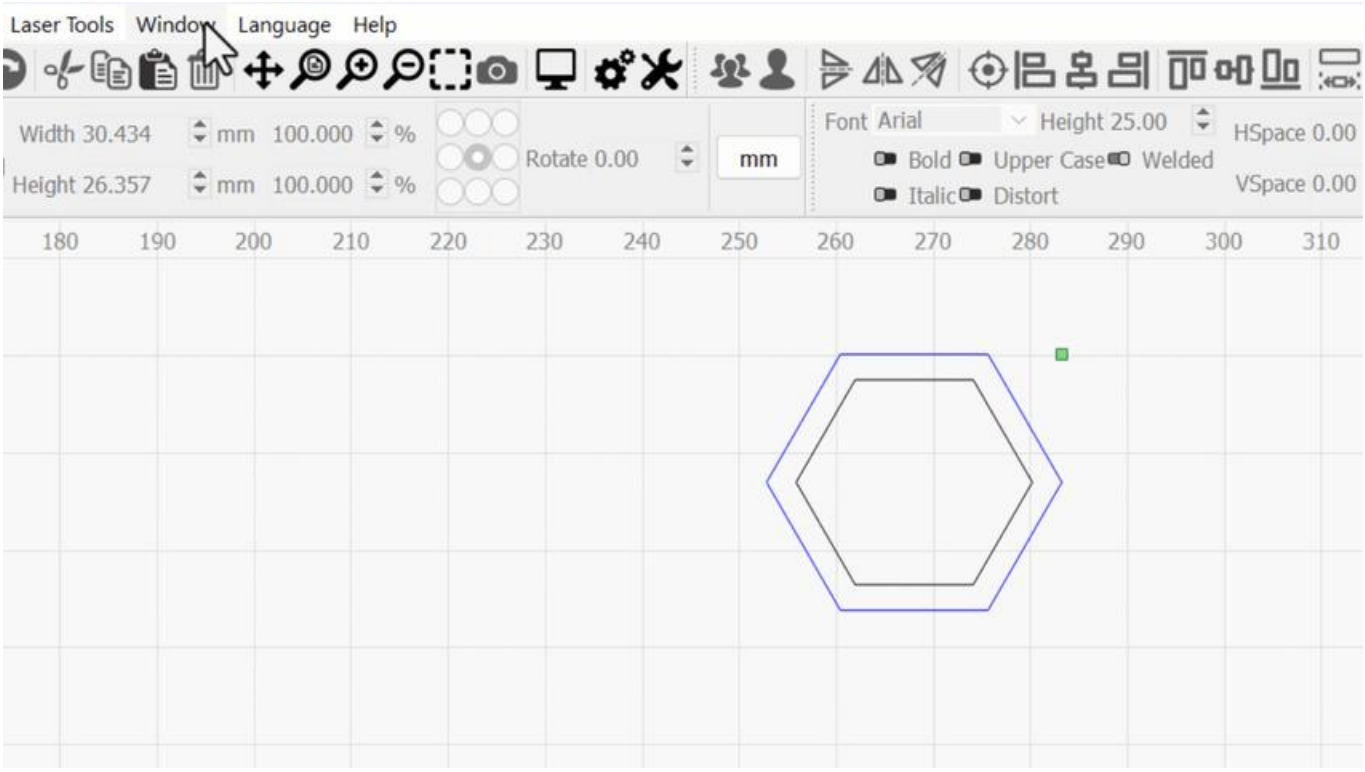
You can change LightBurn's default **View Style** from a **Wireframe** mode to a **Filled** mode to check that your **Fill Mode** layers are arranged properly as you work.

In order for graphics set to a Fill Mode layer to appear filled in your Workspace, go to Window → **Filled / Coarse** (or **Filled / Smooth**), or press `Alt` / `Option` + `Shift` + `W` on your keyboard.

Select **Wireframe / Coarse** or **Wireframe / Smooth** from the same menu, or use the same keyboard shortcut again, to switch back to Wireframe mode.



There is a drawback to using a Filled View Style — while it's helpful for checking on the relationship of graphics set to the same Fill Mode layer, if your graphics are set to different layers, they can end up hidden behind one another — but they'll still be sent to your laser.



For that reason, it's best never to rely on the appearance of graphics in your Workspace alone, and always check the Preview window. We used the Preview window to make many of the examples shown above, and we'll explain how to use it in more detail later on.

Cut Settings

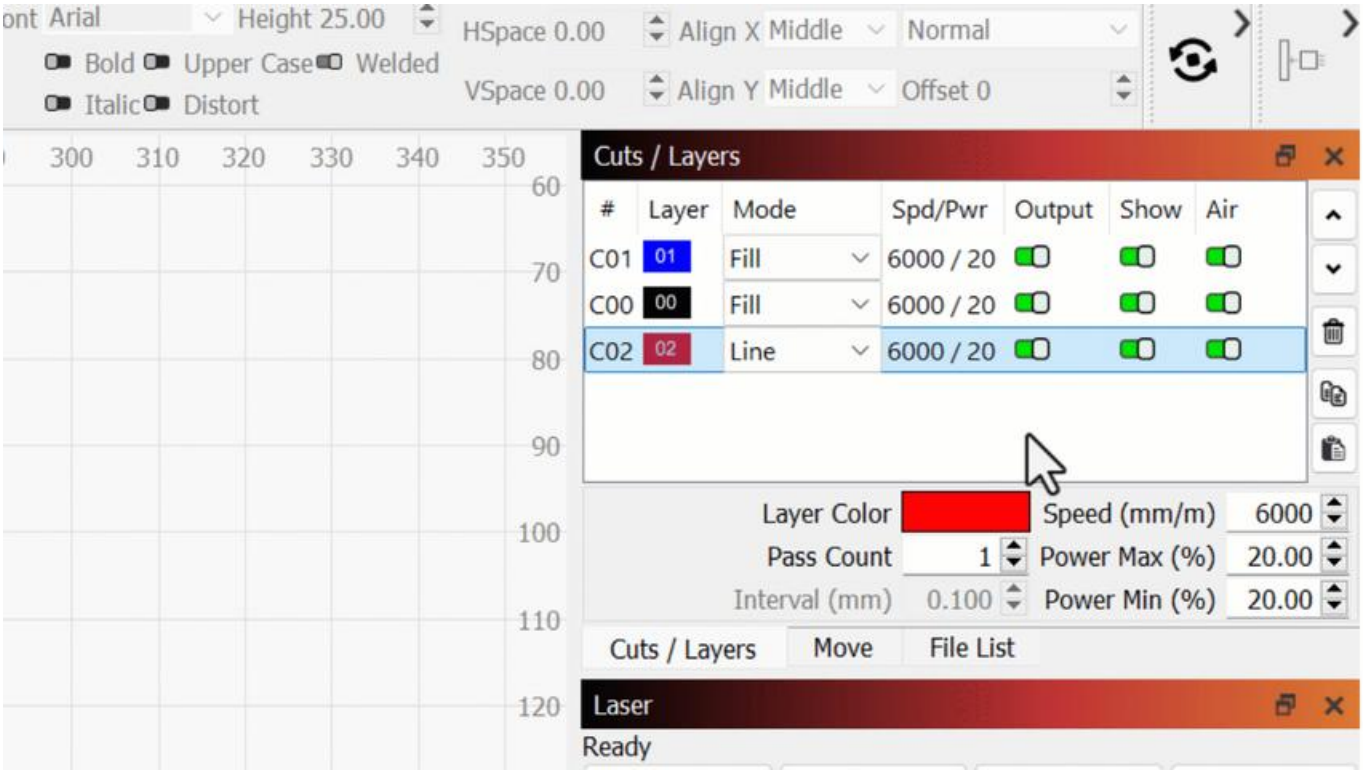
Cut settings are a big topic, and we need to begin with an important caveat: the exact settings required to achieve good engravings, or to cut through material, are highly dependent on your type of laser.

We'll provide some rough guidelines on the relationships between different settings, but the specific settings shown in our examples are not necessarily appropriate for your machine and material. Consult your laser's manufacturer for recommendations directly tailored to your machine.

> **Always pay attention to units of distance and time when entering Speed settings**
>
> When entering **Speed** values recommended by your laser's manufacturer or other LightBurn users, make sure to use the same units of distance and time as those from the recommendation, or to convert the values to your preferred units.
>
> A given number of **millimeters per second** is much faster than that same number in **millimeters per minute**. Mixing up units can lead to reduction in power output due to unexpectedly high speeds, or excessive power output — and even fire — due to unexpectedly slow speeds.
>
> Change your Control Units in the **Dimensions / Units** tab of the **Device Settings** window. LightBurn automatically converts any existing values when you switch between units.

Essential Settings

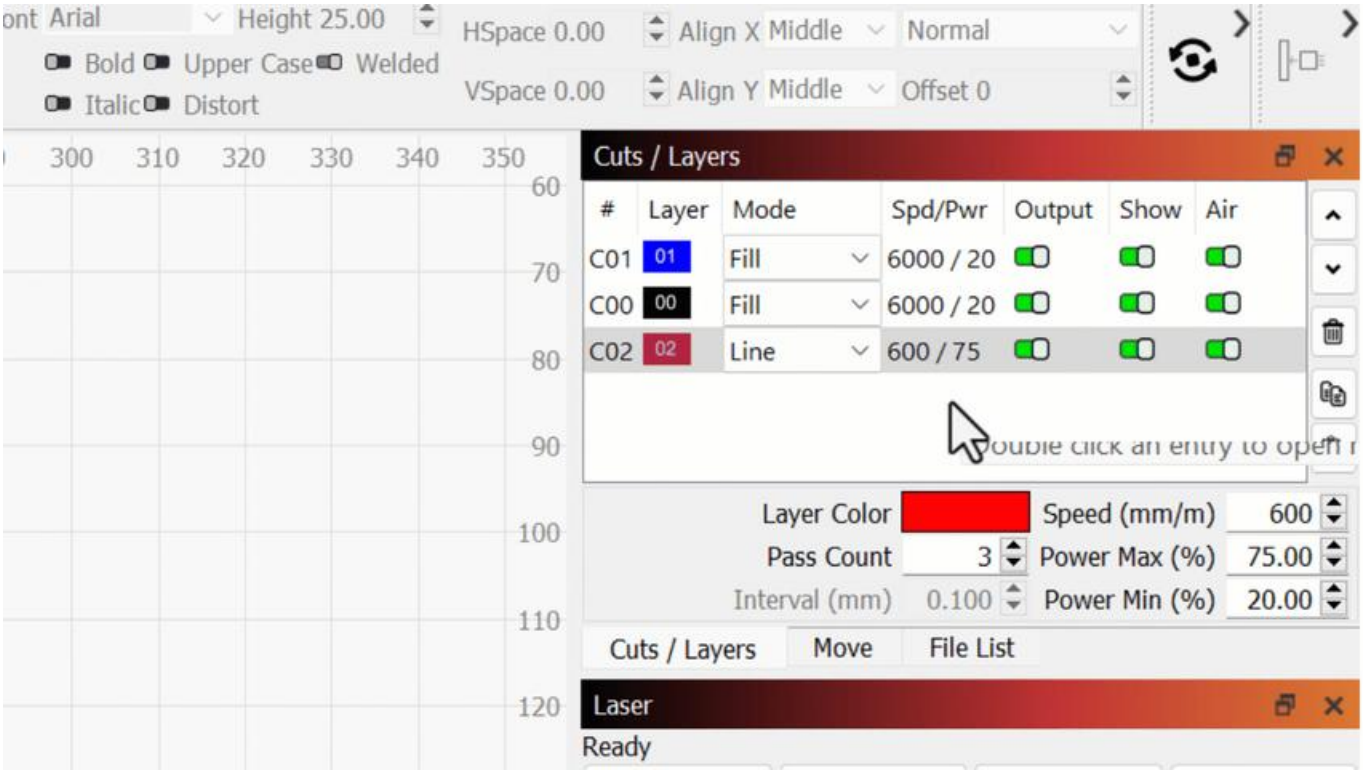
The most important settings for layers set to Line or Fill Mode are accessible directly in the Cuts / Layers Window. Click in any of the fields to adjust a given setting:



Tip

Double-click in a field to select and overwrite current settings.

You can also double-click any layer in the Cuts / Layers Window to access the Cut Settings Editor. You'll see the same essential settings that are available in the Cuts / Layers Window in the Cut Settings Editor, along with several more advanced settings. For now you can ignore the more advanced settings — we just want you to know they're there for down the road.



Output, Show, and Air

You'll see three switches next to each layer in the **Cuts / Layers Window** — turning them on or off changes the behavior of those layers in different ways.

- The **Output** switch controls whether a given layer is sent to your laser when you start a job. In other words, if you turn Output off, none of the graphics on that layer will be cut or engraved by your laser.
- Disabling **Show** hides all the shapes assigned to that layer in your **Workspace**. Hiding shapes like this is sometimes helpful while you're designing, but be careful, hiding a layer isn't the same as turning off the Output switch — it'll still be sent to your laser when you start a job.
- Enabling the **Air** switch tells LightBurn to signal your laser to turn on an air assist, when it begins cutting or engraving a given layer. Disabling it tells LightBurn not to send the signal for that layer.

  Air assists blow air onto the surface of your material at the spot where the laser's beam strikes, which can help prevent soot accumulation and charring, and aid in cutting.

  > **Note**
  >
  > Many lasers do not come with air assists, and others have manual air assists that you can't trigger through software. In those cases, toggling the Air switch will have no effect either way. If you have a Galvo laser, you won't see the Air switch at all.
  >
  > Consult your laser's manufacturer if you're not sure if your laser has a built-in air assist.

Line Mode

For **Line Mode**, when you're just starting out you only need to worry about a few settings: **Speed**, **Power**, and **Passes**. As a general guideline, relatively slow speeds and relatively high powers produce the darkest burns.

If you command the laser to move quickly, or with low Power, it will likely just etch the surface (sometimes called scoring). If you move slowly and with high power, especially with a CO_2 laser, the laser will cut deeper, or even completely through the material. The only difference between surface marking and cutting with Line Mode is the Power and Speed you set.

If you have a high-powered laser, it should be able to cut through many types of materials with a single Pass, but if you have a relatively low-powered laser, you may need to set multiple Passes to achieve a clean cut. The exact number depends on your machine and material.

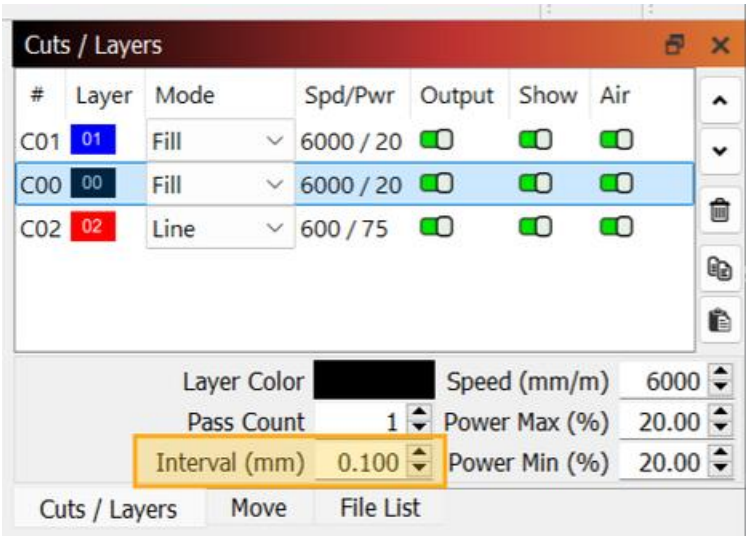
> **Note**
>
> Not all lasers are capable of cutting through all or even any materials. Consult your laser's manufacturer for information on your laser's maximum cutting capability.

Fill Mode

For **Fill Mode**, the darkness and depth of an engraving is also determined by the **Speed** and **Power** settings, and follows the same principle: relatively slow speeds and high powers tend to produce the darkest, deepest etches.

But there's another crucial setting for layers set to Fill Mode — **Line Interval**. The Interval setting determines the distance between the parallel lines your laser scans as it engraves your graphics. Smaller Line Intervals lead to lines that are closer together. As a rule, you want the scan lines to be as close together as possible without overlapping (which can lead to overburning).

The necessary Line Interval for optimal results depends on your laser and the material you're engraving, but LightBurn's default value of **.100** is a reasonable starting point for most lasers and materials.



Material Testing

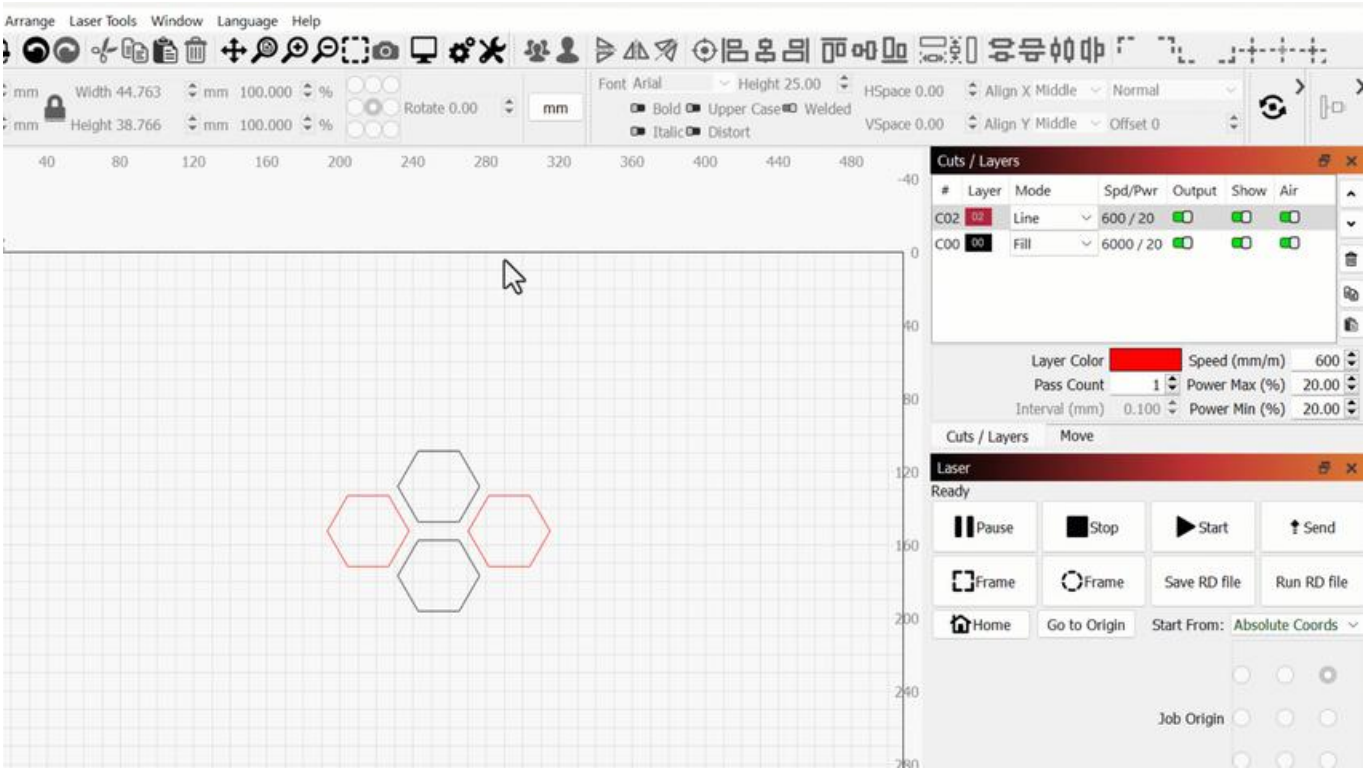
While we've noted that manufacturer recommendations are the best place to begin looking for cut settings, LightBurn also has a built-in Material Test that you can use to test different combinations of **Speed**, **Power**, **Passes**, **Line Interval**, and other settings. The Material Test works by automatically setting up a grid of boxes that will be cut or engraved at a range of different settings.

For more information, check out our guide on completing your First Material Test before starting the First Project at the end of this walkthrough.

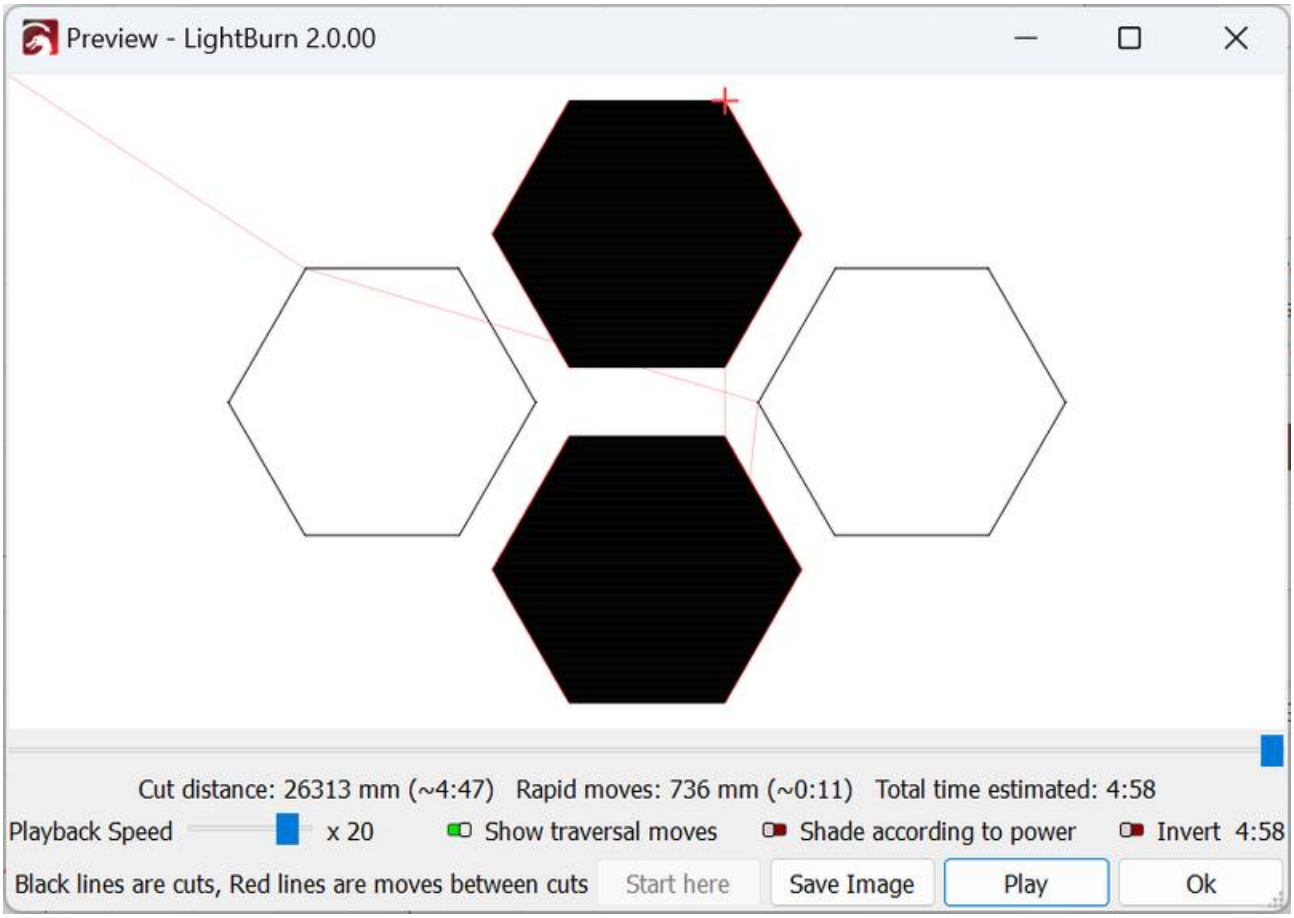
Previewing

Because it can be difficult even for experienced users to keep track of which layers are set to which mode, and whether all of their cut settings are in order, we strongly recommend previewing every project, before sending it to your laser.

To open the Preview window, go to Window → **Preview**, click the Preview icon in the Main Toolbar, or press `Alt`/`Option`+`P`.



Understanding the Preview Window



When you first open the **Preview** window, you'll see a still image of the final result of your project — the contours to be scored or cut by Line Mode will be indicated by black lines, and the areas to be etched or engraved by Fill Mode will be shown as solid black areas, or, if your **Line Interval** is relatively large (or you zoom in close enough) as parallel lines running across the filled shapes.

You'll also see red lines — these lines indicate the path your laser will travel while not firing. You can toggle **Show traversal moves** at the bottom of the Preview window to switch between showing these moves or not.

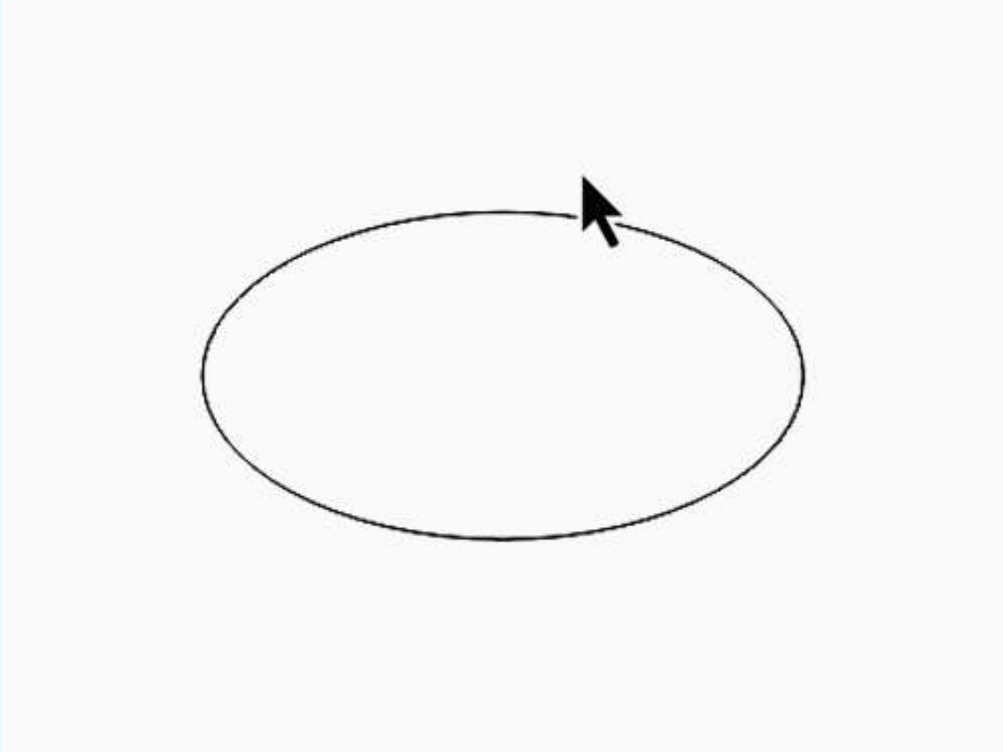
If what you see here doesn't align with your expectations, you've got some editing to do. Make sure all of your shapes are set to the correct layers, and those layers are set to the correct modes. If you're working with layers set to Fill Mode, make sure they're overlapping only as you intend.

Click here for additional tips on fixing issues you identify in the Preview window

Troubleshooting

- If you have duplicate copies of graphics directly on top of each other, there are two types of behavior you might see in the Preview window:
  - If they're both set to Line Mode, you'll see the same contour being cut twice (or more, if you have multiple Passes set for the layer they're on).

- If they're both set to Fill Mode, neither will appear in the Preview window, or be sent to your laser.
- Go to Edit → Delete Duplicates or press `Alt`/`Option`+`D` to delete duplicate shapes.



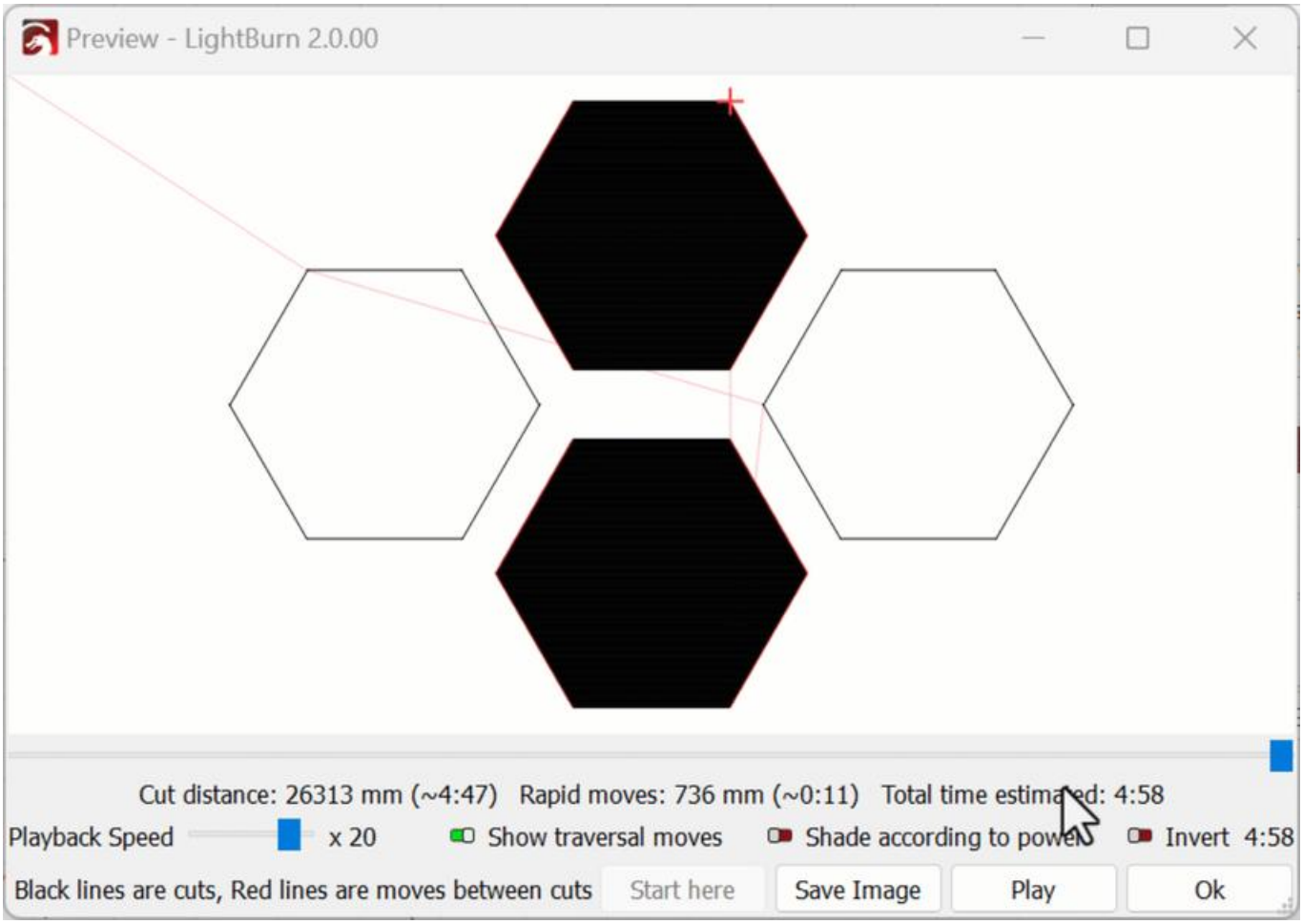
- If you see several strange things and can't identify the cause, double-click the problematic layer to open up the Cut Settings Editor, click Reset to Default in the lower left corner, then re-apply just the essential settings such as Speed and Power.



- See Preview Troubleshooting for help identifying other potential issues in the Preview window.

Simulation

Press **Play** to see a simulation of the movements and lasering operations your laser will perform when you start the job. In addition to making sure the laser will outline or engrave the correct graphics and areas, you'll be able to make sure it performs those actions in the order you want.



By default, the simulation will play in an approximation of the job's real runtime, and **Total time estimated** is shown in the bottom right. You can adjust the **Playback Speed** slider to speed up or slow down the simulation.

If you don't like the order the operations are performed, change their position in the Cuts / Layers Window. As a reminder — by default, the first layer in the Cuts / Layers Window is performed first, followed by the second, and so on.

> **Note**
>
> More advanced methods of controlling the order in which your graphics are cut or engraved are available in Optimization Settings — but we don't recommend adjusting those until you've mastered the basics.

Coordinates and Job Origin

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=aphiLTUWcjE

This next section covers a fundamental concept in the world of lasering. Your laser has a work area — the bounds within which it can cut or engrave graphics — so how do you tell it exactly where to do that for any given project?

The answer depends on your type of laser, personal preferences, and needs. We'll run through your choices, and explain how they work, below.

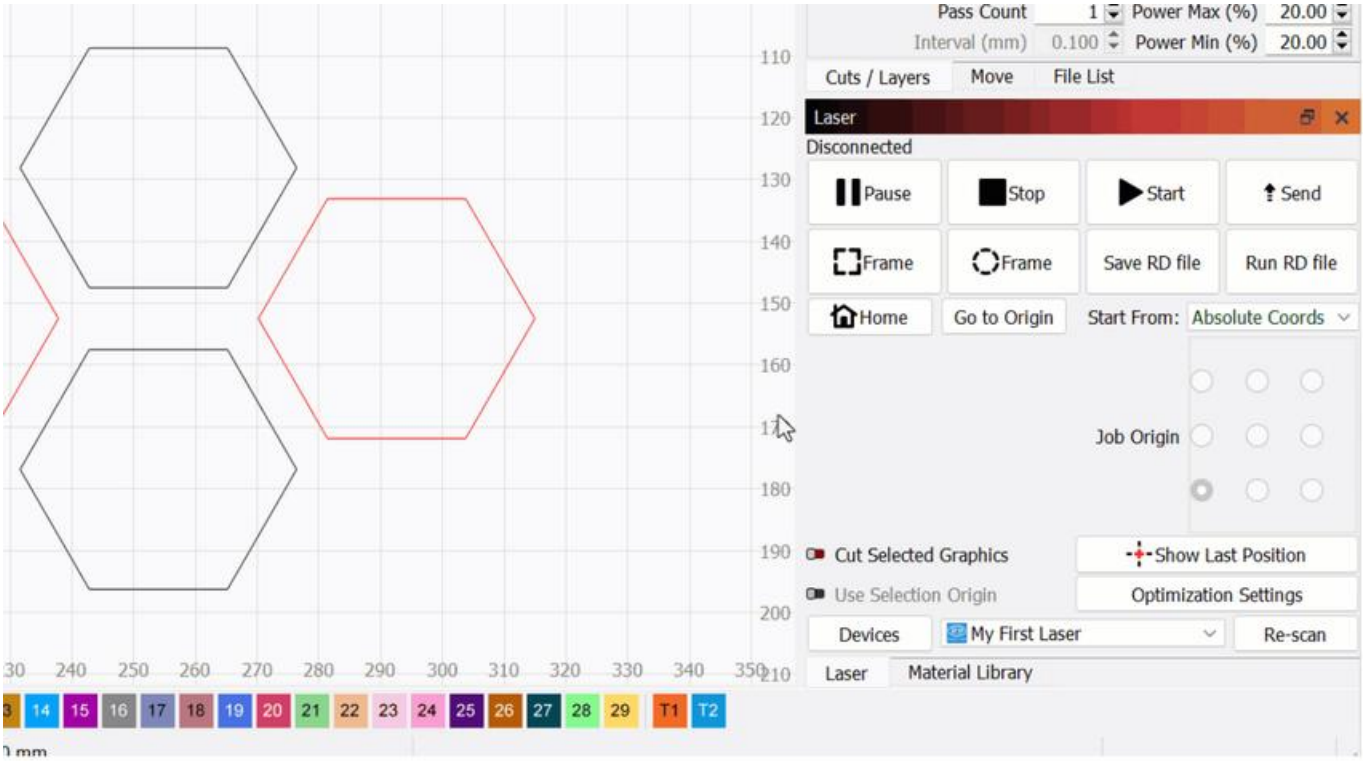
Selecting a Start From Mode

There are three options for determining where graphics will output in your laser's work area.

- **Absolute Coordinates** tells Lightburn to output graphics in your laser's work area at a location that matches their position in your Workspace.
- **Current Position** tells LightBurn to output graphics relative to the position of the laser head at the moment you start a job.
- **User Origin** tells LightBurn to output graphics relative to a custom-defined origin that you set before starting a job.

All three are available in the Start From dropdown menu in the Laser Window.

The 9-dot **Job Origin** selector beneath the dropdown menu determines the orientation of output relative to the Current Position or User Origin. It will be grayed out if you have Absolute Coords selected (Absolute Coords always uses the same pre-determined Job Origin), but clickable if you choose Current Position or User Origin.



> **Note for Galvo Users**
>
> If you have a Galvo laser, you won't see the Start From mode dropdown menu or 9-dot Job Origin selector in the Laser Window — that's because Galvo lasers always use Absolute Coordinates positioning.

Absolute Coordinates

Absolute Coordinates is often the most inuitive option. The grid in your LightBurn Workspace has a coordinate system that matches the physical work area of your laser. The 0,0 position in your Workspace's coordinate system is the same as the location of your laser's **Machine Origin**.

With **Absolute Coords** selected as your Start From mode, anything you place in the LightBurn grid will be output in the corresponding location in your machine's work area, relative to that shared 0,0 point.

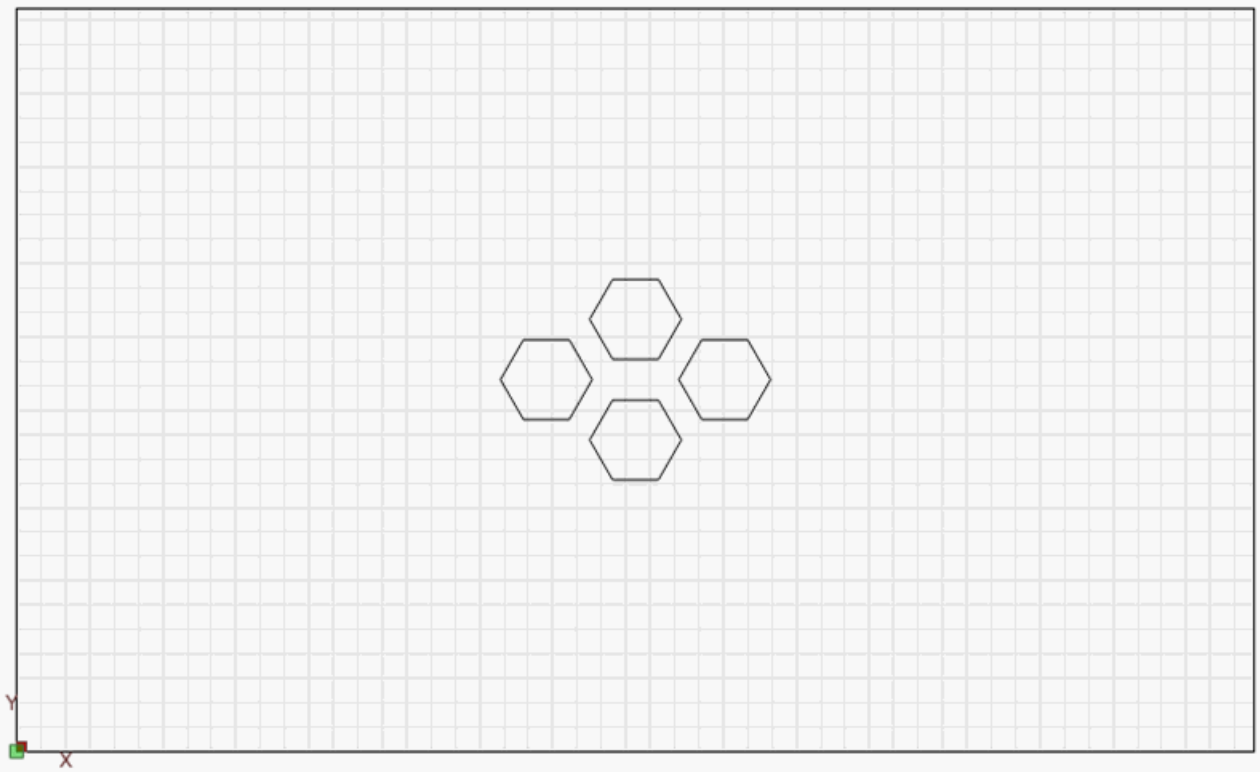
> **Not sure where your Machine Origin is? Click here**
>
> The Machine Origin is the same origin LightBurn identified (or you selected) when you first set up your laser. If LightBurn identified it for you, you most likely won't ever need to change it again.
>
> Almost all GCode-based diode lasers use the lower left corner as their origin, and, for DSP lasers, the origin is the same as the location the laser homes to when you first turn it on (usually the back right corner).

If output to your laser is mirrored or backwards, you may need to change the **Origin** setting in the Device Settings window.

Note

Using absolute positioning requires a laser with a a fixed origin. If you have a small diode laser that does not have homing switches, you will need to manually home the machine. See: Machines without homing sensors / limit switches.

In the image below, the four hexagons placed in the middle of the LightBurn grid will be cut in the middle of the machine work area. The green square in the lower left of the image represents the **Job Origin**, and the red square in the same place shows the **Machine Origin**. When using Absolute Coords, these are always in identical locations.



Using a Camera With Absolute Coordinates Positioning

If your laser came with a camera, or you install one yourself, you'll be able to take a picture and produce an **Overlay** of your laser's work area in your LightBurn **Workspace**.

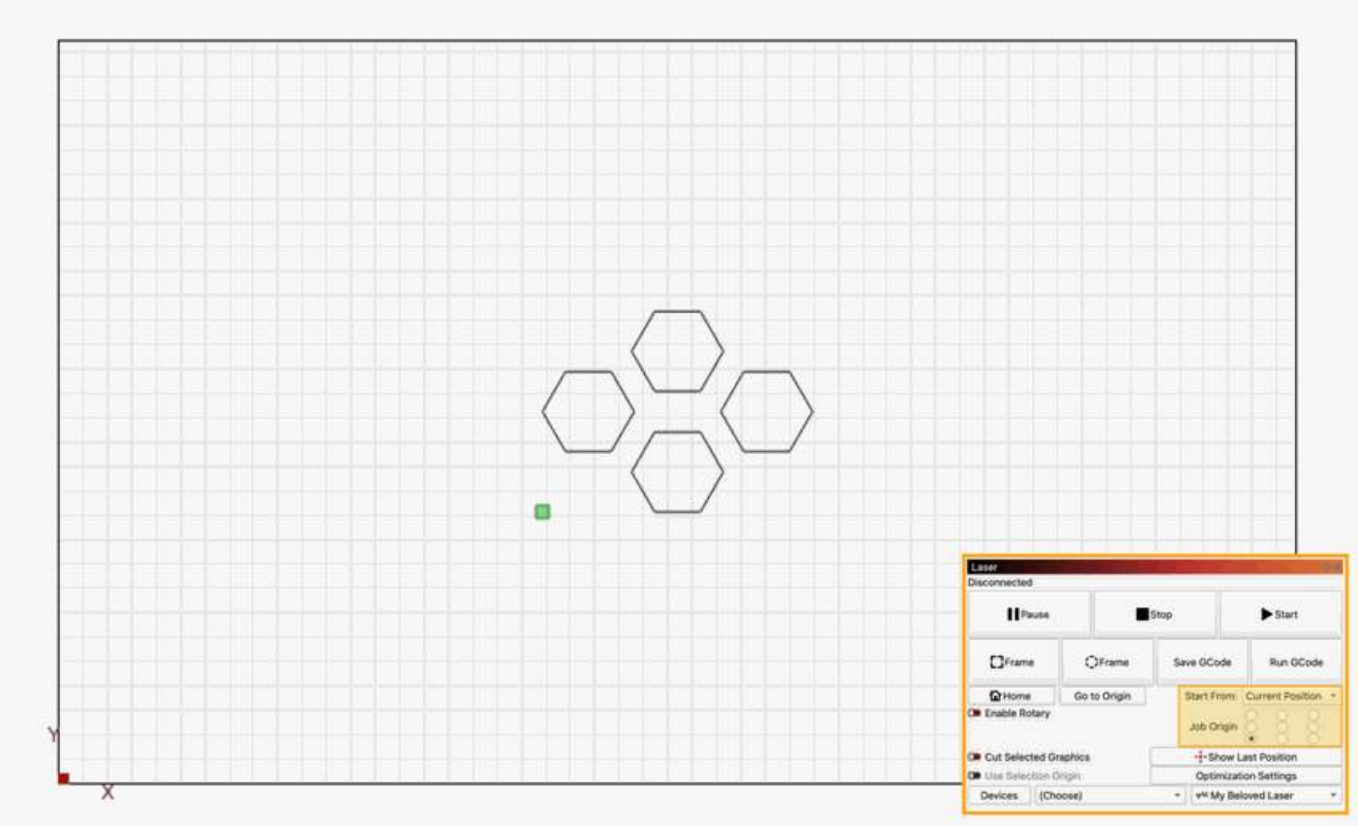
This allows you to position graphics directly over material you've placed in your laser, and use Absolute Coordinates positioning to cut or engrave them in the correct location. Be sure not to select a different Start From mode when you're working with a camera, or things won't line up properly!

See Cameras / Overlays Window for more information on using a camera.

Current Position

With **Current Position** selected, your job outputs relative to wherever the laser head is located when you press the Start button. The **Job Origin** control in the **Laser Window** tells LightBurn how to orient output relative to the laser head.

In this image, we're starting from the Current Position, with the Job Origin set to the lower left:

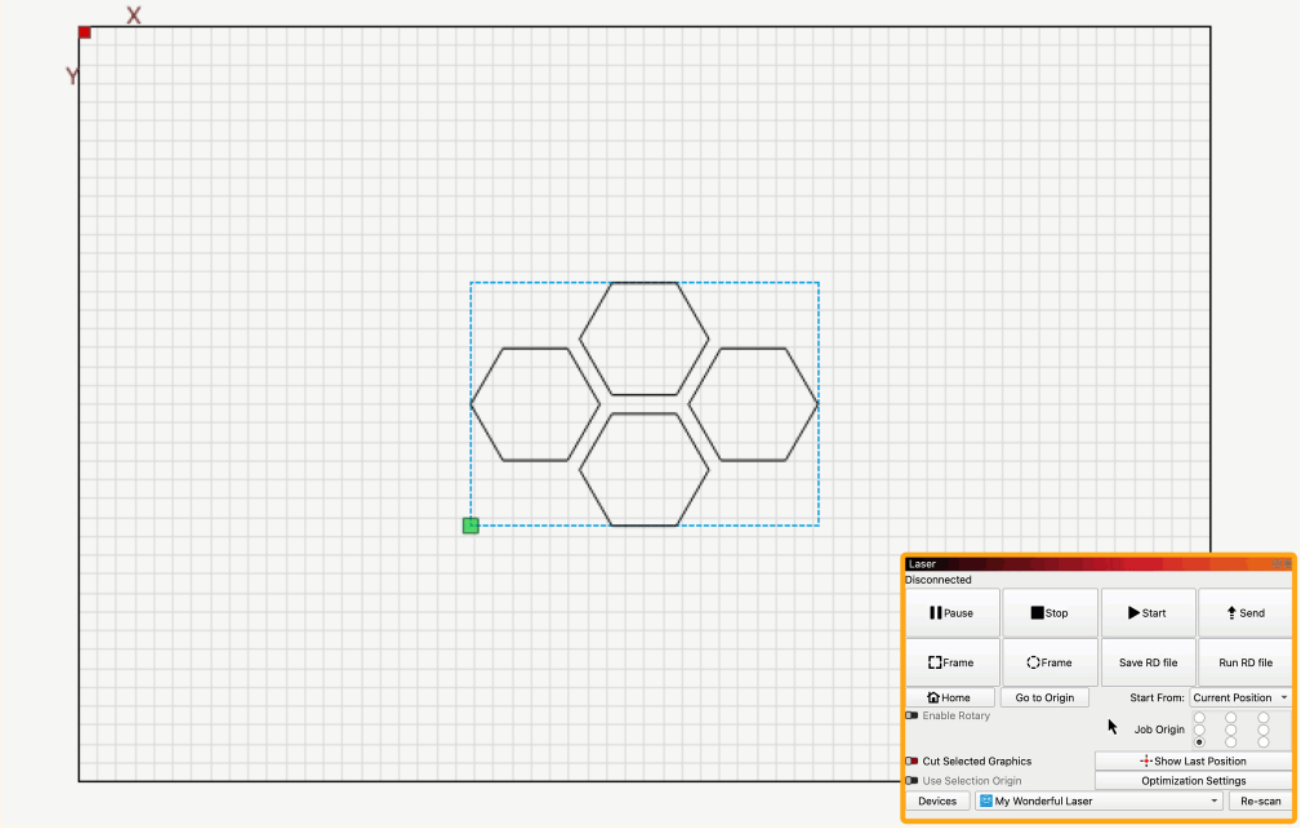


Notice that the green Job Origin indicator has moved. This represents the position of the laser when you start the job, so the laser is going to move slightly up and to the right from wherever it is, cut the four hexagons, and go back to where it started.

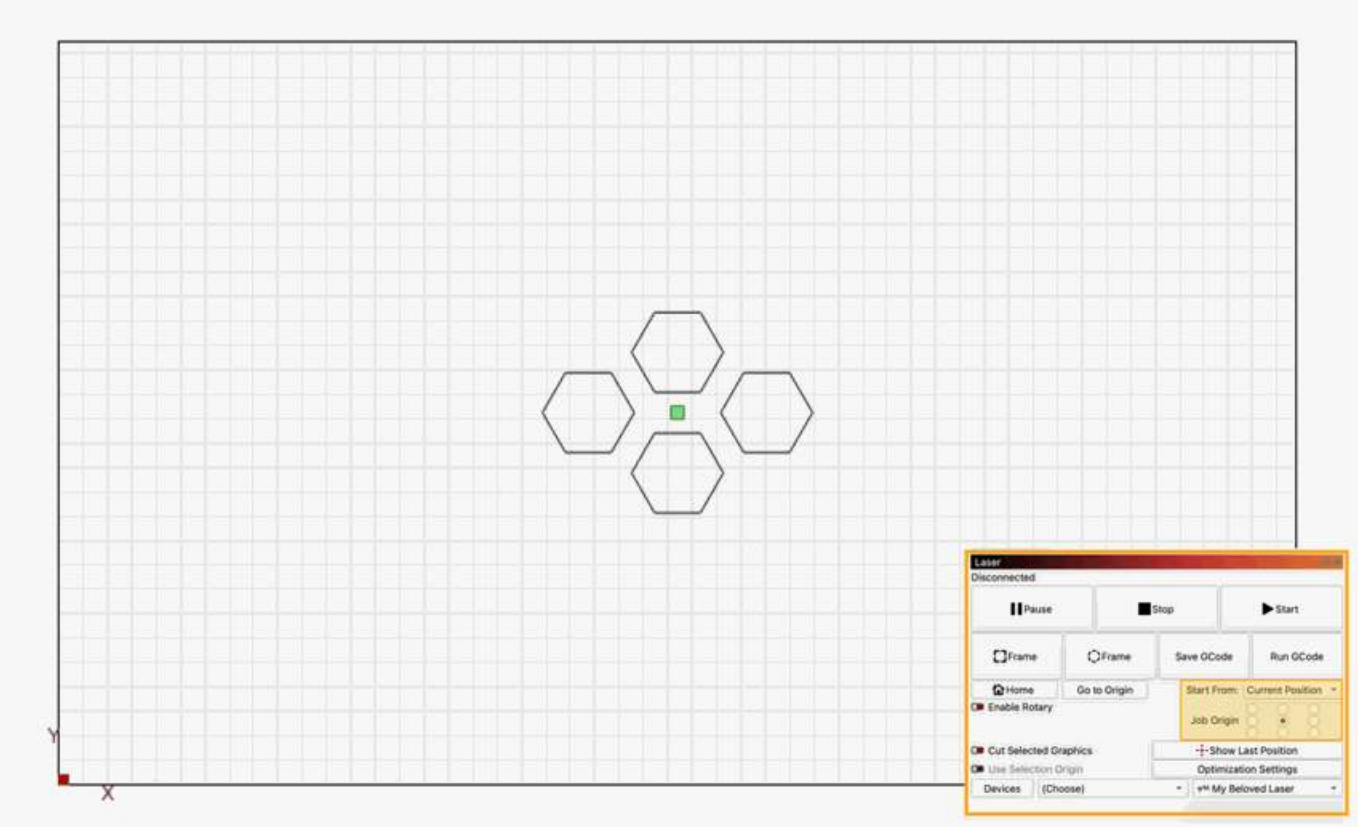
Why is the Job Origin offset from the graphics?

When using Current Position or User Origin, the green Job Origin indicator is always located on a corner, side, or at the center of an imaginary box — the smallest possible rectangle that can fully contain all graphics you're sending to the laser.

Below, we've added a Tool Layer to represent that imaginary box. Notice how the Job Origin moves around the box as we change the location in the 9-Dot selector.



Now imagine that you want to etch this four hexagon pattern onto a coaster or a phone case. Lining it up from the lower left is not easy. If you change the Job Origin setting to center, you get this instead:

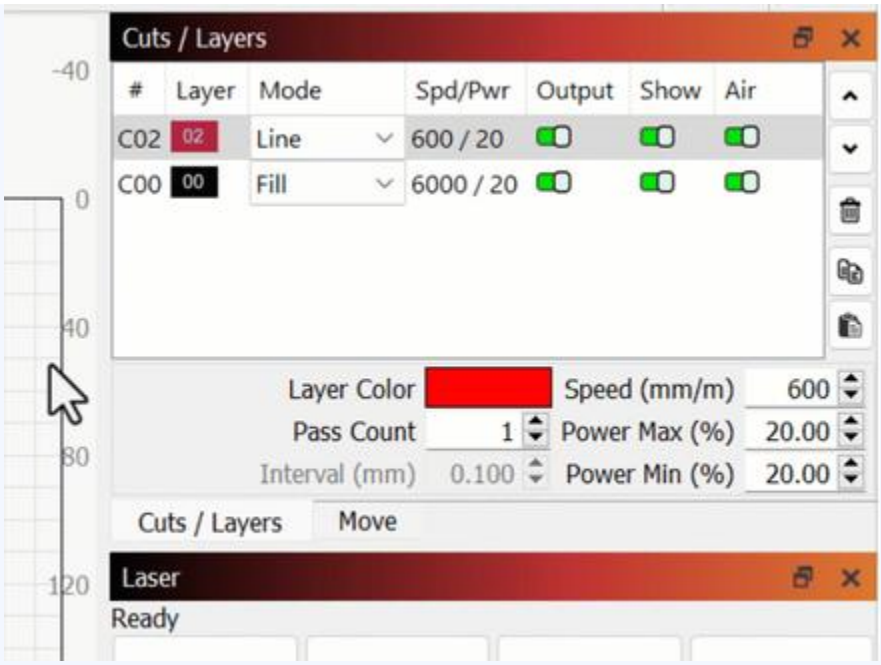


The job will be centered around the current position of the laser head. If you position the laser head directly over the center of the item you want to etch, the output will be centered on the item.

Jogging Your Laser

Both **Current Position** and **User Origin** require you to jog your laser into a desired position to define a custom origin. LightBurn has a few tools for adjusting the location of your laser's head within its work area — the most straightfoward option for beginners is to use the arrow keys in the Move Window. By default, the Move Window is hidden behind the Cuts / Layers Window — click the **Move** tab to expose it.

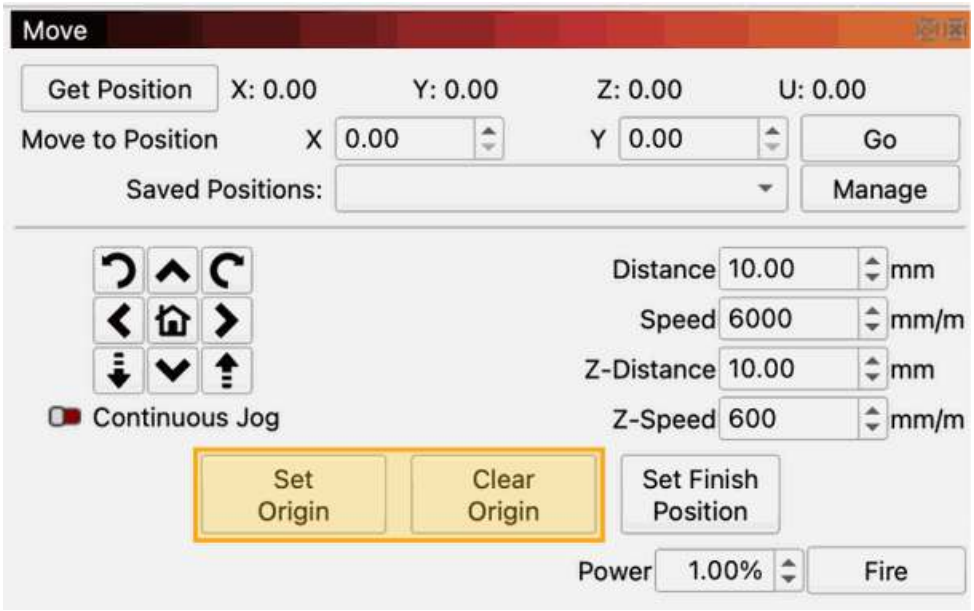
The arrow keys move your laser up, down, right, or left. You can set the amount the laser travels with each click in the **Distance** field, and the rate at which it travels in the **Speed** field.



User Origin

User Origin works almost exactly the same as **Current Position**, except that the starting location is programmable. To set a User Origin, you must first jog your laser to the location you want your job to start from, then set the origin.

Some lasers with digital displays, such as those with Ruida controllers, have an `Origin` button which you press to set the origin. GCode-based systems use the **Set Origin** button in the Move Window in LightBurn to do the same thing.



For supported systems, use **Clear Origin** to remove and reset a custom-set origin.

After setting the origin, you are free to jog your laser around its work area. With User Origin selected as the **Start From** mode, the laser will always move back to start the project in a position relative to the programmed location.

You can also use the **Go to Origin** button in the Laser Window to command your laser to travel back to the origin you've set.

As with Current Position, the **Job Origin** setting determines how output will be oriented relative to the user-defined origin.

The best way to get familiar with the way the different Start From modes is to test them out. On the next page we'll explain how to do that using LightBurn's **Framing** features.

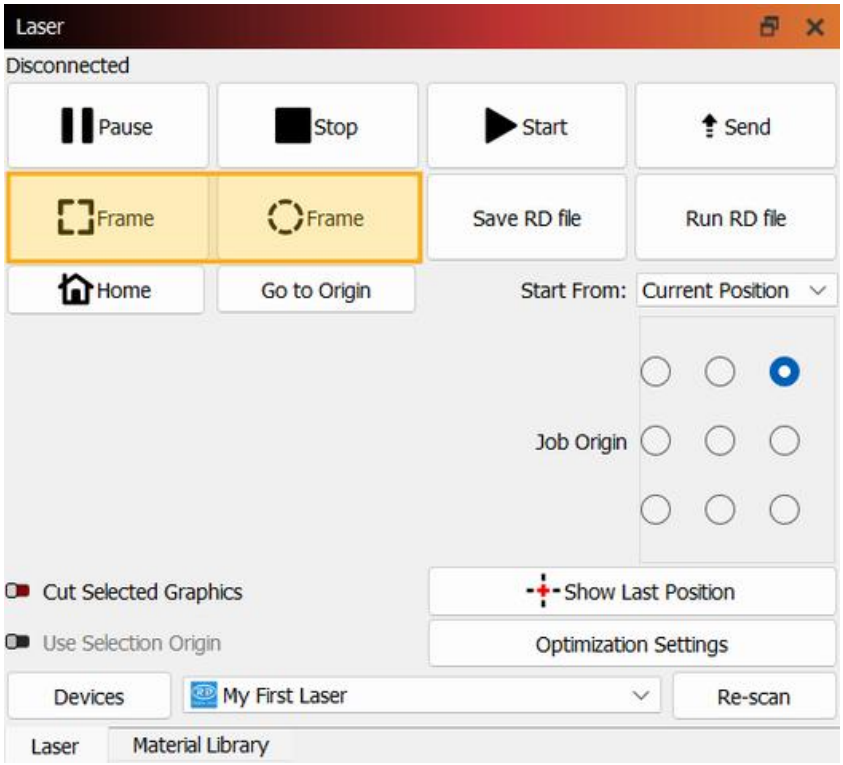
Framing

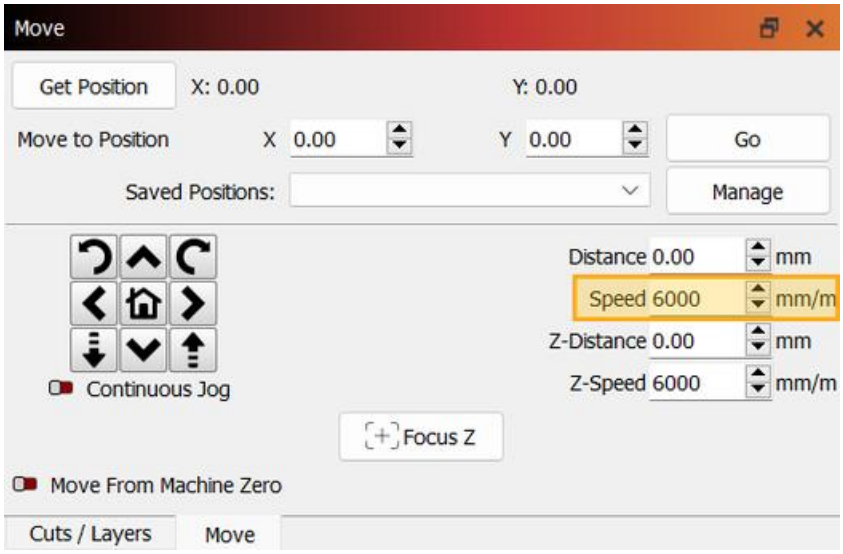
> **Note for Galvo users**
>
> This section covers Framing behavior for Gantry lasers. If you have Galvo laser, see Galvo Framing.

Just as previewing helps you verify that you've applied your layer settings properly, Framing lets you check that you've set your **Start From** mode and **Job Origin** (if applicable) correctly.

There are two **Frame** buttons in the Laser Window — clicking either will command your laser to trace a path around the area where your project will output.

The speed at which the laser travels during Framing is determined by the value set in the **Speed** field in the Move Window.



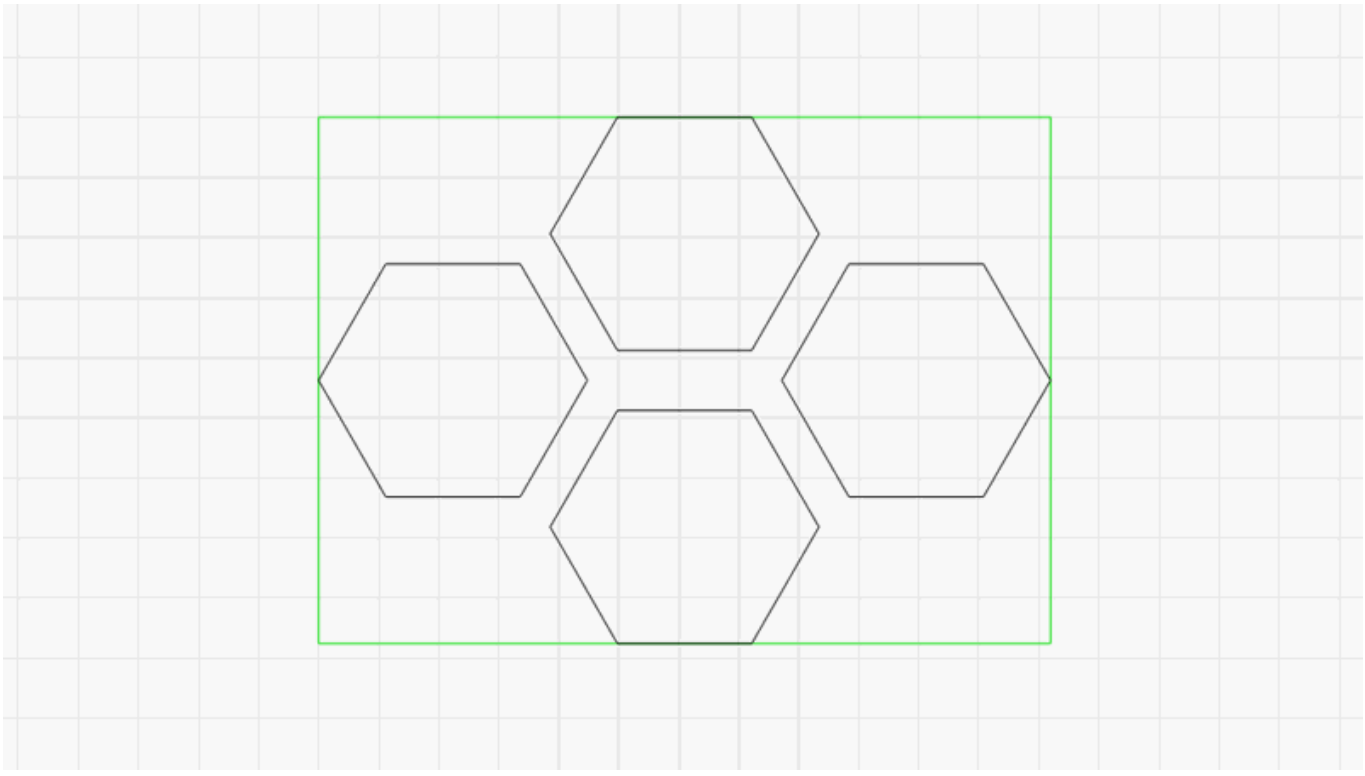


Bounding Box Frame



Bounding Box Framing commands the laser to trace a path defined as the smallest possible rectangle that will fully contain all graphics you're sending to the laser.

Here's a representation of that path in LightBurn:



And here's the laser following that same path during Framing:

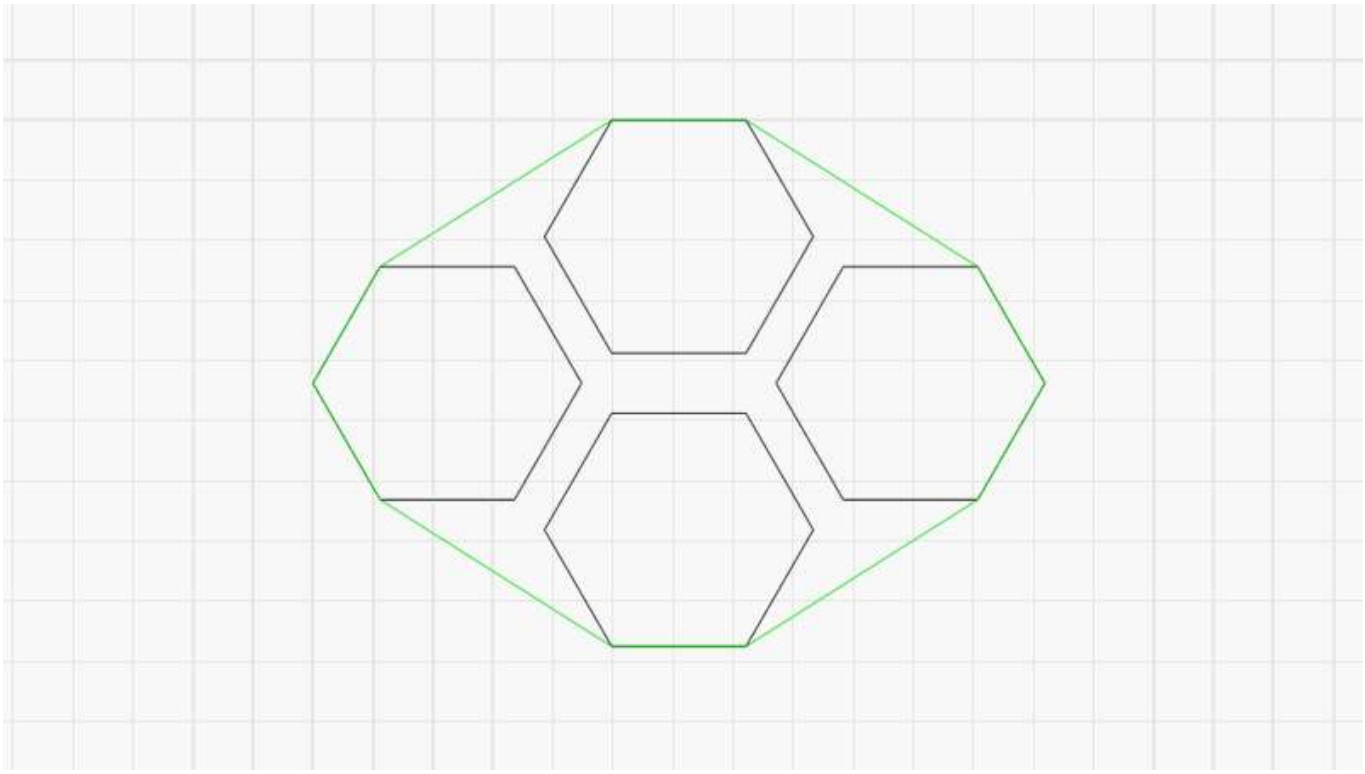


Rubber Band Frame



Rubber Band Framing commands the laser to trace the smallest possible path that fully contains all graphics in the design you're sending to the laser, as if a rubber band were stretched around them.

> **Tip**
>
> Rubber Band Framing is useful for lining up jobs with irregular shapes where a box outline doesn't fit well. For example, a long, thin diagonal shape, or a triangle.

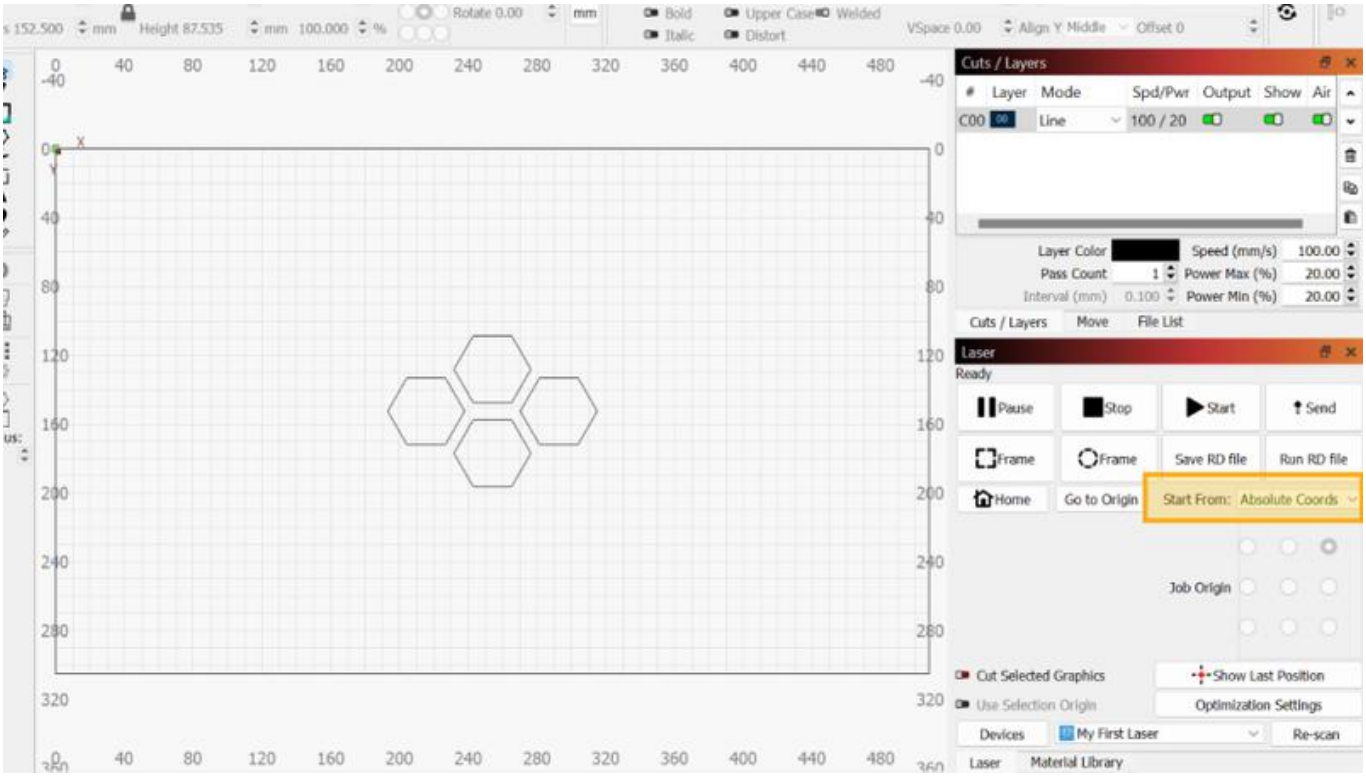




Start From Mode, Job Origin, and Framing

Now let's look at a few examples of how we can test out various **Start From** mode and **Job Origin** combinations. To simplify things, we'll use **Bounding Box Framing** for all of these examples.

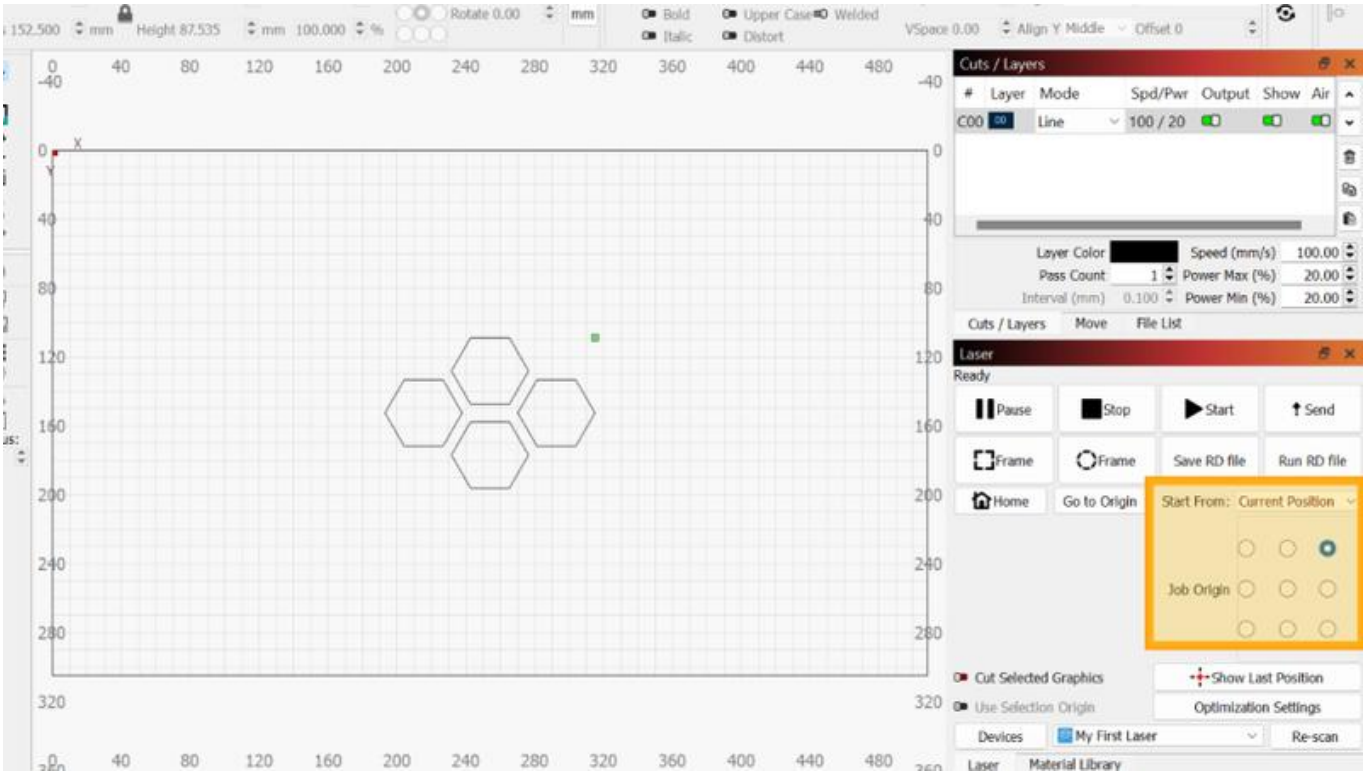
In the first example, we have **Absolute Coords** selected, and our four hexagon design placed directly in the center of the LightBurn Workspace, so when we press the **Frame** button, the laser traces a rectangle in the exact center of its work area:

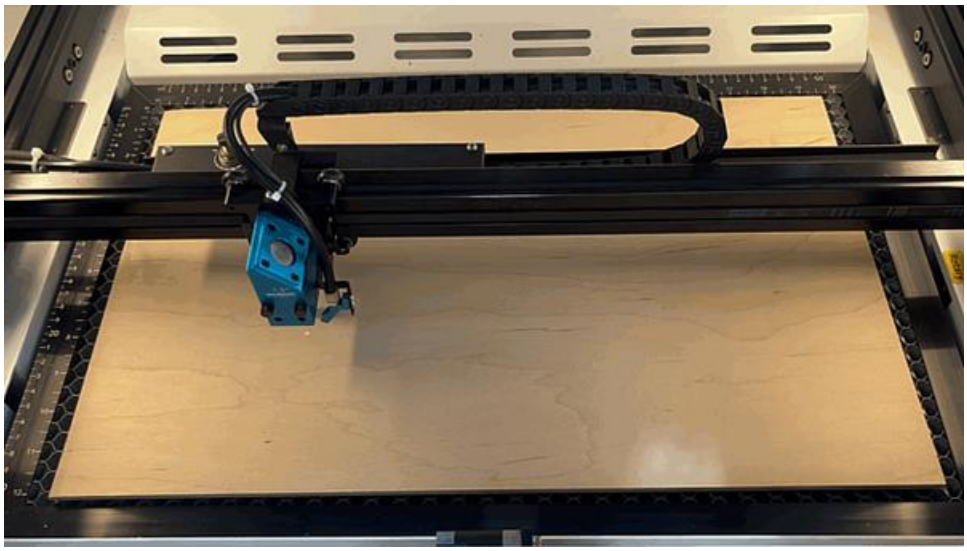




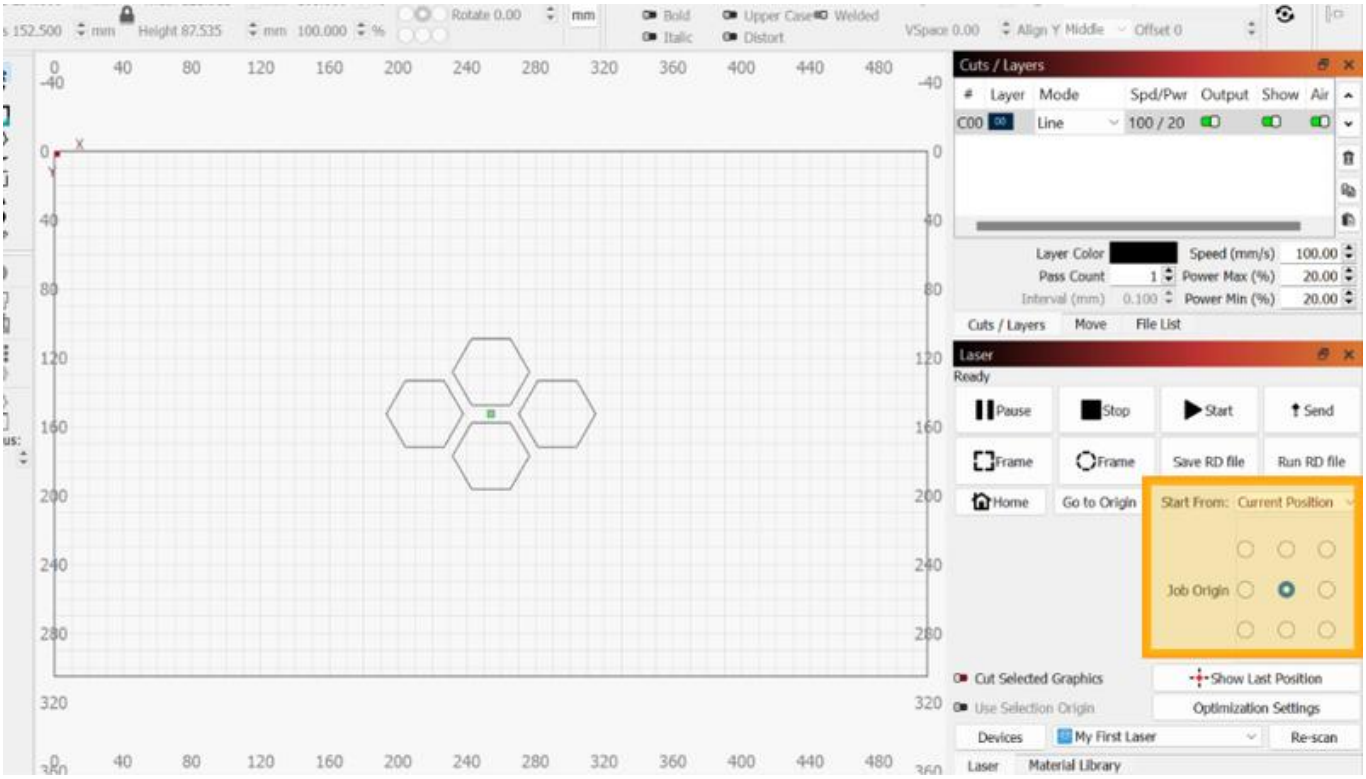
Now we'll leave the design in the same position in our Workspace, but change the Start From Mode, and see how the Framing location changes.

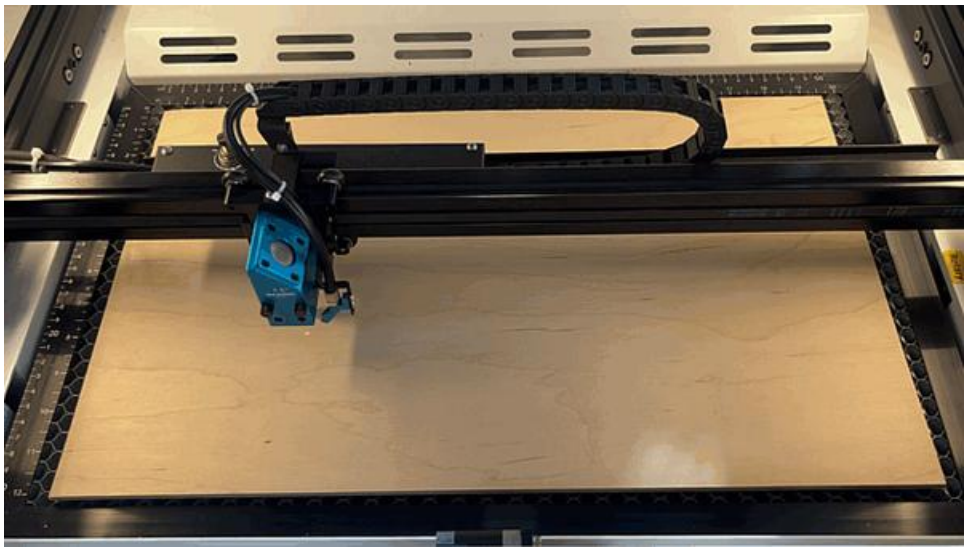
Here we've selected **Current Position** as the Start From mode, and the top right Job Origin position, so the laser moves down and to the left from its starting position:





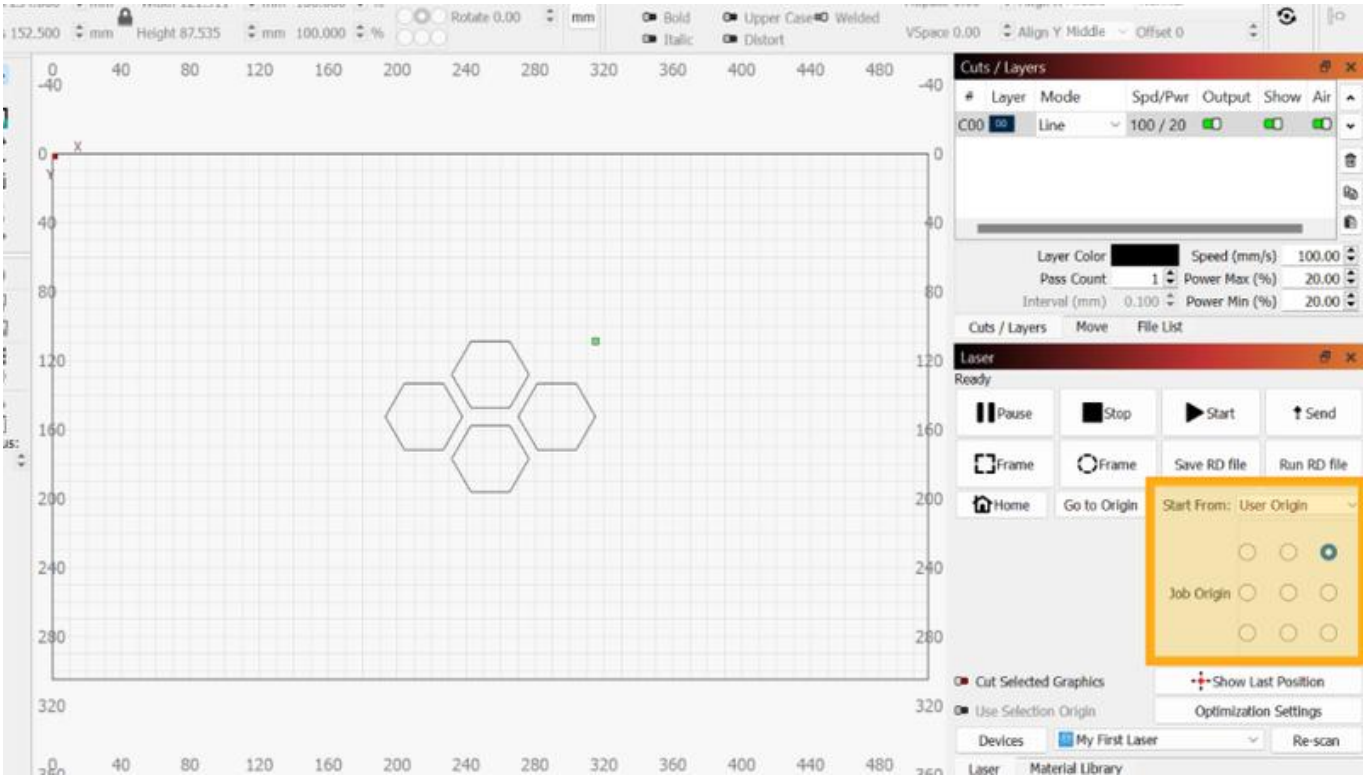
Here's Current Position again, but with the center Job Origin position selected — now the laser moves out from where it's sitting, and traces a rectangle centered on the location where it began:

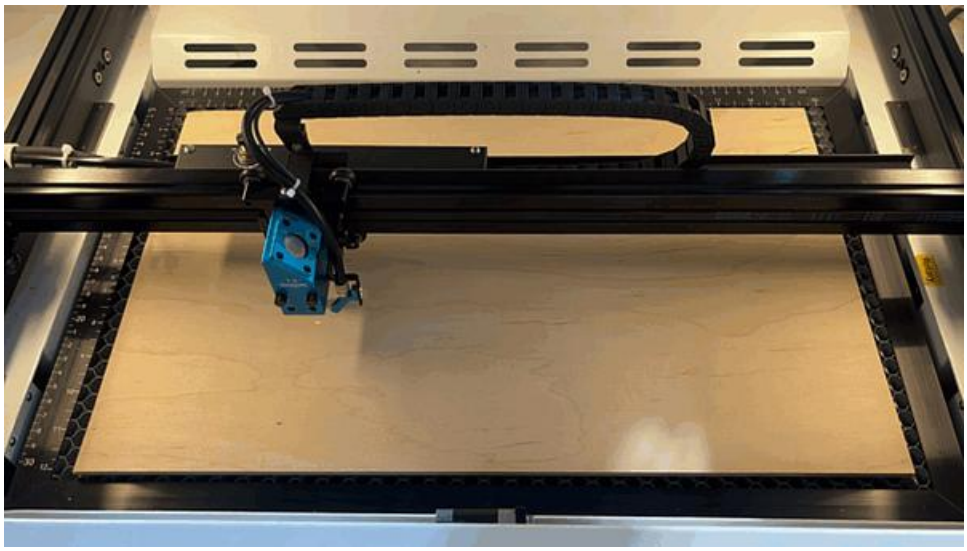




Finally, here's **User Origin** and a top right Job Origin — prior to Framing we'll jog the laser to the top right of its work area, then set the origin (in this case, we used a DSP laser and set the origin from the controller).

Note how even though we jog the laser down to the bottom left after setting the origin, when we press Frame it returns to the top right, to trace a rectangle determined relative to where we set the origin:





Framing Tips

Laser On When Framing

We used a laser with a red dot pointer for the above examples. Pointers are extremely helpful for aligning your laser to material, but many diode lasers do not come with them installed.

Because of this, LightBurn offers the ability to run diode lasers at a very low power to aid in **Framing**. To do so:

1. Go to Edit → Device Settings to open the Device Settings window.

2. In the Basic Settings tab, toggle on the **Enable laser fire button** and **Laser on when framing** settings. Click **OK** to apply.
3. The Move Window should now have a **Fire** button and a box to enter a power level. Keep this value very low to avoid burning the material — start at 0.25%, and only increase that value if the beam is not visible.
4. The **Fire** button will toggle the laser on and off. Now you can use the **Frame** command with the laser visible.

> **Danger**
>
> The Fire button should only be used for diode lasers, which generally don't have a red dot pointer for Framing.
>
> Always wear proper eye protection when Firing your laser. Consult your laser's manufacturer for information on the proper eye protection required for your laser.
>
> This should never be used for a CO_2 laser, which has an invisble beam that could blind you or start a fire.

Frame Continuously

Frame Continuously is an option in the Device Settings window that tells your laser to repeatedly **Frame** the same path until you tell it to stop.

Sometimes you won't get the positioning just right on your first try, but a little nudge of your material is all it takes to get it aligned. Other times, you'll just want to want to double or triple check the position. In either case, Frame Continuously will be a big help.

To enable it:

1. Go to Edit → Device Settings to open the Device Settings window.
2. In the Basic Settings tab, toggle on the **Frame Continuously** setting. Click **OK** to apply.
3. Now when you click either of the Frame buttons, a dialog window will open, letting you know that continuous Framing is active, and to press OK when you're ready to stop.

Using a Camera

With a properly calibrated and aligned camera, you won't need to rely on Framing to align output to material in your laser's work area. Instead, you can position graphics over objects shown in your Workspace Overlay, and they'll be cut or engraved on the object, using **Absolute Coords** positioning.

See Cameras / Overlays Window for more information on using a camera.

Although a properly set up camera is very reliable for positioning, Framing is still a helpful check to make sure you have the correct **Start From** mode selected.

Galvo Framing

> **Note for Gantry users**
>
> This section covers Framing behavior for Galvo lasers. If you have Gantry laser, see Framing.

Just as previewing helps you verify that you've applied your layer settings properly, **Framing** lets you check that your design is set to output in the right spot in your laser's work area.

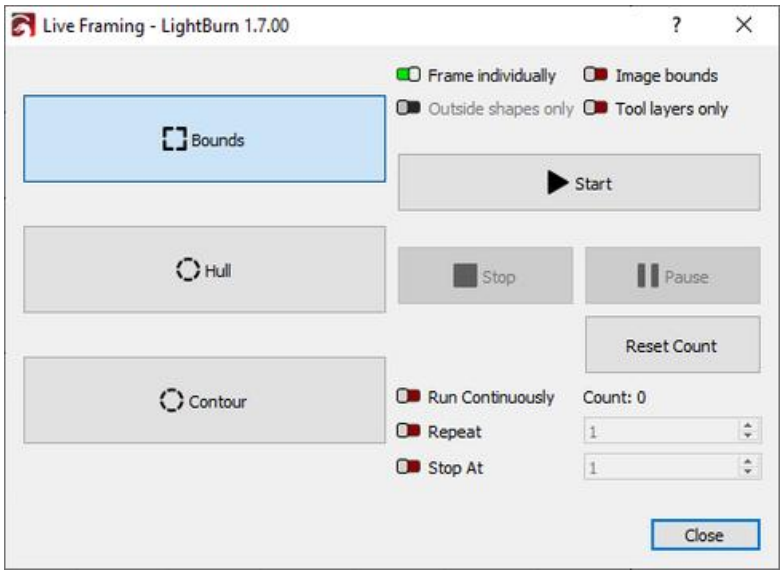
As we noted on the previous page, Galvo lasers always output graphics according to an **Absolute Coordinates** positioning system — your graphics will be marked in your laser's work area in the location that corresponds to their position in your LightBurn Workspace.

Galvo lasers use a red dot light that outlines projects in your laser's work area, allowing you to see where your design will output on a physical object, and you have a few options to determine just how the red dot outlines your project.

Click the **Frame** button in the Laser Window or press `F1` to open the **Live Framing** window and start framing your job.

> **Note**
>
> Mac users need to hold the `Fn` key while pressing `F1` to open the Live Framing window.

By default, the Live Framing window will also open when you press the **Start** button in the Laser Window, but you can disable **Require framing before start** in your Device Settings.



There are quite a few settings in the Live Framing window, but many are advanced options you don't need to worry about just yet. We'll focus on the different styles of Framing, and a few other controls.

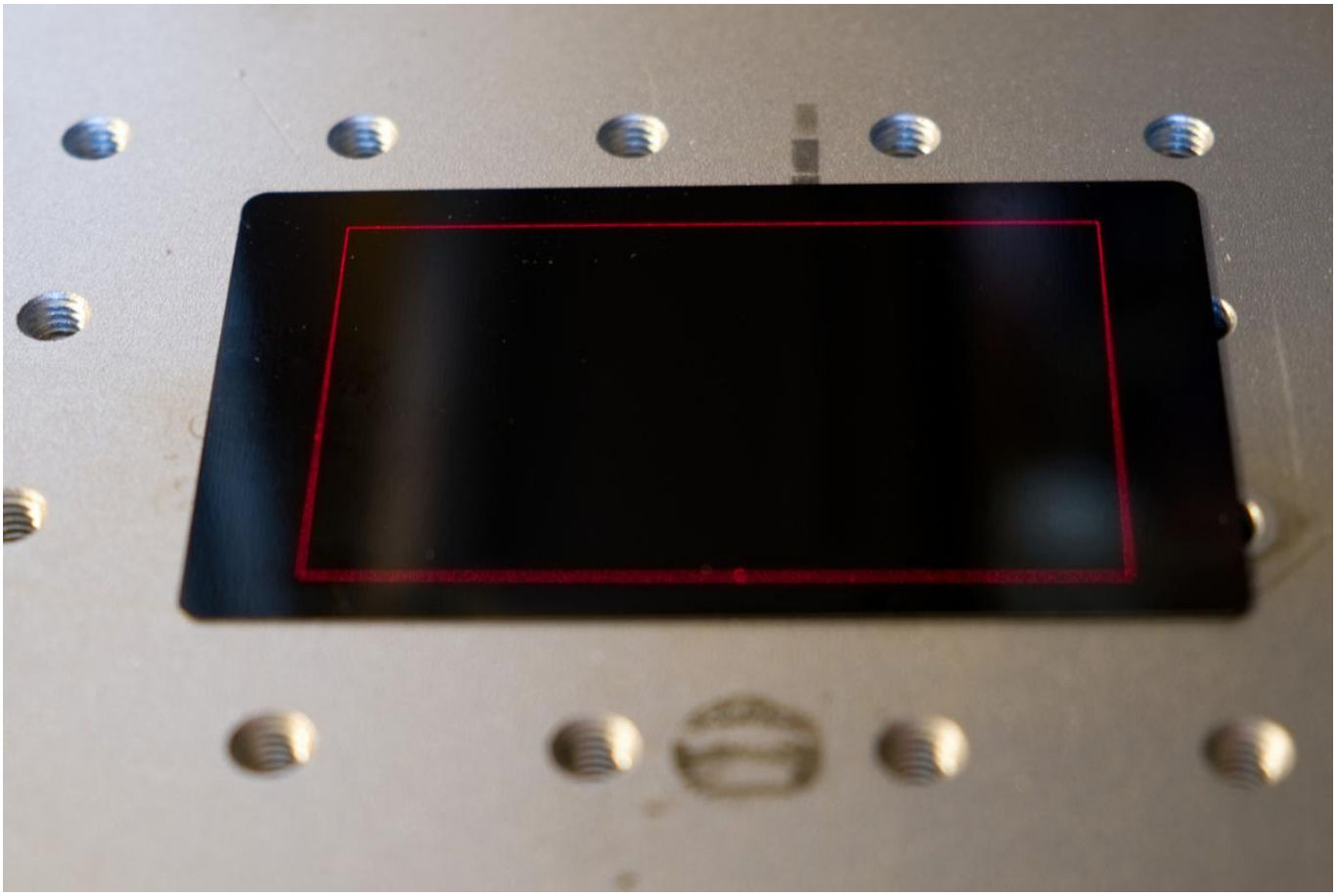
Framing Styles

The different options determine how the red dot light represents your shapes, and which shapes are shown.

Bounds

Commands the red dot light to trace a path defined as the smallest possible rectangle that will fully contain all graphics you're sending to the laser.

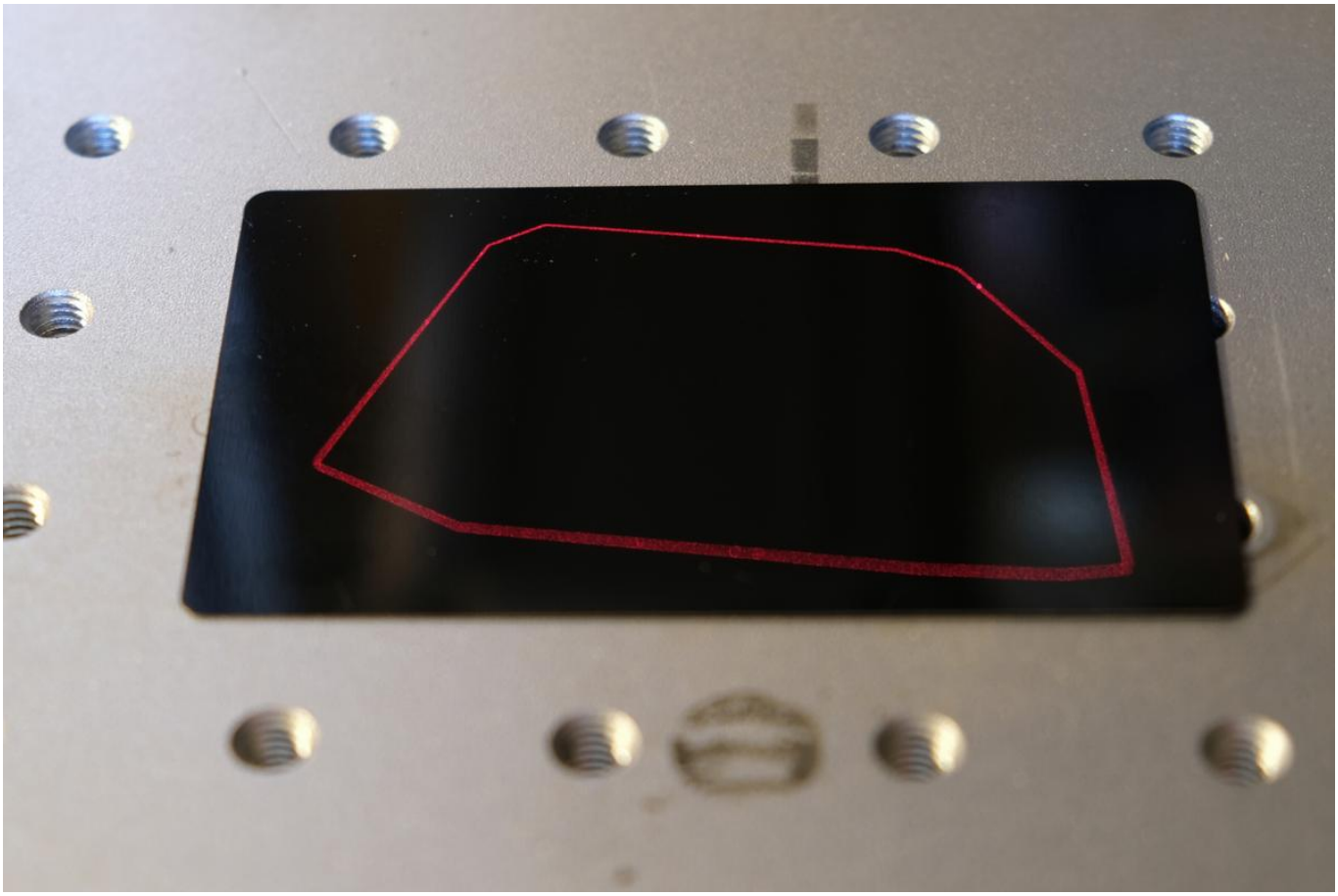
This mode is extremely fast, but doesn't represent complex shapes well.



Hull

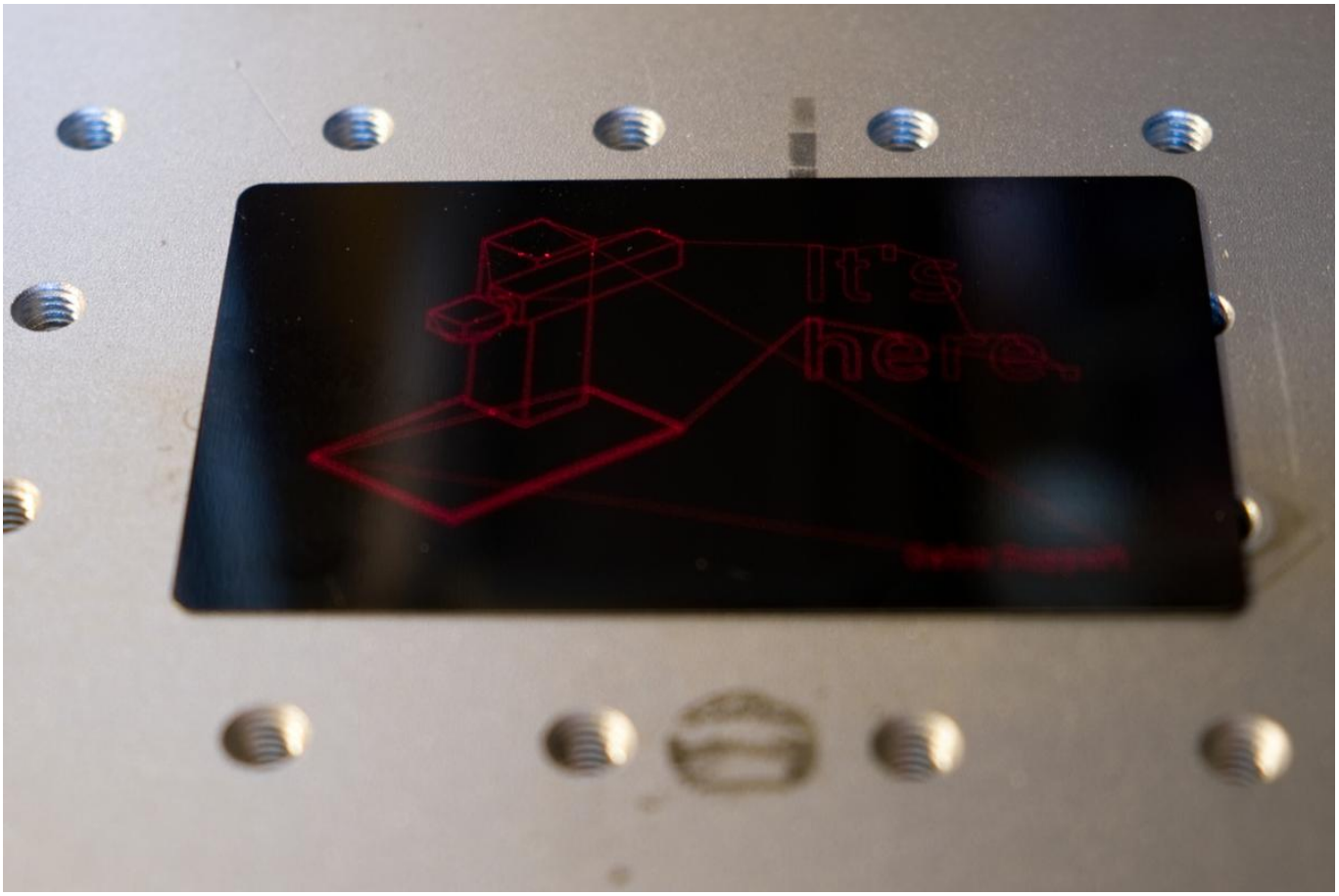
Commands the red dot light to trace the smallest possible path that fully contains all graphics in the design you're sending to the laser, as if a rubber band were stretched around them.

It is nearly as fast as **Bounds**, but gives a much closer fit for rounded or smooth shapes.



Contour

Commands the red dot light to trace a path that follows the exact contours of your shapes — it takes the longest to draw, but it's the most accurate.



Frame Individually

When you're using **Bounds** or **Hull** mode, this option determines whether to trace a single frame around everything in your project, or a separate frame around each individual shape.

This option is grayed out when using **Contour** framing.

Nudging, Rotating, and Rescaling

While framing, you can nudge, rotate, and rescale graphics, adjusting them in your Workspace and live-updating them in your laser's work area.

Key(s)	Result
`Left` / `Right` / `Up` / `Down`	Move graphics by 1 mm
`Shift` + arrow keys	Move graphics by 5 mm
`Ctrl` + arrow keys	Move graphics by .2 mm
`Page Up`	Scale graphics up
`Page Down`	Scale graphics down
`.` (period)	Rotate graphics 15° clockwise

Key(s)	Result
`,` (comma)	Rotate graphics 15° counter-clockwise
`Shift`+`.` (period)	Rotate graphics 1.5° clockwise
`Shift`+`,` (comma)	Rotate graphics 1.5° counter-clockwise
`Ctrl`+`.` (period)	Rotate graphics 45° clockwise
`Ctrl`+`,` (comma)	Rotate graphics 45° counter-clockwise

If your keyboard doesn't have `Page Up` or `Page Down`, try using `Fn`+`Up` or `Fn`+`Down`.

Troubleshooting

- If the framing location or scale is inaccurate, you may need to adjust your laser's focus, or fine-tune the values found in Device Settings → Galvo and Basic Settings → Red Dot.

  See our video guide for more information on adjusting red dot alignment.

  ▶ Watch the video on YouTube: https://www.youtube.com/watch?v=2Ec4-F49aV0

- If **Contour** framing doesn't work, try toggling off **Enable Blanking**, in Device Settings, under **IO Port Settings**.
- For information on framing behavior when using a rotary, see Rotary Mode (Galvo).

Job Control Options

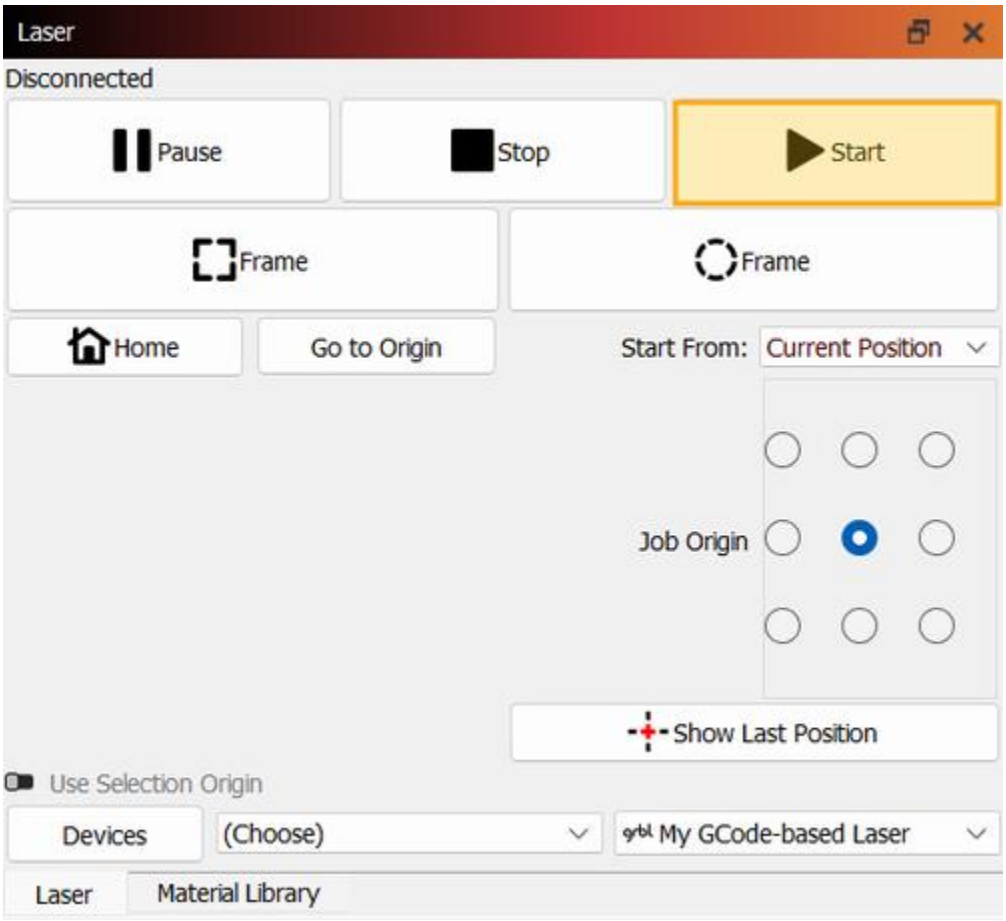
You can also **Start**, **Stop**, and **Pause** your projects from the **Live Framing** window, which remains open while your job is running. These same options are available in the Laser Window, and we cover them in a bit more detail on the next page.

Job Control

After you've created or imported your design, assigned appropriate layer modes and cut settings, and checked (or double-checked) your project using the Preview window and Framing functions, you'll be ready to start the job. Once the job has begun, you'll have a couple of additional options for controlling your machine.

> **Danger**
>
> Before starting any job, make sure you have taken all safety precautions advised by your laser's manufacturer, including wearing proper eye protection, providing adequate ventilation of dangerous fumes, and keeping fire supression equipment close at hand. Never allow your laser to run unattended.

Start



The **Start** button in the Laser Window is the most common method of starting a project, and is available for all types of lasers.

When you press Start, LightBurn will immediately send the project to your laser and it will begin cutting or engraving.

There are a few exceptions to this behavior:

- If you've made a settings or job positioning choice that could cause problems, LightBurn may give you a warning message, and ask you whether you'd like to go ahead with the job, or cancel and correct the issue. We strongly recommend correcting the issue. The warning message will tell you what the problem is, and offer advice on correcting it — be sure to read it in full!
- If you have a Galvo laser, the Live Framing window will open instead. The same job control buttons that area available in the Laser Window appear in the Live Framing window as well, so you can Start the job from there, after checking positioning again, if necessary. You can bypass Framing by toggling off Require Framing Before Start in the **Device Settings** window.
- For any type of device, you can toggle on Enable Jog Checklist in your Device Settings, to have LightBurn show a checklist that you'll need to confirm before starting the job. This is a great way to make sure you haven't forgotten about any crucial safety procedures before starting your job.

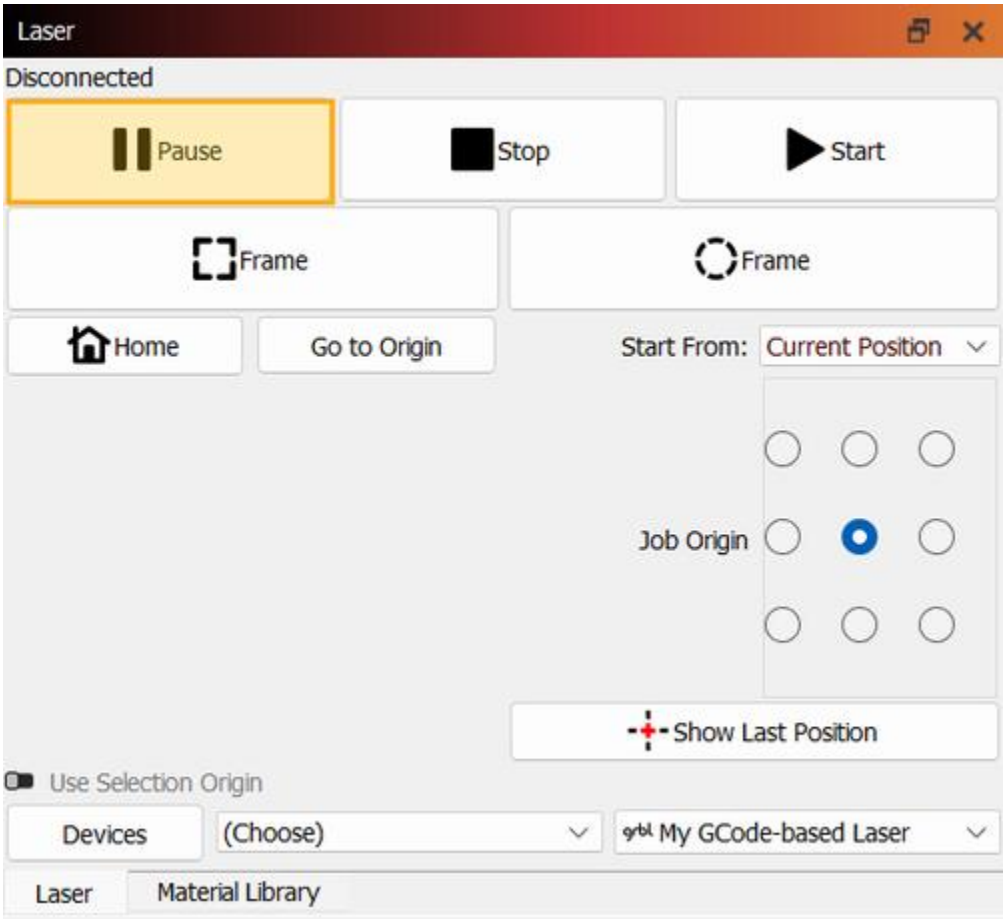
Click here to learn about alternate methods of starting a job

Some types of lasers have additional options for starting projects. If you have a DSP laser, you'll see the option to Send your project to your laser — this option will transfer the probject to your laser's controller. You can use the control panel to access the file and start it from there.

Both DSP and GCode-based lasers have the option to Save in Machine-Specific Format. If you Save in Machine Specific format, LightBurn will save the project with a file extension such as `.nc` or `.rd`. You can then transfer the saved file to your machine using a USB stick. This option is not available for Galvo lasers.

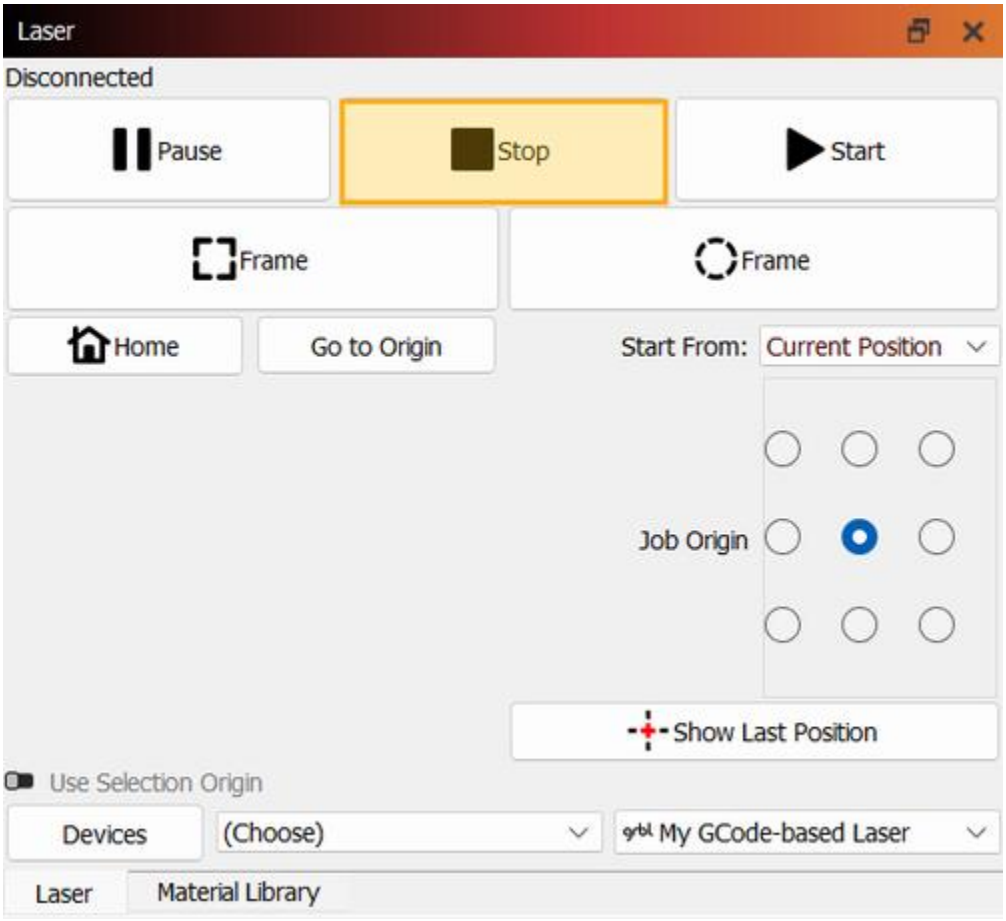
If you're using Beginner Mode you won't see this button in the Laser Window, but the option is also available in the File Menu.

Pause



Click the **Pause** button while your laser is working to temporarily halt the job. You'll be able to click again to **Resume**.

Stop



Clicking the **Stop** button will permanently halt your job, meaning you won't be able to resume right where you left off.

> **Warning**
>
> The Stop button should never be your only means of stopping a job in progress. Locate your laser's emergency stop switch, if available, or be prepared to use another method to compeletely cut power to the machine in the event of a fire. Consult your laser's manufacturer for guidance specific to your machine.

First Project

Now that you know how to create, set up, Preview, Frame, and **Start** a project, you're ready to try running your laser for real. You have two options for the next step:

- You can learn how to use the Material Test tool to dial in optimal settings for your laser and material, before moving on to the our First Project guide.
- You can proceed directly to the First Project, if you have settings recommendations from your laser's manufacturer that you're confident in.

First Material Test

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=pTx0RG_4Wbs

Getting recommendations directly from your laser's manufacturer is usually the best option for finding cut settings, but if those recommendations aren't available or aren't yielding great results, there's another option.

LightBurn has a built-in Material Test that you can use to test different combinations of **Speed**, **Power**, **Passes**, **Line Interval**, and other settings. The Material Test works by automatically setting up a grid of boxes that will be cut or engraved at a range of different settings.

We'll walk you through the process of setting up a Material Test grid to test both engraving and cut settings.

What You'll Need

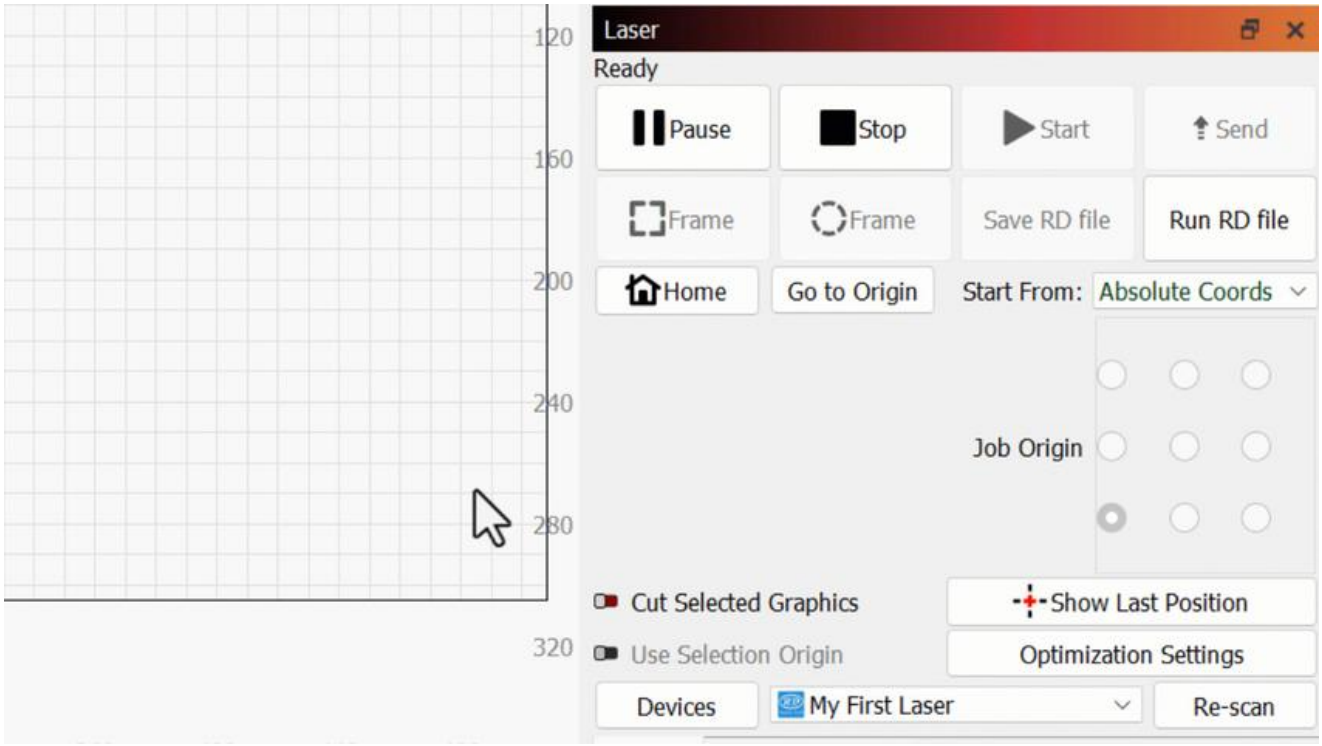
- LightBurn installed on your computer.
- A laser added to LightBurn.
- A spare piece of material you want to test, placed in your laser's work area. Proper settings differ between materials, so you'll need to peform the test again if you change material.

Material Test Positioning

> **Note for Galvo laser users**
>
> Galvo lasers always use **Absolute Coordinates** positioning, and, by default, the test grid will output in the center of your laser's work area. You can adjust its location by using the nudge hotkeys while Framing, or manually changing the **Y Center** and **X Center** values in the Material Test Generator window.

The output location of the test grid in your laser's work area is determined by the Start From mode you select before opening the **Material Test Generator** window. If you choose **Absolute Coords**, you'll have additional controls within the Material Test Generator window, but we'll be using **Current Position** in this guide.

1. In the Start From drop down menu in the Laser Window, select **Current Position**.
2. In the 9-dot **Job Origin** selector beneath the drop down menu, select the center position.



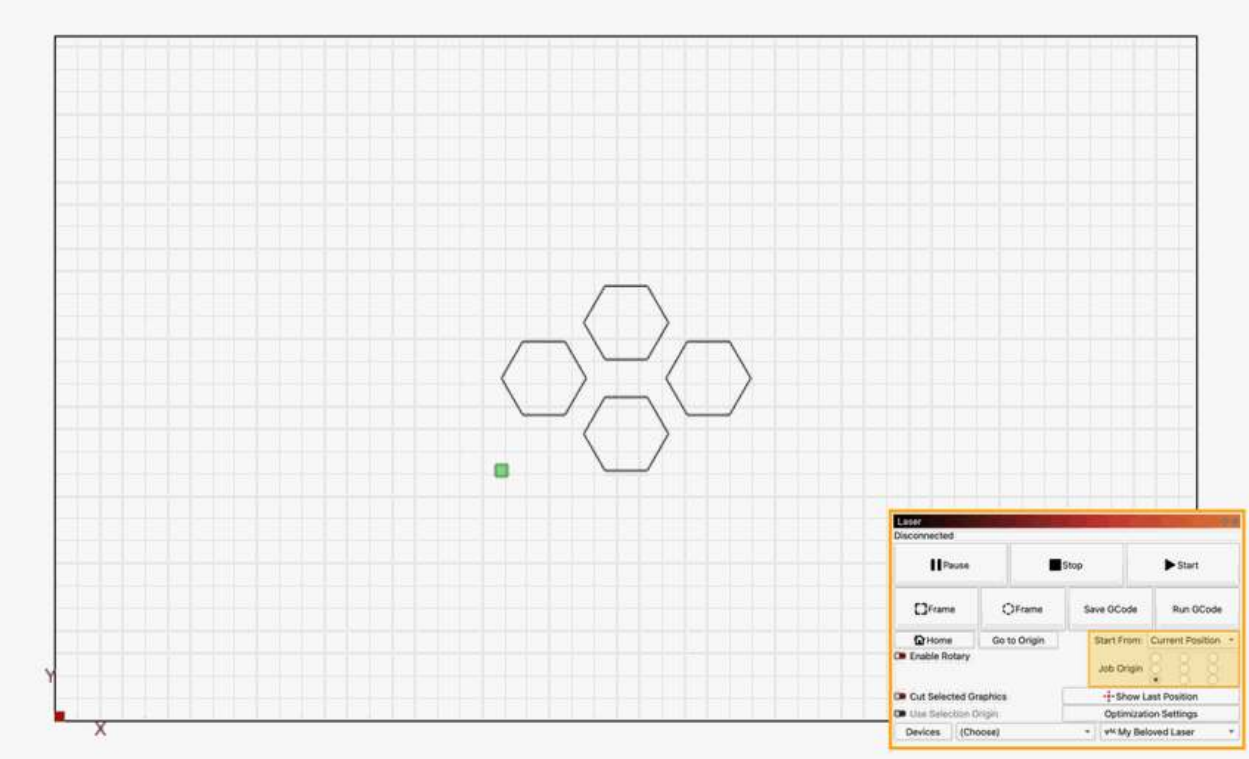
3. Selecting Current Position and the center Job Origin location means the Material Test grid will be centered wherever the laser head is located when you press the **Start** button later on.

Click here to learn more about Current Postion and other Start From modes

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=aphiLTUWcjE

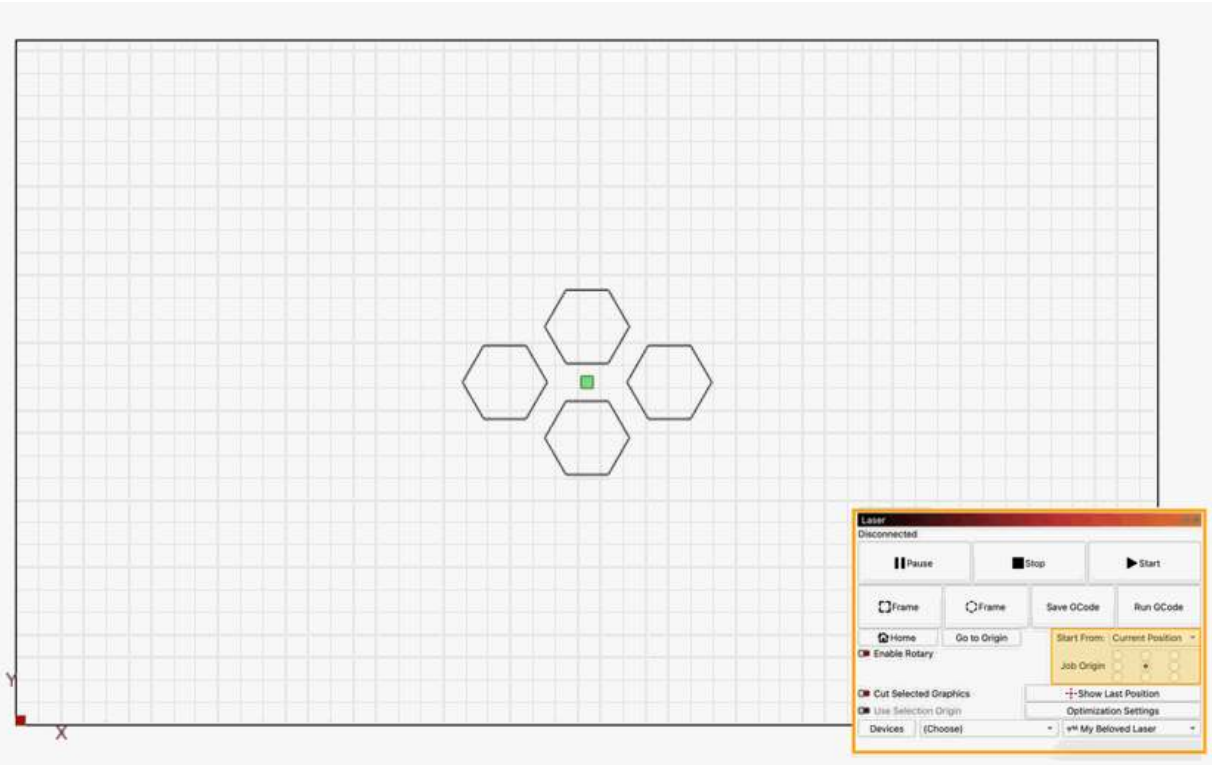
With **Current Position** selected, your job outputs relative to wherever the laser head is located when you press the **Start** button. The **Job Origin** control in the **Laser Window** tells LightBurn how to orient output relative to the laser head.

In this image, we're starting from the Current Position, with the Job Origin set to the lower left:



Notice that the green Job Origin indicator is positioned to the lower left of our graphics. This represents the position of the laser when you start the job, so the laser is going to move slightly up and to the right from wherever it is, cut the four hexagons, and go back to where it started.

Now imagine that you want to etch this four hexagon pattern onto a coaster or a phone case. Lining it up from the lower left is not easy. If you change the Job Origin setting to center, you get this instead:



The job will be centered around the current position of the laser head. If you position the laser head directly over the center of the item you want to etch, the output will be centered on the item.

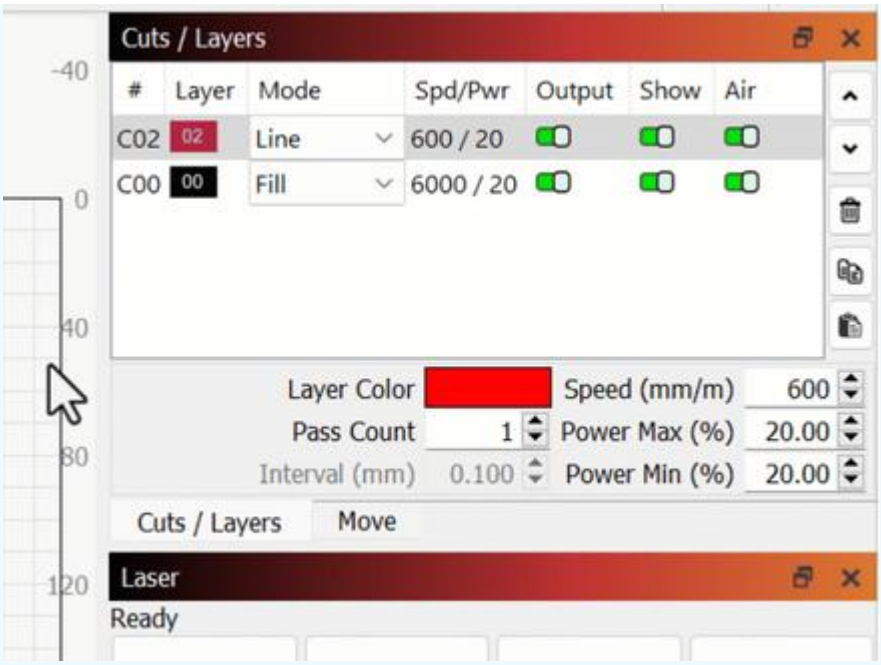
4. Jog your laser over the material. Make sure there is enough room around your laser's head to accomodate an approximately 80 mm x 95 mm rectangle — the default size of the Material Test grid.

   If your laser has a controller display and keypad, you can use the buttons there to jog. If not, use the buttons in the Move Window.

Click here to learn more about jogging your laser

LightBurn has a few tools for adjusting the location of your laser's head within its work area — the most straightfoward option for beginners is to use the arrow keys in the Move Window. By default, the Move Window is hidden behind the Cuts / Layers Window — click the **Move** tab to expose it.

The arrow keys move your laser up, down, right, or left. You can set the amount the laser travels with each click in the **Distance** field, and the rate at which it travels in the **Speed** field.



Focusing the Laser

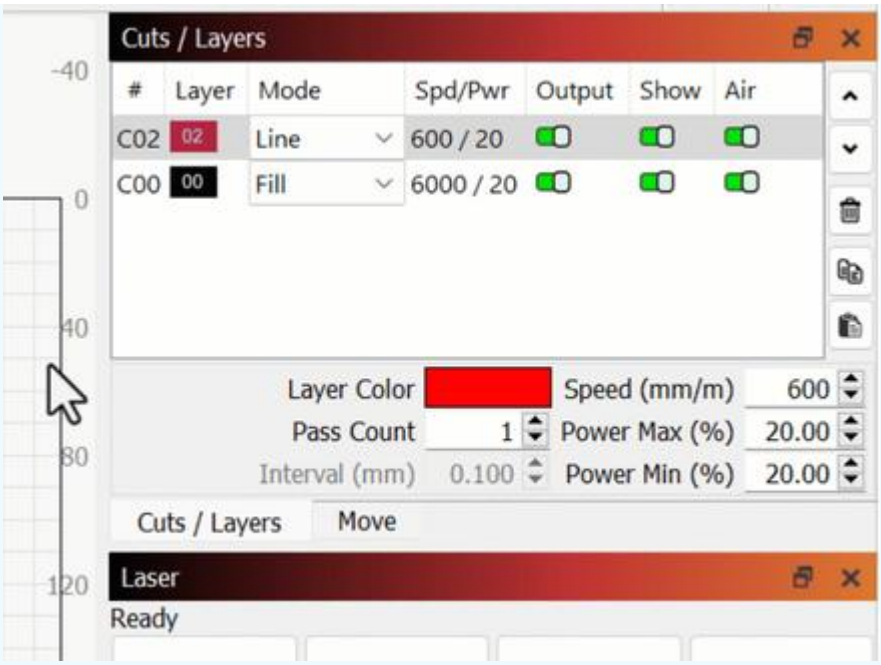
Lasers need to be focused correctly to achieve good results.

1. Place the material in the laser and move the head over the material. LightBurn has jog controls in the Move Window to make this easier. You can ignore this step if you've got a Galvo laser.

Click here to learn more about jogging your laser

LightBurn has a few tools for adjusting the location of your laser's head within its work area — the most straightfoward option for beginners is to use the arrow keys in the Move Window. By default, the Move Window is hidden behind the Cuts / Layers Window — click the Move tab to expose it.

The arrow keys move your laser up, down, right, or left. You can set the amount the laser travels with each click in the **Distance** field, and the rate at which it travels in the **Speed** field.



2. Focus the laser according to your laser's user guide:
   - GCode-based Diode lasers usually have a focusing spacer to set the distance between the laser and the material.
   - DSP-type CO_2 lasers may require manual focusing with a focusing spacer or may support auto-focus, either through the controller, or with the Focus Z button.
   - Galvo lasers require manual focusing. Many galvo lenses are marked with their optimal focal distance, and you can use a ruler to set the correct distane between the lens (or mirrors) and your material. Some are equipped with a two red dot focusing system — adjust the height of the scan head until the dots meet, then the laser is focused. Finally, see our guide on Galvo Laser Focusing for a manual method of determining optimal focus.

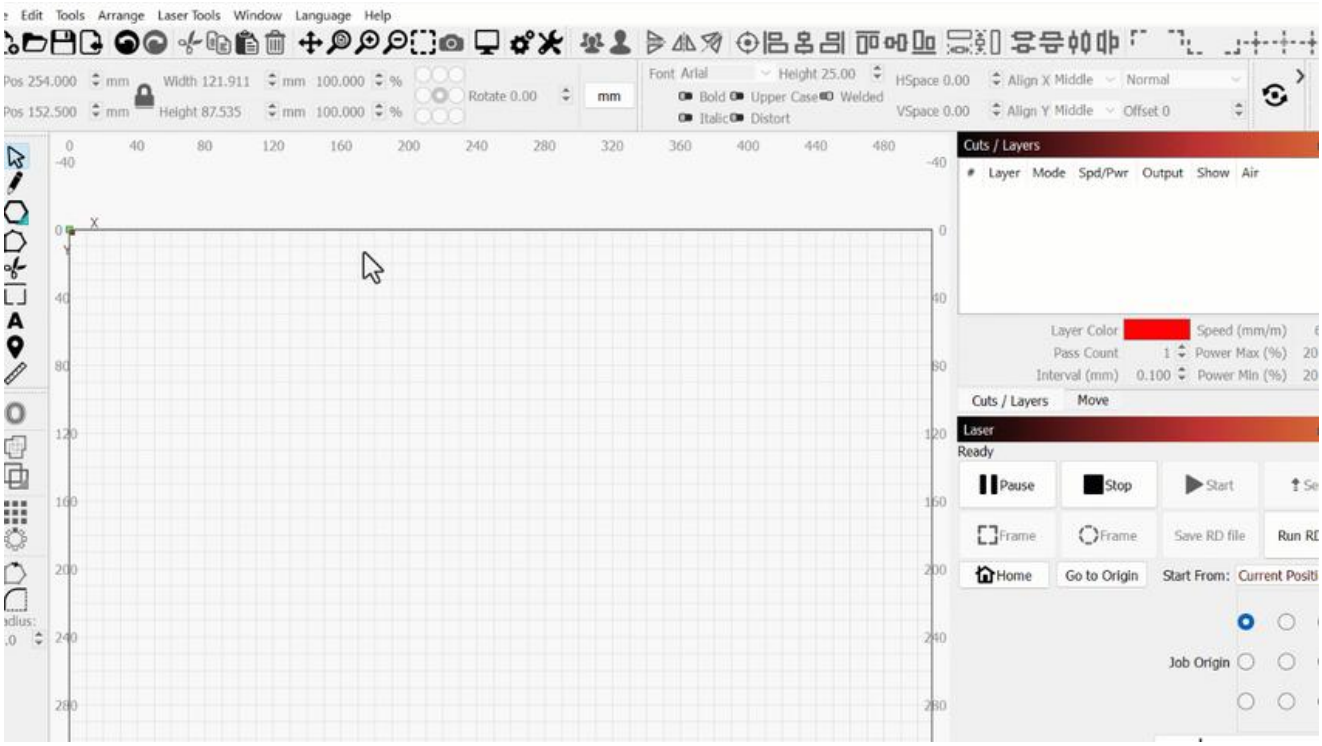
Material Test Setup

LightBurn has a handful of **Built-In Presets** that make setting up your first **Material Test** straightforward.

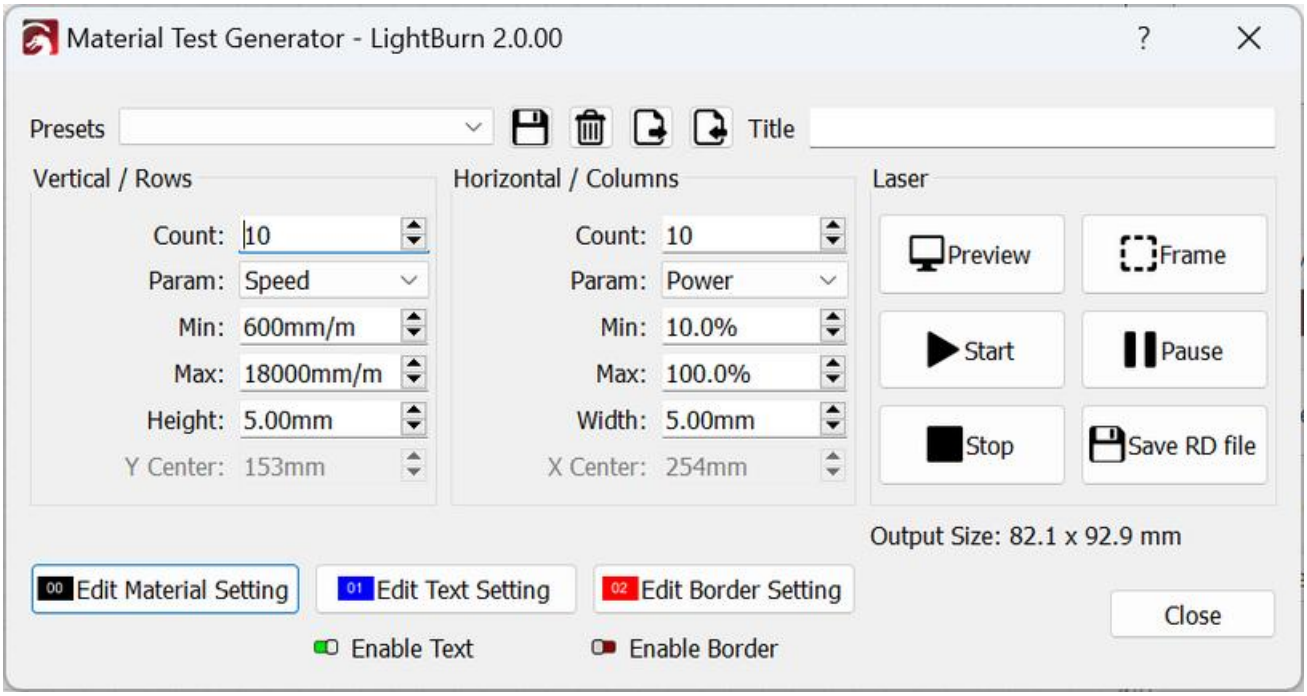
> **Note for Galvo laser users**
>
> The range of reasonable settings for Galvo lasers is too wide for LightBurn to include Galvo presets. Consult your laser's manufacturer, or search our forum for guidance from other users. Visit our Material Test reference page for information on configuring a test grid manually.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=TfJ5fI2J-LI

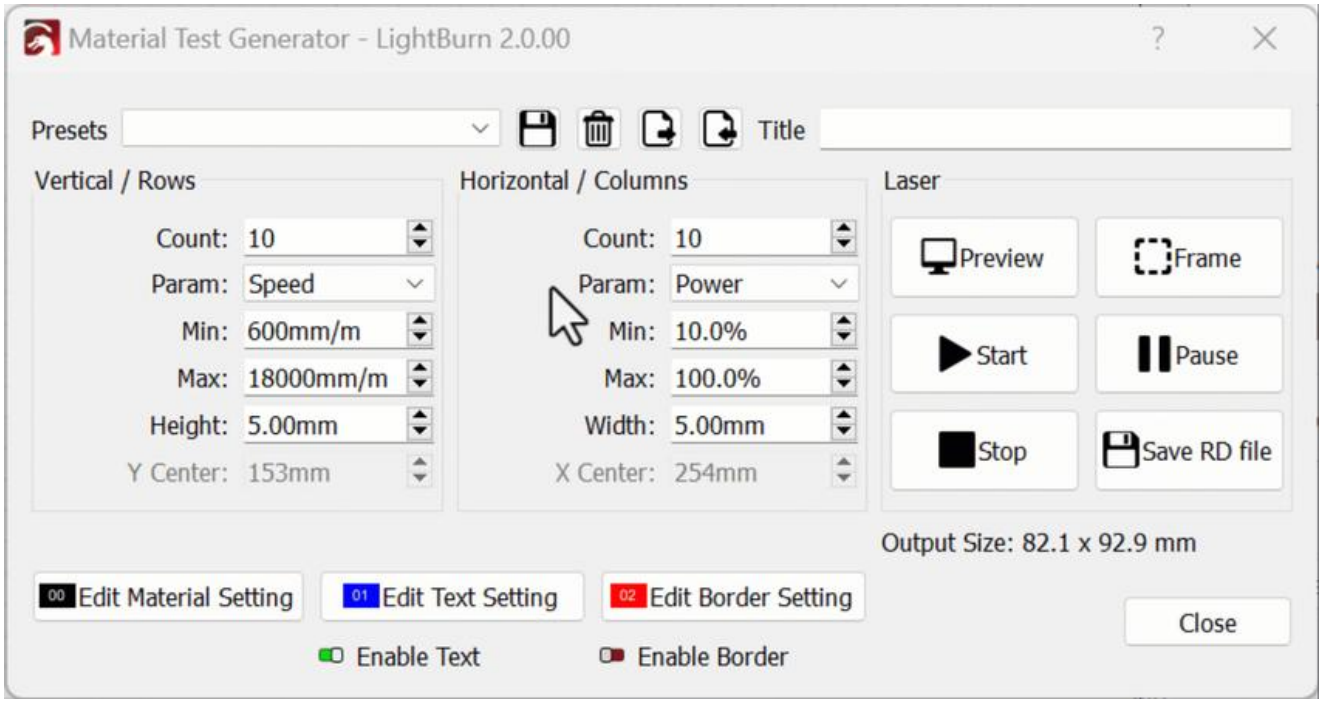
1. To open the Material Test, go to Lasers Tools → Material Test.



2. You'll see fields to select from a list of parameters to test, adjust the range of tested values, and change the size, position, and number of test boxes.



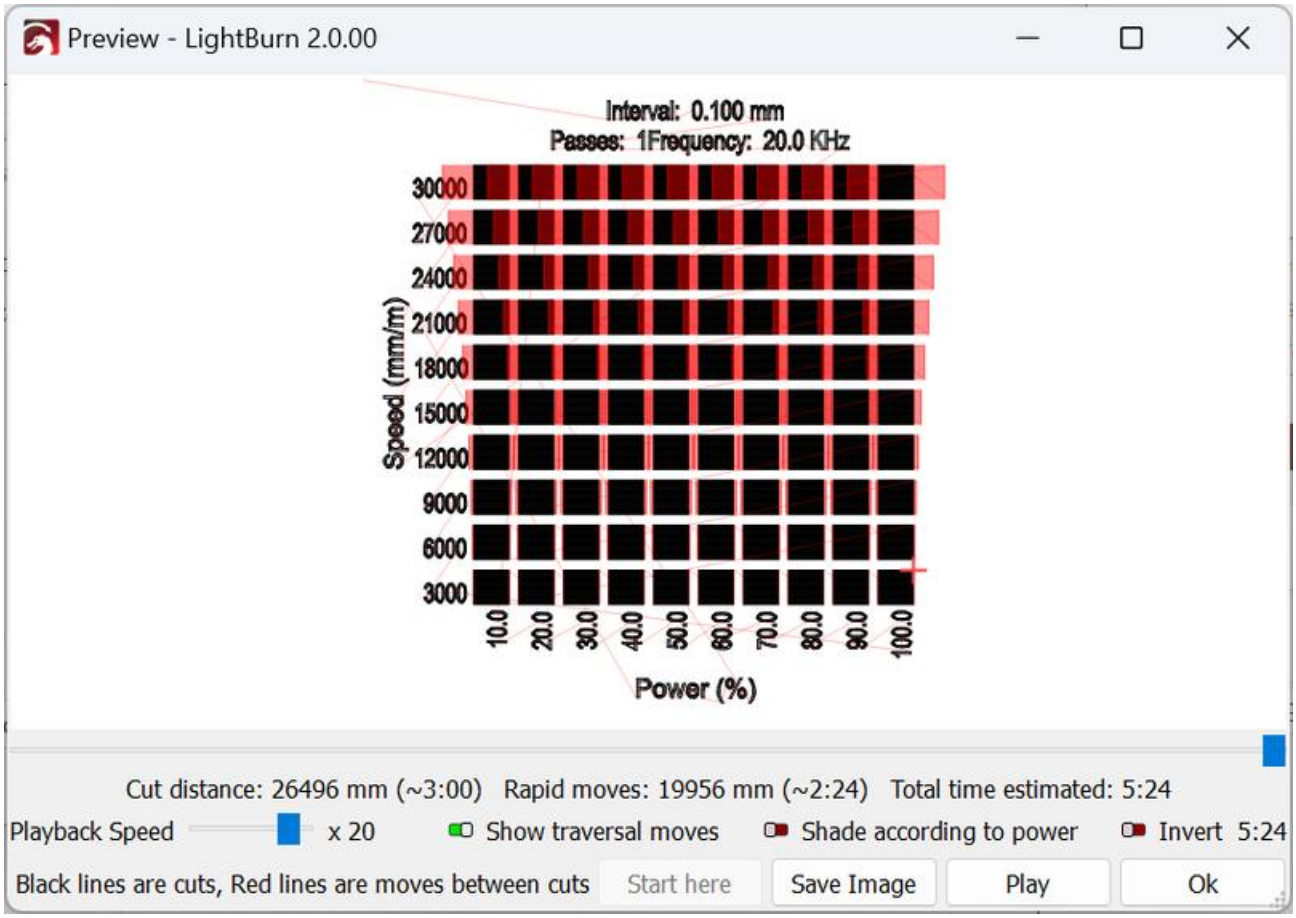
3. Instead of adjusting those fields manually, select a preset from the dropdown menu. You'll see two **Built-In Presets** each for Diode and CO_2 lasers, one **Cut test**, and one **Engrave test**. Select the appropriate Engrave preset for your type of laser.
4. LightBurn will automatically populate the Material Test fields with a range of values that should provide a good starting point for almost all lasers of your selected type.



Previewing

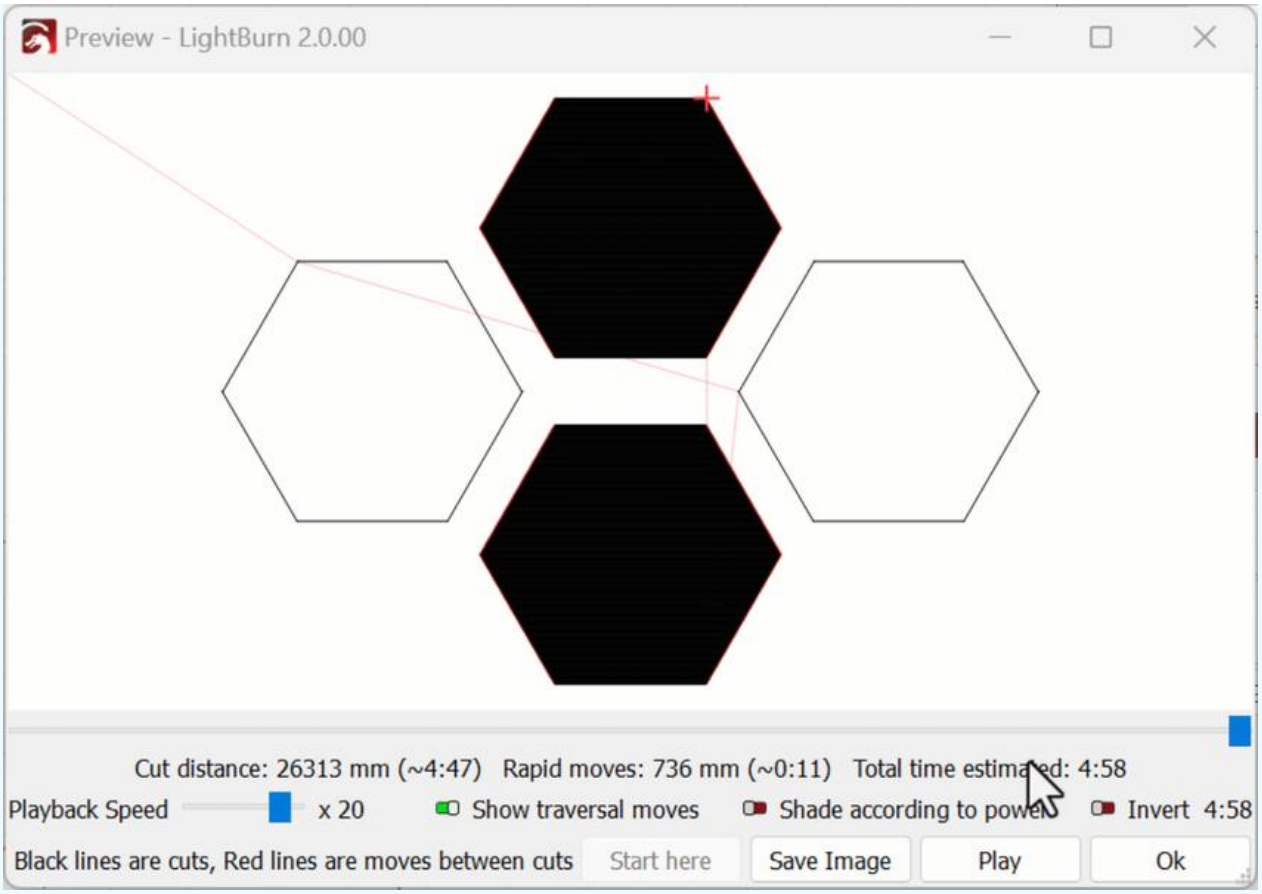
LightBurn has automatically applied all of the important settings for you, but checking the Preview window before you start any job is still a good habit to get into.

1. Click the Preview button to open the Preview window.



2. You'll see a still image of what the final test grid will look like. Press **Play** to see a simulation of the order in which the grid will be engraved.

> **Click here to learn more about the Preview window**
>
> When you first open the Preview window, you'll see a still image of the final result of your project — the contours to be scored or cut by Line Mode will be indicated by black lines, and the areas to be etched or engraved by Fill Mode will be shown as solid black areas, or, if your Line Interval is relatively large (or you zoom in close enough) as parallel lines running across the filled shapes.
>
> You'll also see red lines — these lines indicate the path your laser will travel while not firing. You can toggle **Show traversal moves** at the bottom of the Preview window to switch between showing these moves or not.
>
> Press **Play** to see a simulation of the movements and lasering operations your laser will perform when you start the job. In addition to making sure the laser will outline or engrave the correct graphics and areas, you'll be able to make sure it performs those actions in the order you want.



By default, the simulation will play in a close approximation of the job's real runtime, and **Total time estimated** is shown in the bottom right. You can adjust the **Playback Speed** slider to speed up or slow down the simulation.

3. The order in which test boxes are cut or engraved is determined according to an ascending list of potential risk of burning or charring material.

   In other words, boxes with the highest **Speed** and lowest **Power** will run first, followed by the next highest, lowest, and so on.

Framing

The final check before you run the **Material Test** is to make sure that it will output in the location you're expecting.

1. Click the **Frame** button to tell your laser to trace a box around the area the test grid will be engraved.

   Click here to learn more about Framing

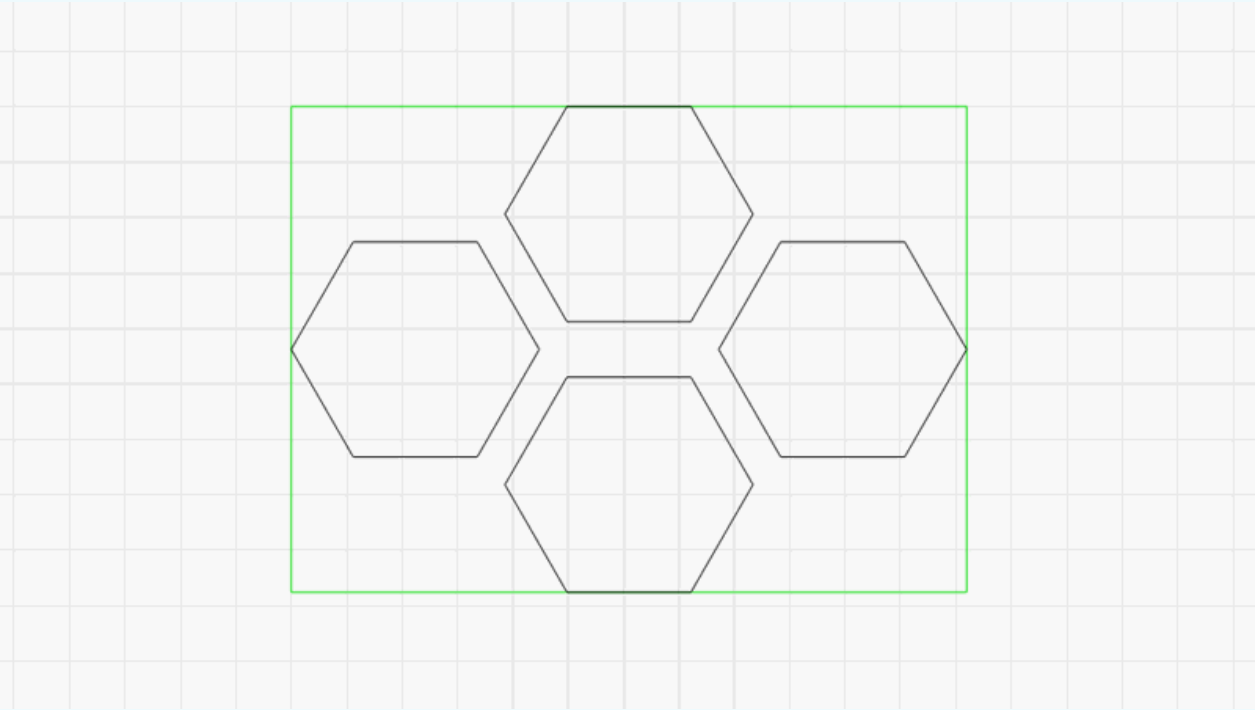
   Bounding Box Frame

   Bounding Box framing

commands the laser to trace a path defined as the smallest possible rectangle that will fully contain all graphics you're sending to the laser.

Here's a representation of that path in LightBurn:

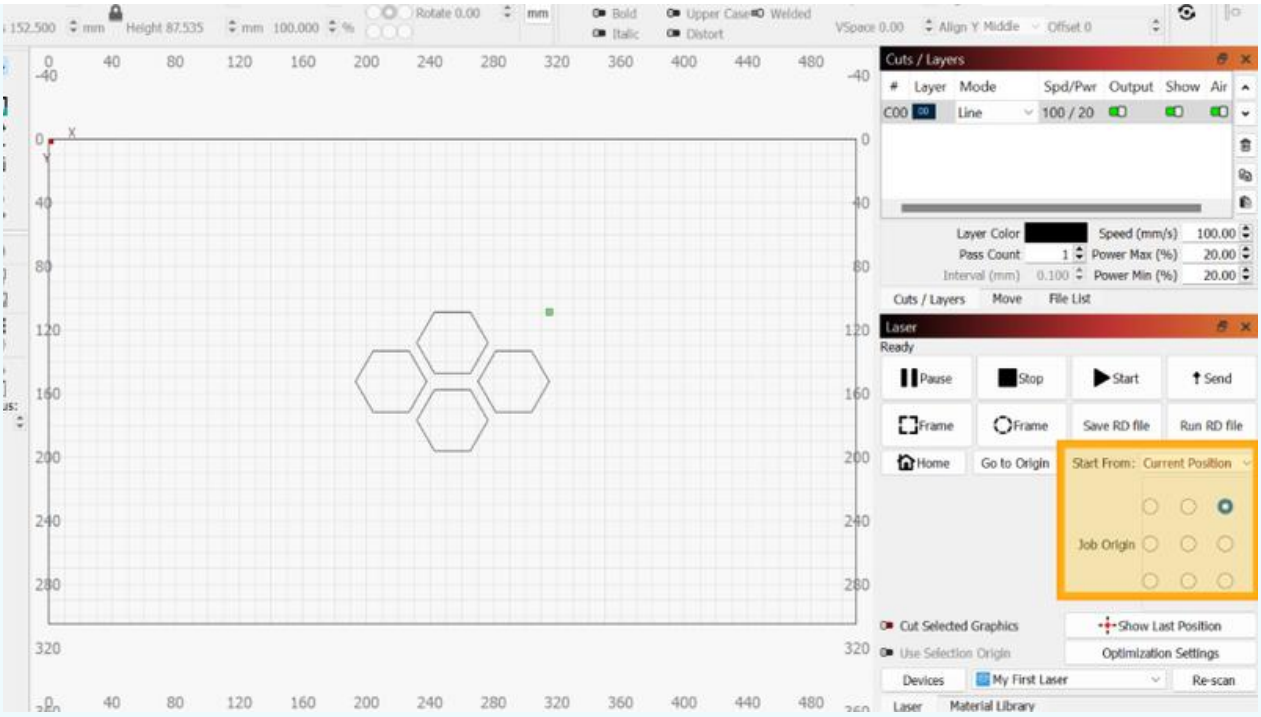


And here's the laser following that same path during Framing:



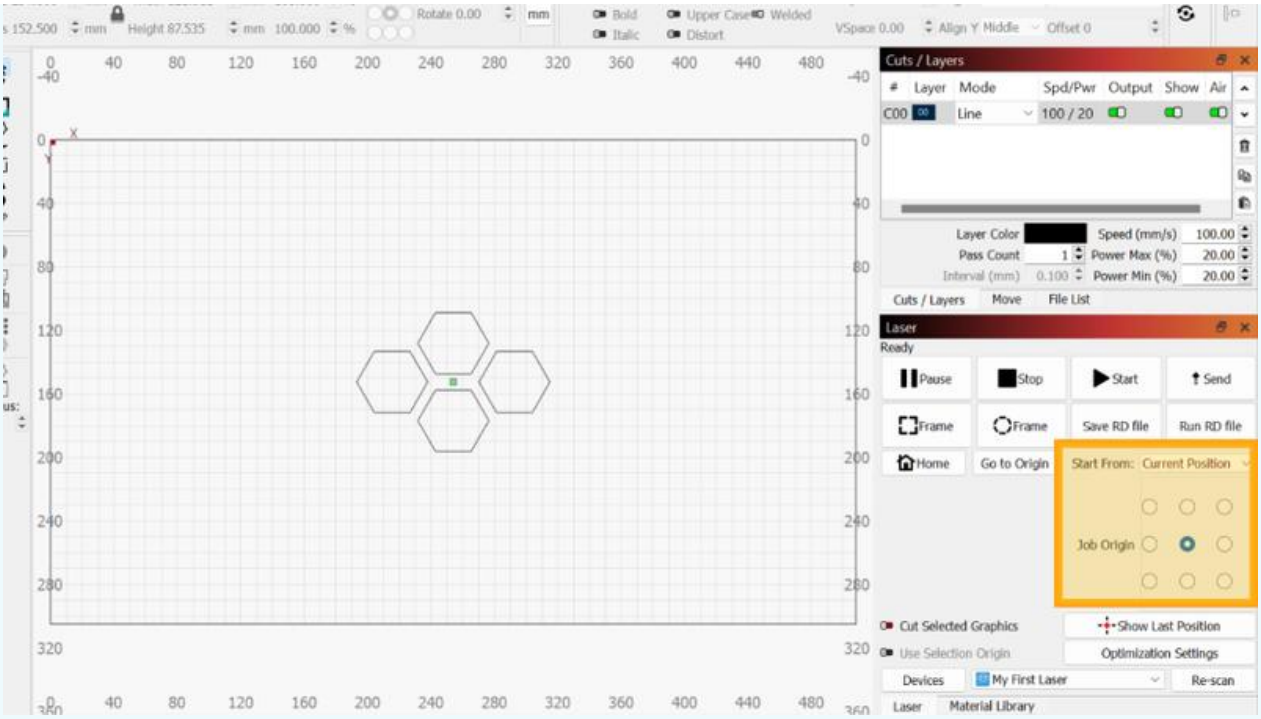
Framing with Current Position

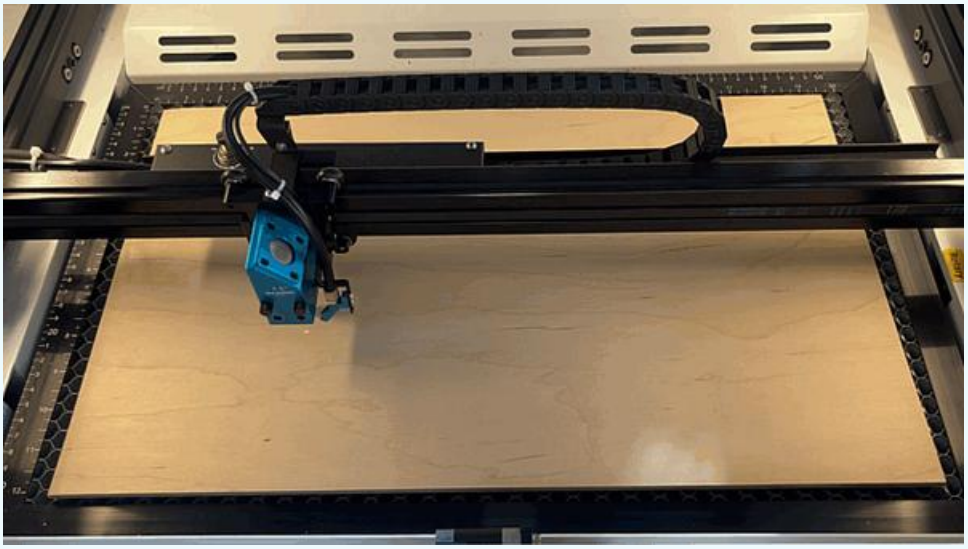
Here we've selected Current Position as the Start From mode, and the top right Job Origin position, so the laser moves down and to the left from its starting position:





Here's Current Position again, but with the center Job Origin position selected — now the laser moves out from where it's sitting, and traces a rectangle centered on the location where it began:





Note for Galvo laser users

If you have a Galvo laser, the grid will be outlined by your laser's red dot light instead. See Galvo Framing for more information.

2. If the area your laser frames is correct, you're ready to start the job. If not, what you do next depends on how far off it is.

Learn More: Tips for improving Framing visibility

> **How to fire your laser at low power during Framing (diode lasers only)**

If your laser has a red dot pointer, it makes framing easier to see. Otherwise, your laser will travel around with the laser turned off. LightBurn does offer the ability to run diode lasers at a very low power to aid in Framing. To do so:

1. Go to Edit → Device Settings to open the Device Settings window.
2. In the Basic Settings tab, toggle on the **Enable laser fire button** and **Laser on when framing** settings. Click **OK** to apply.
3. The Move Window should now have a **Fire** button and a box to enter a power level. Keep this value very low to avoid burning the material — start at 0.25%, and only increase that value only if the beam is not visible.
4. The **Fire** button will toggle the laser on and off. Now you can use the **Frame** command with the laser visible.
5. Adjust the position of the material and/or the laser head until you're happy with the layout of the design.

> **Danger**
>
> The Fire button should only be used for diode lasers, which generally don't have a red dot pointer for Framing.
>
> Always wear proper eye protection when Firing your laser. Consult your laser's manufacturer for information on the proper eye protection required for your laser.
>
> This should never be used for a CO_2 laser, which has an invisble beam that could blind you or start a fire.

Frame Continuously

Frame Continuously is an option in the Device Settings window that tells your laser to repeatedly Frame the same path until you tell it to stop.

Sometimes you won't get the positioning just right on your first try, but a little nudge of your material will do the trick. Other times, you'll just want to want to double or triple check the position. In either case, Frame Continuously will be a big help.

To enable it:

1. Go to Edit → Device Settings to open the Device Settings window.
2. In the Basic Settings tab, toggle on the **Frame Continuously** setting. Click **OK** to apply.
3. Now when you click either of the Frame buttons, a dialog window will open, letting you know that continuous Framing is active, and to press OK when you're ready to stop.

3. If it's close, but a little bit off the material, try nudging the material (if you have room) to align it to the framed area. If you don't have room, you'll need to exit the Material Test Generator window and jog your laser to a better position.
4. If it's very far off, it's likely you didn't select the correct Start From mode or Job Origin position. Revisit the Material Test Positioning step, or see Coordinates and Origin.
5. After making corrections, be sure to Frame again. Repeat as many times as necessary, until you've got everything lined up properly.

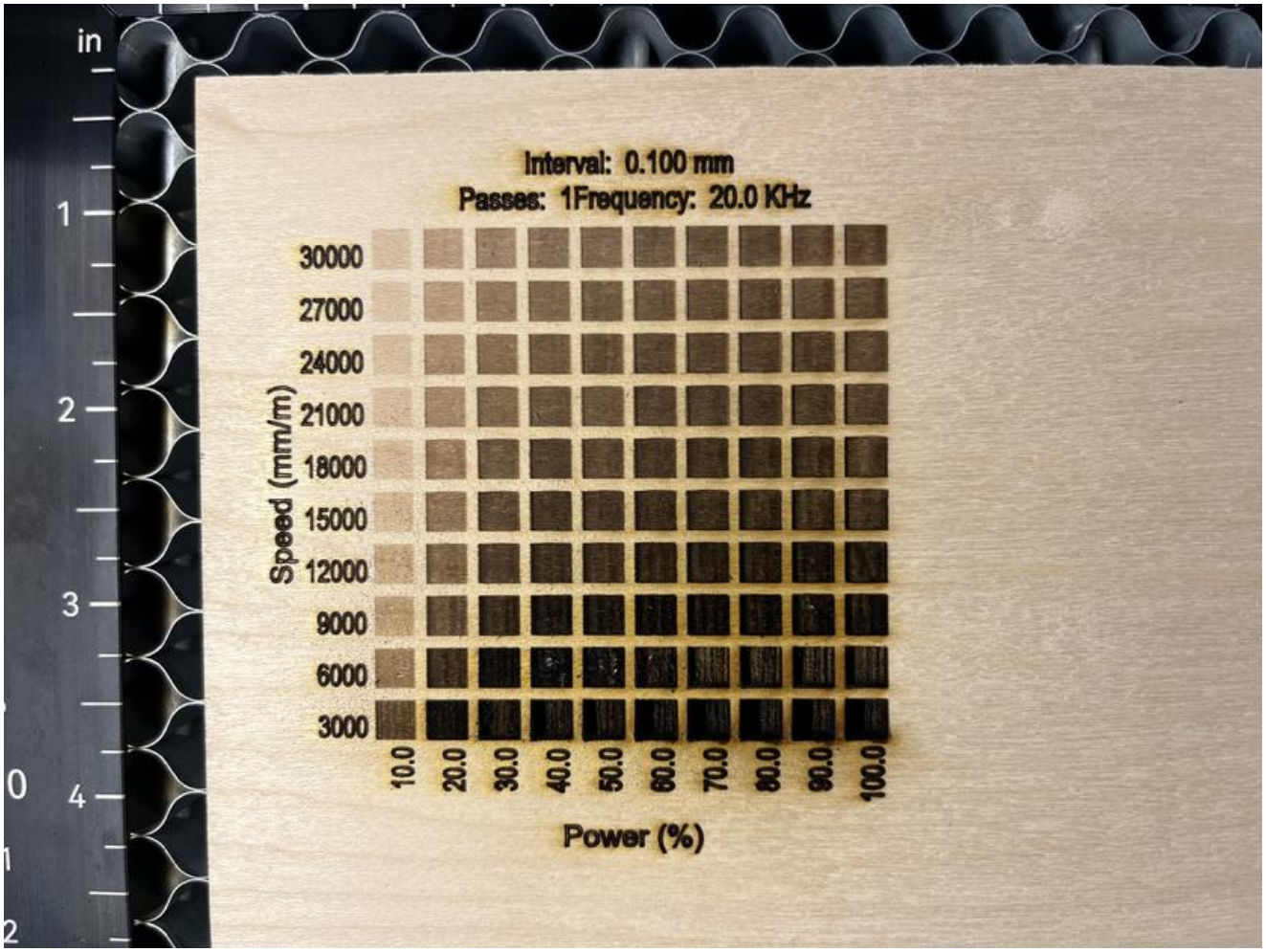
Running the Job

It's time to start your laser — this is the fun part!

1. Press the **Start** button in the **Material Test Generator** window to tell your laser to begin engraving the test grid.

   > **Warning**
   >
   > Never leave a Material Test or any other job running unattended. If you are working with a relatively flammable piece of material such as cardboard, pay special attention when the job reaches the test boxes with the lowest **Speeds** and highest **Powers**. Be ready to stop the job and use fire supression equipment.

2. When it's finished, examine the results — find the box that achieves the nicest engraving and make note of them, apply those settings to a Fill layer in the Cuts / Layers Window, or create and save an entry in the Material Library.



Repeating for Cuts

> **Note for Galvo laser users**
>
> If you have a Galvo laser, its cutting capability is most likely extremely limited. We recommend skipping this test.

Once you've identified optimal engrave settings, you may want to test again for cuts.

> **Avoid damaging your work area**
>
> Since you'll be cutting through the material, make sure that whatever is underneath your material is either scrap that you don't mind burning into or won't be damaged by lasering. This might be a thin sheet of aluminum, a honeycomb bed, or a piece of scrap wood.

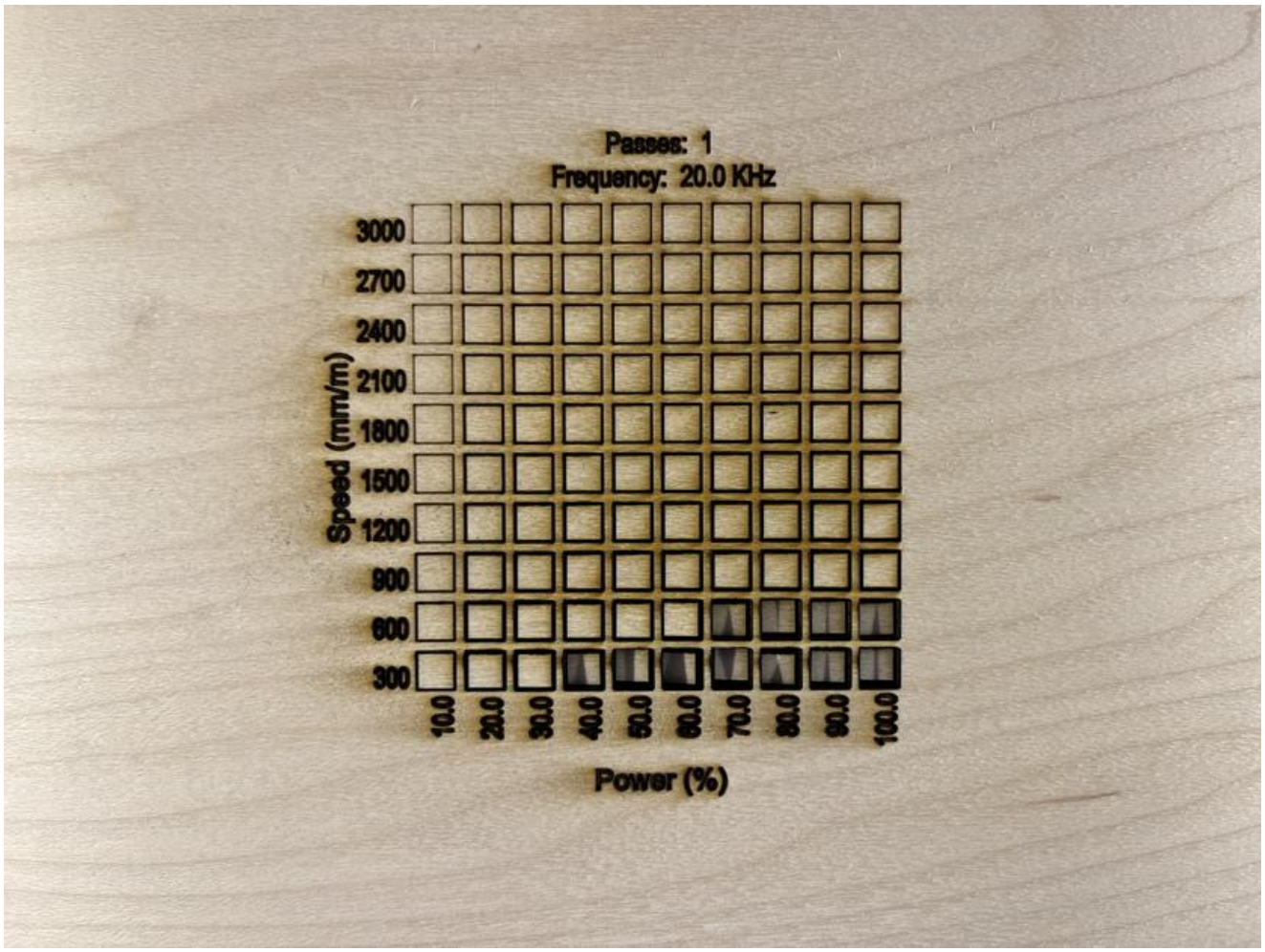
1. Re-position your laser over an un-used section of your material, or place a new piece of material in your laser's work area.
2. Go to Lasers Tools → Material Test to open the **Material Test Generator** again.
3. Select the appropriate **Cut test** preset for your type of laser from the **Presets** dropdown menu.

4. Preview and Frame the job.
5. Run the job.

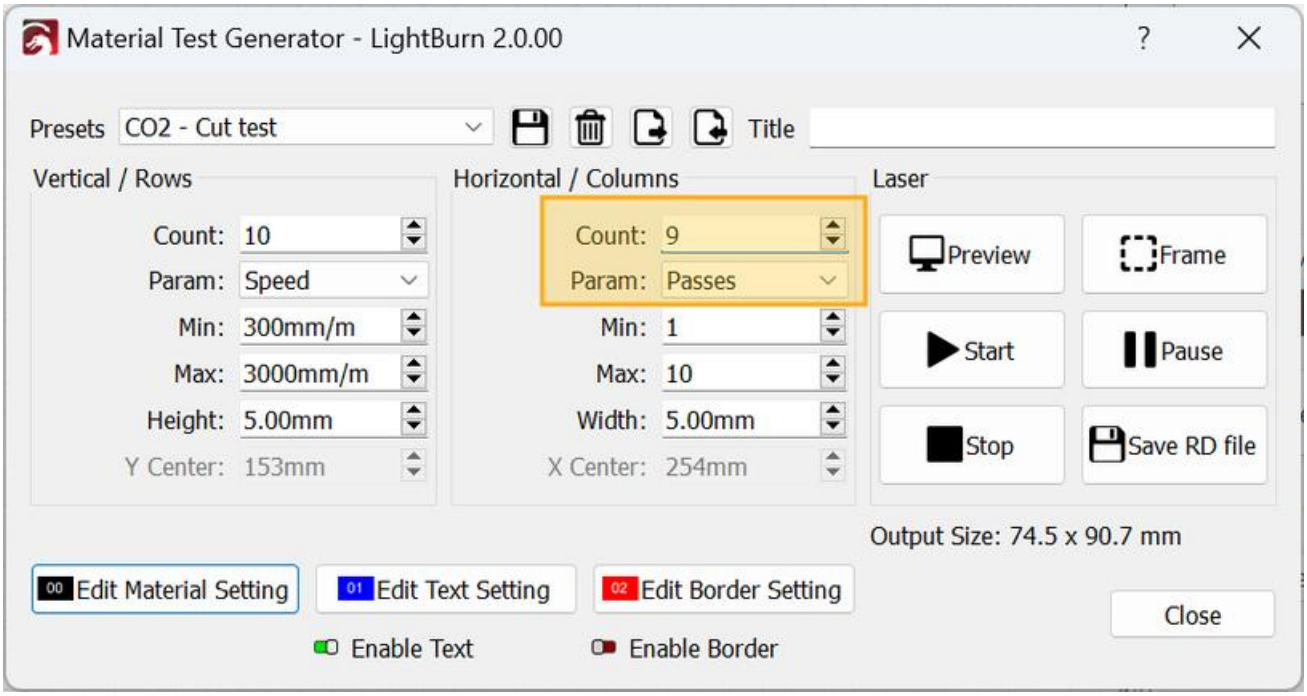
> **Warning**
>
> Never leave a Material Test or any other job running unattended. If you are working with a relatively flammable piece of material such as cardboard, pay special attention when the job reaches the test boxes with the lowest Speeds and highest Powers. Be ready to stop the job and use fire supression equipment.

6. When it's finished, examine the results. You're looking for the box that cuts through the most cleanly, without charring. If several cut through with similar amounts (or lack of) charring, you can select the one that achieved a clean cut at the fastest speed.

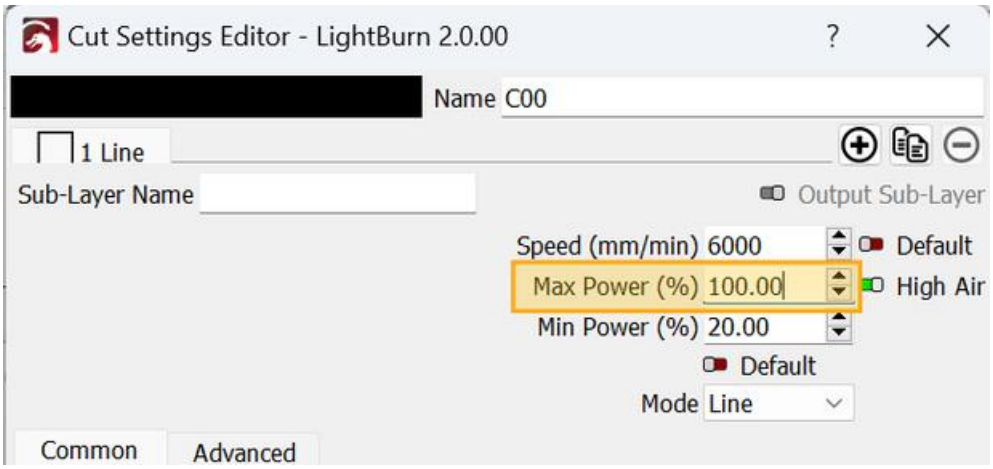
   Make note of those settings, apply them settings to a layer set to Line Mode in the Cuts / Layers Window, or create and save an entry in the Material Library.



7. If none cut through, you'll need to make some manual adjustments.
8. In the second column, open the dropdown menu to replace the **Power** parameter with **Passes**.
9. In the same column, set **Count** to 9, and **Min** to 2.



10. Click **Edit Material Setting** in the lower left to open the Cut Settings Editor, and set the **Max Power** to 100.0%.



11. After Previewing and Framing, run the test again.

> **Warning**
>
> Never leave a Material Test or any other job running unattended. If you are working with a relatively flammable piece of material such as cardboard, pay special attention when the job reaches the test boxes with the lowest Speeds and highest Powers. Be ready to stop the job and use fire supression equipment.

12. Select the box that cut through with the fewest Passes and highest Speed. Make note of those settings, and, along with the 100% Max Power setting, apply them to a Line layer in the Cuts / Layers Window, or create and save an entry in the Material Library.

    If none of the boxes cut through, your laser most likely isn't capable of cutting through the material you're testing.

First LightBurn Project

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=zfEt-kazd_w

One of the best ways to learn to use Lightburn is to use it. We recommend following along with our laser-engraved QR Code WiFi coaster to start exploring LightBurn. By the end, you'll be familiar with LightBurn's user interface and general workflow, and you'll have a coaster your friends and family can scan to log into your WiFi network without needing to type anything in.

The video above shows the same project in an older version of LightBurn. The overall process is the same, but some tools will look a little different, and the QR Code tool is replaced by the Bar Code tool.

Before you begin, familiarize yourself with your laser and how to operate it safely according to manufacturer instructions. Never run your laser unsupervised.

What You'll Need

- LightBurn installed on your computer.
- A laser added to LightBurn.
- Suitable material for the coaster. Our video uses laser-safe plywood, but you can use any laser-safe material you have on hand if you don't have that available.

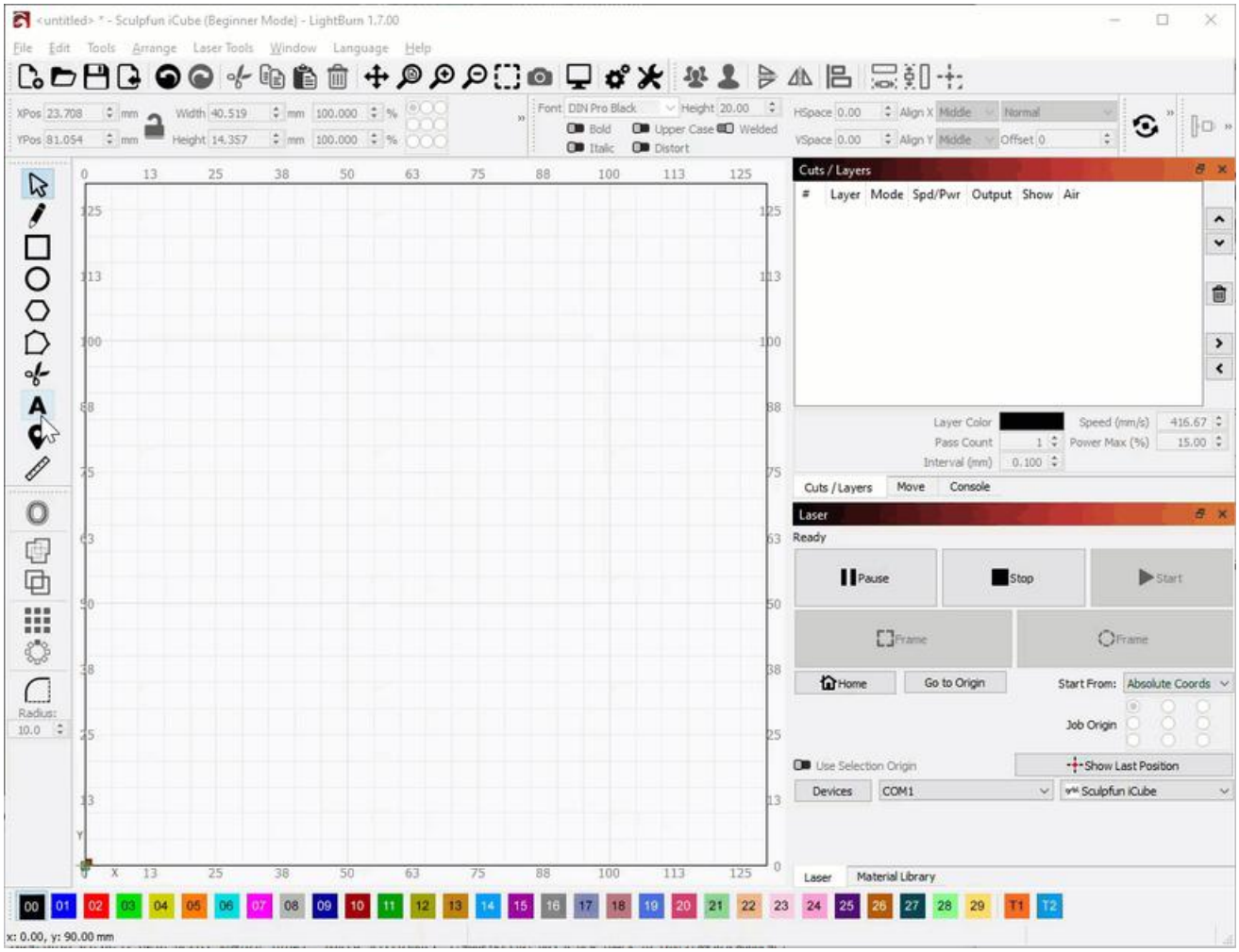
  > **Note**
  >
  > You can use a pre-made coaster blank instead of cutting out your own. Some lasers — including most Galvo lasers — aren't capable of cutting through any materials.

- Your WiFi information to enter into the QR Code.

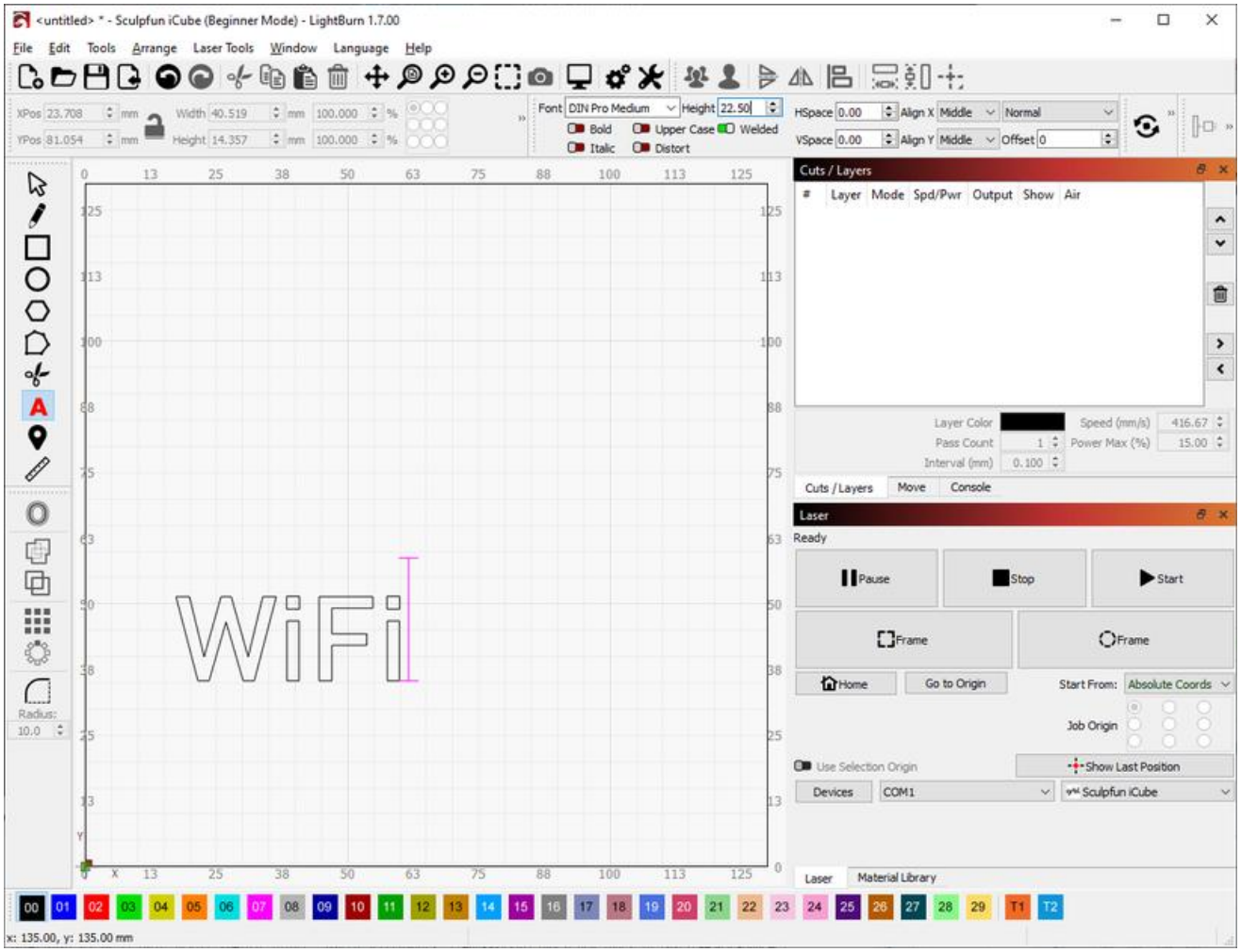
Designing Your Coaster

Create Text

You're going to start by creating text to engrave onto your coaster.

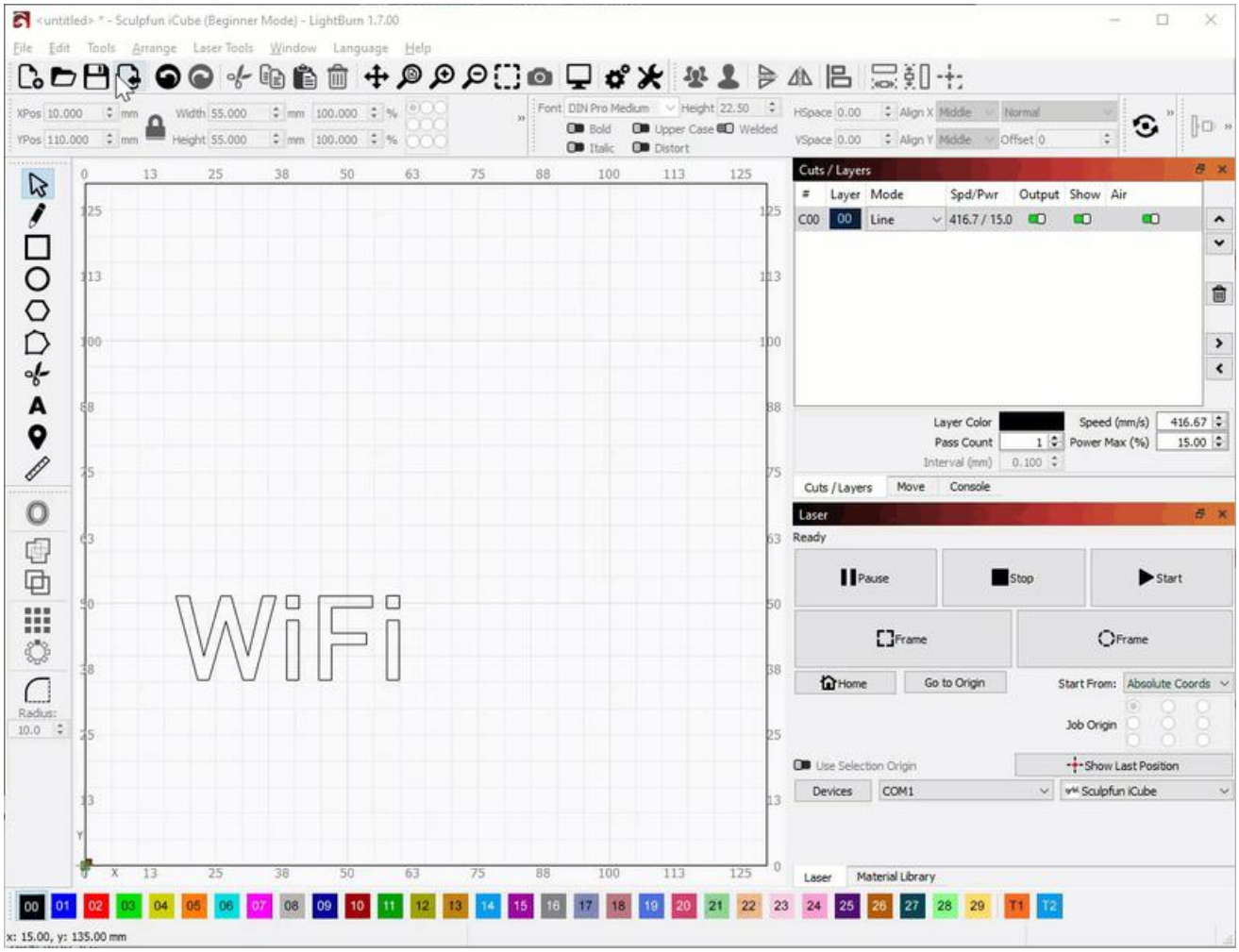


1. Click on the Text tool **A** in the Creation Toolbar.
2. Click in the Workspace to place a cursor and type something out. Our demo coaster will say "WiFi", since that's what the QR Code will be for.
3. Find the Text Options Toolbar near the top of the screen above the Workspace. When you have text selected, this toolbar becomes active, and provides several options for modifying the selected text. For this project, try the following steps:
   - Select a font you like from the **Font** dropdown. LightBurn can use any font installed on your computer.
   - Adjust the size with the **Height** box.
4. When you're done typing, return to the Select tool by pressing `Esc` or clicking the cursor icon in the Creation Toolbar. You can come back and edit the text later by double-clicking it with the Select tool active.



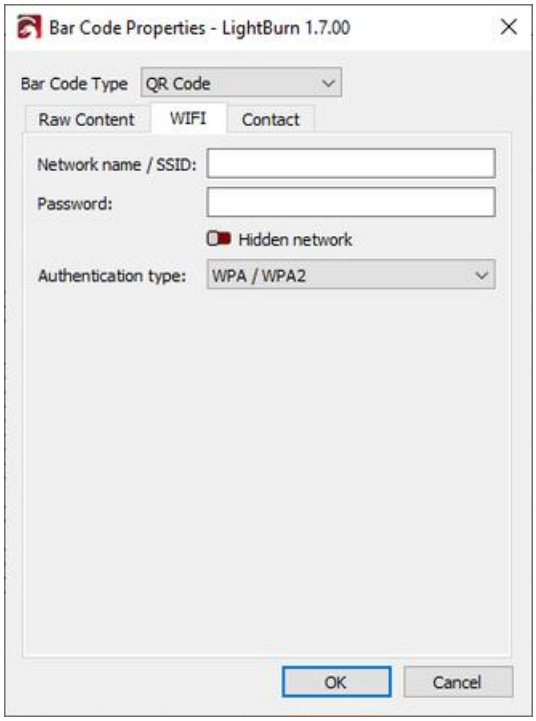
Create a QR Code

LightBurn has a built-in Bar Code tool, which can generate QR Codes as well as a variety of other scannable codes. You'll use this to generate a QR Code that automatically logs someone into your WiFi network.

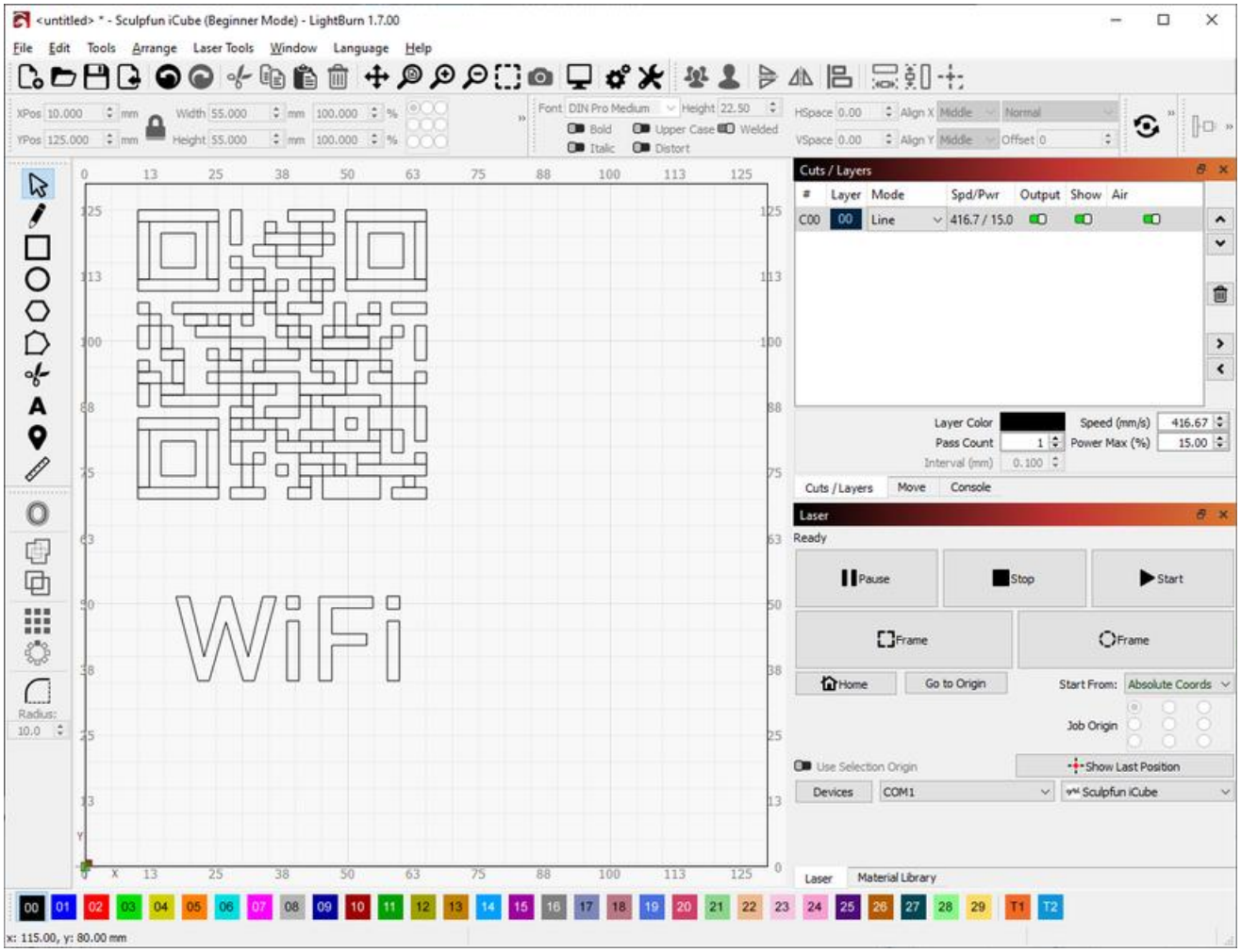


1. Go to Tools → Create Bar Code to open the Bar Code tool. You'll see the cursor change to a QR Code. Click and drag in the Workspace to set the initial size of your QR Code and start creating it. Put it somewhere near where you want it — you'll improve the placement in the next step.

2. Make sure the **Bar Code Type** is set to **QR Code** and click on the **WIFI** Tab.

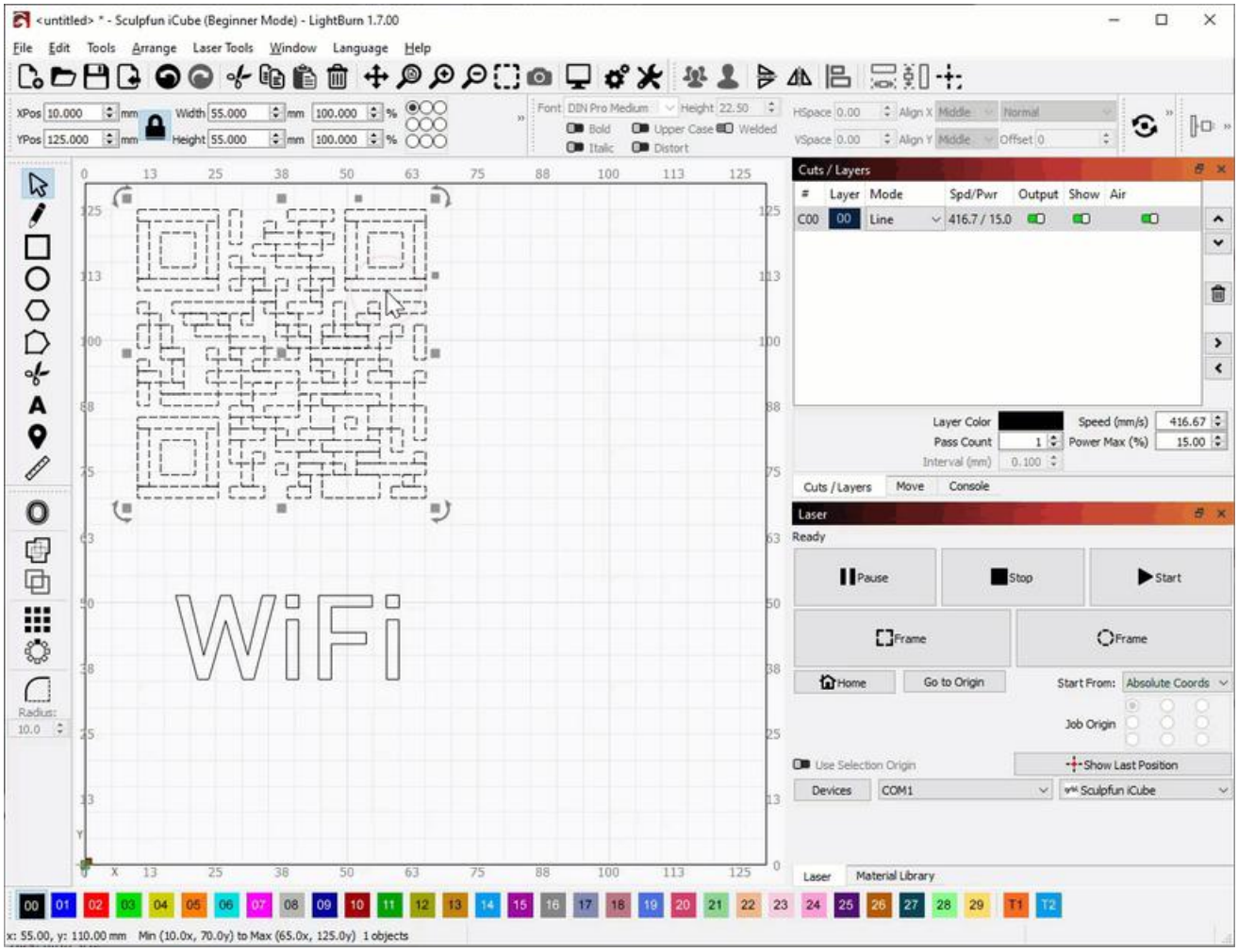


3. Enter your WiFi login info into the window – be very careful with your spelling and capitalization – and click **OK** to create the QR Code.
4. Adjust the size with the **Width** and **Height** boxes in the Numeric Edits Toolbar. Make sure the lock icon is closed so the code's aspect ratio is maintained as you make adjustments, and it doesn't get distorted. Enter a size into either Width or Height — somewhere between 40 and 60 mm is generally a good size.



Aligning and Grouping

Now that you have text and a QR Code, you'll want them to line up nicely. This is a great opportunity to try out the Align and Group tools.



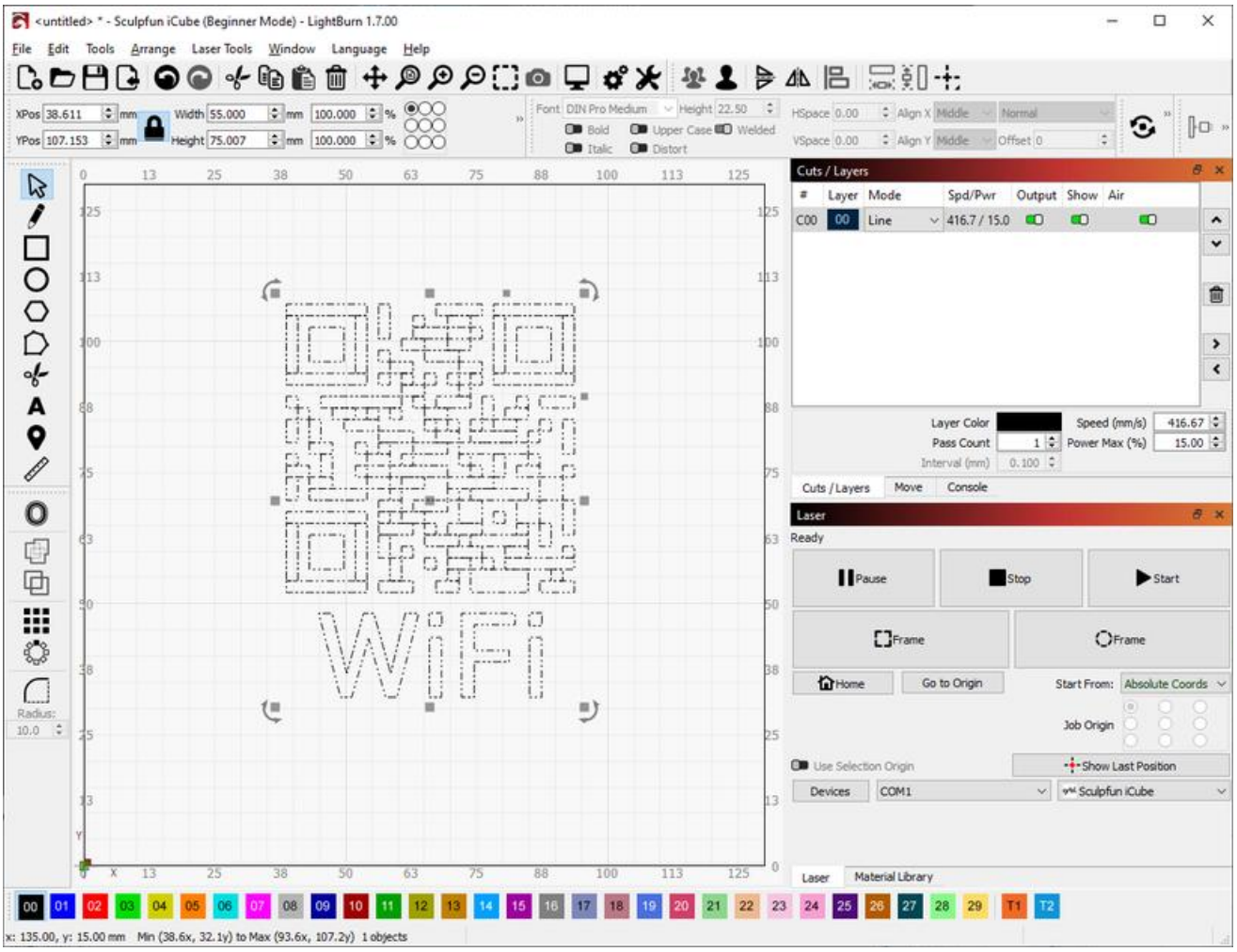
No Alignment Tools Option?

New LightBurn installations start in Beginner Mode, which simplifies the user interface by hiding less-used options. Beginner Mode combines all of LightBurn's alignment tools into the Align Tools Helper.

If you aren't in Beginner Mode, go to **Arrange → Align → Align V-Center** for this step, or enable Beginner Mode in your Settings.

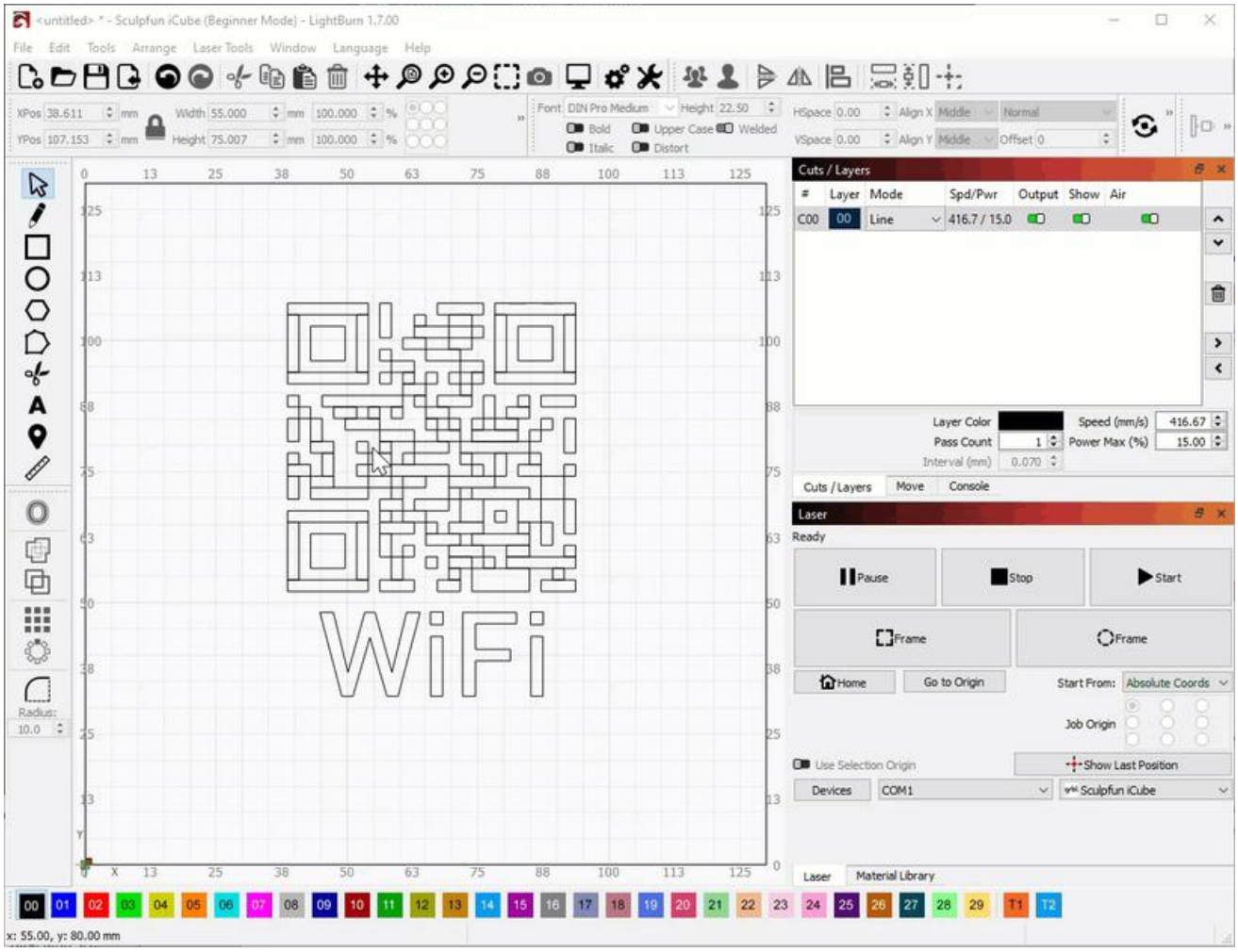
1. Make sure you have the Selection tool active by clicking on the icon or pressing `Esc`.
2. Select the QR Code by clicking on it. Drag to place it where you want it relative to the text. You can quickly spot selected objects by looking for the animated "marching ants" pattern on the outline of the shape.
3. With the QR Code still selected, hold down the `Shift` key and click on the text to select that as well.
4. Go to Arrange → **Alignment Tools** to open the Align Tools Helper, which will preview different alignment options as you hover over them. Use the bottom **Align Centers** button to vertically center both shapes.

5. With both objects still selected, press `Ctrl`/`Cmd`+`G` to Group them. This way both shapes will move together without having to remember to select them both. To move them individually, you'll need to **Ungroup** them by selecting them and pressing `Ctrl`/`Cmd`+`U`.



Layer Modes

Now we'll preview your design and edit your cut settings. We'll be introducing the Preview function and the Cuts / Layers Window.



View Styles

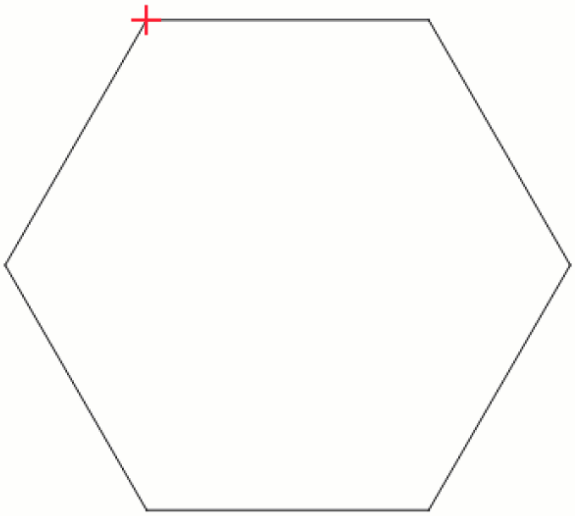
LightBurn offers **Wireframe** and **Filled** options for the **View Style**. After setting Layer 0 to Fill, if you don't see the shape filled in, you can use `Alt` / `Option`+`W` to toggle between Wireframe and Filled view styles. Read more at View Style Types.

1. Open the Preview window with `Alt`/`Option`+`P`. This window shows your design as it will be sent to the laser, and can even simulate the engraving to estimate time and visualize what the laser will be doing. Try pressing play to see how it works.
2. By default, your design will be on Layer 00 and set to Line Mode, which means the laser will trace the outside of the shapes rather than filling them in. Click on the dropdown under **Modes** and select Fill. Next to **Speed** and **Max Power**, set appropriate speed and power settings for the material you're using. Check your laser's user guide for recommended settings.

Learn More: Line Mode versus Fill Mode

Line Mode tells your laser to trace a path around the contours of vector graphics.

If you command the laser to move quickly, or with low Power, it will likely just etch the surface (sometimes called scoring). If you move slowly and with high power, especially with a CO_2 laser, the laser will cut deeper, or even completely through the material. The only difference between surface marking and cutting with Line Mode is the Power and Speed you set.



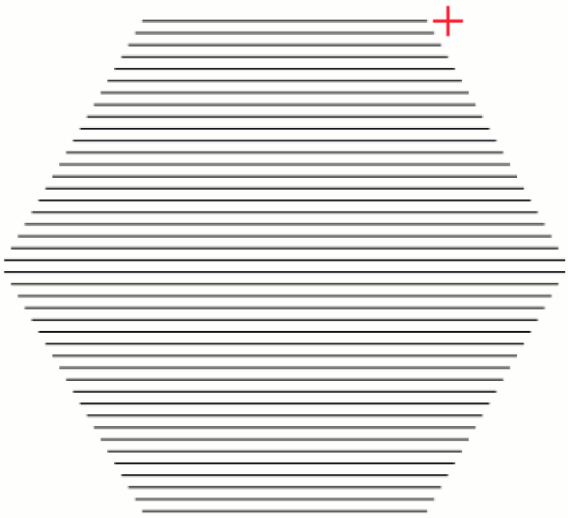
Closed Shapes Only

This feature only works with shapes that are closed. A shape is closed when it is a complete, continuous loop whose start and end points are the same. When a shape's start and end points are different, it is open.

See Open vs. Closed Shapes for more information.

Fill Mode tells your laser to etch parallel lines within the boundaries of vector graphics.

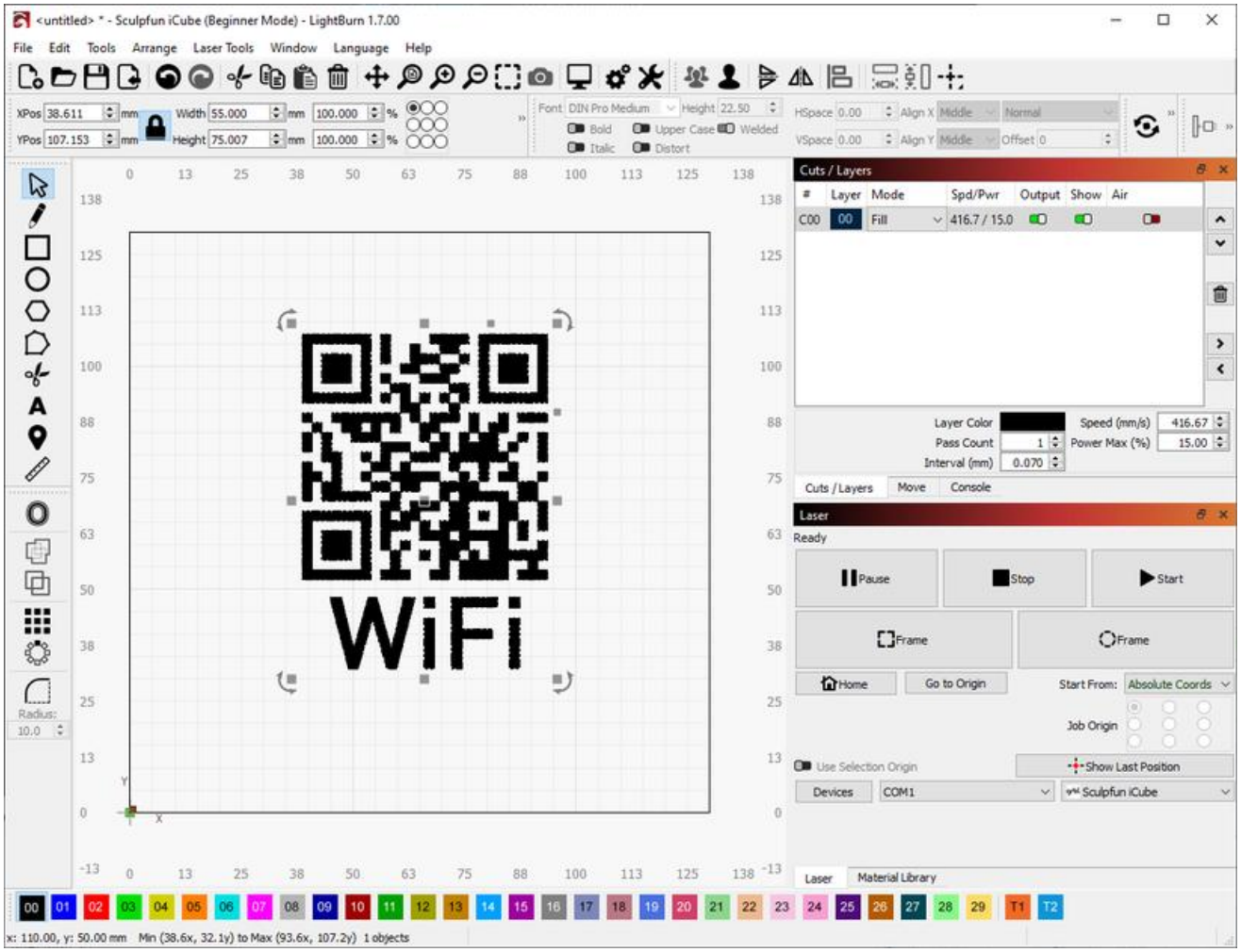
The distance between each line is determined by the Line Interval setting, and the Speed and Power settings control the darkness or depth of the etch. Use low Intervals, low Speeds, and high Powers for deep engravings or dark marks, or tell your laser to move quickly at low Power to lightly etch or mark the surface.



3. Turn off the **Air** toggle to disable air assist for this layer. It's helpful for cutting, but less useful for engraving.

> **Learn More: Air Assist**
>
> Air assists are a feature of some types of lasers that blow air onto the surface of your material at the spot where the laser's beam strikes, which can help prevent soot accumulation and charring, and aid in cutting processes.
>
> Some lasers have automatic assists, which LightBurn can turn on or off when a job begins. Other models have external air assists which you must enable manually when you start a job.
>
> If your laser has a manual air assist or no air assist at all, toggling the Air switch will have no effect.

4. Preview the design again, and you should see the design filled in instead of outlined.

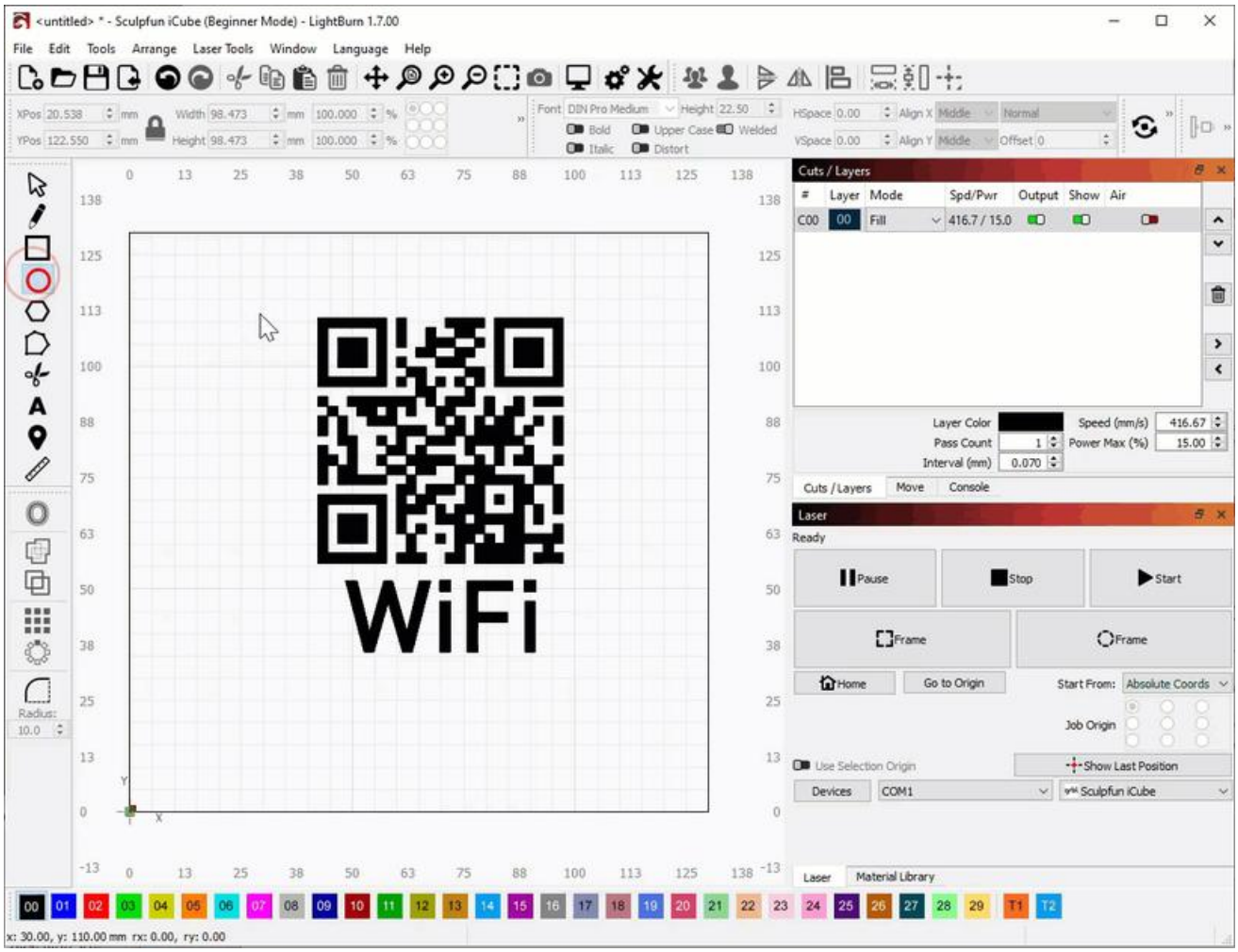


Create a Decorative Border

With the QR Code and text ready to engrave, we'll move onto a different style of engraving.

When you set Layer 00 to **Fill**, the preview shows that the laser will move back and forth over the area to engrave, very much like printing on paper. A raster engraving like this is the most common way to engrave, but you can also engrave (or score) in **Line Mode** by using a lower power and higher speed than you would use to cut through the material. This tends to be faster and produce cleaner results on thin lines than a raster engraving. It's particularly effective with single line fonts.

Draw and Position the Circle



1. Select the Ellipse tool from the Creation Toolbar.
2. Click near the top left of your QR Code and text to start drawing the circle. Drag toward the bottom right, holding down the `Shift` key as you go to keep the width and height the same, and then release to draw the circle. Don't worry too much about the exact size and position yet.
3. Press `Esc` to exit the Ellipse tool. This should leave the circle you just drew selected. Hold down the `Shift` key and click to select the Grouped text and QR Code as well.
4. Center the circle around the rest of your design by going back to Arrange → **Alignment Tools** and choosing **Align Center**.
5. If you need to adjust the size of the circle, use the **Width** and **Height** boxes in the Numeric Edits Toolbar to adjust. You may need to repeat Step 4 after doing this.

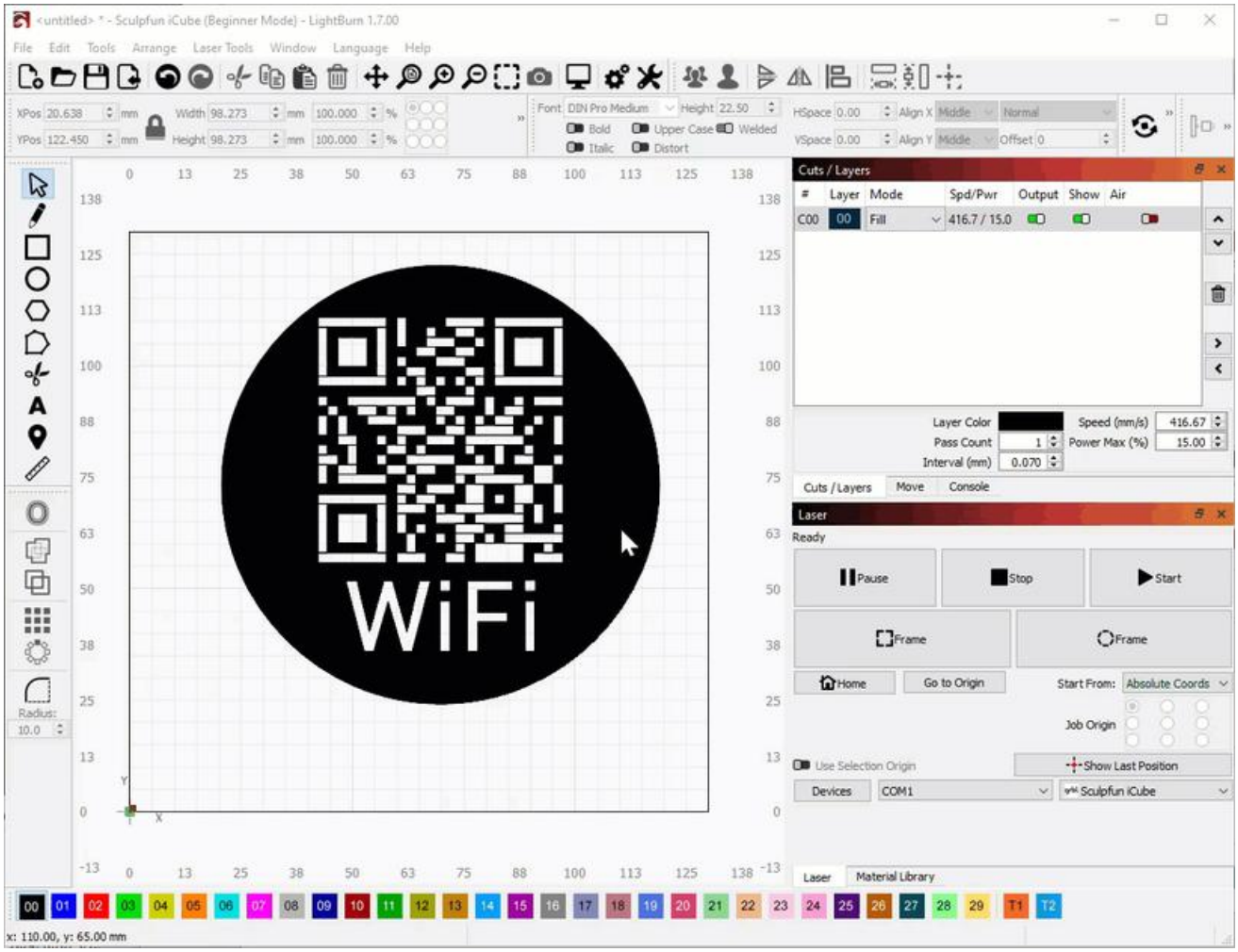
Layers and Cut Settings

> **Always pay attention to units of distance and time when entering Speed settings**
>
> When entering **Speed** values recommended by your laser's manufacturer or other LightBurn users, make sure to use the same units of distance and time as those from the recommendation, or to convert the values to your preferred units.

A given number of **millimeters per second** is much faster than that same number in **millimeters per minute**. Mixing up units can lead to reduction in power output due to unexpectedly high speeds, or excessive power output — and even fire — due to unexpectedly slow speeds.

Change your Control Units in the **Dimensions / Units** tab of the **Device Settings** window. LightBurn automatically converts any existing values when you switch between units.

Right now, your circle is on the same layer as the QR Code and text. This means it'll be filled in like them as well. We'll need to move it to a different layer and give it its own cut settings.



My design is reversed!

A filled shape inside of another filled shape will act as a cutout, similar to a donut hole. We'll fix this in a moment. You can also see a more detailed explanation in How to Invert a Vector Engraving.

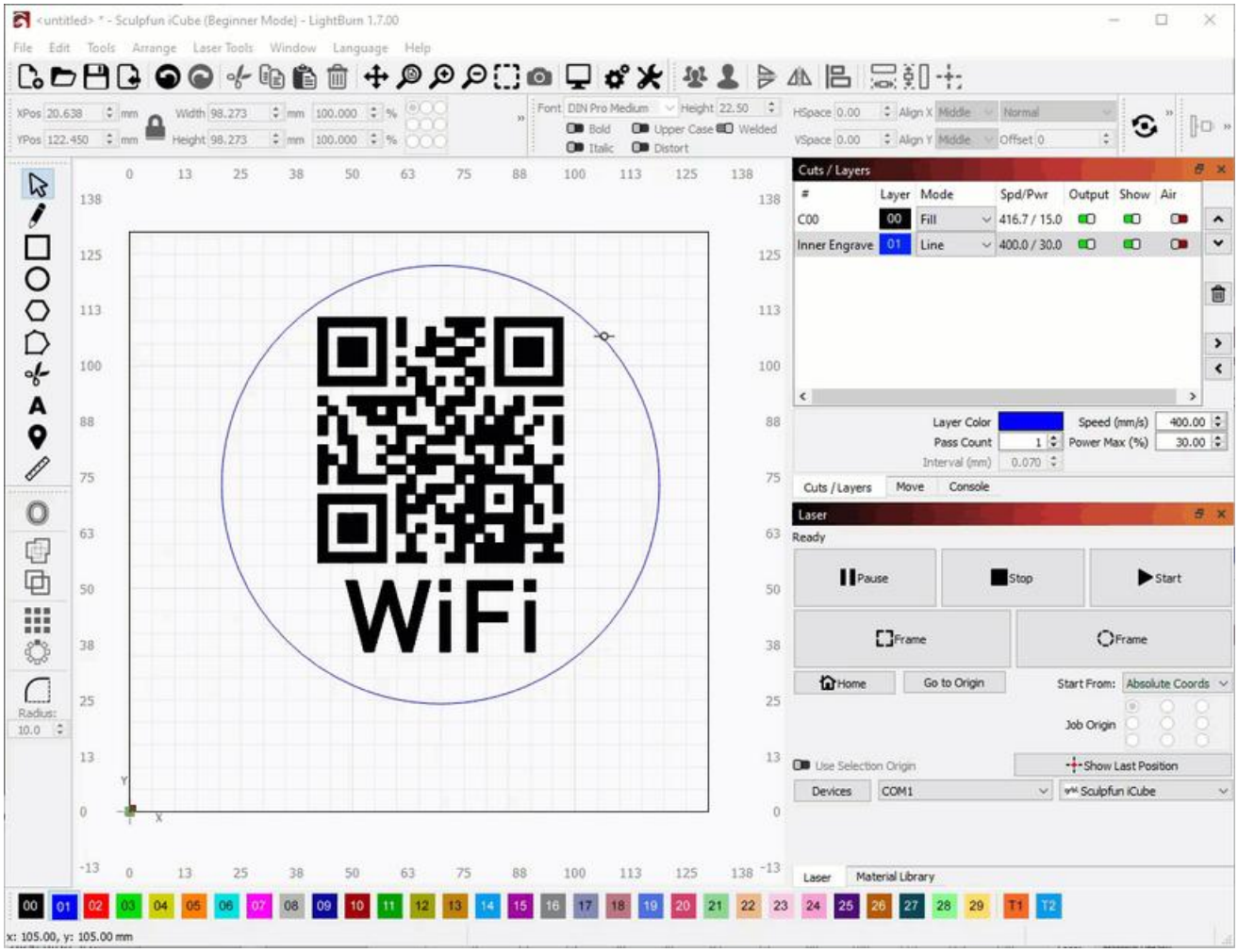
1. Make sure the circle – but nothing else – is selected.
2. At the bottom of the main window, look for the Color Palette. Select a layer other than Layer 00 or the Tool Layers, **T1** and **T2**.

3. In the Cuts / Layers Window, adjust the **Speed** and **Max Power** settings to a higher speed and lower power than your manufacturer recommends for cutting through the material you're using. This will let you engrave (or score) a line without cutting through.
4. Preview your design. Your QR Code and text should still be filled in, but the circle should be just a line that the laser travels along.

Creating the Cut Line

> **Note**
>
> If you're using a pre-made coaster blank, you can skip this step.

The last piece of the design is the circle to cut out. You could do the same thing we did for the previous step – draw a circle and center it with the Alignment tools – but this is a wonderful opportunity to get familiar with the Offset Shapes tool.



1. Make sure the circle is selected.
2. Go to Tools → **Offset Shapes** or press `Alt`/`Option`+`O` to open the **Offset** window.
3. Make sure the **Direction** is set to **Outward** and **Optimize / Simplify results** is enabled. Adjust the **Distance** so that it looks good to you – LightBurn will give you a live preview as you adjust it.

Somewhere around 6 mm or ¼" should work well, but this is your coaster and it's up to you. Press **OK** to create the new circle.

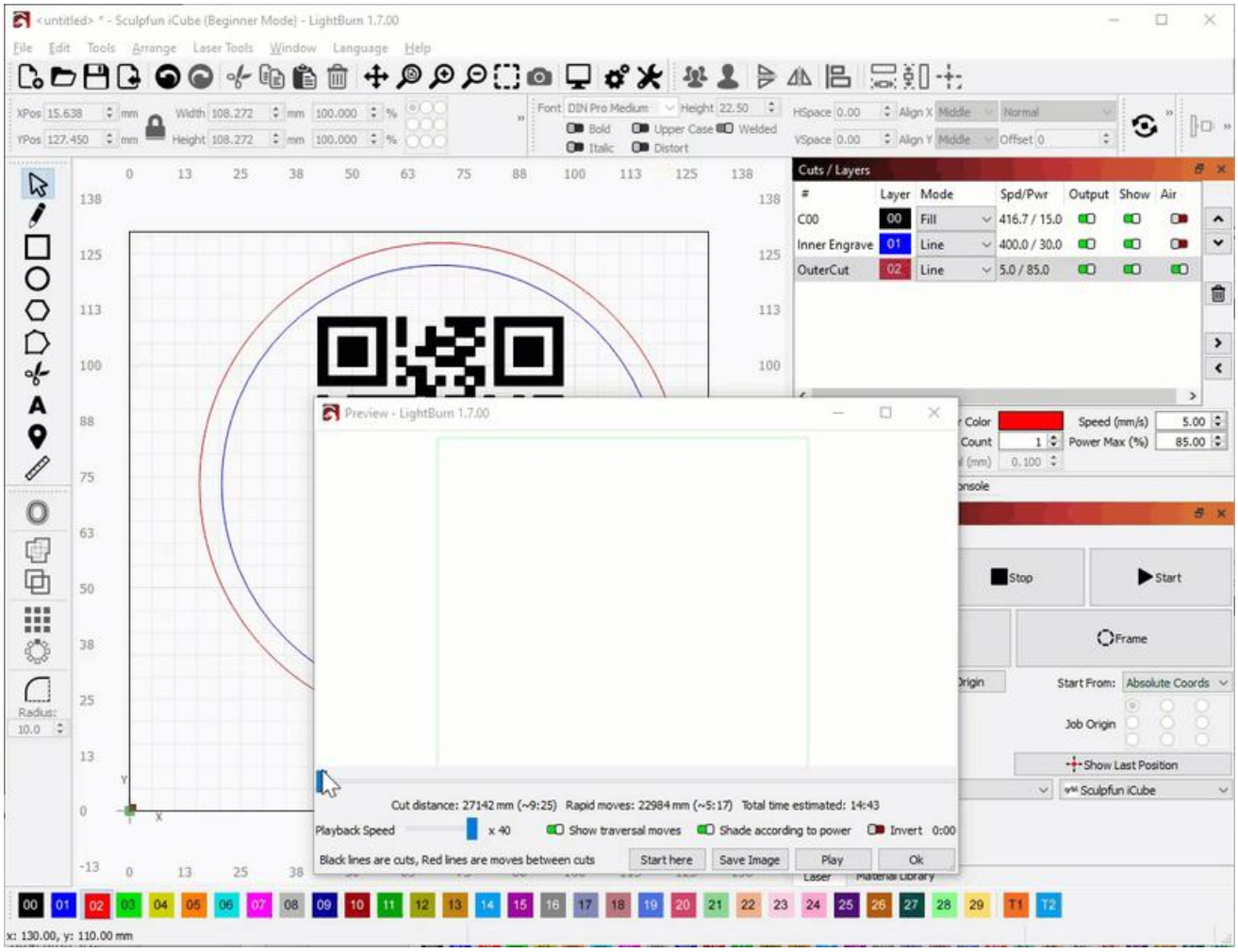
4. As you did before with the first circle, move this circle to a different layer and adjust the cut settings.
5. See your laser's user guide for recommended speed and power settings to cut through the material you're working with.
6. If your laser has an air assist, keep the **Air** switch on to enable air assist for this layer. Air assists are helpful during cutting operations.

> **Learn More: Air Assist**
>
> Air assists are a feature of some types of lasers that blow air onto the surface of your material at the spot where the laser's beam strikes, which can help prevent soot accumulation and charring, and aid in cutting processes.
>
> Some lasers have automatic assists, which LightBurn can turn on or off when a job begins. Other models have external air assists which you must enable manually when you start a job.
>
> If your laser has a manual air assist or no air assist at all, toggling the Air switch will have no effect.

Previewing Your Design

> **Tip**
>
> It can be tempting to start a design without previewing it, especially when you're in a rush, but it's one of the best ways to catch errors before you start burning your material.

Now that your design is complete, preview it one more time. Remember that you can open the Preview window with `Alt`/`Option`+`P`. Press **Play** to preview what your laser will do. You can use the slider at the bottom to speed up or slow down the simulation.



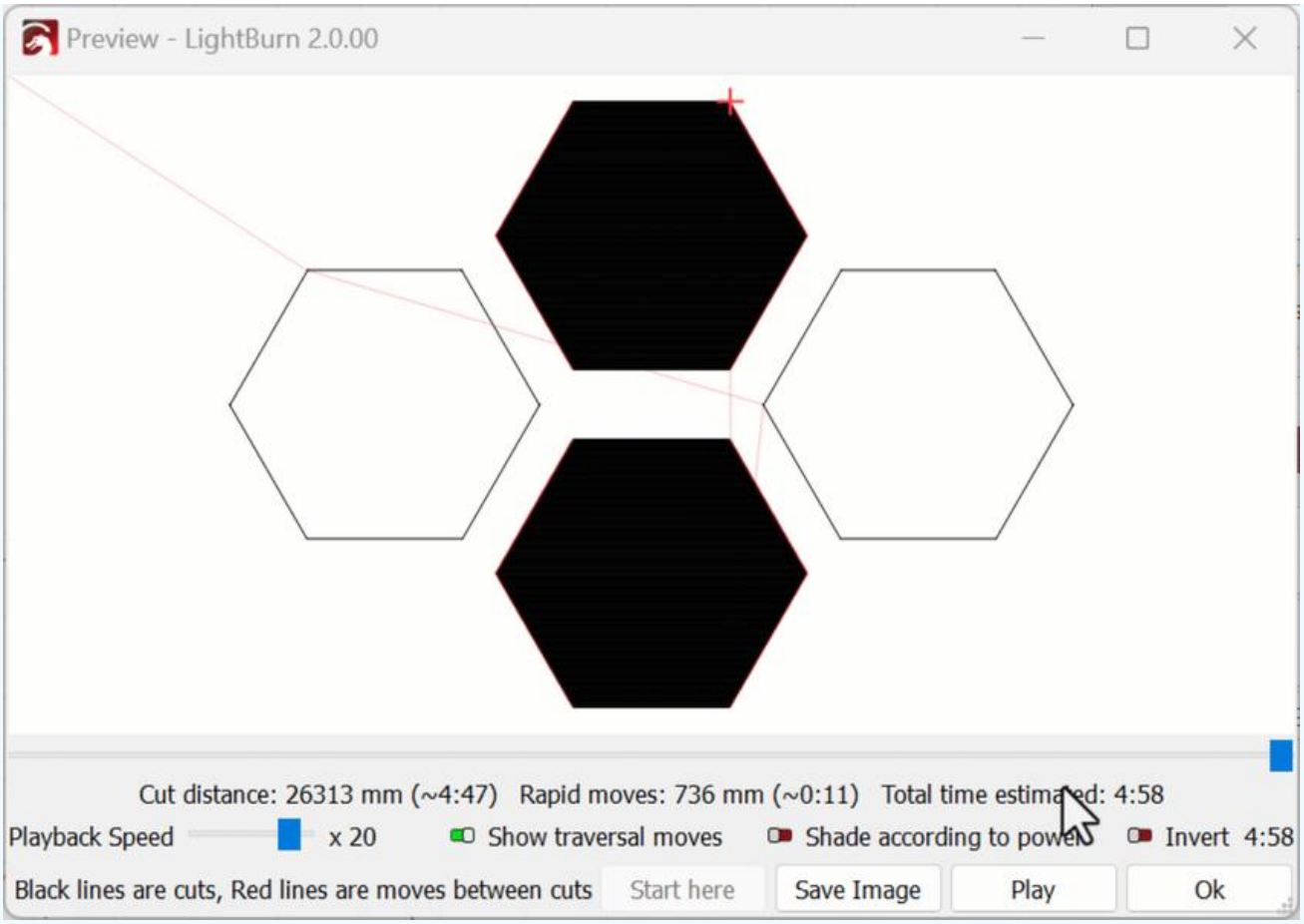
In the simulation, you should see the following happen in order:

1. Raster engraving to fill in the text and QR Code.
2. Vector engraving the decorative border — the laser should follow the outline of the circle.
3. Vector cutting the outer circle.

It's generally best to do engravings before cutting out your design. If you followed along with the steps above, your layers should already be in the right order. If they aren't, you can drag them around in the Cuts / Layers Window to change the order – by default, LightBurn will run each layer in order from top to bottom.

If you make changes, preview them again before moving on to cutting your design.

> **Learn More: Preview**



When you first open the Preview window, you'll see a still image of the final result of your project — the contours to be scored or cut by Line Mode will be indicated by black lines, and the areas to be etched or engraved by Fill Mode will be shown as solid black areas, or, if your Line Interval is relatively large (or you zoom in close enough) as parallel lines running across the filled shapes.

You'll also see red lines — these lines indicate the path your laser will travel while not firing. You can toggle **Show traversal moves** at the bottom of the Preview window to switch between showing these moves or not.

Press **Play** to see a simulation of the movements and lasering operations your laser will perform when you start the job. In addition to making sure the laser will outline or engrave the correct graphics and areas, you'll be able to make sure it performs those actions in the order you want.

By default, the simulation will play in a close approximation of the job's real runtime, and **Total time estimated** is shown in the bottom right. You can adjust the **Playback Speed** slider to speed up or slow down the simulation.

See Preview Troubleshooting for help identifying and resolving any potential issues you spot in the Preview window.

Engraving and Cutting Your Design

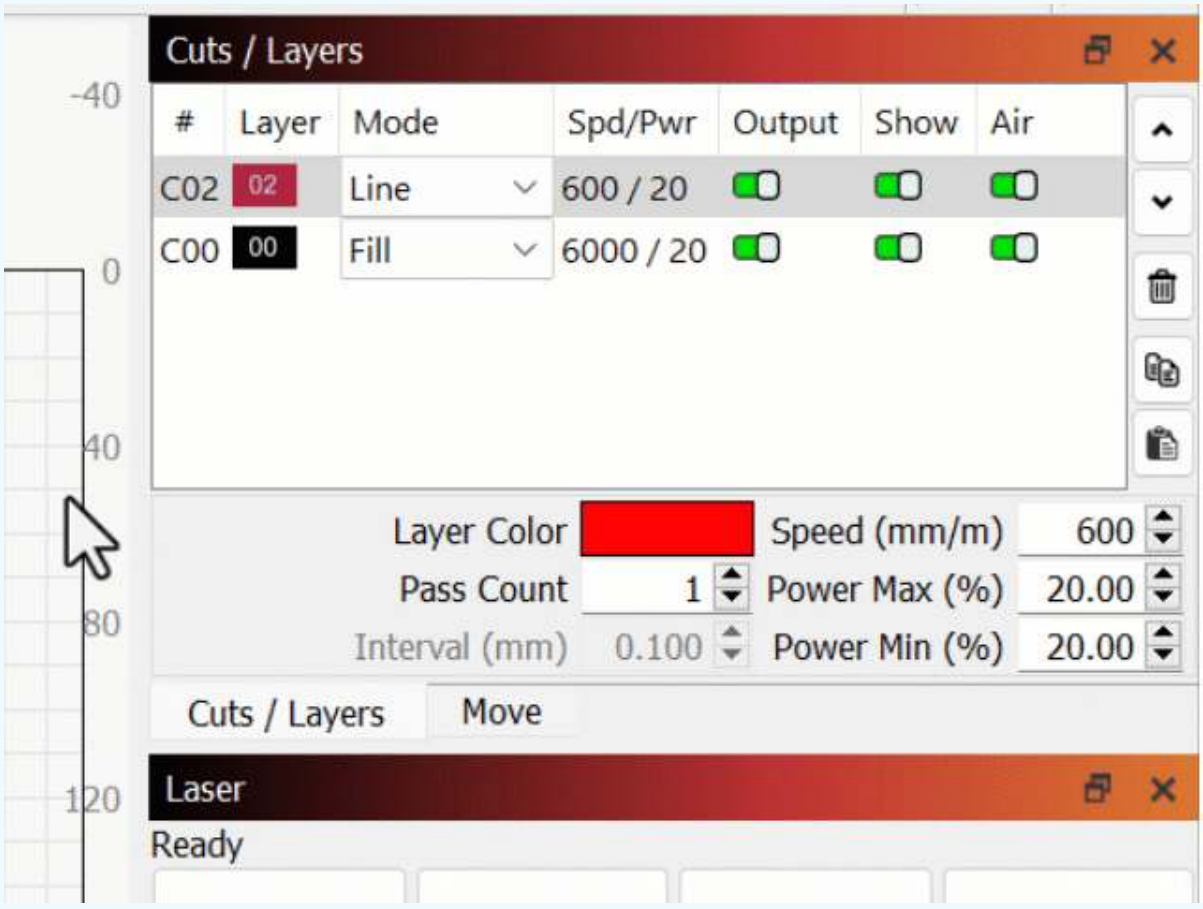
Now that you've created and previewed your design, it's time to load material into your laser and begin cutting.

> **Avoid damaging your work area**
>
> Since you'll be cutting through the material, make sure that whatever is underneath your material is either scrap that you don't mind burning into or won't be damaged by lasering. This might be a thin sheet of aluminum, a honeycomb bed, or a piece of scrap wood.

Focusing the Laser

Lasers need to be focused correctly to achieve good results.

1. Place the material in the laser and move the head over the material. LightBurn has jog controls in the Move Window to make this easier.

> **Quick Reference: Jogging**
>
> 
>

You can use the arrow keys in the Move Window to move your laser up, down, right, or left. You can set the amount the laser travels with each click in the **Distance** field, and the rate at which it travels in the **Speed** field.

If you have a DSP laser you can also jog your laser using buttons on its onboard controller.

More Info → Move Window

Location

- Move Window
- Onboard controller (DSP-only)

Keyboard Shortcuts

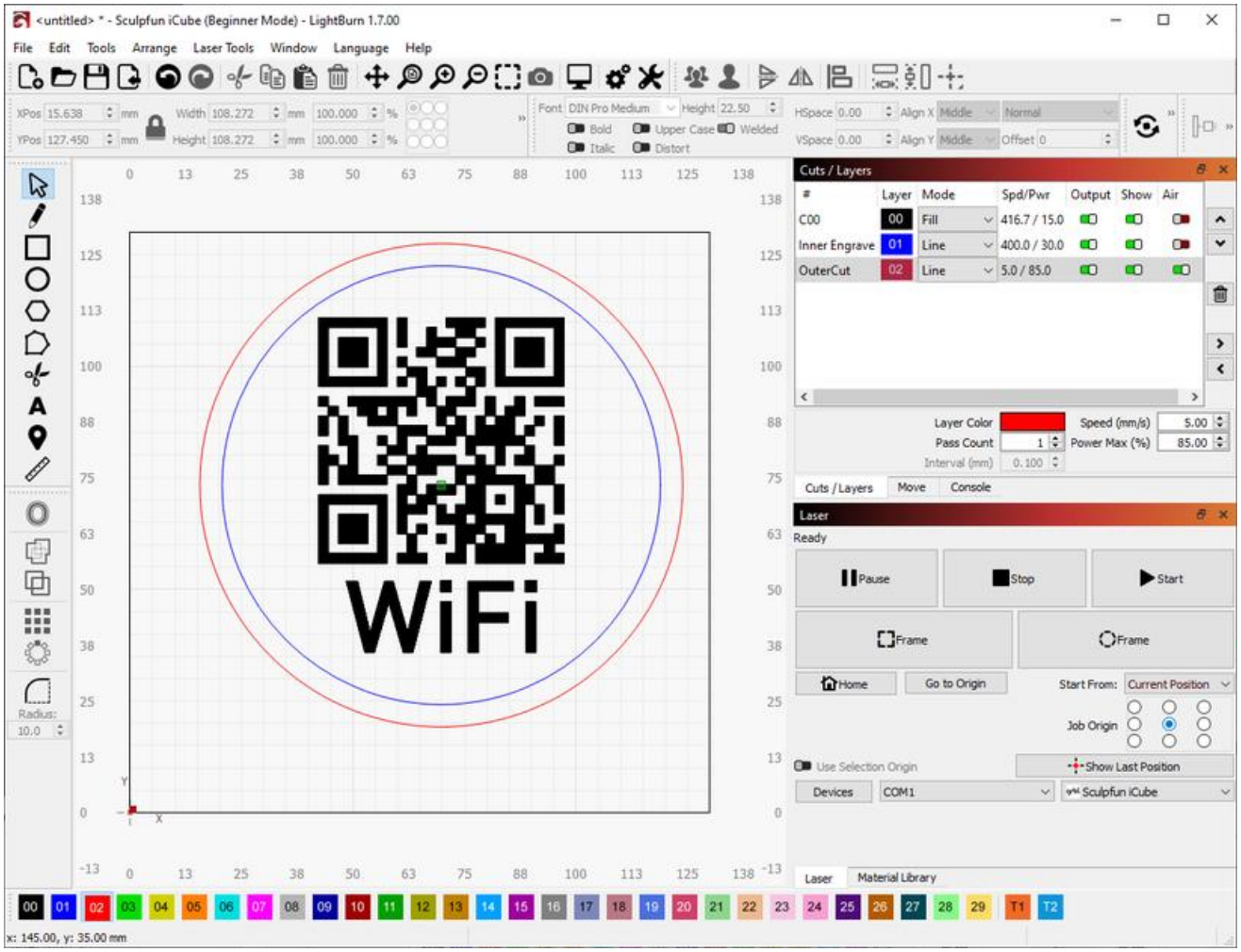
Each keyboard shortcut will move your laser by the amount entered in the **Distance** field, at the rate set in the **Speed** field.

- **Jog Laser Left**: `Alt`/`Option` + `Ctrl`/`Cmd` + `[`
- **Jog Laser Right**: `Alt`/`Option` + `Ctrl`/`Cmd` + `]`
- **Jog Laser Up**: `Shift` `Ctrl`/`Cmd` + `]`
- **Jog Laser Down**: `Shift` `Ctrl`/`Cmd` + `[`

2. Focus the laser according to your laser's user guide:
    - GCode-based Diode lasers usually have a focusing spacer to set the distance between the laser and the material.
    - DSP-type CO_2 lasers may require manual focusing with a focusing spacer or may support auto-focus, either through the controller, or with the Focus Z button.
    - Galvo lasers usually require manual focusing. Many galvo lenses are marked with their optimal focal distance, and you can use a ruler to set the correct distane between the lens (or mirrors) and your material. Some are equipped with a two red dot focusing system — adjust the height of the scan head until the dots meet, then the laser is focused. Finally, see our guide on Galvo Laser Focusing for a manual method of determining optimal focus.

Start From

LightBurn has three different **Start From** modes: **Current Position**, **User Origin**, and **Absolute Coordinates**. For this project, you'll be using Current Position.



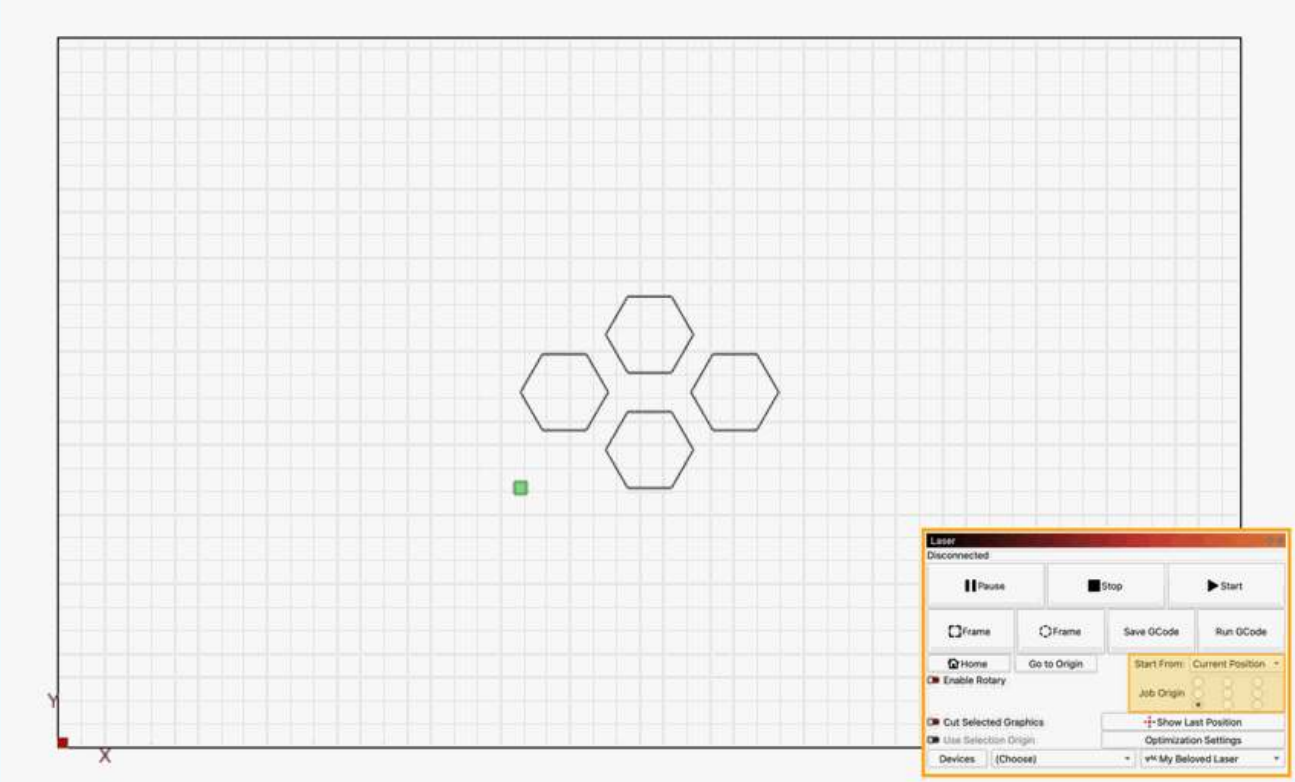
1. Set **Start From** to **Current Position** using the dropdown in the Laser Window.
2. Set the **Job Origin** in the Laser Window to the center of the artwork.
3. With these two settings, your engraving will be centered wherever the laser is positioned when you start the job.

Learn More: Current Position

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=aphiLTUWcjE

With Current Position selected, your job outputs relative to wherever the laser head is located when you press the **Start** button. The **Job Origin** control in the **Laser Window** tells LightBurn how to orient output relative to the laser head.

In this image, we're starting from the Current Position, with the Job Origin set to the lower left:

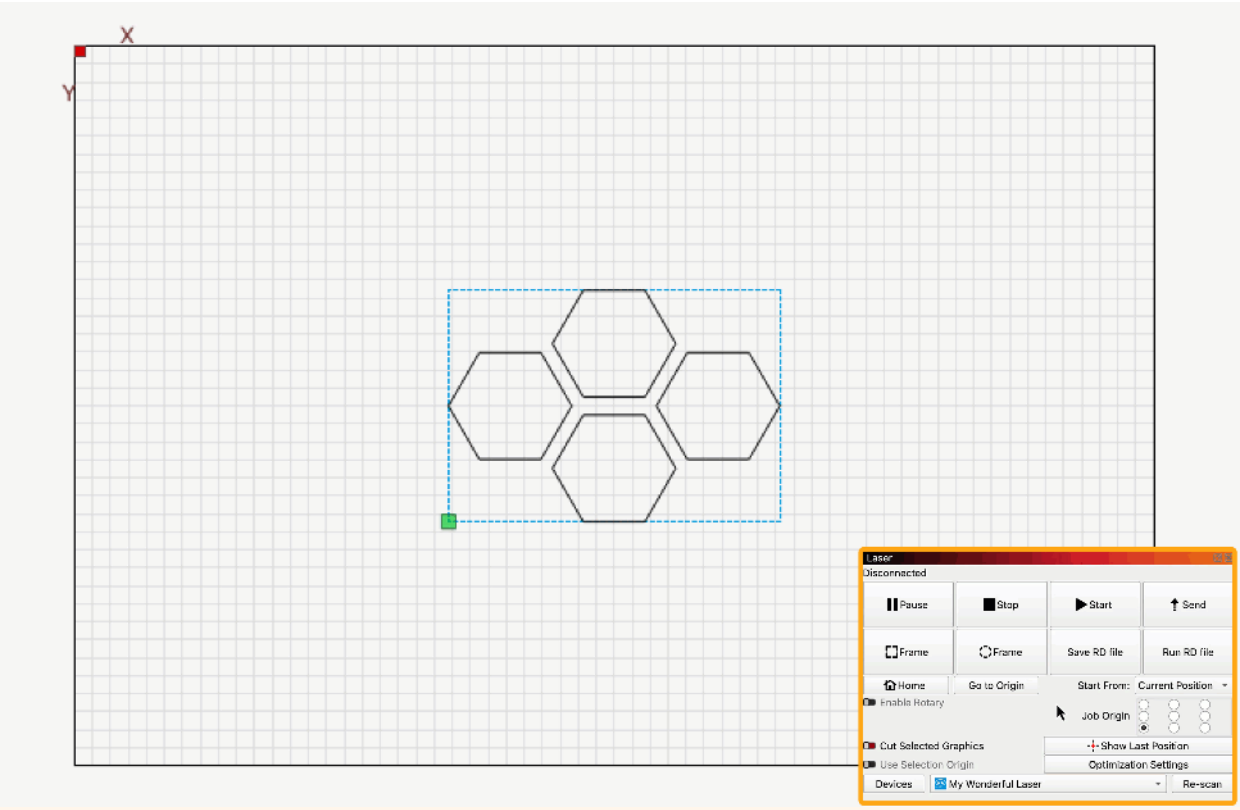


Notice the location of green Job Origin indicator. This represents the position of the laser when you start the job, so the laser is going to move slightly up and to the right from wherever it is, cut the four hexagons, and go back to where it started.

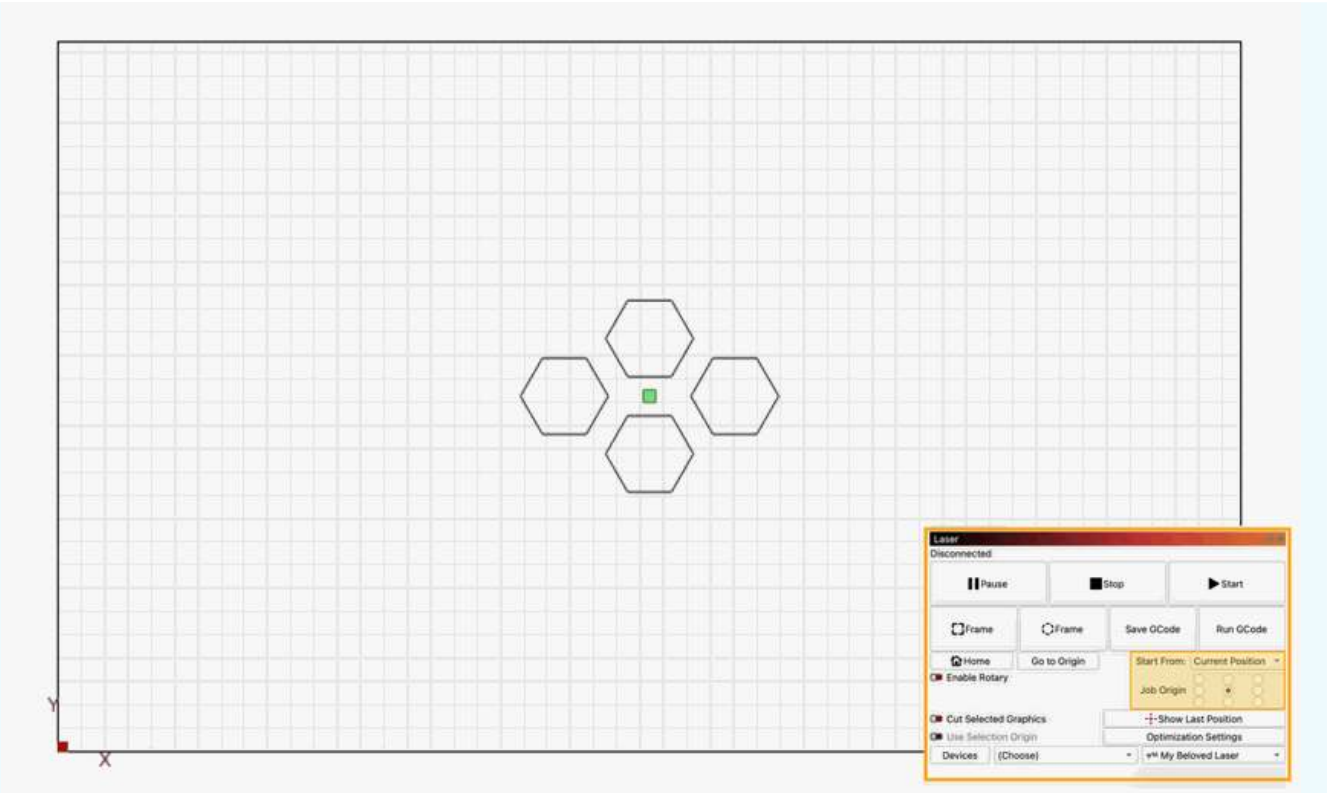
Why is the Job Origin offset from the graphics?

When using Current Position or User Origin, the green Job Origin indicator is always located on a corner, side, or at the center of an imaginary box — the smallest possible rectangle that can fully contain all graphics you're sending to the laser.

Below, we've added a Tool Layer to represent that imaginary box. Notice how the Job Origin moves around the box as we change the location in the 9-Dot selector.



Now imagine that you want to etch this four hexagon pattern onto a coaster or a phone case. Lining it up from the lower left is not easy. If you change the Job Origin setting to center, you get this instead:



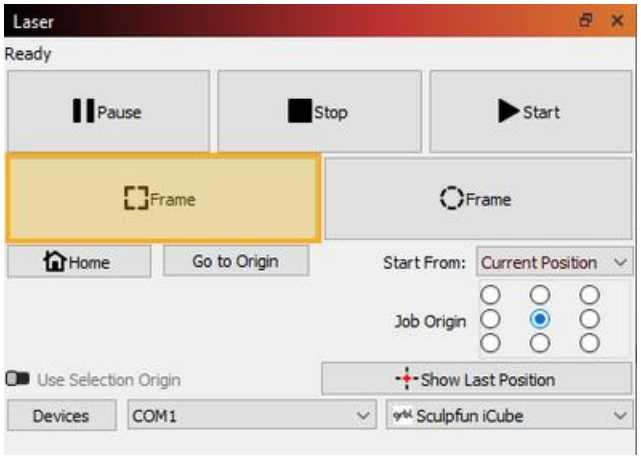
The job will be centered around the current position of the laser head. If you position the laser head directly over the center of the item you want to etch, the output will be centered on the item.

1. Jog your laser to the center of your coaster. If your laser has a controller display and keypad, you can use the buttons there to jog. If not, use the buttons in the Move Window.

Framing

The final check before you can run the job is to make sure that it will output in the location you're expecting.

1. Click the **Bounding Box Frame** button to tell your laser to trace a box around the area the coaster will be cut and engraved.



Learn More: Framing

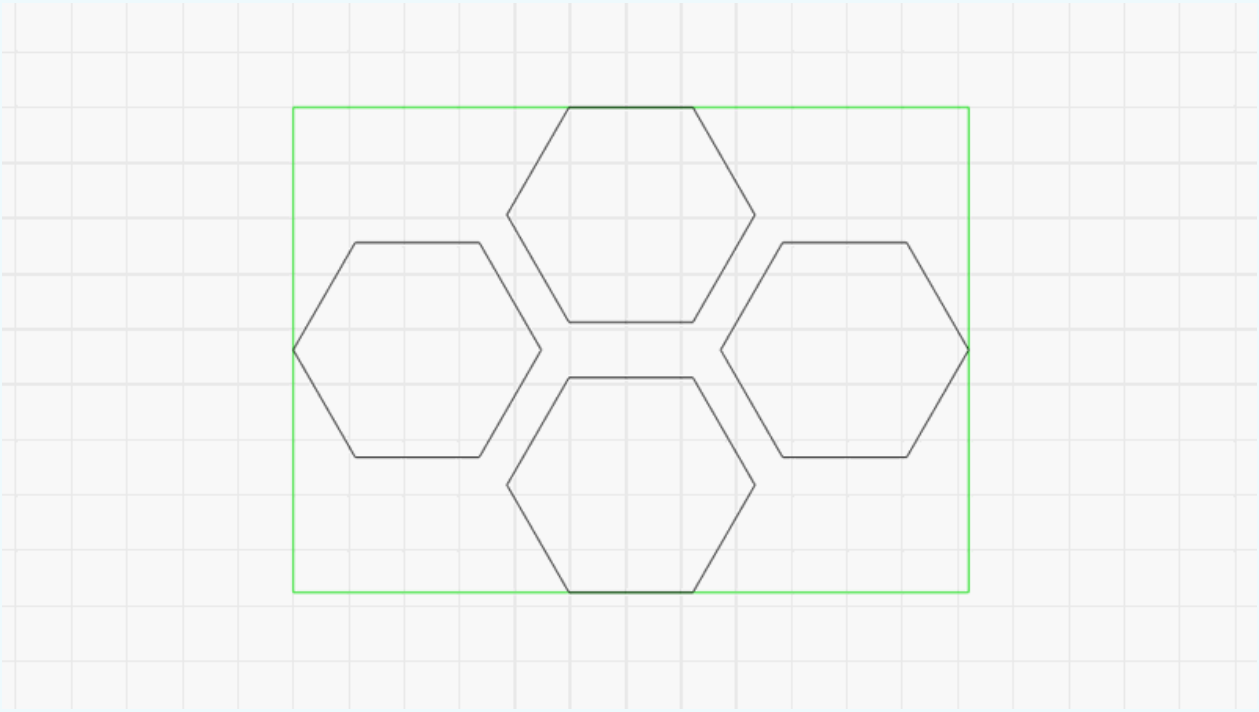
Bounding Box Frame

Bounding Box framing



commands the laser to trace a path defined as the smallest possible rectangle that will fully contain all graphics you're sending to the laser.

Here's a representation of that path in LightBurn:

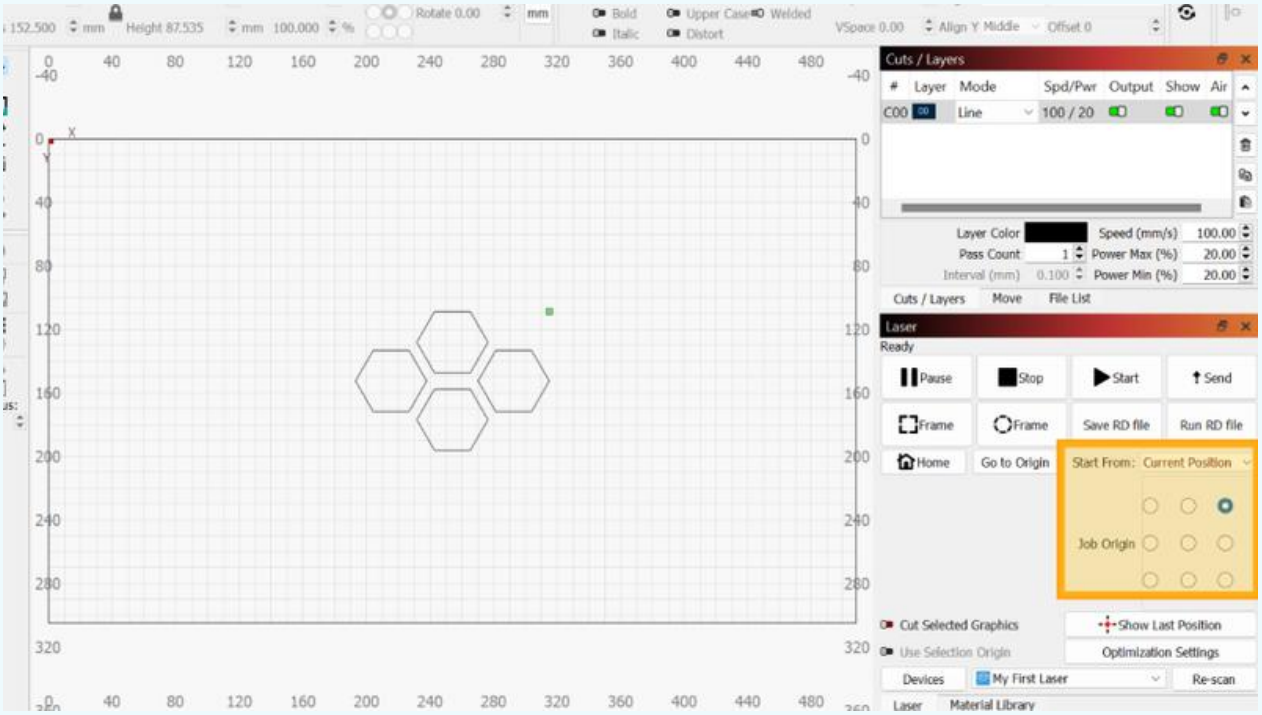


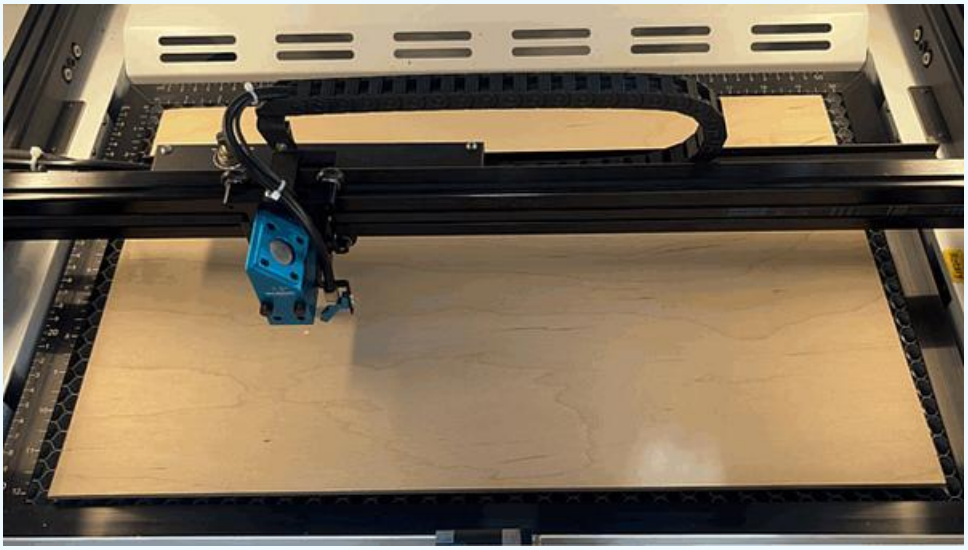
And here's the laser following that same path during Framing:



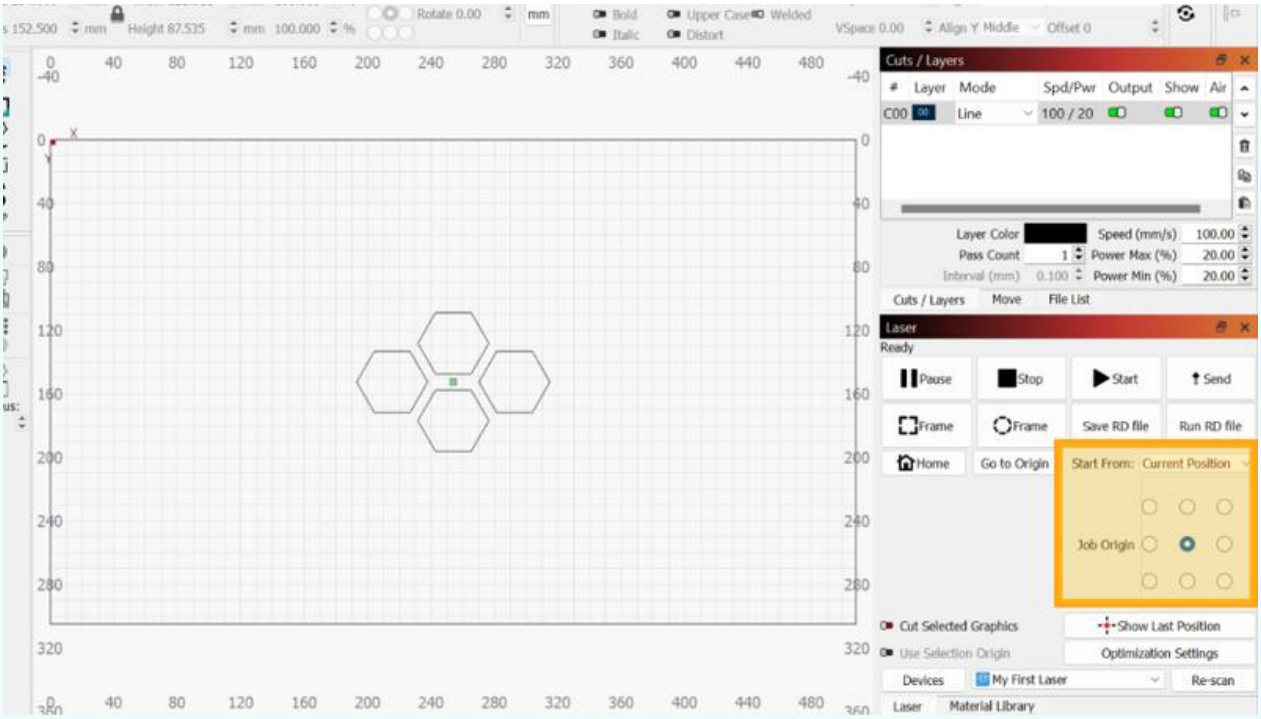
Framing with Current Position

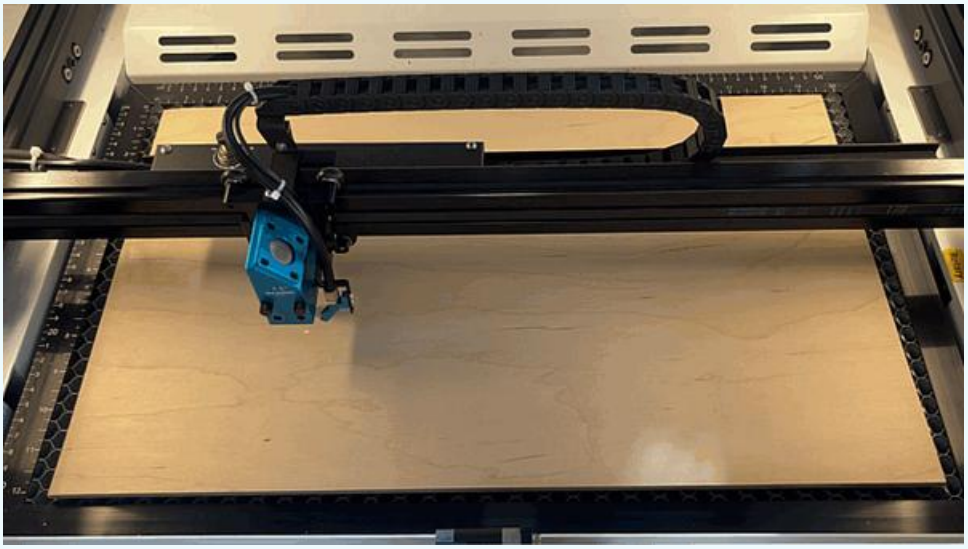
Here we've selected Current Position as the Start From mode, and the top right Job Origin position, so the laser moves down and to the left from its starting position:





Here's Current Position again, but with the center Job Origin position selected — now the laser moves out from where it's sitting, and traces a rectangle centered on the location where it began:





Note for Galvo laser users

If you have a Galvo laser, the design will be outlined by your laser's red dot instead. See Galvo Framing for more information.

2. If the area your laser frames is correct, you're ready to start the job. If not, what you do next depends on how far off it is.

Learn More: Tips for improving Framing visibility

> **How to fire your laser at low power during Framing (diode lasers only)**

If your laser has a red dot pointer, it makes framing easier to see. Otherwise, your laser will travel around with the laser turned off. LightBurn does offer the ability to run diode lasers at a very low power to aid in Framing. To do so:

1. Go to Edit → Device Settings to open the Device Settings window.
2. In the Basic Settings tab, toggle on the **Enable laser fire button** and **Laser on when framing** settings. Click **OK** to apply.
3. The Move Window should now have a **Fire** button and a box to enter a power level. Keep this value very low to avoid burning the material — start at 0.25%, and only increase that value only if the beam is not visible.
4. The **Fire** button will toggle the laser on and off. Now you can use the **Frame** command with the laser visible.
5. Adjust the position of the material and/or the laser head until you're happy with the layout of the design.

> **Danger**
>
> The Fire button should only be used for diode lasers, which generally don't have a red dot pointer for Framing.
>
> Always wear proper eye protection when Firing your laser. Consult your laser's manufacturer for information on the proper eye protection required for your laser.
>
> This should never be used for a CO_2 laser, which has an invisble beam that could blind you or start a fire.

Frame Continuously

Frame Continuously is an option in the Device Settings window that tells your laser to repeatedly Frame the same path until you tell it to stop.

Sometimes you won't get the positioning just right on your first try, but a little nudge of your material will do the trick. Other times, you'll just want to want to double or triple check the position. In either case, Frame Continuously will be a big help.

To enable it:

1. Go to Edit → Device Settings to open the Device Settings window.
2. In the Basic Settings tab, toggle on the **Frame Continuously** setting. Click **OK** to apply.
3. Now when you click either of the Frame buttons, a dialog window will open, letting you know that continuous Framing is active, and to press OK when you're ready to stop.

3. If it's close, but a little bit off the material, try nudging the material (if you have room) to align it to the framed area. If you don't have room, you'll need to exit the Material Test Generator window and jog your laser to a better position.
4. If it's very far off, it's likely you didn't select the correct Start From mode or Job Origin position. Revisit the Start From step, or see Coordinates and Origin.
5. After making corrections, be sure to Frame again. Repeat if necessary, until you've got everything lined up properly.

Running the Job

1. Press the Start button in the Laser Window to tell your laser to begin running your project.
2. If all goes well, you'll have a new coaster in a few minutes.
3. Go and make more stuff!

> **Note**
>
> If the result isn't what you were expecting, revisit previous steps to make sure you followed them all, visit our Troubleshooting section, join our forum to look for advice, or email us at support@lightburnsoftware.com.



Beginner Resources

If you've been following along, you've installed LightBurn, added a laser, learned a great deal about the way LightBurn works, and created your first project. At this point, you know enough to explore LightBurn on your own, but we have some additional resources to help you along.

How to Use These Docs

We're proud to say our documentation is comprehensive, but you might need a few pointers to find your way around. We've got you covered in this brief guide.

How to Use These Docs

Main Window

Find any tool or feature you're looking to learn more about by clicking through interactive UI screenshots, starting from the Main Window.

LightBurn UI

Collections

Sometimes you don't know what tool you're looking for, you just know what you're looking to do. Here we've assembled a range of tools, features, and resources, organized according to the goals they'll help you accomplish.

Curated Collections

LightBurn's YouTube Channel

Our YouTube channel has a variety of videos to help you get familiar with what LightBurn can do and how to make it happen.

LightBurn on YouTube

LightBurn Forum

Talk with other users and LightBurn staff, see and share finished projects, and ask questions on our forum.

LightBurn Forum

Guides

Guides

How-To Guides

Our step-by-step guides will walk you throw how to accomplish a range of tasks in LightBurn.

5 Steps to Perfect Image Engravings

Follow these 5 steps to help you improve your image engravings in LightBurn.

Read More

Kerf Testing & Adjustment

How to test and adjust for your laser's kerf to make parts that fit together perfectly.

Read More

Calibrate Your Camera Lens

How to calibrate your camera lens in LightBurn.

Read More

Full Wrap Tumbler Engraving

Learn how to design, prepare, and engrave a tumbler with a full wrap design on your rotary in LightBurn.

Read More

Customize the LightBurn Window

Learn to customize the LightBurn Window to better suit your needs.

Read More

Move LightBurn to Another Computer

Learn the best way to set up LightBurn on a new computer without losing settings from your old computer.

Read More

Update LightBurn

Learn how to update to a newer version of LightBurn.

Read More

Install an Older LightBurn Version

Learn how to go back to an older version of LightBurn.

Read More

Run Multiple LightBurn Instances

Learn how to open multiple independent LightBurn windows.

Read More

3D Sliced Images on Galvo

Learn how to use LightBurn's 3D Sliced image mode to create 3D engravings on a galvo laser.

Read More

Set Up the Corel Draw Macro

Learn to set up the CorelDRAW macro to make it easier to send work from CorelDRAW directly to LightBurn.

Read More

Adjust the Scanning Offset

Learn to adjust your scan offset to fine tune your results on engravings.

Read More

Configure GRBL Devices

Learn to configure a GRBL machine that wasn't originally intended as a laser.

Read More

Network Connections for GRBL Devices

Learn to configure a GRBL machine to connect over a network.

Read More

Adjust Red Dot Laser Position

Learn to fine tune the position of your red framing laser to accurately frame designs in LightBurn.

Read More

Adjust Offset For a Second Laser Head

Learn to configure a DSP laser with both dual laser tubes and dual laser heads in LightBurn.

Read More

Connect to a Ruida or Trocen Laser over Ethernet

Learn to connect to a DSP laser such as Ruida or Trocen via ethernet using LightBurn.

Read More

Set Up a LightBurn Bridge

Learn to set up and use the LightBurn Bridge to connect to a DSP laser.

Read More

Advanced LightBurn Bridge Setup

Learn to set up LightBurn Bridge on a Raspberry Pi you already have or check out more advanced configuration options.

Read More

Configure a Ruida Controller from Scratch

Learn how to configure a Ruida controller if you need to start from scratch with a new controller or one that's been factory reset.

Read More

Automation with UDP

Learn to use UDP commands to automate basic functionality in LightBurn and on your laser.

Read More

Install Galvo Drivers

Learn to install the drivers needed to connect a Galvo device to LightBurn on a Windows computer.

Read More

Focus a Galvo Laser

Learn how to focus your Galvo laser properly to get the best results from your engravings.

Read More

Change a Galvo Laser Lens

Learn to change the lens on your Galvo laser and adjust your settings in LightBurn to work with the new lens.

Read More

Calibrate and Align the WeCreat Camera

Learn to calibrate and align your WeCreat Vision camera with LightBurn.

Read More

How to Use These Docs

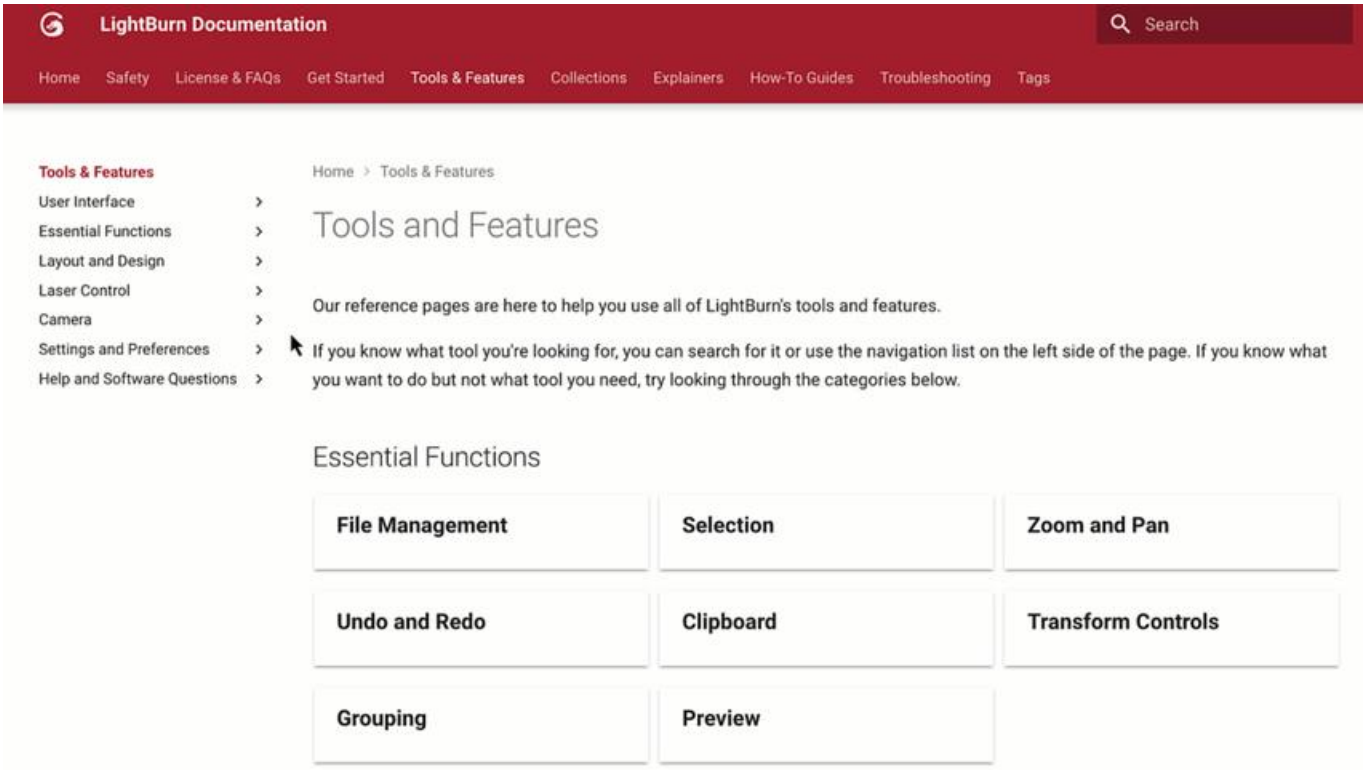
If you want to learn more about LightBurn, you've come to the right place. Now that you're here, you might not know exactly where to go to find just the information you're looking for, so we created this brief guide with tips on navigating and getting the most out of our documentation.

Finding Information

Tools & Features

Every LightBurn tool or feature has a page listed in our Tools & Features section. We've categorized them all by type — click in the index on the left to expand a category. You'll see specific tools and features listed below, or, in some cases, sub-categories which you can also click to expand.

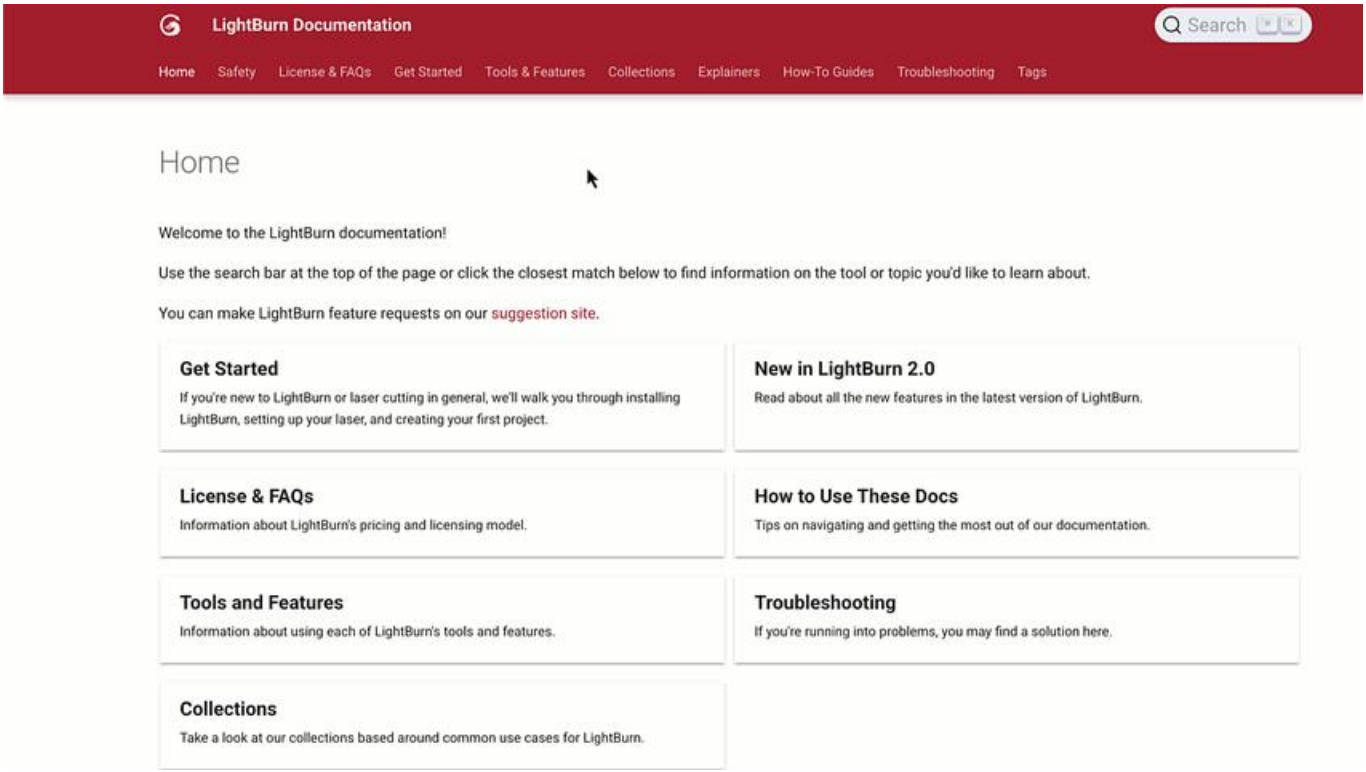
Each category, sub-category, and individual tool or feature is also listed on the Tools & Features landing page.



Main Window

If there's something you see in LightBurn that you want to learn more about, but don't know what it's called, or can't find it in our categorization, our interactive Main Window page is the place to go. You can reach it by clicking **User Interface** in the index on the left when you're in the Tools & Features section.

Click any of the main UI elements in the screenshot to go to the page for that element, where you can click again on specific tools or features it contains.



Finding Help

LightBurn also has some built-in features to make it easier to learn more:

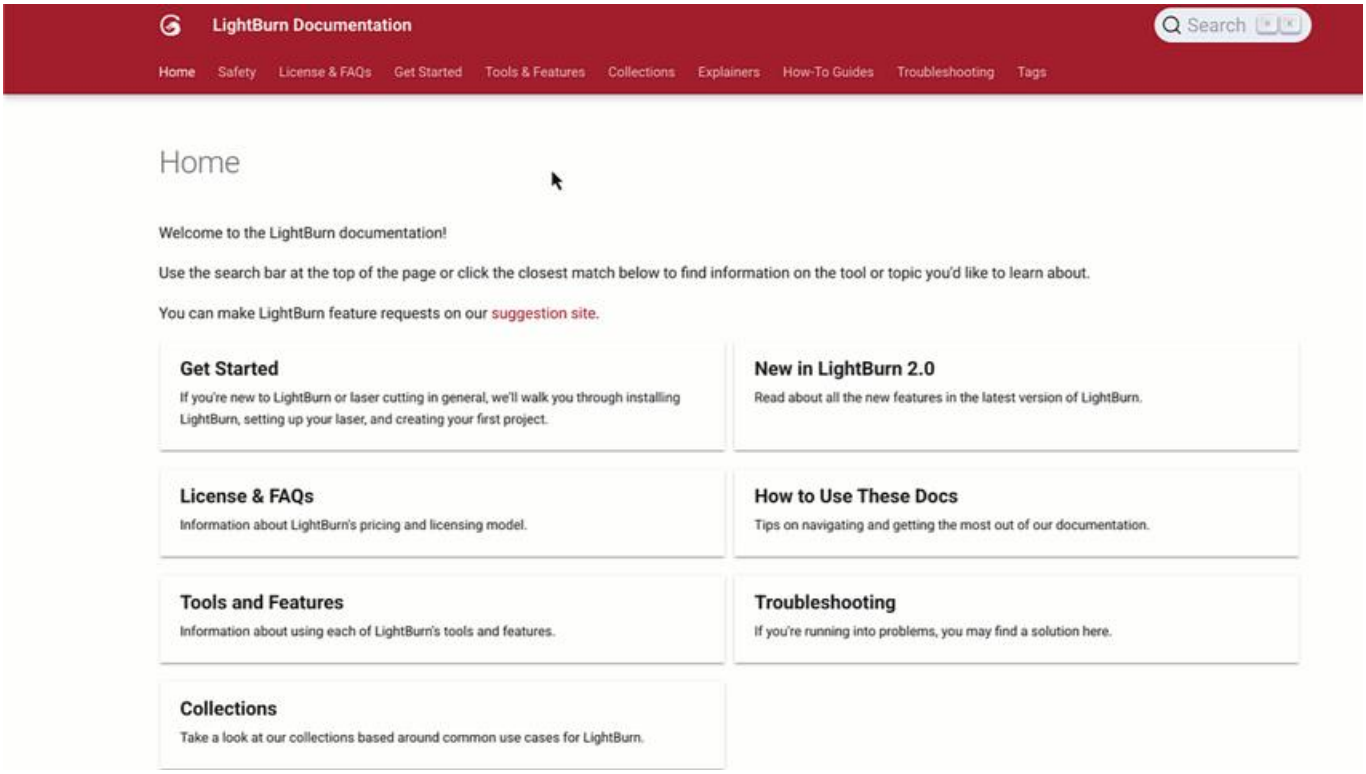
- **Easy Access Help Pages**: Press `F1` while hovering to launch the documentation page for that feature in your default web browser.
- **Tools Tips**: Hover your mouse over a control to see helpful text pop up, with a description of that button or feature.

Related Topics

Almost all of our Tools & Features pages have a **Related Topics** section at the bottom, so if you didn't find exactly what you were looking for on any given page, be sure to scroll to the bottom or click Related Topics in the table of contents to the right for suggestions on where to look next.

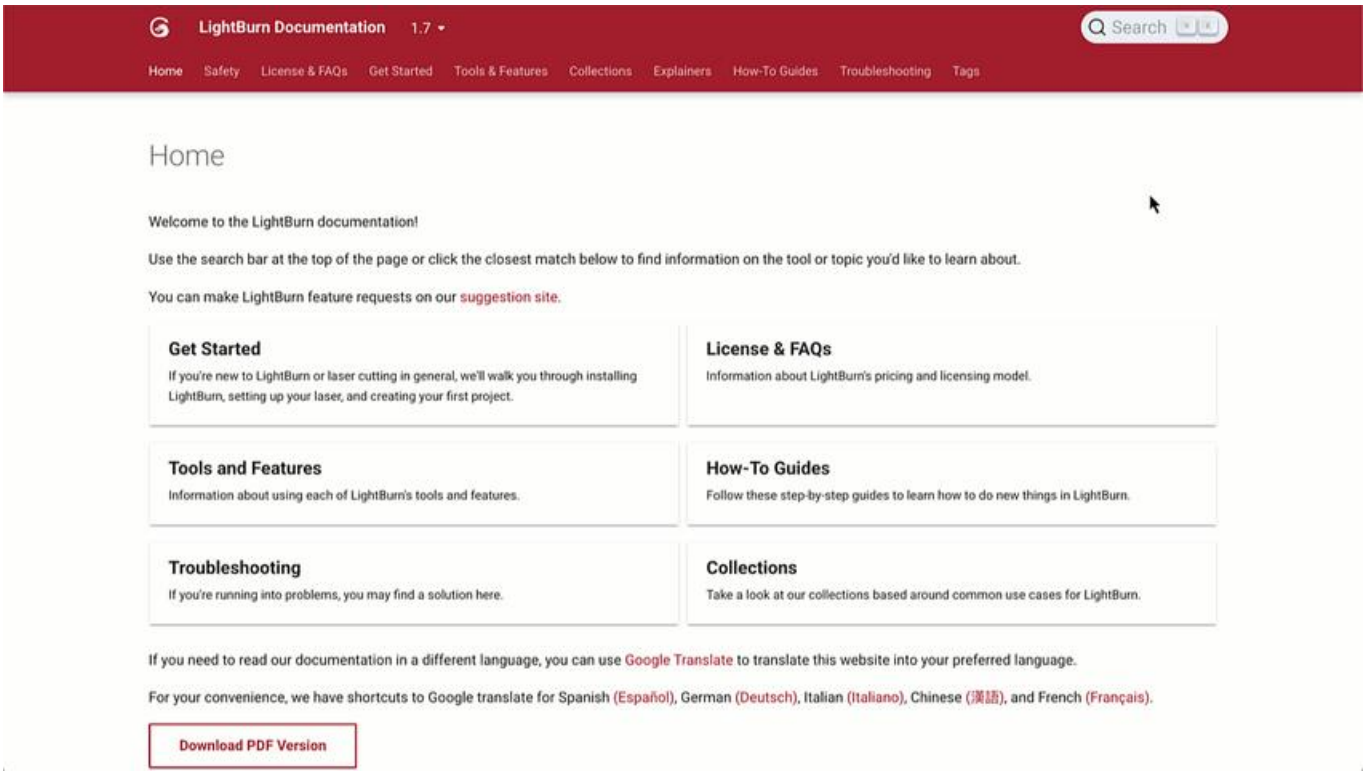
Collections

Sometimes you don't know what tool you're looking for, you just know what you're looking to do. In our Collections, we've assembled a range of tools, features, and resources, organized according to the goals they'll help you accomplish.



Search

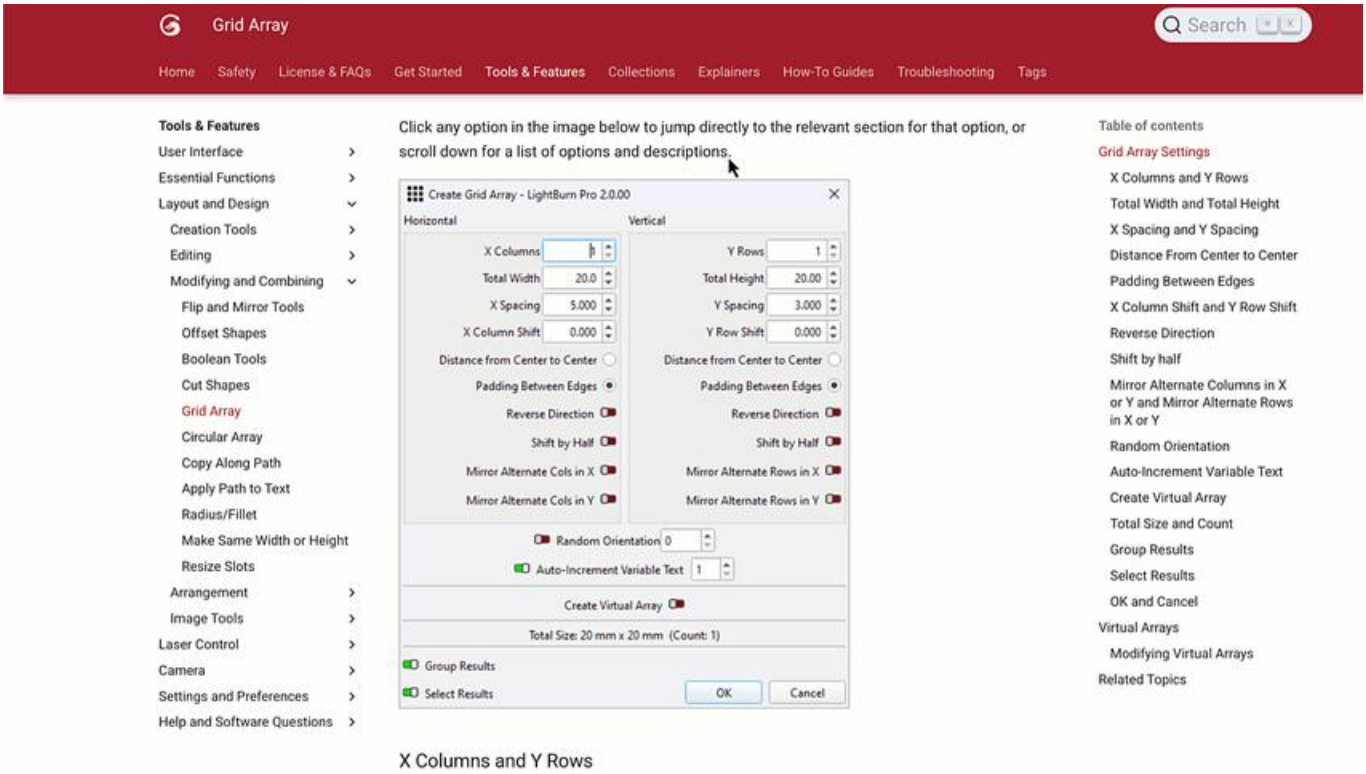
At the top right of any page you'll see the **Search** option. Enter the name of the tool you're looking for there to search for the relevant page in our documentation.



Interactive Features

Clickable Screenshots

In addition to the Main Window page, several other pages contain clickable screenshots — as you hover over buttons and tools within them, they'll be highlighted. If you click where something is highlighted, you'll either jump to a new page, or to a section within the current page that explains the specific feature you were hovering over.



X Columns and Y Rows

Information Boxes

We've placed certain information in special colored boxes throughout our documentation. In some cases they contain important warnings that we want to make very noticeable, but in other cases these boxes contain special notes or tips for specific types of users only.

To keep things tidy, some of those special notes are collapsed by default, meaning the information contained within them is hidden until you click to expand them. Click the box below to see how it works!

> **Tip**
>
> how it works!

Link Previews

If you hover over any link in our documentation that points to another page, you'll see a preview of the linked-to page. That preview may provide all you need to know if you're just looking for a quick summary of a term in context, while in other cases it will let you know if clicking the link will take you to correct page for a feature you're looking to learn more about.

Hover over this link to try it out.

Getting Help

We've assembled a range of common laser or LightBurn issues we've seen users run up against over the years in our Troubleshooting section. You'll find advice on resolving issues connecting to your laser, using LightBurn, or with the quality of your laser's output.

If you can't find the information you're looking for in our documentation, or need clarification on something you've read, email support to ask for help, or join our forum to get advice from experienced users and LightBurn staff.

You'll find those same links at the bottom of every page in our documentation — including just below here!

Advanced Hardware Setup

Laser 2 Offset Setup

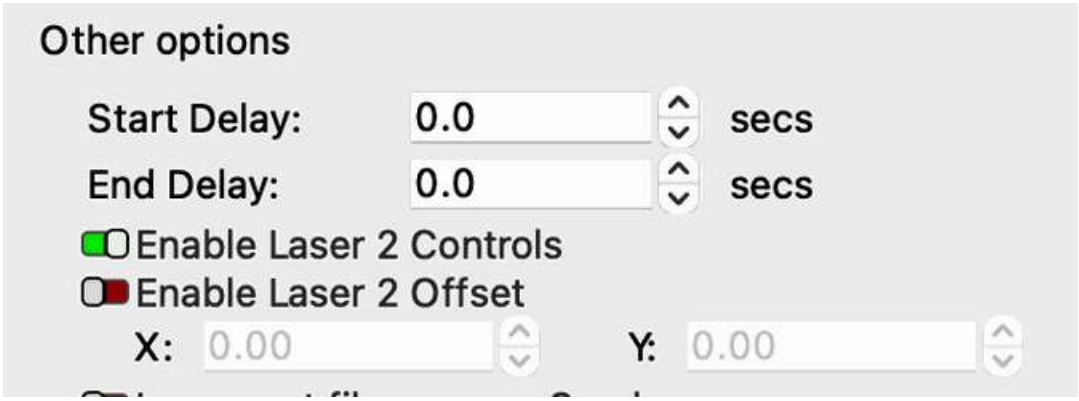
If you have a DSP laser with dual laser tubes and dual laser heads, the directions below will help you set your **Laser 2 Offset** values.

Laser 2 Offset

The **X** and **Y** fields beneath the **Enable Laser 2 Offset** switch in the Device Settings window are used to compensate for the distance between two laser heads. By shifting output to the 2nd laser head by specified X and Y values, output from Laser 2 will line up with output from Laser 1.

Follow the instructions below to measure the distance between Laser 1 and Laser 2 and apply appropriate offsets.

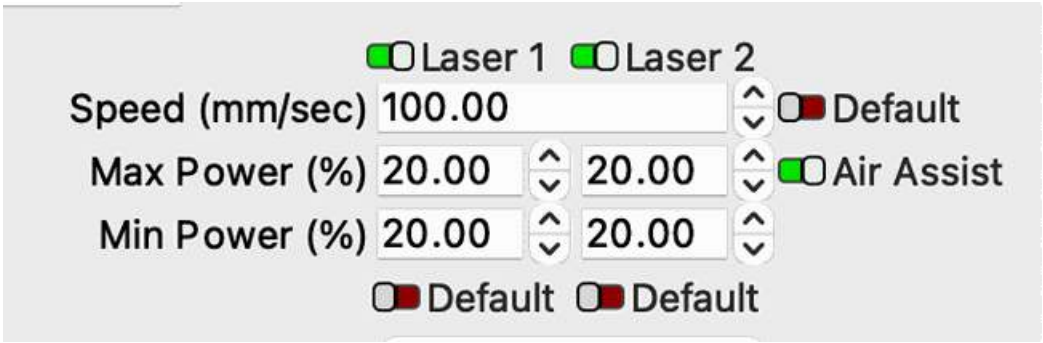
1. Before beginning this process, make sure **Enable Laser 2 Controls** is toggled on.



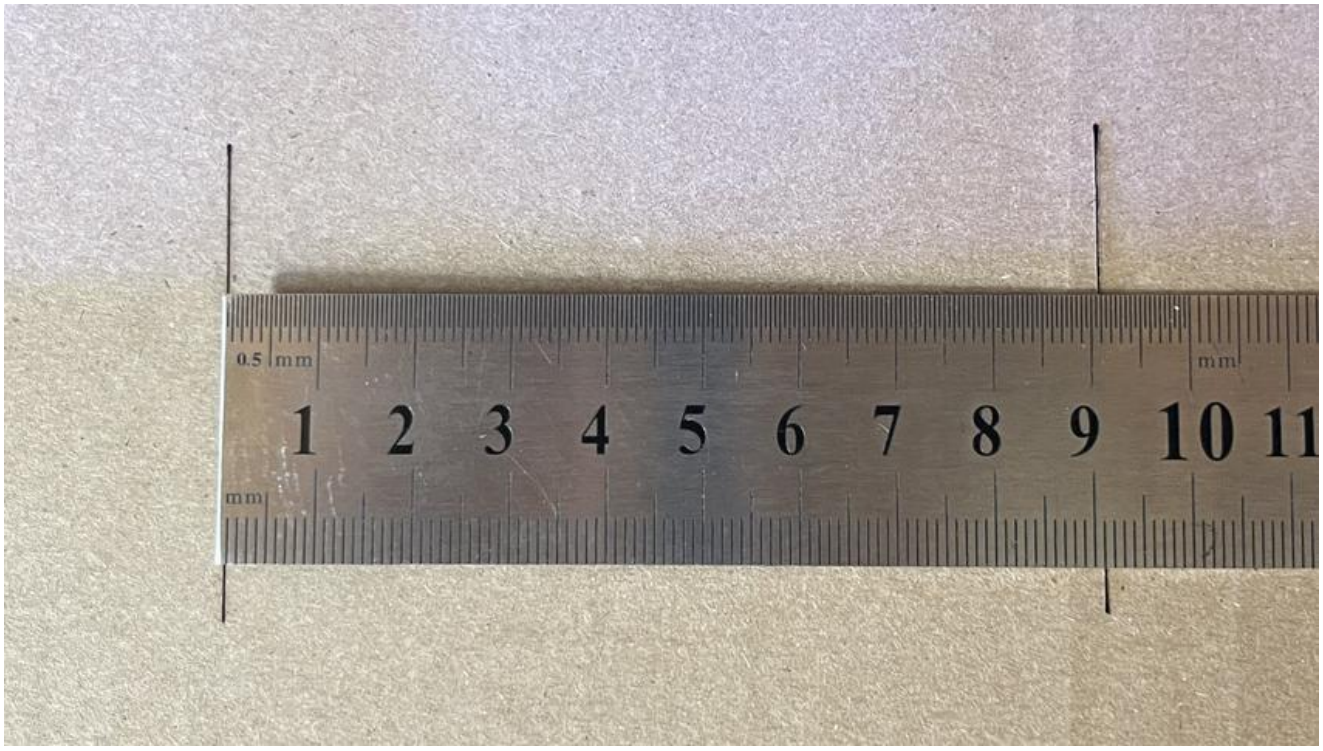
2. Load an inexpensive, expendable material into your laser's bed. Make sure it is large enough for both laser heads to mark it at once. Position it beneath both laser heads.
3. Use the Draw Lines tool to create a short, perfectly vertical line in your LightBurn workspace.

> **Tip**
>
> Hold `Shift` while drawing to ensure the line is perfectly vertical.

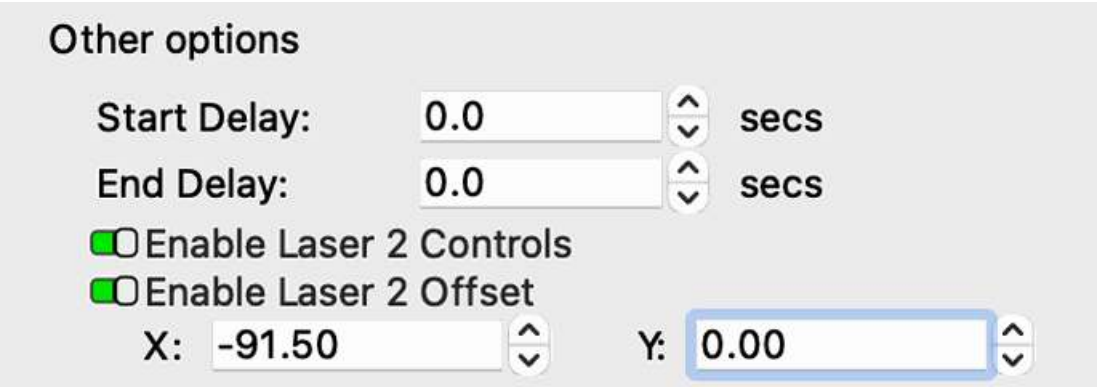
4. Assign the vertical line to a layer set to Line Mode.
5. In the **Cut Settings Editor**, enable both Laser 1 and Laser 2, and apply appropriate settings to mark the vertical lines on the material without cutting through.



6. Select Current Position as your **Start From** mode.
7. Press Start to mark the lines on the material.
8. Measure and record the distance between the lines.



9. In the Device Settings window, turn on **Enable Laser 2 Offset**.
10. If Laser 2 is farther from your machine's origin than Laser 1, enter the measured distance between the lines as a negative value in the **X** field. If it is closer to the origin, enter the distance as a positive value.

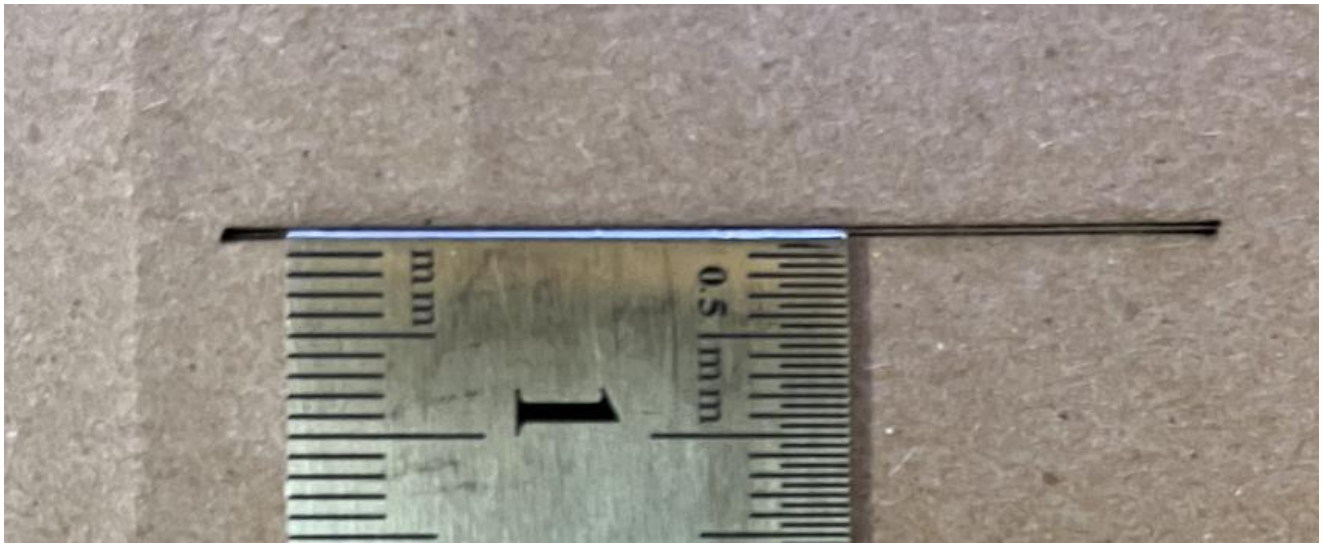


> **Note**
>
> The machine's origin is the location it homes to when it first starts.

11. Rotate the vertical line 90 degrees to make it horizontal.

> **Tip**
>
> Press the . key to rotate objects 90 degrees clockwise.

12. In the Cut Settings Editor, add a **Sub-Layer** to the layer the horizontal line is assigned to.
13. You'll now have two Sub-Layers. Keep both set to Line Mode.
14. Disable Laser 2 for the first Sub-Layer, but enable it for the second Sub-Layer. Disable Laser 1 for the second Sub-Layer.
15. On the second Sub-Layer, apply appropriate settings for Laser 2 to mark your material.
16. Select Current Position as your Start From mode, and make sure the first laser head is positioned over the material.
17. Press Start.
18. If you measured the distance between the vertical lines accurately in Step 8, each horizontal line should line up perfectly in the X dimension.
    - If they are misaligned and the output from Laser 2 shifted in the opposite direction of what you expected, reverse the sign of the X dimension Laser 2 Offset in the Device Settings window. In other words, if the value is positive, make it negative, or vice versa.
    - If they are misaligned but Laser 2 output was shifted in the correct direction, repeat Steps 1-8, taking care to measure as precisely as possible.
19. If the horizontal lines are aligned in both the X dimension and the Y dimension, the process is complete. If they are aligned in X but misaligned in Y, measure the distance between the 2 lines.



20. If the line output by Laser 2 is farther from your machine's origin than the one output by Laser 1, enter the measured distance between the lines as a negative value in the X field. If it closer to the origin, enter the distance as a positive value.

> **Note**
>
> The machine's origin is the location it homes to when it first starts.

21. Mark the horizontal lines again to verify alignment.
    - Reverse the sign of the Y offset if output was shifted in the incorrect direction.
    - If output was shifted in the correct direction in Y but is still misaligned, reset the Y offset to 0 and repeat Steps 17-20.

Adjust Red Dot Framing Offset

If your laser has a red dot pointer that is not aligned to the location of your laser's beam, use the Laser Offset in the Device Settings window to shift your laser's output to match the location of the red dot.

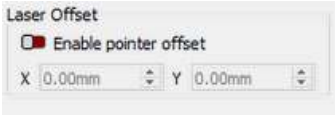
With the Laser Offset properly set up, you can use the red dot location to:

- Align output to material using Current Position or User Origin
- Frame output location

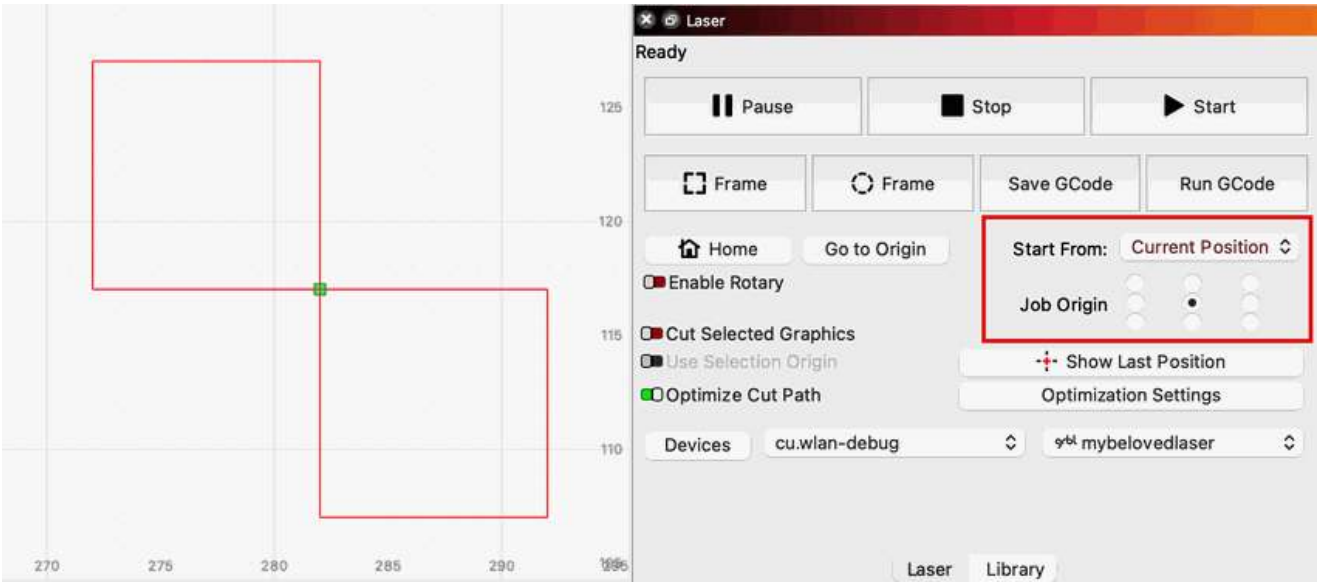
When you run your project, the location of the laser head will shift to match the location of the red dot when you performed one of the above actions.

Establishing a Laser Offset

1. In the Device Settings window, make sure **Enable pointer offset** is disabled before beginning this process.



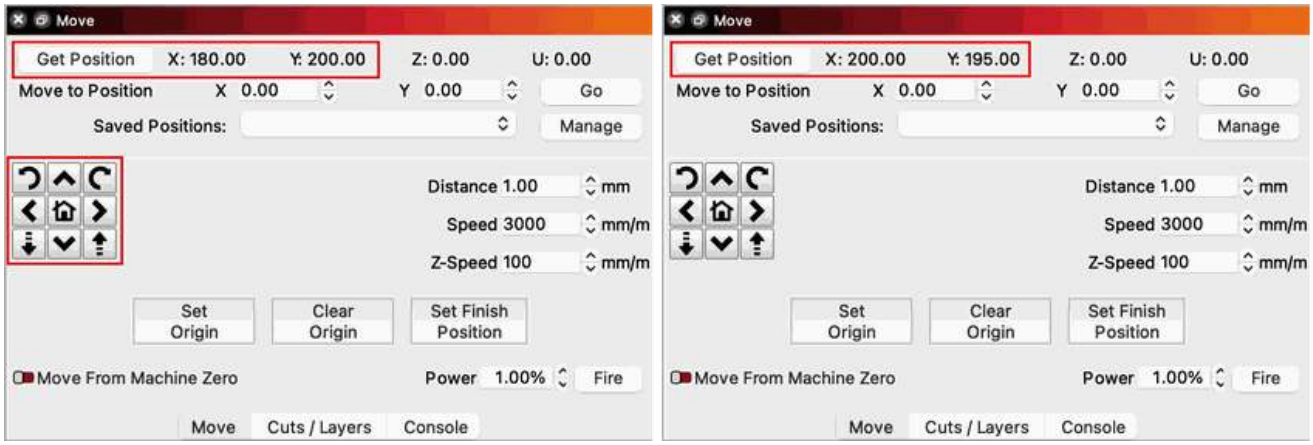
2. Load an inexpensive, expendable material into your laser's bed. Place it in a location that both the red dot and laser's beam can reach. Position your laser over the material.
3. Download this 2 square pattern, and Import it into LightBurn.
4. Make sure the pattern is assigned to a layer set to Line Mode, and apply appropriate settings to mark it on the material you loaded into your laser, without cutting through.
5. Select Current Position as the **Start From** mode, and center for the **Job Origin** position.



> **Note**

> If you are using a DSP device, go to Laser Tools → Machine Settings and make sure **Return Position** is set to **No Return**.

6. Press **Start** in the Laser Window to run the job.
7. In the Move Window, press **Get Position**, and record the **X** and **Y** positions reported.
8. Jog your laser until your pointer's red dot is perfectly aligned to the center of the 2 square pattern — where the squares' bottom right and top left corners meet.
9. Press Get Position once again, and record the X and Y positions reported.



10. To calculate your X and Y offsets, subtract the X and Y positions you recorded in **Step 9** from the ones you recorded in **Step 7**.

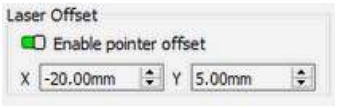
    For example, if you recorded X: 180 in **Step 7** and X: 200 in **Step 9**:

    ```
    180 - 200 = -20
    ```

    And if you recorded Y: 200 in **Step 7** and Y: 195 in **Step 9**:

    ```
    200 - 195 = 5
    ```

11. In the Device Settings window, enter the results from **Step 10** in the X and Y fields in the **Laser Offset** section and toggle on Enable pointer offset.



> **Note**
>
> If you are using a DSP device, reverse the sign of the values you calculated in **Step 10** — make negative numbers positive, and vice versa.

12. Test your offset by re-running the same job, exactly as it was set up in **Step 6**, with the red dot still directly centered over the pattern, where the points of the 2 squares meet.

    The laser head should shift to match the location of the red dot, and the output should overlap with the first engraving.

Troubleshooting

- If output shifts in the wrong direction in one or both dimensions, reverse the sign of the offset you entered in the Device Settings window — make it positive if it is negative, or vice versa.
- If it is close, but slightly misaligned, re-run the setup process, taking care to align the red dot as perfectly as possible to the center of the 2 square pattern in **Step 8**.
- If alignment is lost between uses, make sure the red dot pointer is mounted securely, so that its location cannot drift, then re-run the setup process.

Scanning Offset Adjustment

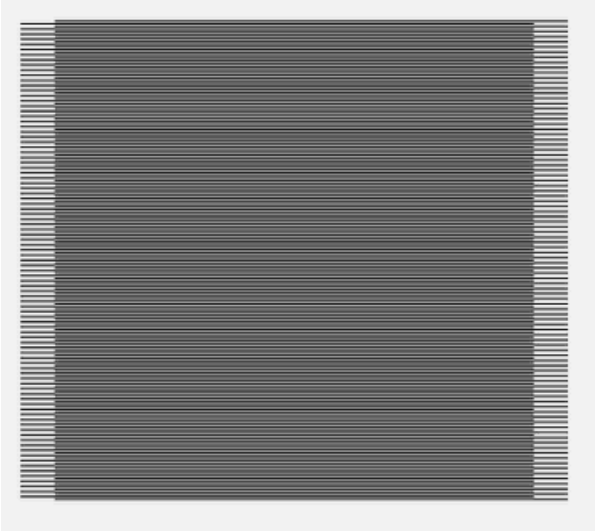
Modern lasers are capable of moving very fast, and with remarkable precision, but firing the beam still takes time. Some power supplies and tubes may respond in microseconds, but others take longer. At higher speeds, belts will pull slightly as well, and all of these things can cause the output of the laser to shift slightly when engraving.

> **Don't forget to check for mechanical problems**
>
> Before adjusting **Scanning Offset**, make sure you check for mechanical problem such as loose belts which can also cause slipping and shifting.

At 100 mm/second, 254 dots per inch means your dots are 0.1 mm in length, fitting 1000 of them in 100 mm. At 100 mm/sec, if your power supply and tube take 1 millisecond to fire, your engraving will be offset by a full dot width.

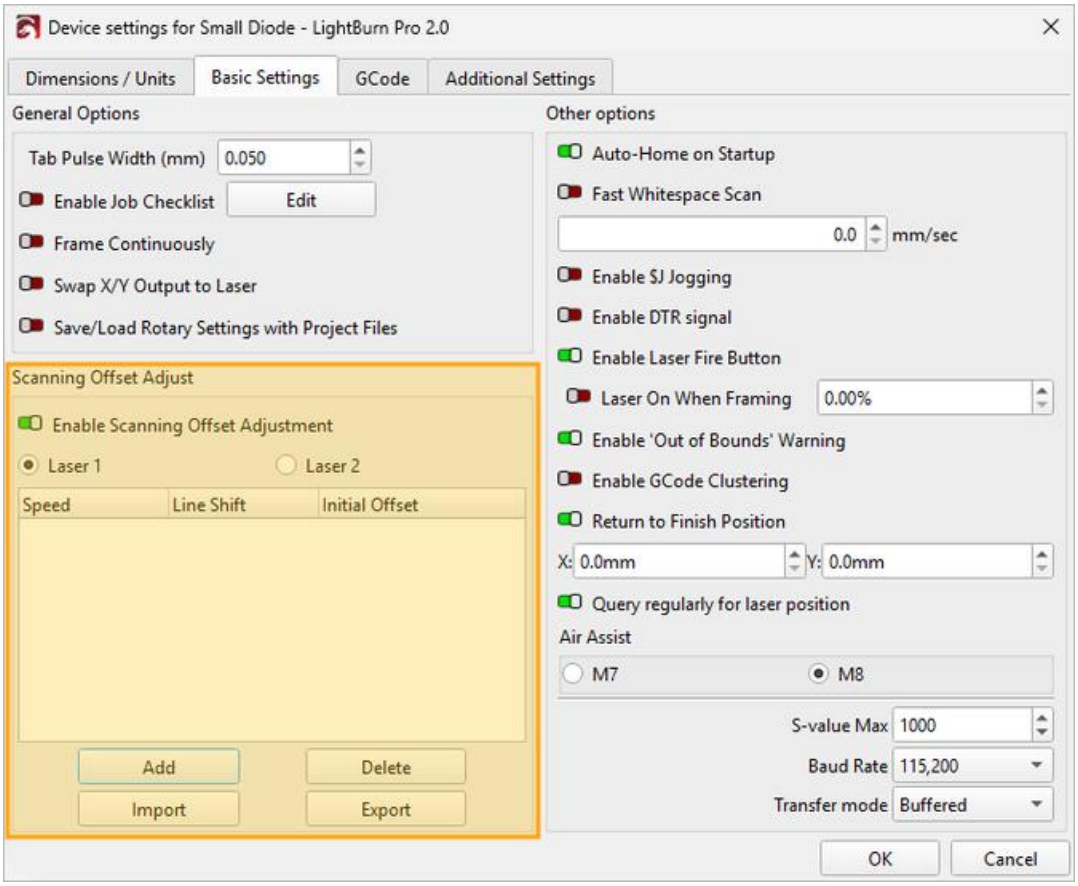
At 500 mm/second, that 1 millisecond delay means you'll be off by 5 dots, or ½ a millimeter. Still not very much, but visible. Adding a bit of mechanical stretch to this will increase the offset.

The result often looks like ghosted edges. The image below is a 20 mm square at 1000 mm/sec, with a 1ms delay, resulting in a full mm of skew between scans:



LightBurn has a setting to counter this, called **Scanning Offset Adjustment**, in the **Basic Settings** tab of Device Settings.

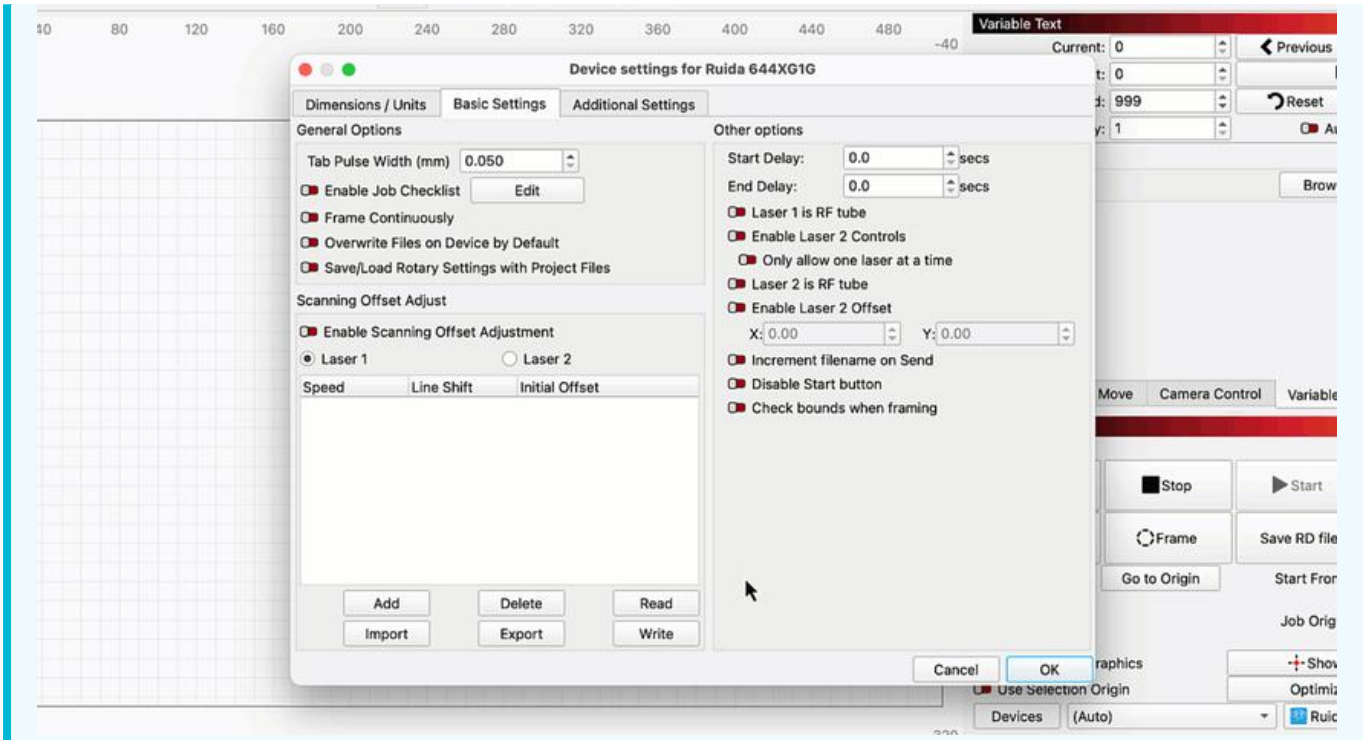
> **Note**
>
> Other software may refer to it as "Reverse Interval" or "Reverse Compensation".



Read and Write Buttons

Some Ruida controllers allow the ability to directly **Read** Scanning Offset Adjustments stored in the controller's settings, or **Write** new settings from LightBurn.

These buttons only appear when LightBurn is directly connected to a Ruida controller that supports the Read/Write functions.



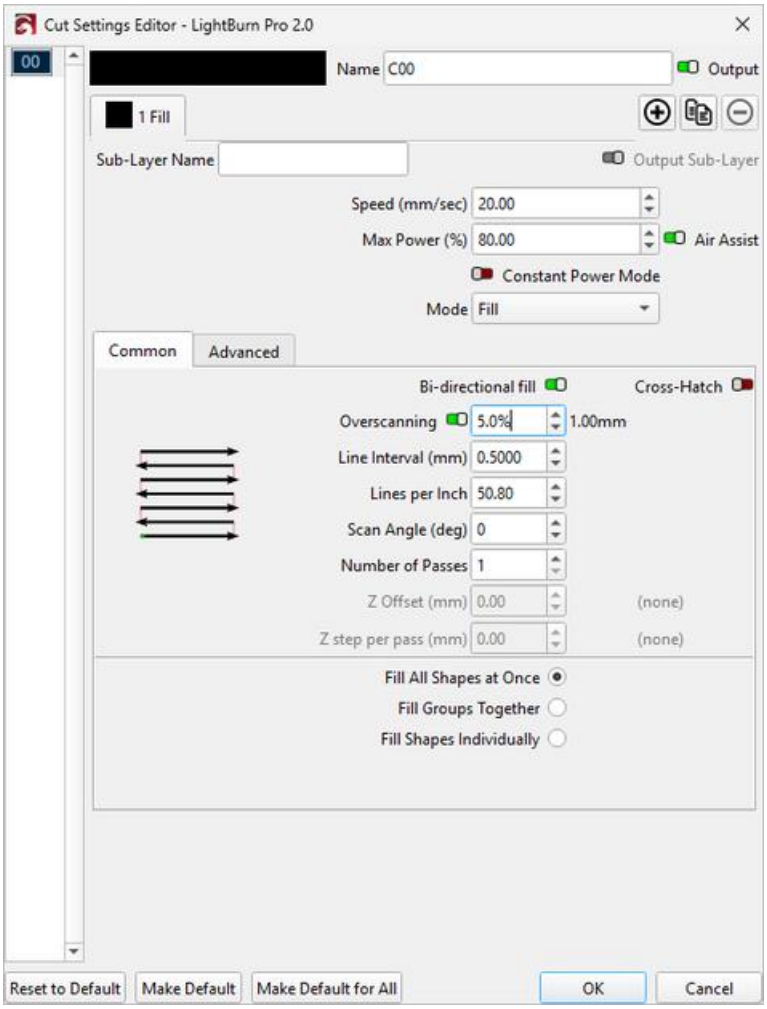
Determining Scanning Offset Adjustments

Already have Scan Offset settings in a `.lbso` file?

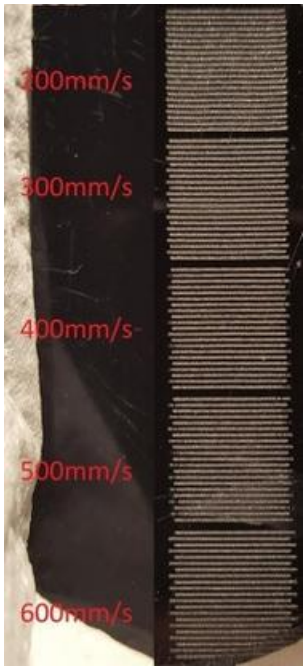
If your laser manufacturer provided a `.lbso` file or you've exported your own settings, skip ahead to Importing and Exporting Scan Offset Settings.

To use this feature, you need to measure the response of your machine at a few different speeds.

1. Create a rectangle in LightBurn, 50 mm wide and 10 mm high,
2. Make sure the rectangle is assigned to a layer set to Fill Mode, and apply an appropriate Power setting to visibly engrave the material you loaded into your laser without charring.
3. Set the Line Interval to 0.5 mm.
4. If you are on a GCode based device, enable Overscanning, and set it to 5% or greater to be sure the machine is not slowing down before reaching the ends. (Ruida devices overscan automatically).

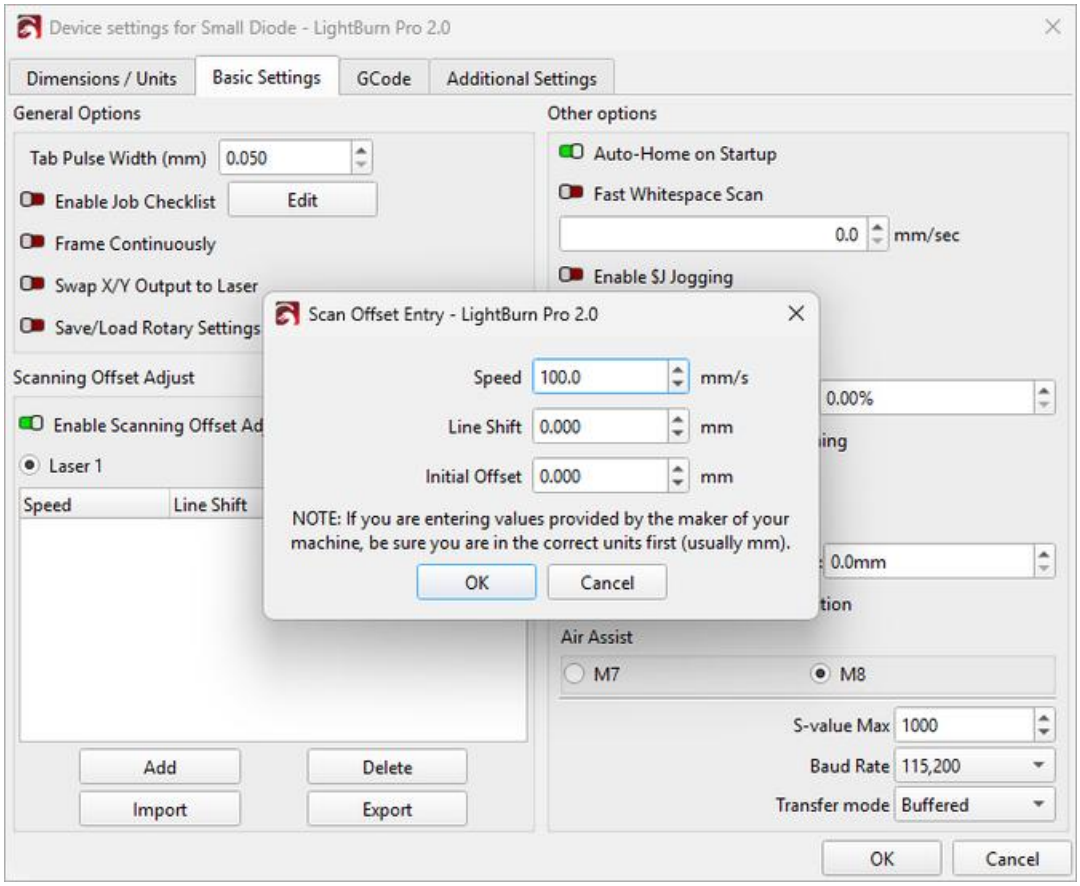


5. Run the rectangle at multiple Speeds. At least two will be required for this process, but using more will improve the quality of your results. Make sure you use speed settings your laser is capable of reaching, and keep track of the speed used for each run. The exact speed settings you use will depend on your machine, but you should test several different speeds representing a range of speeds your laser can achieve.



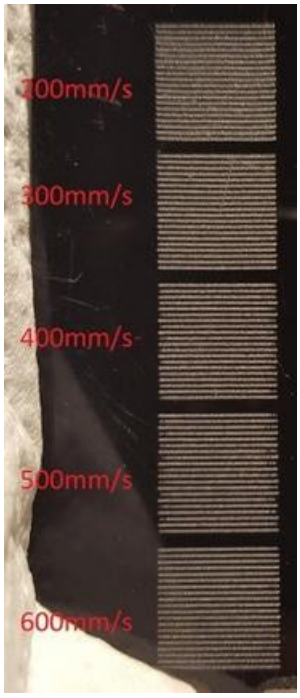
6. To compensate for the misalignment of lines, measure the distance between the ends of the lines at each Speed. You'll need to enter half that distance in the next step.
7. In the **Scanning Offset Adjust** section of the **Device Settings** window, click **Add** to enter the **Speed** and **Line Shift** (distance between ends of lines) values into the Scanning Ofset Adjustment table. LightBurn will use this information to compute the correct adjustments for other speeds as well. The **Initial Offset** value is generally not needed, but can be used to shift the entire engraving left or right if needed to compensate if your engravings are shifted relative to vector cuts or engravings.

> **Note**
>
> You will need to enter half the measured distance between lines — LightBurn moves each line by the amount you specify, so each pair of lines only needs to move half the distance.



8. Make sure to toggle **Enable Scanning Offset Adjustment** on, and click **OK** to exit the Device Settings window and save your changes.
9. You may need to perform this process multiple times, making minor adjustments to get clean results at each speed. After entering the measurements for the above speeds, the resulting corrected output looks like this:

> **Note**
>
> If your machine has two laser sources, you may need to repeat this process and apply unique adjustments for each laser. Click the button next to **Laser 2** to apply settings for the second source.



Importing and Exporting Scan Offset Settings

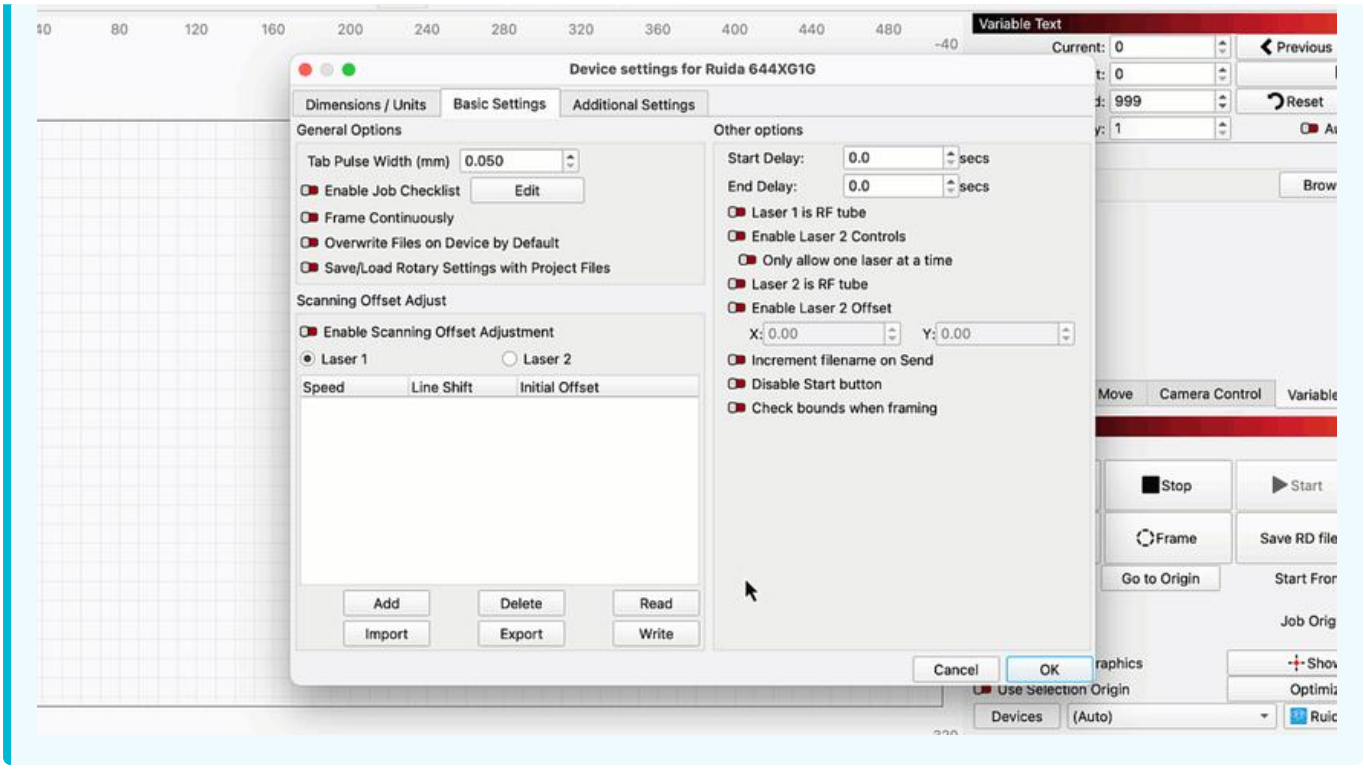
Importing

Use the **Import** button in the **Scanning Offset Adjustment** section of **Device Settings** to load and apply an existing `.lbso` file containing Scan Offset settings. Some manufacturers provide these files to simplify the process. You can also export and import your own Scan Offset settings to move them between different computers.

Exporting

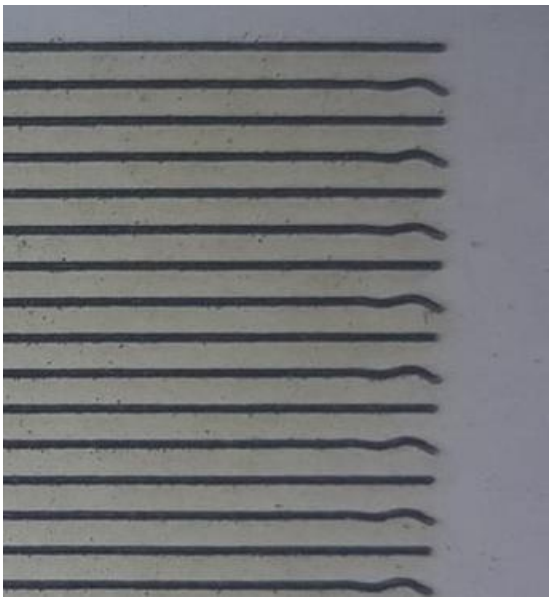
Use the **Export** button in the **Scanning Offset Adjustment** section of **Device Settings** to create a `.lbso` file that can later be imported on another computer to transfer Scan Offset settings.

> **Read and Write Buttons**
>
> Some Ruida controllers allow the ability to directly **Read** Scanning Offset Adjustments stored in the controller's settings, or **Write** new settings from LightBurn.
>
> These buttons only appear when LightBurn is directly connected to a Ruida controller that supports the Read/Write functions.



Line Wobble

A different, but equally common problem is line wobble, often caused by too high an acceleration setting. When performing the test engraving above, you may notice lines that look like this:



If so, your machine is moving too quickly between the rows, and you're seeing physical "bounce" in the gantry because of it. Lowering the acceleration setting for your Y Axis can correct this.

Related Topics

- Mechanical Issues
- Fill Mode

WeCreate Camera Calibration & Alignment

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=wDxY-taFj80

These instructions are no longer necessary for LightBurn 2.1 or newer. If you are using a WeCreat laser, please update to LightBurn 2.1 or newer and use the general instructions for Camera Calibration and Camera Alignment.

For the convenience of users with LightBurn 2.0, we are keeping these directions available. You can also jump to the documentation for LightBurn 2.0 by clicking the version dropdown at the top of this page or going to https://docs.lightburnsoftware.com/2.0/.

wecreate

> **Note**
>
> These instructions are only for users with WeCreat Vision lasers. Follow our standard camera Calibration and Alignment instructions if you do not have a WeCreat vision laser.

Preparation

In order to complete this process, you will need:

- WeCreat firmware version 2.0.24 or later. To update, open WeCreat MakeIt and go to **Settings → Firmware → Check for Updates**. If you need assistance with the firmware update process, contact WeCreat support.
- LightBurn version 1.7.00 or later.
- A WeCreat device profile in LightBurn, created by importing the `.lbdev` file supplied by WeCreat.
  - Download the file for your machine directly from WeCreat here.
  - For instructions on setting up the device profile, see here.
- A piece of material in your laser that is large enough to engrave the target marker pattern onto — it must be at least 200 mm x 200 mm (roughly 8" square).

Steps

1. Select the WeCreat device from the dropdown menu in the Laser Window to make it active.
2. Place the material you're using for calibration in your laser's work area and make sure the laser is focused to your material. If your laser isn't focused:
   - Use the jog controls in the Move Window to move the head of the laser over your material.
   - If you're using a laser that supports auto focus, use the Focus Z button to automatically focus the laser. Currently, this is only supported for the WeCreat Vista.

     > **Warning**

> You must have WeCreat firmware version 2.0.24 or later in order to use the Focus Z button.

- For lasers that don't support auto focus, please check your user manual and focus according to WeCreat's directions.

3. Open the Cameras / Overlays Window and make sure it is displaying an image from the WeCreat camera, indicating that it's connected.
4. Right-click in the Cameras / Overlays Window and select **Calibrate Camera Lens**.
5. In the **Lens Calibration Wizard**, select the WeCreat camera from the list, then select **(None - precalibrated camera)** from the **Use Preset** dropdown menu.

   > **Note**
   >
   > This option is used for the WeCreat camera because the device automatically corrects for the lens distortion — no additional corrections are needed.

6. Click **Next**, then **Align Camera** to open the **Camera Alignment Wizard**. Make sure the WeCreat camera is still selected.
7. If your laser is not currently focused to the material in your work area, exit the Camera Alignment Wizard and go back to step 2 to focus your laser, then go to Laser Tools → **Calibrate Camera Alignment** to re-open the Camera Alignment Wizard.
8. Select **Camera is over the work area, in a fixed position**. Select the WeCreat camera from the list and click Next.
9. Choose between **Fit Workspace** or **100% Scale**. The 100% Scale option is roughly A4 or US letter sized pattern. Fit Workspace scales the pattern up larger, if your laser's working area can accommodate a larger size, which can sometimes improve the accuracy of your alignment.
10. Enter appropriate **Speed** and **Power** values for both **Fill** and **Line**. The precise values you need will depend on your model of laser, and the material you're using. For the WeCreat 20W, we chose 3000 mm/minute for Speed, and 30% power.

    > **Warning**
    >
    > Make sure to adjust these settings to appropriate values for your material and laser, and if you use WeCreat's recommendations note that they use mm/second, not mm/minute, and convert appropriately. If you have the 40W model, be sure to adjust to appropriate settings for your machine — you can lower the power, set speed a bit higher, or some combination of the two.

11. Press **Frame** to make sure the area the pattern will engrave is aligned to your material.
12. Once the engraving is done, click **Cancel** to close the window, then click the **Home** button in the Laser Window to return your laser to its top Z height.
13. Once it has reached the top position, re-open the Camera Calibration Wizard.
14. Make sure the WeCreat camera is still selected then click Next, and Next again, to proceed past the window where you set up engraving parameters and ran the job.
15. Click **Capture Image**, then Next.

16. Tag each of the target markers by double-clicking in the center of each one in order.

    You can pan and zoom around the image using the same controls as the LightBurn Workspace and Preview window, or use the **Zoom In** and **Zoom Out** buttons.

    When you double-click, a red "+" mark will appear. Place a tag in the center of each of the four targets, in the order they are numbered (1, 2, 3, 4). If you place one incorrectly, you can double-click near it to shift it around, or click **Undo Last** to remove it and try again.
17. Once you've marked each target, click Next and then **Finish** to exit the Camera Alignment Wizard.
18. Click Update Overlay in the Cameras / Overlays Window to capture an Overlay for your Workspace. You can use this Overlay to align graphics over the material in your laser's work area.

    > **Note**
    >
    > Your WeCreat must always be at its top Z height prior to capturing an Overlay, or the resulting capture will be misaligned.

Firmware Configuration

GRBL Configuration

Most GCode-based lasers are pre-configured to work with LightBurn, but if you have a less common machine like a Shapeoko, EleksMaker, X-Carve, or Acro system, you might need to make some simple changes to get the most from LightBurn.

Quick Reference

- You might need to adjust your spindle max RPM value (`$30`) to match the LightBurn default (1000) or vice versa. The value in LightBurn is called **S-Value Max** in Device Settings.
- You might need to enable **Laser Mode** if you have GRBL 1.1f or later (`$32=1`)
- If you have an older version of GRBL (prior to 1.1f) upgrading the firmware is highly recommended, as Laser Mode also prevents the machine from pausing with every power change. The pause, which happens on older versions, or when not using Laser Mode, will cause excessive burn spots when engraving images.
- If your machine uses negative workspace coordinates you'll need to enable CNC Machine in Device Settings.
- Make sure the controller is reporting positions in mm, as expected by LightBurn (`$13=0`)
- If your machine does not have homing switches (also called limit switches) you will need to home it manually if you want to use Absolute Coords or User Origin modes
- LightBurn works best with the G54 Workpiece Coordinate Offset set to 0,0,0. You can set this in the console by entering `G10 L2 P1 X0 Y0` and pressing enter, while connected to your machine. If you do need to use the G54 offset, enabling Machine Coordinates will fix some problems, but not all. In particular, User Origin workflows and Continuous Jogging may not work correctly with a G54 offset.

GRBL Flavors

GRBL firmware was originally designed for CNC machines and 3D printers, with laser support added more recently. It is highly configurable, and this is both a blessing and a curse. The "standard" way a CNC machine is configured is somewhat different from the way laser machines often are. Luckily this is easy to change, and easy to switch from one to the other.

More recent versions of GRBL (1.1f and up) support two laser-oriented features. The first is Laser Mode, enabled by setting $32=1 in the firmware settings. Laser mode eliminates the pauses that happen when changing power output, because GRBL knows it's controlling a laser which reacts instantly, instead of waiting for a spindle to change RPM.

The second is a feature called variable power mode, or the M4 command. In this mode, GRBL adjusts the laser power as the machine speeds up and slows down, making for very consistent cutting and marking. Older versions of GRBL do not have this feature, and simply run the laser at a constant power output for the duration of a cut. Since the machine needs to slow down to take sharp corners, this means corners get over-burnt, while long straight lines end up lighter. This also has the benefit

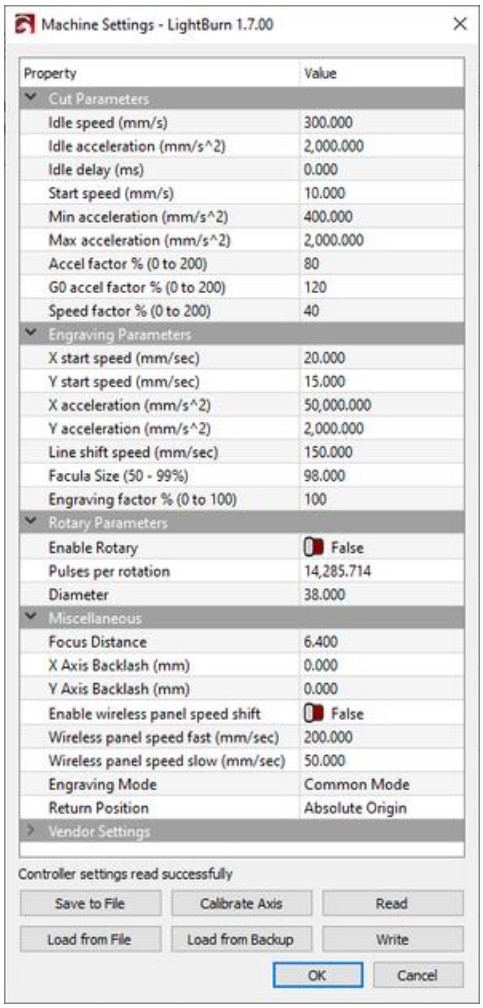
that when the laser comes to a complete stop, the beam turns off (zero speed equals zero power), meaning that pausing a job automatically turns off the laser. This is not always true with other versions of GRBL.

If you aren't already running GRBL 1.1f (or later) on your controller, we highly recommend it for laser use. If this isn't an option, that's ok, but your results won't be as good, and pausing the laser runs the risk of leaving the beam on and ruining the job.

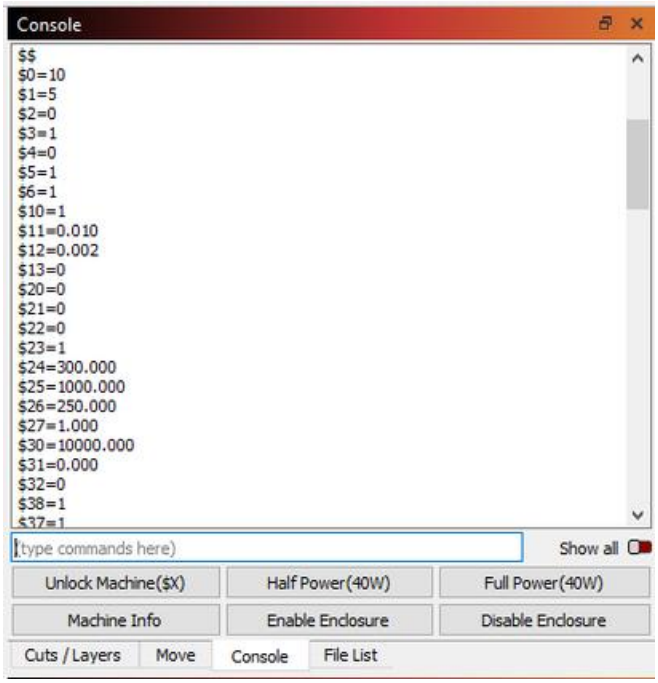
Getting Machine Settings

Many GRBL based machines allow the settings of the machine to be easily exported or viewed. For advanced configuration like adding limit switches, updating firmware, or setting up a new controller, you'll need to interact with these values. You can do this in LightBurn by using the **Console** window or the **Machine Settings** window to copy, update, or reset your machine settings.

The Machine Settings window provides you with a formatted table with the parameters labeled. You can also export these settings to a file for later restoration.



In the Console Window, enter `$$` to have the controller return the machine's configuration. This will show the settings in plain text, requiring more familiarity with what each command means but allowing you to view the configuration directly, without any chance of mislabeling based on custom vendor configurations.



Other Machines

If you aren't sure how to configure your machine, the following steps will help you figure it out.

Figure out which firmware you're running

In LightBurn, when you first connect to the machine the console will typically show a "hello" message from the controller. For Smoothieware boards it is just `Smoothie`. For GRBL, it will be `GRBL 1.1f [$ for help]` or similar - this tells you it's GRBL, and which version. Machines using GRBL 1.1f or later will support the M4 variable power command, and just use the "GRBL" driver in LightBurn. GRBL 1.1e or older (GRBL 1.0, GRBL 0.9, etc) must use the GRBL-M3 device in LightBurn.

Find the machine origin and coordinate direction

1. Home the machine by pressing the **Home** button in the **Move** window.
2. In the console window, type `G0 X0 Y0` and press `Enter`. This will command your machine to head toward its origin position - the location represented by (0, 0) in the machine's coordinate system. This is not always the same as the home position. The origin is usually either the front left corner or the rear right corner of the machine, but can be a different corner or the center of the work area. If your machine moved to the center, skip to Center Origin Machines for more information.
3. After the machine has stopped moving from the previous command, type `G0 X10 Y10` and press `Enter`. If your machine moves 10 mm into the work area on both axes, your machine uses positive workspace coordinates. You can set the origin in LightBurn to match the machine origin you found in step 2. If your machine bumped the rails, it uses negative coordinates.

Machines without homing sensors or limit switches

If your machine does not have homing switches (also called limit switches) you will need to home it manually if you want to use Absolute Coords or User Origin modes. There are two ways to do this:

- With the machine off, manually move the laser head to the origin position (usually front-left), then power up the machine. Until you tell it otherwise, the power-on location of the controller is treated as the zero position.
- With the machine powered, jog the laser head to the origin position. In the console window, type: `G92 X0 Y0` and press `Enter`. The `G92` command tells GRBL to set the current location as the specified coordinate, so you're telling the machine "this is zero". You will also need to set `$10=0` for this to work correctly.

If you use the second option frequently, you may want to save the command as a macro.

xTool Wi-Fi Upload

We do not currently support uploading GCode to the xTool family of lasers via Wi-Fi. The upload mechanism is significantly different from the currently supported methods, and that development hasn't yet been completed. Thank you for understanding.

Legacy Instructions: Negative Coordinate Machines

Legacy Instructions: Negative Coordinate Machines

If you're using a version of LightBurn released before 1.6, follow these instructions to work with a machine that uses negative coordinates.

LightBurn expects positive workspace coordinates. If your machine is configured to use negative workspace coordinates, you can set a workspace offset to enable compatibility with LightBurn. If you are switching between LightBurn and CNC software that expects a negative coordinate system, you can clear the workspace offset. You can also set macros to toggle between the two modes.

Set A Workspace Offset

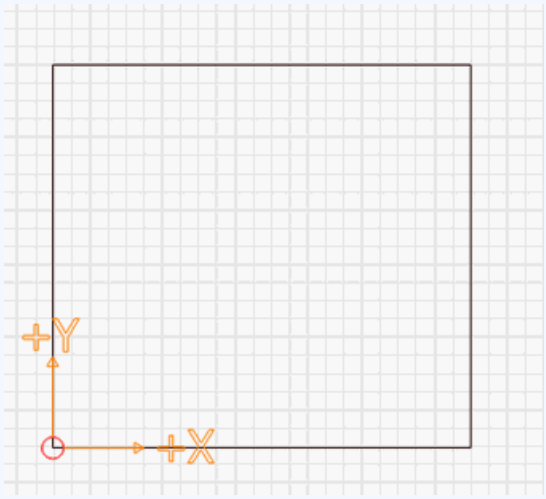
In the console, enter `G10 L2 P1 X-WIDTH Y-HEIGHT` and hit `Enter`. Replace `WIDTH` and `HEIGHT` with the actual X and Y travel distances of your machine. Set your machine to use workspace coordinates with the `$10=0` command.

Example: `G10 L2 P1 X-310 Y-180` `Enter`

Clear a Workspace Offset

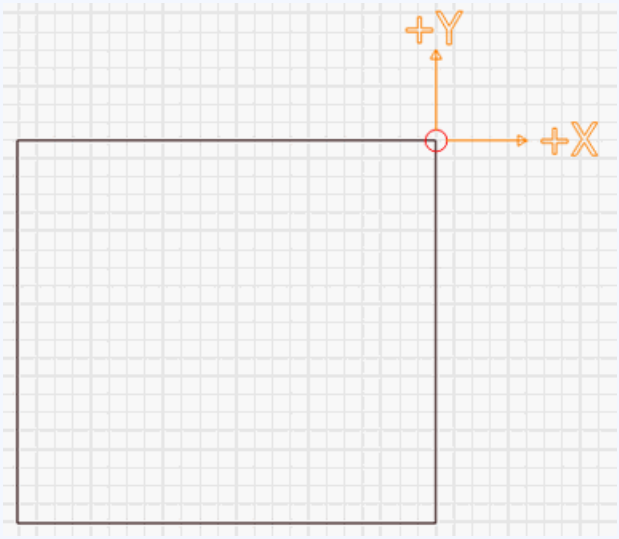
In the console, enter `G10 L2 P1 X0 Y0` . This is generally only required if switching between using your machine as a laser (with an offset) and as a CNC machine in software that expects negative coordinates.

Detailed Explanation.



This image shows the origin at the front left, with positive X values moving to the right, and positive Y values heading to the rear of the machine.

A negative workspace system looks like this:



In this image, the origin is at the rear right of the machine. The X and Y directions are the same as before, but now you would need to use negative numbers to move into the work area.

You'll need to know the total distance your machine can travel in both axis. For a Shapeoko XXL, for example, it's 812 mm in X and Y. For a 1000 mm 2nd Generation X-Carve, it's 750 mm in X and Y - note the difference between overall size and travel distance. By applying a workspace offset that is the size of your machine area, you can shift the offset to the opposite corner, like this:

Entering `G10 L2 P1 X-812 Y-812` means set an offset (`G10 L2`) in the first coordinate system (`P1`) with a distance of -812 mm in each direction (`X-812 Y-812`).

If you enable a workspace offset, you will also need to make GRBL report its location relative to this shifted origin, instead of in "machine space", by setting $10=0. Some systems, like Easel or Carbide Motion, may need a different value, so it's a good idea to make a note of the existing setting.

Set Macros

For a one click way to set or clear workspace coordinates, you can create macro buttons for these commands in the Macros Window. See the linked page for directions.

Once the macro is saved, you can use the command by pressing the appropriate button.

Center Origin Machines

Some systems have their origin in the center of the workspace. To confirm that this is the case, after homing your machine enter `G0 X0 Y0` in the console and press `Enter`. This command says "rapid move to coordinate 0,0". If your machine travels to the center of the workspace, it's configured for a center origin.

A center origin machine requires a workspace offset similar to what's described in Negative Coordinate Machines, but the offset should be half the travel distance. For instance, a machine with a travel distance of 500 mm x 500 mm and a center origin would require `G10 L2 P1 X-250 Y-250`.

Configuring a Ruida Controller

> **Warning**
>
> A factory reset of a Ruida controller resets it to the stock configuration supplied by Ruida, not the configuration your laser's manufacturer applied to it. Because of this, a factory reset is not recommended. If you've already done it, this guide is here to help.

When you purchase a laser with a DSP controller installed, the manufacturer configures the laser controller to match the machine configuration.

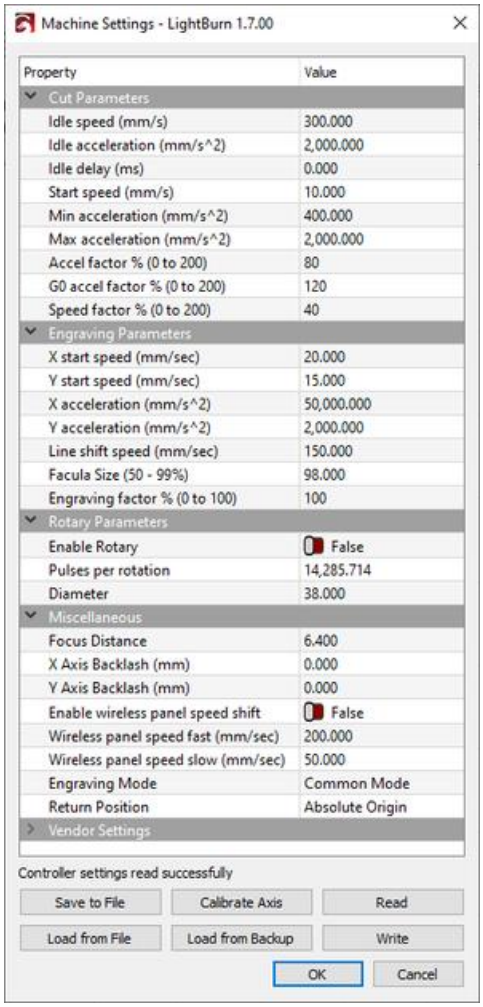
This guide is here to help you if you need to do this configuration yourself, either because you are installing a new controller into a machine or because you did a factory reset of your controller.

Ruida Machine Settings in LightBurn

> **Tip**
>
> It's a good idea to use the **Save to File** button to back up your settings before making changes, so you can revert to what you had before if needed.

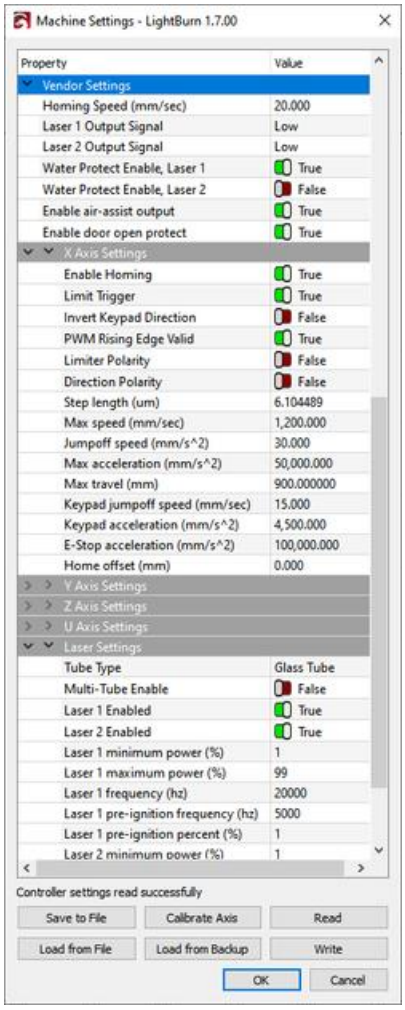
With the laser/controller connected to your computer and powered on, go to Laser Tools → **Machine Settings** to open the Machine Settings window. LightBurn will automatically read the settings from the controller when this window is opened.

As you're making changes, the **Read** button will tell LightBurn to read the configuration from the controller, and **Write** will save the settings in LightBurn back to the controller. For more information on the Machine Settings window, see our page on Machine Settings.



The top section of the list is referred to as user settings – these are settings that are intended to be tunable by the user if necessary. The lower section, under **Vendor Settings**, are things usually configured for you by the factory and should only be changed if you have a good reason for doing so.

Click to expand the Vendor Settings section, and, after acknowledging the warning, you'll see the below settings.



The first section is general configuration, and then there are settings that are specific to each axis, followed by a Laser Settings section, for configuration specific to the laser type.

Homing, Directions, and Travel

A DSP laser will have homing switches – one on each axis. They might be physical lever-type micro switches, optical beam-break switches, or inductive proximity sensors. Locate the switch at the end of the X axis, and the one for the Y axis. Note that a Ruida controller homes using the X- and Y- limit switches.

When you power up the laser, the first thing it will do is move toward the corner it thinks those switches are at to home itself. If it's moving in in the wrong direction, you'll have to hit the ESC button on the controller itself to stop it.

To fix this, you'll need to go to the **X Axis Settings** and **Y Axis Settings** under Vendor Settings. Near the top of each will be three check boxes:

- Invert Keypad Direction: Controls which way the arrow buttons move the laser.
- Limiter Polarity: Tells the controller whether the limit switch triggers high (normally open) or low (normally closed) when contacted.
- Direction Polarity: Controls which way the motor moves in general.

Step 1: Correct direction of travel

Before continuing onto other steps, make sure that the X axis and Y axis move properly when you power on the machine.

There are only 4 possible combinations of **Limiter Polarity** and **Direction Polarity** for each axis. Without access to your machine, we can't tell you the correct combination, but change those settings for the X axis until it moves properly and recognizes the homing switch when you power up the machine. Once the X axis is correct, do the same for the Y axis.

Step 2: Correct keypad controls

When the limiter and direction settings are correct, check that the keypad arrows on the machine are moving the laser in the correct direction. If not, toggle the **Invert Keypad Direction** button for whichever axis is wrong.

Step 3: Set Max travel

You will also need to set the **Max Travel** value for the X and Y Axes - these values dictate the length of each axis, and together define the size of the work area of the machine.

Step Length Calibration

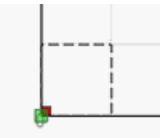
The next part is figuring out how far the laser moves when you tell it to, and how far off it is. The controller needs to know how far a single step moves when it sends a step pulse to the motors so it can translate real measurements into the proper number of steps.

Rough Calibration

You'll need to start by doing rough adjustments to get close enough to the correct values to be able to calibrate the step length properly.

In LightBurn, set the **Start From** setting in the Laser Window to **Absolute Coords**. See Coordinates and Job Origin for help.

Draw a small rectangle at the origin. Start with 10 x 10 or 20 x 20 mm, like this:



> **Tip**
>
> Note that your origin corner might be in a different spot than shown here. You should set it in Device Settings to match the origin corner of the machine.

Now, use the **Frame** button and compare the size of the frame to the size of the rectangle.

- If the laser is barely moving, your step size is too big – the controller thinks it's moving more than it actually is.
- If the framing is much too large, the step size is too small – the controller thinks it needs to take more steps than are actually necessary.

Adjust the X Axis and Y Axis settings for **Step Length** based on what you saw with the framing. The changes are proportional to the difference between the size of the rectangle and the amount of travel:

- If the laser moved twice as far as it should, double the step length value.
- If the laser only moved half as far as it should, cut the step length in half.

Frame again and repeat the adjustment process until the size of the framing is reasonably close to the size of the rectangle you drew.

Final Calibration

Draw a box similar to the first box, but make it larger – 100 mm or 200 mm is generally a good size to work with. The larger box makes it easier to get precise enough measurements without specialized equipment.

Set the Min Power and Max Power values low, but high enough to make a mark. Run the job to engrave the rectangle on a piece of scrap material. Measure the result as accurately as you can.

This time, you'll need to do just a of math to calculate the with the step size:

- New Step Size = (Current Step Size * Measured Length) / Requested Length

As you are dialing in the above settings, if the steppers skip or make buzzing sounds, you might need to reduce the **Idle Acceleration** or **Idle Speed** settings near the top. Those things will require tuning with some trial and error.

Fixing Skewed Engraving

An occasional problem with new setups, and sometimes even existing machines, is lines cutting correctly, but engravings coming out slanted or skewed, like this:



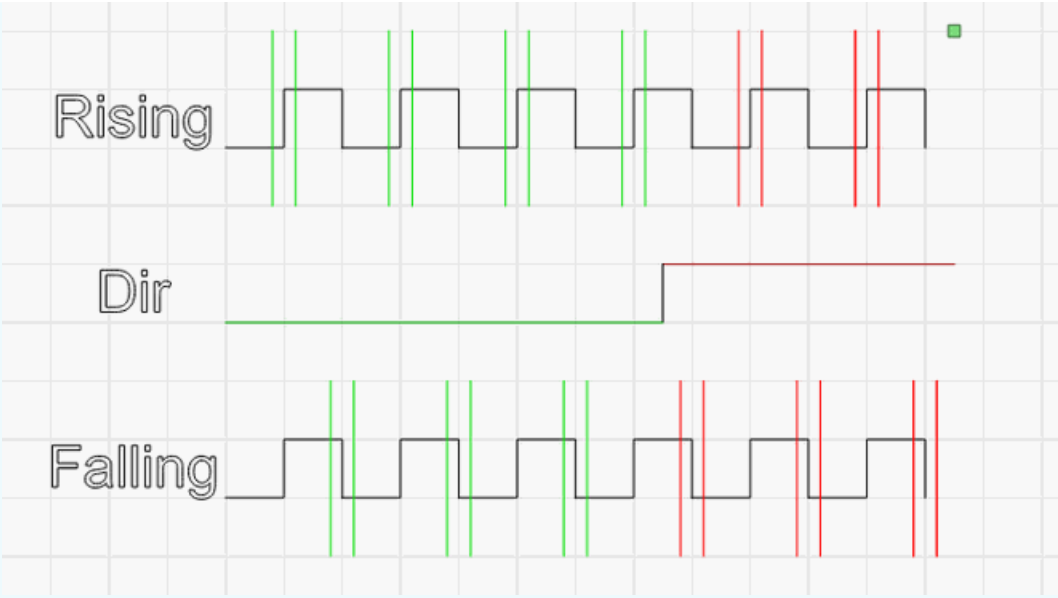
If your output looks like this, you likely have your motor pulse step polarity set incorrectly. On Ruida controllers, each axis has a setting called **PWM Rising Edge Valid**. This setting tells the controller whether the rising edge or falling edge of a step pulse is what the motor driver is looking for, and fixing it may correct skewed engraving.

To fix this problem, Find the **PWM Rising Edge Valid** setting on the X axis, and toggle the value – enable it if it's disabled, or disable if it's enabled. Do the same for the Y axis.

What is this setting and why does it matter?

A step pulse is a transition from low to high, or high to low. The controller will hold the line low, and pulse it high, or hold the line high, and pulse it low. The transition itself is what matters, and motor drivers will either look for a transition from low to high (rising edge) or high to low (falling edge) to accept as a **Step**.

If the laser controller believes that the motor driver is looking for the leading edge signal (when it transitions from low to high), it will pulse the line, and could change the direction line immediately after that. If the motor driver is waiting for the falling edge, it will see the direction change BEFORE the falling edge of the pulse, meaning that it will change direction one step too soon.



In the image above, the upper line of steps is interpreted as 4 steps in one direction, then two in the other. The lower line is interpreted as 3 and 3, and the only difference is which side of the step signal the driver is looking for.

Start and End Delay

This feature adds in a delay at the start or end of jobs. This is often used to give fants time to get to speed at the start of job or finish venting the air in the laser before stopping.

Related Topics

- Machine Settings

Galvo Laser Guides

Changing a Galvo Laser Lens

To ensure optimal performance following a physical lens swap on a Galvo laser, there are several changes and settings you'll need to confirm. Some manufacturers provide the calibration information for your new lens, and we strongly recommend you check their website before proceeding to perform the calibration manually.

Physical Lens Swap

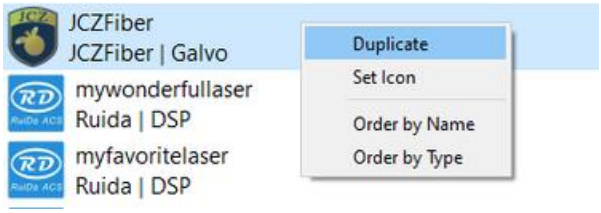
This video guide from Laser Everything covers the physical installation of the lens, as well as many of the items outlined below with EZCAD analogues. We suggested you review this video by Laser Everything to help ensure the lens swap doesn't damage your machine or hurt you in the process.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=EdSDWhuYcCc

Duplicate Original Laser

In many cases, changing the galvo lens means that you'll need to adjust many settings and recalibrate it, effectively making it a "new" machine when you do so. To duplicate a device in LightBurn, click Devices in the Laser Window, right-click on a given laser, and select Duplicate.

> **Tip**
>
> Be sure to give the duplicate a descriptive and accurate name to contrast it to the original, and ensure that it's the on you edit later.



Focus

After changing lenses physically, you'll need to adjust the focus of the laser, even before other performing the calibration process, to ensure the results are as good as possible.

Review the steps outlined in Galvo Laser Focusing to ensure your laser's output is crisp and in focus — keep in mind your red dot framing may no longer be in focus due to the focal length changes.

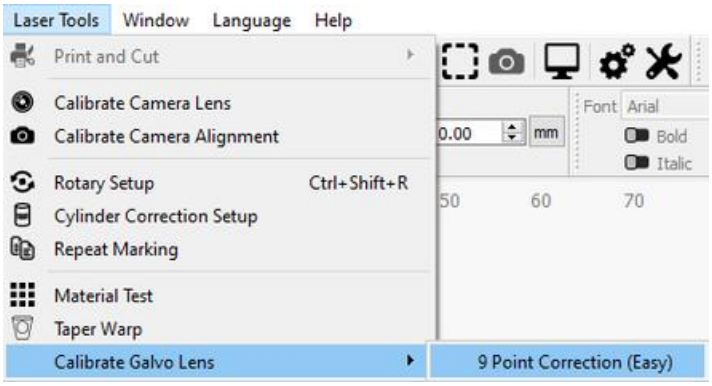
Field Size and Lens Calibration

All Galvo lenses have distortion which must be corrected in order for output from your laser to have an appearance that's true to the design you see in your LightBurn Workspace.

When working with a new laser, you can usually import lens correction values provided by your machine's manufacturer when you first set up your laser with LightBurn.

If your manufacturer didn't provide the necessary file, or you purchase a new lens for a laser you already own, however, you may need to use LightBurn's **Galvo Lens Calibration** wizard to find the appropriate corrections.

To open the Galvo Lens Calibration wizard, go to Laser Tools → **Calibrate Galvo Lens** → **9 Point Correction (Easy)**.



For more information on this process, see Calibrate Galvo Lens — Galvo Lens Calibration Wizard.

Related Topics

- Add a Galvo
- Red Dot Offset

Installing Galvo Drivers

> **Note**
>
> These instructions are for Windows users only — macOS users do not need to install a driver to connect to a Galvo laser.

If you're using a Galvo laser with Windows, you'll need to make sure the correct driver is installed.

If you have an EZCAD-based laser, the easiest way is to check the **Install EzCad2** or **Install EzCad3 Driver** box in the LightBurn installer. If you've already installed LightBurn without installing the driver or if you need to update the driver to support an EZCad3 laser, you can re-run the installer with the driver checked to install or update it.

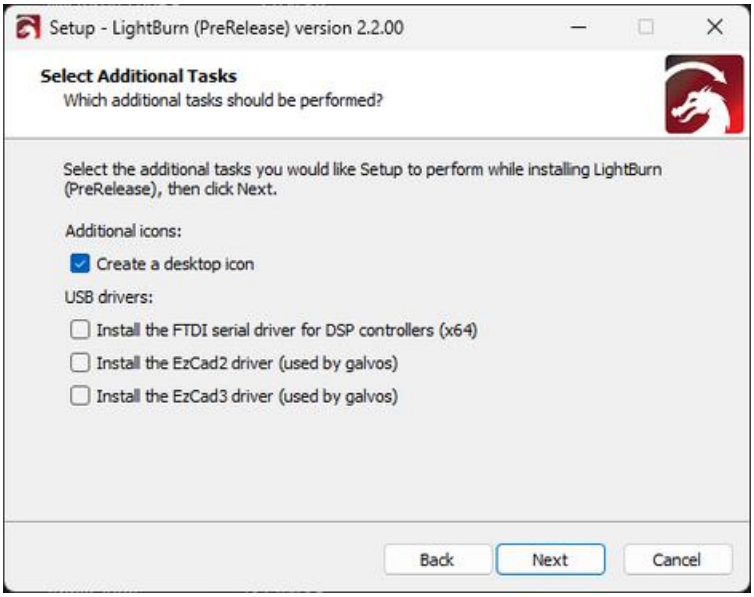
> **EZCad2 and EZCad3 Driver Support**
>
> LightBurn 2.1 installed a combined EZCad 2 and 3 driver. LightBurn 2.2 has separate drivers for EZCad2 and EZCad 3. If you're using older drivers, please update to the new one using the LightBurn installer.

If your laser uses a **BSL** control board, you may need to install a different driver by installing your laser's stock software, or searching the files provided by your machine's manufacturer for an executable file named `CypressDriverInstaller.exe` or similar. Both the stock software and driver are usually provided with your laser on a USB drive.

> **Not sure if your laser has an EZCAD2, EZCAD3, or BSL board?**
>
> If your laser uses EZCAD2 as its stock software, it has an EZCAD2 control board.
>
> If your laser uses EZCAD3 as its stock software, it has an EZCAD3 control board.
>
> If your laser uses SeaCAD, SimpleCAD, or BSLCAD as its stock software, it most likely has a BSL control board. There are many other variants — consult your laser's manufacturer for further assistance identifying your type of laser.
>
> If your laser uses different software, you may need to investigate further. Some lasers use a rebranded version of one of the programs listed above, such as the GWeike G2 Pro, which uses GLaser and is a BSL machine. You can see our compatibility matrix for more information, or contact support for help.

Installing the Driver (EZCad2 and EZCad3 Lasers)

When installing LightBurn, the last screen of the installer has a set of checkboxes to install additional drivers needed for certain types of machines. You'll need to select **Install EZCAD2 Driver** or **Install EZCAD3 driver** in order for LightBurn to communicate with the Galvo laser you're trying to set up.



If you didn't check this box during initial installation, you can run the installer again, and select the box to ensure the driver is installed. Windows will prompt you for permissions escalation to install it, and replace the driver provided by default with one LightBurn can use to communicate with your galvo-based laser.

If you prefer to install the driver manually, you can find the installer at `C:\Program Files\LightBurn\EzCad2Driver\` and can be run again via the `dpinst-x64.exe` file directly.

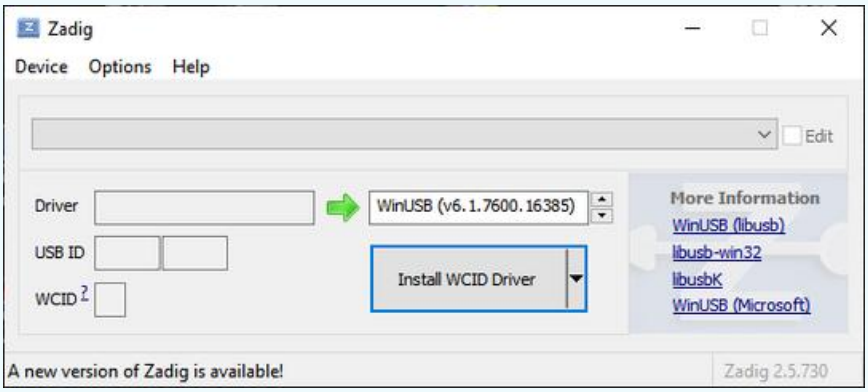
Legacy Zadig-based Driver Swap

Legacy Zadig-Based Driver Swap

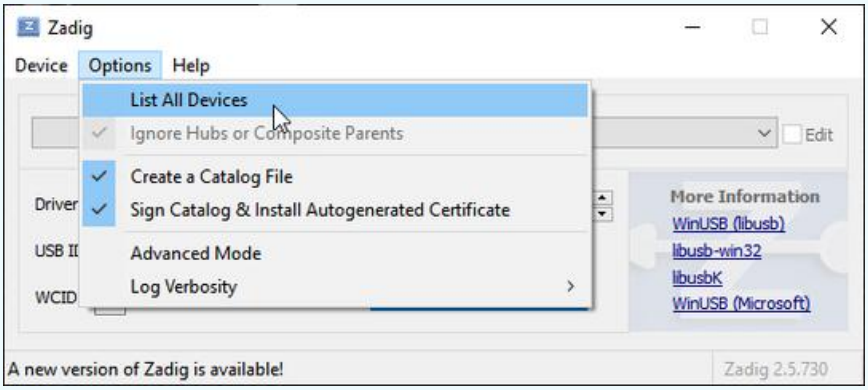
If the driver installed automatically isn't working, you can try using Zadig to replace the manufacturer's driver with a generic one so LightBurn can communicate with it. This occurs most often on Windows 7 computers.

Download Zadig, the driver replacement tool from https://zadig.akeo.ie/ and run it. Zadig doesn't need to be installed.

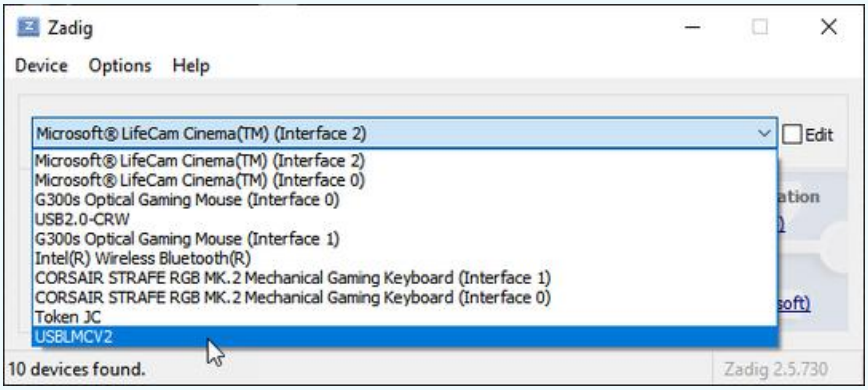
You'll see a window that looks like this:



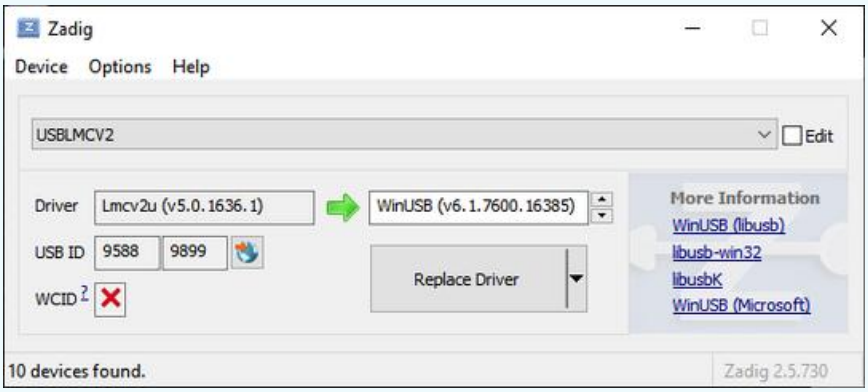
In Zadig, go to **Options → List All Devices**.



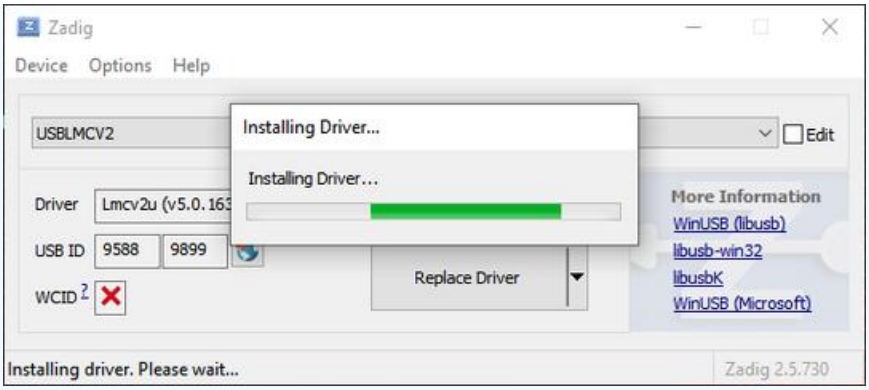
If your EZCAD laser is powered up and connected with the USB cable, you should see **USBLMCV2** (or very similar) in the device list, like this:



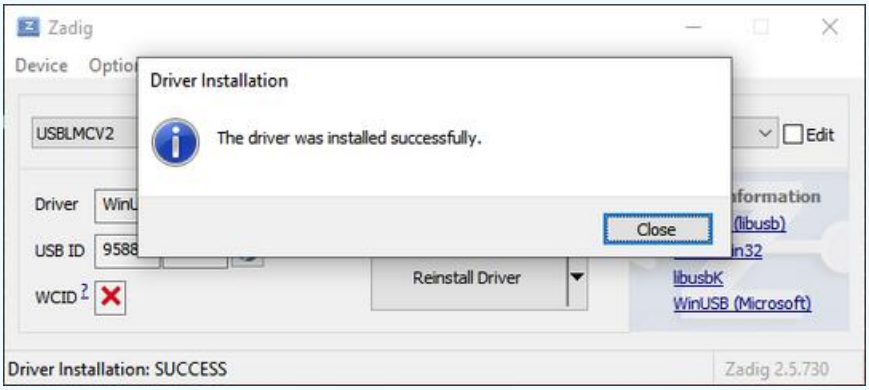
Select that device and choose **WinUSB** as the target driver, like this:



1. Double check that it says **LMCV2U** in the **Driver** box, and that the USB ID under it matches what you see above. If you haven't used the laser with this computer before, the **Driver** box may be empty instead.
2. Click **Replace Driver** and wait for the process to complete. Make sure that you are not running EZCAD when you do this — if anything is actually using the hardware, the driver swap will likely not complete, and time-out after five minutes. If everything goes as expected, it takes about a minute, often less.
3. You'll see some different messages cycle through while it does the install. When it's done, it might ask you to reboot, but that's rarely required.



4. On the rare occasion that Zadig fails, try it again — it almost always works, and sometimes reports a failed installation erroneously, when it actually succeeded. You should see this:



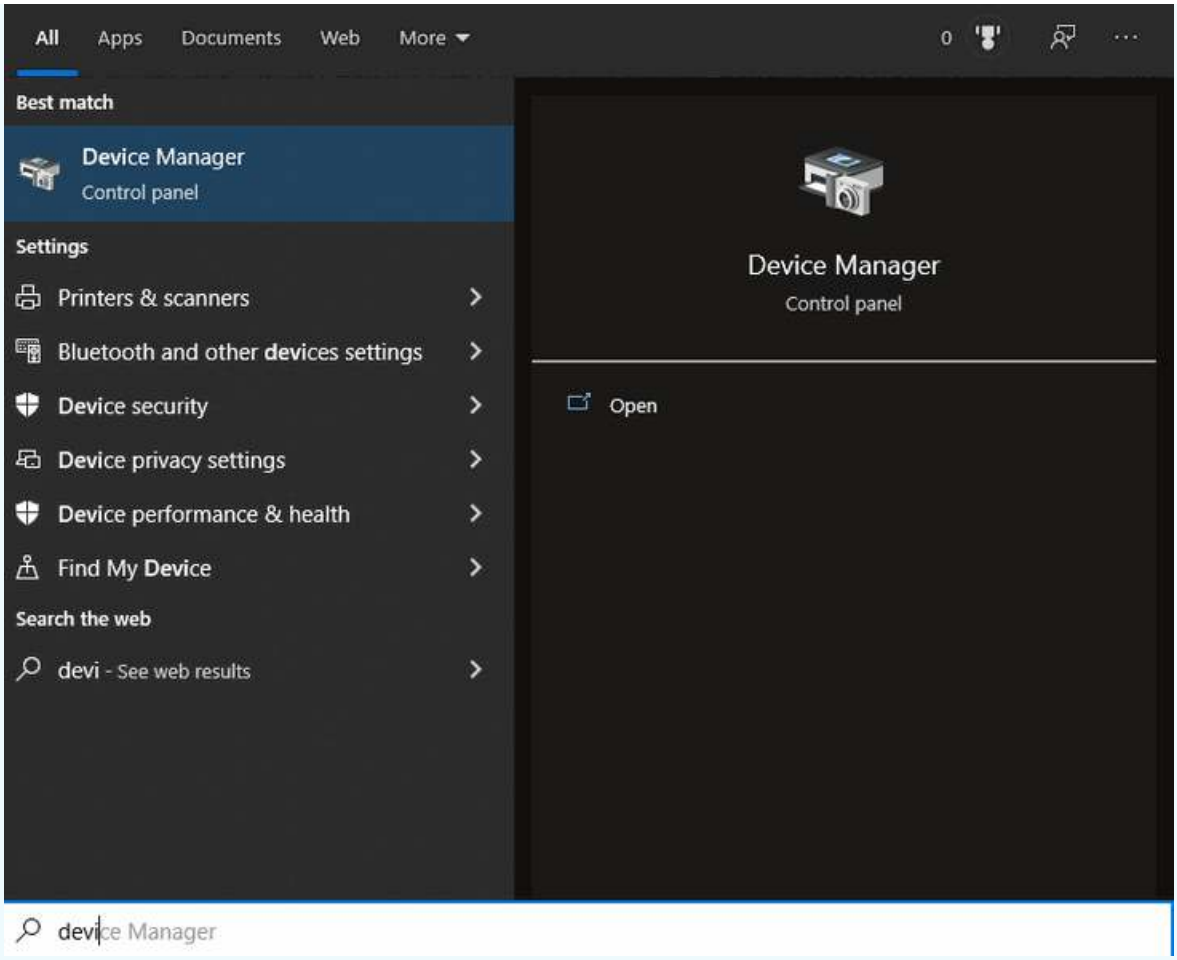
Restoring the EZCAD Driver

Restoring the EZCAD Driver

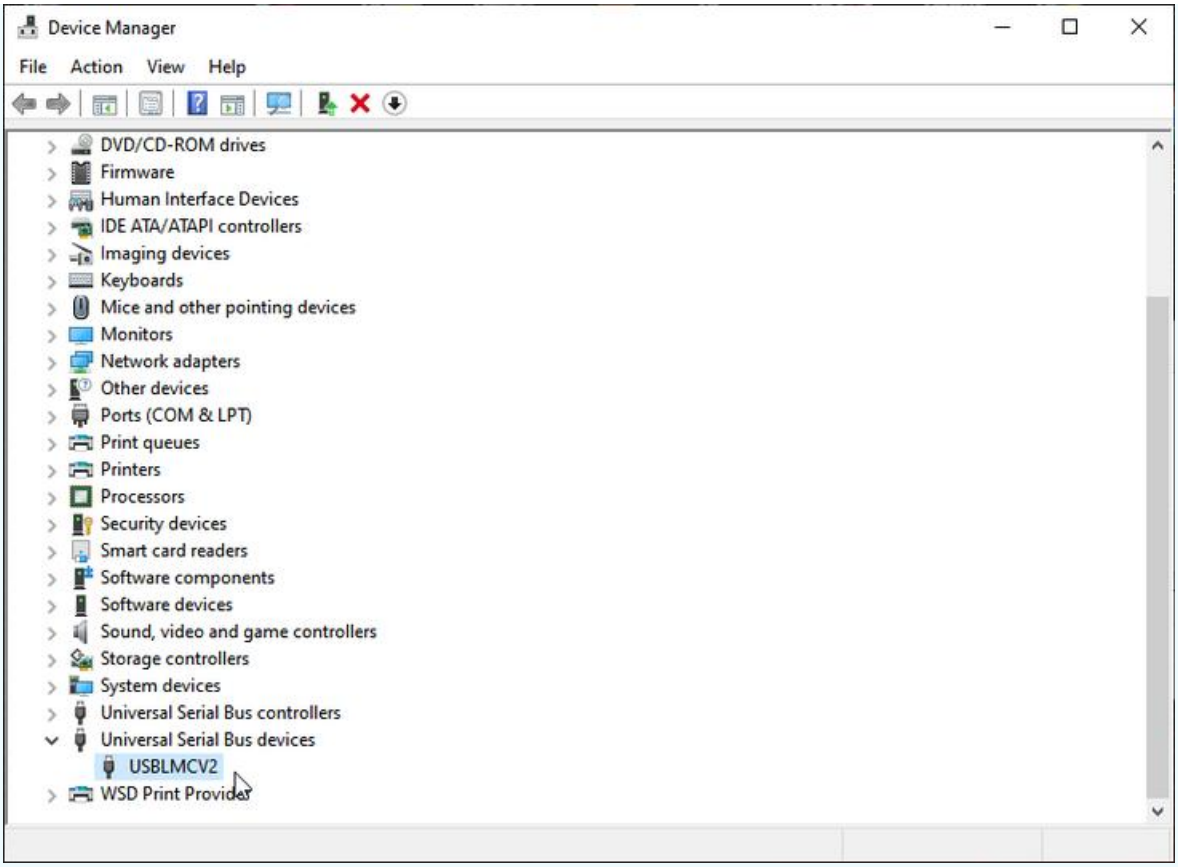
In the event you want to use EZCAD, you'll need to return to the original driver. You can switch back and forth as many times as you need to.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=J7nsXEUQ1_8

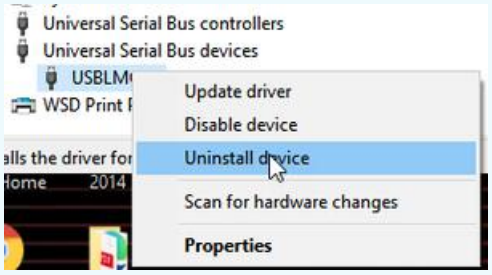
Open the Windows Device Manager.



Next, find the **USBLMC** device in the list - it should be near the bottom, under **Universal Serial Bus devices**, shown below:



Right-click the device and choose **Uninstall Device** from the pop-up menu.



In the next window, check the box to delete the driver. This doesn't actually delete anything from your system, it just completely disassociates the driver from this device.



Click the **Uninstall** button. The process should finish quickly.

If you power off the laser and power it back on, Windows will reconnect, realize that it already has a driver for this device (the old LMC driver) and it will appear in the Devices list again, near the top, as a **BJJCZ Device**. Now you can use EZCAD again.

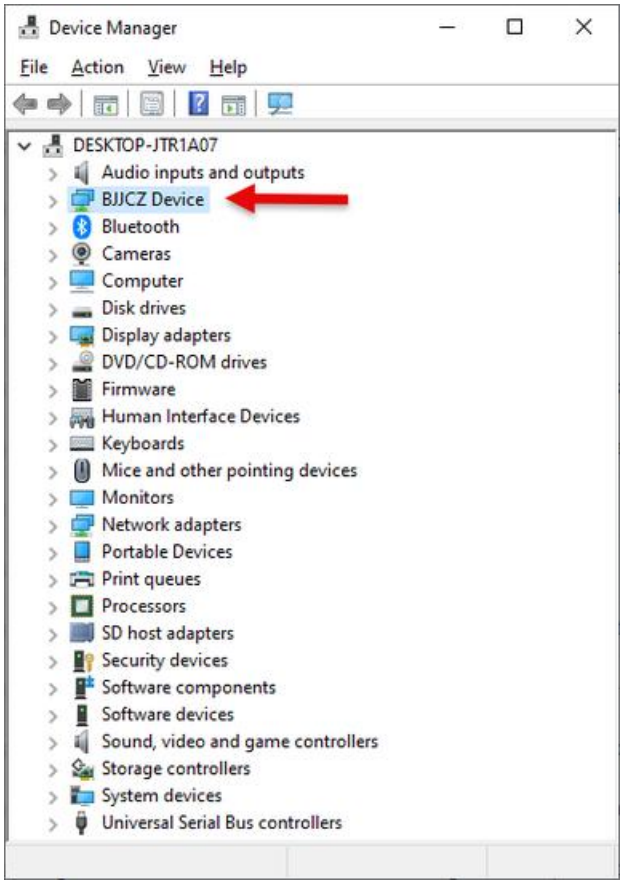
Verify Driver Installation

Opening Device Manager

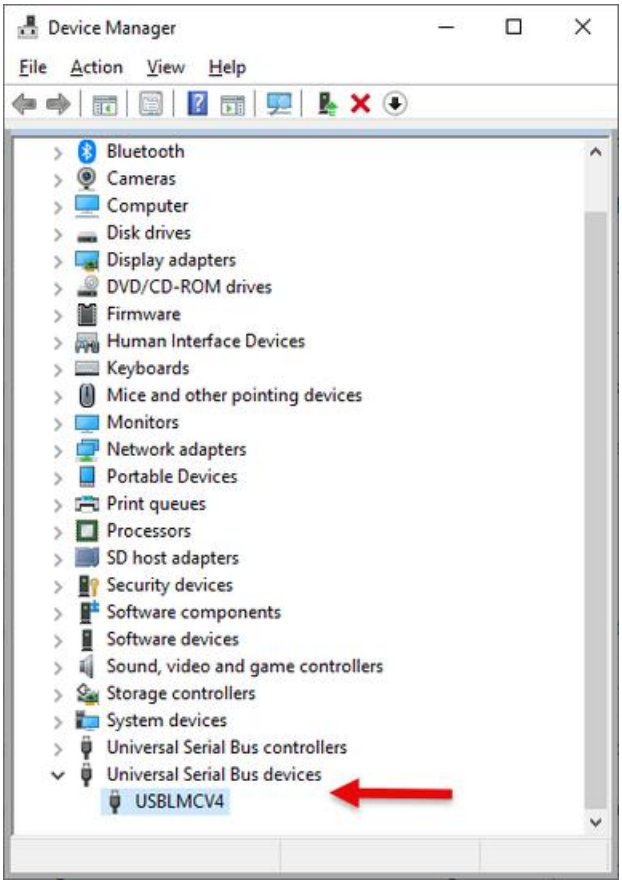
To open the Device Manager, type "device manager" in the search box on the taskbar, then select Device Manager from the menu.

Your laser will appear differently in the Windows **Device Manager** before and after the driver has been successfully installed.

- Before installing the driver, the laser will appear as a **BJJCZ Device**. If you see this, the correct driver is not installed.
- After installing the driver, the laser will appear under **Universal Serial Bus devices**:
- For EZCad2 devices, you should see something along the lines of **USBLMCV2**, **USBLMCV4**, or something else with **LMC** in it.
- For EZCad3 devices, you should see a name that contains some variation of **DLC**.



An EZCAD-based Galvo device as it appears before the driver has been installed



Success! An EZCAD-based Galvo device as it appears after the driver has been installed

Installing the Driver (BSL-Based Lasers)

To connect a BSL Galvo laser to LightBurn, you may need to install the same driver used by the stock software that came with your laser, in one of two ways.

Not sure if your laser has an EZCAD2, EZCAD3, or BSL board?

If your laser uses EZCAD2 as its stock software, it has an EZCAD2 control board.

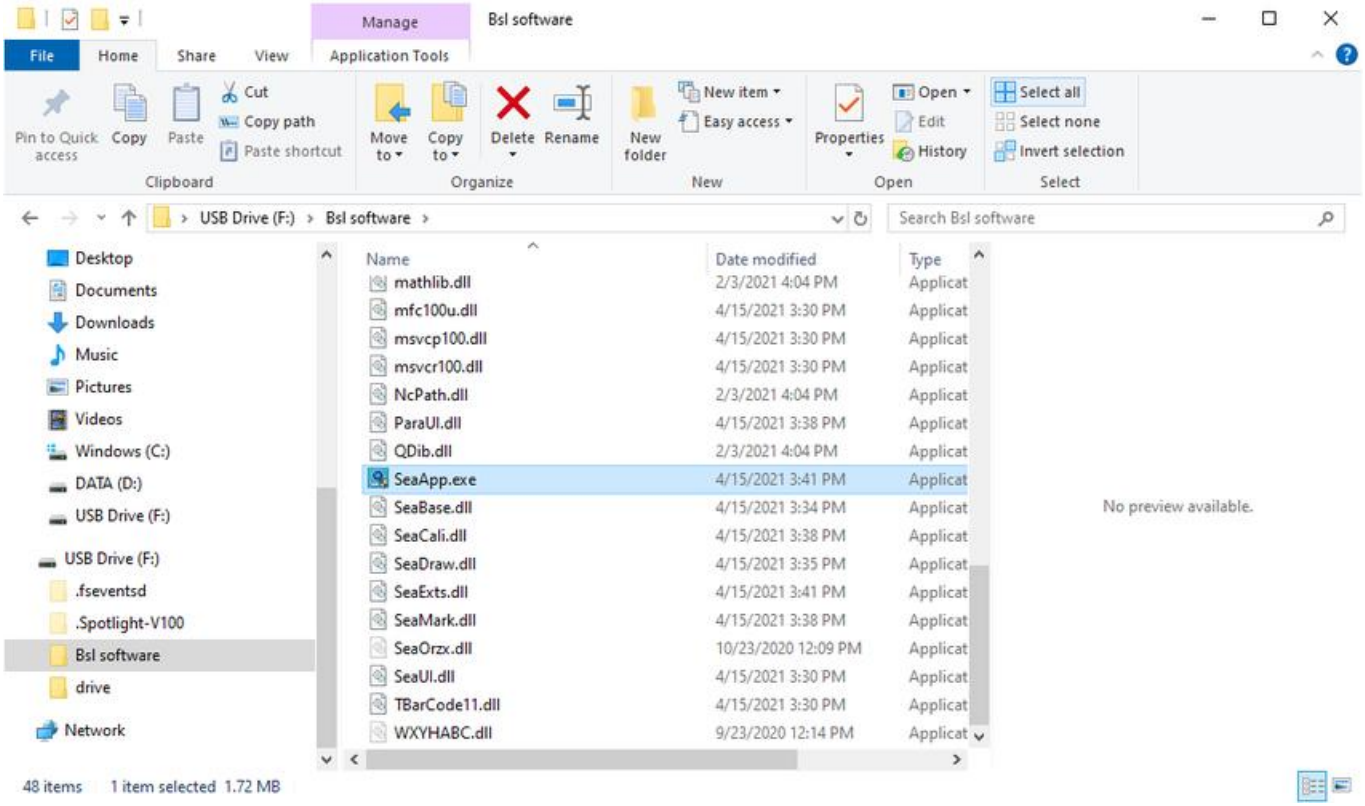
If your laser uses EZCAD3 as its stock software, it has an EZCAD3 control board.

If your laser uses SeaCAD, SimpleCAD, or BSLCAD as its stock software, it most likely has a BSL control board. There are many other variants — consult your laser's manufacturer for further assistance identifying your type of laser.

If your laser uses different software, you may need to investigate further. Some lasers use a rebranded version of one of the programs listed above, such as the GWeike G2 Pro, which uses GLaser and is a BSL machine. You can see our compatibility matrix for more information, or contact support for help.

Install Stock BSL Software

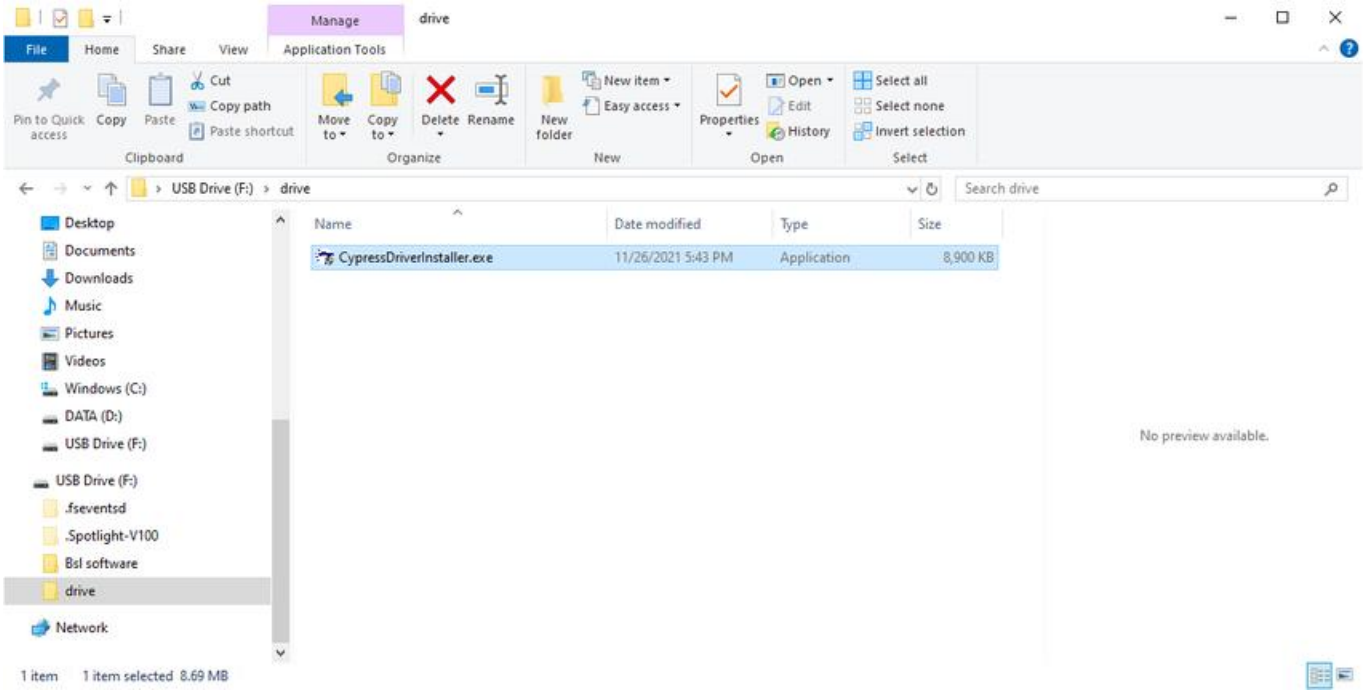
Search the USB drive that came with your laser for the executable file that installs your laser's stock software. Double-click the file to begin the installation process.



The SeaAPP.exe file will install SeaCAD, along with the necessary driver. The specific name of this file and stock software may vary for your laser.

Install the Cypress Driver

Search the USB drive that came with your laser for the executable file that installs the Cypress driver. Double-click the file to begin the installation process.



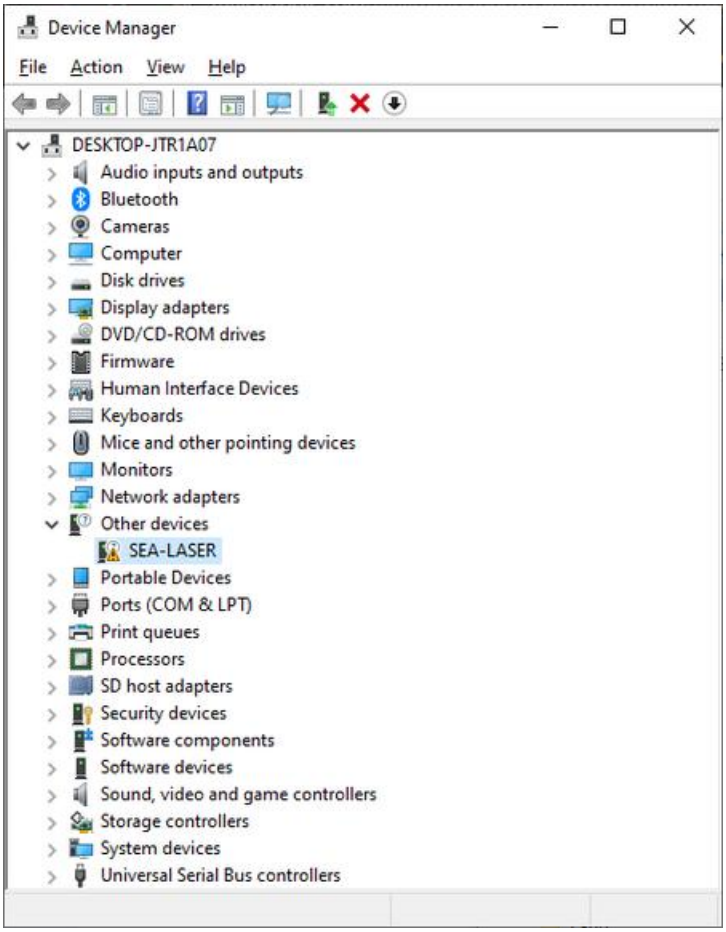
The CypressDriverInstaller.exe file will install the necessary driver for this laser. The specific name of this file may vary for your laser.

Verify Driver Installation

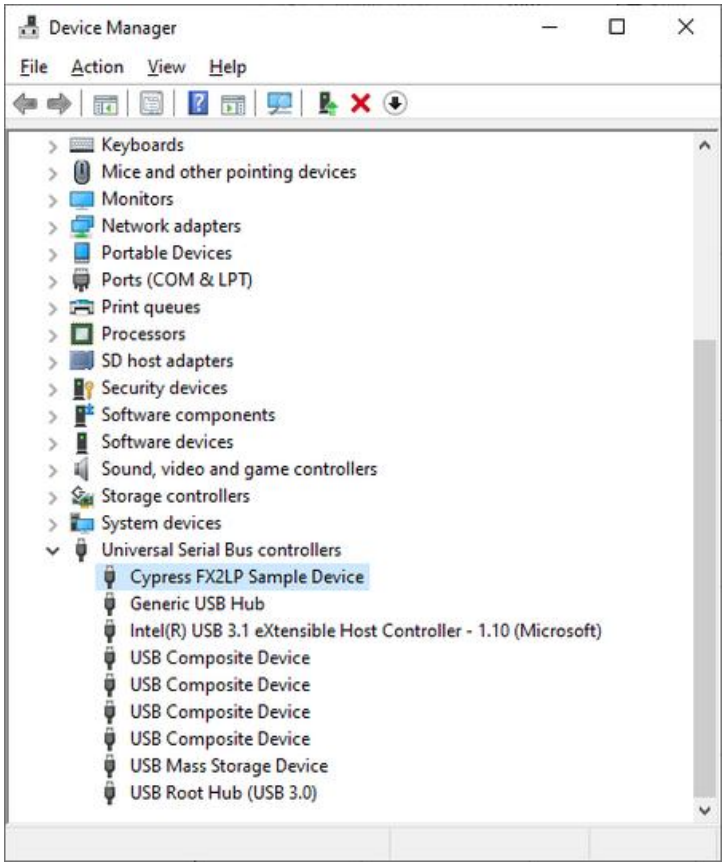
> **Opening Device Manager**
>
> To open the Device Manager, type "device manager" in the search box on the taskbar, then select Device Manager from the menu.

Your laser will appear differently in the Windows **Device Manager** before and after the driver has been successfully installed.

- Before installing the driver, the laser will appear with a caution/warning symbol next to its name. If you see this, the correct driver is not installed.
- After installing the driver, the laser will appear under **Universal Serial Bus controllers** as **SEA-LASER** (with no caution/warning symbol), **Cypress FX2LP Sample Device**, or something else similar.



A BSL-based Galvo device as it appears before the driver has been installed



Success! A BSL-based Galvo device as it appears after the driver has been installed

Related Topics

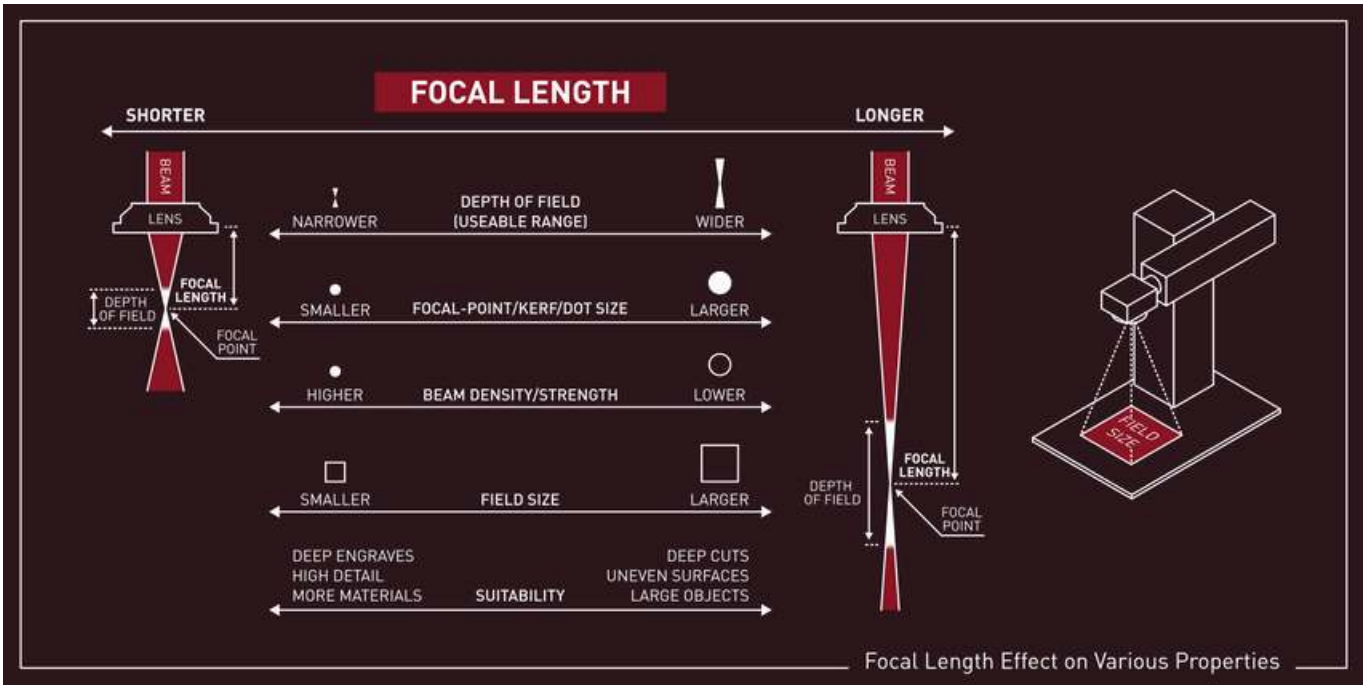
- Add a Galvo Laser
- Galvo Troubleshooting

Focusing Galvo Lasers

> **Danger**
>
> Any use of a laser as outlined in this section requires effective and useful laser safety glasses, calibrated to your laser's output wavelength range.
>
> Laser safety glasses should come with the wavelengths they protect against permanently marked on the glasses themselves.

To use your Galvo laser effectively, you'll need to have it in focus, well tuned, and well calibrated. This enables effective use of features like Cylinder Correction, 3D Sliced Engravings, and more.

Finding Your Focal Point



Each lens should be labeled with a focal length, such as F330 for 330 mm lenses or F160 for 160 mm lenses. This is a good starting point for focusing, but manufacturing differences result in discrepancies between lenses, so you should still fine-tune the focus by looking for crisp outlines in framing or listening for audio cues.

To use "crisp outlines" to find the focal length, you'll need to Frame a design and raise/lower the galvo until the edges are as crisp as possible. This requires your galvo to have an in-line red dot.

With audio cues, you'll instead listen for louder burning or sizzling noises while the job is running. The correct focal length will generally make the loudest sounds.

While the depth of field is a range that produces engraved results, finding the optimal focal point within that field is important to ensure the best results, and is best found through trial and error.

Step-By-Step Guide for Focusing Flat Objects

1. Raise your galvo head as high as it can go.
2. Start a Repeat engraving.
3. Lower the head until it starts making noise.
4. Continue farther down slowly until the engraving noise is as loud as it gets.
5. If the noise tapers off, raise it back up and fine tune based on that.
6. Your flat surface is now focused!
7. If your laser has two diode-red-dots that are manually moved, re-adjust those at this point. See your laser's user manual for help.

Measuring Focal Length

> **Tip**
>
> Make sure you keep track of what part of the laser your focal length is measured to! You might find it convenient to measure to a point other than the bottom of the lens, and that's fine as long as you keep measuring to the same height.

Accurately measuring the focal length above is a useful process to speed up swaps of different and diverse jobs, given the lens. Use an acrylic ruler (preferably one with the edge of the ruler starting from 0"/0 mm), to measure up to the point the ruler meets the outside lip of your lens.

If your acrylic ruler has a lip, you can measure the offset with calipers to produce the formula needed to measure the focal length accurately.

To make it quicker and easier to focus the laser when working with objects of different sizes, it's a good idea to make a tool that you can place between the lip of your lens and your object, so you can easily raise/lower your galvo head to it. This can be as simple as a dowel of the right length marked to indicate which lens it goes with (if you have multiple lenses). Another popular option is to laser cut an acrylic focusing tool for the size you need.

Video Guide For Focus Sticks

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=wOEdciQtPVI

Focusing For Cylinder Correction

Cylinder correction within LightBurn maximizes the depth of field usable for engraving by intentionally defocusing your laser. See here for more information.

Related Topics

- Changing a Galvo Laser Lens
- Red Dot Offset

- Add a Galvo

LightBurn Configuration

Customizing the LightBurn Window

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=t0p4U2eSFQs

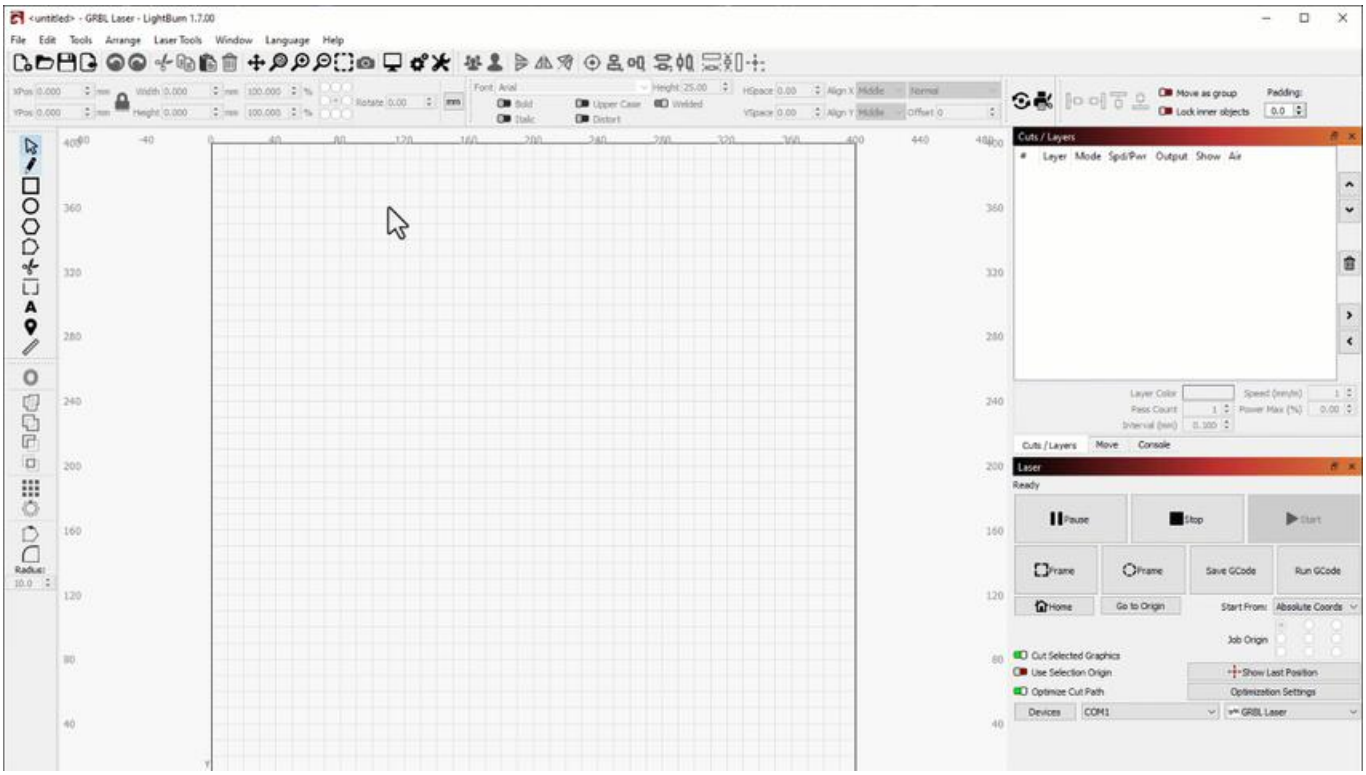
The LightBurn layout is highly customizable — you can choose which windows and toolbars to expose and hide, or undock, move, and re-dock them to a location other than their default position.

Enabling and Disabling UI Elements

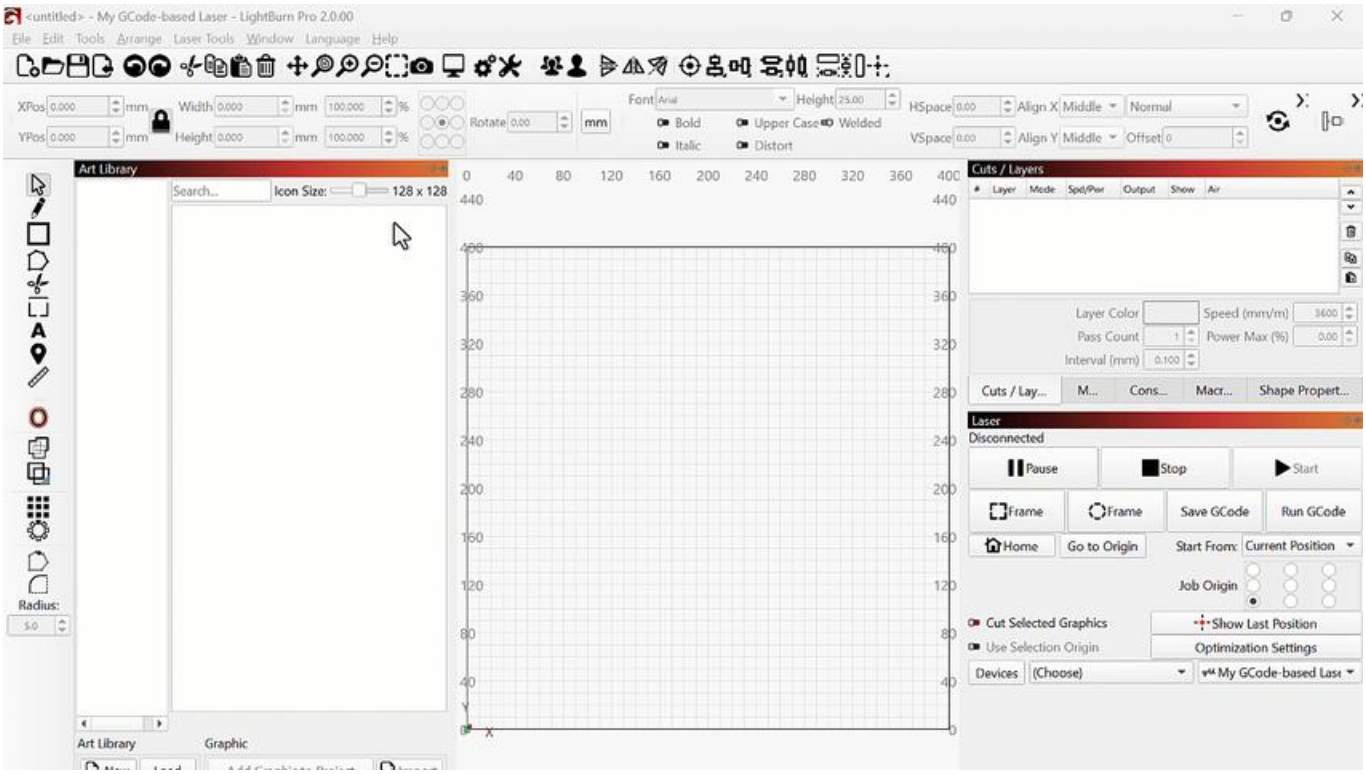
Use the Window Menu to enable or disable any toolbar or window in LightBurn. Several windows are not shown by default, and you'll need to enable them in order to access the tools and options they contain.

Enabled windows and toolbars have checkmarks next to their names, and disabled windows do not.

Click any unchecked window or toolbar in the list to enable it.

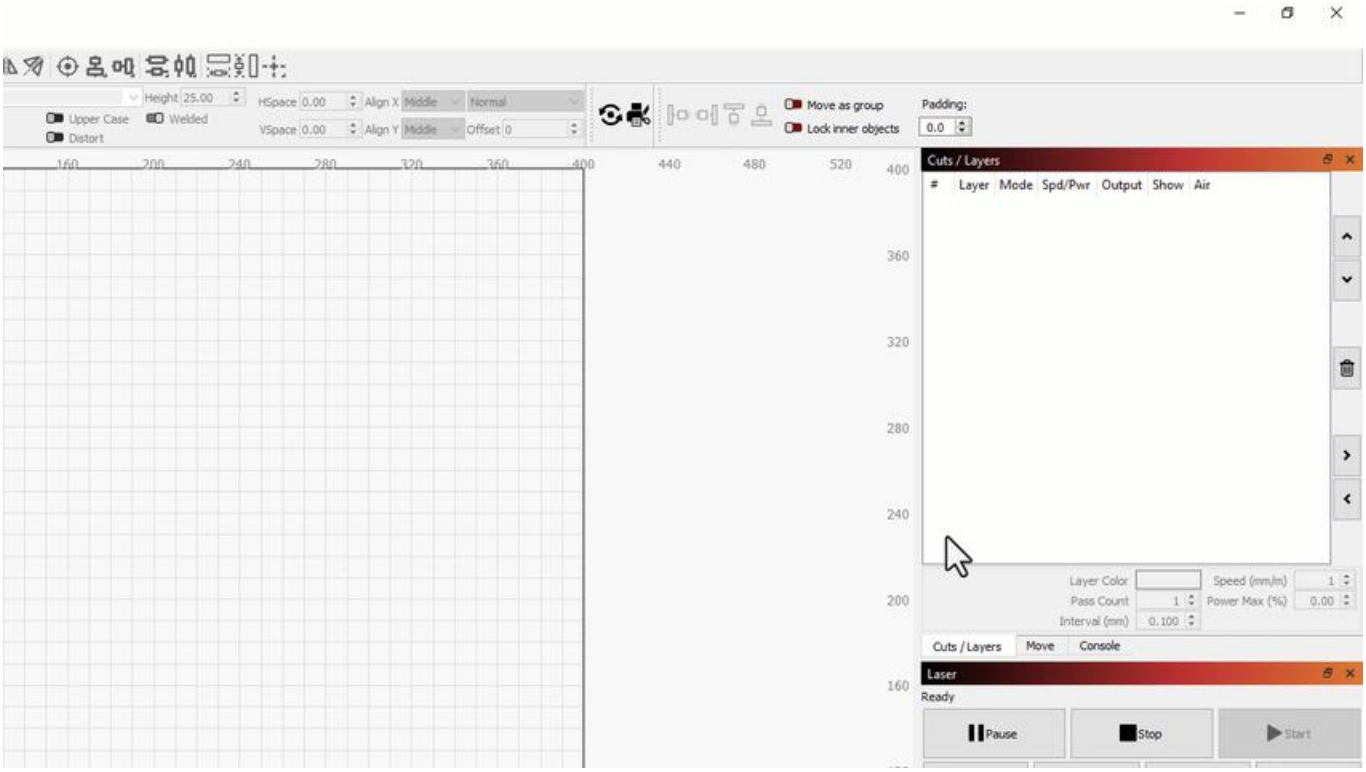


Click any checked window or toolbar to disable it. You can also click the X in the upper right corner of any window to disable it.



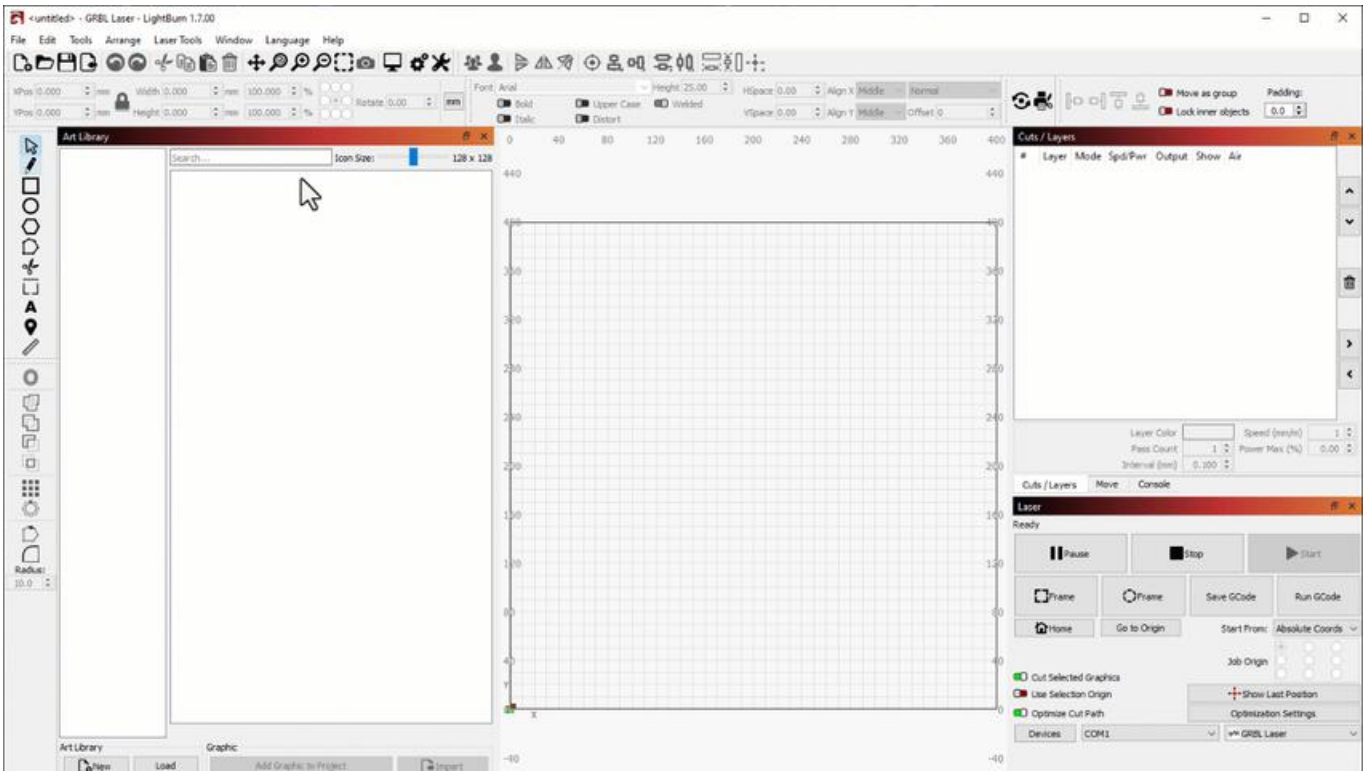
Right-Click Context Menu

Right-click over any toolbar, or over the top bar of any window, to open a window and toolbar selection context menu. Just as in the Window Menu, enabled windows and toolbars have checkmarks next to their names, and disabled windows do not. Click any unchecked window or toolbar to enable it, and any checked window or toolbar to disable it.



Toggle Side Panels

Click **Toggle Side Panels** or press F12 to disable all windows that are docked on the left or right side of the Workspace. Click or press F12 again to re-enable them.



Docking and Undocking

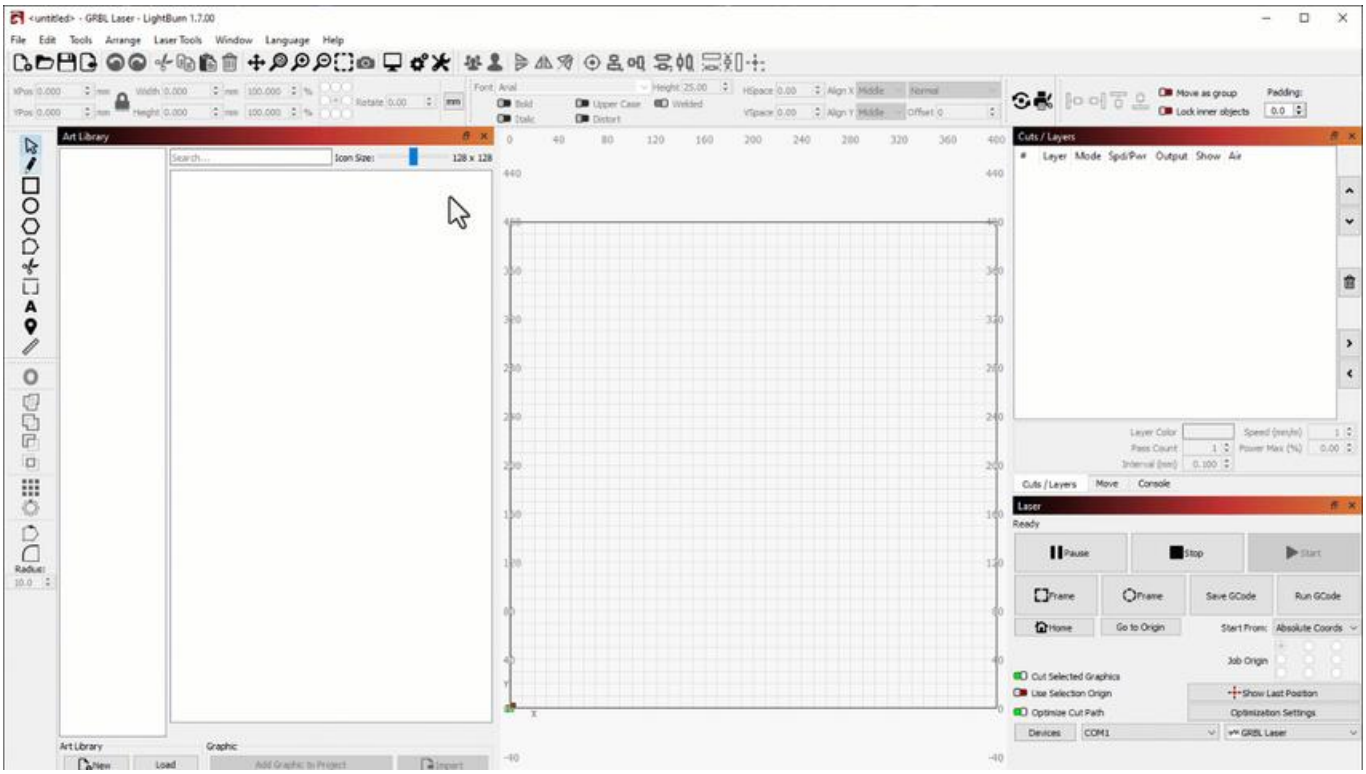
Windows

Click and drag any window from its top bar to undock it — you can leave the window floating anywhere you'd like, or re-dock it to another position.

The dockable area for windows is to the left, right, top, or bottom of the Workspace. You can stack multiple windows on top of one another vertically, or dock them so that they are tabbed together in a shared space.

By default, several windows are stacked and tabbed together on the right side, and no windows are visible on the left.

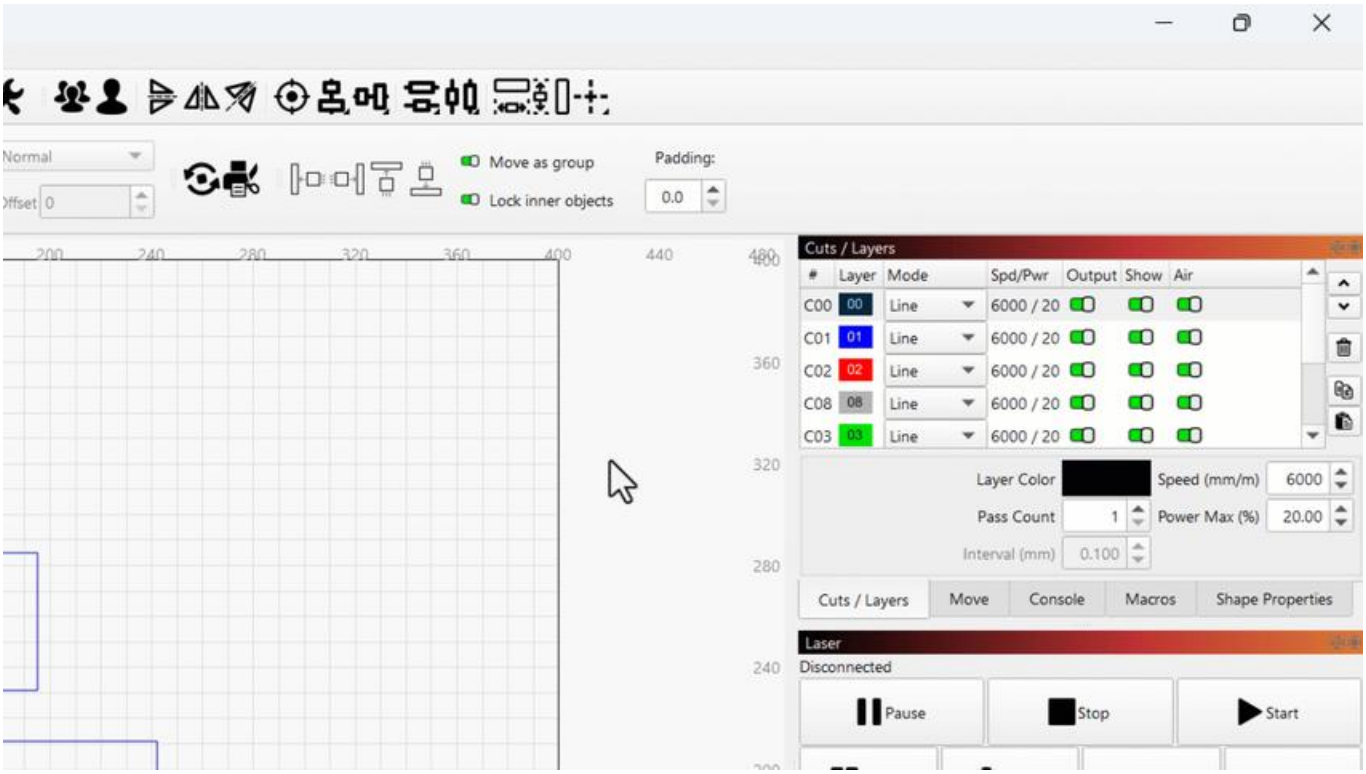
When you've dragged a window over an area it's possible to dock it to, the area will turn blue or light gray. Release to dock the window.



Note

The current positions of existing toolbars and windows, size of your computer's display, and the size of the LightBurn window all determine where and how many windows can be stacked on top of one another — if you're unable to dock an additional window above another, it is most likely not a valid location. If your LightBurn window is not expanded to the full size of your screen, try maximizing it.

Click and drag the side or bottom of a window or docking area to adjust its width. Note that each window has a minimum size requirement, and every window in that docking area must fit within the new width (including tabbed windows that aren't currently visible).

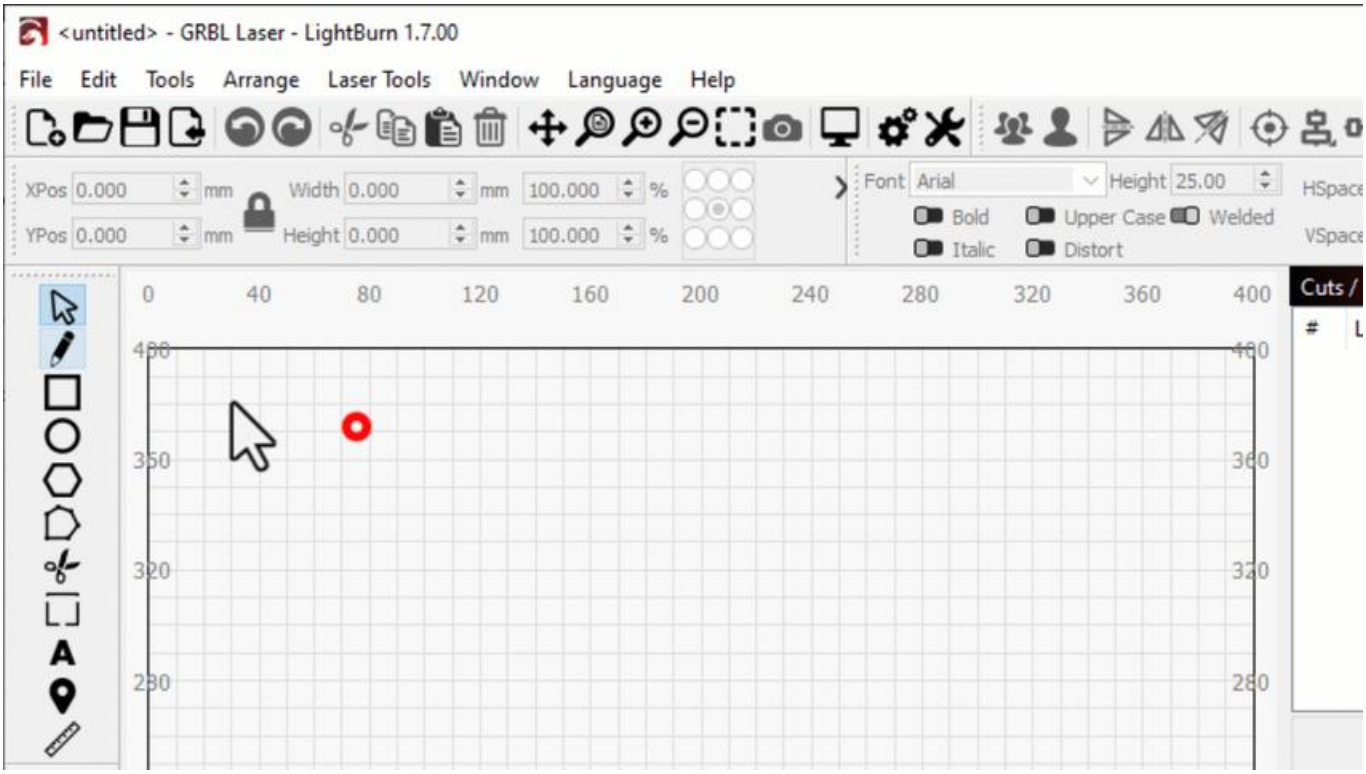


Toolbars

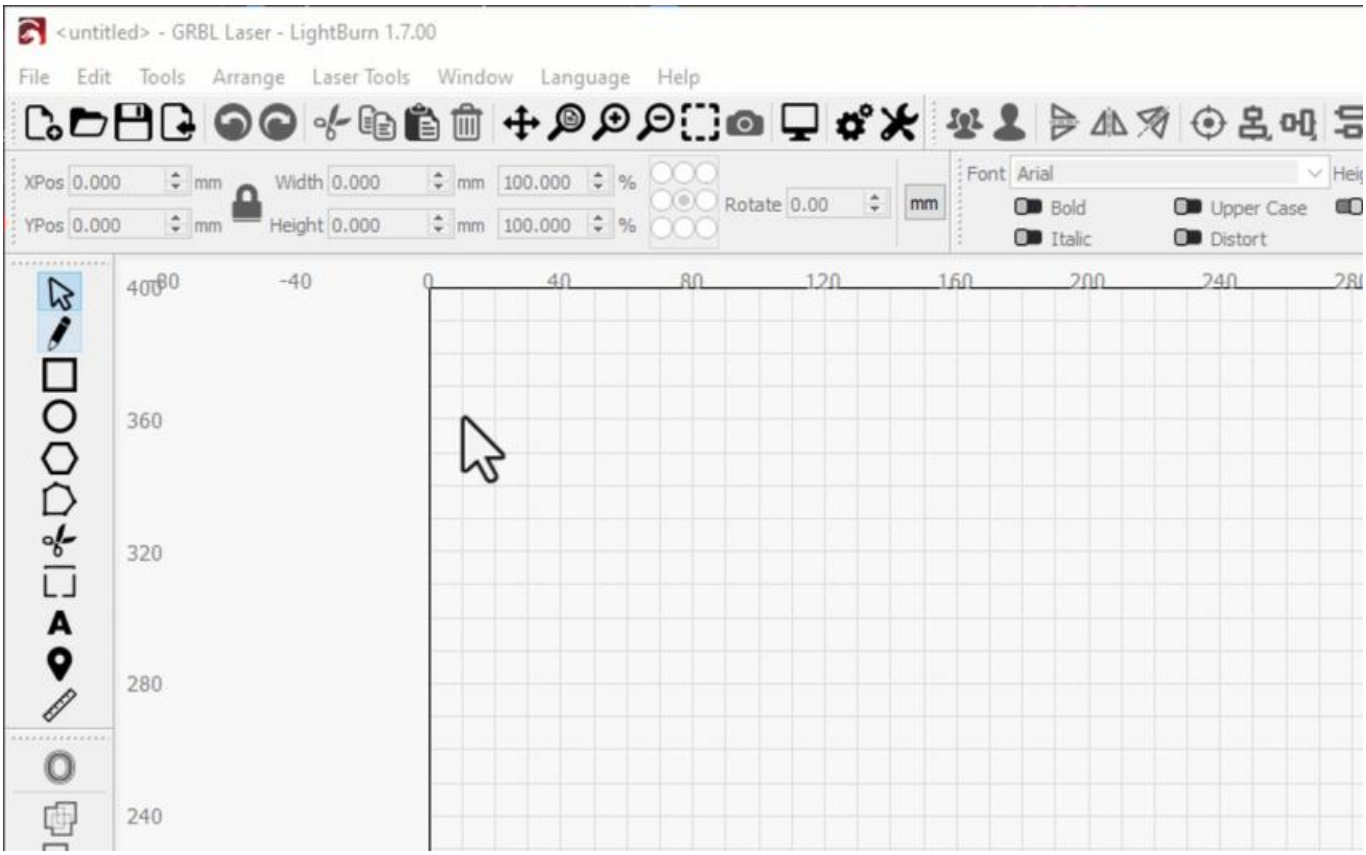
Click and drag any toolbar from the dotted line to undock it — you can leave the toolbar floating anywhere you'd like, or re-dock it to another position.

The dockable area for toolbars is above, below, to the left, and to the right of the Workspace. Toolbars can be stacked on top of one another or docked next to each other.

If the space in which several toolbars are docked is too small to show all of the buttons the individual toolbars contain, you'll see an arrow symbol in a collapsed version of the toolbar. Drag the dotted line to adjust the size of a toolbar, or click the arrow to temporarily expand a toolbar and show all tools it contains.



Space permitting, you can stack toolbars in spaces that are not occupied by any toolbars in the default layout.



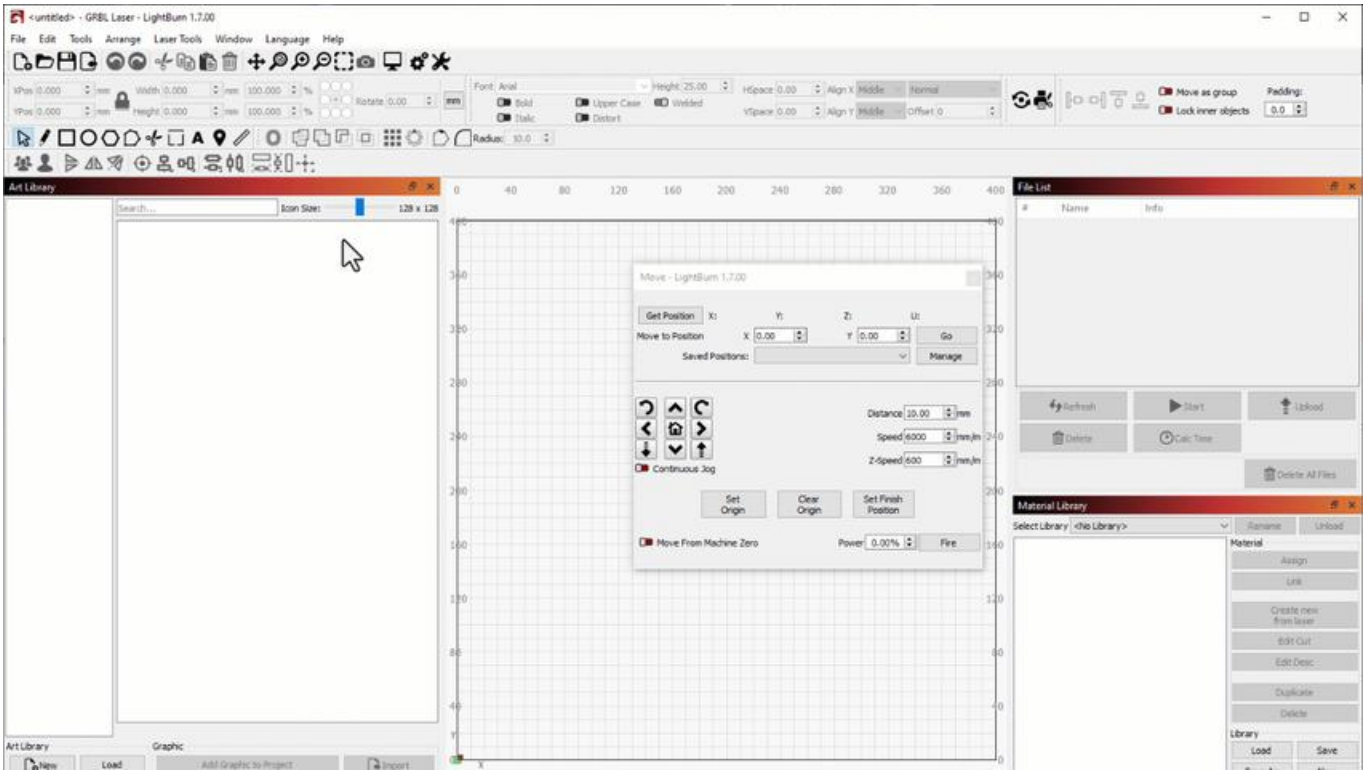
Reset to Default Layout

Reset to Default Layout restores the layout of the full LightBurn window to the same format it was in when you first installed and opened LightBurn. If you've made lots of changes to your layout that you want to reverse all at once, or you've lost sight of the Laser Window, Cuts / Layers Window, Color Palette, or any other default windows or toolbars, use this option to get them back.

Resetting the layout doesn't reset anything else — your preferences, devices, and libraries are all preserved.

Go to Window → **Reset to Default Layout** to re-open and re-dock all default windows in their original order, while closing any that are not visible by default.

Holding `Shift` while clicking to open LightBurn will also reset the layout.



Related Topics

- Workspace
- Main Window
- Reset to Default Settings

Updating LightBurn

Upgrading to the latest version of LightBurn gets you new features, improved machine support, and bug fixes.

This guide will help you upgrade your current LightBurn installation to a new version. It's meant primarily for updating LightBurn on the same computer, but we also have information on migrating LightBurn to a new computer and Downgrading to an Older Version.

Install LightBurn on a New Computer Install an Older Version

Upgrading to a New Version

Upgrading your LightBurn installation on the same computer shouldn't affect your LightBurn configuration, but it's still always a good idea to back up your LightBurn configuration with a User Bundle and wait until you finish any critical, time-sensitive projects before upgrading.

Your project files aren't managed by LightBurn and aren't touched by the upgrade process, but consider this your reminder to back them up as well.

> **Installing Release Candidates**
>
> If you want to try a Release Candidate alongside the current live version of LightBurn, don't worry! The Release Candidate will install in a separate directory, so you can run it alongside the current version.

Update LightBurn

Latest Version Release Candidate

1. Create a User Bundle and save it as a backup. You shouldn't need it, but if something goes wrong during the upgrade process, you'll be able to restore your settings in a clean installation.
    1. Go to **File** → **Bundles** → **Export Bundle**.
    2. Make sure all the components of your LightBurn installation that you want to backup are selected and click **Export**.
    3. Save the `.lbzip` file somewhere you can find it again easily.
2. The upgrade process doesn't affect your project files, but if you don't regularly back up your project files, this is a great time to back them up as well. Save copies of your project somewhere you can find again easily.
3. Download the new installer from the LightBurn website. Links are provided above.
4. Close LightBurn, if it's open.
5. Install LightBurn.

After installing a new version, LightBurn should still have all your pre-existing settings. If you're installing a Release Candidate alongside your existing LightBurn installation, LightBurn will offer to import settings from your existing installation, if there aren't pre-existing settings.

If, for some reason, there is a problem with your LightBurn configuration on the new installation, follow the directions for Importing a User Bundle to restore settings from your previous LightBurn installation. This also works to bring settings from your main LightBurn installation into a Release Candidate installation.

Downgrading to Older Versions

Although we always encourage users to run the current release of LightBurn, there are times when you may need to either downgrade to an older release, or share files with someone using an older LightBurn version. With that in mind, we have some tips to help with both scenarios.

Backwards Compatibility

Project files, User Bundles, `.lbzip` device profiles, and other files from older versions of LightBurn are fully compatible with versions of LightBurn newer than the ones they were made in. However, we can't guarantee backwards compatibility — files created in current versions of LightBurn may not work in older ones.

Why Use an Older Version?

Users might want to install an older version of LightBurn for one of the following reasons:

- If you encounter issues with a new release of LightBurn, it's sometimes helpful to temporarily downgrade to a previous release so you can continue working. Please report issues to support so we can help you resolve any problems, and escalate bugs to our developers.
- If the upgrade period on your license has expired, your license remains valid for any LightBurn version released before the end of your upgrade period. This means you can keep using LightBurn forever, but you'll need to download an older version of LightBurn from release.lightburnsoftware.com.

  > **Tip**
  >
  > To use the latest release, you can add another year of updates to your license key, no need to purchase a whole new license.

- Downgrading a previous release can ensure compatibility if you need to share files with a user who is limited to using an outdated version of LightBurn.
- If you're running an operating system that is not supported by the current release of LightBurn, you'll need to download an older version that is.

  > **Info**
  >
  > - Current versions of LightBurn run on 64-bit Windows 10 (build 1809) or later and macOS 11 or later.
  > - LightBurn version 1.7 is the last version compatible with Linux, versions of Windows prior to 10, and versions of macOS prior to 11.

Downgrading to an Older Version

> **Looking for release candidates?**

> To download release candidates, please go to our release candidate download page.

Whether you're installing the current version of LightBurn, a release candidate, or an older version, the best place to download is always directly from LightBurn. The current release is always available at release.lightburnsoftware.com - latest. Older releases, going back to LightBurn 0.6, are available at release.lightburnsoftware.com.

Click on the version you want to download. In the screenshot below, you can see the download page for LightBurn 1.5.06.



Operating System

Icons on the left side of the page show at a glance what operating system each download is for:

- for Windows
- for macOS
- for Linux (LightBurn 1.7 and earlier)

Some versions may have more than one download available for the same operating system. See below for more information.

Installer Version

If there's more than one download available for your operating system, the file name describes the differences.

Windows

32 bit installers will have `32Bit` in the filename, and the main Windows installer will not. Most modern computers do not need the 32 bit version, and should run the main Windows installer. If you're unsure, try the main installer first. 32 bit support was discontinued with the release of LightBurn 1.6.

macOS

Some versions will have a distinction based on your computer's macOS version.

Linux

Check the file extension for the installer type. Linux support was discontinued with the release of Lightburn 2.0.

Move LightBurn to a New Computer

If you've been using LightBurn and need to set up a new computer, this guide will help you move your LightBurn settings and devices from an existing installation to a new computer.

Before you begin, please make sure that you have:

- An existing, already configured LightBurn installation running at least LightBurn 1.6.
- A new LightBurn installation that you wish to migrate to, also running at least LightBurn 1.6.
- Some way of moving files between computers. A USB flash drive is typically a convenient way to do this.

What if I'm using an older version of LightBurn?

This guide makes use of LightBurn's User Bundles feature, introduced in LightBurn 1.6. If you are using an older version of LightBurn, please either upgrade before migrating or see our legacy documentation for assistance.

License Management

Each LightBurn license key comes with 3 seats, allowing you to activate LightBurn on up to 3 computers simultaneously. If you will no longer be using LightBurn on a particular computer, you can deactivate LightBurn to free up the seat for use on another computer. See License Management for more information.

You can add additional seats without purchasing a separate license here.

Migration Overview

Most of the migration process is handled by LightBurn's User Bundles feature, which can export and import the following:

- LightBurn Settings
- Custom Hotkeys
- Material Test Presets
- Image Presets
- Devices
- Material Libraries
- Art Libraries

Please be aware that **User Bundles** do not contain:

- Your project files, which you'll need to move yourself.
- Your SHX fonts folder, which you'll need to move yourself.
- Ordinary fonts, which are handled by your computer, not by LightBurn.

Migrate Settings and Devices With User Bundles

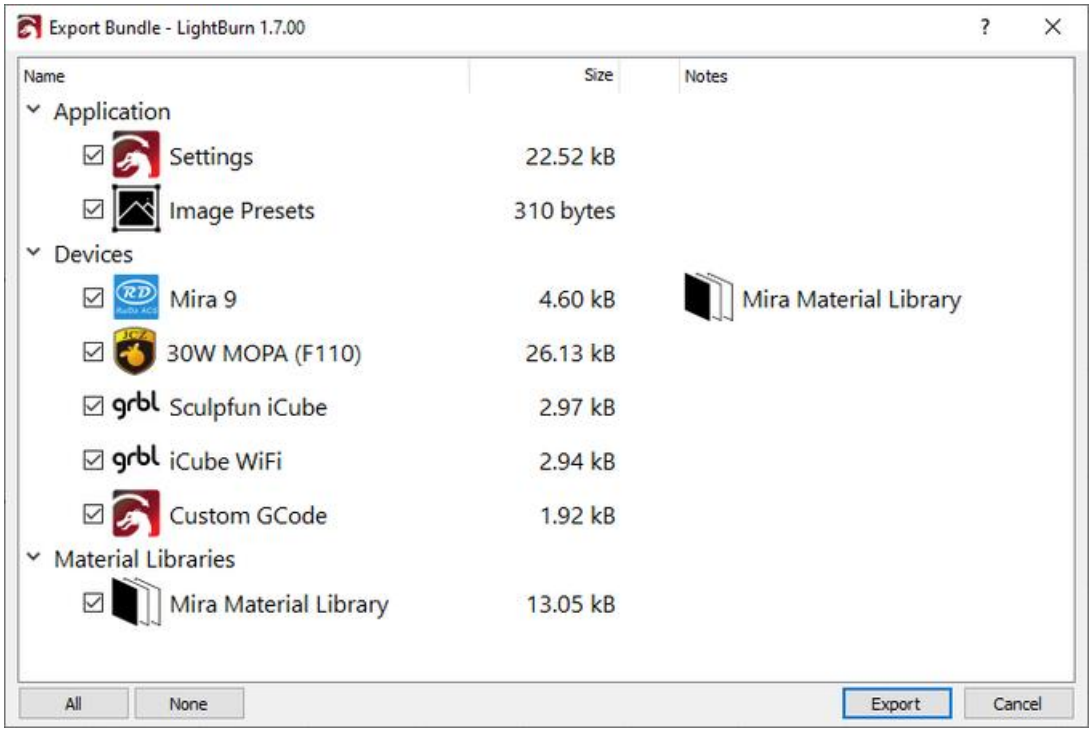
> **What about `.lbdev` files?**
>
> Guides and videos based on older versions of LightBurn may still reference `.lbdev` files, which were previously created by exporting a device profile. As of LightBurn 2.0, this button now creates a User Bundle for the device. User Bundles can be imported in the Devices window just like the older `.lbdev` files.

The migration process starts with exporting a **User Bundle** from the existing installation and importing it into the new installation.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=S0QrokNepCQ

Start by exporting user bundles containing all the settings you want to bring into the new installation:

1. In an existing and already configured LightBurn installation, go to **File → Bundles → Export Bundle**.

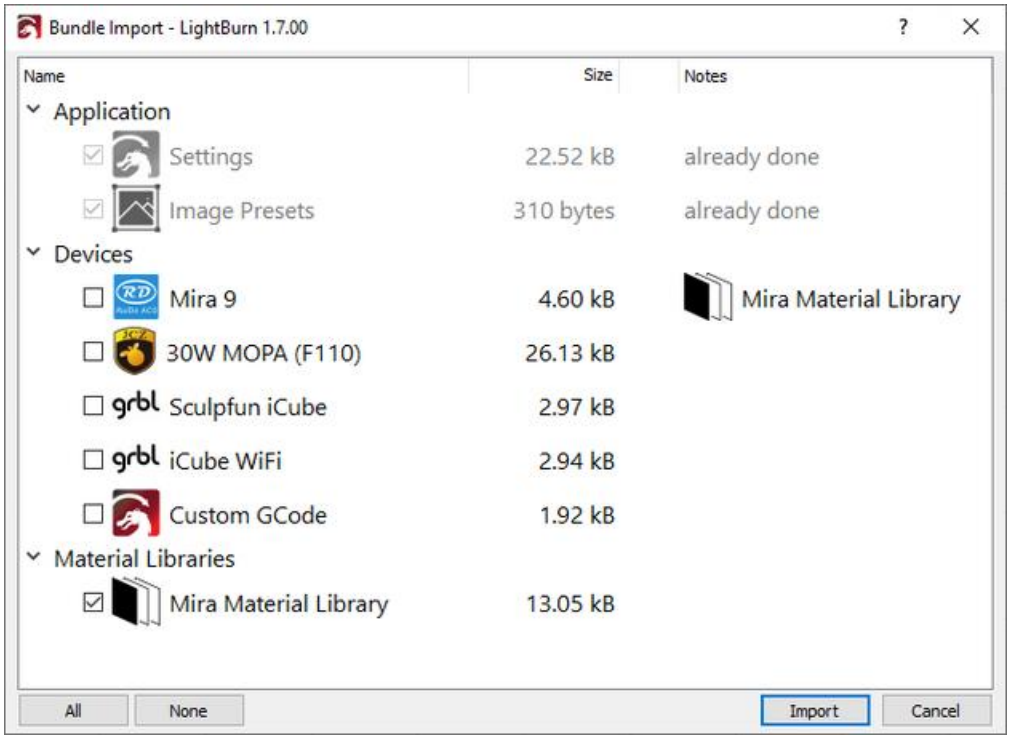


2. Make sure that all components of the current installation that you want to include are selected and click **Export**. Save the `.lbzip` file somewhere you can find it again easily.

Next, import the bundle into the new installation:

1. Move the `.lbzip` file from **Export Bundle** to the new computer.

2. Open LightBurn on the new computer, and drag the `.lbzip` file into LightBurn to load it.



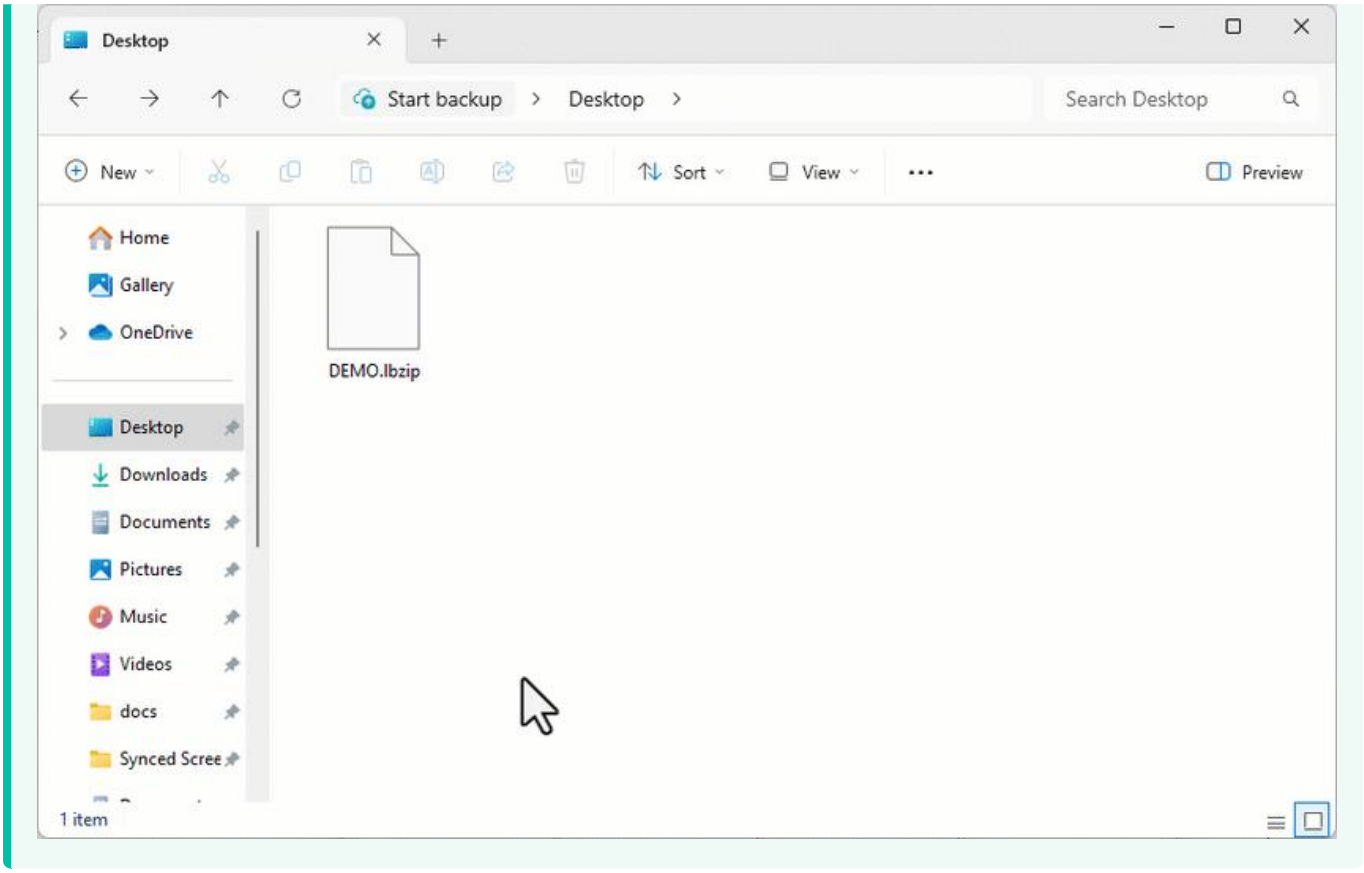
3. Make sure that all components you want to import are selected, and uncheck any components you want to ignore. Components that are already detected on the computer will be grayed out and unavailable, with two exceptions:
   - Devices that conflict with an existing one will create a duplicate to preserve the existing profile.
   - **Material Libraries** and **Art Libraries** with the same name but different contents will create a new copy of the library without overwriting the original.
4. Click **Import** to complete the process.

Creating `.lbdev` files

Warning: `.lbdev` files are not guaranteed to be backwards compatible with older versions of LightBurn.

If you need to create a `.lbdev` file, you can extract one from a User Bundle if needed. To do this:

1. Export a User Bundle as described above.
2. Open the `.lbzip` file in Windows Explorer or in Finder.
3. Change the extension of the file to `.zip` instead of `.lbzip`.
4. Open the zip folder and copy the `.lbdev` file you're looking for to a suitable location.



Move Project Files

Because users are free to store and organize their project files however they want, LightBurn is unable to move project files automatically. Please make sure that you copy any project files you want over to your new computer yourself.

Move SHX Fonts

SHX Fonts are handled separately from other fonts installed on your computer. To move your SHX fonts to the new computer, please do the following:

1. Copy your **SHX Fonts** folder to the new computer.
2. If your SHX fonts aren't recongized, update the SHX Font path on the new computer.

Related Topics

- User Bundles
- SHX Fonts
- Devices
- LightBurn Settings

Opening Multiple LightBurn Instances

Open Additional LightBurn Windows

Each LightBurn window can open a single project and communicate with a single device at a given time. To jump quickly between projects or run jobs on multiple lasers simultaneously, open a new window by going to File → New Window.

All windows will share the same settings and preferences. Be careful if you're changing your preferences in LightBurn – the last window closed will save changes to the preferences file, overwriting any changes you may have made in other open windows.

If you need fully independent LightBurn instances, with separate settings and preferences, we have directions for running multiple independent instances below.

Running Multiple Independent Instances of LightBurn

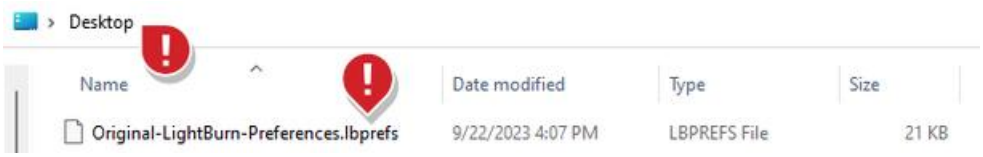
> **Info**
>
> These directions will only work on Windows. The steps required to adjust this behavior and replicate it on macOS will not be the same.

> **Danger**
>
> This guide is for advanced users comfortable making changes to their operating system. Follow these directions carefully and at your own risk.

To run multiple fully independent instances of LightBurn on the same computer, you will need to create separate profiles. This workflow is not suggested, but is documented for the users who need it.

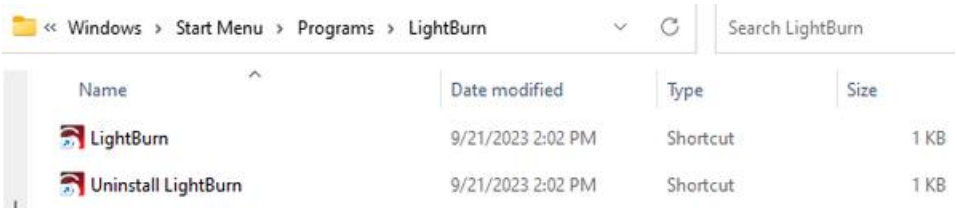
Steps

1. Open LightBurn and export your current preferences by going to File → Preferences → **Export Prefs**.
2. Save the `.lbprefs` file and note its location.

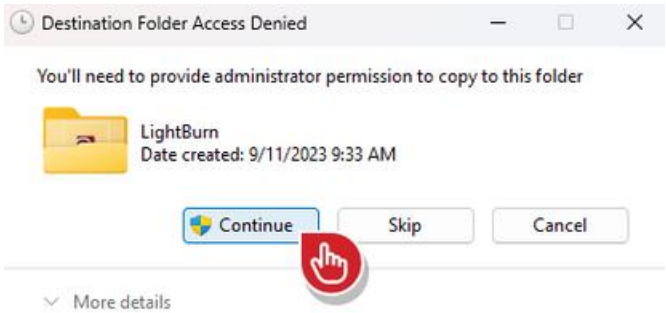


3. Close all LightBurn windows.
4. Open File Explorer and navigate to the Windows Start Menu links storage location. The easiest way is to enter the file path below into the File Explorer address bar.

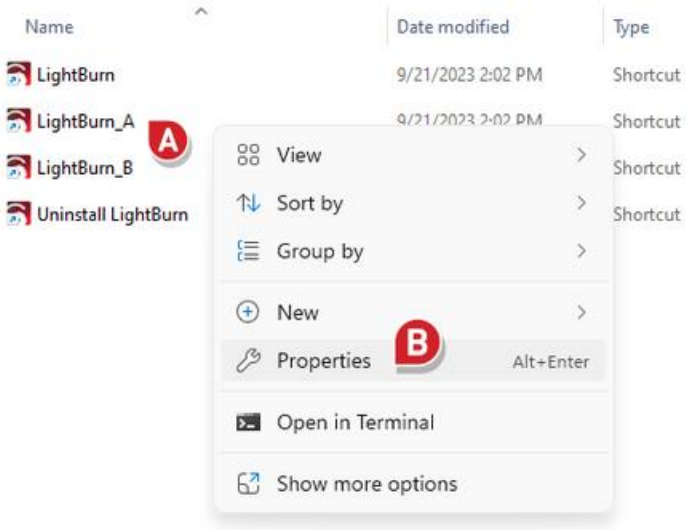
```
C:\ProgramData\Microsoft\Windows\Start Menu\Programs\LightBurn\
```



5. Copy the LightBurn shortcut and paste it twice. Rename each file – this guide uses `LightBurn_A` and `LightBurn_B`. Grant administrator privileges in the User Account Control window by selecting **Ok** and then **Continue** to save each change.



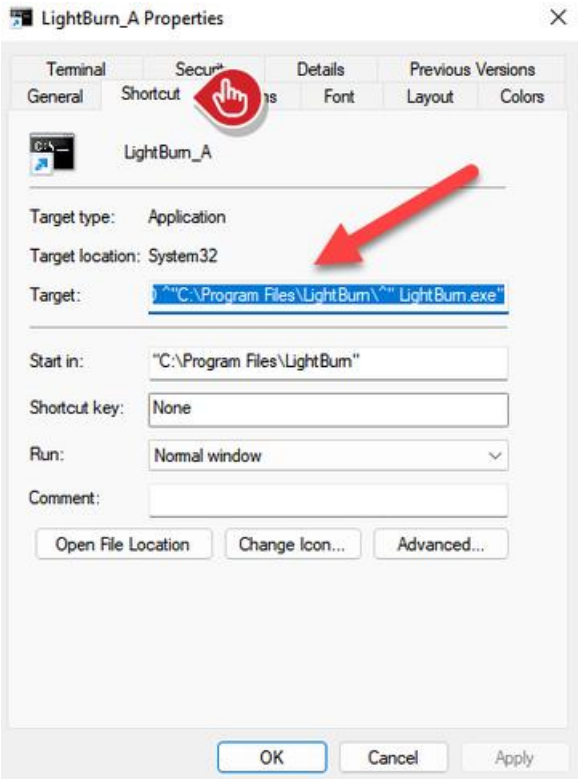
6. Right-click on the first shortcut file and select **Properties**.



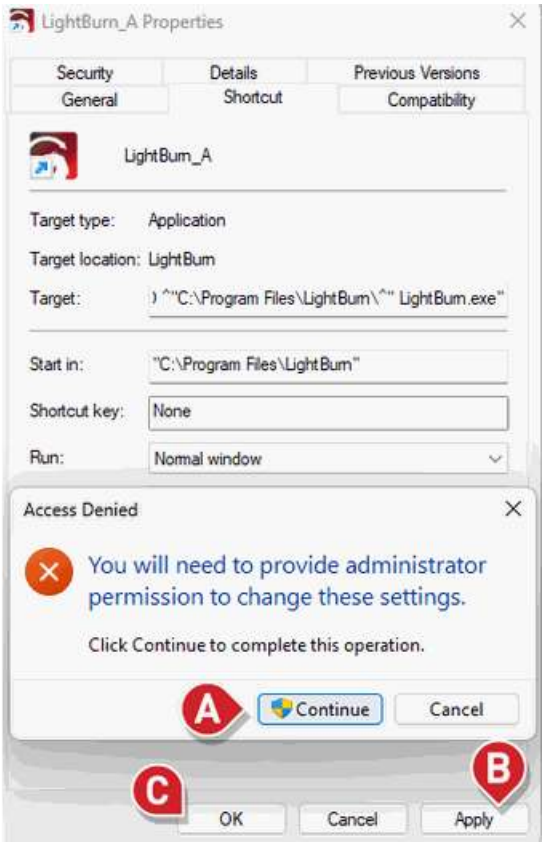
7. Select the **Shortcut** tab and replace the text in the **Target** field to match.

> **Info**
>
> Replace `LightBurn_A` with whatever name you want your first profile to use. Do not include a space between the file name and the `&&` characters as formatting and syntax matters.

```
C:\Windows\System32\cmd.exe /c "SET LB_SETTINGSFOLDER=%USERPROFILE%\LightBurn_A&& START /D
^"C:\Program Files\LightBurn\^" LightBurn.exe"
```



8. Click **Apply** followed by **OK** and grant admin permissions again to save the changes.



9. Repeat **Step 6** through **Step 8** for the second shortcut, this time using this target link:

> **Info**
>
> Replace `LightBurn_B` with whatever name you want your first profile to use. Do not include a space between the file name and the `&&` characters as formatting and syntax matters.

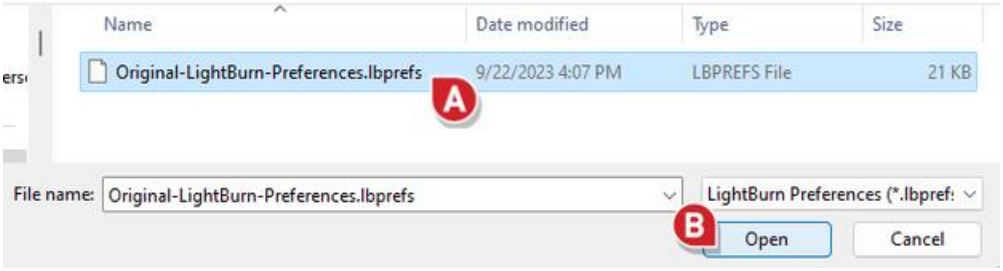
```
C:\Windows\System32\cmd.exe /c "SET LB_SETTINGSFOLDER=%USERPROFILE%\LightBurn_B&& START /D
^"C:\Program Files\LightBurn\^" LightBurn.exe"
```

Two new LightBurn shortcuts are now located in the Windows Start menu. Reentering your license key will not be required.

10. Open the Windows Start Menu and click on the first newly created LightBurn shortcut – `LightBurn_A` in our example – to launch the first independent instance of LightBurn.



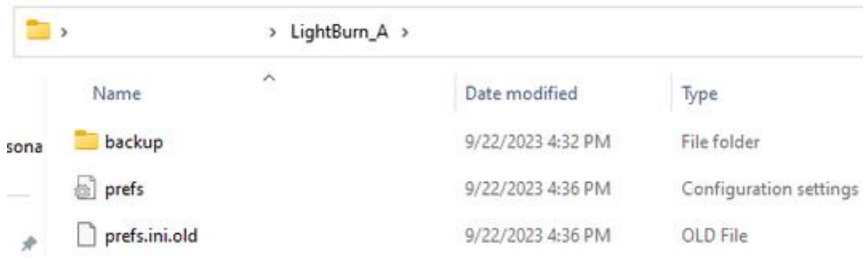
11. Click **OK** to close the first run Help & Notes window.
12. Click on **Import Preferences** in the bottom left corner of the Devices window.
13. Select and load the preference file you saved in **Step 1** to load all devices in the preferences file.



14. Select and remove any devices you do not want associated with this LightBurn instance and click **OK**.
15. Open the second application link – `LightBurn_B` in our example – and repeat **Step 11** through **Step 14** to load the original preference file into the second independent instance of LightBurn.

Summary

Two new preferences directories can now be found in your Windows user profiles folder at `%USERPROFILE%\LightBurn_A` and `%USERPROFILE%\LightBurn_B`. The `%USERPROFILE%` shortcut will resolve to the full file system path on for your user directory.



Starting LightBurn with the new shortcuts will launch multiple LightBurn instances that are completely independent of one another. They will not overwrite the other's preferences file and any changes you make to the device on each will persist, and will include completely independent recent file lists, rotate/shear options, and more. Each LightBurn instance will be completely independent.

> **Info**
>
> While LightBurn allows for simultaneous use of multiple devices without conflicts, cameras and lasers cannot be used by multiple instances of LightBurn concurrently. For example, a DSP controller set up as "Ruida A" will not be able to be accessed in the `A` and `B` configurations simultaneously.

Origin

This guide originated as a post on our forum by one of our developers. For reference, that guide can be found here.

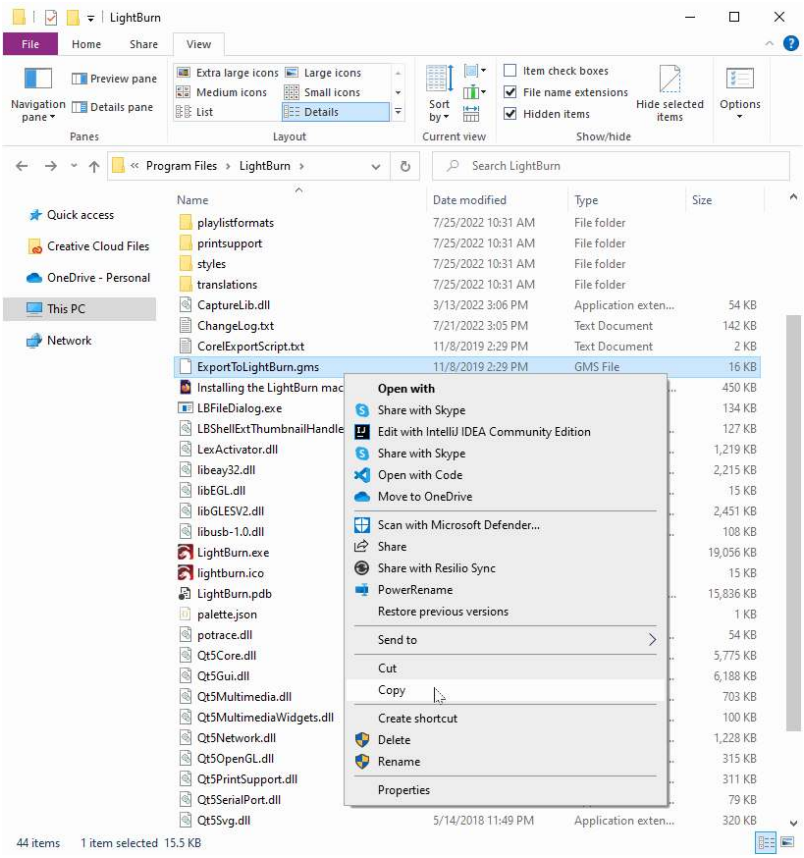
CorelDraw Macro Setup

LightBurn offers CorelDRAW users on Windows an optional macro to simplify and speed up the process of sending work to LightBurn directly through CorelDRAW.

> **Note**
>
> macOS users can **Export** work out of CorelDRAW as an .AI or an .SVG file and Import into LightBurn.

Installation

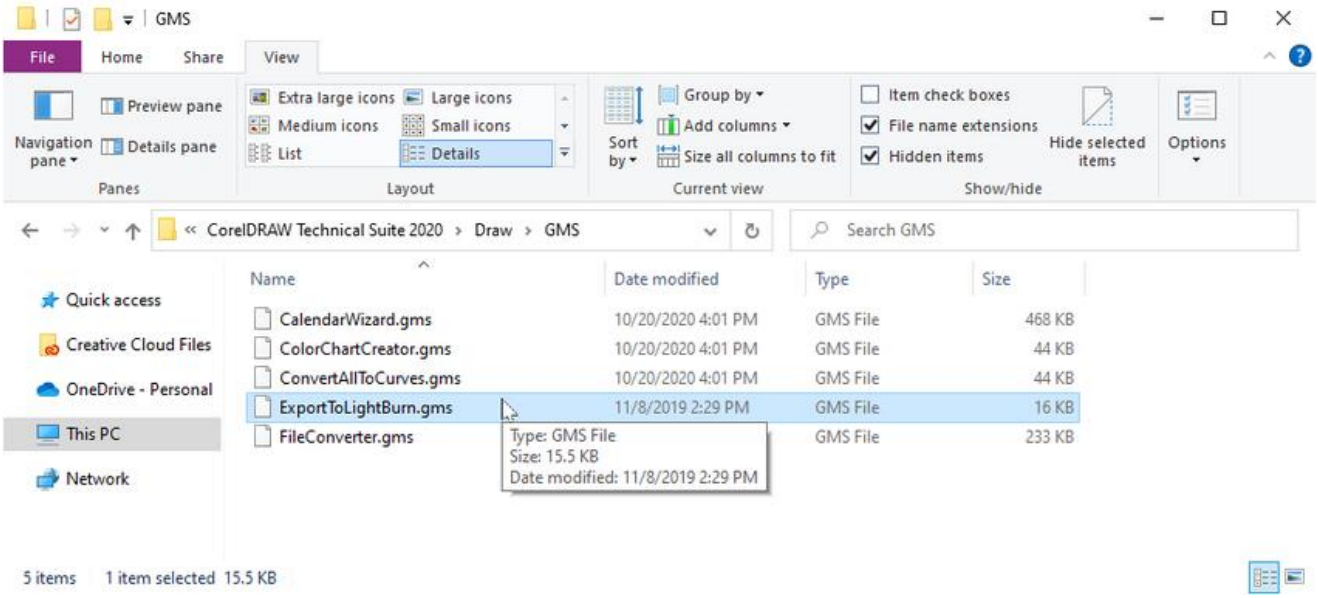
1. Make sure LightBurn and CorelDRAW are both installed on the same computer.
2. Open the LightBurn install folder, typically located at `C:\Program Files\LightBurn`. Find the `ExportToLightBurn.gms` file inside this LightBurn folder.
3. Copy the `ExportToLightBurn.gms` file by right-clicking and choosing **Copy** or by pressing `Ctrl`+`C`.



Copy the ExportToLightBurn.gms file

4. Open the CorelDRAW install folder, typically found at `c:\Program Files\Corel`. Locate the folder called `CorelDRAW Graphics Suite ###` or `CorelDRAW Technical Suite ###`, with `###` being your CorelDRAW program version number.

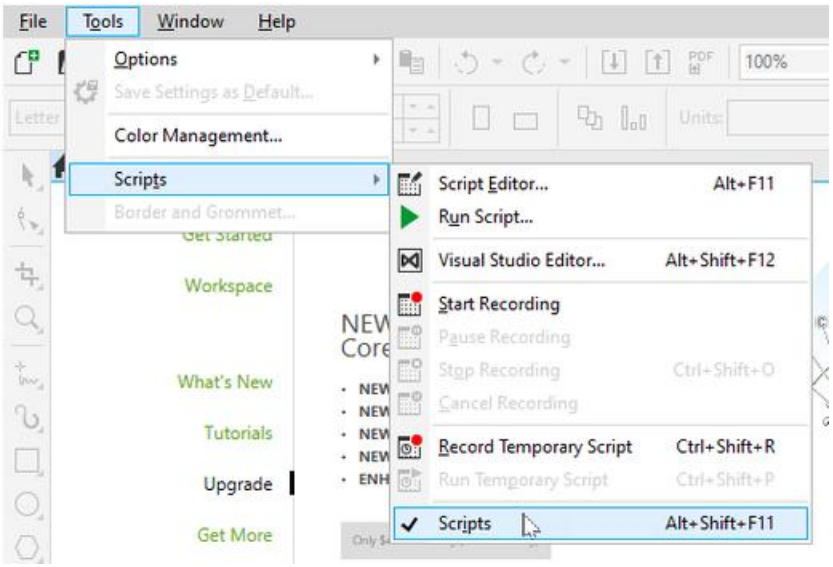
5. Open the `CorelDRAW Technical Suite ###` folder, with `###` being your CorelDRAW program version number, the `Draw` folder, and finally the `GMS` folder. Paste the `ExportToLightBurn.gms` file copied earlier into this `GMS` folder.
6. Grant permission to copy into the `GMS` folder if prompted.



Paste into CorelDRAW's GMS folder

Loading the Macro

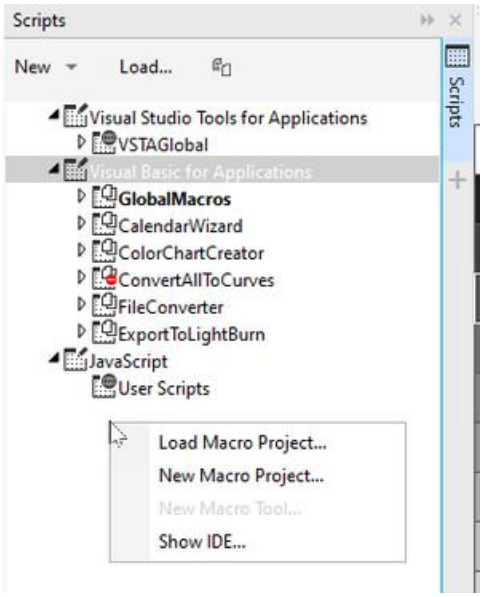
1. Open CorelDRAW.
2. Open the **Script Manager** (**Macro Manager** on older versions) by going to **Tools → Scripts → Scripts** or by using `Alt`+`Shift`+`F11`. Older versions can open the Macro Manager by selecting **Tools → Macros → Macros Manager**.



Open the Macro/Script Manager

3. If the **ExportToLightBurn** macro doesn't show up in the **Visual Basic For Applications** category, load it manually.

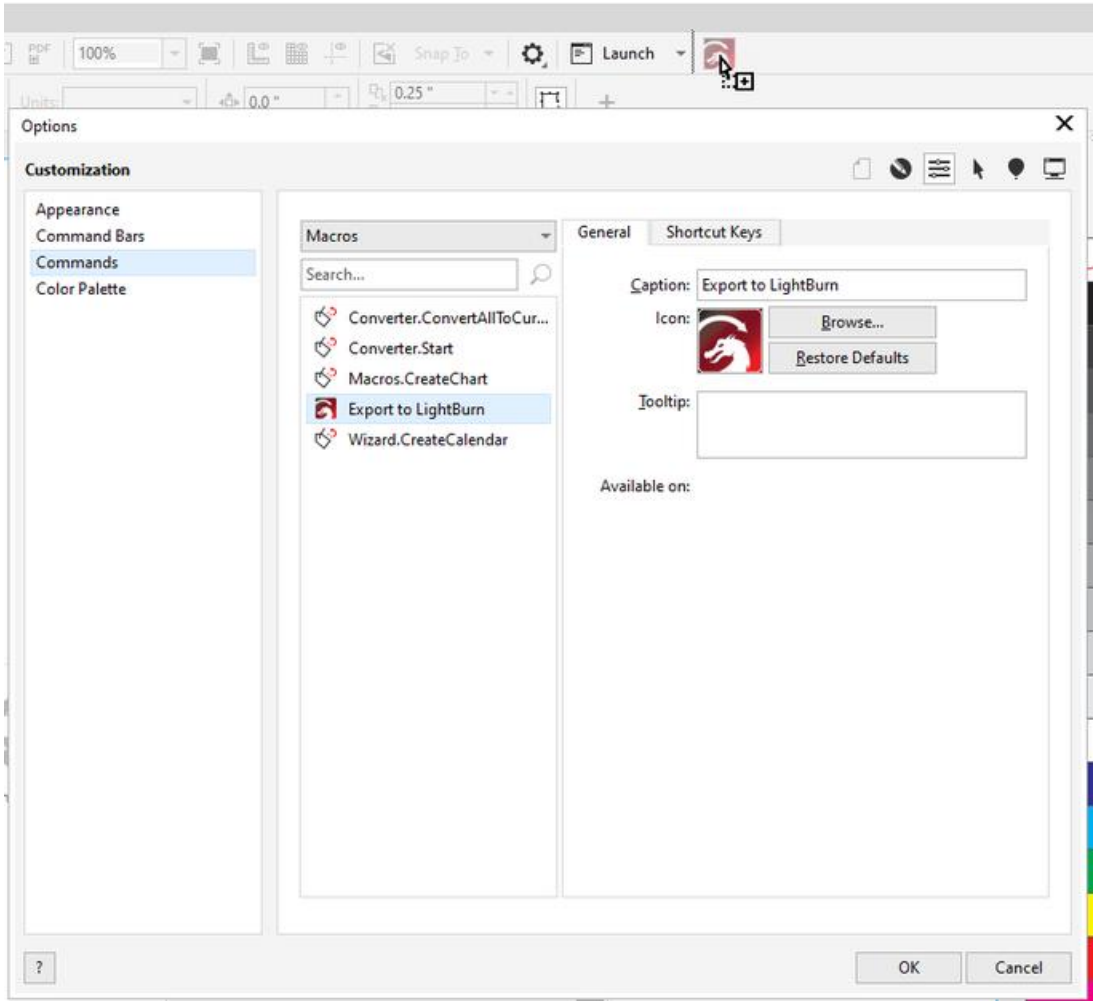
   A. Right-click on the background of the Script Manager window, select **Load Macro Project**, and select `ExportToLightBurn.gms` from CorelDRAW's GMS folder (located earlier in this guide).



Load the macro if needed

Configuration

1. Open CorelDRAW's customization window from **Tools → Options → Customizations**. On older versions select **Tools → Customizations**.
2. Open the **Commands** section on the left. On older versions, select **Workspace → Customization → Commands**.
3. Select **Macros** from the pane that appears on the right.
4. Select the LightBurn entry.
5. Click on the **Browse** button next to the icon. On older versions, select the **Appearance** tab to select an icon.
6. Select the LightBurn icon found in the `C:\Program Files\LightBurn` folder and click **Open**.
7. Grab the macro from the customization window and drag it to a toolbar for an easier workflow in the future.
8. Set the caption to `Export to LightBurn`.
9. Click **OK** to close the Script/Macro Manager window.



Suggested configuration for the macro

Using the Export to LightBurn Macro

Click the Export to LightBurn macro button to launch LightBurn with your current selection from CorelDRAW loaded in your Workspace. If nothing is currently selected, the entire file will load instead.

Related Topics

- File Management
- Workspace

Automation with UDP

LightBurn has a network command interface that allows you to launch LightBurn, load a file, run the laser, and perform several other basic operations. It can be used to run commands on the same computer you're using, or on a different networked computer. Some uses for the UDP system include:

- As part of a workflow for generating files in specialized software and sending those files to LightBurn for cutting.
- Loading a file on a different computer used for controlling the laser.

Using UDP

LightBurn will listen for UDP commands on the IP address you specify. To communicate with LightBurn on the local computer (the computer you're sending commands from), use localhost (127.0.0.1). To communicate with a separate computer, use the IP address of that computer. LightBurn listens on port 19840 and responds on port 19841. The listener socket should be available on all supported operating systems, but there is an additional Windows-only helper utility called "SendUDP" to simplify launching the software and loading a file.

Using SendUDP

SendUDP runs in the command line and can launch the software on the local computer and (optionally) load a file.

Calling `"C:\Program Files\LightBurn\SendUDP.exe" "<filename>"` will load LightBurn with the specified file, or tell the currently running LightBurn instance to load the file.

> **Why the quotes?**
>
> Because the Program Files folder has a space in the name, it's necessary to put the path to the file in quotes. They can be omitted for files with no spaces in the path, but including them unnecessarily is harmless.

If you're already familiar with the command line, great! If not, here are tips for using SendUDP in both Windows Command Prompt and PowerShell.

Running in Command Prompt:

In Command Prompt, you only need to provide the path to `SendUDP.exe` followed by the path to your file like in the example below:

```
"C:\Program Files\LightBurn\SendUDP.exe" "C:\Users\JG\Projects\UDPTest.lbrn2"
```

Running in PowerShell:

Start with a `.\` and use a relative path to `SendUDP.exe` like in the example below:

```
.\"..\..\Program Files\LightBurn\SendUDP.exe" "C:\Users\JG\Projects\UDPTest.lbrn2"
```

Sending UDP Commands

Although SendUDP is Windows-only, the listener socket should work on any support operating system. Any method that allows you to communicate with the listener socket should work for sending commands. Our examples will use Python.

Basic Example

The example below will command LightBurn to start running the currently open file on the laser.

```
import socket
sock = socket.socket(socket.AF_INET, socket.SOCK_DGRAM) # (1)
sock.sendto(b"START", ("127.0.0.1", 19840)) # (2)
```

1. This line configures the script to communicate with the listener socket.
2. Change the command from **START** to one of the other valid commands to perform other actions.

Advanced Example

This is a more complete example of using UDP commands in Python, adapted from a post in the

```
import socket

UDP_IP = "127.0.0.1"
UDP_OUT_PORT = 19840
UDP_IN_PORT = 19841
MESSAGE = b"LOADFILE:F:\\BeeTest.ai" # (1)
# MESSAGE = b"FORCELOAD:F:\\BeeTest.ai"
# MESSAGE = b"IMPORT:F:\\BeeTest.ai"
# MESSAGE = b"LASER:Laser1"
# MESSAGE = b"CLOSE"
# MESSAGE = b"FORCECLOSE"
# MESSAGE = b"START"
# MESSAGE = b"STATUS"
# MESSAGE = b"PING"

print("UDP target IP:", UDP_IP)
print("UDP target port:", UDP_OUT_PORT)
print("UDP input port:", UDP_IN_PORT)
print("message:", MESSAGE)

outSock = socket.socket(socket.AF_INET, socket.SOCK_DGRAM)
inSock = socket.socket(socket.AF_INET, socket.SOCK_DGRAM)

inSock.bind( (UDP_IP, UDP_IN_PORT) )

outSock.sendto(MESSAGE, (UDP_IP, UDP_OUT_PORT))
```

```
data, addr = inSock.recvfrom(1024)
print(data, addr)
```

1. The `b` before the string converts it into a bytes-like object. The socket protocol doesn't accept strings here.

Available UDP Commands

All commands will return `OK` if successful, `!` if unsuccessful, or `?` if invalid.

`STATUS` returns `!` if the laser is busy and `OK` if it isn't busy, allowing a script to wait for the laser to finish running.

Command	Action
LOADFILE:<path>	Loads the specified file in the currently open LightBurn window. Do not include `<>` in the command.
FORCELOAD:<path>	Loads the specified file in the currently open LightBurn window, skipping the warning dialog for unsaved changes to the current file. Do not include `<>` in the command.
IMPORT:<path>	Imports the specified file into LightBurn and places it in the center of the workspace. Do not include `<>` in the command.
LASER:<name>	Selects the named laser.
CLOSE	Closes LightBurn.
FORCECLOSE	Closes LightBurn, skipping the warning dialog for unsaved changes to the current file.
START	Starts running the currently open job.
STATUS	Checks the status of the laser. Returns `OK` if the laser is connected and not busy.
PING	Checks the status of LightBurn. Returns `OK` if LightBurn is open and does not have a dialog window open.

Shared Settings for Multiple Users

For schools, makerspaces, businesses, and other users who may need to configure multiple computers to share Devices, Settings, Material Libraries, and other configuration, we have some tools available to help you.

Licensing

To set up LightBurn licenses from the command line, see the directions for System Locked License Setup

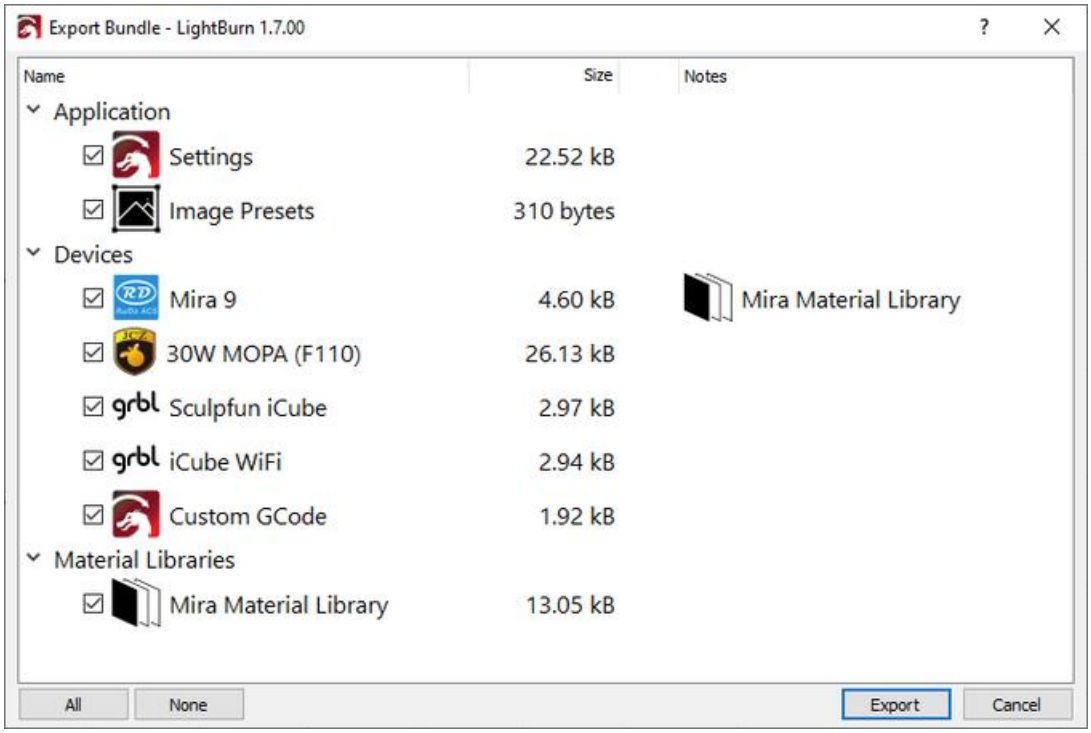
Initial Configuration

The easiest way to configure multiple computers is with LightBurn's User Bundles, which are described further in our guide to Migrating Between Computers.

User Bundles can contain any or all of the following:

- LightBurn Settings
- Custom Hotkeys
- Material Test Presets
- Image Presets
- Devices (including cameras)
- Material Libraries
- Art Libraries

To share your settings:



1. Make LightBurn is configured the way you want on your first computer.
2. Create a User Bundle by going to **File → Bundles → Export Bundle**.
3. Save your User Bundle somewhere accessible to all the other computers you want to configure. The exact way you do this will depend on your setup, but a few possibilities:
   - If you have user profiles managed by your IT department, they may be able to place the `.lbzip` file somewhere accessible (such as the desktop) for all users.
   - Save the file to a network drive. If you're in a school or other environment with an IT department, they may be able to help you set one up if you don't currently have one.
   - Save the file to a shared folder in a cloud storage service like Google Drive, OneDrive, or Dropbox.
   - Use shared folders through your operating system or a service like Syncthing.
   - If you don't have any network-based sharing options, save to a USB drive or similar.
4. On each other computer you want to configure, open LightBurn and load the User Bundle by doing one of the following (or have users do this):
   - Go to **File → Bundles → Import Bundle** and select the `.lbzip` file you created.
   - Drag the `.lbzip` file into the LightBurn window.
5. Select the components of the bundle you want to use and click **Import**.

If you keep the User Bundle available for users, they can repeat the import process if they need to restore their configuration.

> **defaultprefs.ini**

> In earlier forum posts, there are references to using a `defaultprefs.ini` to set default Lightburn settings each time the program starts up. At the present time, this is not currently recommended as it may not work reliably.

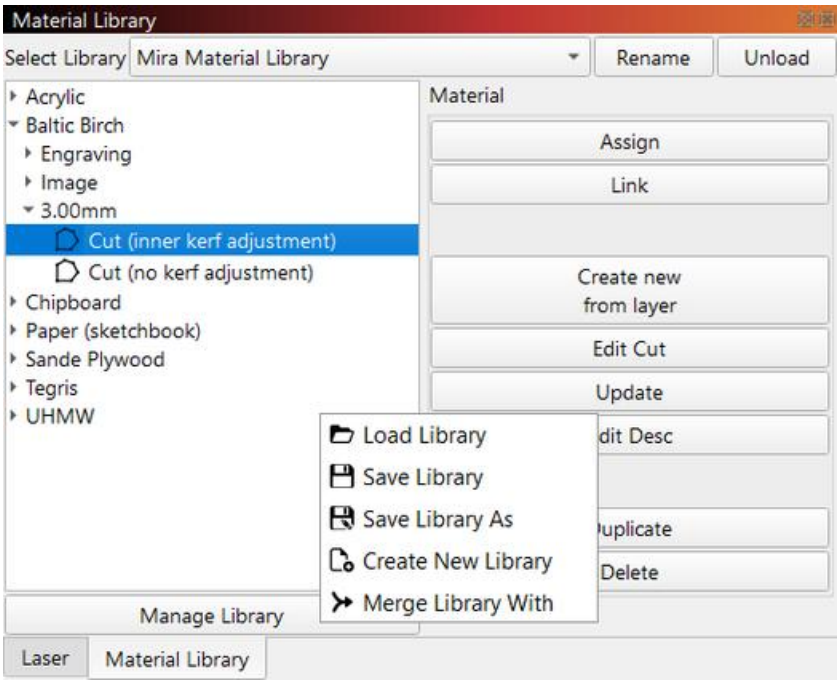
Ongoing Updates

While you probably won't be changing your Device List very often, you may want to update Material Libraries or Art Libraries more often and share those changes with users.

Libraries are reloaded every time LightBurn starts up, so you can have each computer load a Material Library or Art Library from a shared file, and at every startup, the most recent version will be loaded.

Material Library

Material Libraries make it easy to store, look up, and apply cut settings. You can use Material Libraries to share your cut settings between computers. Once a Material Library is loaded, applying your cut settings is as easy as opening the Material Library Window, selecting the correct Material Library entry, and choosing either **Apply** or **Link**.

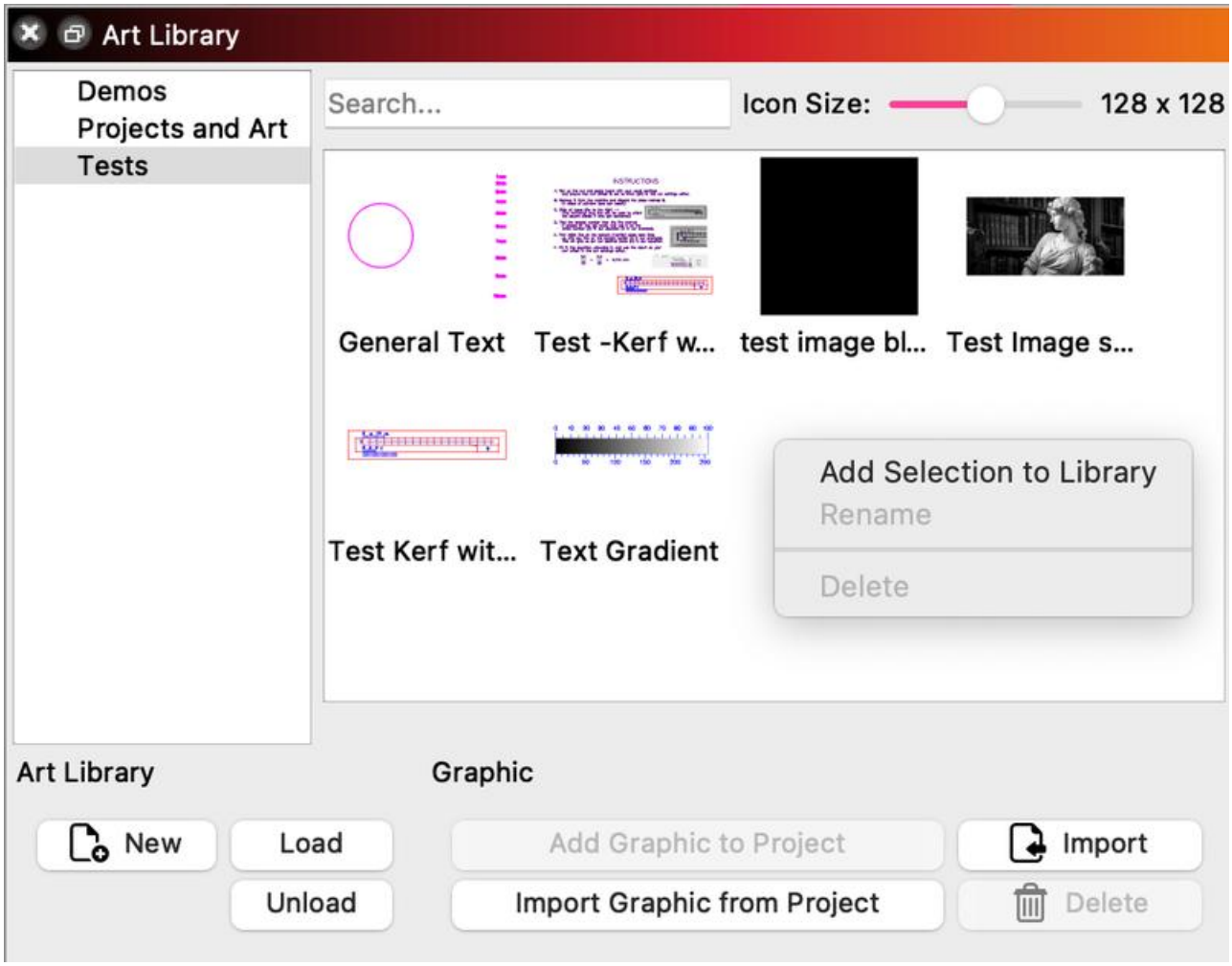


Use the Manage Library button to manage Material Libraries.

- Use Save Library As to save a copy of your active Material Library to a location of your choice. If you're sharing your Material Library with other users, this may be better than sharing your Material Library directly, in case other users change the entries unexpectedly.
- Have users on other computers use Load Library to load your shared Material Library file from a shared file.

Art Library

For art resources you want to share, consider using a shared Art Library. This can be useful for shared artwork, but also for creating placeholders representing engraving blanks you use, various cutting templates, or material sizes.



- To share an Art Library, find the `.lbart` file for the Art Library you want to share and copy it to a shared location.
- Users can load an Art Library by clicking the **Load** button in the bottom of the Art Library window and selecting your shared `.lbart` file.

Related Topics

- Educational Info
- System Locked and FLoating Licenses
- User Bundles

Network and Wireless Connections

Connect to a Ruida or Trocen Laser with Ethernet

If you have a laser using a Ruida or Trocen controller, you may want to connect via Ethernet to avoid using excessively long cables. This guide will help you determine an appropriate IP address to use and set that IP address on the laser controller itself.

Networking

Ethernet on macOS 15+

If you're using macOS 15 or higher, you'll need to allow LightBurn access to the local network.

DSP controllers operate over IPv4, or Internet Protocol Version 4, which consists of unique external and unique local addresses within a network. They need a static – or unchanging – IP address for LightBurn to be able to find them.

In order to connect a laser over Ethernet, you'll need to select a static IP address that doesn't interfere with other devices on your network. If you're comfortable doing that on your own, select an IP address appropriate for your network and jump ahead to setting the IP address on the controller. Otherwise, keep reading for help.

More on Networking

Most modern networking involves DHCP, where addresses are loaned temporarily to devices, like laptops and phones. More in-depth networking information can be found at Google Support, Simple Wikipedia's explanation, or other technical resources like Cisco or Meraki's websites. This is a complex topic with a wide range of possible setups, so we are unable to cover every situation.

Choose an IP Address

Choosing a static IP address that doesn't interfere with other devices on your network can be a little tricky if you aren't already comfortable with computer networking, but home networks are usually small enough that we can get by with picking an address that's unlikely to be used already.

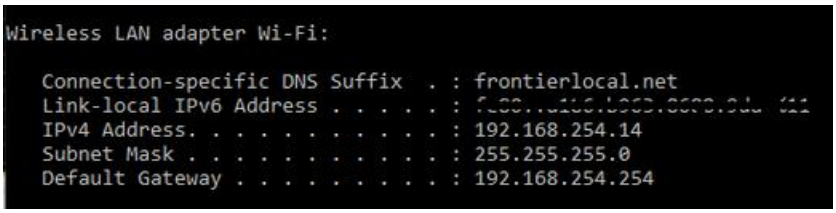
For more complex network situations, such as a school or work network, we recommend working with your IT/networking team to select an IP address and connect your laser to ensure that you don't cause problems for your network infrastructure.

1. Find Your Computer's IP Address

To start, you'll want to find your computer's local IP address. It will be four sets of numbers separated by periods, similar to `192.168.1.19`. See below for tips on how to do that on Windows and Mac.

Windows

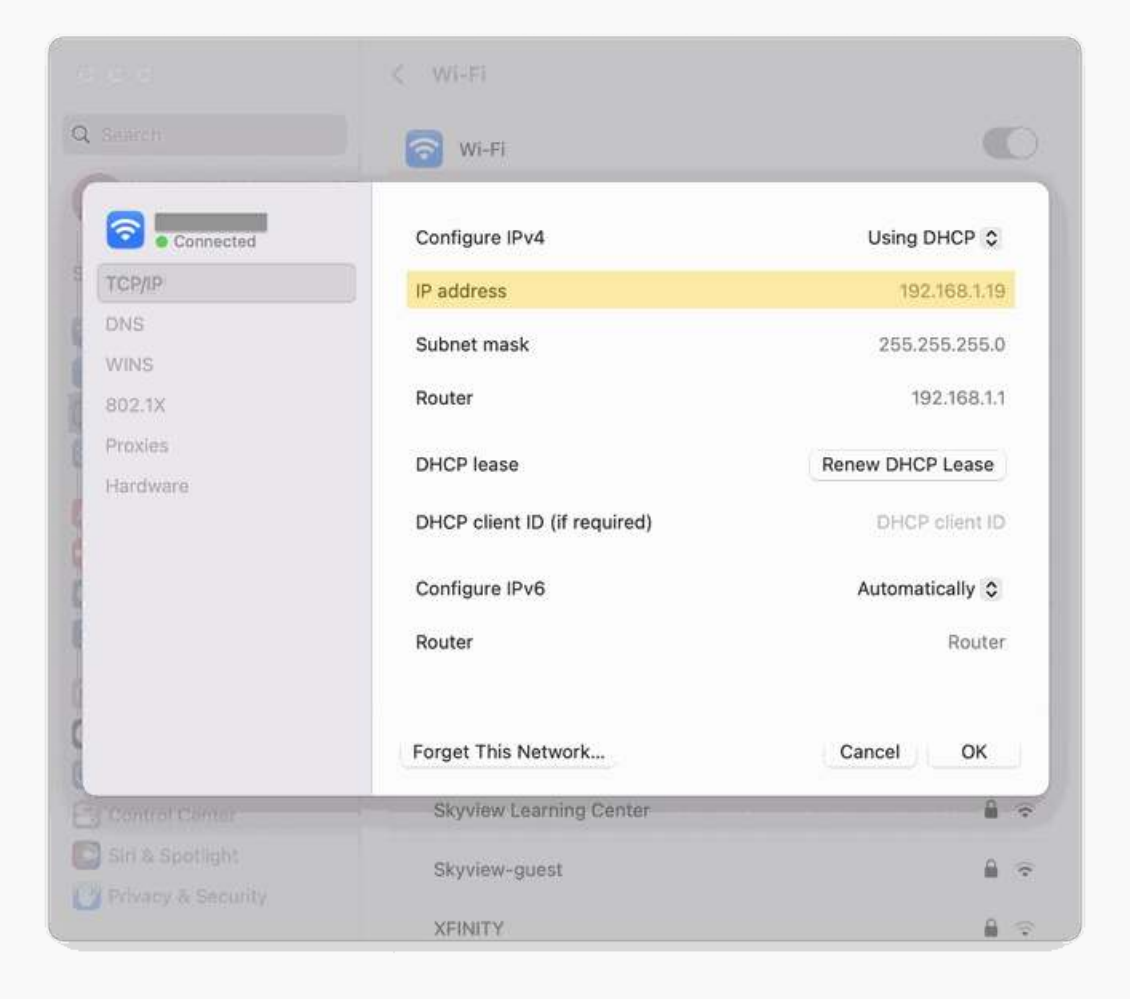
Open the command prompt (type "cmd" in the start menu or in the `Win`+`R` search box). At the command prompt, type `ipconfig`, and press `Enter`. Your results will look similar to this:



The line that says `IPv4 Address` is what you're looking for. Make a note of this address and move on to choosing a unique IP address.

Mac

Open Spotlight with `Cmd`+`Space` and type `IPv4` into the search bar to bring up your IP address in system settings.



Make a note of this address and move on to choosing a unique IP address.

2. Choose a unique IP address for your laser

Start with the IP address you found in the previous step. The first three segments need to stay the same, but the last segment needs to be a value from 1 to 255 that isn't already used on your network. An easy trick that usually works is to add or subtract 100 from your computer's address. This makes it less likely that another device connecting to your network will automatically be given the same IP address you choose.

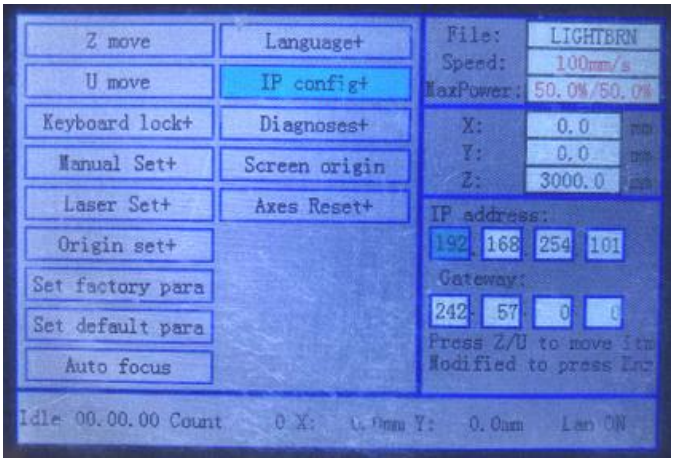
Starting IP address	Try this IP address
`10.10.1.154`	`10.10.1.54` (subtracted 100)
`192.168.2.201`	`192.168.2.101` (subtracted 100)
`10.100.50.14`	`10.100.50.114` (added 100)

This trick doesn't guarantee a unique address, so if it's not working, try a few different IP addresses by varying the last segment.

Set the IP address on the controller

> **Note**
>
> Ruida and Trocen controller configurations can vary. This is a guideline to how it typically works, but if you're having trouble finding it, please either check the manual for your controller or contact support for help.
>
> These directions are for Ruida controllers. We will include directions and screenshots for Trocen in the near future.

On the Ruida controller, use the Z/U or Menu buttons to enter the menu, navigate to **IP config+** , and press enter to access that menu setting. On the right side of the screen, the IP address field and gateway will be visible. Use the left/right arrows to move through the octets (sections), and the up/down arrows to change the value. Press and hold the up/down arrows to change the value.



When done, press the **Enter** button.

Add the laser in LightBurn

DSP controllers accessed over a network are a separate type of connection than connecting directly over USB, and must be added separately, even if you've already added the laser to LightBurn over a USB connection.

Networked lasers can't be detected by **Find My Laser**, so you'll need to add the laser manually.

> **Warning**
>
> In the event this communication method proves unreliable, you may want to consider using a LightBurn Bridge, especially if you're using macOS. The Bridge provides a much more stable network connection than the Ruida controller's UDP connection.

Next: Add a Laser

Source

This guide was originally made for our forums on October 20th, 2019. You can find the original guide here.

GRBL Network Connection Setup

This guide explains how to set up a GRBL laser over a local network (e.g. your home router).

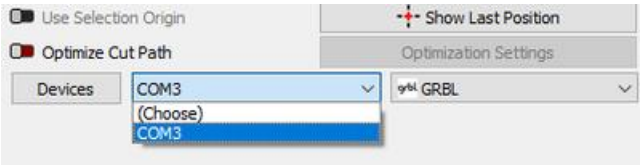
The Sculpfun iCube is used as an example, but the process should be similar for any GRBL device capable of network connections.

We've attempted to point out anything that is device-specific throughout, but GRBL network configuration is not fully standardized. If your device's network settings don't match this guide, please email support or visit our forum, where we can help you with your device and gather the information we need to update this page to help other users.

Part 1: Connect Over USB

> **Why do I need USB? Isn't this about WiFi?**
>
> You'll need to connect with USB to update a few settings on your laser. If your laser is already connected with USB, you can skip to Part 2.
>
> If your laser is already connected to WiFi and you just want to connect a second computer to it, you can skip to Part 4.

1. Turn on the device and connect to the computer with a USB cable.
2. Use the USB connection to add the device as described in Find My Laser.
3. Select the device in the Laser Window:
   - If you see **Ready** right below the title bar of the window, your laser is connected and ready to use.
   - If you see **Busy** and a green progress bar, your device is busy doing something. Many GRBL devices will home themselves when first connected, so you'll see the Busy message until homing is complete.
   - If you see **Disconnected**, your device is not connected. Make sure you have the correct serial port selected (the dropdown that says **COM3** in the screenshot below). If your laser still isn't connecting, select **(Choose)** in the same dropdown and then reselect the serial port to reset the connection. If you're still unable to connect, see Troubleshooting: Connection Problems.

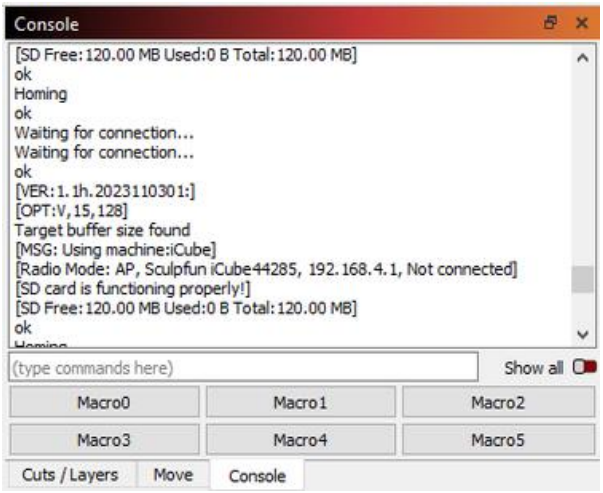


4. Open up the Console Window:
   - By default, the Console window is in the same section of the screen as the Cuts / Layers and Move Windows.
   - If you don't see the Console window, go to the **Window** menu and make sure **Console** is checked.

- In the next two sections of this guide, you'll use the Console Window to configure your network connection.

Part 2: Find Network Info in Console

Immediately after you connect to your laser, you should see a series of messages in the console describing your laser's status.



You need information about your laser's network status, shown in the line starting with `Radio Mode`:

```
[Radio Mode: AP, Sculpfun iCube44285, 192.168.4.1, Not connected]
```

Your laser won't send the exact same message, but you should see something similar. Here's what each part of the message means, based on the example above:

Message	Meaning	Explanation
`Radio Mode: AP`	Network type, typically either `AP` or `STA`	If you see `AP` here, your laser is in Access Point mode, allowing you to connect directly to the laser over WiFi. If you haven't changed your laser's network settings, it will likely be in this mode. If you see `STA` here, your laser is in Station mode and will attempt to connect to an existing WiFi network. This is not universal, and some devices are configured differently.
`Sculpfun iCube44285`	WiFi network name, or SSID	The name of the WiFi network the laser is either broadcasting (in Access Point mode) or attempting to connect to (in Station mode).
`192.168.4.1`	Laser's IP address	This is the IP Address of the machine — it's how a computer knows where to find it on a network.
`Not Connected`	Connection Status	This tells you whether your laser is connected with any devices over the network connection.

Your specific device may not match this exactly. Please email support or visit our forum if you need help with your specific device.

Part 3: Configure Network Connection

This process varies between devices. If you're not sure how to change WiFi settings and the directions below don't work for you, try checking your laser's user manual or seeking support from your laser's manufacturer. These settings are stored on the laser, so once your laser connects to the network, you shouldn't need to repeat this process until your WiFi network changes.

Your laser may come with a mobile app or web interface that allows you to configure network settings. If you aren't able (or don't want) to use these, skip ahead to Configuration in LightBurn.

Configuration With App or Web Interface

> **Tip**
>
> Many devices will automatically disconnect from a WiFi network without an internet connection. You may need to modify your device's network settings to connect directly to your laser. See the support resources for your phone or computer for help with your network settings.

If your laser is in Access Point mode, you may be able to configure the network connection using a mobile app provided by the manufacturer or a web interface. This isn't always possible and the process varies between lasers, but in general you will need to start by connecting a phone or computer to the laser's WiFi network. If you need a password to connect, check your laser's user manual for the default password. For the Sculpfun used as an example here, the default password is "12345678".

If your laser uses a web interface, enter the IP Address from Part 2 where you would usually type in a website's URL to access it.

Here are the settings you will need to configure, although they may be displayed differently for your laser:

Name	Value	Notes
Network type	Typically STA or Station	You want to connect to an existing network. You don't want AP or Access Point
SSID	Your WiFi network's name, or SSID	Newer routers may have 2.4GHz and 5GHz connections available. If you see both, choose 2.4GHz
Password	Your WiFi network's password	Enter this carefully to avoid entering it incorrectly

Configuration in LightBurn

To configure your laser's network settings, you will need to use the Console Window.

Network configuration varies between devices, so check your laser's user manual for more information. The Sculpfun used as an example here came with a sheet of paper listing the appropriate parameters. Regardless of what laser you're using, you will need to change the settings listed below, but the number after `$` may be different for your specific laser:

Name	Parameter	Value
Network type	`$50`	Typically `STA`
SSID	`$53`	The name of the WiFi network to connect to
Password	`$54`	The password to connect to the WiFi network

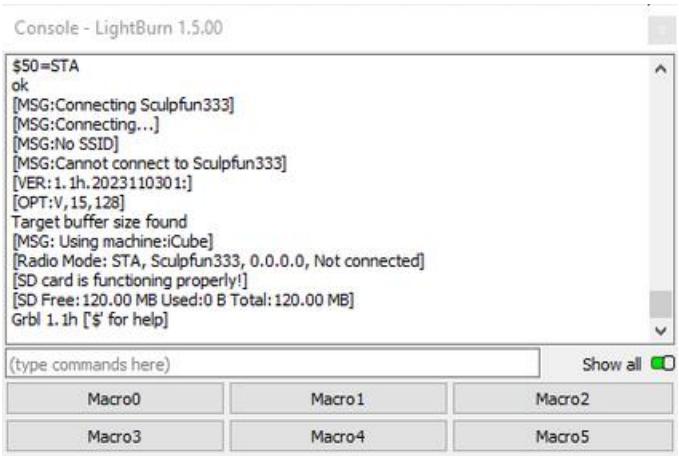
Each command given is entered into the text box at the bottom of the **Console Window** and followed by pressing `Enter`.

To check the value for any of these parameters before changing it, you can use the parameter without assigning a value. To change the value, add `=YOURVALUE` after the parameter. As an example, `$53` will show the name of the WiFi network to connect to. `$53=MyWifiNetwork` will configure the laser to connect to a network called "MyWifiNetwork".

Each of the following steps will use the parameters shown above. If your laser's manufacturer provides different parameters, use those parameters.

> **Known Exceptions**
>
> The Spider X1 requires that `$50` be set to `2` to enable WiFi. As we identify additional devices that differ from the settings shown above, we will add them here.

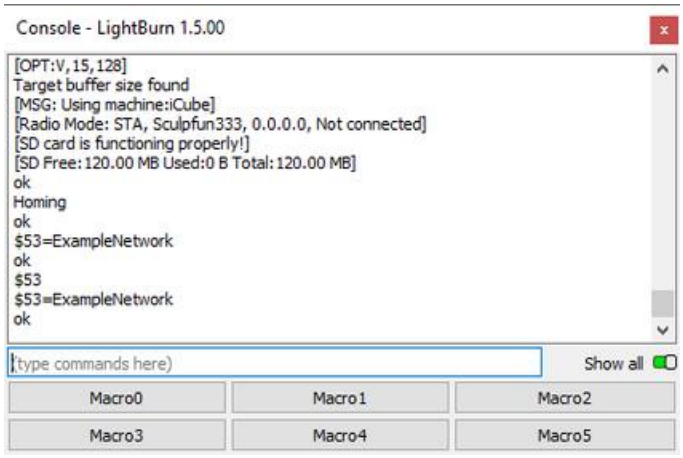
Set Network Type



Use `$50=STA` to set your laser to Station mode. This tells it to connect to an existing network.

Don't worry for now if you see `[MSG: No SSID]` or a similar message in the Console window. This is your laser's way of telling you that it doesn't know what network to connect to yet.

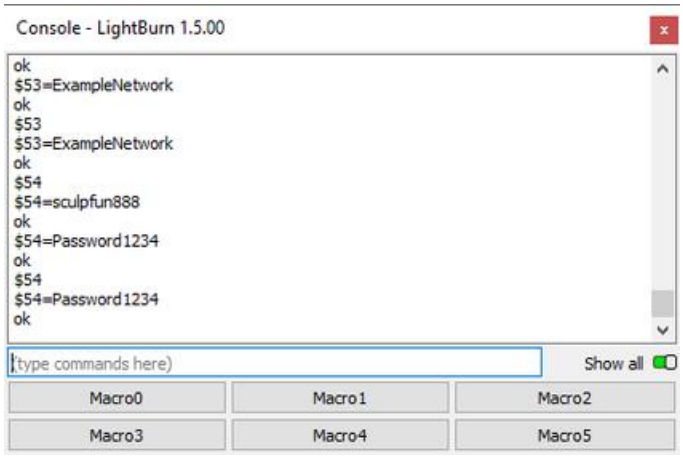
Set Network Name



Use `$53=ExampleNetwork`, to set the SSID, replacing "ExampleNetwork" with the actual name of your WiFi network.

Make sure you type accurately, including capitalization.

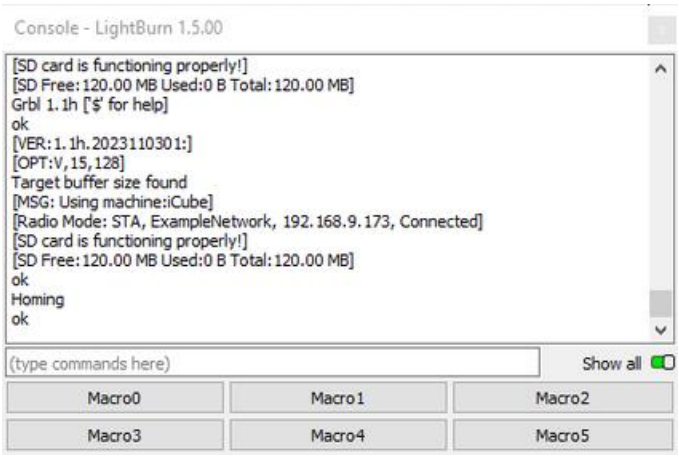
Set Password



Use `$54=Password1234` to set the WiFi password, replacing "Password1234" with your actual password. The screenshot above also shows how to use `$54` to check the saved password.

Make sure you type in your password accurately, including capitalization. This should be the password to connect to WiFi, not the login info for your computer or any accounts you may have.

Check Connection



After configuring network settings, you may need to restart your laser to get it to connect. The `Radio Mode` status message now shows updated network settings and ends with `Connected`:

```
[Radio Mode: STA, ExampleNetwork, 192.168.9.173, Connected]
```

Make sure you write down the IP Address (`192.168.9.173` in the example above) for easy access in Part 4.

Part 4: Network Connection in LightBurn

Once you have an IP address, you can set up a network connection in LightBurn.

1. In the Laser Window, click the Devices button and choose Create Manually. Click **Next** to continue.
2. Select **GRBL** to create a new GRBL device and click **Next**.
3. Select **Ethernet or Wifi/TCP** as your connection type and click **Next**.
4. Enter your device's IP address or hostname from Part 3 and click **Next**.
5. Give the device a name that makes sense to you and set the size of the work area. Click **Next**.
6. Check that the origin and homing settings are correct and click **Next**.
7. Make sure the summary looks correct and click **Finish**.

At this point, you should be able to connect to your laser through the network connection by selecting the device you've just added in LightBurn.

Troubleshooting

- If your laser isn't able to connect to the WiFi network in Part 4:
  - Check that the SSID and password are correct.
  - If you are using a 5GHz WiFi network, try using a 2.4GHz network instead. Most routers that have 5GHz WiFi will also have a second SSID for 2.4GHz.
- Ensure your computer is connected to the same network as your laser.

- If you have trouble connecting, you may need to change the network port LightBurn is using to communicate with your laser.
- If you can connect but the laser is stuttering, stopping, or restarting mid-job, you may need to switch to Synchronous Transfer Mode.

LightBurn Bridge

Connecting to the LightBurn Bridge

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=LaNQv5C33s8

Follow the steps below to connect to your laser using a LightBurn Bridge.

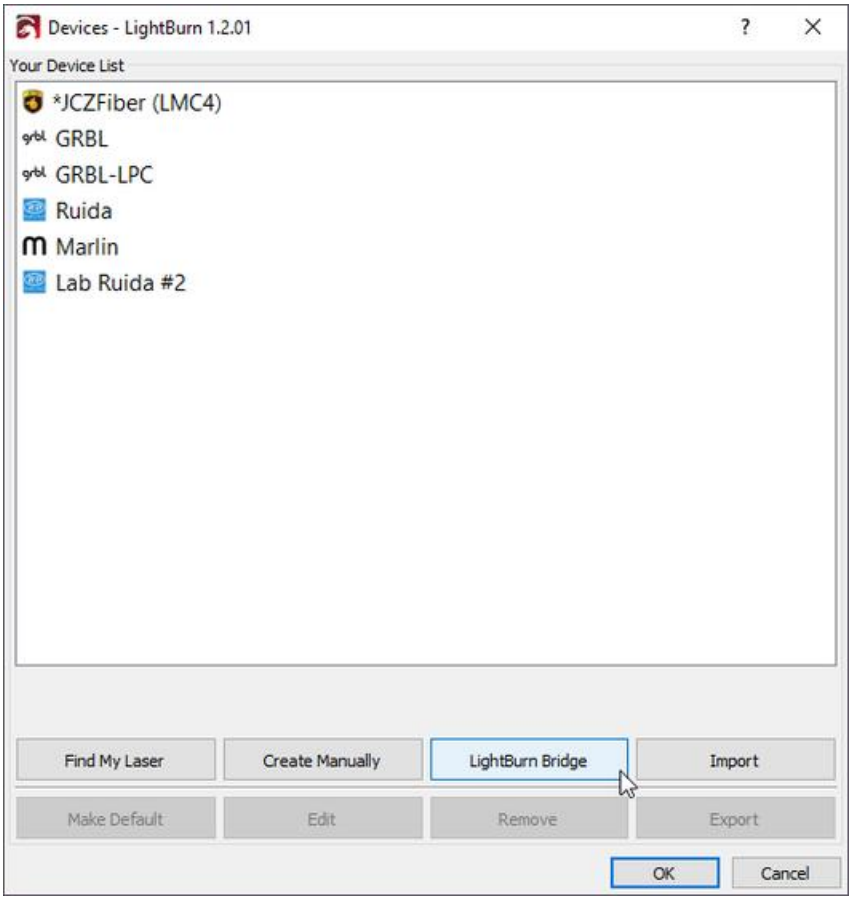
> **Note**
> The LightBurn Bridge is only compatible with lasers using Ruida controllers.

Ruida Controller Configuration

The LightBurn Bridge is pre-configured to connect to a laser configured with an IP address of **10.0.3.3**. In the Ruida control panel, set the IP of your laser to this value, then connect a network cable from the controller to the Raspberry Pi, and plug in the Pi with its power adapter. If you need to change the IP address, see LightBurn Bridge Advanced Setup — Configuration Options, and please note the caveats to doing this, listed there.



In LightBurn, in the Devices screen, click **LightBurn Bridge** to set it up:



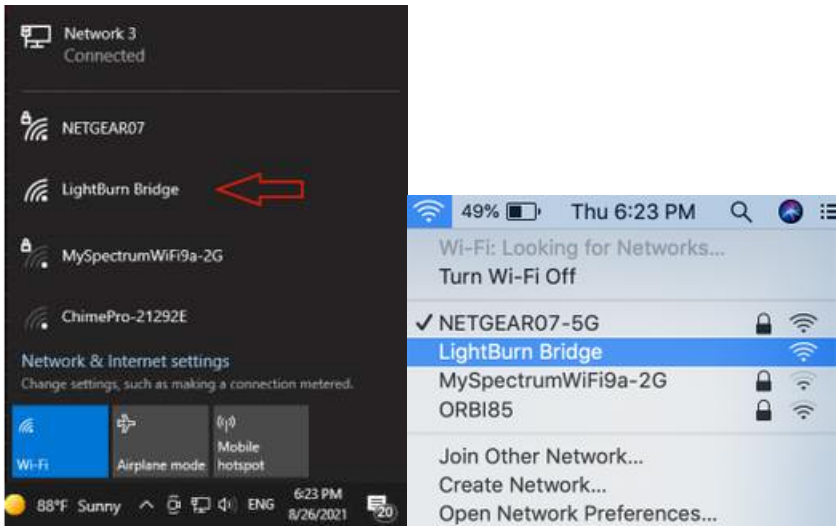
Connecting the LightBurn Bridge to Your Network

> **Ethernet on macOS 15+**
>
> If you're using macOS 15 or higher, you'll need to allow LightBurn access to the local network.

LightBurn will open the LightBurn Bridge discovery wizard to guide you through the setup.



On your computer or a mobile device, use the WiFi settings to find the LightBurn Bridge device, and connect to it:



Note

The access point name you find will typically have a four-character random string at the end of the name, such as `LightBurn Bridge E4B7`. This is to prevent conflicts if you have multiple LightBurn Bridge devices in the same location.

When you've selected the LightBurn Bridge access point, click **Next**.

Note

You may also configure the WiFi manually, as shown on the Advanced Setup page.

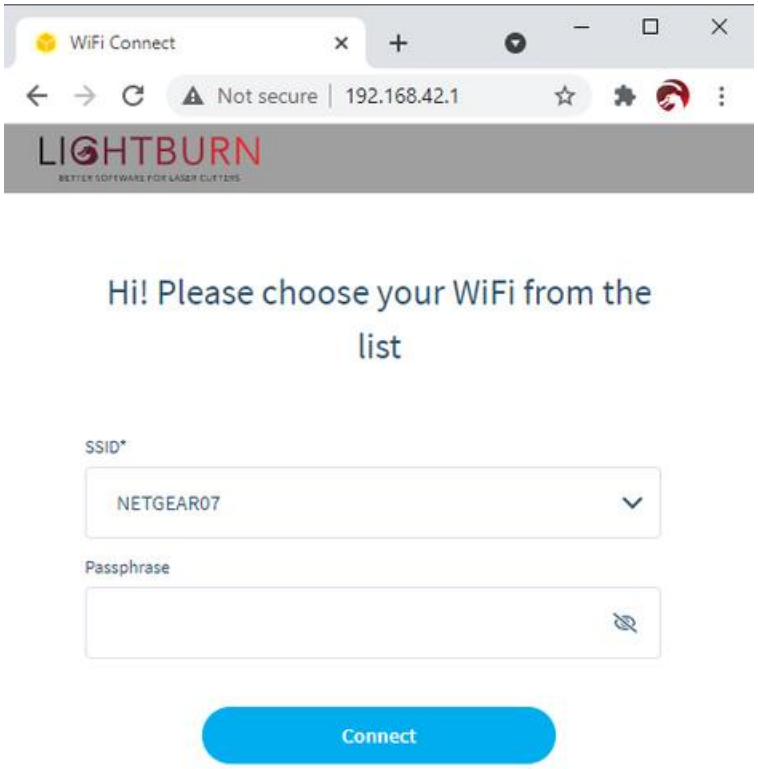
Configuring the Bridge in LightBurn

Now you need to tell the LightBurn Bridge which WiFi network to connect to, and give it the network password. In most cases, the configuration page on the Bridge will open automatically. If it does not, browse to `http://192.168.42.1` as shown in the wizard page below. If you want to use your PC to configure the Bridge, click the button shown to open your browser to the correct page:



Setting the WiFi name and password

In the browser window, you'll see a page like this:

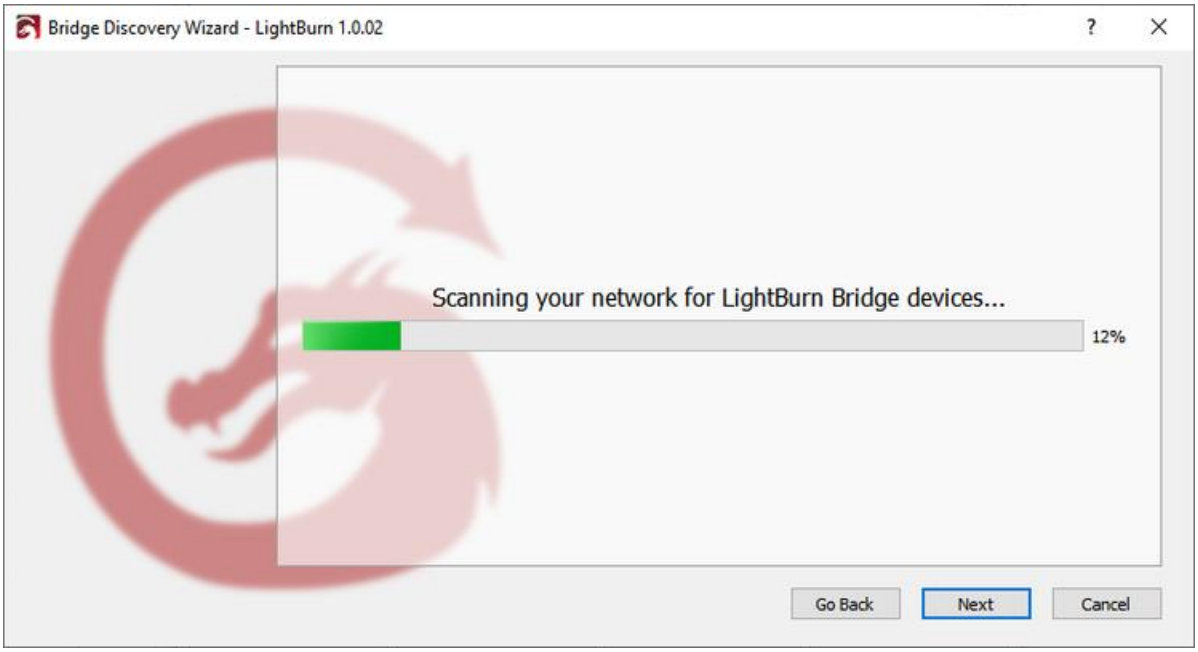


Select your WiFi network from the list of options, enter your password, then click the **Connect** button at the bottom. You'll see a message appear near the top, telling you that the bridge is connecting to your WiFi network. You can now close the browser window and go back to the LightBurn wizard.

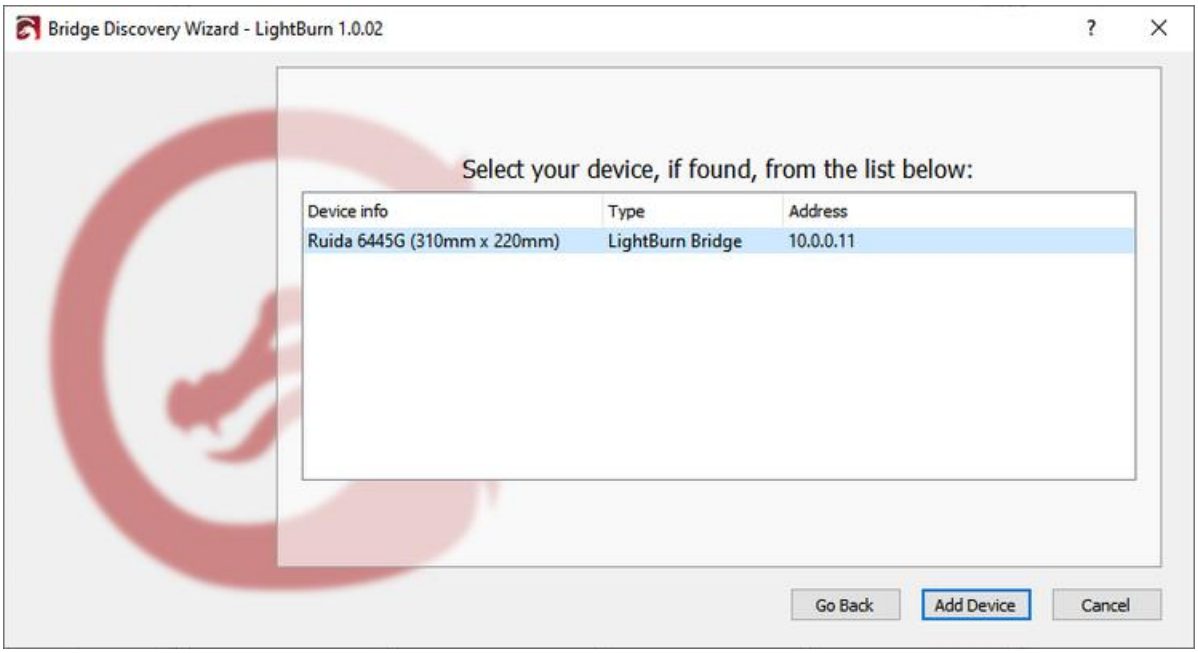
If you were using your PC to configure the LightBurn Bridge, re-connect the PC to your WiFi network if you need to.

Finding the LightBurn Bridge From Your PC

LightBurn will now scan your network, looking for connected devices, to find the new LightBurn Bridge. You'll see a progress bar appear as it does this:



If LightBurn is able to connect to the Bridge device, and the Bridge was able to talk to your laser, you should see something like this:



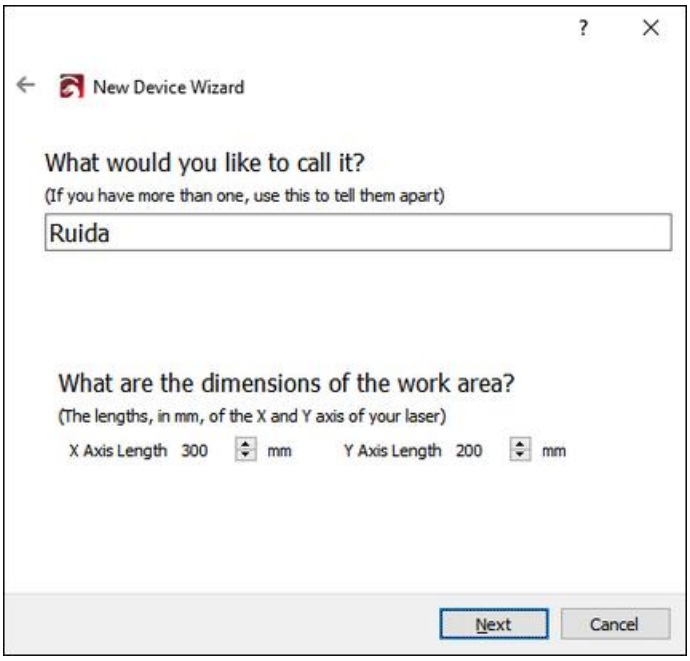
If everything looks good, click **Add Device**, and you'll be taken to the setup screen for the device, asking you to enter the name, check the size, and so on.

If the either the Bridge or the laser are not found, you may need to change the IP address on your laser to 10.0.3.3 (the default for the LightBurn Bridge), or, optionally, you can configure the bridge to use a different address for your laser.

Name and Work Area Size

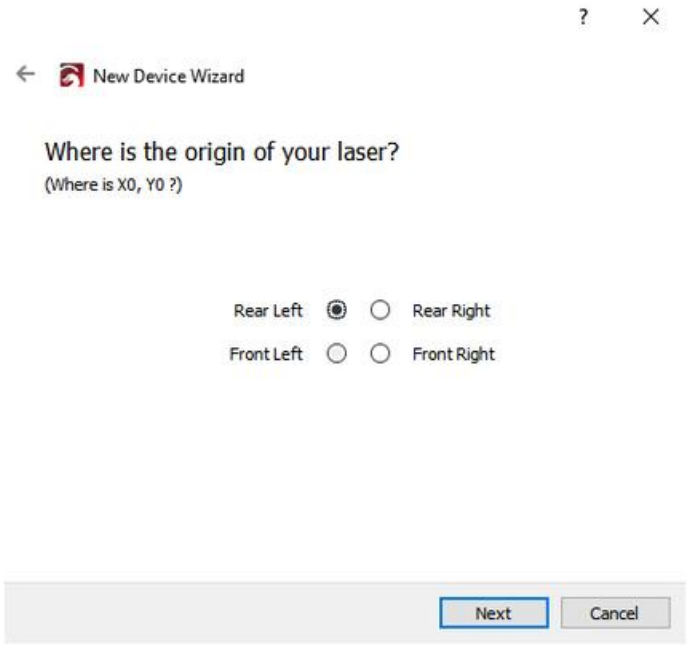
You can name the laser, which is very useful if you have more than one, or just leave it as is.

You must set the size of the work area for your laser so that LightBurn can try to prevent things from going out of bounds. If you don't know the exact size, you can change this later in the Device Settings page.



Laser Origin

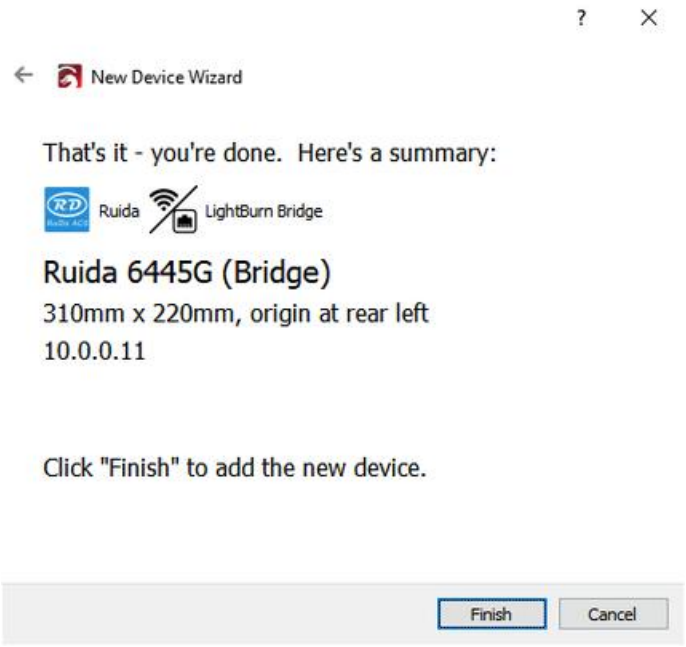
The origin setting is where the zero point of your X and Y Axes meet. If you get this wrong, you can change it later in Device Settings page. This setting also controls the orientation of the output — if it's wrong, the output from your laser may be mirrored or upside down.



With a Ruida controller, the origin corner will be the corner that the laser head seeks out when you power it up.

All Done!

That's it — the final page will show you a summary of your choices. You can go back and fix anything if necessary, or click **Finish** to create the new device entry.



Next Steps

If your use of LightBurn Bridge is more complex than indicated here, you may need to take a look at our Advanced LightBurn Bridge Setup guide.

LightBurn Bridge Kit Setup

Whether you want to roll your own LightBurn Bridge with your own Raspberry Pi or just want to dive deeper into configuration options, this page is for you.

First, please note that LightBurn Bridge is only supported on Raspberry Pi **3b, 3b+, or 4** models. Any others will not work. We can only provide limited support for self-built LightBurn Bridge devices.

We do not provide the option to install LightBurn Bridge on an existing Raspberry Pi OS installation, as it's designed as a single purpose appliance device. Any modification to the installation, aside from those mentioned below, will void any warranty or support obligations from LightBurn Software.

Components

You will need a small number of components to set up your bridge kit.

- Rapsberry Pi model **3b, 3b+, or 4**. The 1GB model is sufficient.
- MicroSD card with at least 4GB storage.
- Raspberry Pi case
- Power supply
- Ethernet cord

You can find approved sellers of Raspberry Pi boards, along with most other components sold as parts of a kit, on the official Raspberry Pi website.

Flashing the LightBurn Bridge Image to an SD Card

Download the LightBurn Bridge disk image file from here and save it to your computer.

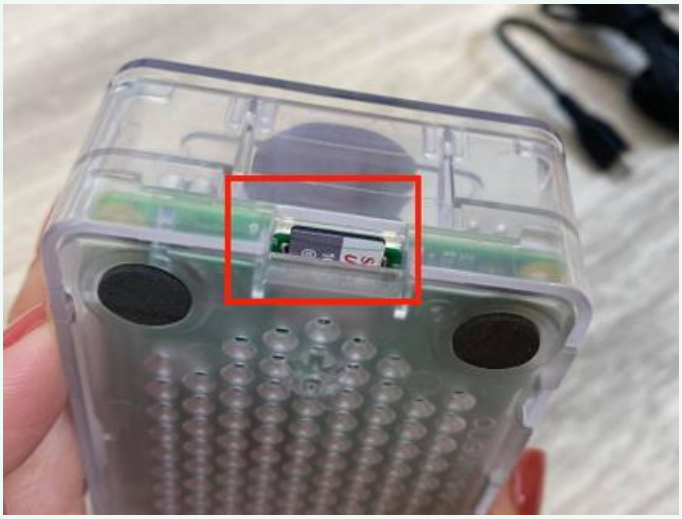
After you've downloaded the LightBurn Bridge image file, you will need to flash it to a 4GB or larger micro SD card.

For that task, we recommend you download Raspberry Pi Imager. It's cross platform and will handle the task with ease. Install the tool and open it:



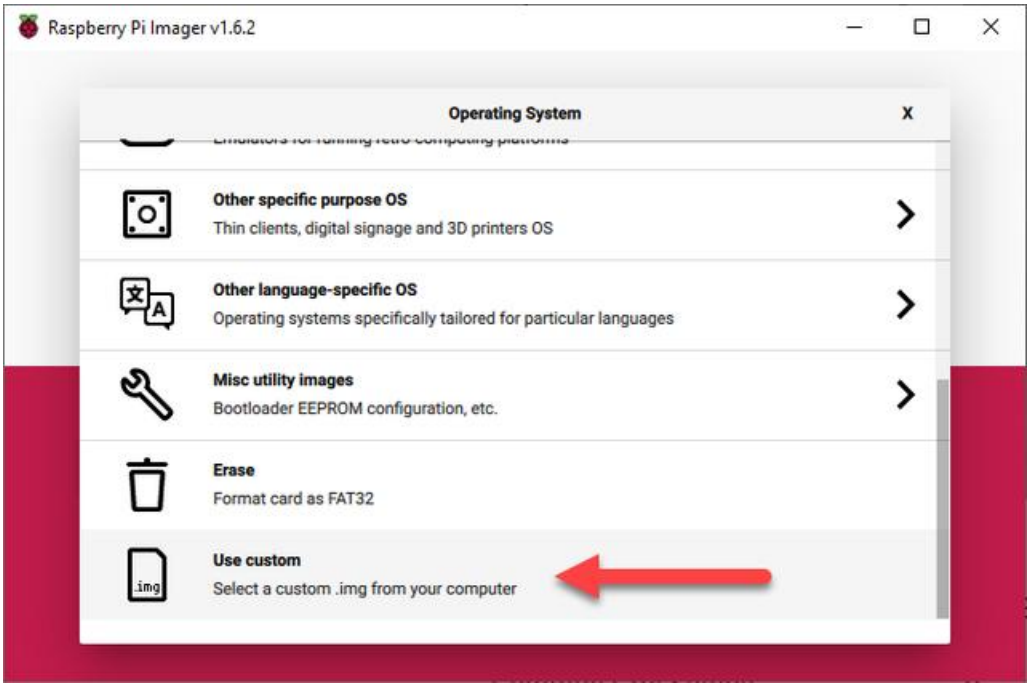
Tips on removing the SD card from an existing kit

If you're upgrading an existing Pi, power off the unit and remove the SD card from the bottom, shown here:

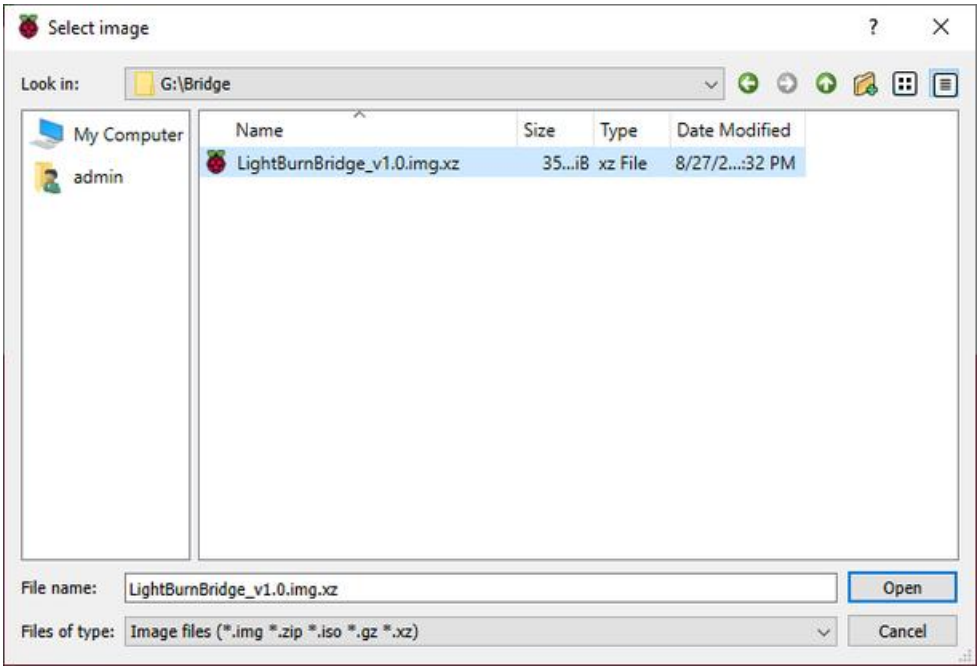


To remove the card, use a fingernail, the tip of a pen, or a pair of tweezers to hook the small lip and the end of the card and pull it out.

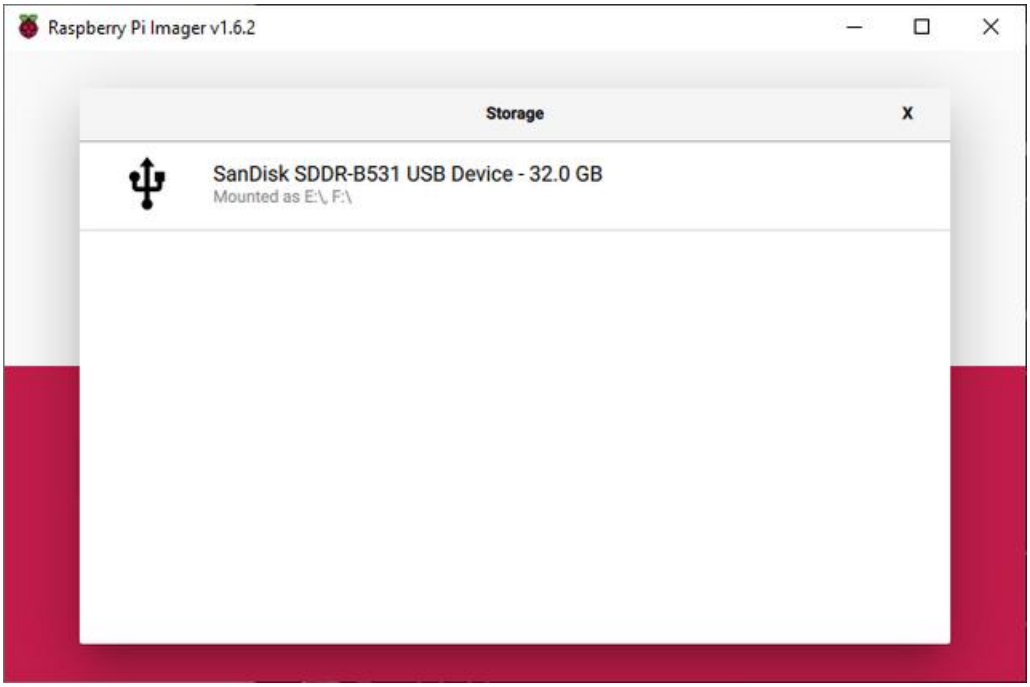
Click on **CHOOSE OS** (the left-most button in the middle), then scroll to the bottom and click on **Use custom**:



Browse to the folder you downloaded the `.img.xz` file to (most likely your Downloads folder), select the file, then click **Open**.



Insert your SD card into your computer and, back on the main Raspberry Pi Imager window, click on **CHOOSE STORAGE**. Select the correct device — in most cases it will be the only option. If you don't see it, try removing the card and re-inserting it, or unplugging your card reader and connecting it again. You can leave the **Storage** window open when you do this — when the SD card appears, select it.



Finally, click the **WRITE** button. It will warn you that you're about to overwrite the SD contents — click **OK**, and wait for it to write the image. At the end of the process, it will verify it was written correctly.

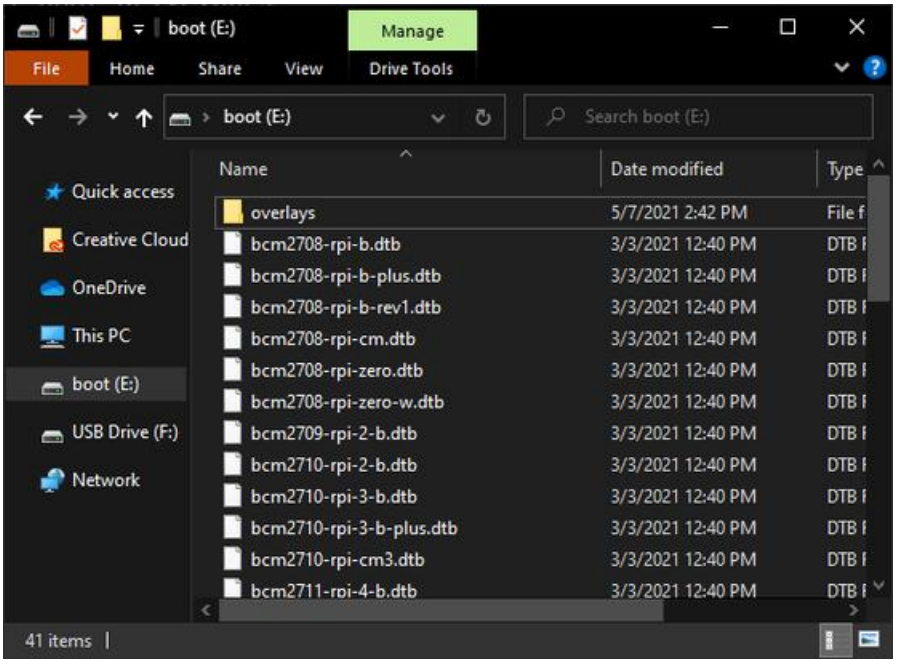
That's it! You now have your own LightBurn Bridge SD card to use in your own Raspberry Pi. If you want to do any sort of advanced configuration ahead of time, head on to the section below. Otherwise just insert the card into the Raspberry Pi, then connect it to your laser as detailed in the main guide, turn it on, and continue with the LightBurn Bridge setup.

Advanced Configuration

Manual WiFi Setup

By default, the LightBurn Bridge automatically presents a WiFi access point which you can connect to and configure your WiFi connection through. However, if that option is not working for you — for example, if you have a network with a hidden SSID — or if you would simply like to pre-configure your device, you can use the `wifi.txt` option.

Insert the micro SD card into your computer (either one you made or a pre-flashed card you purchase) and find the newly mounted drive labeled `boot` as shown below.



This will, of course, be a little different on each operating system. Some systems may show two drives/partitions and the other will likely be unreadable. That's ok, just find the one labeled `boot` and open it.

Now create a file called `wifi.txt` and open it in the text editor of your choice. Note, this must be a simple text editor like notepad, TextEdit, or gedit, and not a rich document editor like Word.

Now add two lines as follows:

```
ssid=YourNetworkSSID
pass=YourNetworkPassword
```

Replace the values after the `=` with whatever matches your network.

> **Note**
>
> Both of these fields **are case-sensitive**. Double check your network name and password for proper capitalization!

If your network is not password protected, you may leave out that line of the file.

> **Warning**
>
> Do not include anything else in this file!

Now eject the drive, remove the SD card, re-insert it into your Raspberry Pi, and power it back on.

It will read this file on boot and attempt to connect to the network.

If it succeeds, it will be renamed to `wifi.txt.loaded`. If it fails, it will be renamed to `wifi.txt.failed` and the WiFi configuration access point will make itself available as normal. The access point will generally be the first indication that the connection failed, but you can confirm by checking that file name after re-inserting the card into your computer.

If it fails, recheck the SSID and password in the file, rename it back to `wifi.txt` and try again.

LightBurn Bridge Configuration Options

You can also configure a few items manually via another file on the `boot` partition (as described above) called `bridge.json`. This file will not be created on the `boot` partition until the device has been booted up at least once, but you can create it yourself ahead of time, if desired, using the template below.

```
{
    "laser_ip": "10.0.3.3",
    "device_type": "ruida",
    "ap_name": "LightBurn Bridge E4B7"
}
```

These options are as follows:

- `laser_ip`: The static IP address configured on your Ruida laser controller. By default you will need to set it to `10.0.3.3` which we've chosen because it's unlikely to use this IP subnet for a private WiFi network. For complicated reasons, the IP address of the Ruida must be on a different subnet (the `10.0.3` part of the address) than your WiFi network. In general, you should never change this value unless this conflict exists.
- `device_type`: At the time of writing this `ruida` is your only option. There will likely be options added in the future though, so don't change this!
- `ap_name`: By default the WiFi configuration Access Point name will be `LightBurn Bridge` followed by four random letters or numbers, for example: `LightBurn Bridge E4B7`. This is so that you can have multiple devices in the same area and their access points won't conflict. But you can also set it to be whatever you want here. You may use any alpha-numeric characters and symbols, with the exception of `?"$[\]+`, and it must be between 6 and 32 characters.

After making changes, save it back to the `boot` drive/partition that is mounted from the SD card on your computer. Then insert the card back into your Pi and turn it on.

Projects and Techniques

5 Steps to Perfect Image Engraving

Getting an image to engrave well is a task that often stumps even experienced laser-folk. Often images appear too dark, too light, or "muddy" and lacking detail.

There are many settings that influence the quality of an engraved image, and each one needs to be refined for the overall image to engrave well. In this article we briefly explain the purpose of these settings and, more importantly, how to get them perfect.

Steps:

1. Focal Length

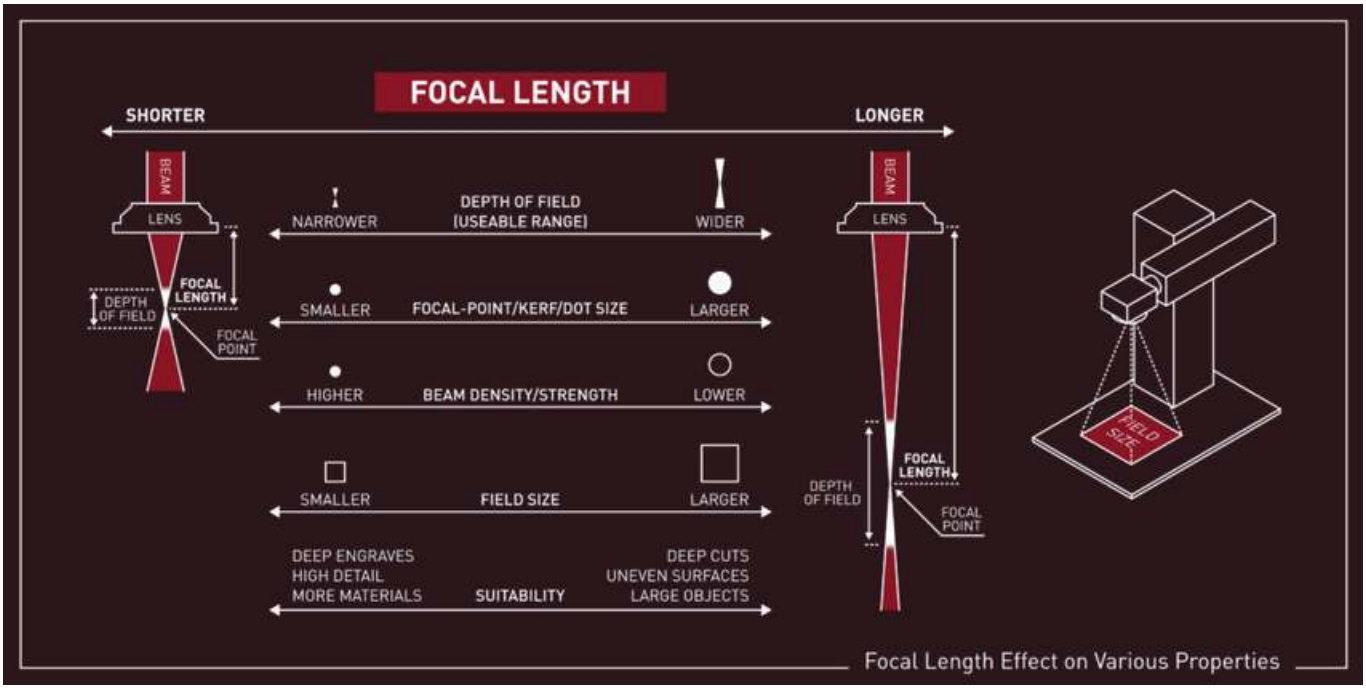


Diagram of the effect of focal length on image engravings

Focal length affects the strength and size of the laser dot where it meets the material. A laser that is perfectly focused will have the smallest dot possible, with the highest strength of the beam.

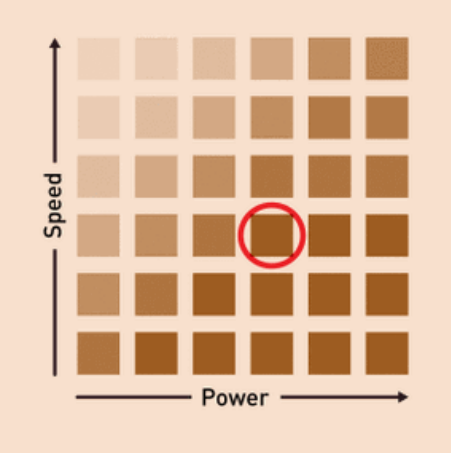
To find the correct focal length for your hardware, use Laser Tools → Focus Test, if your machine has a motorized Z Axis.

For lasers without a motorized Z Axis, use the method of focusing described by your machine's manufacturer.

> **Tip**

> Some people prefer the look of a slightly defocused laser for engraving, as it softens the edges between scan lines. However, adjusting settings is easiest with a focused laser. We recommend keeping the laser focused while conducting the following steps, and playing with defocusing as an optional step at the end.

2. Speed & Power



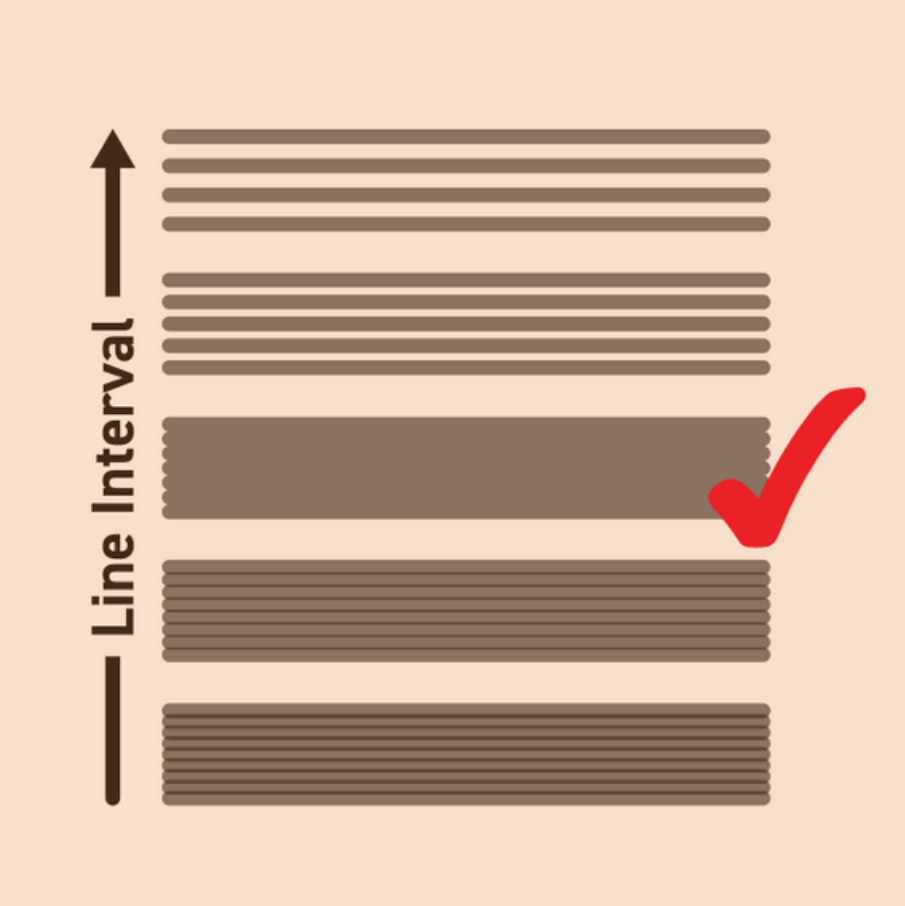
Relationship between speed and power in laser engraving

The Speed and Power settings affect the depth and darkness of an engraving.

Go to Laser Tools → Material Test to test varying Speed and Power settings, and find a value that engraves nice and dark but doesn't cut too deep. Dark engravings are generally more difficult with CO_2 lasers than diode lasers — CO_2 users will typically have more success with slow speeds and low power.

Apply the optimal Speed and Power values you identified to a layer set to Image Mode in the Cut Settings Editor, or save them to an entry in the Material Library.

3. Line Interval/DPI



Ideal settings for Line Interval/DPI

Line Interval and DPI are tied together. They change the distance between each pass (scan) of the laser head. If Line Interval is too low (and DPI too high), images will be very dark (as the lines overlap). Too high and the image will be too light.

Use Laser Tools → Material Test to find the ideal interval. You're looking for the value that creates lines that just touch (don't overlap, nor have a gap). Try values between 120 and 300 dpi (0.08-0.2 mm). You will need a microscope or strong magnifying glass.

If you don't wish to use the Interval Test, or you are having trouble seeing the overlap of lines on your material, try engraving a gradient and manually varying the Line Interval in the Cut Settings Editor.

When you have found an ideal Line Interval setting, apply it to a layer set to Image Mode in the Cut Settings Editor, or save it to an entry in the Material Library.

Tip

- When working with material with a grain (e.g wood), orient the test so the scan lines are perpendicular to the grain — parallel makes it difficult to distinguish between engraving texture and material texture.
- A raking light can help to differentiate scan lines.

- For best results, create future images with the same DPI as dictated by the Line Interval you found in this step.

4. Dot Correction

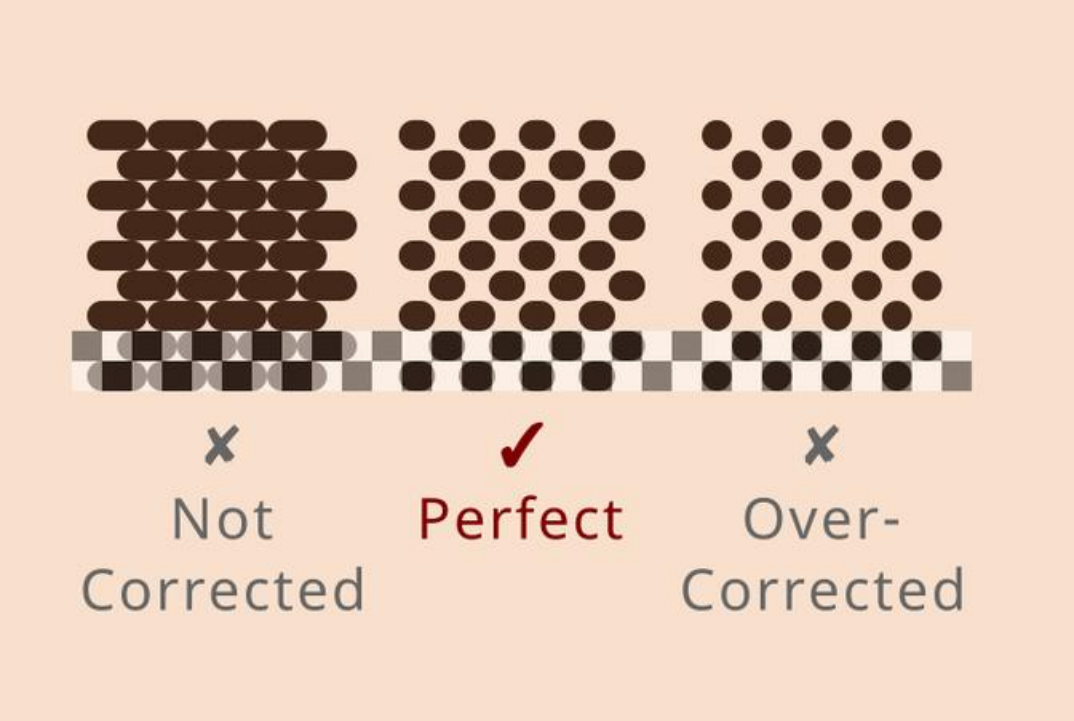


Diagram showing different levels of Dot Correction

Beams have a thickness which makes dots longer than they should be — we can correct that with Dot Width Correction. Make sure you adjust your settings in Cut Settings Editor as described below:

- Use the settings for **Speed**, **Power**, and **Line Interval/DPI** you found in the previous steps.
- Set the Image Mode to Ordered.
- Turn off Bi-directional scanning.
- Make sure Negative Image is turned off.

Additional information on these settings

Using a smoother/more organic image mode (e.g. Jarvis) would not produce this consistent pattern. Bidirectional scanning can result in every other layer being slightly offset, unless you've already perfected your Scanning Offset Adjustment table. This doesn't matter for the more organic image modes, but it's very visible in Ordered.

Ordered Image Mode on a 50% gray rectangle should produce a checkerboard pattern that is 50% engraved and 50% not. Test out the same image with different Dot Width Correction values — the number should always be smaller than whatever you're using as your Line Interval setting. The valid range is from 0 to your interval value.

Tip

> Use Convert to Bitmap to create a 50% gray rectangle.

Once you have found the optimal Dot Width Correction value, apply it to a layer set to Image Mode in the Cut Settings Editor, or save it to an entry in the Material Library.

Make sure to switch back to Jarvis Image Mode, and turn Bi-directional scanning back on (it’s a lot quicker).

5. Adjust Image

Try engraving your image. If you are still not getting the look you desire, it's likely because the image you're using needs some adjusting, which you can do using the Adjust Image tool. Try changing the Brightness, Contrast, and Gamma, along with the Enhance features.

Kerf - What it is and How to Compensate

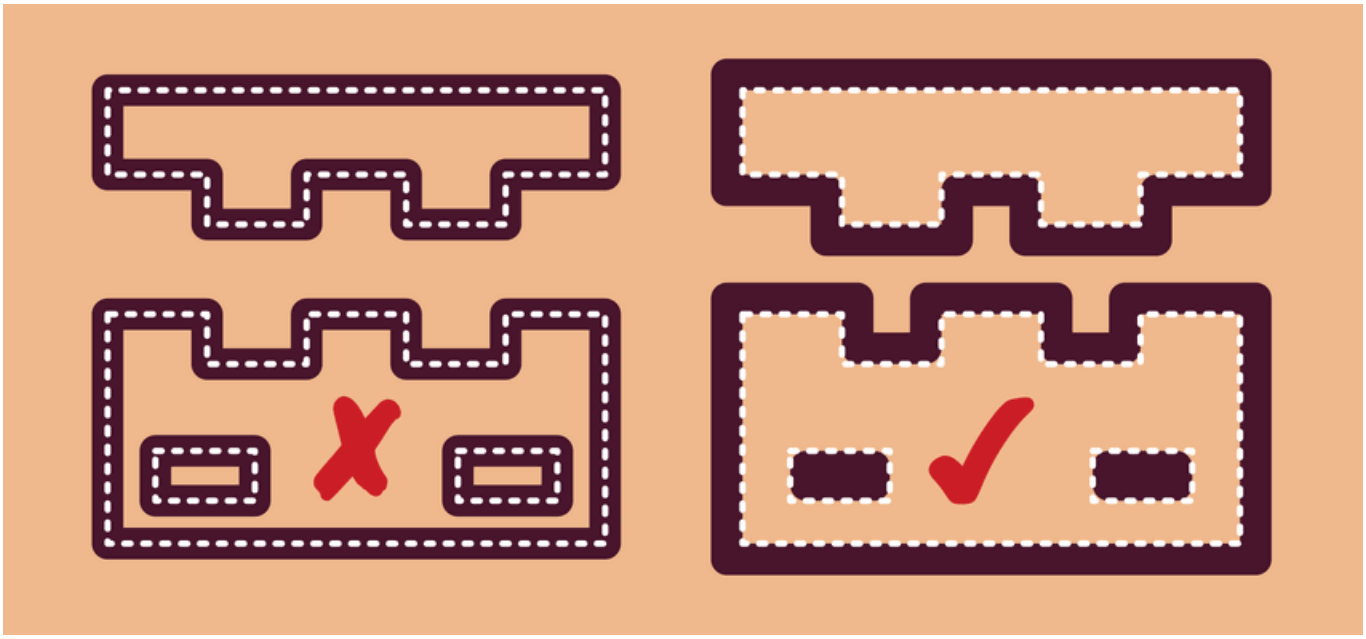
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=BKWHWXi6Wz0

Well-managed **Kerf** is the key to perfectly-fitted parts such as intricate marquetry or secure finger-joints in boxes.

Kerf describes the material removed by a tool as it works. In woodworking it's the thickness of a saw, but in laser cutting it's the gap of material removed by the beam. The width of this gap — its Kerf — varies based on the material being cut, the thickness of that material, and the Settings used to cut, such as speed and power.



When Kerf is not taken into account, and a beam is run along the center of a path, it removes material from either side of that line, and so the resulting parts are undersized (or in the case of slots, oversized). However, if you offset the path to compensate for the Kerf, perfect dimensional-accuracy can be achieved. This prevents your pieces from fitting together either too tightly or too loosely.

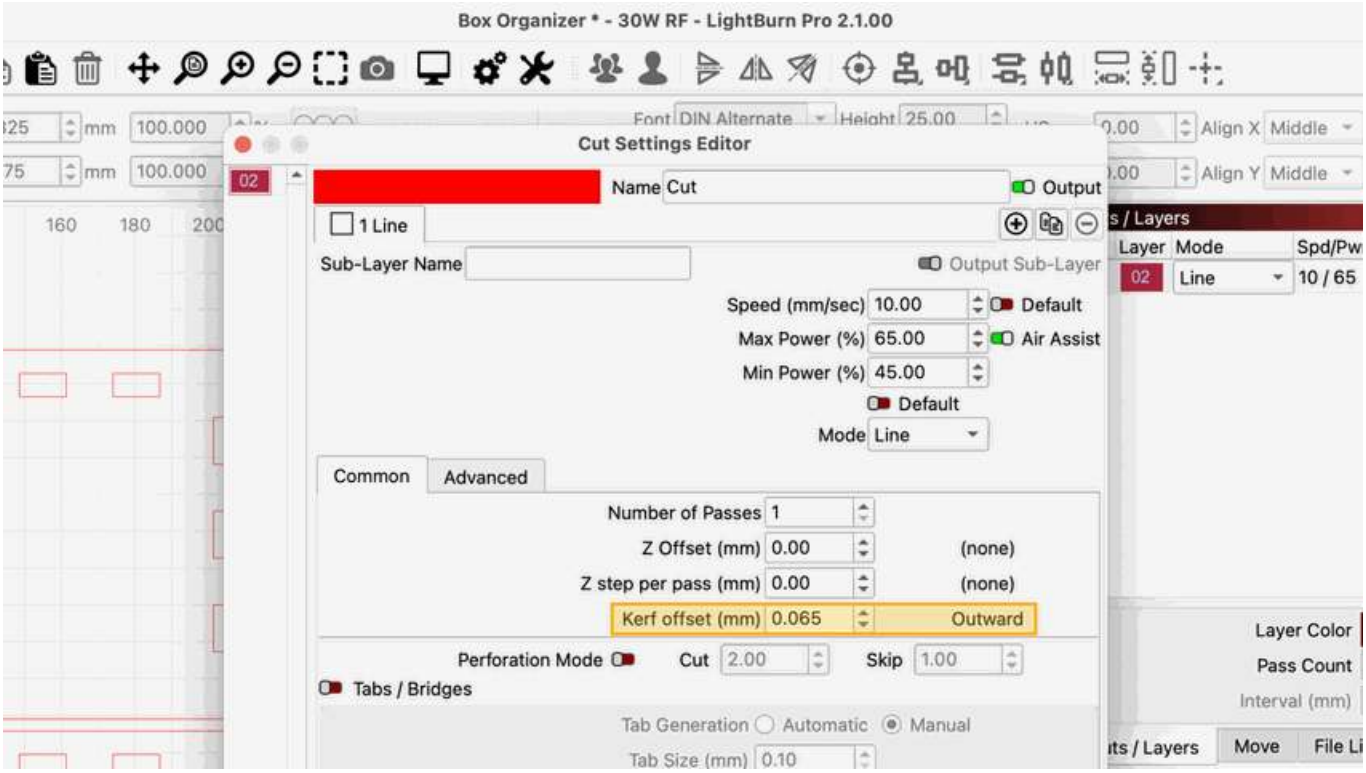


Applying a Kerf Offset in LightBurn

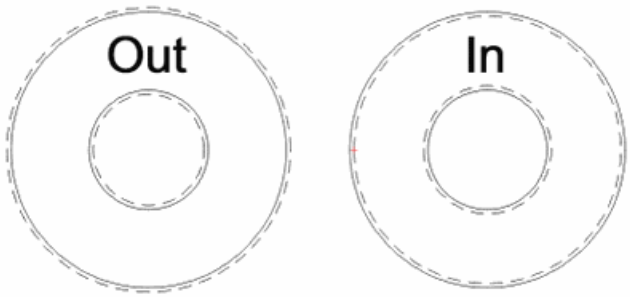
> **Closed Shapes Only**
>
> This feature only works with shapes that are closed. A shape is closed when it is a complete, continuous loop whose start and end points are the same. When a shape's start and end points are different, it is open.
>
> See Open vs. Closed Shapes for more information.

LightBurn makes it easy to apply the correct offset for a particular material through the Kerf Offset setting. You can find this option in the Cut Settings Editor, for layers set to Line Mode.

A correct Kerf Offset expands or shrinks the border so that the edge of the cut matches the edge of the art. Positive Kerf Offset values expand the border, and negative values shrink it — you'll see **Inward** or **Outward** appear next to the Kerf Offset field to let you know in which direction your value is adjusting the border.



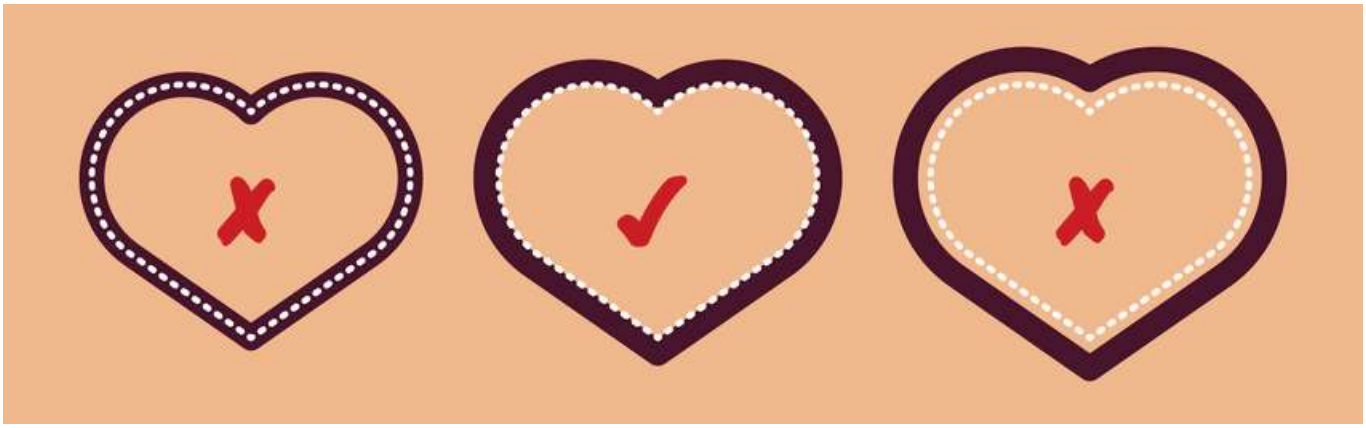
The Kerf Offset will be applied to all closed shapes assigned to the layer. Shapes that lie within other shapes on the same layer will have the offset direction inverted, to ensure that holes become the correct size.



Kerf Offset doesn't affect the actual geometry of the art — the offset is only applied to the code that gets sent to the machine and shown in the Preview window. This means you can re-use your art with different materials or different lasers by applying different Kerf Offsets, without changing the geometry of your design.

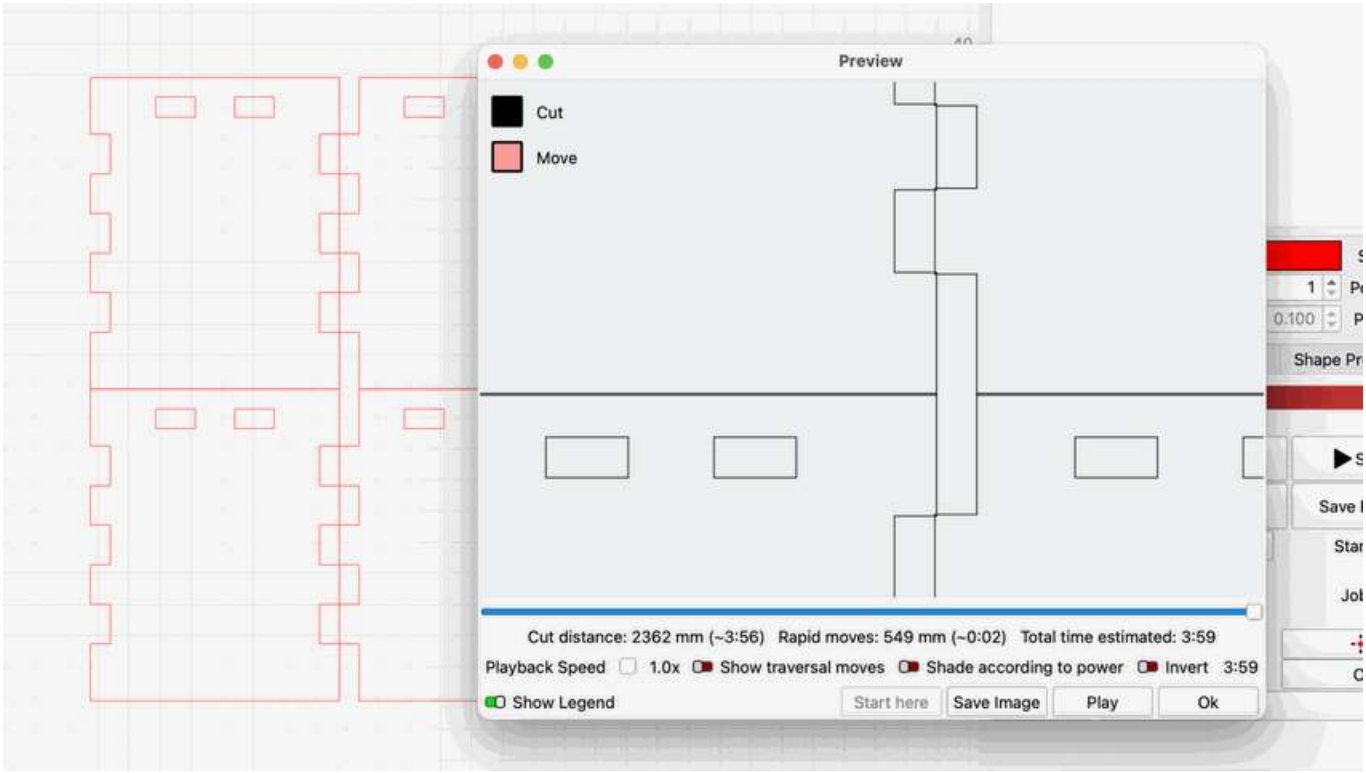
Kerf Offset is different from scaling a design, as scaling increases the size over the whole object, not just at the edges where the material is being lost to the Kerf. Scaling an object usually results in pieces that do not fit together, hence the need for an offset.

It's possible to overcompensate for Kerf by increasing the Kerf Offset value too far. This results in pieces that are too tight to fit together, so it's important to find the "sweet spot" for Kerf Offset.



Because Kerf Offset takes more space, it's important to leave room between the parts.

Kerf Offsets are shown in the Preview window, so it's a good idea to check it before sending the job to the machine.



These shapes are set too close together, and the cut lines will overlap once the Kerf Offset is applied.

Finding the Correct Kerf Offset Value

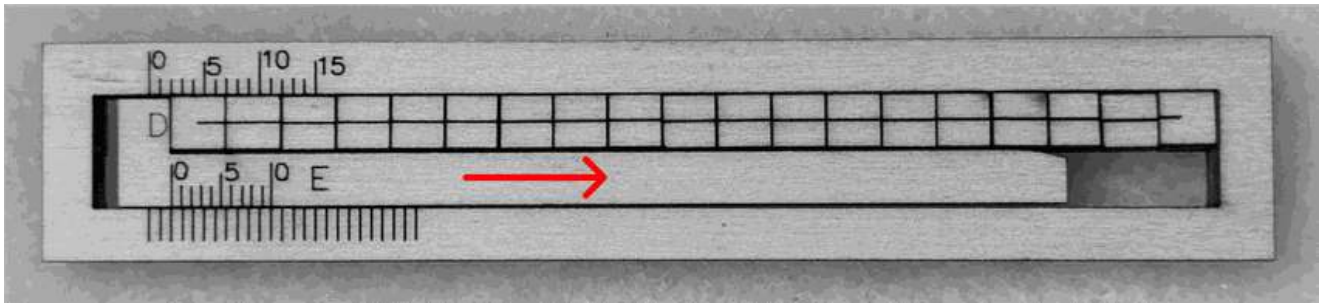
Some people try to measure the size of their pieces and compare them to the desired size to find their **Kerf Offset**, but the differences are usually a fraction of a millimeter, and even with calipers measuring this accurately is difficult.

We have created a tool to help you calculate the proper Kerf Offset value much more reliably and easily:

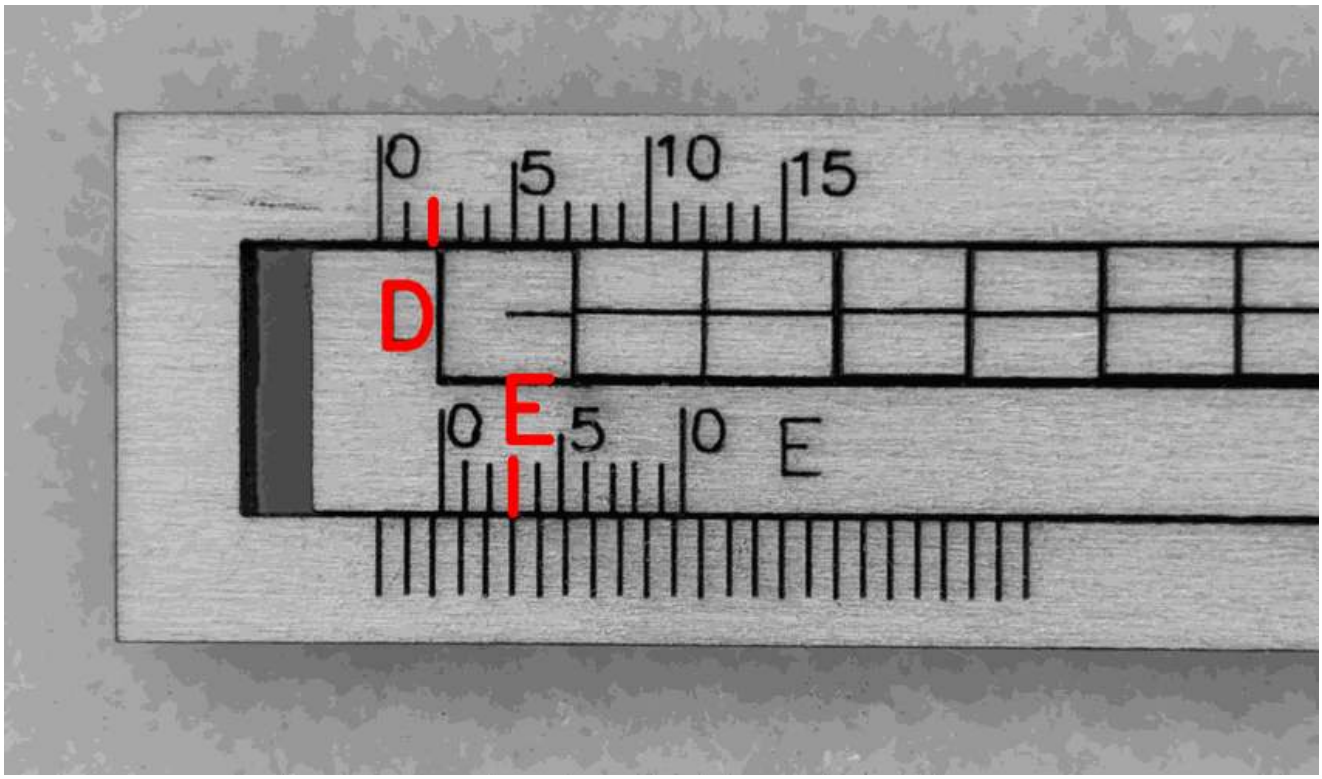
Download Kerf Test

Instructions

1. Set up the Cut (red) and Score (black) layers with your usual settings, and ensure the **Kerf Offset** is set to 0 mm (off) in the Cut Settings Editor. Position and run the Vernier Kerf Offset Test on your machine as normal.
2. Remove all pieces from the machine (a piece of painter's tape can assist), and remove the piece marked DISCARD.
3. Slide all loose bits to the right →. The horizontal black line can be used to orient the rectangular pieces if they get scattered.

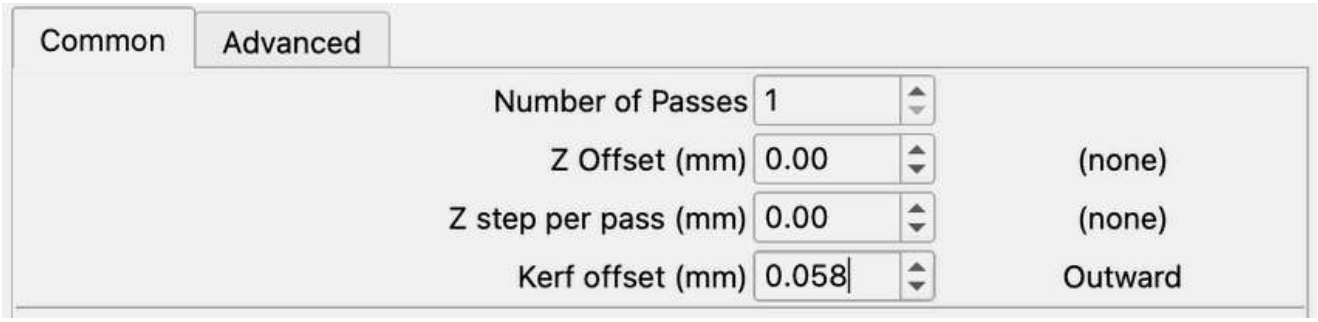


4. Find the largest number that the line marked D passes on the top scale, and use it as the whole number (D) in our equation (2 in the example).
5. Find which line on the bottom (Vernier) scale best lines up with any line on the bottom border (3 in our example). This will give us the decimal place (E) in our equation.



6. Fill in the equation, calculate it, and use the result as your Kerf Offset in the Cut Settings Editor.

```
D . E        2 . 3
-----   =    -----   =  0.0575 mm
  40           40
```



7. Optional: You may then use the box joints included to test your new Kerf Offset.

Once you have found your Kerf Offset, it's a good idea to save the settings to your Material Library, and the test itself to the Art Library, for future use.

Troubleshooting

Dealing With Irregular Beam Shapes

Some laser sources have beams that are not circular. For example, some diodes have a rectangular-shaped dot that result in a different Kerf (and power) on the vertical and horizontal axes. There are a couple of ways to manage this:

- Find the axis with the larger kerf and use that value, knowing some areas will be loose.
- Find the Kerf with the test turned on a 45 degree angle, and then cut all parts on a 45 degree angle, which will give more consistent results but limit the size of objects cut.



Adjusting Tabs For Different Thicknesses of Material

Kerf Offset is not the right tool for changing the height of tabs and adjusting slots to suit different thicknesses of material. See the Tab and Slot Resizer for that purpose.

Engraving Full Wrap Tumblers

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=LsHBv3QCD0c

Whether you're looking to make a personalized gift for a friend, add a new product to your lineup, or just do something fun with your laser, full wrap tumbler engraving is a rewarding technique to master.

Full wraps utilize a design that goes all the way around the cup, so you can’t tell where it begins and ends. They can be simple or complex, tightly grouped or loose, but they will all require similar steps to ensure they come out the way you expect.

Learn how to start tackling this popular design style and make it yours.

What You'll Learn

- Important considerations when choosing a tumbler to engrave on
- Things to think about when selecting a full wrap design, or creating your own
- How to create a template and full wrap design using the following tools:
  - Tool Layers, to make a template of your tumbler in your LightBurn Workspace
  - Grid Array, to duplicate a design until it fits the circumference of your tumbler
- Configuring your rotary in LightBurn
- Combining Tool Layers, Cut Selected Graphics, and Framing to position your engraving around a pre-existing logo

What You'll Need

- A laser with a rotary axis
- A rotary (Chuck style recommended)

> **What's a Chuck style rotary and why is it recommended?**
>
> There are two styles of rotary: **Chuck** and **Roller**. Chuck rotaries have adjustable jaws that grab and hold objects, while Roller rotaries have revolving rollers or wheels that objects are placed on top of.
>
> Chuck rotaries are able to grasp objects very firmly, greatly reducing the risk of slippage, and allowing you to rewind and fix mistakes while guaranteeing consistent placement.
>
> You can complete full wraps using a Roller rotary, but you need to take care to keep the tumbler from slipping on the rollers, and it's more challenging to rewind and go over mistakes.
>
> Wrapping the rollers with a non-slip material, or placing weights inside the object can help reduce slippage.

- A powder-coated tumbler (or any surface you're sure your laser can engrave on)
- A full wrap design, either imported or made from scratch
- Seamstress' (flexible) tape or calipers
- A level
- Painter's tape (and lots of it)

Choosing a Tumbler

Your choice of tumbler affects the difficulty level of completing a full wrap engraving. We recommend starting with the easiest — a straight-sided tumbler — and that's what we'll use for the rest of this guide. Below we've outlined some considerations for more challenging tumblers as well.

Easy

Straight-sided tumblers are the easiest to work with, and what we recommend for beginners. With a straight-sided tumbler, you don't have to deal with multiple focus levels, work around any tapers, or compensate for distortions.



Intermediate

Tapered tumblers are somewhat more difficult. You'll need to ensure your laser is in focus at multiple distances from the cup's surface, and you may see some distortion of your graphics at the taper.

> **Tip**
>
> You can compensate for distortion caused by tapers using our Taper Warp tool.



Hard

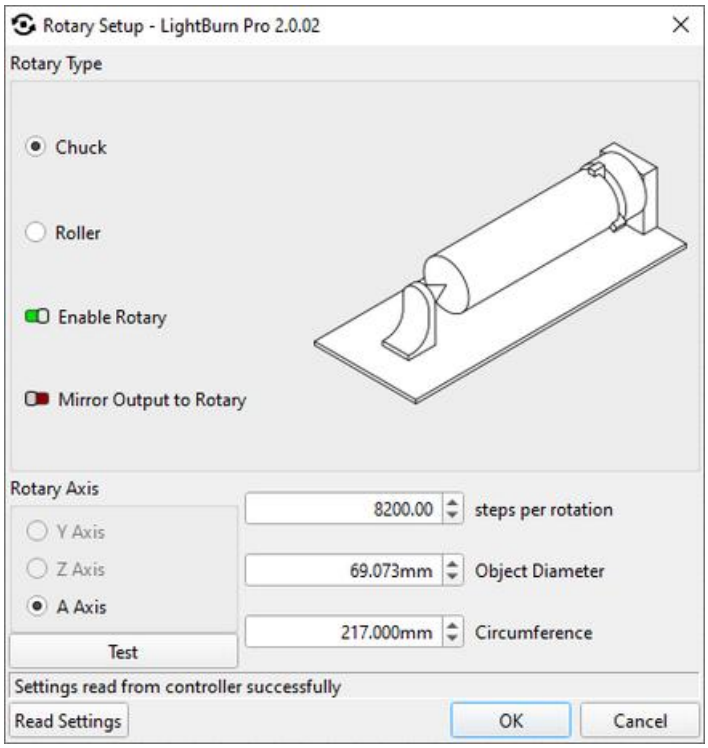
Tumblers with step downs are the most difficult to work with. The dramatic differences in circumference make it more challenging to get all your measurements and focus right, in addition to distorting your designs.



Configuring the Rotary

Before running your first rotary job you'll need to open the **Rotary Setup** window to establish some important values, and enable Rotary Mode.

1. Go to Laser Tools → Rotary Setup to open the Rotary Setup window.



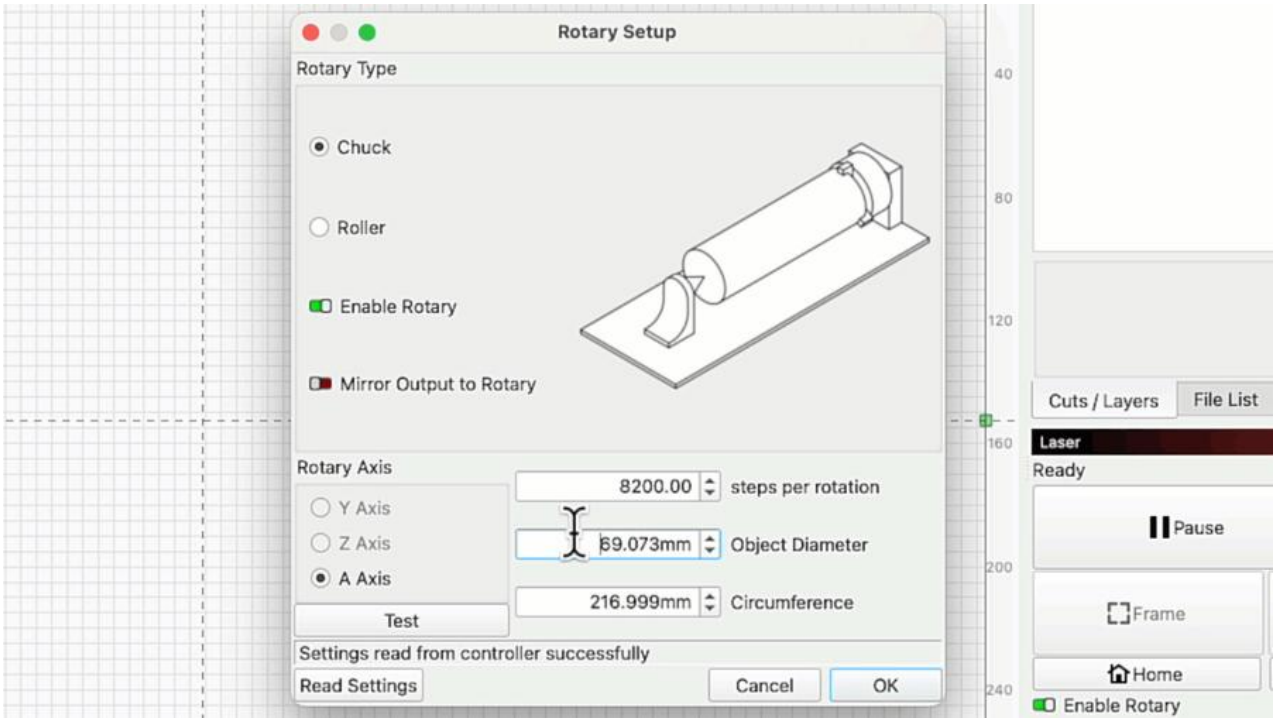
2. In the **steps per rotation** field (**mm per rotation** if you're using a GCode-based device), enter the number of motor steps required to turn your rotary 360°.
   - In most cases, this value will be provided by the manufacturer of your rotary or machine. If you can't find it in their public resources, we recommend reaching out to them for assistance.
   - If you're still not sure, you can use the **Test** button to find the correct value. When it's correct, pressing the Test button will cause your rotary to make one complete rotation, then return to the start.
   - Once this value is set correctly, you never need to change it again.
   - See Rotary Mode for more information on configuring a rotary with your model of laser.
   - If you're using a Galvo laser, you'll see additional options to control **Split Size** and **Overlap**. See Rotary Mode (Galvo) — Split Setup for information on these settings.
3. If you're working with a **Chuck** style rotary, you will also need to enter the Circumference of the object you're placing in your rotary, to ensure your laser's output is scaled properly.

> **Note for Roller style rotaries**
>
> For **Roller** style rotaries, Circumference does not affect output scaling, but you'll still need to know this value when you set up your full wrap design.

> You will also need to accurately enter the diameter of your rollers. This value is usually provided by your rotary's manufacturer, but you can measure it with high-precision calipers if not.

1. Use seamstress' tape to wrap around the tumbler and measure its circumference, then enter it in the Rotary Setup window.





2. If you're using calipers, measure the tumbler's diameter, and enter it in the Object Diameter field. LightBurn will automatically calculate the Circumference based on the diameter.

4. Make sure the **Enable Rotary** switch is toggled on, so LightBurn will tell your laser to use its rotary axis, and scale output appropriately.

Creating the Design

Next we'll explain how to create a template for your tumbler, go over some important considerations to keep in mind when you're selecting or creating a design, and show how you can use Grid Array to duplicate a design within the template to make a full wrap.

Making a Template

Before laying out our design, we'll create a template to represent the engraveable area of our tumbler.

1. Measure your tumbler. In addition to the circumference measurement you took earlier (or derived from the diameter), you'll need to know the length of the cup.
2. Use the Rectangle tool to create a rectangle of any size.

Quick Reference: Draw Shapes



Create and edit rectangles, ellipses, polygons, or stars to build designs out of vector objects.

More Info → Primary Shapes

Location

Creation Toolbar

Tools → Primary Shapes

Keyboard Shortcuts

Rectangle/Square

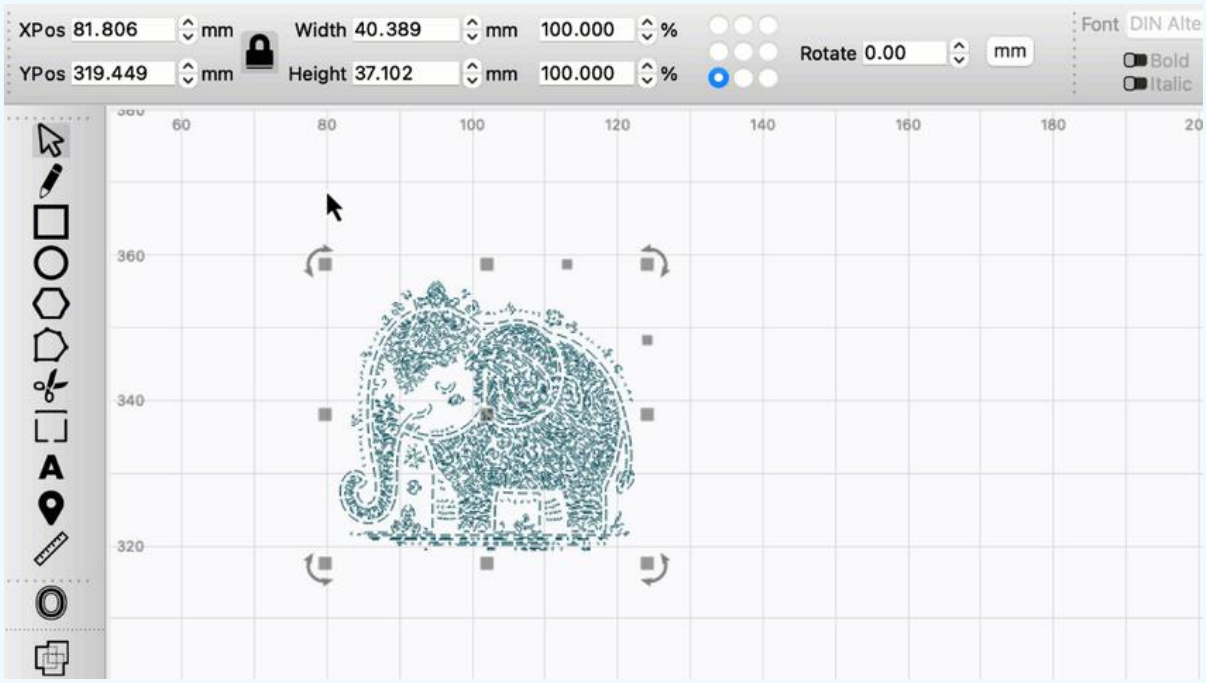
`Ctrl` / `Cmd` + `R`

Ellipse/Circle

`Ctrl` / `Cmd` + `E`

3. In the Numeric Edits Toolbar, set one of the rectangle's sides to match the cup's length, and the other to match the cup's circumference.

Quick Reference: Numeric Edits Toolbar



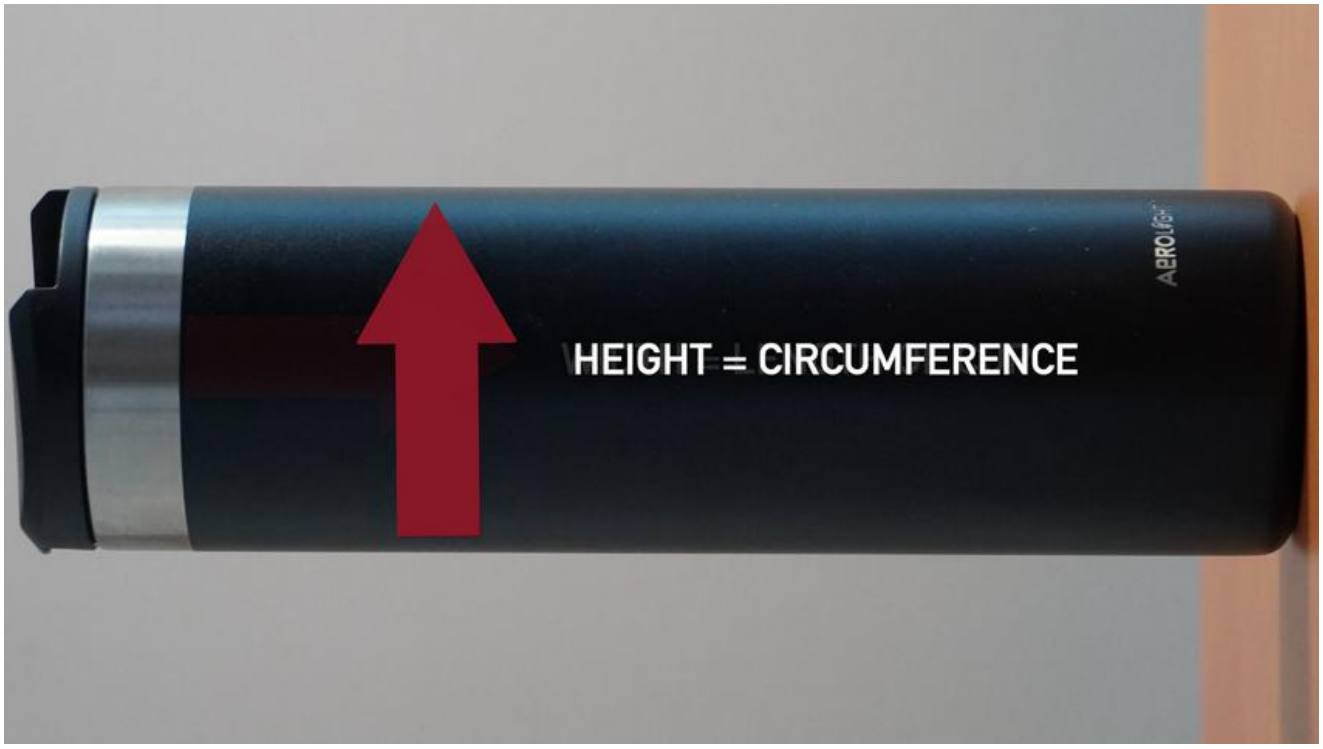
Use the **Numeric Edits Toolbar** to adjust the size, position, and orientation of shapes in your selection using precise numeric values.

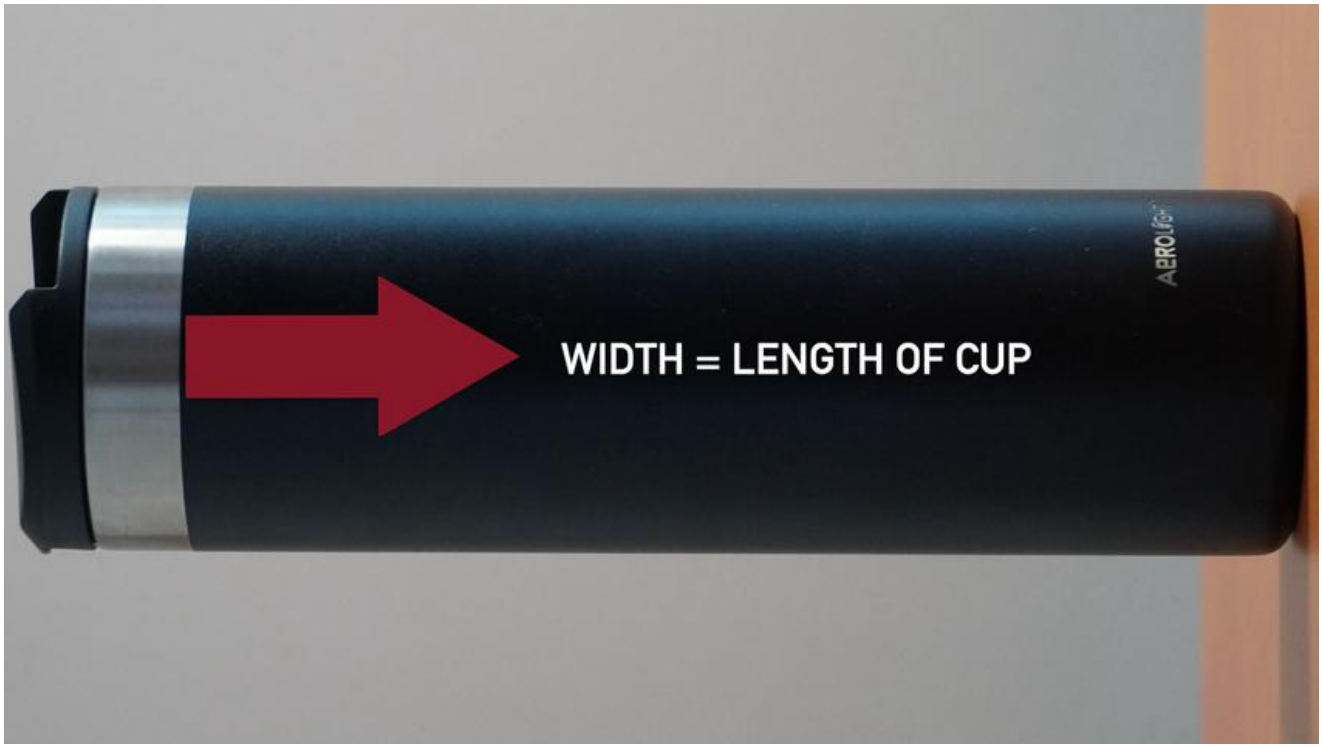
More Info → Numeric Edits Toolbar

Note for Galvo Users

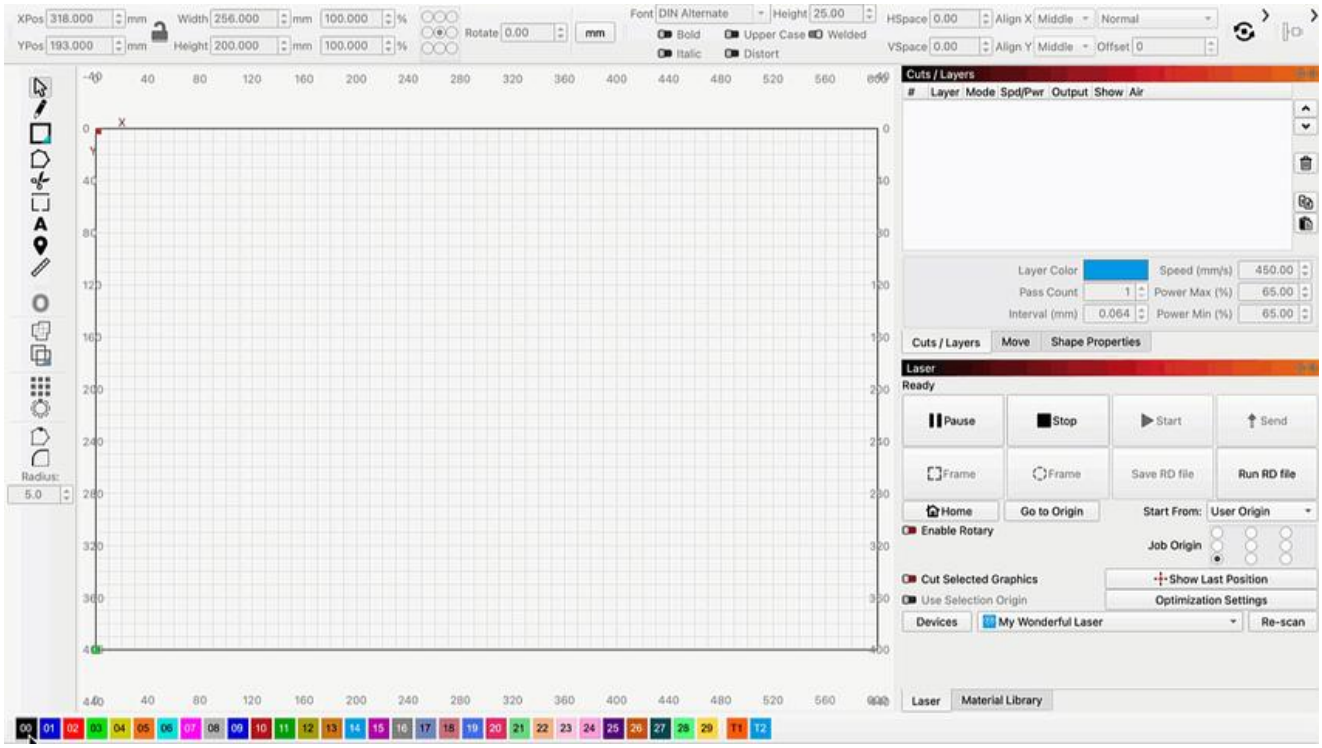
If you're using a Galvo laser, the correct orientation depends on whether you have set your rotary to the X or Y axis. See Rotary Mode (Galvo) — Rotary Axis for more information.

You can orient the rectangle however you'd like while you design, but remember which side is which, and, if necessary, take care to rotate it 90 degrees later, before sending the job to your laser. In the final orientation, the rectangle's Height should match the cup's circumference, and its **Width** should match the cup's length.





4. Set the rectangle to a Tool Layer. Tool Layers are special layers that are never sent to your laser, so this rectangle will now serve as a guideline only, with no risk of accidentally engraving it onto your tumbler.

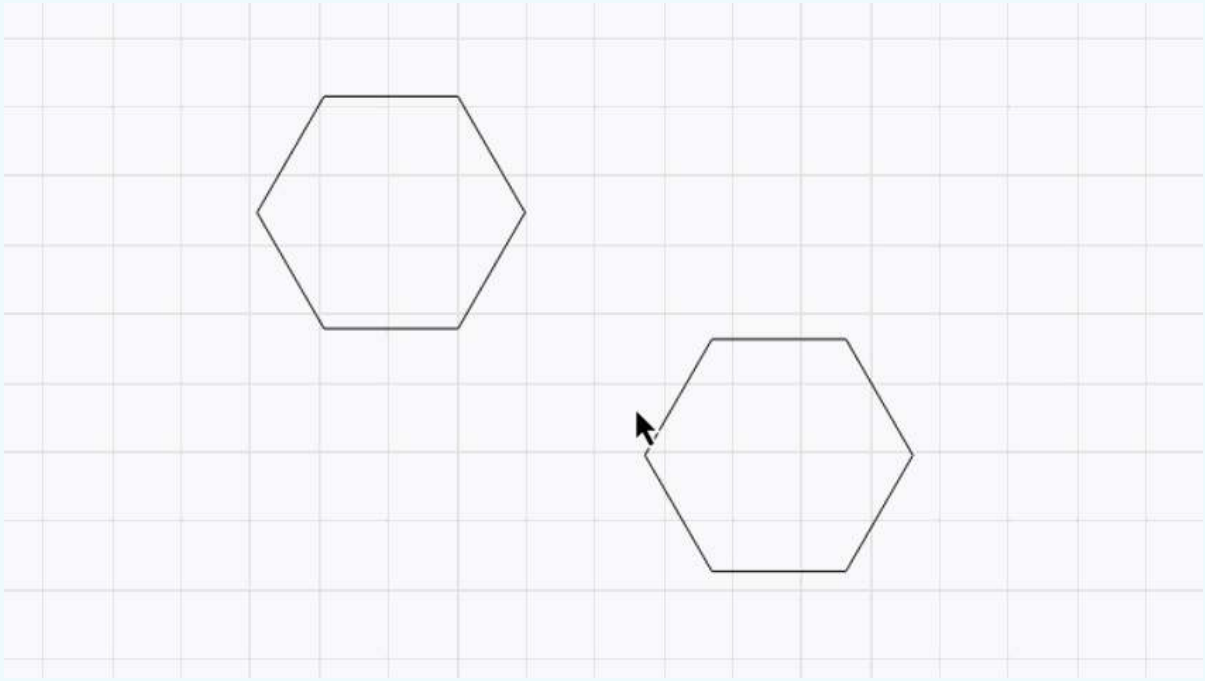


5. If your tumbler has a logo you'd like to work your design around, get a rough measurement of its dimensions, and its distance from the top of your cup.

> **Tip**
>
> If you're using a CO_2 laser and your tumbler has a logo that you don't want to work your design around, you can cover it with aluminum tape instead. CO_2 lasers can't penetrate aluminum.

6. Create another rectangle, also set to a Tool Layer, and set its dimensions to match the measurement of the logo — making sure that the Height and Width values are entered correctly for your template's current orientation.
7. With the Selection tool active, hover over the side of the logo outline that is oriented parallel to the side of the template representing the cup's circumference, until you see the midpoint Snapping icon appear.

> **Quick Reference: Snapping**



LightBurn's automatic **Snapping** tools help align objects to other shapes in your Workspace, or ..to the Workspace grid itself — when you move objects close to a valid location, they will snap to that point.

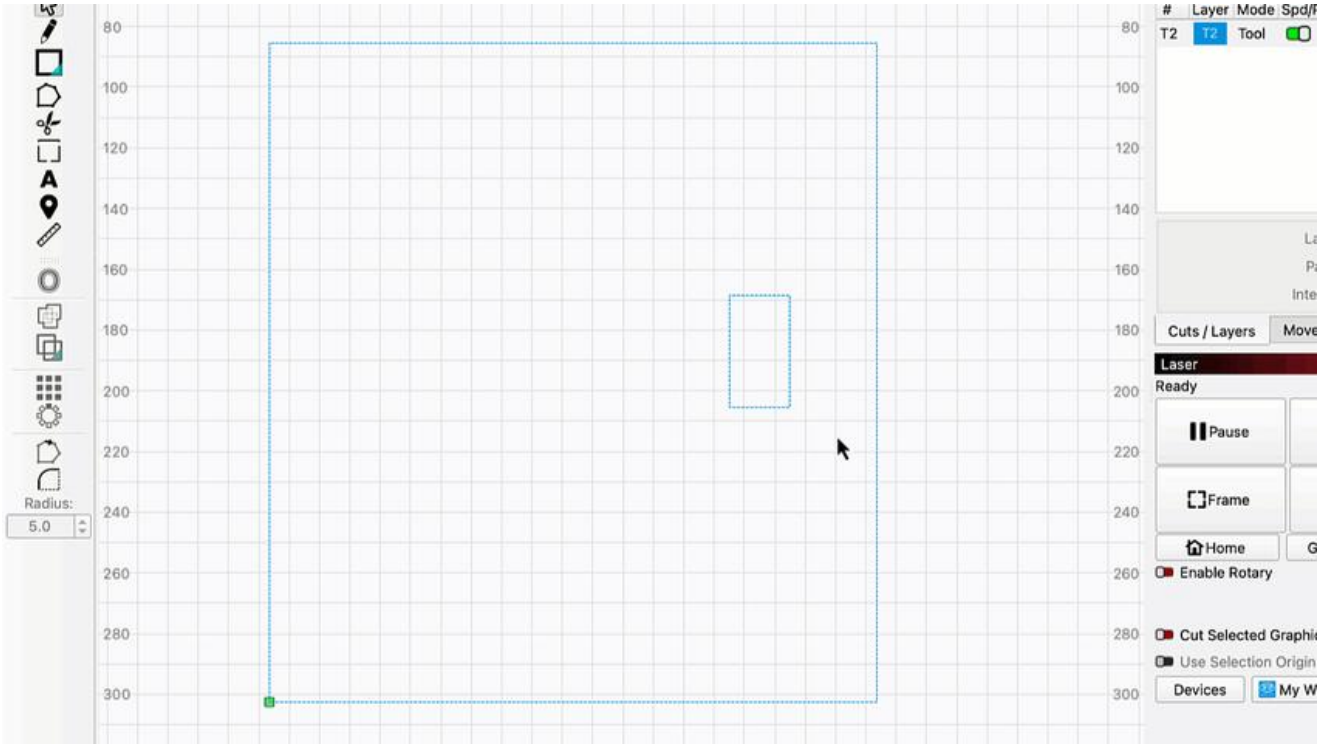
As you hover your cursor over an object, the icon will change to indicate when it is over a node, line, midpoint, center, or intersection. Use these Snapping points when selecting, moving, or creating objects to ensure exact alignment.

More Info → Snapping

Object Snapping Points

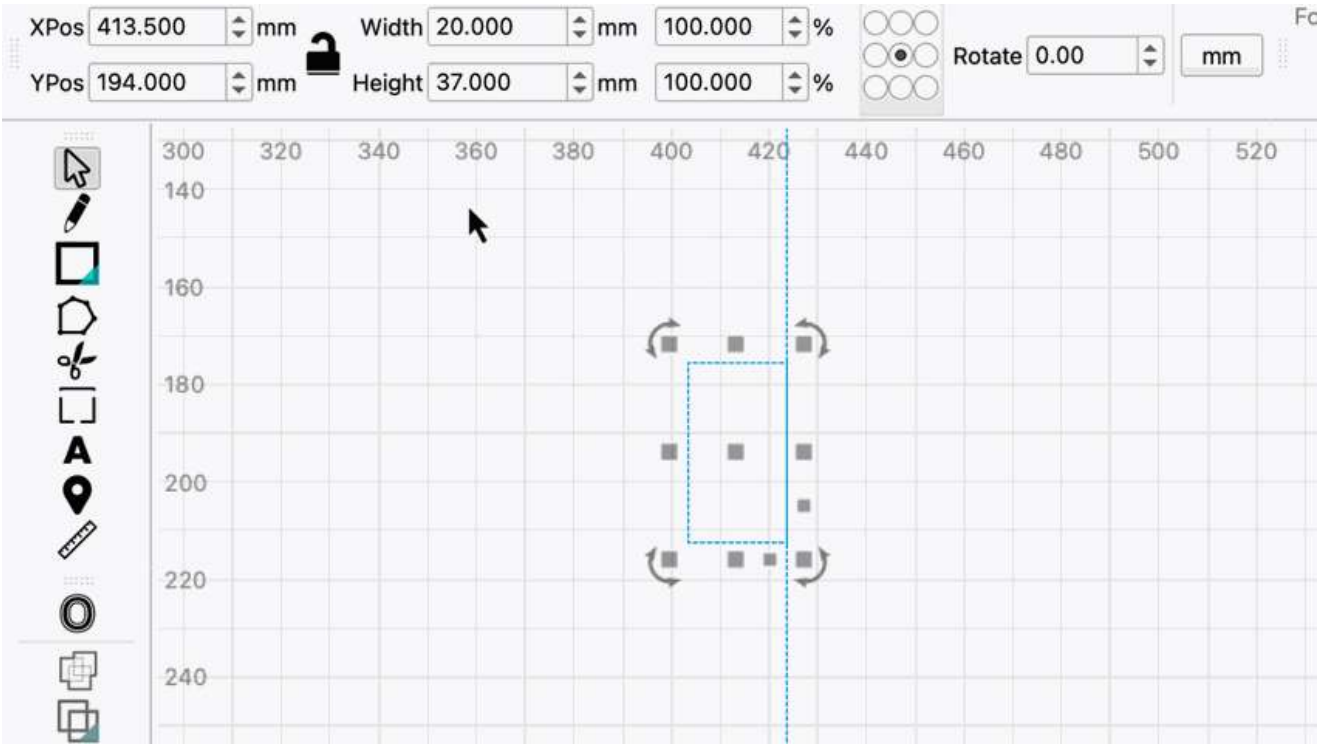
Snapping Point	Icon	Description
Node		The cursor is over any node in a vector graphic, or the corner of an image.
Midpoint		The cursor is over the exact halfway point between two nodes.
Center		The cursor is over the center point of an object.
Intersection		The cursor is over the intersection of two lines.
Line		The cursor is over a line, but not a node, midpoint, or intersection on that line.

8. Click and drag the logo outline from that point, until you see the midpoint snapping icon appear again on the side of the larger template, then release to place the outline there.



9. In the Numeric Edits Toolbar, adjust the X Position or Y Position of the outline to move it as far in from the side of the template as the logo is from the end of the cup.
    1. Without overwriting the existing value, use the `+` or `-` symbols to add or subtract the distance you took from edge to logo. Press `Enter` once you've input the symbol and value.

       Whether you need to add or subtract depends on the position of the logo outline relative to the edge, and where your machine's origin is.
    2. If the logo outline moves the wrong way, press `Ctrl`+`Z` to undo the move, then use the opposite math operation.

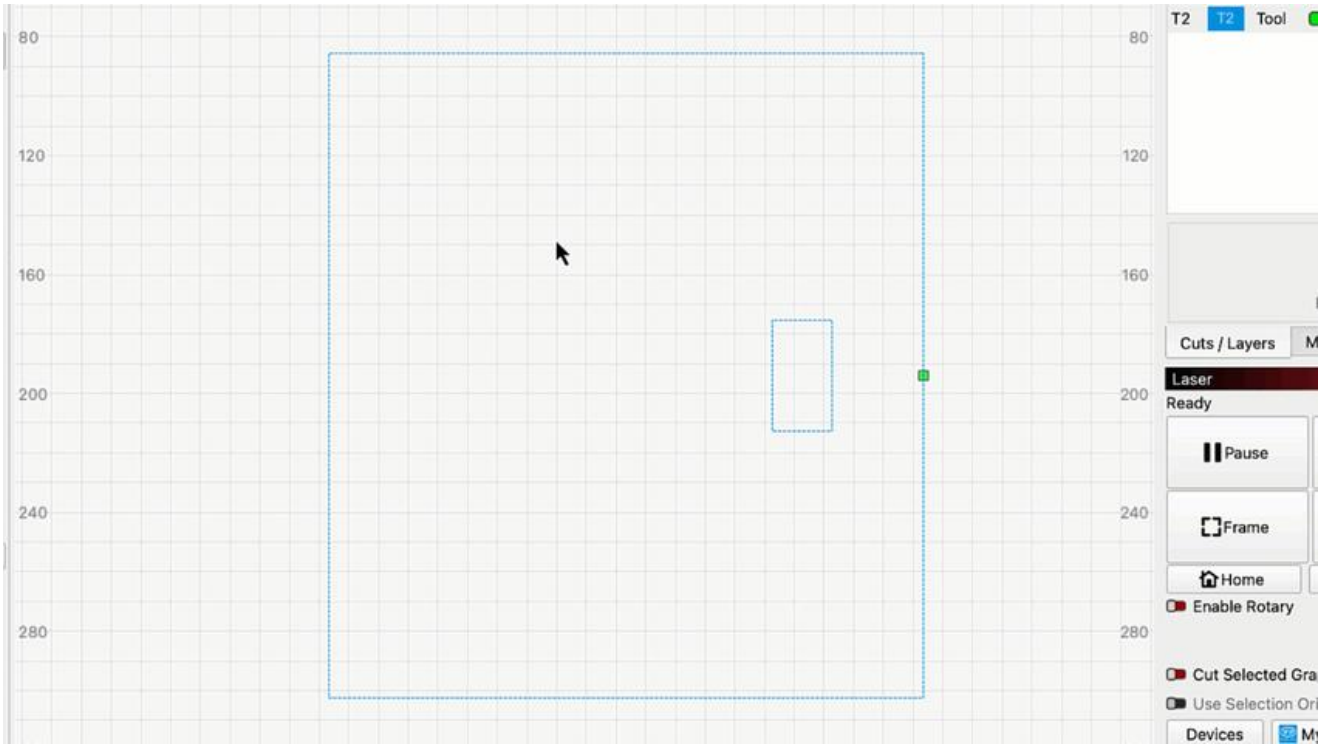


Testing the Template

Before moving forward, it's a good idea to test the template to make sure it is sized correctly, and the logo outline is positioned appropriately.

1. Create a thin rectangle that spans the length of your template in the dimension matching your cup's circumference.

   You can use Snapping to make sure the rectangle starts at exactly one side and ends at exactly the other.



2. Set the layer the thin rectangle is on to appropriate **Speed** and **Power** settings to lightly etch the surface of the painter's tape, without going through.

   The precise settings for that will depend on your laser, but we recommend starting with a very low power to begin with. If it's too low, you can always increase and test again. But if it's too high, you'll burn through the painter's tape and might waste a tumbler.

> **What cutting/engraving settings should I use?**
>
> The exact settings required to achieve good engravings, or to cut through material, are highly dependent on your type of laser.
>
> While we can provide some rough guidelines, the best source for specific recommendations is usually your laser's manufacturer.
>
> > **Always pay attention to units of distance and time when entering Speed settings**
> >
> > When entering **Speed** values recommended by your laser's manufacturer or other LightBurn users, make sure to use the same units of distance and time as those from the recommendation, or to convert the values to your preferred units.
> >
> > A given number of **millimeters per second** is much faster than that same number in **millimeters per minute**. Mixing up units can lead to reduction in power output due to unexpectedly high speeds, or excessive power output — and even fire — due to unexpectedly slow speeds.

> Change your Control Units in the **Dimensions / Units** tab of the **Device Settings** window. LightBurn automatically converts any existing values when you switch between units.

The two primary settings controlling the depth of a cut or the darkness of an engraving are power and speed.

Power controls the laser's intensity. Higher powers allow you to cut deeper or make darker marks for engravings, but at the cost of potentially charring or melting edges, producing excessive smoke, and making a wider kerf.

Speed controls how fast the laser moves. Lower speeds will deliver more power and heat to the material, as the laser takes longer to pass over any one point. Higher speeds can complete a job faster, but will require higher power. High speeds can also exacerbate or make any existing mechanical issues more apparent.

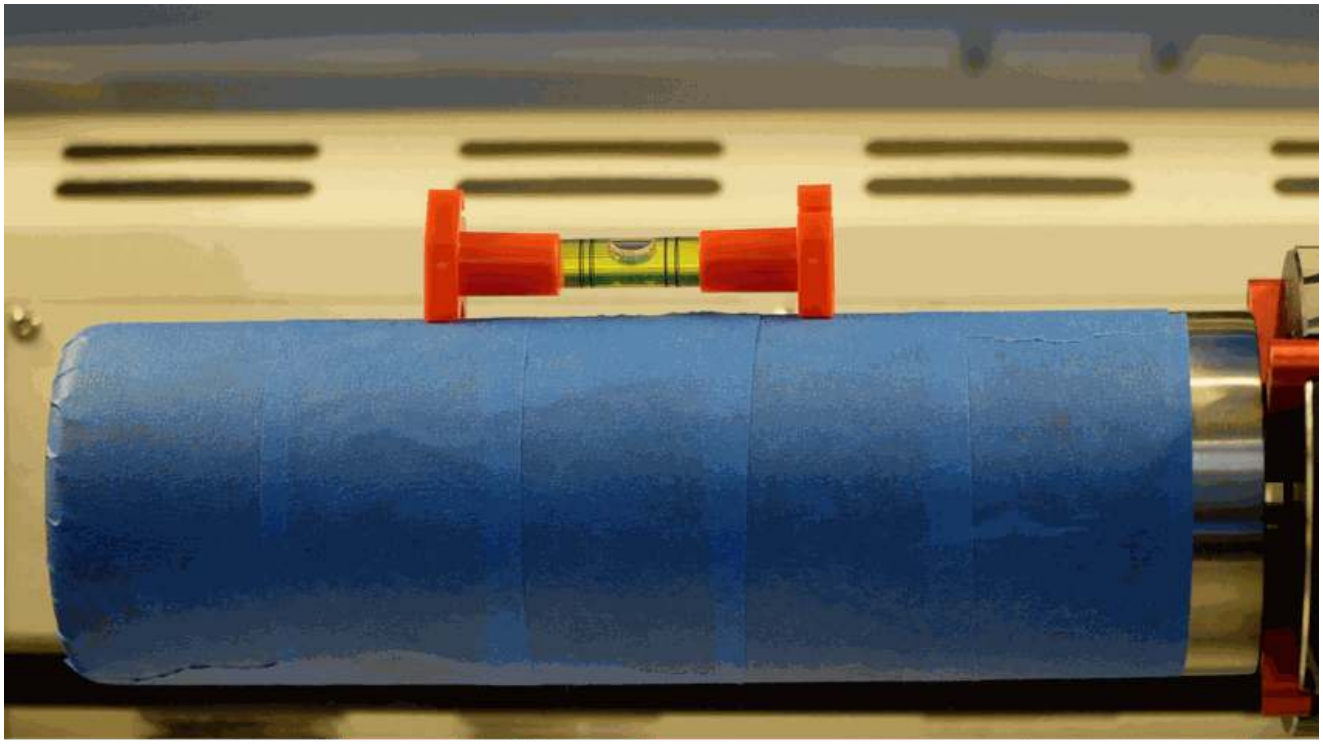
> **Info**
>
> For a deeper dive into the relationship between these settings, see Speed vs. Power.

There's another crucial setting for layers set to Fill Mode — **Line Interval**. The Interval setting determines the distance between the parallel lines your laser scans as it engraves your graphics. Smaller Line Intervals lead to lines that are closer together. As a rule, you want the scan lines to be as close together as possible without overlapping (which can lead to overburning).

While we've noted that manufacturer recommendations are the best place to begin looking for cut settings, LightBurn also has a built-in Material Test that you can use to test different combinations of **Speed**, **Power**, **Passes**, **Line Interval**, and other settings. The Material Test works by automatically setting up a grid of boxes that will be cut or engraved at a range of different settings.

See Material Test for more information.

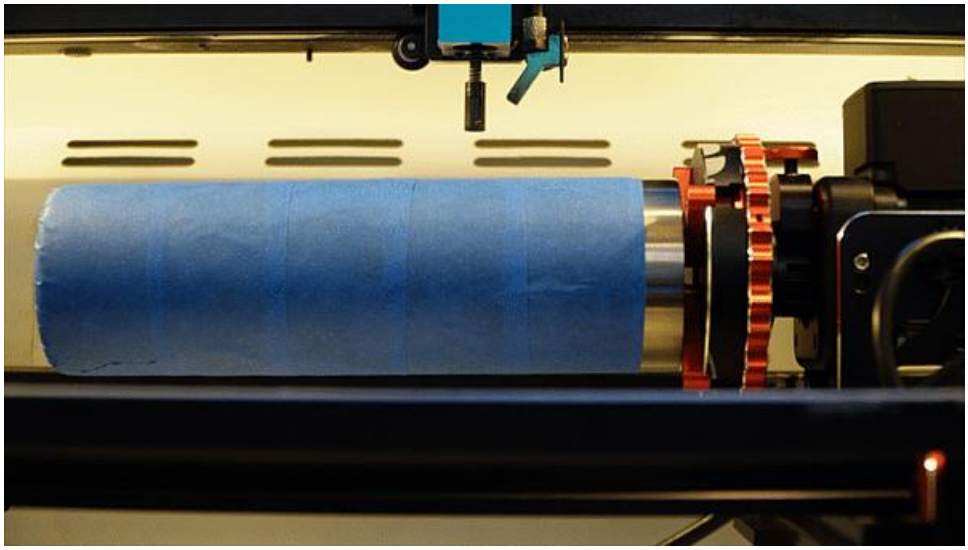
3. Completely wrap your tumbler with painter's tape, and place it on your rotary.
4. Make sure your laser is connected to LightBurn, and Rotary Mode is enabled.
5. To get a good engraving on your tumbler, it will need to be perfectly level on the rotary. Place the tumbler on the rotary, then a level on top of the tumbler, and adjust the angle of the rotary until it's precisely level.



6. Lasers need to be focused correctly to achieve good results — the space between the laser and the surface of your material must be set to precise distance. Focus the laser according to your laser's user guide — we used a DSP laser, with auto-focus capability.

Learn More: Focusing Methods by Laser Type

- GCode-based Diode lasers usually have a focusing spacer to set the distance between the laser and the material.
- DSP-type CO_2 lasers may require manual focusing with a focusing spacer or may support auto-focus, either through the controller, or with the Focus Z button.
- Galvo lasers usually require manual focusing. Many galvo lenses are marked with their optimal focal distance, and you can use a ruler to set the correct distance between the lens (or mirrors) and your material. Some are equipped with a two red dot focusing system — adjust the height of the scan head until the dots meet, then the laser is focused. Finally, see our guide on Galvo Laser Focusing for a manual method of determining optimal focus.



Note for Galvo Users

Steps 7 - 9 do not apply to Galvo lasers, which always work in Absolute Coordinates, and handle positioning differently. See Rotary Mode (Galvo) — Positioning Graphics for more information.

7. In the Laser Window, set the Start From mode to **User Origin**, and the **Job Origin** to either the center-right or center-left.

   With these options selected, the rectangle will be engraved to the left (for a center-right Job Origin) or the right (for center-left) of your custom origin, at the same distance from the edge of the cup as the edge of the template.

Learn More: User Origin

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=aphiLTUWcjE

User Origin works almost exactly the same as Current Position, except that the starting location is programmable. To set a User Origin, you must first jog your laser to the location you want your job to start from, then set the origin.
Some lasers with digital displays, such as those with Ruida controllers, have an `Origin` button which you press to set the origin. GCode-based systems use the **Set Origin** button in the Move Window in LightBurn to do the same thing.

For supported systems, use **Clear Origin** to remove and reset a custom-set origin.

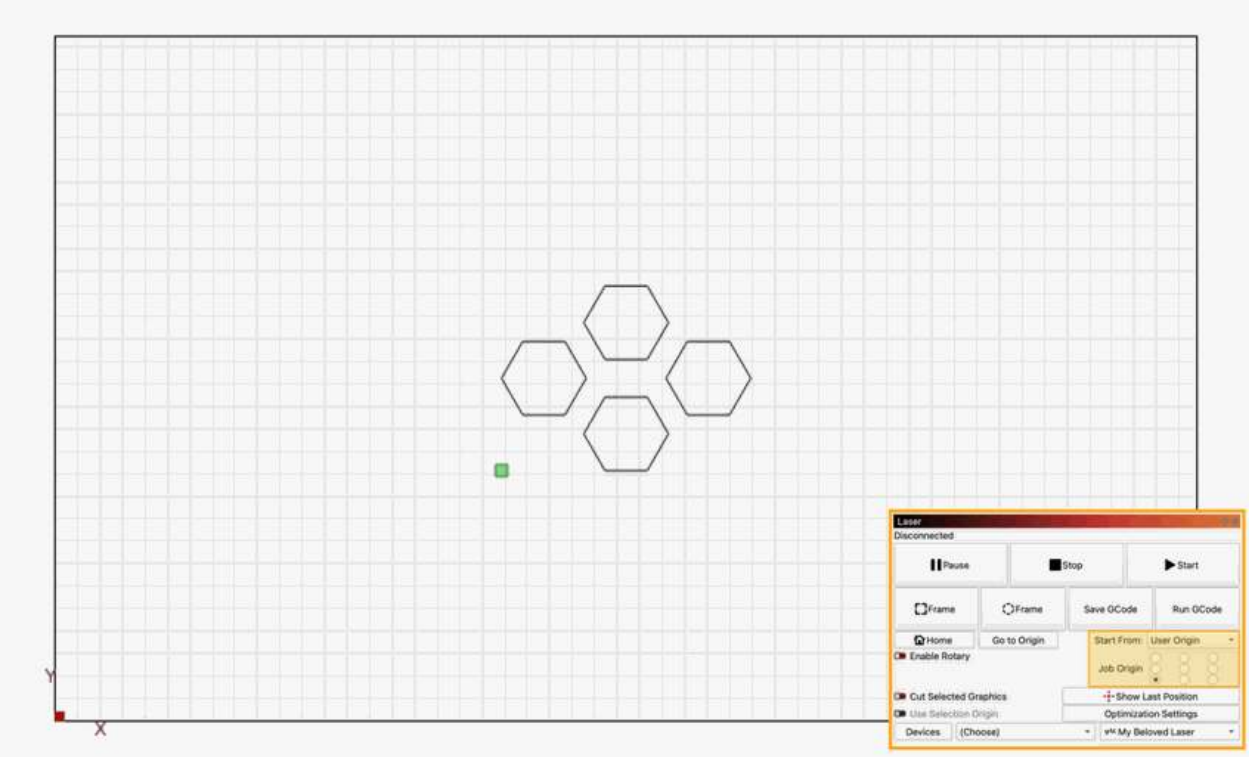


After setting the origin, you are free to jog your laser around its work area. With User Origin selected as the **Start From** mode, the laser will always move back to start the project in a position relative to the programmed location.

You can also use the **Go to Origin** button in the Laser Window to command your laser to travel back to the origin you've set.

As with Current Position, the **Job Origin** setting determines how output will be oriented relative to the user-defined origin.

Here, the Job Origin is set to the lower left:

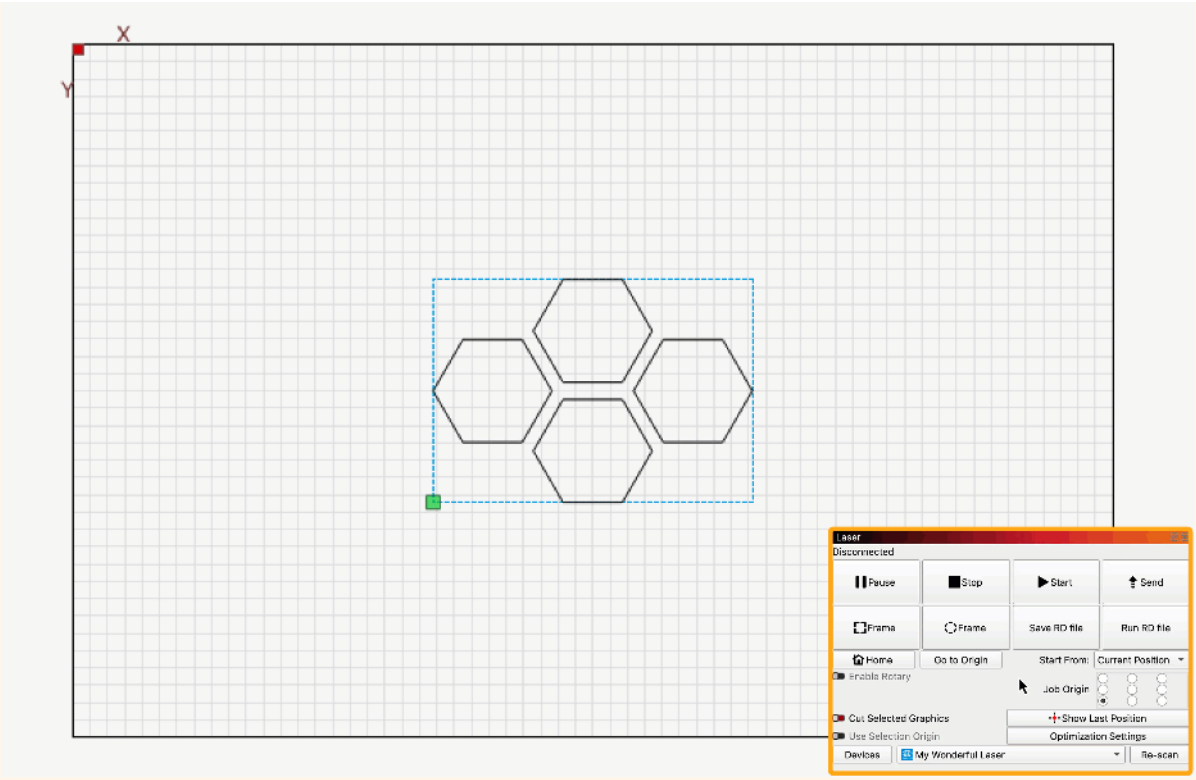


Notice the location of green Job Origin indicator. This represents whatever location you've set for your custom origin. When you start the job, the laser is going to move from wherever it is currently located and cut the four hexagons up and to the right from the programmed origin location.

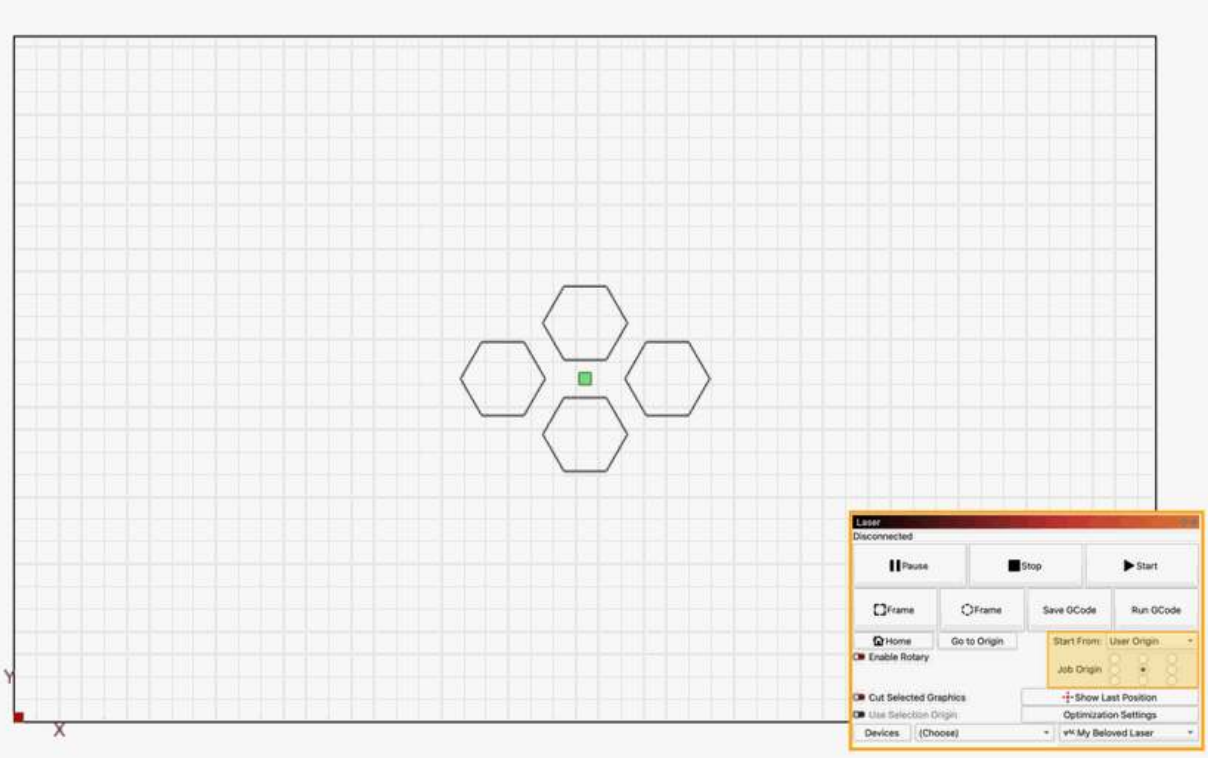
Why is the Job Origin offset from the graphics?

When using Current Position or User Origin, the green Job Origin indicator is always located on a corner, side, or at the center of an imaginary box — the smallest possible rectangle that can fully contain all graphics you're sending to the laser.

Below, we've added a Tool Layer to represent that imaginary box. Notice how the Job Origin moves around the box as we change the location in the 9-Dot selector.



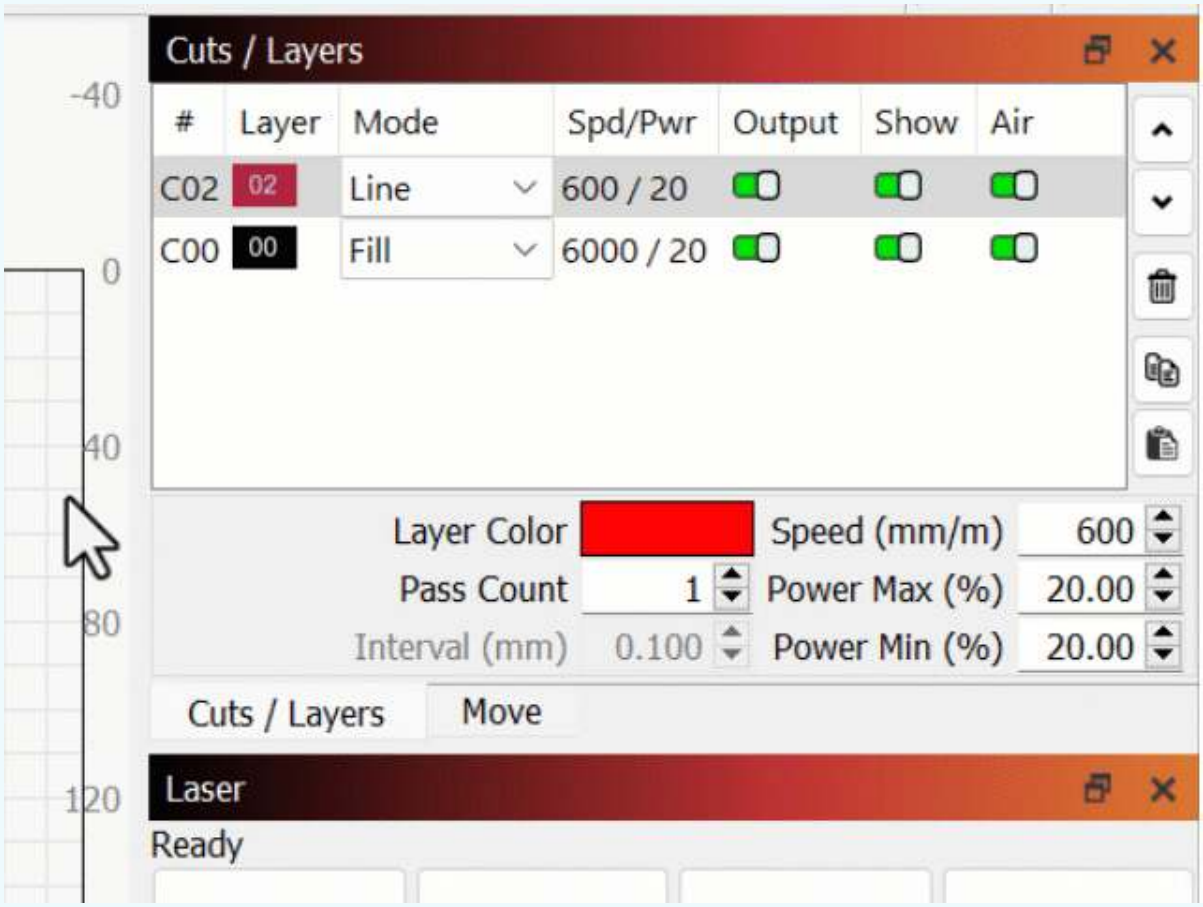
Now imagine that you want to etch this four hexagon pattern onto a coaster or a phone case. Lining it up from the lower left is not easy. If you change the Job Origin setting to center, you get this instead:



> The job will be centered around the custom origin you set. If you position the laser head directly over the center of the item you want to etch before setting the User Origin, the output will be centered on the item.

8. Jog your laser to edge of the engravable area of your tumbler. If you selected the center-right Job Origin position, jog it the right edge. If center-left, jog to the left edge.

   If your laser has a controller display and keypad, you can use the buttons there to jog. If not, use the buttons in the Move Window.

> **Quick Reference: Jogging**
>
> 
>

You can use the arrow keys in the Move Window to move your laser up, down, right, or left. You can set the amount the laser travels with each click in the **Distance** field, and the rate at which it travels in the **Speed** field.

If you have a DSP laser you can also jog your laser using buttons on its onboard controller.

More Info → Move Window

Location

- Move Window
- Onboard controller (DSP-only)

Keyboard Shortcuts

Each keyboard shortcut will move your laser by the amount entered in the **Distance** field, at the rate set in the **Speed** field.

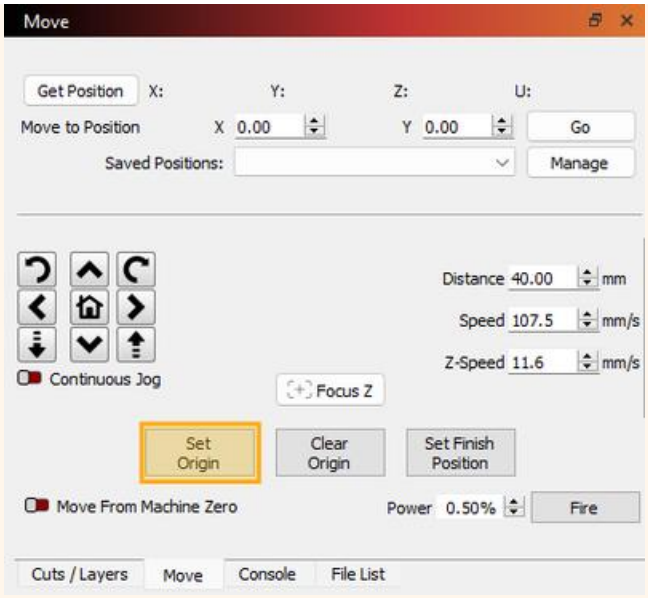
- **Jog Laser Left**: `Alt`/`Option` + `Ctrl`/`Cmd` + `[`
- **Jog Laser Right**: `Alt`/`Option` + `Ctrl`/`Cmd` + `]`
- **Jog Laser Up**: `Shift` `Ctrl`/`Cmd` + `]`
- **Jog Laser Down**: `Shift` `Ctrl`/`Cmd` + `[`

9. Press the **Set Origin** button in the Move Window, or the `Origin` button on your machine's display, if your laser has a Ruida controller.

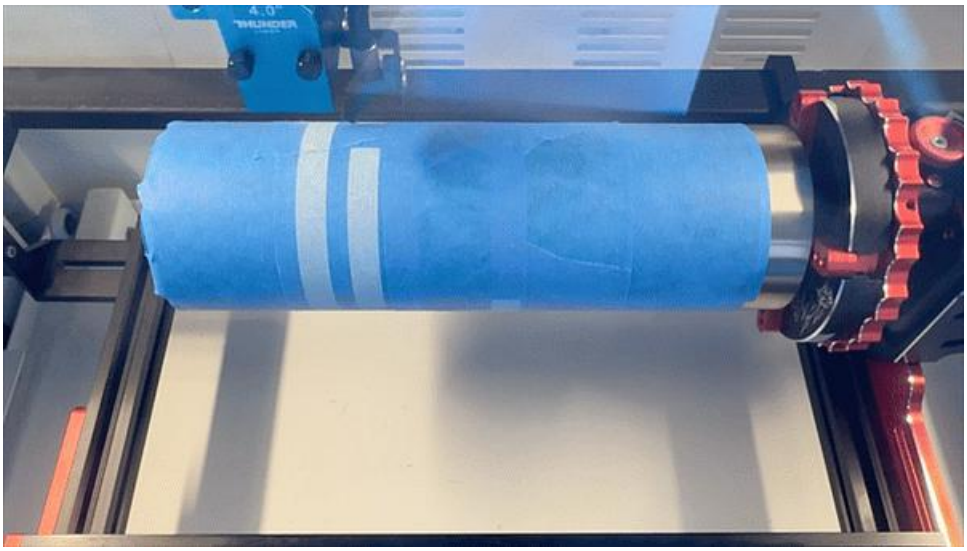
Where's the Set Origin button?

The **Set Origin** button is presented in the Move Window, which is docked behind the Cuts / Layers Window by default. If you don't see it, go to Window → **Move** to re-enable.

This button is only available if you are using a GCode-based laser. If you're not sure what kind of laser you have, see Identifying Your Laser.



10. Press the **Start** button in the Laser Window to run the job.



11. If your rotary set up is correctly, and you've measured your object accurately, the starts and ends of the thin rectangle engraving should line up perfectly.

    If they come up short, you need to increase the Height of the template. If they overlap, you need to decrease it. Then run the test again. This may require some trial and error, but once you've got your template sized correctly for a particular tumbler, you can Save and reuse it in the future.

Design Considerations

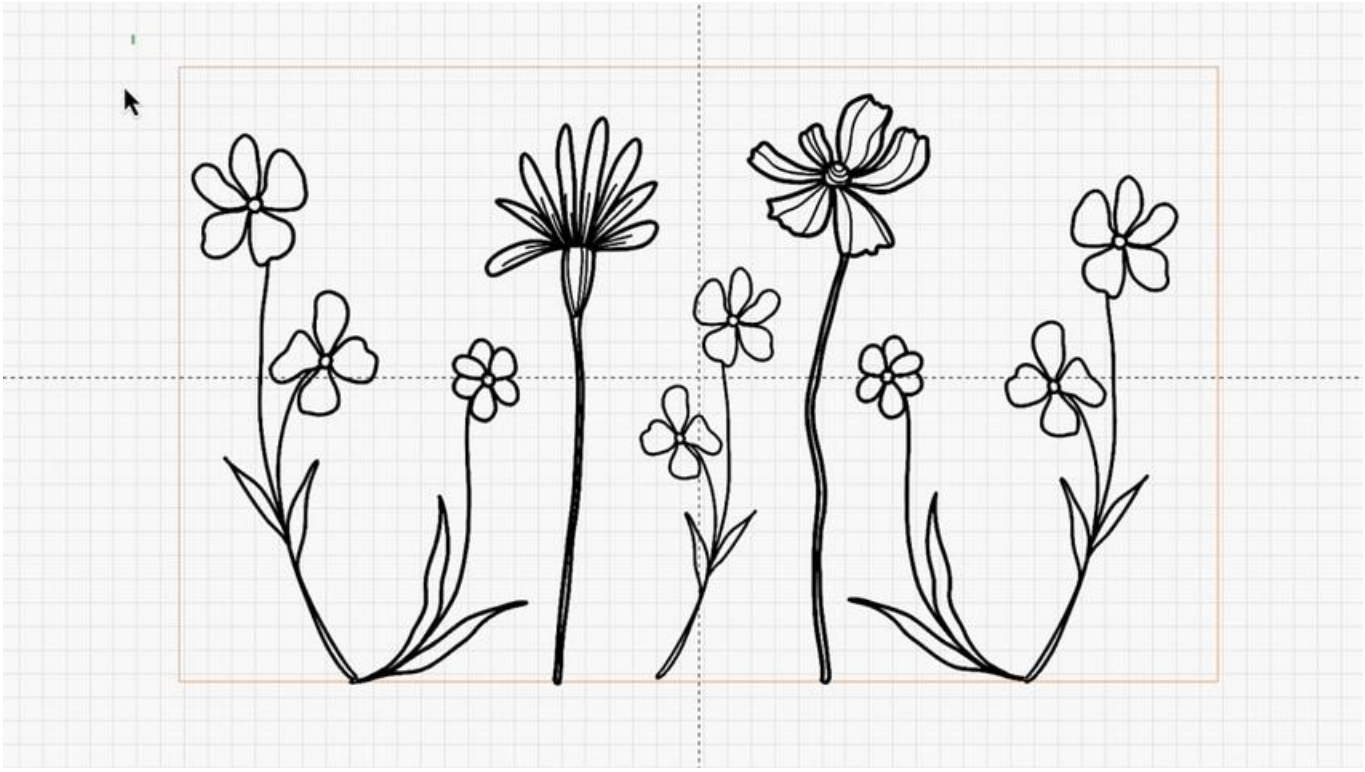
Before we move on, we'll go over some important considerations to keep in mind when you're selecting or creating a design.

The level of complexity of your design affects how easy it will be to complete your engraving successfully. You can create your design completely from scratch, or Import multiple design elements into LightBurn, and then pattern them appropriately, as we will do in this guide.

This guide will focus on an easy design, while incorporating one intermediate element — we'll show how to work your design around a logo on the tumbler.

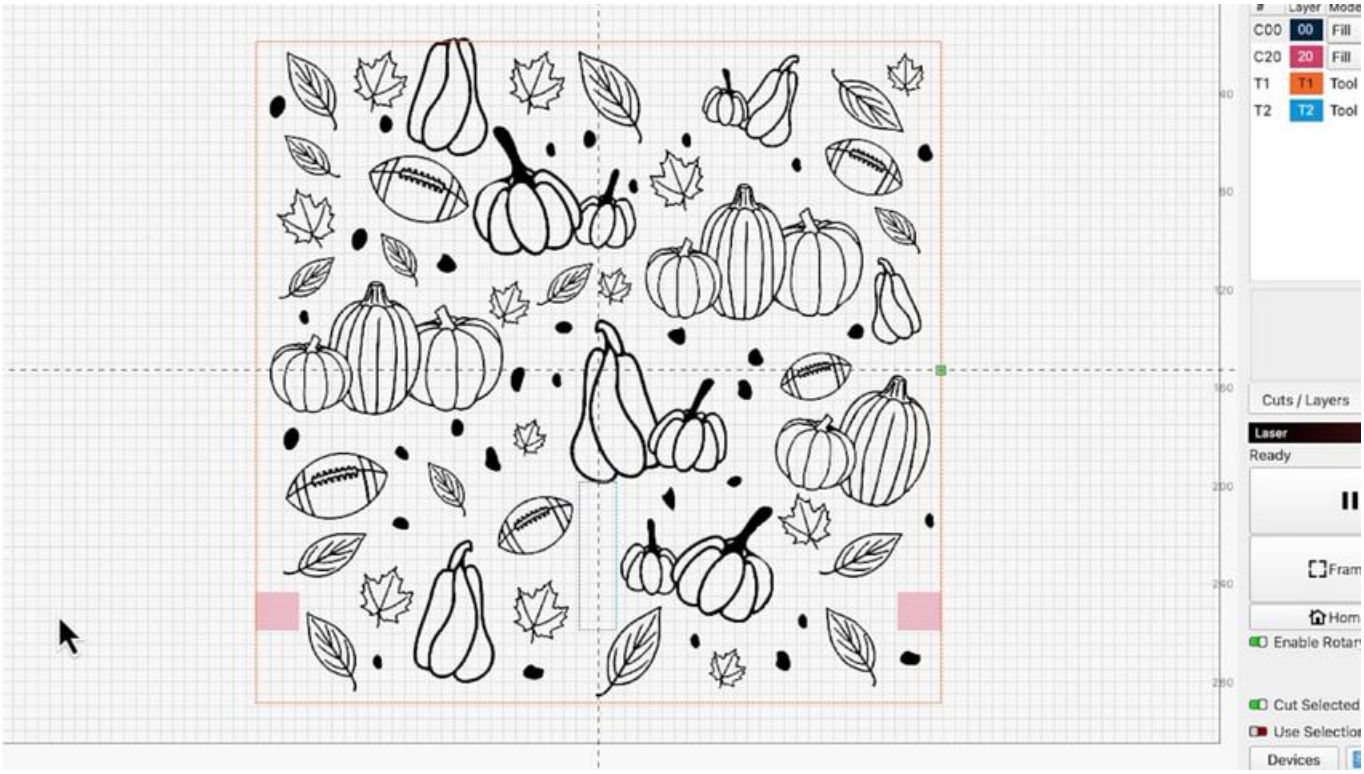
Easy

- Very loose, meaning all graphics in the design are well spaced apart, with little risk of elements overlapping or crashing into each other if your object slips on your rotary.
- No shapes that will look obviously distorted if your rotary setup isn't perfect, or there is a taper to your cup that you haven't compensated for. Distortion will be most obvious on standard geometric shapes like squares or circles.
- No logos on the cup that you need to work your design around.
- No seamless patterns. Seamless patterns are meant to line up at specific distances that won't necessarily match your tumbler's length or circumference. Fitting them onto your tumbler without distorting them can be a challenge (unless you design your own from scratch).



Intermediate

- Relatively loose but with graphics closer together, leading to more potential for overlap.
- Includes some shapes that will show clear distortions if your setup isn't perfect.
- Designs that must work around existing logos on the cup.



Hard

- Graphics that are very tightly spaced, with little to no margin for error.
- Lots of straight lines or geometric shapes that will show clear distortions.
- Seamless patterns — these not only leave zero room for error, they are also challenging to size correctly to any given object.



Laying Out and Patterning the Design

Now we're ready to place our elements into our design. First we'll place our graphics, and then we'll use Grid Array to make a repeating pattern from them.

1. You can use any graphics you like for this step — Import SVG artwork, or create shapes, text, or custom paths using LightBurn's Creation Tools.

   We chose to work with imported artwork.

2. We'll keep all of these shapes assigned to the same Fill Mode layer, since we're going to engrave them all with the same Speed, Power, and Interval settings.

   What cutting/engraving settings should I use?

   The exact settings required to achieve good engravings, or to cut through material, are highly dependent on your type of laser.

   While we can provide some rough guidelines, the best source for specific recommendations is usually your laser's manufacturer.

   > **Always pay attention to units of distance and time when entering Speed settings**
   >
   > When entering **Speed** values recommended by your laser's manufacturer or other LightBurn users, make sure to use the same units of distance and time as those from the recommendation, or to convert the values to your preferred units.

> A given number of **millimeters per second** is much faster than that same number in **millimeters per minute**. Mixing up units can lead to reduction in power output due to unexpectedly high speeds, or excessive power output — and even fire — due to unexpectedly slow speeds.
>
> Change your Control Units in the **Dimensions / Units** tab of the **Device Settings** window. LightBurn automatically converts any existing values when you switch between units.

The two primary settings controlling the depth of a cut or the darkness of an engraving are power and speed.

Power controls the laser's intensity. Higher powers allow you to cut deeper or make darker marks for engravings, but at the cost of potentially charring or melting edges, producing excessive smoke, and making a wider kerf.

Speed controls how fast the laser moves. Lower speeds will deliver more power and heat to the material, as the laser takes longer to pass over any one point. Higher speeds can complete a job faster, but will require higher power. High speeds can also exacerbate or make any existing mechanical issues more apparent.

> **Info**
>
> For a deeper dive into the relationship between these settings, see Speed vs. Power.

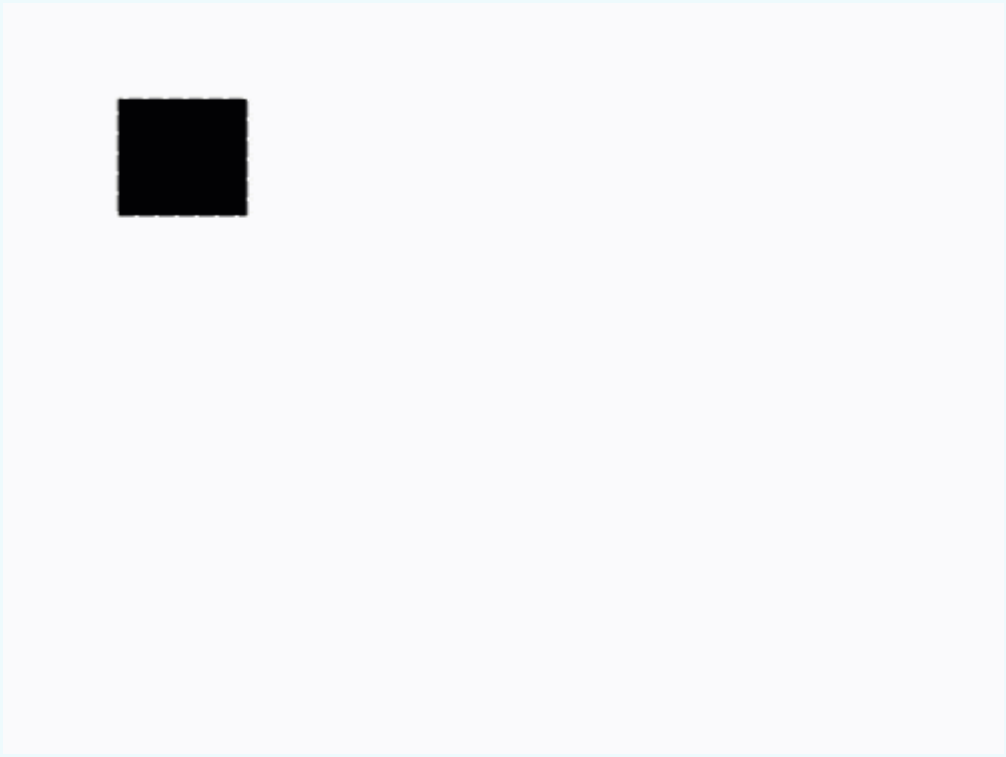
There's another crucial setting for layers set to Fill Mode — **Line Interval**. The Interval setting determines the distance between the parallel lines your laser scans as it engraves your graphics. Smaller Line Intervals lead to lines that are closer together. As a rule, you want the scan lines to be as close together as possible without overlapping (which can lead to overburning).

While we've noted that manufacturer recommendations are the best place to begin looking for cut settings, LightBurn also has a built-in Material Test that you can use to test different combinations of **Speed**, **Power**, **Passes**, **Line Interval**, and other settings. The Material Test works by automatically setting up a grid of boxes that will be cut or engraved at a range of different settings.

See Material Test for more information.

3. Arrange the artwork until you have a grouping you're happy with.
4. Next select all the graphics and open Grid Array. Increase the number of rows and columns until the pattern fits across the Width and Height of your template.

> **Quick Reference: Grid Array**



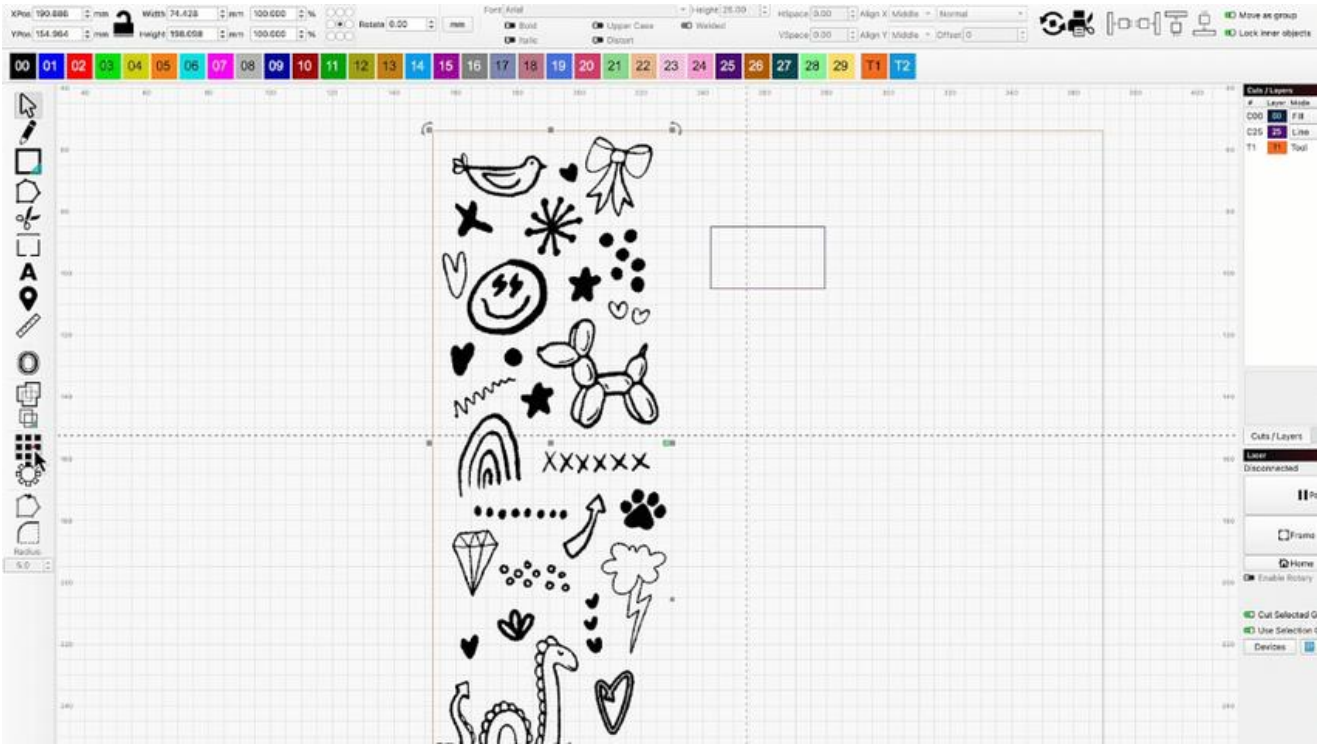
The Grid Array tool creates copies of an object (or objects) in regularly spaced rows and columns, and includes options to adjust spacing, mirror the shapes, randomize orientation, and more.

More Info → Grid Array

Location

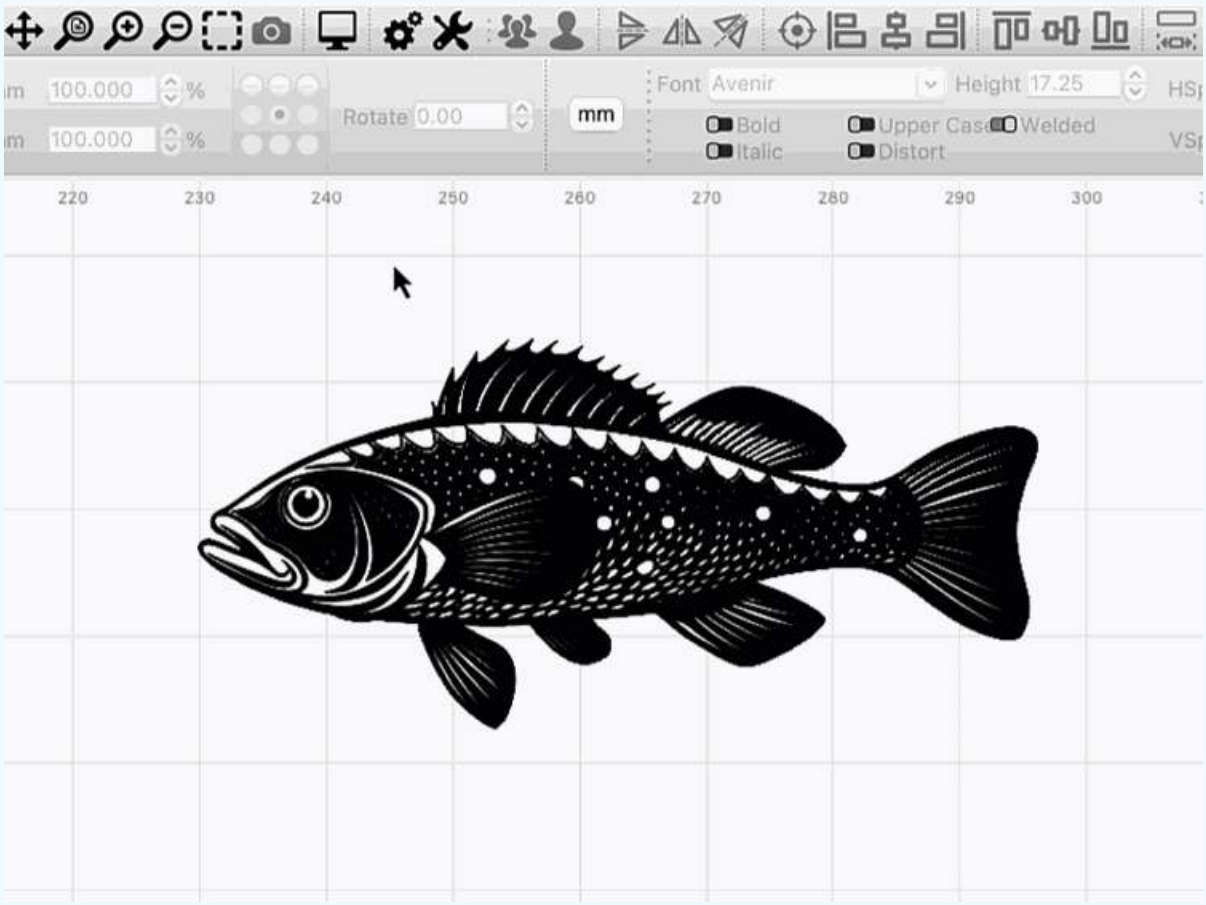
Modifiers Toolbar

Arrange → Grid / Array



5. Delete any graphics that are overlapping with the logo outline box, and rearrange anything nearby, if necessary to make the graphics surrounding the box look more organic.
6. Once you're happy with the placement of your design, Select everything, then Group it and Duplicate it.

> **Quick Reference: Grouping**



The **Group** tool combines collections of objects into single units, and the **Ungroup** tool splits Groups back into the individual objects that make them up.

Normally, all objects are independent of one another, and can be manipulated individually. But it's sometimes helpful to treat a collection of things as a single unit, to make sure they keep their relative position and size when you're moving them around.

More Info → Grouping

Location

Arrange Toolbar

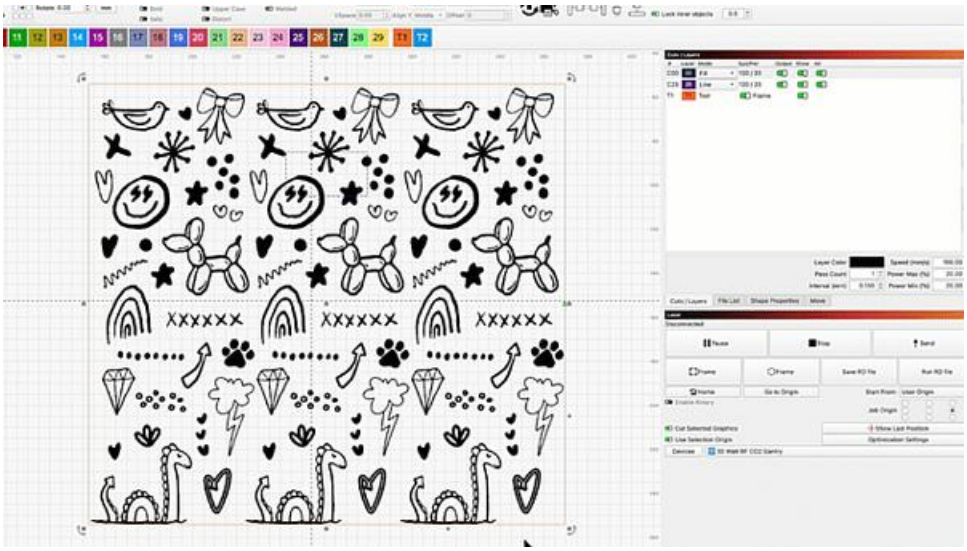
Arrange → Group/Ungroup

Right-click → Group/Ungroup

> **Keyboard Shortcuts**
>
> **Grouping**
>
> Windows: `Ctrl` + `G`
>
> Mac: `Cmd` + `G`
>
> **Ungrouping**
>
> Windows: `Ctrl` + `U`
>
> Mac: `Cmd` + `U`

7. We're going to Snap the Grouped copy to an edge of the original, so we can get a sense of how the design will look when it's wrapped around our tumbler, and make sure there is no obvious seam.

   If we're not happy with how it looks, we can make some adjustments until we are.



8. Make sure to Delete the copy once you're done making adjustments.

Positioning and Engraving

Once you're satisfied with the layout of your design, it's time to verify the positioning of your graphics around the logo outline, and run the engraving.

Verifying Position

> **Note for Galvo Users**
>
> These steps do not apply to Galvo lasers, which always work in Absolute Coordinates, and handle positioning differently. See Rotary Mode (Galvo) — Positioning Graphics and Galvo Framing for more information.

1. Make sure your laser is connected to LightBurn, and Rotary Mode is enabled.

> **Note**
>
> If you aren't working around a logo on your tumbler, you can skip the rest of this section, and Frame your entire design before moving on to engraving it.

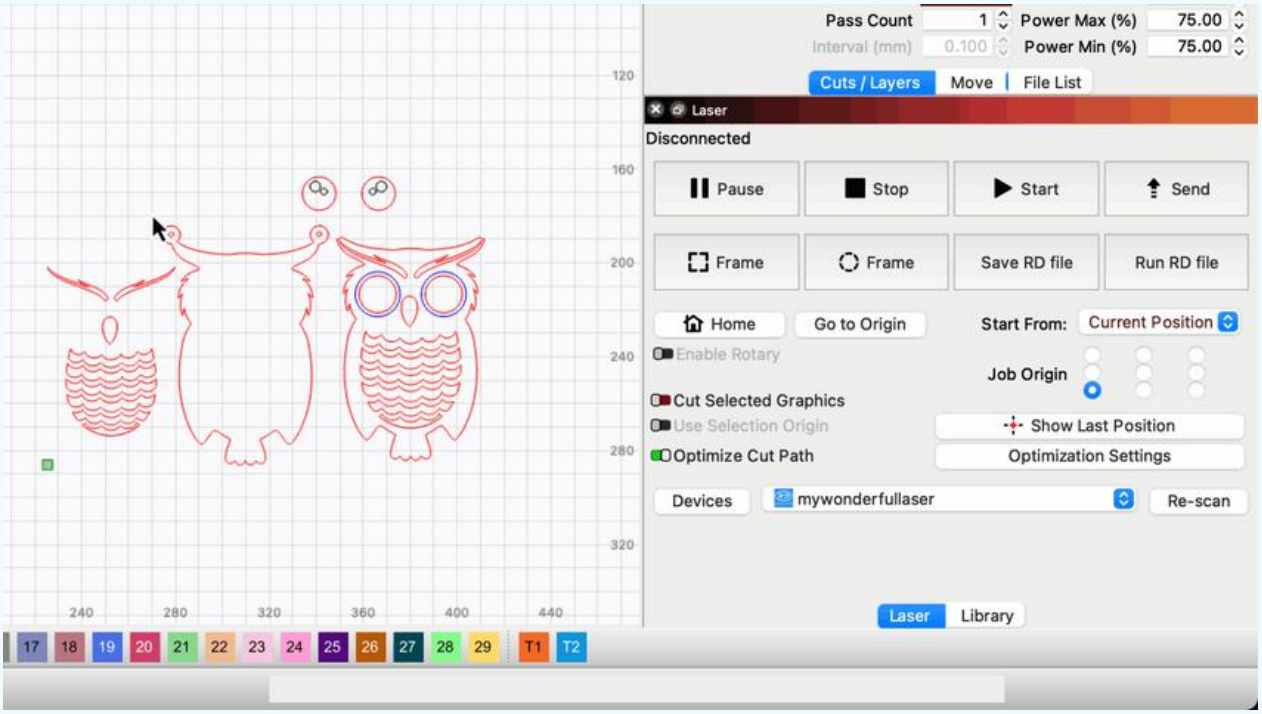
2. In the Laser Window, enable Cut Selected Graphics, but make sure Use Selection Origin is disabled.
   - Using Cut Selected Graphics will tell LightBurn to only send objects currently in your Selection to the laser when you Frame or Start the job. For this test, we only want to Frame one thing — the outline that represents the tumbler's logo.
   - Keeping Use Selection Origin disabled will mean LightBurn still includes everything that isn't being sent when calculating Job Origin. We want to make sure we include the entire tumbler template when we calculate origin, rather than just calculating it based off the logo outline.

Learn More: Cut Selected Graphics and Use Selection Origin

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=Om-6FRoy6Z0

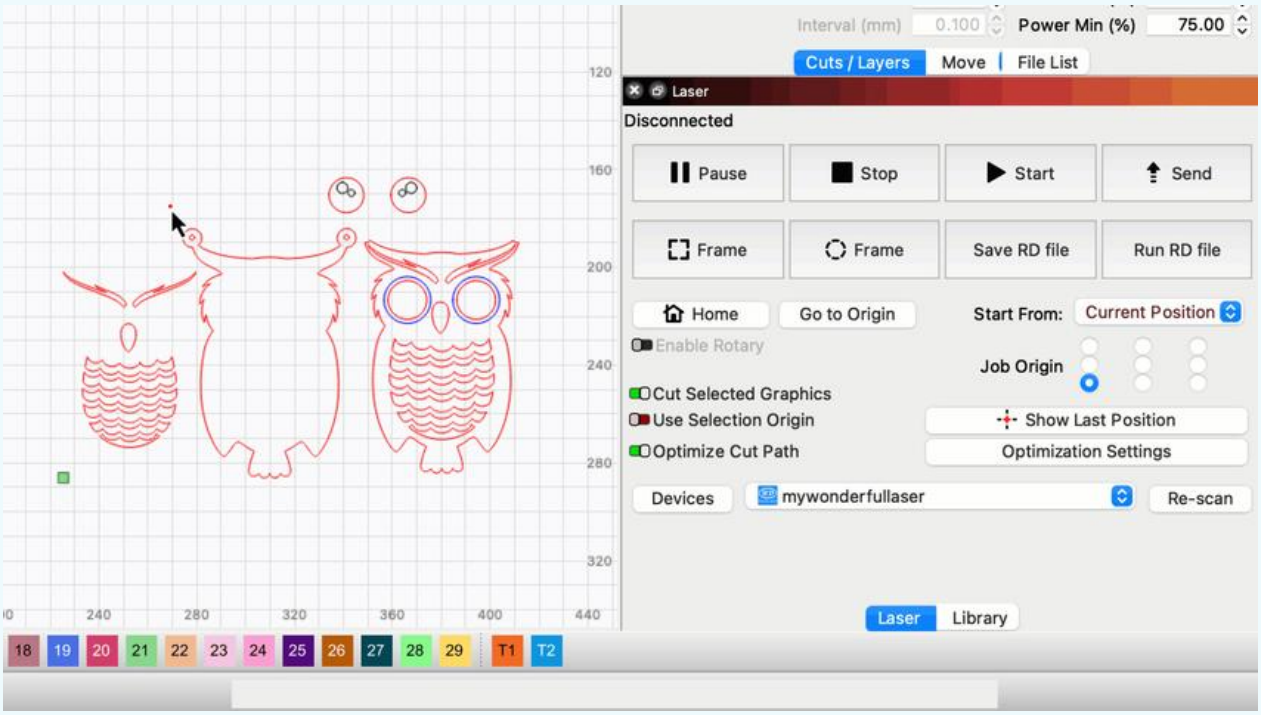
By default, LightBurn outputs all graphics in your Workspace when you Preview, Frame, Start, Send, or save a project in a machine-specific file format. This works well if you have one design in each project, but is less ideal when storing multiple designs within the same file.

In order to output only specific design elements, you can enable **Cut Selected Graphics** instead.



Use Selection Origin allows you to choose whether to orient output relative to the entire project's Origin, or just the Origin of the graphics in your selection.

The Job Origin is indicated on your design as a green square. You'll see its position move as you toggle Cut Selected Graphics and Use Selection Origin on and off in different combinations.



See Cut Selected Graphics / Use Selection Origin for more information.

3. Select the logo outline, and the logo outline only.
4. As before, set the Start From mode to **User Origin**, and the **Job Origin** to either the center-right or center-left — the same side of your cup that is closest to the logo.

Learn More: User Origin

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=aphiLTUWcjE

User Origin works almost exactly the same as Current Position, except that the starting location is programmable. To set a User Origin, you must first jog your laser to the location you want your job to start from, then set the origin.
Some lasers with digital displays, such as those with Ruida controllers, have an `Origin` button which you press to set the origin. GCode-based systems use the **Set Origin** button in the Move Window in LightBurn to do the same thing.

For supported systems, use **Clear Origin** to remove and reset a custom-set origin.

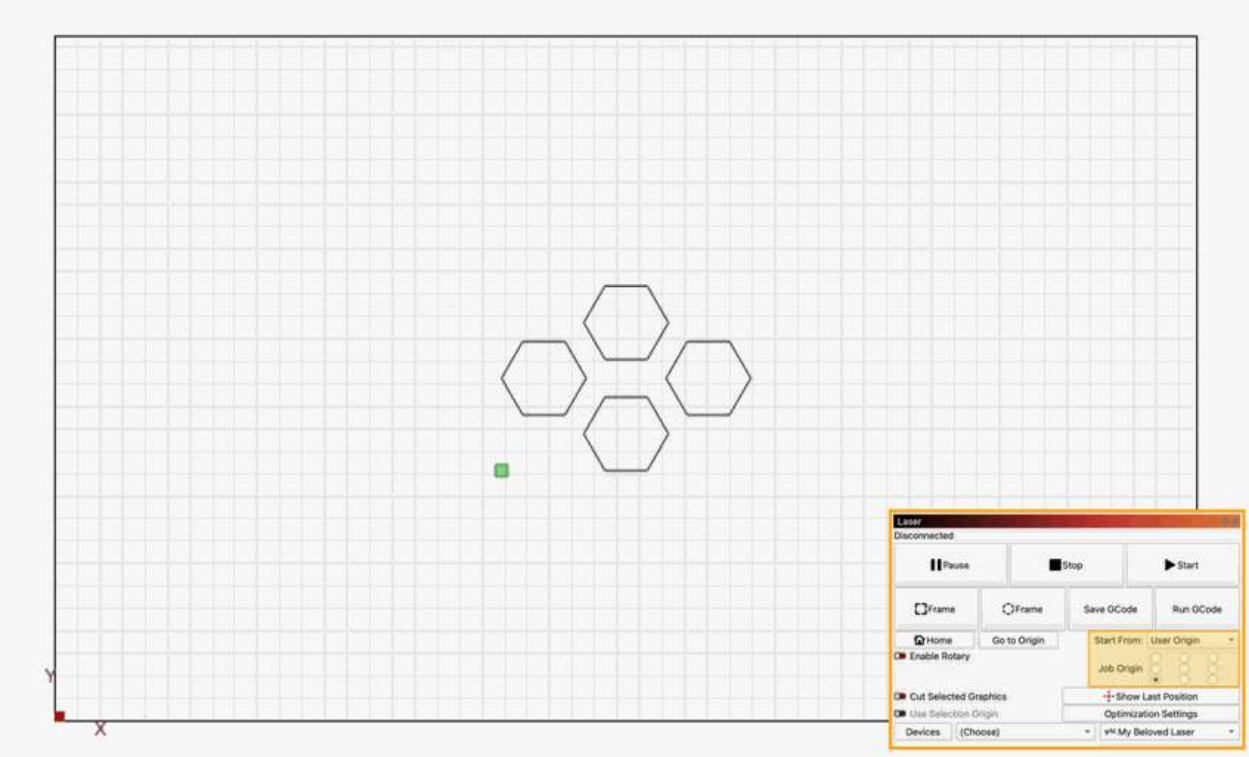


After setting the origin, you are free to jog your laser around its work area. With User Origin selected as the **Start From** mode, the laser will always move back to start the project in a position relative to the programmed location.

You can also use the **Go to Origin** button in the Laser Window to command your laser to travel back to the origin you've set.

As with Current Position, the **Job Origin** setting determines how output will be oriented relative to the user-defined origin.

Here, the Job Origin is set to the lower left:

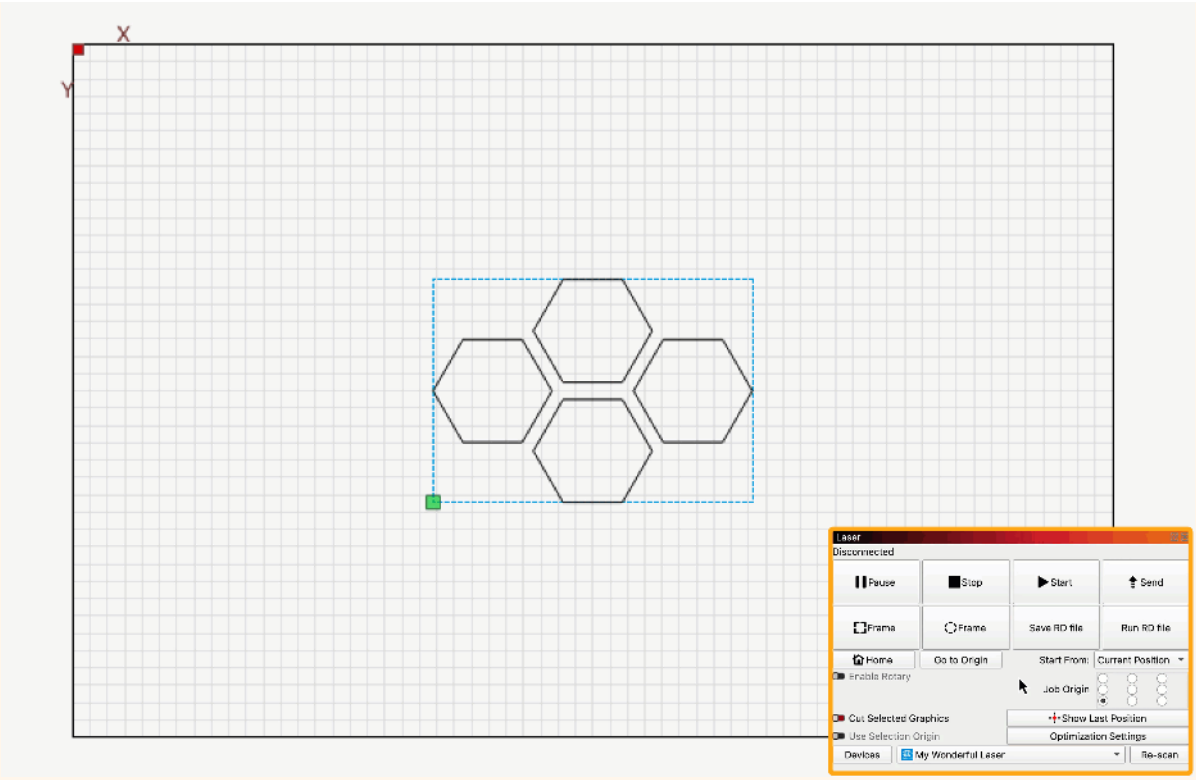


Notice the location of green Job Origin indicator. This represents whatever location you've set for your custom origin. When you start the job, the laser is going to move from wherever it is currently located and cut the four hexagons up and to the right from the programmed origin location.

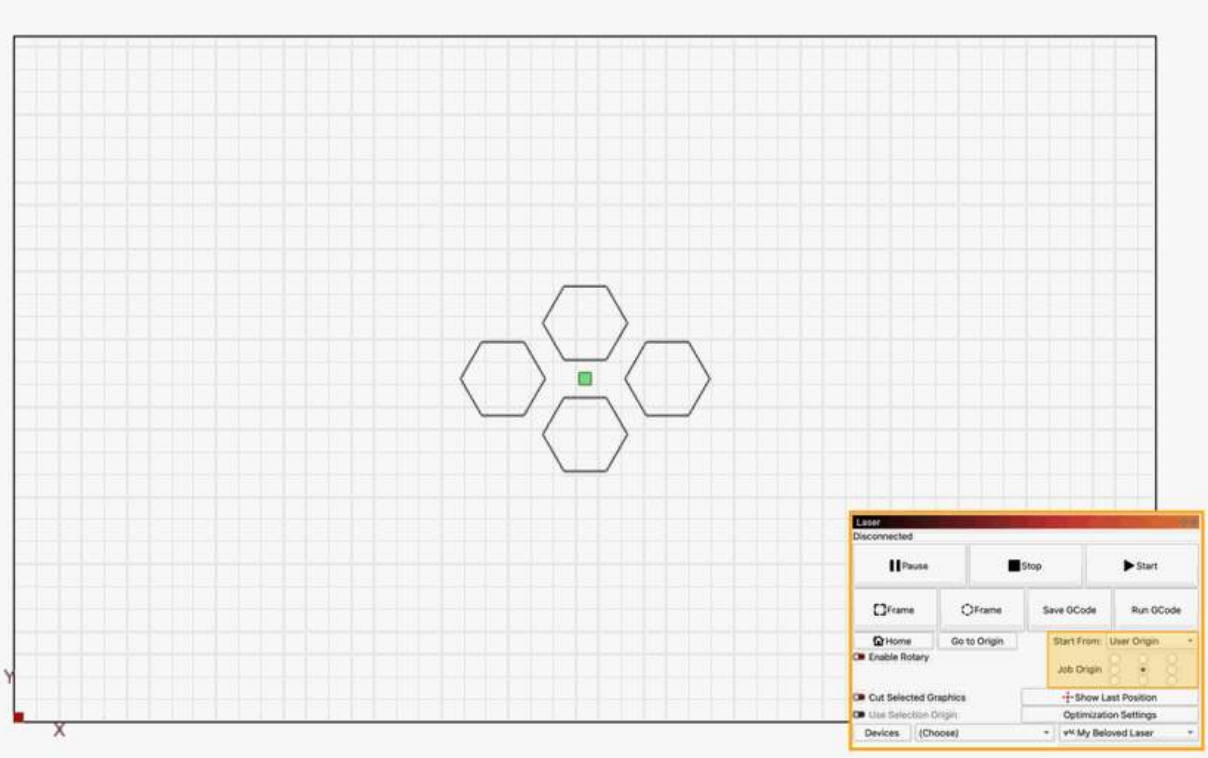
Why is the Job Origin offset from the graphics?

When using Current Position or User Origin, the green Job Origin indicator is always located on a corner, side, or at the center of an imaginary box — the smallest possible rectangle that can fully contain all graphics you're sending to the laser.

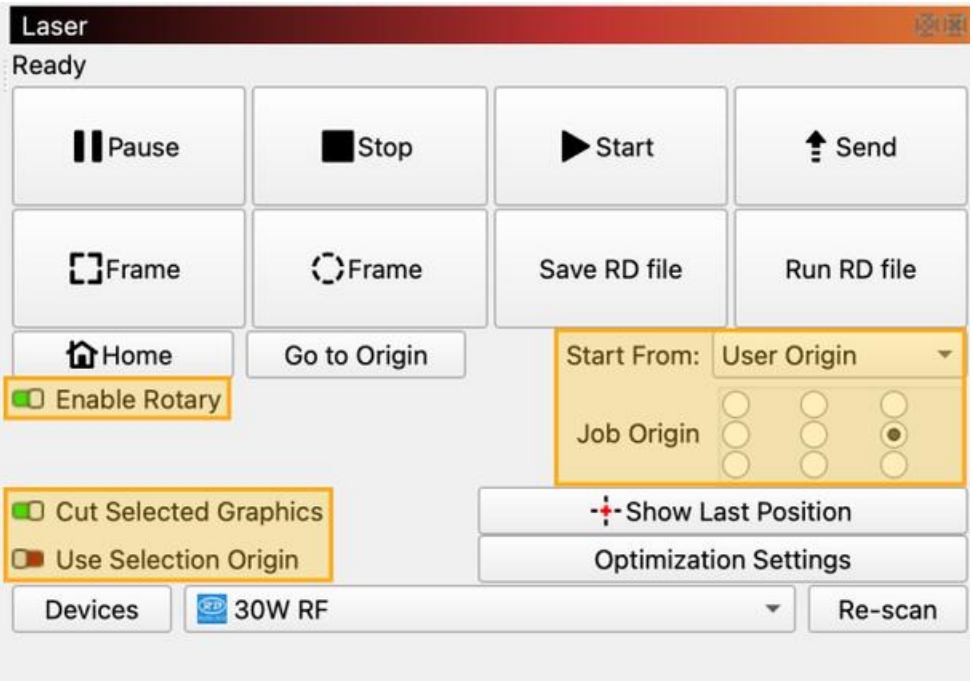
Below, we've added a Tool Layer to represent that imaginary box. Notice how the Job Origin moves around the box as we change the location in the 9-Dot selector.



Now imagine that you want to etch this four hexagon pattern onto a coaster or a phone case. Lining it up from the lower left is not easy. If you change the Job Origin setting to center, you get this instead:



> The job will be centered around the custom origin you set. If you position the laser head directly over the center of the item you want to etch before setting the User Origin, the output will be centered on the item.



5. Jog the laser head so that it is to the right or left of the logo, and still roughly centered between its top and bottom.

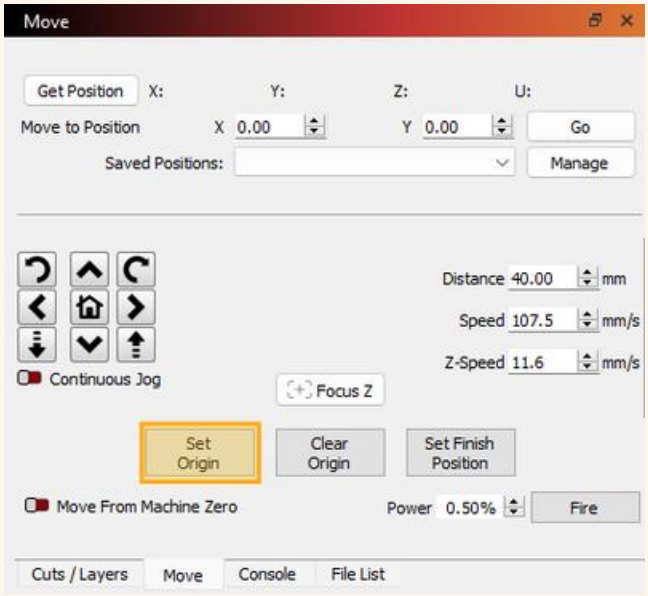


6. If you're using a GCode-based laser, press the **Set Origin** button in the Move Window. If you're using a DSP laser, use the `Origin` button on your laser's controller.

Where's the Set Origin button?

The **Set Origin** button is presented in the Move Window, which is docked behind the Cuts / Layers Window by default. If you don't see it, go to Window → **Move** to re-enable.

This button is only available if you are using a GCode-based laser. If you're not sure what kind of laser you have, see Identifying Your Laser.



7. Press the Frame button in the Laser Window — you should see the laser trace a path around the logo on the cup. If it's off by a bit, jog the laser, update the User Origin, and try again, until you get it as close as possible, and are sure your engraving won't overlap with the logo.

Note on Framing with Tool Layers and Cut Selected Graphics

Normally, when you Frame a project that includes Tool Layers, the Tool Layers are used when calculating Job Origin, but are not themselves Framed — the laser will only travel around graphics that will actually be sent to the laser.

However, when Cut Selected Graphics is enabled, and a Tool Layer is selected, the laser will include the Tool Layer in anything it physically Frames around. In this case, since we only have an object set a Tool Layer selected, that is the only thing that's Framed.

Learn More: Tips for improving Framing visibility

> **How to fire your laser at low power during Framing (diode lasers only)**
>
> If your laser has a red dot pointer, it makes framing easier to see. Otherwise, your laser will travel around with the laser turned off. LightBurn does offer the ability to run diode lasers at a very low power to aid in Framing. To do so:
>
> 1. Go to Edit → Device Settings to open the Device Settings window.
> 2. In the Basic Settings tab, toggle on the **Enable laser fire button** and **Laser on when framing** settings. Click **OK** to apply.

3. The Move Window should now have a **Fire** button and a box to enter a power level. Keep this value very low to avoid burning the material — start at 0.25%, and only increase that value only if the beam is not visible.
4. The **Fire** button will toggle the laser on and off. Now you can use the **Frame** command with the laser visible.
5. Adjust the position of the material and/or the laser head until you're happy with the layout of the design.

> **Danger**
>
> The Fire button should only be used for diode lasers, which generally don't have a red dot pointer for Framing.
>
> Always wear proper eye protection when Firing your laser. Consult your laser's manufacturer for information on the proper eye protection required for your laser.
>
> This should never be used for a CO_2 laser, which has an invisble beam that could blind you or start a fire.

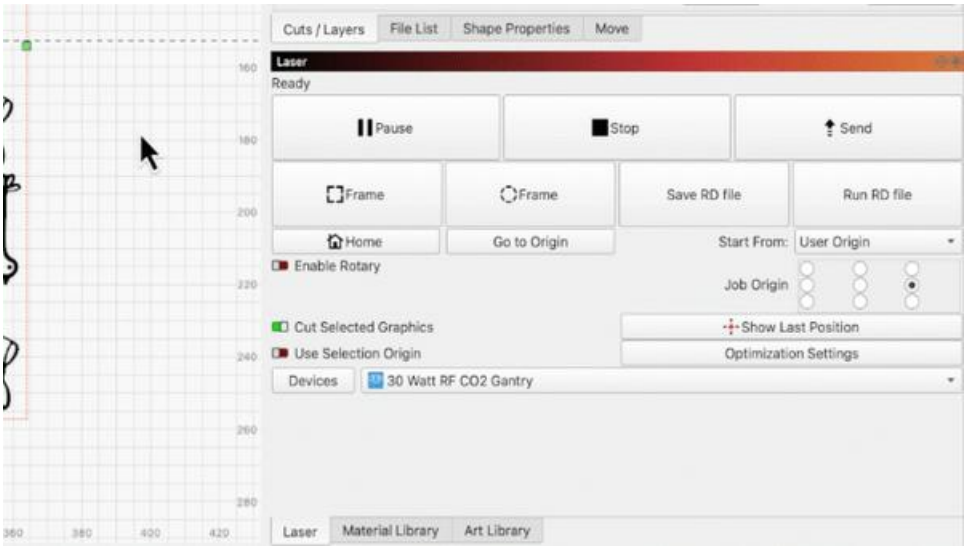
Frame Continuously

Frame Continuously is an option in the Device Settings window that tells your laser to repeatedly Frame the same path until you tell it to stop.

Sometimes you won't get the positioning just right on your first try, but a little nudge of your material will do the trick. Other times, you'll just want to want to double or triple check the position. In either case, Frame Continuously will be a big help.

To enable it:

1. Go to Edit → Device Settings to open the Device Settings window.
2. In the Basic Settings tab, toggle on the **Frame Continuously** setting. Click **OK** to apply.
3. Now when you click either of the Frame buttons, a dialog window will open, letting you know that continuous Framing is active, and to press OK when you're ready to stop.

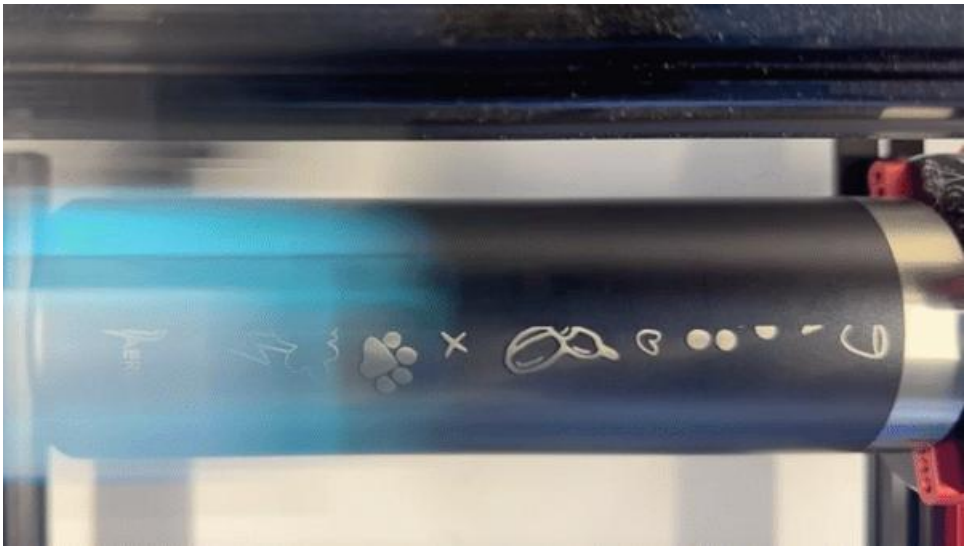


Engraving

> **Tip**
>
> Make sure your tumbler is still level on your rotary and your laser is still in focus. Refer back to Testing the Template Steps 5-6 if you're not sure.

This is the big moment — we're going to engrave our tumbler.

1. Ok, not quite yet. If you're nervous, you can keep the painter's tape from the earlier test on, and adjust your settings to lightly etch it, as a final step to verify your design, rotary setup, and alignment.
2. If you're ready, now it's really the big moment. Press Start in the **Laser Window**. If you're using a DSP laser, you can also choose to Send the project to your laser, and start it from the controller.



3. How does it look? Prior to removing it from the rotary, make sure the engraving made it all the way through the powder coating. With a Chuck style rotary, you can run a second pass, if needed, to clean up any spots before you take it off.

4. After we remove the tumbler from the rotary, we'll give it quick spray with a cleaning product and a scrub with a magic eraser to clean up remaining powder coat residue, and we've got a great looking tumbler.
5. We did it! Look at that perfect wrap!



What's Next

- Next time, you can try upping the difficulty level and engraving a more challenging tumbler or design.
- Try building names or other personalized features into your design — you can pattern the graphic elements into a full wrap, then add in custom text, and rearrange the graphics around it.
- Instead of working around the logo completely, see if you can come up with any fun ideas for incorporating the logo into your design.
- Try a seamless design. We mentioned before that they're the most challenging type of design to master, but they're also deeply satisfying to get perfect!

  Check out our **Seamless Tiling Patterns** workshop video to learn how to make your own from scratch:

  ▶ Watch the video on YouTube: https://www.youtube.com/watch?v=-mdueIcFtQ8

3D Sliced Engravings

LightBurn's 3D Sliced Image Mode enables Galvo users to create engravings with both visual and physical depth. This 2.5D engraving mode does not require a live Z Axis and can produce highly detailed engravings, but does not offer precise control over the engraving's depth.

3D Sliced mode uses multiple passes to engrave a design one layer at a time. This mode is intended for use with depth maps – sometimes called height maps – which are specially designed greyscale images in which a pixel's brightness represents depth, rather than shading.

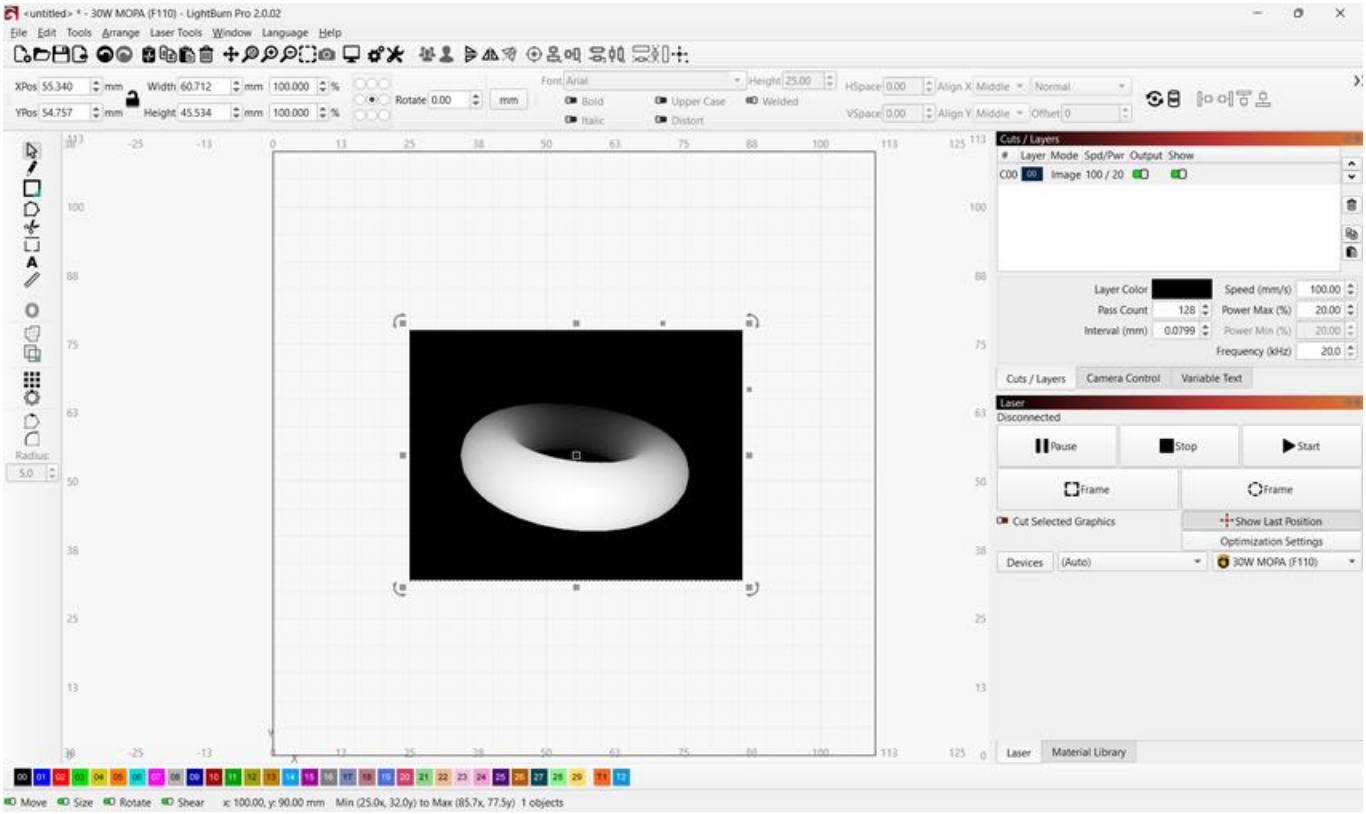
Lighter pixels are engraved with fewer passes than darker pixels, resulting in a deeper engrave in darker areas of the image. Enabling Negative Image inverts brightness, leading to lighter areas engraving deeper than darker areas.

Depth maps can be created by hand, although tools like STL2PNG can generate raster images from STL files to create more complex depth maps. Once a depth map is created, it can be imported to LightBurn like any other image.

Using 3D Sliced Mode

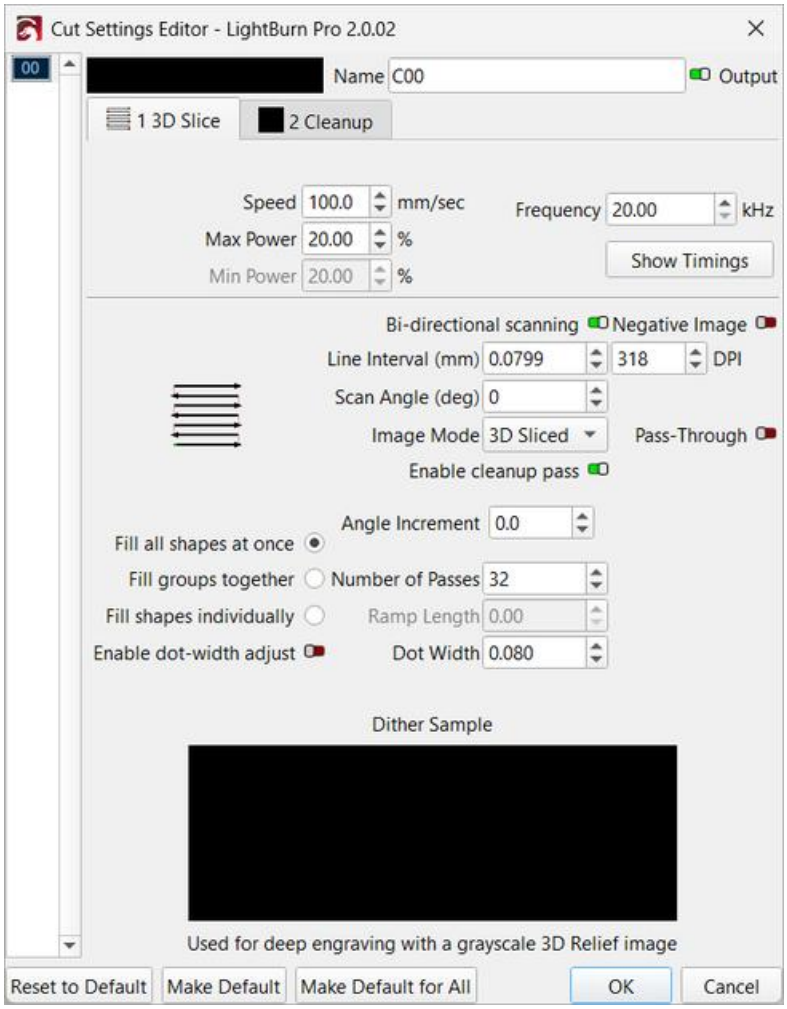
Create a Depth Map

Produce a greyscale depth map image in any tool you choose and Import it into LightBurn.



Enable 3D Sliced Mode

Use the Cut Settings Editor to set the Image Mode to 3D Sliced. This mode is only available when using a Galvo laser.



Adjust Settings

Most settings are shared with the other Image Modes, but there are a few details specific to this mode.

Number of Passes

In **3D Sliced** mode, raising the **Number of Passes** splits the depth map into more layers, increasing the effective resolution of the engraving. The darkest shades in your depth map will be engraved with the full Number of Passes, and all lighter shades will be engraved with progressively fewer passes, down to zero passes for pure white shades.

Using a large Number of Passes may require decreasing power to avoid burning deeper than intended. This has a similar effect to decreasing layer height on a 3D printer.

> **What about 24-bit depth maps?**

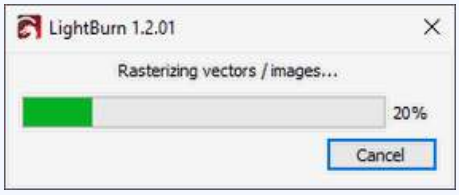
If you've tried to use a 24-bit depth map in LightBurn 2.1, you may have been confused that it shows up as an 8-bit depth map. This is because a 24-bit depth map is really using 3 channels of 8-bits each.

As of LightBurn 2.1, LightBurn support 16 bit depth maps. If you plan to run more than 256 passes, a 16 bit image is better, as it can contain more than 256 shades of gray. You can see the bit depth of an image that's been imported into LightBurn by selecting only that image and checking the status bar, as shown in the image below.



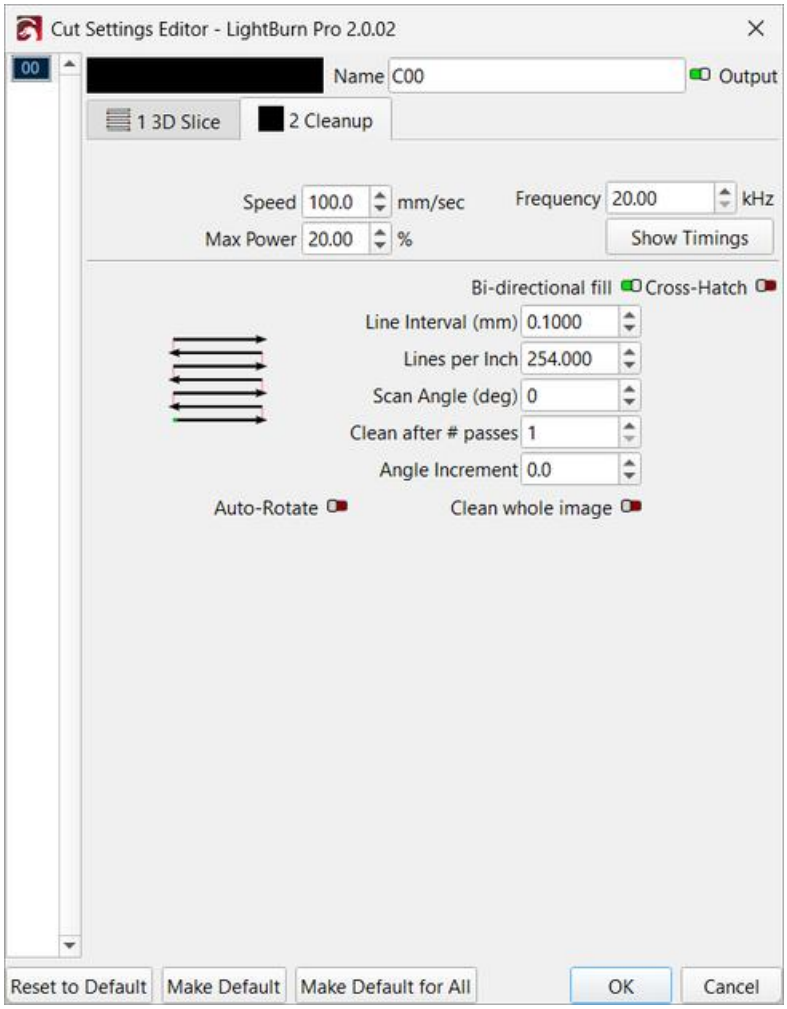
Note

Larger, more intricate designs may take more time to process, due to the complexity of generating each pass. If you see this dialog, please be patient as your computer may need some time to prepare your design.



Cleanup Passes

Toggling on **Enable cleanup pass** will automatically add a **Cleanup** Sub-Layer in the **Cut Settings Editor**, where you can specify how often automatic cleanup passes should run, and adjust the settings for those passes.



Cleanup passes are typically run with a tighter **Line Interval**, higher **Frequency**, and lower **Max Power** than the main engraving passes, and are used to clear debris and produce a better finish.

Angle Increment

> **Process Change: Angle Increment**
>
> This setting is new in LightBurn 2.0. Previous LightBurn versions used crosshatching by default for 3D slicing. Now that users can set angle increment themselves, crosshatching is no longer automatic.
>
> To reproduce the crosshatching effect from previous LightBurn releases, set the **Angle Increment** to 90°.

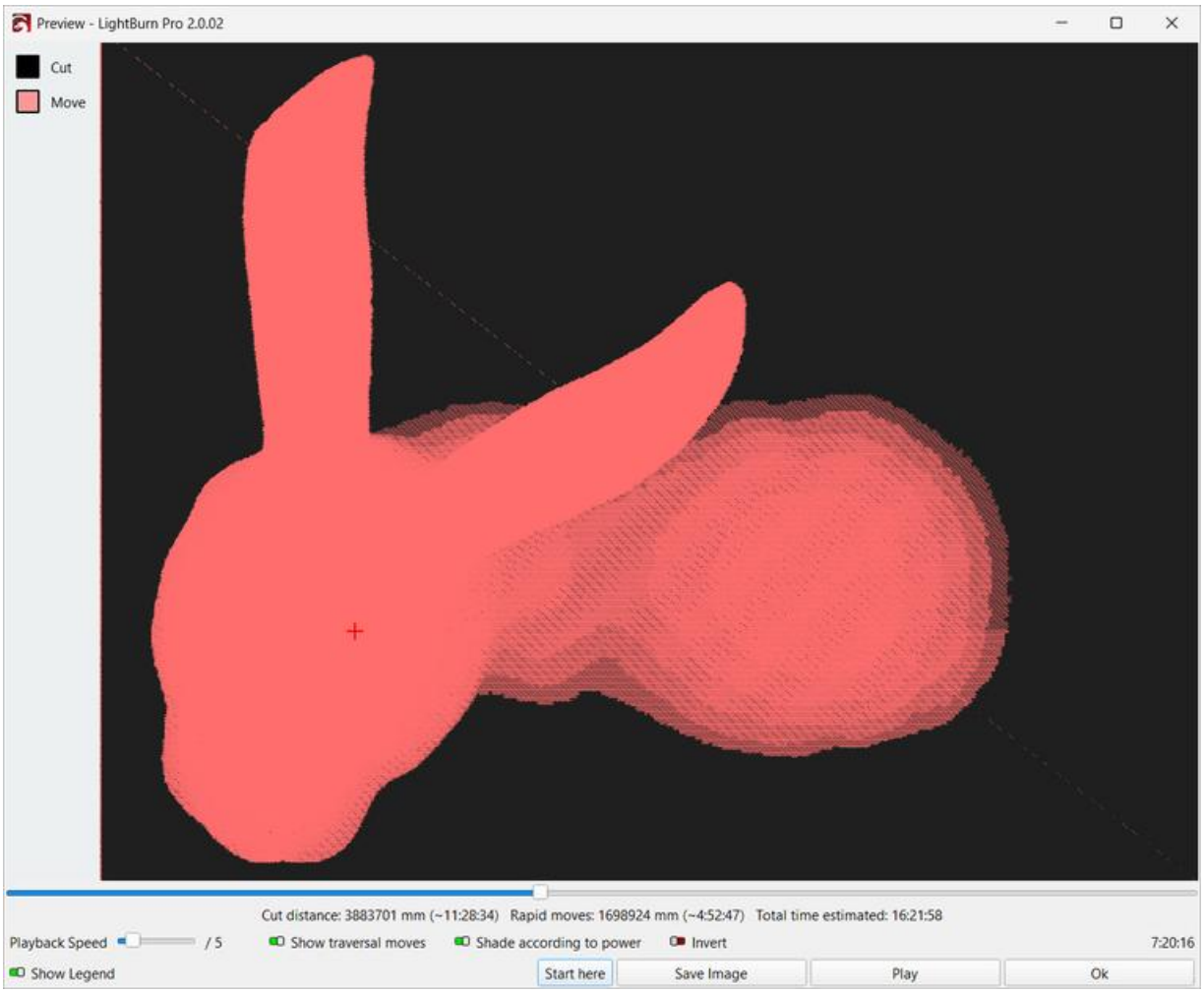
The **Angle Increment** setting will determine how much the Scan Angle rotates between passes.

With the default setting of 0, all passes will be engraved in the same direction. Setting a non-zero number will tell LightBurn to rotate between passes. As an example, if the Scan Angle is set to 0 and the Angle Increment is set to 30°, subsequent passes will be at 0°, 30°, 60°, 90°, and so on.

Preview and Run

Check the Preview to confirm your design is outputting as you're expecting. If you click the **Play** button, you should see your design render from the bottom to the top, with black indicating the engraved portion for each layer.

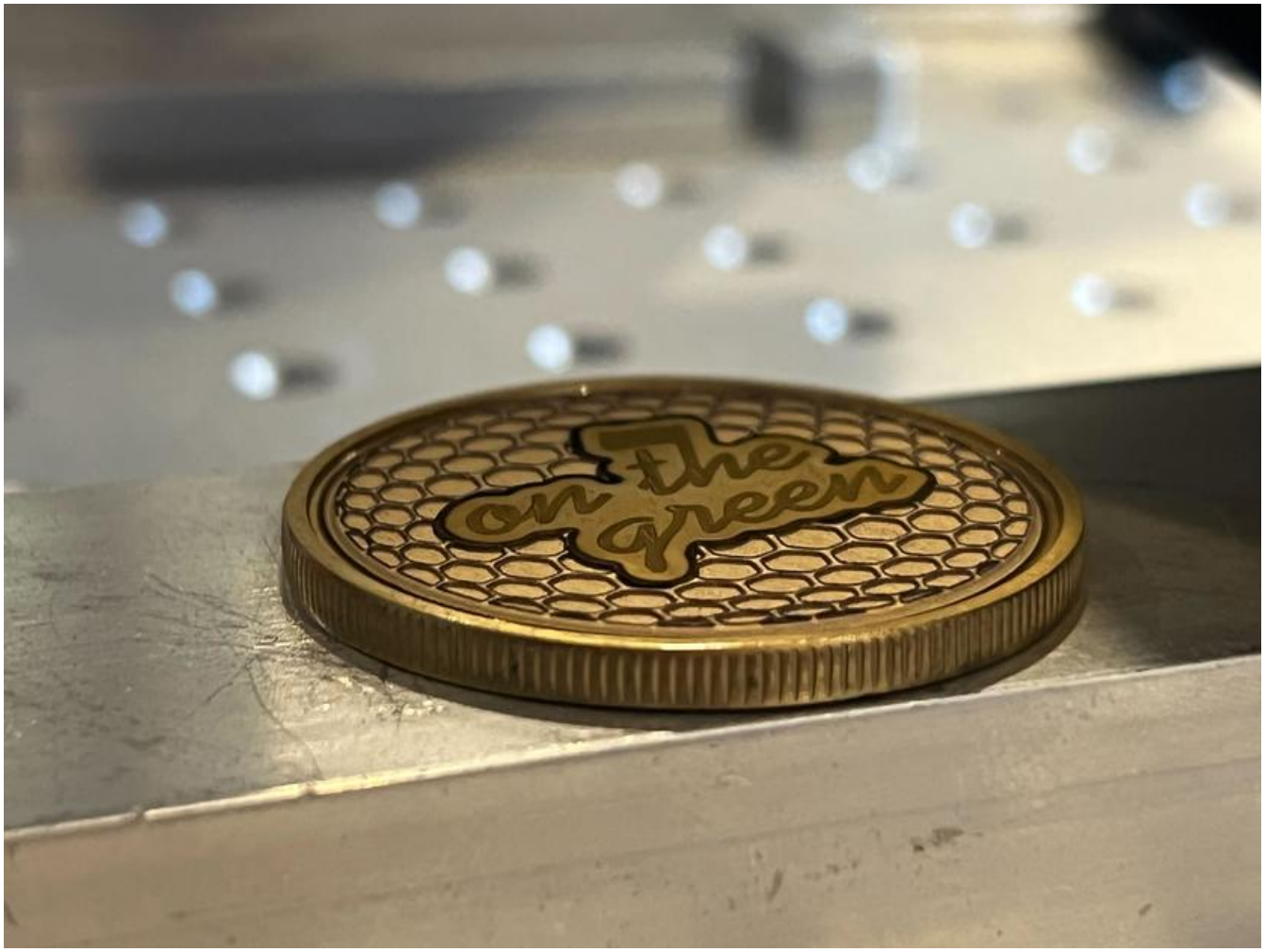
Enable **Show traversal moves** and check to make sure that the red areas expand with each pass. The red areas are made up of lines indicating movements made while the laser is off, so, as the job progresses, the laser should be passing over increasingly large areas of lighter shading, while continuing to engrave increasingly small darker-shaded areas.



More Examples






The
on Green

Licensing

LightBurn is not a subscription service. Buy LightBurn once and use it forever, with a year of updates included. Purchase another year of updates if you want new features after that.

Unlike subscription services that require you to pay monthly or yearly to continue using them, LightBurn does not require you to keep paying to keep accessing the software you've already purchased.

When you purchase a LightBurn license, it includes a year of free updates. After that year is up, you can unlock future updates by adding a year of updates to your existing key at any time.

A standard LightBurn license has three seats, meaning it can be installed and activated on up to 3 computers at a time.

What is a License Key?

A LightBurn license is a digital key used to unlock the software for permanent use.

Every new license comes with a year of free updates from the date you first activate it. After a year, you'll need to renew your software update period in order to receive updates for another year. If you renew before your key expires, you'll receive an additional two months on your one-year renewal.

If you choose not to renew, you will always be able to continue using the most recent version of LightBurn that is compatible with your operating system and was released within your license's valid update period.

License Portal

Buy LightBurn

Add a Year of Updates to Your LightBurn License Key

To download an earlier version of LightBurn that is compatible with an operating system no longer supported by current versions of LightBurn, please see our OS compatibility table or click on the blue box below for direct links to the most recent version of LightBurn for your operating system.

Support for Other Operating Systems

- LightBurn version 1.7 is the last version compatible with Linux, versions of Windows prior to 10, and versions of macOS prior to 11. Download LightBurn 1.7.08 and see our 1.7 documentation for information and installation instructions.
- Users with macOS 12 should download LightBurn 2.1.04.
- Users with macOS 11 should download LightBurn 2.0.05.
- macOS users running versions older than 10.13 should download LightBurn 1.6.03, which supports macOS 10.11 or newer. See our legacy documentation for more information and installation instructions.

- Users limited to 32-bit versions of Windows should download LightBurn 1.5.06, which is the final 32-bit release of LightBurn. See our legacy documentation for more information and installation instructions.

License Types

The type of license you need depends on the controller used by your laser. LightBurn supports three device types: GCode, DSP, and Galvo.

- LightBurn Core supports GCode-based devices.
- LightBurn Pro supports both DSP and Galvo devices, in addition to GCode-based devices.

If you're not sure what type of license you need, see Identify Your Laser.

If you started with a license that supports one type of device, and now need to use LightBurn with a different type, you can add support for the new device type to your existing key. Please note that adding support for additional devices does not renew your key.

A LightBurn license automatically comes with 3 seats, meaning you can activate it on three computers at one time. You can use our License Portal to view and remove activations on your key, freeing up activations in use on old or inoperable computers so you can activate on new ones.

If you already have a key, you can purchase additional seats here. If you would like to purchase a new key with more seats than the default of 3, or are interested in a Floating license setup, see our Educational and Volume Licensing page.

If you sell your laser and wish to sell your license along with it, that is permitted. Please note that it is not permitted to sell part of your license to someone else.

Have questions? Read our FAQ or contact us at Support@LightBurnSoftware.com

Additional License Information

License FAQ

Frequently asked questions about the LightBurn License.

Read More

License Management Guide

How to access the license portal, manage your license key, and activate your license offline.

Read More

Educational and Volume Licensing

Information on licensing options that work well for large numbers of users.

Read More

System Locked and Floating License Setup

Directions for advanced licensing setup options, mostly used in schools or other shared spaces.

Read More

Enterprise Security FAQs

Answers to questions we frequently receive from corporate or educational IT departments prior to purchasing.

Read More

LightBurn End User License Agreement

A copy of the LightBurn EULA, available without downloading the software.

Read More

Accessibility Statement

LightBurn's current accessibility statement.

Read More

License FAQ

Is a LightBurn license a subscription?

No. You pay once and and can use it forever, with a year of updates included.

After a year, you'll need to purchase another year of updates to continue receiving updates, at a significantly lower cost than a new license.

Unlike subscription services that require you to pay monthly or yearly to continue using them, LightBurn does not require you to keep paying to keep accessing the software you've already purchased.

Which version do I need?

The type of license you need depends on the controller used by your laser. LightBurn supports three device types: GCode, DSP, and Galvo.

- LightBurn Core supports GCode-based devices.
- LightBurn Pro supports both DSP and Galvo devices, in addition to GCode-based devices.

If you're not sure what type of license you need, see Identify Your Laser.

Can I use the same license key on any operating system?

Yes, the same key can be used to activate LightBurn on Windows, macOS, or Linux.

How many computers can I use it on?

A LightBurn License automatically comes with 3 seats, meaning you can activate LightBurn on 3 computers at one time. You can use our License Portal to manage your activations.

If you'd like to activate LightBurn on more than 3 computers, you can purchase additional seats for your license here. The cost of each additional seats depends on your license type.

If you would like to purchase a new key with more seats than the default of 3, or are interested in a Floating license setup, see our Educational and Volume Licensing page.

I lost my license key — how do I find it again?

If you've lost your license, you can enter your email address in our license key recovery tool to look it up.

You can also use your email address to access our License Portal and view your license.

License Portal

If you don't turn up any results, or can't access the portal, contact us at support@lightburnsoftware.com

Does the license expire?

No, but your license's update period has an expiration date — a year from when you first activated it. When the update period ends, the key is still valid, but you won't be able to use any version of LightBurn released after the end of the update period. Any version released before the key expired will continue to work.

My license expired, how do I revert to a compatible version of LightBurn?

To download and install a version of LightBurn compatible with your key, visit our release repository.

How do renewals work?

If your key's update period has expired, a renewal will add a year of updates, starting from the date you purchase the renewal.

If you renew before your update period expires, the renewal period will be extended starting from the date your key was set to expire. You'll also receive an additional two months on your one-year renewal.

How much is it to renew?

The price to renew your license's update period and receive another year of software updates is $40.

Renew your license here.

Do renewals cost more if I have more than 3 seats on my license?

No. The price of a renewal is the same regardless of the number of seats on your license.

Where can I find LightBurn's EULA?

You can find the LightBurn EULA within LightBurn by going to **Help → End User License Agreement** or in our documentation.

Can I add support for a new type of laser to my existing key?

Yes. You can upgrade your license here.

Will I lose support for other devices when upgrading?

No. A license upgrade adds supported device types to your key, but doesn't take any away.

Does adding support for a different device type renew my key?

No. Adding support for a different device type does not renew your key, or affect your key's update period in any way.

I already have a LightBurn license. Can I try a different version before upgrading?

Yes. If you are already have a LightBurn license, but would like to try it with a device type your key does not currently support, go to Help → License Management and click **Deactivate License**, then **Extend Trial**.

How do I know when my upgrade has been applied?

Our systems are fully automated, and all upgrades are processed promptly. However, LightBurn will need to contact our licensing server to register any upgrade, and it can take up to 24 hours for LightBurn to sync with the server. You can force LightBurn to check back in by deactivating and then reactivating your key in the License Management window.

How do I know if my license supports my laser?

If you've already activated your license, but aren't sure if it supports your type of laser, go to Help → License Management to see a list of device types supported by your license.

If you're unsure whether your laser is one of the types listed, see Identify Your Laser.

If you're unable to connect to your laser even though it is supported by your license, see Troubleshooting: Connection Problems.

Can I transfer my key to another user?

Yes. You are free to transfer your key to another user, or sell it along with your laser. The current owner of the key needs to email us from the email associated with the key to request the transfer to the new owner's email address. The new owner will be granted access to our License Portal to manage the key.

Please note that selling or sharing individual seats on your license key is not permitted. Keys must be transferred in their entirety, including all seats.

License Management Guide

The license system used by LightBurn has a web portal where you can view the computers activated on your license, deactivate unused computers, or process an **Offline Activation**.

If you purchased your key from LightBurn directly, you will have access to this portal. If you purchased from a 3rd party vendor, we will need to assign your license key to your email address for you to gain access. Contact us at support@lightburnsoftware.com with your full name and license key to have us do so.

Accessing the License Portal

The first time you use the portal you will need to enter the email address assigned to your key and request a password reset.

License Portal

Follow these instructions to access the License Portal:

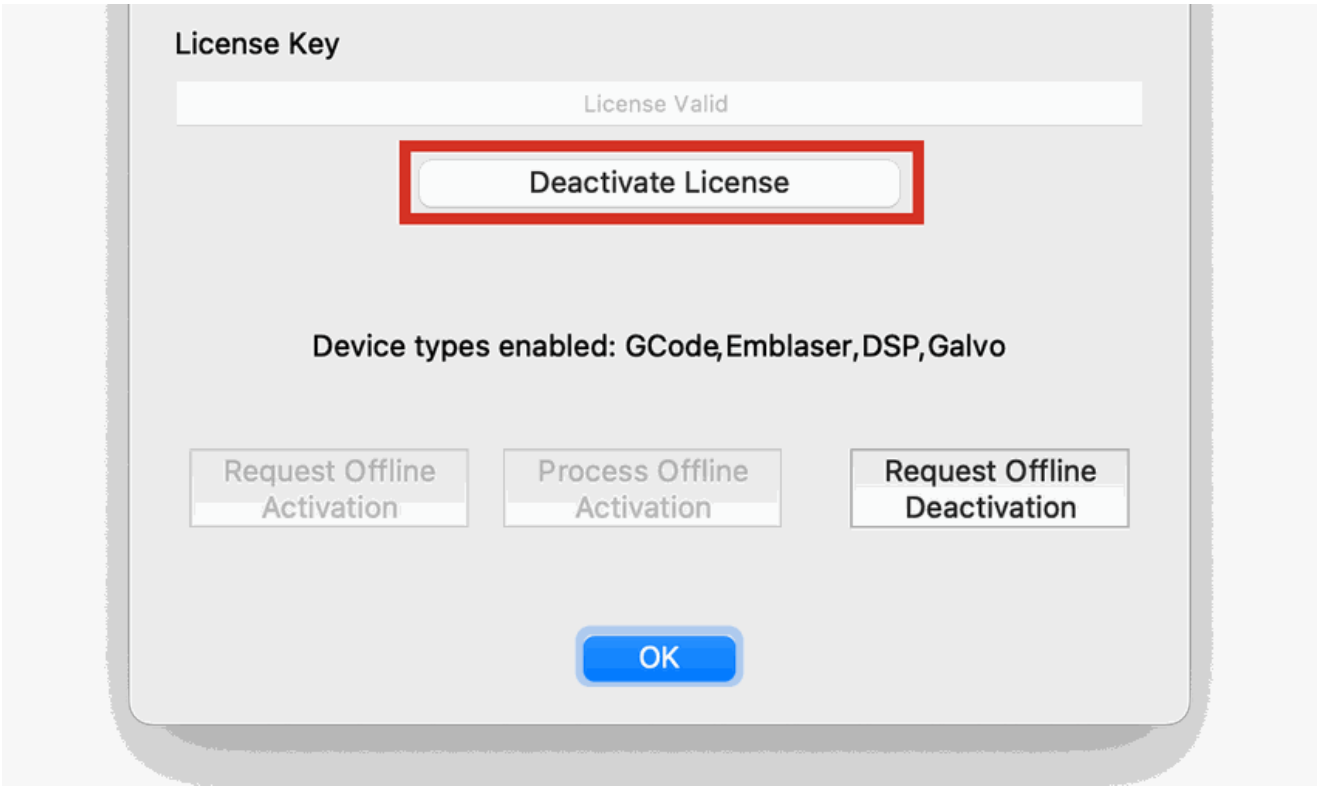
1. Go to https://lightburn.customer.cryptlex.com/.
2. Click **Forgot Password**.
3. Enter the email address where you received your license key, and click **Recover Password**.
4. You will receive an email from Cryptlex with a password reset link.
5. Follow the link, choose your new password, then log in.

> **Note**
>
> If you do not see an email from Cryptlex, check your Promotions or Spam folders — the password reset email sometimes ends up there. If you still cannot find it, your email provider might be blocking the email from ever reaching you. Contact us at support@lightburnsoftware.com to have us reassign your key to an address under a different provider. Gmail accounts work well.

Removing an Old Computer From Your Key

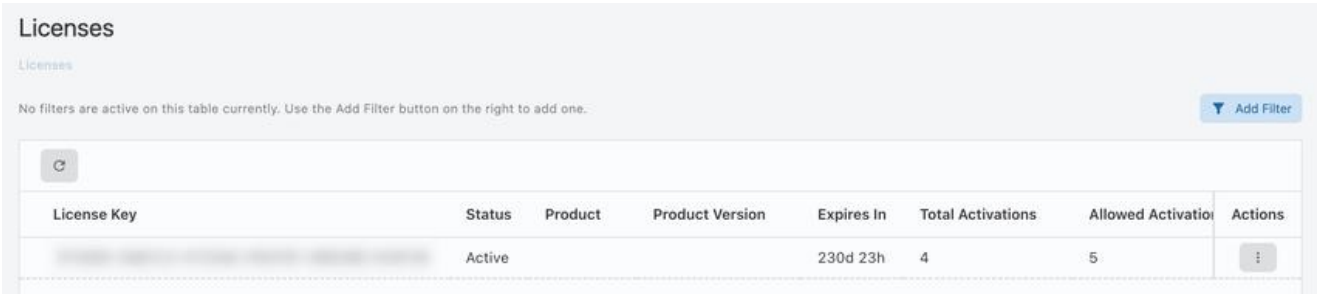
If you still have access to the computer you wish to remove from your license:

1. Open LightBurn on that computer and go to Help → License Management.
2. Click **Deactivate License**.

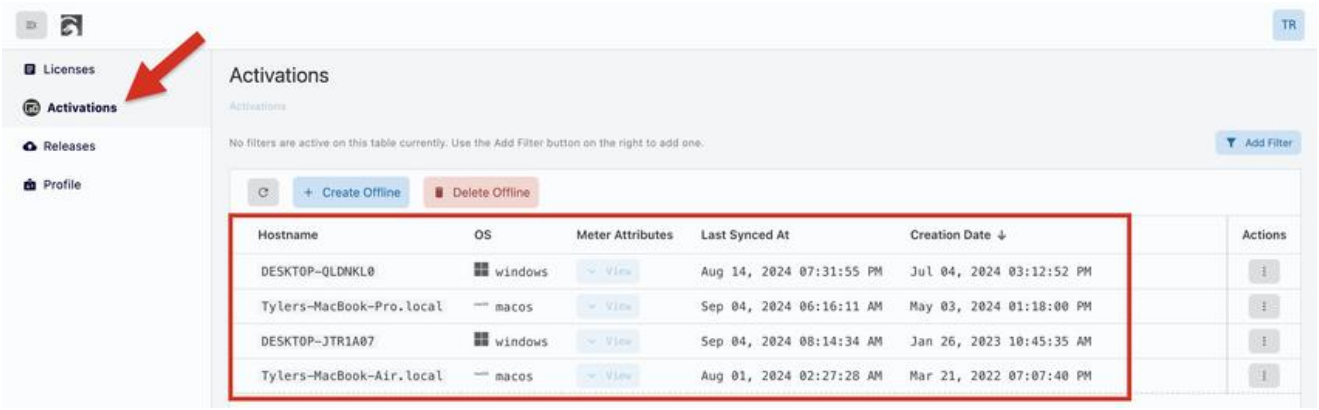


If you no longer have access to the computer you wish to remove:

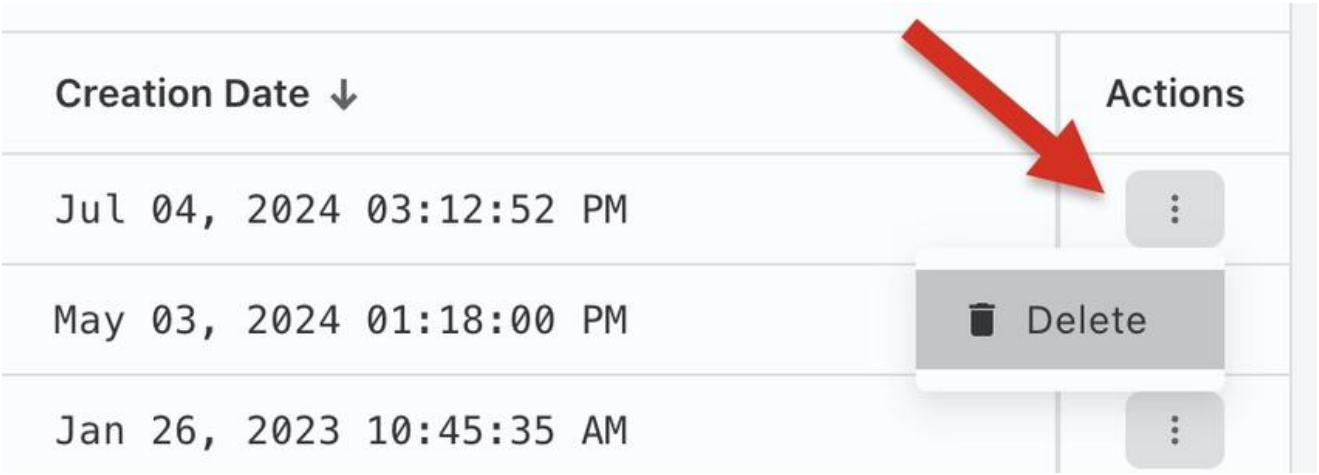
1. Log in to the License Portal. On the main screen you will see any licenses associated with your email address.



2. Click on **Activations** in the left sidebar to view a list of computers assigned to your license key, when they were last used (**Last Synced At**), and when they were first activated (**Creation Date**).



3. Click the three dot icon to the right of any computer in the list, to show the option to **Delete** that computer, removing the license from the computer, and freeing up the license seat for use on another machine.



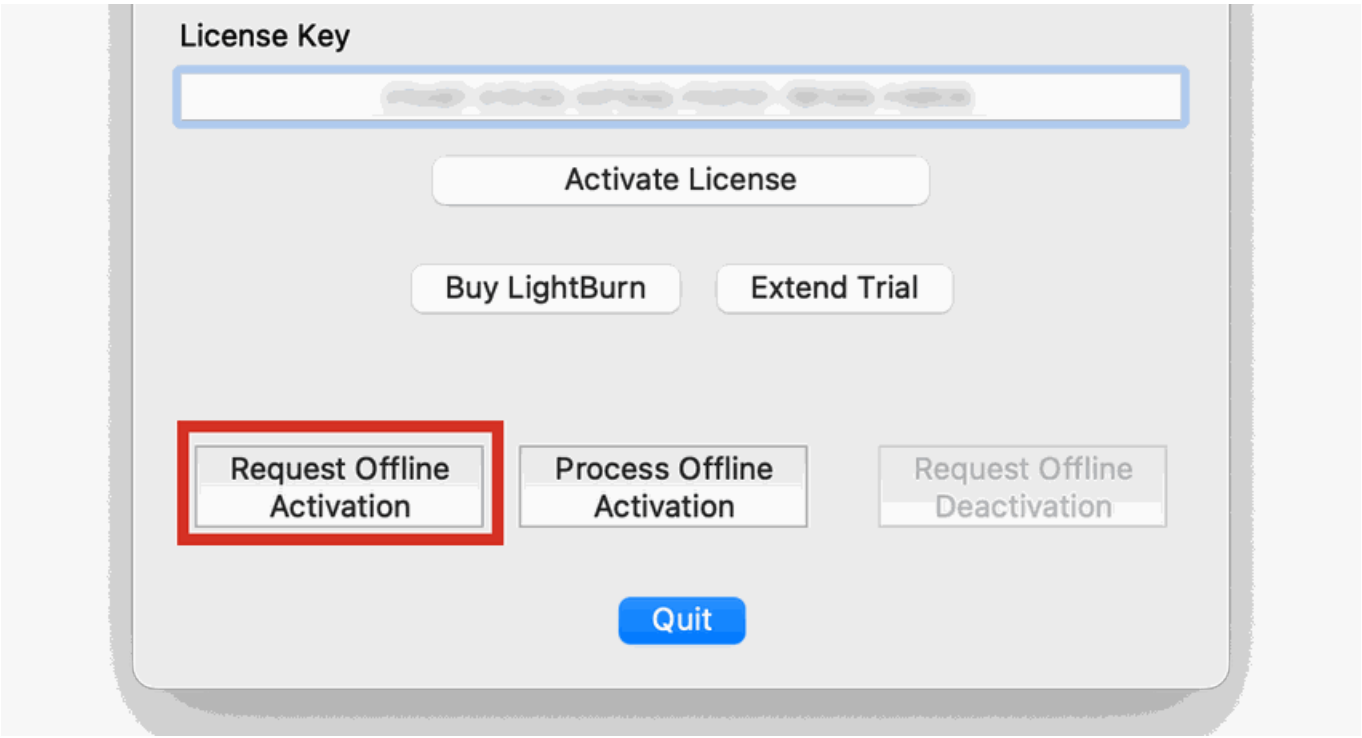
Processing an Offline Activation

Purchased LightBurn licenses — but not free trials — can be activated offline. If your computer can be connected to the internet, even temporarily, this process is not necessary.

Follow the steps below to activate LightBurn on a computer without access to the internet (offline).

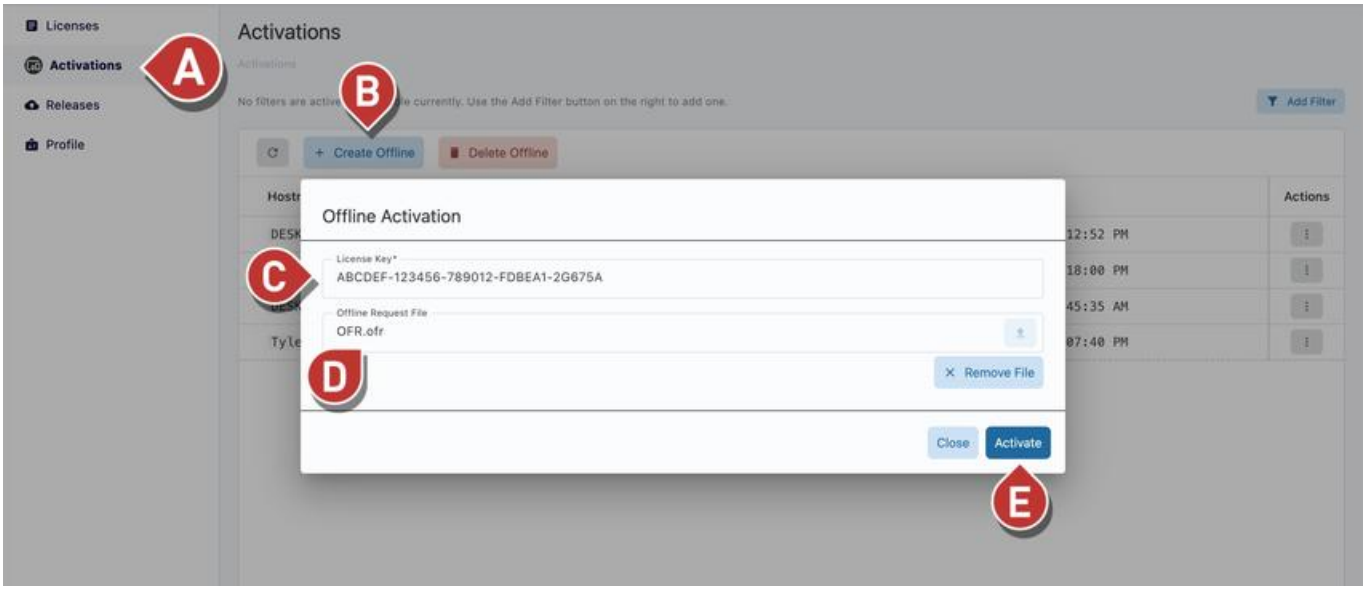
Step 1: Generate a request file

1. Open LightBurn on the offline computer.
2. Go to **Help** → **License Management** to open the **License Management** window.
3. Enter your license key in the License Management window.
4. Click **Request Offline Activation**.
5. Save and copy the generated `.ofr` file to a flash drive or other removable storage.



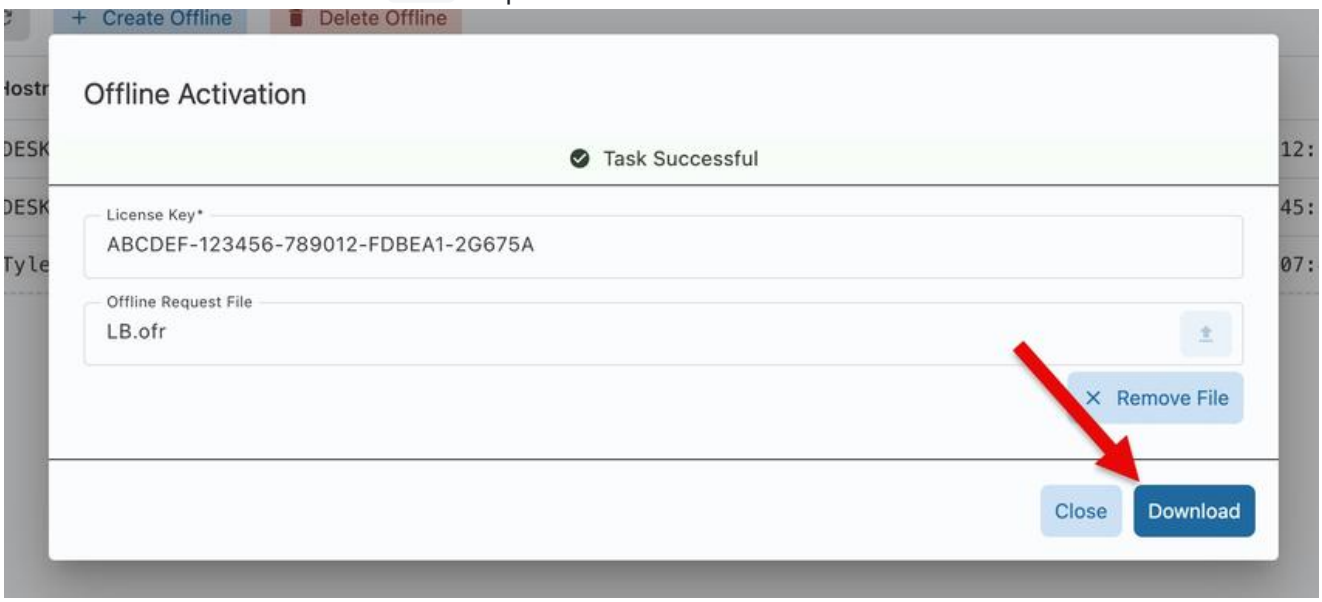
Step 2: Upload the request file to the License Portal

1. Bring the `.ofr` file from Step 1 to a computer with an internet connection.
2. On this computer, open the License Portal and log in. If you're logging into the portal for the first time, see Accessing the License Portal for more instructions.
3. While viewing your license in the portal, go to **Activations** in the left navigation bar.
4. Click on **Create Offline**.
5. Paste your key into the **License Key** field.
6. Click in the **Offline Request** field to open your computer's file browser, navigate to the `.ofr` file you saved to a flash drive, and select it.
7. Click **Activate**. Keep the window open for the next step.



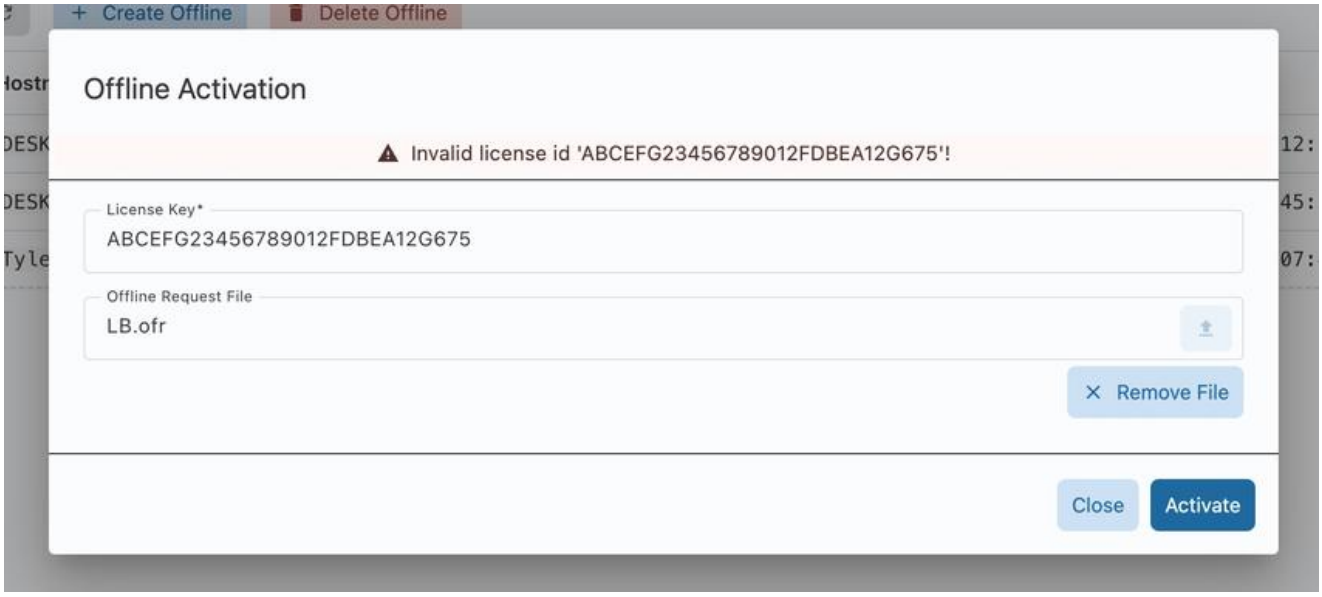
Step 3: Complete activation on the offline computer

1. If you entered the information correctly, you'll see a message saying **Task Successful**. Click **Download** to download the `.dat` response file.

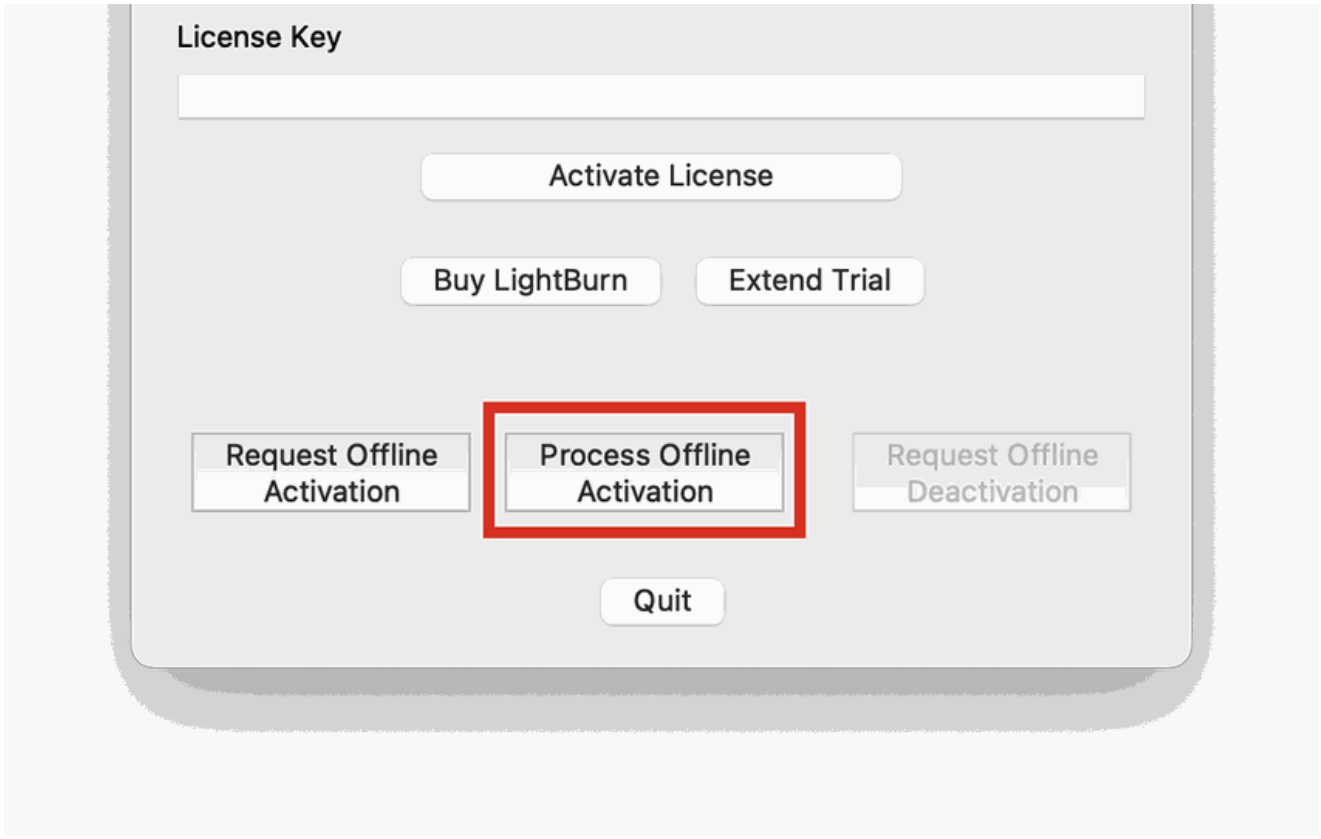


2. If you see an **Invalid license id** message instead, you'll need to repeat the process. Take care to enter your exact license — including hyphens — into LightBurn and the field in the Offline Activation window. Copying and pasting the key directly from your original order email is the best

way to ensure accuracy.



3. Save the file to the flash drive and transfer it back to the offline computer.
4. On the offline computer, in LightBurn's **License Management** window, click **Process Offline Activation**.



5. Navigate to the location you saved the `.dat` response file to in Step 2 and select it, completing the Offline Activation process.

It is not possible to complete this process for a Trial ID. LightBurn must connect to the internet once to activate a trial, after which you are free to take your computer back offline.

If you're not comfortable performing all of the above steps yourself, email us at support@lightburnsoftware.com with the `.ofr` file and license key, and we will process the file and return the `.dat` response file for you to use to complete the activation.

Educational and Volume Licensing

A LightBurn license key comes with 3 seats by default, meaning that it can be installed and run on up to three computers. If you need more than 3 seats, you can buy additional seats at a reduced price. Keep reading for more information, or use the Quote Calculator on our website for volume pricing and purchase links.

For more information on how the LightBurn license works, see our License Info page.

To get started, download our software here and start an unrestricted 30 day free trial.

For more information on purchasing additional seats and a calculator to determine license costs, keep reading.

Volume Licensing

By default, when activating a LightBurn license, the software will be activated for the user account and computer it's activated on, meaning that separate user accounts on the same computer would need to activate a LightBurn license independently.

LightBurn offers two advanced licensing setup options — System Locked and Floating licenses — to accommodate installation in shared spaces such as school computer labs.

System Locked Licenses

A System Locked license is locked to the computer it's activated on, allowing all users on the same computer to use the same license without sharing login info.

Floating Licenses

A Floating license is managed by our license server, allowing the license to be allocated when a LightBurn session is started and released when the session is closed. With Floating licenses, you pay for the number of licenses that will be used at the same time, rather than the number of computers LightBurn will be installed on.

- This is intended for environments such as schools, where LightBurn might be installed on a large number of computers, but only used by a class of 30 at any given time.
- Your key must be specially tagged as a Floating license to use this setup option.
- Floating licenses have a 10 seat minimum and require constant internet connection.
- Floating licenses must be installed according to our directions.

See our Quote Calculator for pricing and purchasing links for both standard and floating licenses.

For setup information, please see System Locked License Setup or Floating License Setup.

System Locked and Floating License Setup

System Locked and Floating licenses are advanced licensing setup options, often required for installations of LightBurn in shared spaces such as school computer labs.

A System Locked license is locked to the computer it is activated on, allowing all users on the same computer to use the same license without sharing login info. Any LightBurn key can be set up as System Locked.

A Floating license is managed by our license server, allowing the license to be allocated when a LightBurn session is started and released when the session is closed. This is intended for environments such as school computer labs where a greater number of computers will need to be activated than instances of LightBurn will be used at once. Your key must be specially tagged as a Floating type in order to utilize this setup option.

See our Educational and Volume Licensing page or contact us via email if you would like to learn more about volume licensing options.

System Locked License Setup

Windows

1. Make sure LightBurn is installed and activated as normal, then close out of the program, or the rest of the process will not work.
2. Open up a **Command Prompt** window:
    1. Click the **Start** menu, then type "CMD".
    2. Windows will automatically find the **Command Prompt** app as the best match. Right-click it and select **Run as administrator**.
3. Type the following into the command prompt and press `Enter`:

    ```
    cd "C:\Program Files\LightBurn”
    ```

4. Next, type the following into the command prompt, replacing YOUR_LICENSE_KEY with your actual license key, and press `Enter`:

    ```
    LightBurn -l YOUR_LICENSE_KEY
    ```

> **Note**
>
> Enter a lowercase letter L, and not the number 1.

Mac

1. Make sure LightBurn is installed and activated as normal, then close out of the program, or the rest of the process will not work.

2. Open up a **Terminal** window.
    1. Press `Cmd`+`Space` then type "Terminal".
    2. Click Terminal to open it.
3. Type the following into the Terminal window and press `Enter`.

```
cd /Applications/LightBurn.app/Contents/MacOS
```

4. Next, type the following into the Terminal window, replacing YOUR_LICENSE_KEY with your actual license key, and press `Enter`:

```
sudo ./LightBurn -l YOUR_LICENSE_KEY
```

> **Note**
>
> Enter a lowercase letter L, and not the number 1.

Once you've completed this process, all users on the same computer will be able to use LightBurn without activating the key individually.

Floating License Setup

Activation

In order to set up a Floating license, first follow the above instructions on setting up a System Locked license.

> **Tip**
>
> Once you've run the command line process successfully on one system, you can copy the "lightburn.ldata" file from the LightBurn program folder on that system to the same location on other systems you want to use the license, so you can avoid having to type the command line on the rest.

Network Restrictions

LightBurn will need to maintain a constant connection with the external license server to verify licenses. If you are experiencing difficulties with a license not being verified, your network may be restricting LightBurn's access.

To ensure LightBurn can contact the license server, whitelist **api.cryptlex.com** for port **443**.

It is also possible to configure LightBurn to access the network through a proxy. You can append the proxy connection address using -p ProxyString.

The proxy format is:

```
[protocol://][username:password@]machine[:port]
```

Examples of valid proxy strings:

```
http://127.0.0.1:8000/
http://user:pass@127.0.0.1:8000/
socks5://127.0.0.1:8000/
```

Troubleshooting

The most common error we see in this process is running the commands without administrator permissions. If you just run it in user mode, it doesn't have permission to create a file in the Program Files folder.

- Open the Command Prompt in administrator mode by right-clicking on the Start Menu entry and selecting **Run as administrator** when opening it.
- On macOS, make sure to include **sudo** when entering the command to activate the key and enter your password correctly.

The second most common error we see is entering the license key or command prompt entries incorrectly. Our instructions show the command line like this:

```
LightBurn -l YOUR_LICENSE_KEY
```

That is "LightBurn", followed by a minus sign and a lower-case letter L, a space, then your license key (without the parentheses around it).

Here is an example entry using a fake key:

```
LightBurn -l ABCDEF-123456-789012-FDBEA1-2G675A
```

Enterprise Security FAQ

This page contains answers to common questions we receive from corporate or educational IT departments prior to purchasing. If your question is not answered here, please email support for further assistance.

Is LightBurn a cloud service? Does it require an internet connection?

LightBurn is a software package that runs entirely on the local computer. An internet connection is not required, except to activate a free trial.

Does LightBurn utilize an internet connection in any way?

If available, LightBurn utilizes an internet connection for three purposes:

- License activation and deactivation. If this is not an option, you can complete an offline activation instead.
- Checking for updates to the software and built-in links. This check takes place once a day when the software is opened, and accesses LightBurn's servers as well as a publicly accessible GitHub repository.
- If you choose to use the Web Integration for Cuttle, an internet connection is used to connect to Cuttle. LightBurn does not receive or send any additional information through the web integration.

LightBurn does not connect to the internet for any other purposes.

How does LightBurn use, collect, and share personal information?

At present, we do not collect or process personal information considered to be "Sensitive Information", "Sensitive Personal Information", or "special category data" in connection with our services.

For customers who license our software or make other purchases, we collect the following information:

- Name and contact information, including company affiliation.
- Order information.
- Payment information, including credit card account or bank account and routing information.

We use this information for the following purposes:

- To process payments and generate license keys.
- Service analysis, security, and administration.

Information is collected directly from the individual, and may be shared with our payment processor, our license key processor, and our sales tax remittance processor. It may also be shared with the following:

- **Processors/service providers:** We share your personal information with third parties that provide services to us. We engage these kinds of third parties with contracts that require them to use your personal information only for the purpose of delivering the services for which we have engaged the third party and as required by law. These kinds of third parties provide website hosting, business, professional, administrative, or technical support functions for us, such as payment processing, billing, data storage, quality assurance, and marketing.
- **Legal compliance recipients:** We disclose personal information to the courts, the government, law enforcement agencies, litigants, and similar recipients when required by law.
- **Successors:** We may disclose personal information associated with a part of our business to a buyer, potential buyer, or other successor to our business.

Where do you store data?

Your personal information will be stored and processed in the United States, where our servers are located. If we transfer personal information to the United States from another jurisdiction, we will do so in a way that complies with applicable legal requirements.

Does LightBurn back up the software configuration?

Yes, LightBurn's preferences are automatically backed up regularly. All backups are saved to the local computer. Read more in our documentation.

Does LightBurn provide regular alerts for software updates?

Yes. If an internet connection is available, LightBurn checks for updates on a daily basis and will notify users of an update when available.

Does LightBurn require any third party agreements or use third party software or components?

Lightburn utilizies the following third party software/libraries:

- **Potrace** image trace library licensed from IcosaSoft Software Inc
- **OpenCV** Copyright © 2026, Intel Corporation Download License
- **zstd** Copyright © 2016-2026, Yann Collet, Facebook, Inc. Download License
- **Sparrow** Copyright © 2025 Jeroen Gardeyn, KU Leuven Download License
- **jagua-rs** Copyright © 2025 Jeroen Gardeyn and contributors Download License

Where can I access more detailed information on privacy and security?

- LightBurn's Privacy Policy
- LightBurn's End User License Agreement (EULA)
- For additional legal documents such as W9s or accessibility documentation, please email support

LightBurn End User License Agreement

Version June 2026

IMPORTANT – PLEASE READ THIS CAREFULLY

This End User License Agreement ("**EULA**") is between you or the LightBurn approved entity on whose behalf you accept this EULA ("**you**" or "**your**") and LightBurn Software Inc. ("**LightBurn**," "**our**" or "**we**") and applies to your access and use of the version of the LightBurn software made available to you ("**LightBurn Software**"), the unique serial number license key(s) we or our authorized reseller(s) provide to you in connection with such version (each a "**LightBurn Key**") and our documentation or user manuals covering the use of the LightBurn Software and LightBurn Key(s), (the "**Documentation**"). We provide an end user license agreement with each version of our LightBurn Software that we make available. Such end user license agreement covers that version only. If you download or otherwise access multiple versions of our LightBurn Software, you may be presented with multiple end user license agreements. You must agree to each end user license agreement to use that version of the LightBurn Software and associated LightBurn Key and Documentation. This EULA covers the version of the LightBurn Software which is downloadable or otherwise made available with this EULA (including if delivered via an email or other link to you) and associated LightBurn Key and Documentation (for this EULA, the "**LightBurn Software Materials**").

LIGHTBURN IS WILLING TO LICENSE THE LIGHTBURN SOFTWARE MATERIALS TO YOU ONLY ON THE CONDITION THAT YOU ACCEPT ALL OF THE TERMS IN THIS EULA. YOU ACCEPT AND AGREE TO BE BOUND BY THE TERMS OF THIS EULA BY DOWNLOADING, INSTALLING OR OTHERWISE USING THE LIGHTBURN SOFTWARE MATERIALS OR OTHERWISE ACCEPTING THIS EULA. IF YOU ACCEPT THIS EULA ON BEHALF OF ANOTHER PERSON, COMPANY, OR OTHER LEGAL ENTITY, YOU REPRESENT AND WARRANT THAT (A) YOU HAVE FULL AUTHORITY TO BIND THAT PERSON, COMPANY OR LEGAL ENTITY TO THIS EULA, (B) YOU HAVE PROVIDED TO LIGHTBURN THE NAME OF SUCH PERSON, COMPANY, OR LEGAL ENTITY THAT WILL BE USING THE LIGHTBURN SOFTWARE MATERIALS, AND (C) UPON RECEIPT OF SUCH NAME, LIGHTBURN HAS NOT DENIED YOU ACCESS TO THE LIGHTBURN SOFTWARE MATERIALS. YOU REPRESENT THAT YOU ARE OF SUFFICIENT LEGAL AGE IN YOUR JURISDICTION OR RESIDENCE TO USE OR ACCESS THE LIGHTBURN SOFTWARE MATERIALS AND TO ENTER INTO THIS EULA. IF YOU DO NOT AGREE TO THIS EULA, YOU SHOULD CEASE ACCESSING OR USING THE LIGHTBURN SOFTWARE MATERIALS, AND, WHERE APPLICABLE, CLICK THE "I DO NOT AGREE" BUTTON BELOW.

This EULA incorporates by reference all terms in other uniform resource locators (URLs) herein, as such terms are modified by us from time to time.

Our Privacy Policy, which can be found at https://lbrn.info/privacy (the "**Privacy Policy**"), provides details about our privacy practices and information about our processing of personal information. You acknowledge you have reviewed the Privacy Policy and consent to the processing of your personal information as described in it. If there is a conflict between the main body of this EULA and the Privacy Policy with respect to use and disclosure of your personal information, the terms of the Privacy Policy controls.

We may change this EULA, the Privacy Policy or other terms in URLs at any time. By continuing to access or use the LightBurn Software Materials you agree to be bound by the most recent version of such terms. If we materially change any such terms, we will let you know by placing a notice within the tool or through other reasonable methods.

The disclaimers, exclusions, and limitations of liability under this EULA will not apply to the extent prohibited by applicable law. Some jurisdictions do not allow the exclusion of implied warranties or the exclusion or limitation of incidental or consequential damages or other rights, so those provisions of this EULA may not apply to you.

1. License

The LightBurn Software Materials are licensed not sold to you.

1.1. Permitted Use.

We offer design-only licenses and commercial licenses. Each type of license has different use rights. You are permitted to use the LightBurn Software Materials under the applicable license you purchase only as follows (each a "**Permitted Use**"): (a) with any of the compatible computer and laser and laser-cutting device(s) listed at the following link, which may be broken out by tier: https://lbrn.info/devices (each a "**Laser Device**"); (b) if you purchase a LightBurn Key for a design-only license you may create laser and laser-cutting designs, which designs may only be implemented on Laser Device(s) under a commercial license; (c) if you purchase a LightBurn Key for a commercial license, you may create laser and laser-cutting designs and implement designs through use of the LightBurn Software with Laser Device(s); and (d) in compliance with the Documentation, this EULA, and all applicable laws and standards of care. A design-only license does not permit you to implement designs through use of the LightBurn Software with Laser Device(s).

1.2. Permitted Users.

You are permitted to allow access to and use of the LightBurn Software Materials under the applicable license you purchase only as follows (each a "**Permitted User**"): (a) if the LightBurn Software Materials are being used in a household setting, by you or any members of your immediate household (e.g. your spouse, partner); (b) if the LightBurn Software Materials are being used in an educational institution, by you or your students, or any other professionals in such educational institution; or (c) if the LightBurn Software Materials are being used in a maker's space or other business, by any individuals using the maker's space or employed, or under contract, at the business. Your access to and use of the LightBurn Software Materials in connection with the uses in subsections (b) and (c) are subject to you providing the name of the company or legal entity with which you are affiliated. You are responsible for all access to and use of the LightBurn Software Materials, including ensuring that all Permitted Users that access or use the LightBurn Software Materials understand and comply with this EULA.

1.3. Types of Licenses.

You may purchase (including for no fee) a trial license to the LightBurn Software or purchase (for a fee) a LightBurn Key for a regular license to the LightBurn Software, with either license being design-only or commercial. Trial licenses are provided on an AS IS basis for thirty (30) days from when you

download the LightBurn Software. A trial license does not require a LightBurn Key. To convert to a regular license from a trial license you must purchase a LightBurn Key, and, when you do so and implement such LightBurn Key, your license will automatically convert to the applicable license under Section 1.5. At the end of the trial period, if you have not purchased a LightBurn Key to convert to a regular license and implemented it, your license to the LightBurn Software Materials and this EULA will automatically terminate.

1.4. LightBurn Keys.

Except for trial use, you cannot access or use the LightBurn Software without a LightBurn Key. Each LightBurn Key permits the type of license you purchase (design-only or commercial), and will be registered against the email address you provide when you make your purchase (if you purchase through reseller(s), we may not receive your email address until later). You are responsible for the LightBurn Keys you receive and any replacement LightBurn Keys will be handled on a case-by-case basis. The determination whether to replace or reissue LightBurn Keys is in our sole discretion. Please contact us at Support@lightburnsoftware.com to seek replacement or reissuance of LightBurn Keys or to add or change the email address associated with your purchases. Each LightBurn Key initially allows you to install the licensed LightBurn Software on three (3) separate computers (each separate activation is an "**Activation**"). You may de-activate an Activation from any computer and install on a different computer without using more than one Activation. You may purchase additional Activations under your LightBurn Key. Such purchased additional Activations are also subject to this EULA (and the licenses and restrictions on use herein). You may allow any Permitted User to install the licensed LightBurn Software on any compatible computer(s) up to the aggregate number of purchased Activations. Except in connection with the transfer of the license in accordance with Section 1.6 below, the resale of Activations or LightBurn Keys is prohibited unless you have an authorized reseller agreement with us.

1.5. License.

Subject to the terms of this EULA, we grant you, during the term of this EULA, a limited, non-exclusive, non-sublicensable (except use by the applicable Permitted Users is permitted), revocable limited license to (a) download, install and use the LightBurn Software and LightBurn Keys, and (b) use and make copies (so long as all copies have our proprietary notices on them) of the Documentation in connection with your authorized use of the LightBurn Software and LightBurn Keys, in each case for the Permitted Use by the Permitted Users.

1.6. Transferability.

You may request to transfer your license to another individual or entity by contacting LightBurn at support@lightburnsoftware.com and providing contact details about the proposed transferee, including the transferee's email address. LightBurn may approve or deny requested transfers in its sole discretion for any reason, including, but not limited to, prevent fraud, abuse, unauthorized resale activity, Activation sharing, circumvention of licensing restrictions, or other misuse of the LightBurn Software Materials. Any approved transfer is subject to the following: (a) you must transfer all rights associated with the LightBurn Key being transferred and all Activations associated with that LightBurn Key to a single transferee, with no splitting of Activations among multiple transferees (any

attempt to split is void); (b) you must cease all use of the LightBurn Software Materials covered by the transferred license and retain no copies in your possession or control; and (c) the transferee must agree to this EULA by accessing or using the LightBurn Software Materials.

1.7. Safety Warning.

We do not manufacture Laser Devices and so we do not control their safety features. The LightBurn Software Materials are intended for use with laser and laser-cutting Laser Device(s). Such Laser Devices are inherently dangerous. Use of incompatible Laser Device(s) or third party accessories may impact the LightBurn Software's performance. Please read all safety instructions for any Laser Device(s) before and when using the LightBurn Software Materials with them, including but not limited to the LightBurn safety instructions provided at https://docs.lightburnsoftware.com/latest/Safety/. When utilizing the LightBurn Software Materials, you must ensure that Laser Device(s) are not directed to perform functions that may cause injury or property damage, such as, without limitation, by directing a laser to perform functions that could ignite flammable materials, by having untrained people use the Laser Device(s), or by leaving the Laser Device(s) unattended or switched on when not in use. You acknowledge that the primary role of the LightBurn Software Materials in Laser Device(s) is to control the motion of the laser head of a compatible Laser Device(s), and not to control the safety features, if any, integrated into the Laser Device(s). Safety features like thermal shutdown, fire detection, and other critical features are typically hardware-level protections built into the Laser Device(s) and are beyond the control of the LightBurn Software Materials. We disclaim all liability arising from any Laser Device(s).

1.8. Your Obligations.

You acknowledge and agree that you will comply with the manufacturer terms applicable to the Laser Device(s) used in connection with the LightBurn Software Materials. You, and not LightBurn, are responsible for the safety and security of your use of any Laser Device(s). You will indemnify and hold us harmless from any and all claims, losses, liabilities, damages, fines, penalties, costs and expenses (including attorney's fees) arising from or relating to your use or misuse of the LightBurn Software Materials or Laser Device(s).

1.9. Restrictions.

We reserve all rights not granted to you. Except as expressly provided herein, you agree not to, and you will not permit others to: (a) license, sublicense, sell, resell, market, rent, lease, assign, transfer, distribute, transmit, host, outsource, display, perform, disclose or otherwise commercially exploit the LightBurn Software Materials or make the LightBurn Software Materials available to any third party; (b) copy or use the LightBurn Software Materials for any purpose; (c) remove or alter any trademark, logo, copyright or other proprietary notices, legends, symbols or labels in the LightBurn Software Materials; (d) modify, make derivative works of, disassemble, reverse compile or reverse engineer any part of the LightBurn Software Materials (except to the extent applicable laws specifically prohibit such restriction for interoperability purposes, in which case you agree to first contact LightBurn and provide LightBurn an opportunity to create such changes as are needed for

interoperability purposes); (e) attempt to access or derive the source code of the LightBurn Software Materials; or (f) use the LightBurn Software Materials except for the applicable Permitted Use by the applicable Permitted User.

1.10. Third Party Software.

Elements of the LightBurn Software may contain or be derived from materials from third parties ("**Third Party Software**"). Such Third Party Software may be subject to additional terms which, if they exist, will be set forth here: https://lbrn.info/thirdparty. You and the Permitted Users are responsible for regularly reviewing such terms to determine the status of, and for complying with, all such applicable Third Party Software terms and conditions. Your use of the LightBurn Software is contingent on compliance with these additional terms of license (however such rights do not expand the Permitted Use and license grants herein). Subject to the observance of the respective Third Party Software terms (as may be amended from time to time), you and the Permitted Users are authorized to use Third Party Software within the Permitted Use of the LightBurn Software only.

1.11. Third Party Services.

The LightBurn Software Materials may provide access to, facilitate interaction with, or embed certain third-party websites, services, APIs, tools, content, or platforms ("**Third-Party Services**"). LightBurn does not control and is not responsible for the availability, content, functionality, security, privacy practices, transactions, or operations of any Third-Party Services. Your use of any Third-Party Services may be governed by separate terms and you and the Permitted Users are responsible for reviewing and complying with such terms. LightBurn may modify, suspend or discontinue your access to any Third-Party Services at any time without liability.

2. License Management

The LightBurn Software may include certain functionality designed to validate your rights and to enforce your compliance with the license granted herein, including Activations, entitlement validation, fraud prevention, abuse detection, security monitoring, and related licensing infrastructure functions. Such technology may be operated by LightBurn directly or through authorized third-party service providers acting on our behalf. You acknowledge that the LightBurn Software may periodically communicate with LightBurn systems or authorized third-party service providers in order to validate licenses, verify compliance with applicable Activation limitations, maintain the integrity and security of the LightBurn Software Materials and related services, detect fraud or abuse, and enforce this EULA. You acknowledge that we may also apply other reasonable technical or operational measures, including rate limits or abuse prevention controls, in connection with Activations, deactivations, or license transfers in order to prevent fraud, abuse, unauthorized sharing, or circumvention of licensing limitations. You also acknowledge that LightBurn or its authorized service providers may maintain records relating to licenses, Activations, update eligibility, account status, and related operational information for purposes including license administration, fraud prevention, security, customer support, legal compliance, and enforcement of this EULA.

3. Updates

We may from time to time provide patches, bug fixes, updates, upgrades, and other modifications to improve the performance, security, functionality, or compatibility of the LightBurn Software. You will be entitled to receive such updates to the LightBurn Software free from charge for twelve (12) months from the date you purchase a LightBurn Key for a version of the LightBurn Software you license from us or our authorized resellers (which updates are also licensed under and subject to the terms of this EULA). Updates issued after such twelve (12) month period may require the purchase of a new LightBurn Key. Updates that require the purchase of a new LightBurn Key will be licensed under a separate end user license agreement covering such LightBurn Software Materials and this EULA does not apply to such updates.

4. Ownership; Feedback

The LightBurn Software Materials and all worldwide copyrights, trade secrets, and other intellectual property rights therein are the exclusive property of LightBurn and its licensors. There are no implied licenses in this EULA. You may provide LightBurn with suggestions or feedback with respect to the LightBurn Software Materials (“**Feedback**”). You acknowledge that any Feedback is provided on a non-confidential basis and that we may use, disclose, reproduce, license and otherwise exploit such Feedback for any purpose. You will not be entitled to any compensation or attribution for LightBurn’s use of Feedback. LightBurn does not waive any rights to use similar or related ideas previously known to us, developed by our employees, or obtained from other sources.

5. Term and Termination

The trial license has the term set forth in Section 1.3 unless earlier terminated as set forth herein. The regular license under Section 1.5 commences when you download the LightBurn Software if there is no trial license or, if there is a trial license, when that trial license converts to the regular license under Section 1.3, and continues for as long as you use the LightBurn Software Materials (including the first twelve (12) months’ worth of updates to the LightBurn Software, except that use after such twelve (12) month period may result in certain functionality, services, validations, or compatibility features no longer being available), unless this EULA is terminated as set forth in this EULA (as noted above, updates beyond the first twelve (12) months may require a separate end user license agreement for such LightBurn Software Materials). We may suspend or terminate this EULA and your license(s) at any time if you fail to comply with any term(s) of this EULA, with written notice to you. LightBurn reserves the right to investigate any actual or suspected violation of this EULA and you shall reasonably cooperate with any such investigation, including by providing relevant information relating to your use of the LightBurn Software Materials. You may terminate this EULA and your license(s) at any time with written notice to us or by ceasing use of the LightBurn Software Materials and destroying all copies thereof (including when you license a new version of the LightBurn Software under a separate end user license agreement from us). Upon termination of this EULA, your license(s) will terminate and you must stop all use of the LightBurn Software Materials and destroy all copies you and your Permitted Users have. The preamble and Sections 1.7, 1.8, 1.9, 1.10, 1.11, 2, 4, 6, 7, 8, 10, 11, and the corresponding sections in 12 and 13 will remain in effect after any termination of this EULA.

6. Warranty Disclaimer

NOTWITHSTANDING ANYTHING TO THE CONTRARY AND TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, WE PROVIDE THE LIGHTBURN SOFTWARE MATERIALS ON AN “AS-IS” AND “WITH ALL FAULTS” BASIS. WE DISCLAIM ALL WARRANTIES AND CONDITIONS, WHETHER EXPRESS, IMPLIED, OR STATUTORY, INCLUDING THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, TITLE, QUIET ENJOYMENT, ACCURACY, AND NON-INFRINGEMENT OF THIRD-PARTY RIGHTS, AS WELL AS WARRANTIES ARISING UNDER COURSE OF PERFORMANCE, COURSE OF DEALING OR TRADE USAGE. WE DO NOT GUARANTEE ANY SPECIFIC RESULTS FROM THE USE OF THE LIGHTBURN SOFTWARE MATERIALS. WE MAKE NO WARRANTY THAT THE LIGHTBURN SOFTWARE MATERIALS WILL BE UNINTERRUPTED, FREE OF VIRUSES OR OTHER HARMFUL CODE, TIMELY, SECURE, OR ERROR-FREE. CERTAIN FUNCTIONALITY ASSOCIATED WITH THE LIGHTBURN SOFTWARE MATERIALS, INCLUDING LICENSING, ACTIVATION, VALIDATION, UPDATES, FRAUD PREVENTION, SECURITY, CLOUD-CONNECTED FEATURES, OR RELATED SERVICES, MAY DEPEND ON THIRD-PARTY INFRASTRUCTURE OR SERVICE PROVIDERS, AND LIGHTBURN SHALL NOT BE RESPONSIBLE FOR INTERRUPTIONS, DELAYS, OUTAGES, ERRORS, OR FAILURES CAUSED BY SUCH THIRD-PARTY SYSTEMS OR PROVIDERS. YOU USE THE LIGHTBURN SOFTWARE MATERIALS AND LASER DEVICE(S) AT YOUR OWN DISCRETION AND RISK. YOU WILL BE SOLELY RESPONSIBLE FOR (AND WE DISCLAIM) ANY AND ALL LOSS, LIABILITY, OR DAMAGES RESULTING FROM YOUR USE OF THE LIGHTBURN SOFTWARE MATERIALS OR LASER DEVICE(S). THE IMPROPER USE OF THE LIGHTBURN SOFTWARE MATERIALS WITH ANY LASER DEVICE(S) MAY BE DANGEROUS OR PRESENT RISKS OF ELECTRIC SHOCK, LASER DEVICE DAMAGE, FIRE, PROPERTY DAMAGE, PERSONAL INJURY, DEATH AND OTHER HAZARDS. YOU ARE RESPONSIBLE FOR ANY ELECTRIC SHOCK, LASER DEVICE DAMAGE, FIRE, PROPERTY DAMAGE, PERSONAL INJURY, DEATH AND OTHER HAZARDS CAUSED BY USE OR MISUSE OF THE LIGHTBURN SOFTWARE MATERIALS OR LASER DEVICE(S), INCLUDING FAILURE TO SWITCH OFF THE LASER DEVICE(S) WHEN NOT IN USE AND LEAVING THE LASER DEVICE(S) UNATTENDED. WE DO NOT CONTROL THE SAFETY FEATURES OF LASER DEVICE(S) AND HAVE NO RESPONSIBILITY FOR THE LASER DEVICE(S). SOME JURISDICTIONS DO NOT ALLOW THE EXCLUSION OF IMPLIED WARRANTIES, SO THE ABOVE EXCLUSION MAY NOT APPLY TO YOU.

7. Limitation of Liability

IN NO EVENT WILL LIGHTBURN OR ITS SHAREHOLDERS, AFFILIATES, CONTRACTORS, SUPPLIERS, LICENSORS, DIRECTORS, OFFICERS, EMPLOYEES AND AGENTS (“LIGHTBURN PARTIES”) BE LIABLE FOR ANY CONSEQUENTIAL, EXEMPLARY, SPECIAL, PUNITIVE, INDIRECT, INCIDENTAL, TREBLE OR OTHER SIMILAR DAMAGES, INCLUDING ANY DAMAGES FOR LOST DATA OR LOST PROFITS, PROPERTY DAMAGE, DEATH OR PERSONAL INJURY, ARISING FROM OR RELATING TO THIS EULA OR THE LIGHTBURN SOFTWARE MATERIALS OR THEIR USE, EVEN IF LIGHTBURN KNEW OR SHOULD HAVE KNOWN OF THE POSSIBILITY OF SUCH DAMAGES. LIGHTBURN’S TOTAL CUMULATIVE LIABILITY ARISING FROM OR RELATED TO THIS EULA OR THE LIGHTBURN SOFTWARE MATERIALS OR THEIR USE, WHETHER IN CONTRACT OR TORT OR OTHERWISE, WILL NOT EXCEED (a) FOR TRIAL LICENSES, ZERO DOLLARS, OR (b) FOR REGULAR LICENSES, THE RETAIL PRICE OF THE APPLICABLE REGULAR LIGHTBURN SOFTWARE LICENSE LISTED ON OUR WEBSITE ON THE DATE OF YOUR PURCHASE OF THE REGULAR LIGHTBURN SOFTWARE LICENSE, WHETHER SUCH LIGHTBURN SOFTWARE LICENSE IS PURCHASED ON OUR WEBSITE OR FROM ONE OF OUR AUTHORIZED RESELLER(S). THIS LIMITATION

WILL NOT BE INCREASED BY THE EXISTENCE OF MORE THAN ONE END USER LICENSE AGREEMENT, INCIDENT OR CLAIM. NOTHING IN THIS EULA AND IN PARTICULAR WITHIN THIS “LIMITATION OF LIABILITY” CLAUSE WILL ATTEMPT TO EXCLUDE LIABILITY THAT CANNOT BE EXCLUDED UNDER APPLICABLE LAW. YOU HAVE NO CLAIM AGAINST ANY OF THE LIGHTBURN PARTIES OTHER THAN LIGHTBURN IN CONNECTION WITH THIS EULA.

8. Confidentiality

“**Confidential Information**” means the LightBurn Software Materials and all other confidential or proprietary information disclosed or made accessible to you by LightBurn, except for information which you can demonstrate is available to the public prior to disclosure to you or becomes available to the public by publication or otherwise without your fault. You will not use or disclose any Confidential Information except as expressly authorized by this EULA. You will protect all Confidential Information using the same degree of care which you use to protect your own confidential information, but in no event less than a reasonable degree of care. You will take prompt and appropriate action to prevent unauthorized use or disclosure of Confidential Information. If you are legally compelled to disclose any Confidential Information, you will provide LightBurn prompt written notice of such requirement so that an appropriate protective order or other relief may be sought. The breach, or threatened breach, of any provision of this Section, or the breach, or threatened breach, of license rights or restrictions or other violation, infringement, or misuse of LightBurn Software Materials or other Confidential Information may cause irreparable harm to LightBurn without an adequate remedy at law. Upon any such breach, violation, infringement, misuse, or threat thereof, LightBurn will be entitled to seek injunctive relief to prevent you from commencing or continuing any action constituting such breach, violation, infringement, misuse, or threat thereof, or other equitable relief, without having to post a bond or other security, and without having to prove the inadequacy of other available remedies. We may do so in any court of competent jurisdiction.

9. For U.S. Government End Users

The LightBurn Software Materials are a “commercial item,” as that term is defined at 48 C.F.R. 2.101 (OCT 1995), and more specifically are “commercial computer software” and “commercial computer software documentation,” as such terms are used in 48 C.F.R. 12.212 (SEPT 1995). Consistent with 48 C.F.R. 12.212 and 48 C.F.R. 227.7202-1 through 227.7202-4 (JUNE 1995). The LightBurn Software Materials are provided to U.S. Government End Users only as a commercial end item and with only those rights as are granted to all other customers pursuant to the terms and conditions herein.

10. Export Compliance

All use of the LightBurn Software Materials by you and your Permitted Users will be in accordance with (a) the Export Administration Act of 1979, as amended (50 U.S.C. app. 2401-2420), and the Export Administration Regulations, 15 C.F.R. Parts 730-774 (“**EAR**”); (b) the Arms Export Control Act, 22 U.S.C. § 2778, and the corresponding International Traffic in Arms Regulations; (c) the economic sanctions, laws and regulations enforced by the U.S. Department of the Treasury’s Office of Foreign Assets Control (“**OFAC**”), 31 C.F.R. Part 500 et seq., and the U.S. Department of State’s Office of

Terrorist Financial and Economic Sanctions Policy; and (d) the anti-boycott regulations, guidelines, and reporting requirements under the EAR and Section 999 of the Internal Revenue Service Code (collectively, "**U.S. Sanctions and Export Control Laws**"). In particular, you agree that you and your Permitted Users are: (i) not located in, headquartered in, or incorporated under the laws of any country or territory subject to U.S. territorial economic sanctions or anti-terrorism restrictions under the EAR, including Belarus, Cuba, Iran, North Korea, Russia, or the Crimea, Donetsk, or Luhansk regions of Ukraine (collectively "**Sanctioned Locations**"); and (ii) not a person identified on U.S. or other applicable government restricted party lists, including but not limited to the Specially Designated Nationals and Blocked Persons ("**SDN**") List maintained by OFAC or the Entity List or Denied Persons List maintained by the U.S. Department of Commerce's Bureau of Industry and Security (collectively, "**Restricted Parties**"), or owned or controlled by or acting on behalf of, any Restricted Party or any party located in, headquartered in or incorporated under the laws of any Sanctioned Locations. You and your Permitted Users will not use or divert the LightBurn Software Materials as follows: (A) to countries for which a specific export license is required; (B) to Restricted Parties or Sanctioned Locations; (C) for activities related to weapons proliferation, such as nuclear, chemical, or biological weapons or missile technology; (D) for end-uses in violation of the EAR relating to a military or military intelligence end-use or end-user, including for incorporation into any military item, for the use, development, or production of military items, for any activity that supports or contributes to the operation, installation, maintenance, repair, overhaul, refurbishing, development, or production of military items, or for use by a military end-user such as the national armed services (army, navy, marine, air force, or coast guard), the national guard and national police, government intelligence or reconnaissance organizations, or any person or entity whose actions or functions are intended to support military end uses in violation of the EAR; or (E) in a manner otherwise inconsistent with end-use and end-user restrictions imposed by the U.S. Sanctions and Export Control Laws implemented by 15 C.F.R. Part 744.

11. General

11.1. Language.

The official version of this EULA is in English.

11.2. Waivers.

All waivers by us will be effective only if in writing. Any waiver or failure by us to enforce any provision of this EULA on one occasion will not be deemed a waiver of any other provision or of such provision on any other occasion.

11.3. Severability.

If any provision of this EULA is held by a court of competent jurisdiction to be illegal, invalid or unenforceable, that will not affect the legality, validity or enforceability of any other provision of this EULA. Any such unenforceable provision will be deemed to be amended to the minimum extent required to accomplish the objectives of such provision. If the illegality, invalidity or unenforceability cannot be cured by such amendment, and such provision materially alters the rights, benefits, or obligations of a party, such party shall have the right to terminate this EULA upon written notice to the other party.

11.4. Governing Law; Venue.

This EULA and any matters arising in connection with this EULA will be governed, interpreted, and construed in accordance with the laws of the State of Delaware, USA without regard to the principles of conflicts of laws. Any legal action or proceeding relating to this EULA and any matters arising in connection with this EULA will be instituted exclusively in a state or federal court in the State of Delaware, USA except for equitable relief by us under Section 8 (Confidentiality). Except as set forth in Section 8 (Confidentiality), the parties will submit to the exclusive jurisdiction of, and accept that venue is proper in, these courts in any such legal action or proceeding.

11.5. Assignment.

Except as permitted in Section 1.6, neither the rights nor the obligations arising under this EULA are assignable by you except with the prior written consent of LightBurn, and any such attempted assignment will be void and without effect. This Agreement may be assigned, in whole or in part, by us. This EULA is binding on and inures to the benefit of any successors and permitted assigns.

11.6. Miscellaneous.

The headings of sections of this EULA are for convenience and are not to be used in interpreting this EULA. This EULA will not be construed as creating a relationship of employment, agency, partnership, joint venture, or any other form of legal association. Neither party has any power to bind the other party or to assume or to create any obligation or responsibility on behalf of the other party. This EULA will not confer any rights or remedies upon any person other than the parties, and their respective successors and permitted assigns and the indemnified parties (with respect to indemnification). All notices and other communications required or permitted under this EULA will be in writing and will be deemed given (a) if to LightBurn, when delivered to 838 Walker Rd Suite 21-2, Dover, DE, 19904; or, (b) if to you, when delivered either by (i) courier service (including via First Class Mail with the United States Postal Service) to the mailing address on file with us, or (ii) e-mail to the e-mail address on file with us. This EULA constitutes the entire agreement between the parties with respect to its subject matter and as of the date of agreement by you supersedes all prior or contemporaneous agreements, negotiations, representations, and proposals, whether written or oral, relating to its subject matter. Except for modifications made by us to this EULA, Privacy Policy or other terms in URLs, no modification of this EULA by you will be effective unless in writing and signed by an authorized representative of LightBurn. The Uniform Computer Information Transactions Act and the United Nations Convention on Contracts for the International Sale of Goods will not apply to this EULA, or any business conducted pursuant to it, regardless of when and however adopted, enacted and amended under the laws of any jurisdiction. In this EULA where we use the words “including” or “include” it means including without limitation or includes without limitation, respectively, and the word “or” means and/or, unless the context dictates otherwise. The parties will execute and deliver any additional contracts, instruments or documents necessary to fully effectuate the terms of this EULA.

12. Variation of EULA Terms for Licensee Based in or Resident of the European Union

12.1. Generally.

This EULA will apply in respect of purchasers of the licenses to the LightBurn Software Materials who, at the time the license for the LightBurn Software Materials is purchased, (in the case of incorporated or unincorporated entities, businesses, institutions and organizations) are based, or (in the case of individuals acquiring the license for the LightBurn Software Materials in their personal capacity) have their main place of residence, in the European Union, subject to the variations set out below in this Section 12.

12.2. Preamble section.

The 3rd, 4th and 5th paragraph in the preamble are replaced in their entirety with the following:

This EULA incorporates by reference all terms in uniform resource locators (URLs) referred to herein. Such additional terms will apply as they may be modified by us from time to time, provided that no such modifications unreasonably limit or place unreasonable conditions on your right to use the LightBurn Software Materials in accordance with this EULA.

Our Privacy Policy which can be found at https://lbrn.info/privacy (the “Privacy Policy”) provides details about our privacy practices and information about our processing of personal information. You acknowledge you have been given access to a copy of the Privacy Policy.

We may change this EULA, the Privacy Policy or other terms in URLs at any time. Any such changes will apply to you in relation to copies of the LightBurn Software Materials to which you already purchased a license before the date of such change only insofar as the change does not unreasonably limit or place unreasonable conditions on your right to use the LightBurn Software Materials in accordance with the terms of this EULA at the time of purchase. All future changes to this EULA, the Privacy Policy or other terms in URLs will apply to you in relation to copies of the LightBurn Software Materials purchased after the date of the change.

12.3. Section 1.6.

The following shall be added to Section 1.6:

Transferability. Where you purchase a perpetual license for the LightBurn Software Materials and the license fees are paid in full in advance, you may transfer the license (on the terms of this EULA) to another person (a “transferee”) without seeking our consent provided that: (a) you must transfer possession of all copies of the LightBurn Software Materials and all Activations to a single transferee and must not retain any copies of LightBurn Software Materials (including any back-up copy of the LightBurn Software) in your possession or control; (b) you must transfer all of the rights that you purchased in connection with the copy of the LightBurn Software Materials, including all Activations, to a single transferee; accordingly, user rights under this EULA including the Activations purchased with the license (including where you purchase a license for use by multiple users) cannot be split between you and a transferee or amongst several transferees; (c) the transferee will not have the benefit of and will not be entitled to receive from us any maintenance or support services, software updates, warranty services or any other services that you may be entitled to receive from us (including updates or other services you are entitled to receive from us under Section 3 or any other

contract and including any warranties or product guarantees that you may purchase or receive in connection with the LightBurn Software Materials) and you may not transfer your right to receive such services to any transferee; and (d) you must provide this EULA to the transferee, and the transferee will be deemed to have agreed to this EULA by accessing or using any of the LightBurn Software Materials. Except as aforesaid (or where we provide our consent to a transfer in accordance with Section 1.6), your license is transferable only if you sell, assign or otherwise transfer the Laser Device with which the LightBurn Software is used and subject to the following: (i) you must transfer all LightBurn Software Materials and all Activations to the same transferee with the Laser Device being transferred, in other words you cannot split up Activations between transferees; and (ii) you must provide this EULA to the transferee, and the transferee is deemed to have agreed to this EULA by accessing or using any of the LightBurn Software Materials. If you do not comply with the foregoing, we may consider the transfer void.

12.4. Section 1.7.

The last sentence of Section 1.7 is replaced with the following:

Subject only to the last sentence of Section 7 below, we disclaim all liability arising from any Laser Device(s).

12.5. Section 1.8.

The following sentence is added at the end of Section 1.8:

The foregoing is subject to and will not derogate from the last sentence of Section 7 below.

12.6. Section 1.9.

The opening words in the second sentence in Section 1.9 are replaced with the following:

Except as expressly provided herein, or as may be permitted under Articles 5(2), 5(3) or 6 of Directive 2009/24/EC of the European Parliament and the Council on the legal protection of computer programs, or under any national legislation implementing such provisions in the European Union member state in which you are based or where you are resident (as such legislation may be varied or re-enacted from time to time), you agree not to, and you will not permit others to:

The remaining provisions of Section 1.9 in sub-sections (a) to (f) are unchanged.

12.7. Section 7.

The following sentence is added at the end of Section 7:

Anything to the contrary herein notwithstanding, nothing in this EULA or in any terms incorporated by reference into this EULA will exclude or limit our liability: (i) for death or personal injury caused by our negligence; (ii) for our fraud or fraudulent misrepresentation; or (iii) for any matter for which it would be illegal for us to exclude or attempt to exclude our liability.

12.8. Section 11.1.

Section 11.1 is replaced in its entirety with the following:

Language. The official version of this EULA is in English, except where you are purchasing a license to the LightBurn Software Materials as an individual consumer, in which case, if a version of this EULA is provided to you in one of the official languages of the country where you have your main place of residence, that language version will be the official version of this EULA.

12.9. Section 11.4.

The following sentence is added at the end of Section 11.4:

The foregoing notwithstanding, if you are an individual and you purchase the license for the LightBurn Software Materials as a consumer, outside your trade or profession, you have the right to bring proceedings against us in the courts of the country where you are domiciled (or, at your choice, if we have a domicile in a member state of the European Union, in the courts of that country, or in a state or federal court in the State of Delaware, USA) and we may only bring proceedings against you in the court of the country where you are domiciled.

13. Variation of EULA Terms for Licensee Based in or Residents of The United Kingdom

13.1. General.

This EULA will apply in respect of purchasers of the licenses to the LightBurn Software Materials who, at the time the license for the LightBurn Software Materials is purchased, (in the case of incorporated or unincorporated entities, businesses, institutions and organizations) are based, or (in the case of individuals acquiring the license for the LightBurn Software Materials in their personal capacity) have their main place of residence, in the United Kingdom, subject to the variations set out in this Section 13.

13.2. Preamble Section.

The 3rd, 4th and 5th paragraph in the preamble are replaced in their entirety with the following:

This EULA incorporates by reference all terms in uniform resource locators (URLs) referred to herein. Such additional terms will apply as they may be modified by us from time to time, provided that no such modifications unreasonably limit or place unreasonable conditions on your right to use the LightBurn Software Materials in accordance with this EULA.

Our Privacy Policy which can be found at https://lbrn.info/privacy (the “Privacy Policy”) provides details about our privacy practices and information about our processing of personal information. You acknowledge you have been given access to a copy of the Privacy Policy.

We may change this EULA, the Privacy Policy or other terms in URLs at any time. Any such changes will apply to you in relation to copies of the LightBurn Software Materials to which you already purchased a license before the date of such change only insofar as the change does not unreasonably limit or place unreasonable conditions on your right to use the LightBurn Software Materials in accordance with the terms of this EULA at the time of purchase. All future changes to this EULA, the Privacy Policy or other terms in URLs will apply to you in relation to copies of the LightBurn Software Materials purchased after the date of the change.

13.3. Section 1.6.

The following shall be added to Section 1.6:

Transferability. Where you purchase a perpetual license for the LightBurn Software Materials and the license fees are paid in full in advance, you may transfer the license (on the terms of this EULA) to another person (a “transferee”) without seeking our consent provided that: (a) you must transfer possession of all copies of the LightBurn Software Materials and all Activations to a single transferee and must not retain any copies of LightBurn Software Materials (including any back-up copy of the LightBurn Software) in your possession or control; (b) you must transfer all of the rights that you purchased in connection with the copy of the LightBurn Software Materials, including all Activations, to a single transferee; accordingly, user rights under this EULA including the Activations purchased with the license (including where you purchase a license for use by multiple users) cannot be split between you and a transferee or amongst several transferees; (c) the transferee will not have the benefit of and will not be entitled to receive from us any maintenance or support services, software updates, warranty services or any other services that you may be entitled to receive from us (including updates or other services you are entitled to receive from us under Section 3 or any other contract and including any warranties or product guarantees that you may purchase or receive in connection with the LightBurn Software) and you may not transfer your right to receive such services to any transferee; and (d) you must provide this EULA to the transferee, and the transferee will be deemed to have agreed to this EULA by accessing or using any of the LightBurn Software Materials. Except as aforesaid (or where we provide our consent to a transfer in accordance with Section 1.6), your license is transferable only if you sell, assign or otherwise transfer the Laser Device with which the LightBurn Software is used and subject to the following: (i) you must transfer all LightBurn Software Materials and all Activations to the same transferee with the Laser Device being transferred, in other words you cannot split up Activations between transferees; and (ii) you must provide this EULA to the transferee, and the transferee is deemed to have agreed to this EULA by accessing or using any of the LightBurn Software Materials. If you do not comply with the foregoing, we may consider the transfer void.

13.4. Section 1.7.

The last sentence of Section 1.7 is replaced with the following:

Subject only to the last sentence of Section 7 below, we disclaim all liability arising from any Laser Device(s).

13.5. Section 1.8.

The following sentence is added at the end of Section 1.8:

The foregoing is subject to and will not derogate from the last sentence of Section 7 below.

13.6. Section 1.9.

The opening words in the second sentence in Section 1.9 are replaced with the following:

Except as expressly provided herein, or as may be permitted under sections 50A, 50B, 50BA and 296A of the Copyright, Designs and Patents Act 1988 (as such legislation may be varied or re-enacted from time to time), you agree not to, and you will not permit others to:

The remaining provisions of Section 1.9 in sub-sections (a) to (f) are unchanged.

13.7. Section 7.

The following sentence is added at the end of Section 7:

Anything to the contrary herein notwithstanding, nothing in this EULA or in any terms incorporated by reference into this EULA will exclude or limit our liability: (i) for death or personal injury caused by our negligence; (ii) for our fraud or fraudulent misrepresentation; or (iii) for any matter which it would be illegal for us to exclude or attempt to exclude our liability.

13.8. Section 11.4.

The following sentence is added at the end of Section 11.4:

The foregoing notwithstanding, if you are an individual and you purchase the license for the LightBurn Software as a consumer outside your trade or profession, you have the right to bring proceedings against us in the courts of the part of the United Kingdom where you are domiciled (or, at your choice, if we have a domicile in any part of the United Kingdom, in the courts of that part of the United Kingdom, or in a state or federal court in the State of Delaware, USA) and we may only bring proceedings against you in the courts of the part of the United Kingdom where you are domiciled.

LightBurn Accessibility Statement

Accessibility Statement for LightBurn Software Inc.

This is an accessibility statement from LightBurn Software Inc.

Measures to support accessibility

LightBurn Software Inc. takes the following measures to ensure accessibility of LightBurn Software:

- We have received a third party accessibility audit from Deqeue Systems, and have a VPAT available on request.
- We are working on improving accessibility based on the results of the accessibility audit.

Conformance Status

The Web Content Accessibility Guidelines defines requirements for designers and developers to improve accessibility for people with disabilities. It defines three levels of conformance: Level A, Level AA, and Level AAA. LightBurn Software is partially conformant with WCAG 2.1 Level AA. Partially conformant means that some parts of the content do not fully conform to the accessibility standard.

Feedback

We welcome your feedback on the accessibility of LightBurn. Please let us know if you encounter accessibility barriers while using LightBurn or the LightBurn documentation.

- Email: support@lightburnsoftware.com

We try to respond to feedback within 3 business days.

Date

This statement was created on 3 April 2026 using the W3C Accessibility Statement Generator Tool.

Downloads and Resources

Downloads and Resources

Our resources section has files and links we hope you'll find useful.

Art Libraries

On this page, you'll find curated Art Libraries with themed clip art and LightBurn-created artwork for you to use in your projects.

Go to Art Libraries

External Resources

On this page, you'll find lists of tools that we've found useful. We hope you'll enjoy them too.

Go to External Resources

Camera Mounts

Endoscope Camera Mount

We have additional designs coming soon, but for users looking to get started with Camera Print and Cut or other uses for Head-Mounted Cameras, we have an easily adaptable design for an endoscope camera mount.

This design uses a v-shaped groove to hold cameras of various sizes in place with zip ties, and can be mounted to your laser with zip ties, double-sided tape, or both.

Downloads and Resources



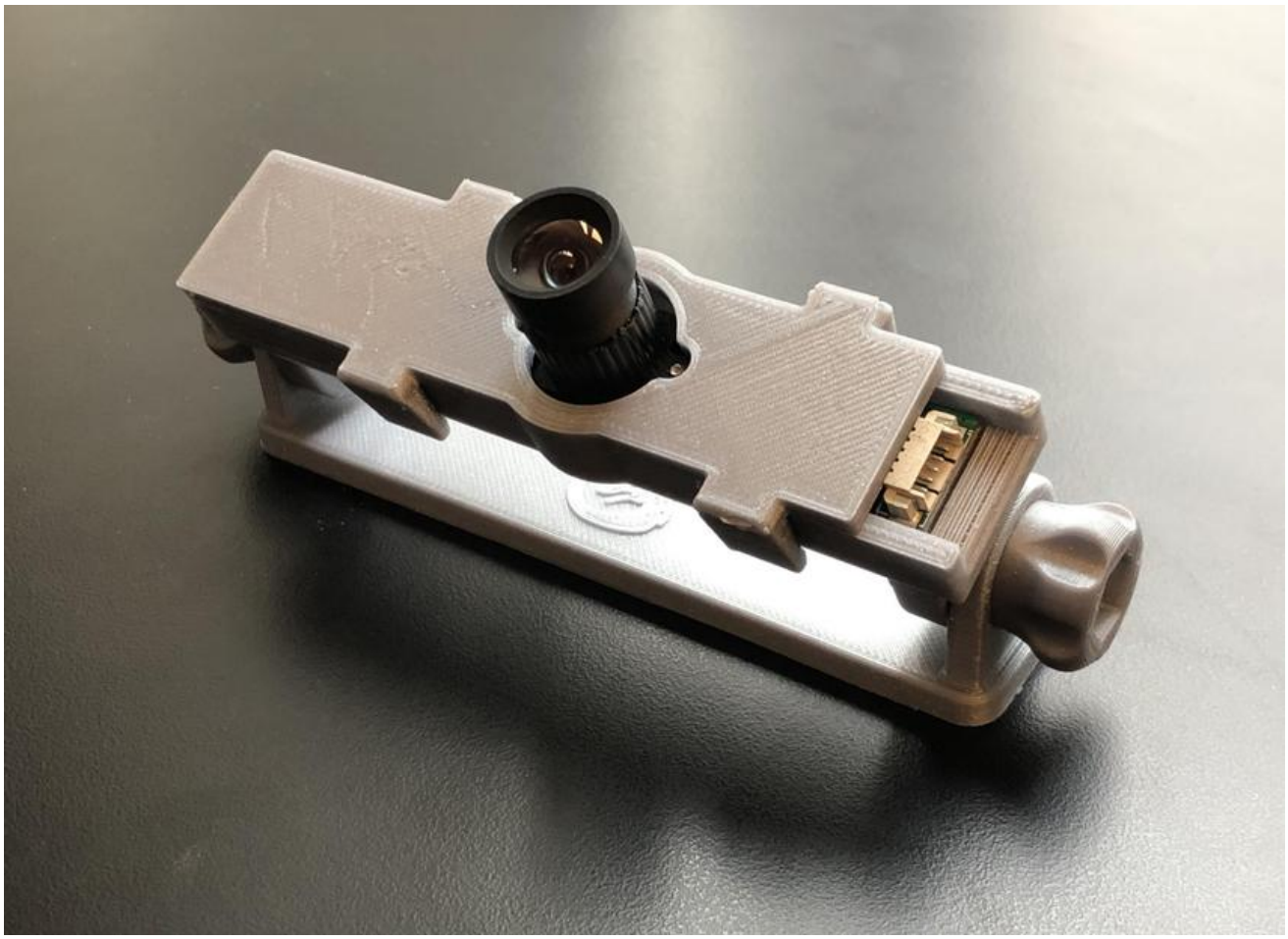
Download the Endoscope Camera Mount

Overhead Camera Mounts

If you're using one of LightBurn's cameras and have access to a 3D printer, you can print your own without having to get it shipped to you.

If you don't have a 3D printer of your own, a local makerspace or library may be able to help you.

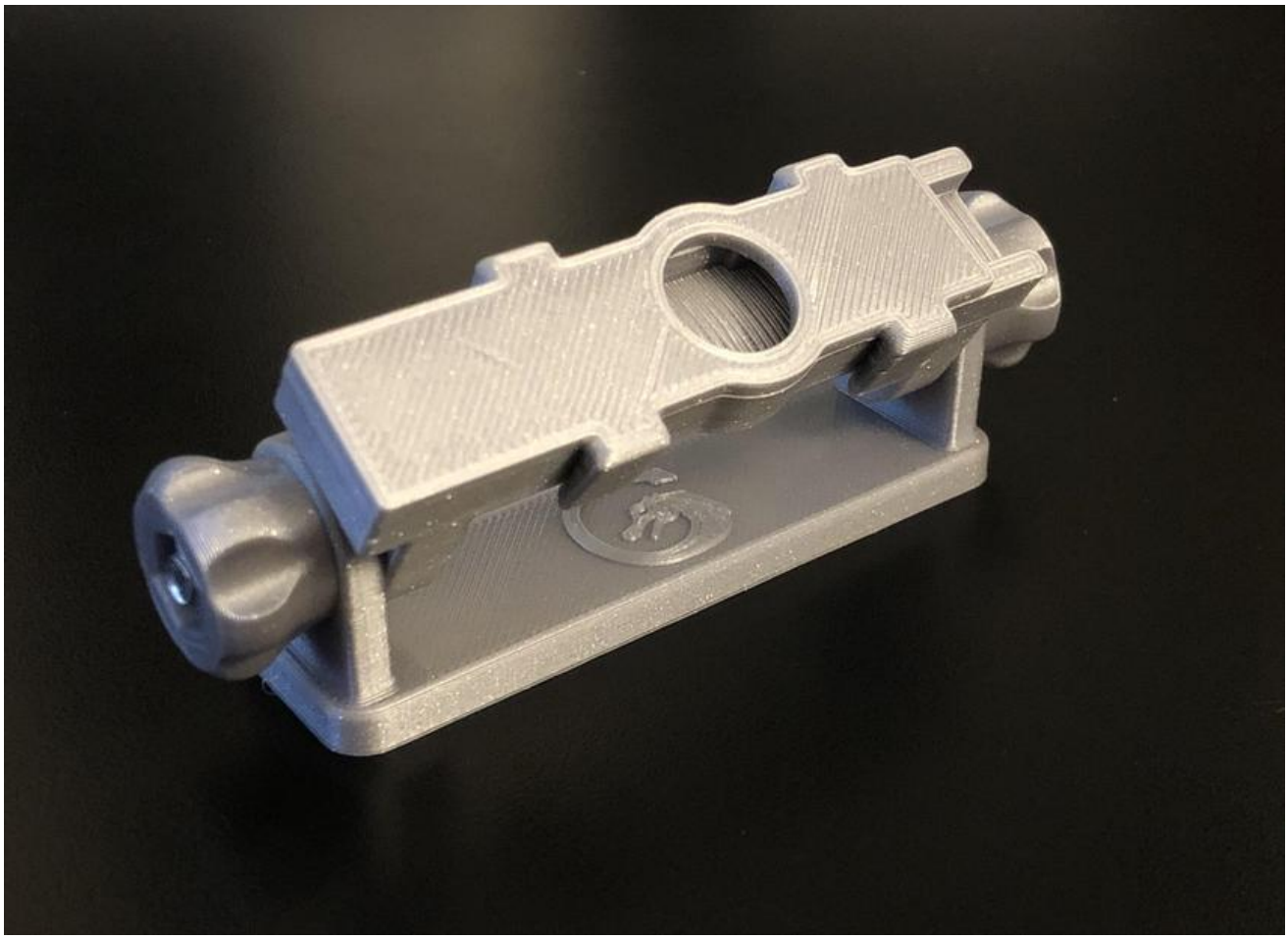
Downloads and Resources



If you have the 8MP 85 or 120W LightBurn camera, please download the large camera mount.

Large Camera Mount

Downloads and Resources



If you have any 5MP LightBurn camera or the 8MP 75 or 120N LightBurn camera, please download the small camera mount.

Small Camera Mount

Downloadable Art Libraries

Welcome to our Art Libraries download page. On this page, you'll find curated Art Libraries with themed clip art and LightBurn-created artwork for you to use in your projects.

If you haven't tried Art Libraries yet, click below for a video and a link to written directions to help you get started.

How to use Art Libraries

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=30hYVXQ-4CA

How to Use Art Libraries

As we release additional art libraries and other downloadable resources, we'll post them here, so check back soon.

Downloads

Latest Release

Our July 2026 release features three new art libraries:

- **Back To School**
- **Sea Creatures**

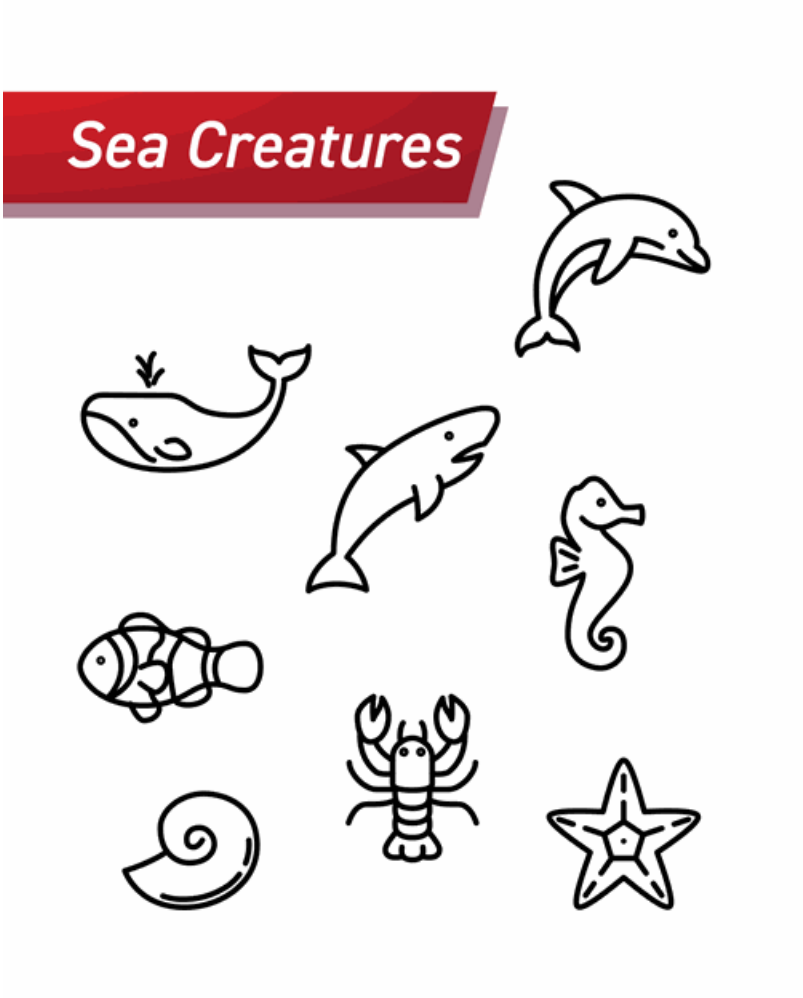
- **Summer Foods**

Back To School



Download

Sea Creatures



Download

Summer Foods



Download

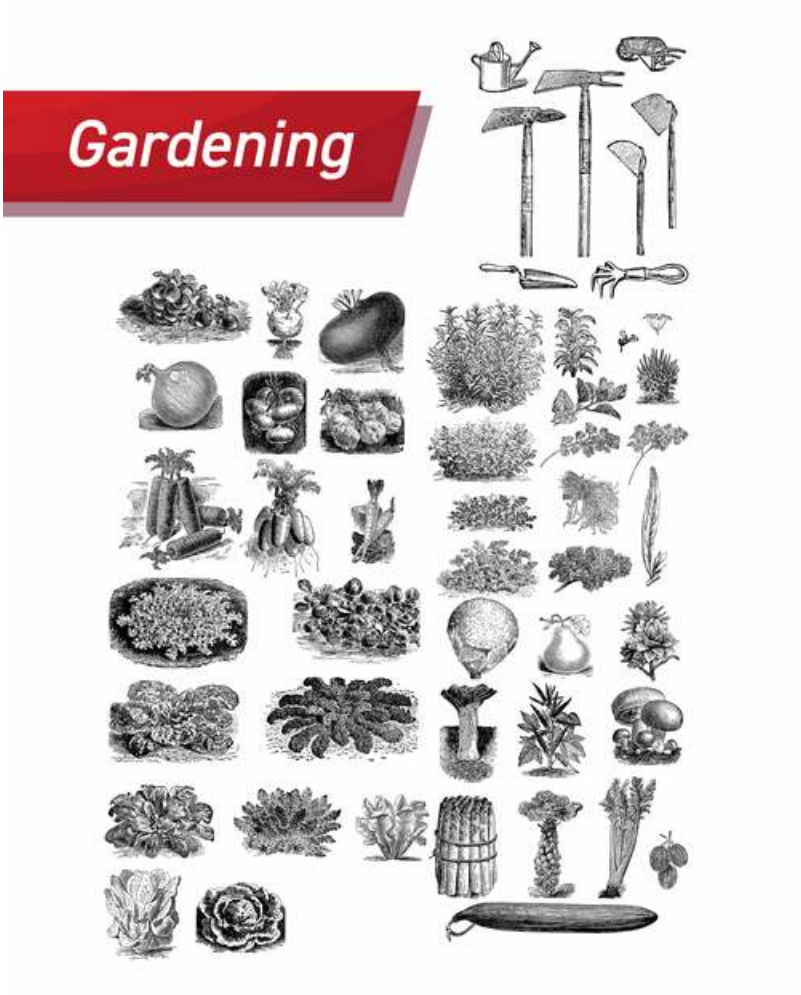
April 2026

Our April 2026 release features:

- **Garden**, featuring plants and gardening tools.
- **Bugs**, which go great with gardens.

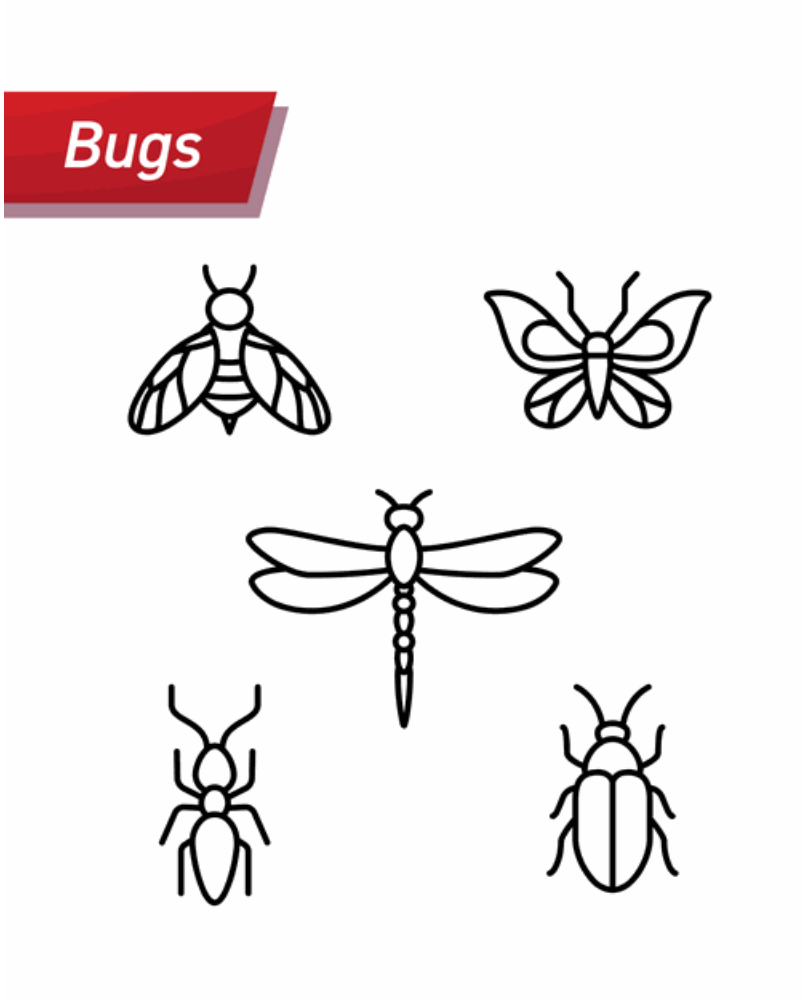
- **Graduation**, with artwork to focus on celebrating graduates of all levels.

Garden



Download

Bugs



Download

Graduation

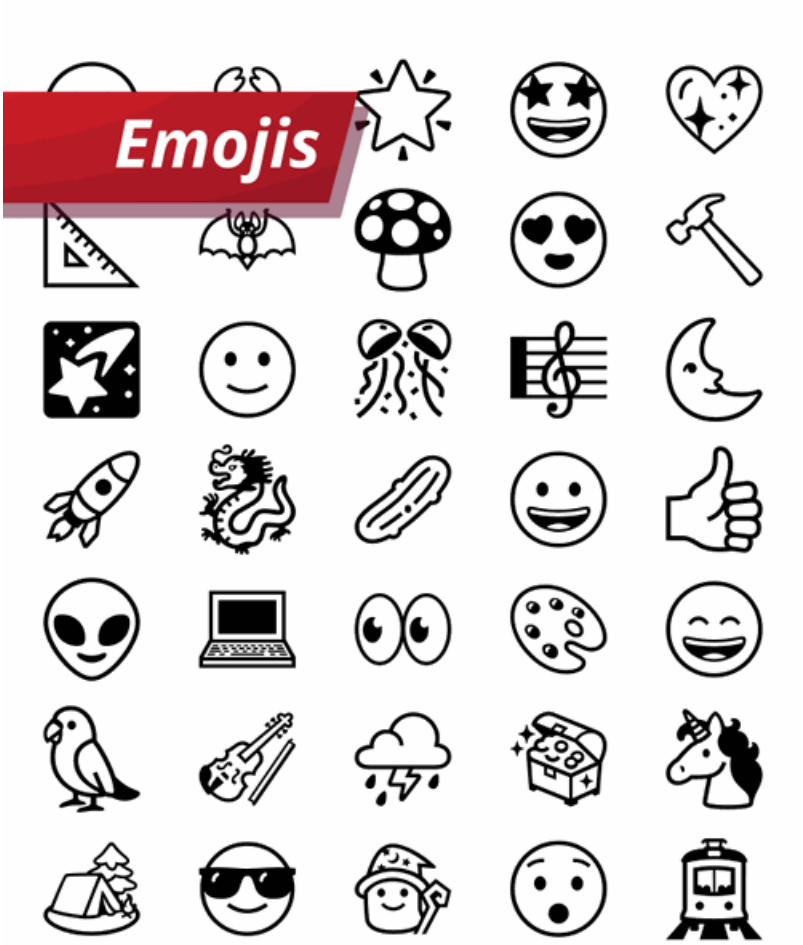


Download

January 2026

Our January 2026 release features an emoji library with 1366 emojis in 5 different weights, countries of the world, and a vintage Valentine's Day library.

Emoji



This Art Library has 1366 emoji symbols in five different weights. Each font weight is a separate library and can be downloaded with the download buttons below.

Light

Regular

Medium

Semibold

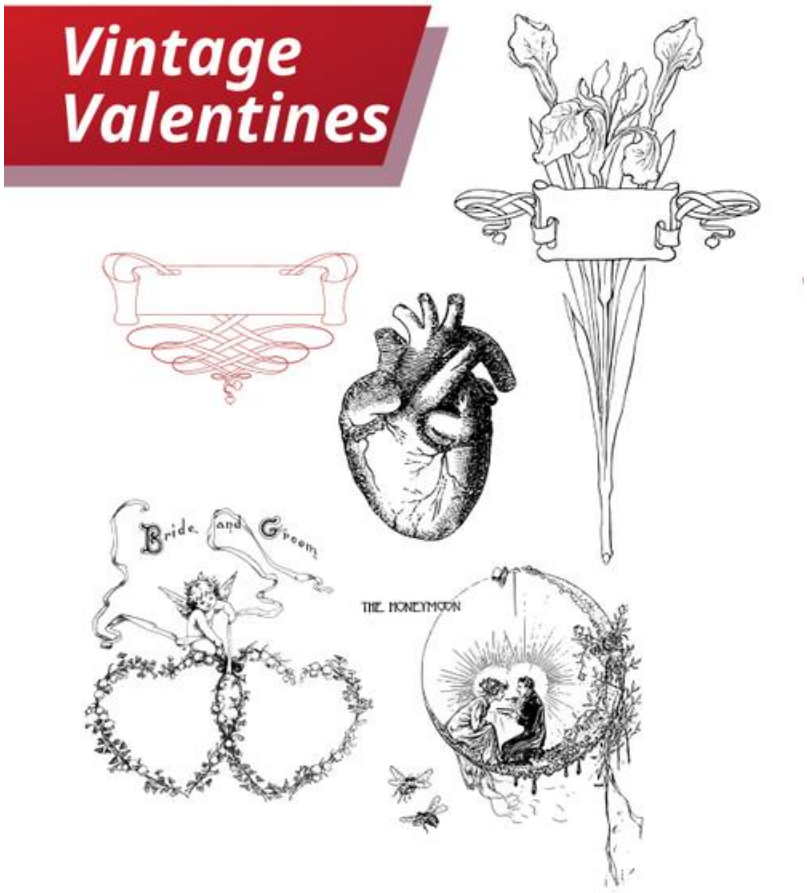
Bold

Countries



Download

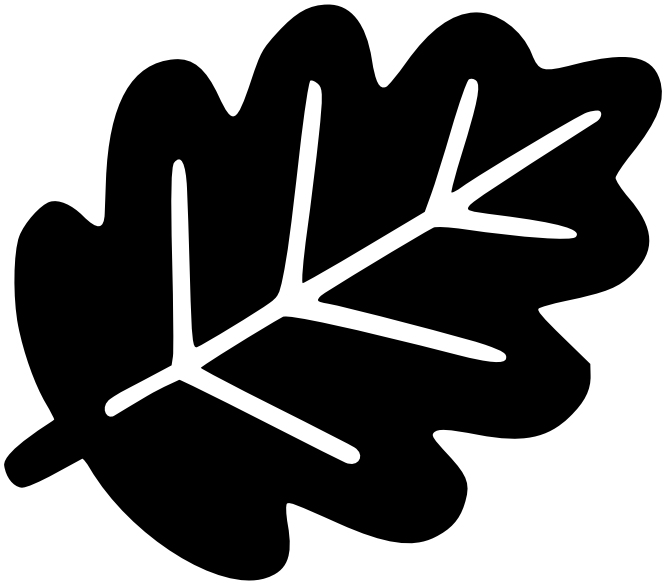
Retro Valentines



Download

September 2025

Fall/Autumn 2025



Download

Halloween 2025



Download

Winter 2025



Download

External Resources

Unsurprisingly, we love using LightBurn here. But there are some other great tools that pair well with it.

The links below are a variety of tools that the people at LightBurn have used and would recommend. We don't make any money off your use of these, and don't have any control over the tools themselves, but want to share them with our users in case they're useful to you as well.

Project Generators

These project generators can generate a variety of customized designs for you and output designs ready for you to laser cut or modify further. Most of them will have a focus, such as box makers or paper templates, but aren't as specialized as some of the more niche tools below.

Boxes.py

Price: Free

An open source generator for boxes, shelves, wall-mounted tool holders, and more. Exports in a wide variety of formats and the designs are good on their own or as a base for further customization.

Go to Boxes.py

Cuttle

Price: Free & paid tiers

A web-based design tool with a variety of pre-made generators you can use as well as tools to create your own. Has free and paid tiers.

This is also one of our included Web Tools

Go to Cuttle

Templatemaker.nl

Price: Free

Generator for a variety of paper patterns, including some unusual styles of gift boxes. Exports files in a wide variety of formats, including LightBurn (`.lbrn`), SVG, PDF, and DXF.

Templatemaker can create perforated fold lines, or export fold lines in a different color for you to score with a lower power or higher speed setting than your cuts.

Go to Templatemaker.nl

My Laser Tools

Price: Free & Paid Tiers

My Laser Tools has a wide variety of generators for laser cutters, CNC routers, and 3D printers. Exports in `.svg`, `.stl`, and `.3mf`.

This is also one of our included Web Tools

Go to My Laser Tools

MakerCase

Price: Free & paid tiers

A generator for a few box types, including divided boxes.

Go to MakerCase

Make A Box

Price: Free

A straightforward open source generator for tabbed boxes.

Go to Make A Box

Fonts and Text Tools

The tools in this section focus primarily on text, currently single-line text tools for faster engraving on your laser.

SHX Fonts

Price: Free

SHX Fonts hosted by burntstuff. Read more about using them here.

Go to SHX Fonts

Single Line Fonts

Price: Paid

A collection of single line fonts and artwork. The fonts are more varied and stylish than the SHX font collection, and have special characters to support a wider range of languages. Fonts are compatible with LightBurn 2.0 or newer or any version of MillMage. Read more, including installation instructions for use with LightBurn, here.

Go to Single Line Fonts

Single-Line Font Renderer

Price: Free

Simple open source web-based tool for creating SVGs of text using single-line fonts.

Go to Single-line Font Renderer

Specialized generators

These more specialized generators aren't the right tool for every project, but sometimes they're exactly what you need.

Joinery

Price: Free

This web-based tool focuses on modifying your existing parts to join them together in a variety of (often unusual) ways

Go to Joinery

Laser Map Maker

Price: Paid

Web-based tool for preparing layered maps.

Go to Laser Map Maker

Involute Spur Gear Builder

Price: Free

Open source web-based tool for generating gears you can cut with your laser.

Go to Involute Spur Gear Builder

Woodgears Gear Generator

Price: Free & paid

Free web-based tool for generating gears, and a more advanced paid Windows program supporting more gear types.

Go to Woodgears Gear Generator

Jigsaw Puzzle Generator

Price: Free

Free web-based tool for generating jigsaw puzzle pieces.

Go to Jigsaw Puzzle Generator

Laser-Cut Maze Designer

Price: Free

Free web-based tool for generating mazes for laser cutting. Includes multiple assembly styles.

Go to Laser-Cut Maze Designer

Maze Generator

Price: Free

Free web-based tool for generating mazes, with multiple shape and style options.

Go to Maze Generator

Vector Ruler Generator

Price: Free

Open source web-based tool for generating rulers in SVG format.

Go to Vector Ruler Generator

Art Files

Public Work

Price: Free

Art Type: Raster

A collection of free to use raster images.

Go to Public Work

Noun Project

Price: Free & paid tiers

Art Type: Vector & Raster

A collection of icons (in SVG and PNG formats) and stock photos.

Go to Noun Project

Internet Archive

Price: Free

Art Type: Vector & Raster

Many collections of free images and other files. Not tightly curated, but there are a lot of good files here.

Go to Internet Archive

Tiled.art

Price: Free

Art Type: Vector

A web-based tesselation design tool. Check out the **Art** section as well for some impressive tesselation examples.

Go to Tiled.art

Voronoi

Price: Free

Art Type: Vector

A free web-based Voronoi pattern generator.

Go to Voronoi

Flaticon

Price: Free & paid tiers

Art Type: Vector (paid) & Raster (free)

A large icon collection. Free tier can access PNG files, while paid tiers can access SVG and other file formats as well.

Go to Flaticon

Troubleshooting

If you're having trouble using LightBurn, our troubleshooting resources are a great place to start looking for help with common problems and frequently asked questions.

> **Note**
>
> If you can't find answers here, you can reach us by email at Support@LightBurnSoftware.com, or join our Support Forum, where you can search to find solutions to your issue, or make a new post asking for help.
>
> See How to Ask for Help for guidelines on getting effective assistance as quickly as possible.

What's Wrong?

Choose a category below for more help.

Job Quality

Resources to help you figure out why your laser's output doesn't look how you expect.

Read More

Preview Window

Solutions to common problems when using the Preview Window.

Read More

Connection Problems

Get help connecting your laser to a computer and sending jobs to it.

Read More

Troubleshooting

LightBurn Editor Help

Frequently asked questions about creating and editing designs in LightBurn.

Read More

GRBL Errors

Get help with GRBL-specific errors.

Read More

GRBL Communications

Get help communicating with GRBL devices.

Read More

License Activation and Management

Solutions to common license activation and management issues.

Read More

Camera Troubleshooting

Solutions to common problems using a camera in LightBurn

Read More

Setting up CNC-Based Lasers

Get help setting up a CNC router as a laser cutter.

Read More

Drivers

Get help finding and installing the appropriate drivers for your laser.

Read More

Electrical problems

Get help identifying and solving electrical problems with your laser.

Read More

Galvo Laser Troubleshooting

Get help identifying and solving problems with your Galvo laser.

Read More

How to Ask For Help

How to ask for help with LightBurn and get useful answers quickly.

Read More

Vector Artwork Imports with Extra Lines

If you get a lot of extra lines and unexpected shapes in your design when bringing in designs made outside of LightBurn, we can help.

Read More

Machine Size Mismatch Error

Get help dealing with the machine size mismatch error in LightBurn.

Read More

macOS-Specific Problems

Get help with problems specific to macOS.

Read More

Windows-Specific Problems

Get help with problems specific to Windows.

Read More

Mechanical Issues

Get help identifying and fixing mechanical problems with your laser.

Read More

Troubleshooting

Reset to Default Settings

Messed up your settings and not sure what to do? Here's how to reset them.

Read More

Serial Port Problems

Get help with serial port problems between LightBurn and your laser.

Read More

USB Cables

Get help identifying and solving problems with your USB cable.

Read More

Device Specific Troubleshooting

Some machines use non-standard commands or configurations, and need special settings to work properly with LightBurn.

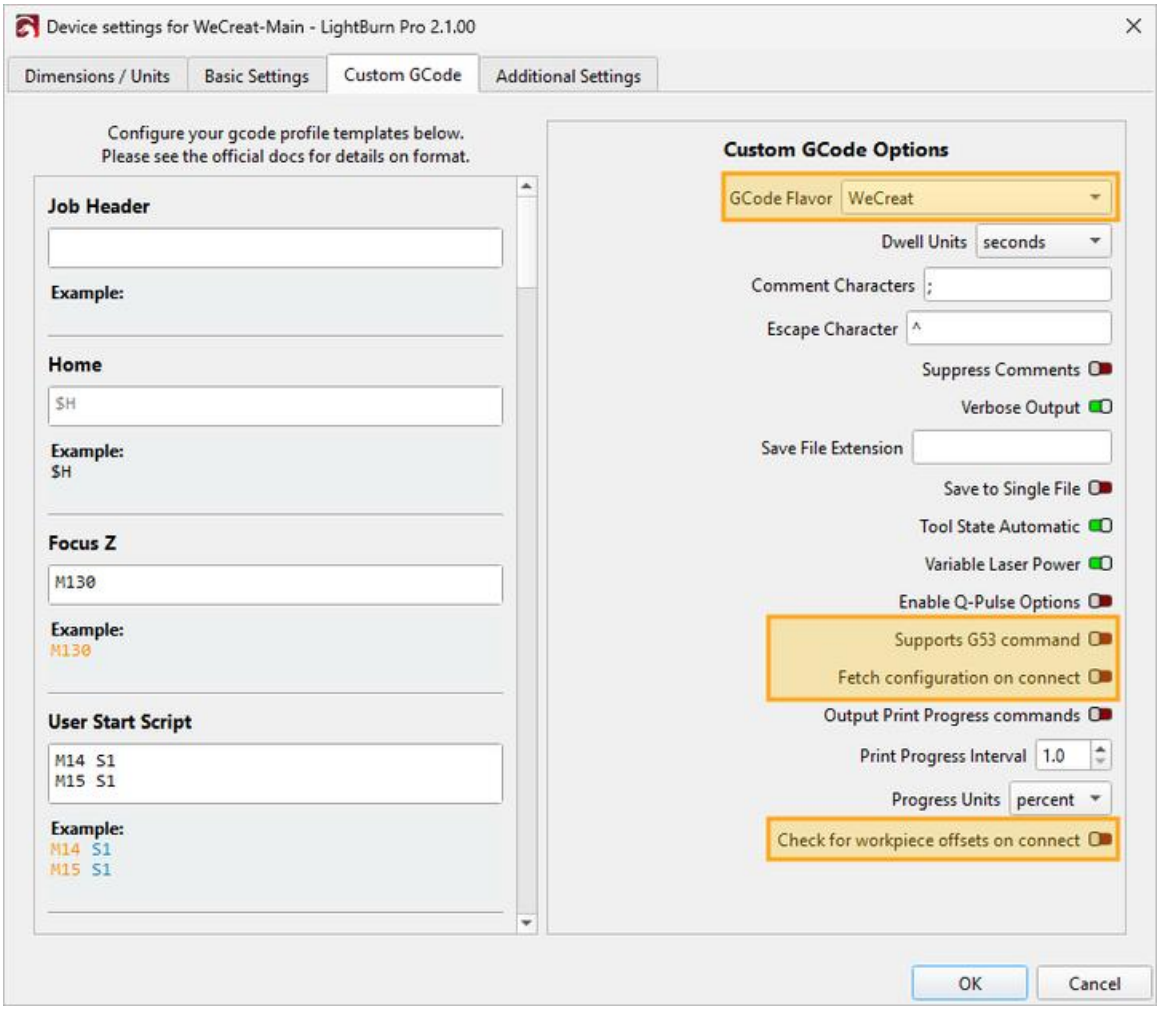
wecreate

WeCreat Vision

The WeCreat Vision needs to be set up as a Custom GCode device, either using Create Manually or by importing a pre-configured bundle.

If you choose to add the machine manually, or already added one and aren't sure the machine is set up correctly, please make sure you use the following settings in the Custom GCode tab of Device Settings.

- **GCode Flavor** must be set to **WeCreat**.
- **Supports G53 command** must be disabled.
- **Fetch configuration on connect** must be disabled.
- **Check for workpiece offsets on connect** must be disabled.



Darkly Labs Emblaser Devices

Overview

To use a Darkly Labs Emblaser with LightBurn, you will need to use LightBurn license that's associated with Darkly Labs (licenses purchased from Darkly Labs with an Emblaser are already set up this way) and set up your device using the correct Emblaser device type for your machine. Keep reading for additional details.

Additional Information

Emblaser devices have some special features in LightBurn, such as:

- WiFi wizard
- Custom focus calibration wizard
- Custom camera calibration and alinment wizards

When you buy an Emblaser, the LightBurn license key that comes with it has a special tag that allows it to work with Darkly Labs machines. When you start LightBurn using a license with this special tag, you'll see the Darkly Labs logo on the splash screen at startup. If this logo is missing and you think this is an error, please contact support for further help.



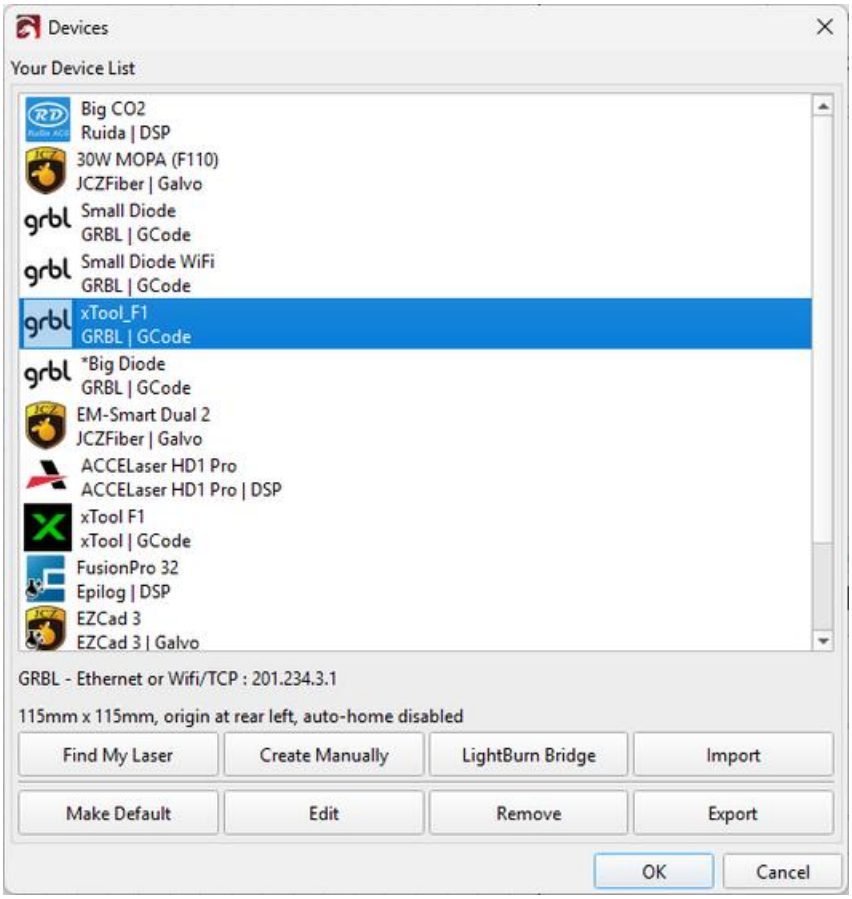
When you add your device to LightBurn, you'll need to choose the correct device type - each Emblaser device type is listed separately.

xTool

LightBurn 2.2 brings significant changes to how xTool devices are configured and used in LightBurn. Please see our xTool page for additional information on setup and device support.

Existing xTool Devices

If you've already set up an xTool device in previous versions of LightBurn, you'll need to re-add the device to LightBurn.



In the screenshot above, you'll see the xTool F1 listed twice. The highlighted one has a **grbl** icon and was created using a `.lbdev` file. This older device setup should not be used for xTool devices in LightBurn 2.2. The new xTool device type, represented with an xTool icon, offers more support for xTool features, an easier installation process, and support for a wider range of machines.

What to Do

If your xTool device was set up before LightBurn 2.2 or has a **grbl** icon:

- Make sure you know where your Material Library is if you're using one.
- Add your xTool device using our updated directions for 2.2.
- Load your **Material Library** if you have one to load from an old xTool device.
- Delete the older xTool device.

Firmware Issues

Previous versions of this page warned that newer firmware updates on certain xTool machines were incompatible with LightBurn. This has been fixed with the new xTool Device Type, which improves support for xTool machines and works with current software versions.

If you're using an xTool laser created in an older version of LightBurn as a GRBL laser, you will need to use a firmware version below the ones listed at xTool Devices: Firmware. We recommend that you instead recreate your laser with the new xTool Device Type.

Job Quality

This page contains a collection of various issues you may see in your completed cuts and engravings, along with their most probable solutions.

Job Mirrored or Backwards



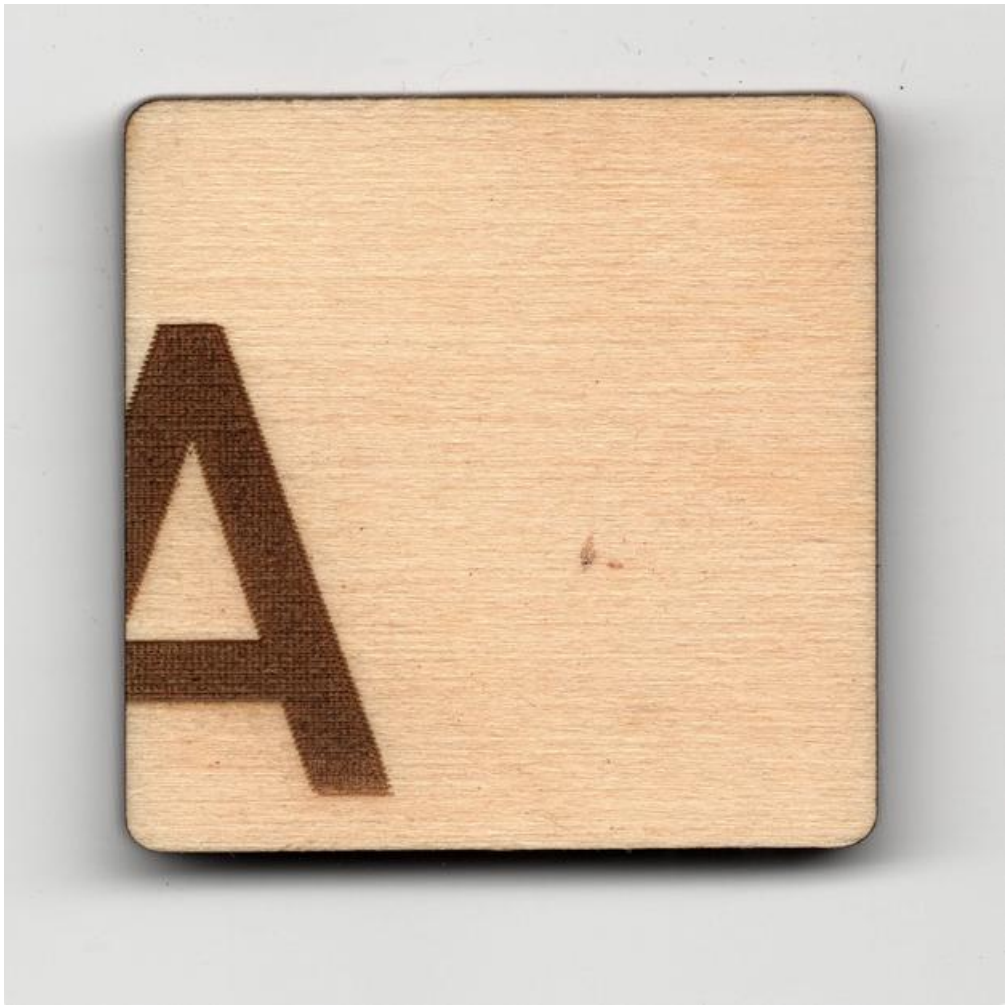
Job Mirrored or Backwards

"GRBL: Low or No Power Output"



"GRBL: Low or No Power Output"

Job Starting in Wrong Location



Job Starting in Wrong Location

Warped or Disconnected Lines



Warped or Disconnected Lines

Move Actions Are Burned



Move Actions Are Burned

Incorrect Size



Incorrect Size

Fuzzy, Blurry, or Overblown Edges



Fuzzy, Blurry, or Overblown Edges

Edges of Engravings Are Dark or Burned



Edges of Engravings Are Dark or Burned

Wobbly Engraving



Wobbly Engraving

Poor Image Quality



Poor Image Quality

Inconsistent Engraving Tone or Color



Inconsistent Engraving Tone or Color

Periodic Defects in Engravings



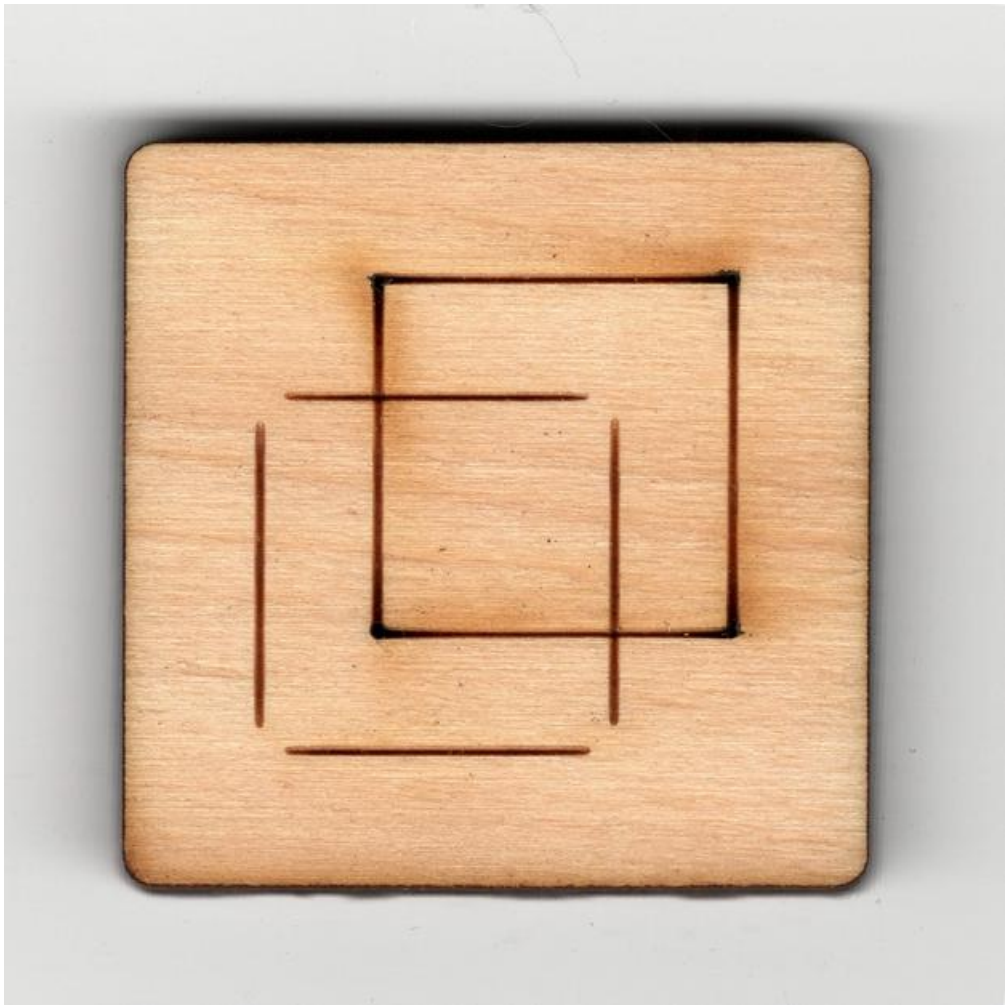
Periodic Defects in Engravings

Jagged Lines



Jagged Lines

Corners Are Too Dark, Too Light, or Missing



Corners Are Too Dark, Too Light, or Missing

Laser is Losing Power or Not Cutting at Low Speeds



Laser is Losing Power or Not Cutting at Low Speeds

Slanted or Skewed Jobs



Slanted or Skewed Jobs

Engravings Within Cuts Are Shifted



Engravings Within Cuts Are Shifted

Layers Shift When Cutting Multiple Objects



Layers Shift When Cutting Multiple Objects

Wiggly Lines at Corners



Wiggly Lines at Corners

Wrong Orientation



Description

Job flipped, mirrored, reversed, inverted, upside down, or reflected when comparing appearance in your LightBurn workspace to the real-world cut/engrave.

Solution

- Check that the Device Origin is set correctly. This setting controls the orientation of the laser's output. If this is set incorrectly, your job will be mirrored or inverted.
  - The Origin for GCode devices (including most diode lasers) is typically in the front left corner, regardless of where limit switches are installed.
  - The Origin for DSP controllers (including Ruida, Trocen, and TopWisdom) should match the corner your laser homes to when first powered up.
  - To correct a mirrored or inverted engraving:
    - If your engraving is flipped upside down, change whether the origin is in the front or back corners.

    - If your engraving is mirrored sideways, change whether the origin is on the left or right side.
    - If your engraving is flipped in both directions, move the origin to the opposite corner.

- In rare circumstances, jobs can be mirrored due to flipped motor cables within the machine. Seek assistance from the manufacturer if you have set your **Device Origin** properly and are still having issues.
- For Galvo lasers, you may need to adjust Per Galvo settings to set the correct Galvo to X, and to **Reverse Direction (Negate)** one or both Galvos.

GRBL: Low Power or No Output



Description

Low or no power output with a GRBL device. The two most common causes of this behavior are:

Setting Speed Too High in Variable Power Mode

The most common cause of this issue is commanding your laser to run faster than its maximum allowable speed, leading to power downscaling. When **Variable Power** mode is enabled, the power level is 0 when the laser is stopped, and increases as the laser head accelerates toward the requested speed. This makes the resulting burn more consistent. However, if you ask the laser to go faster than it is able, it will never reach the requested speed and therefore never reach the requested power level either.

Solution

- Set speed within the allowed range of your machine. Go to Machine Settings or enter `$$` in the Console window to see the **\(110/\)111**, **X/Y Max rates** set in your machine's firmware. The X and Y Maximum rates are the fastest you can command your machine to travel, in mm/minute.
- Double-check the units of speed you are using. A given value in mm/second is much faster than the same value in mm/minute. Most diode manufacturers provide suggested settings in mm/minute. If you prefer to use mm/second, take care to properly convert the values. You can switch to mm/minute, or another unit of measurement, in the Settings window.
- If you want the same behavior as LaserGRBL, which uses **Constant Power** mode (M3), enable the **Constant Power** toggle in the Cut Settings Editor.

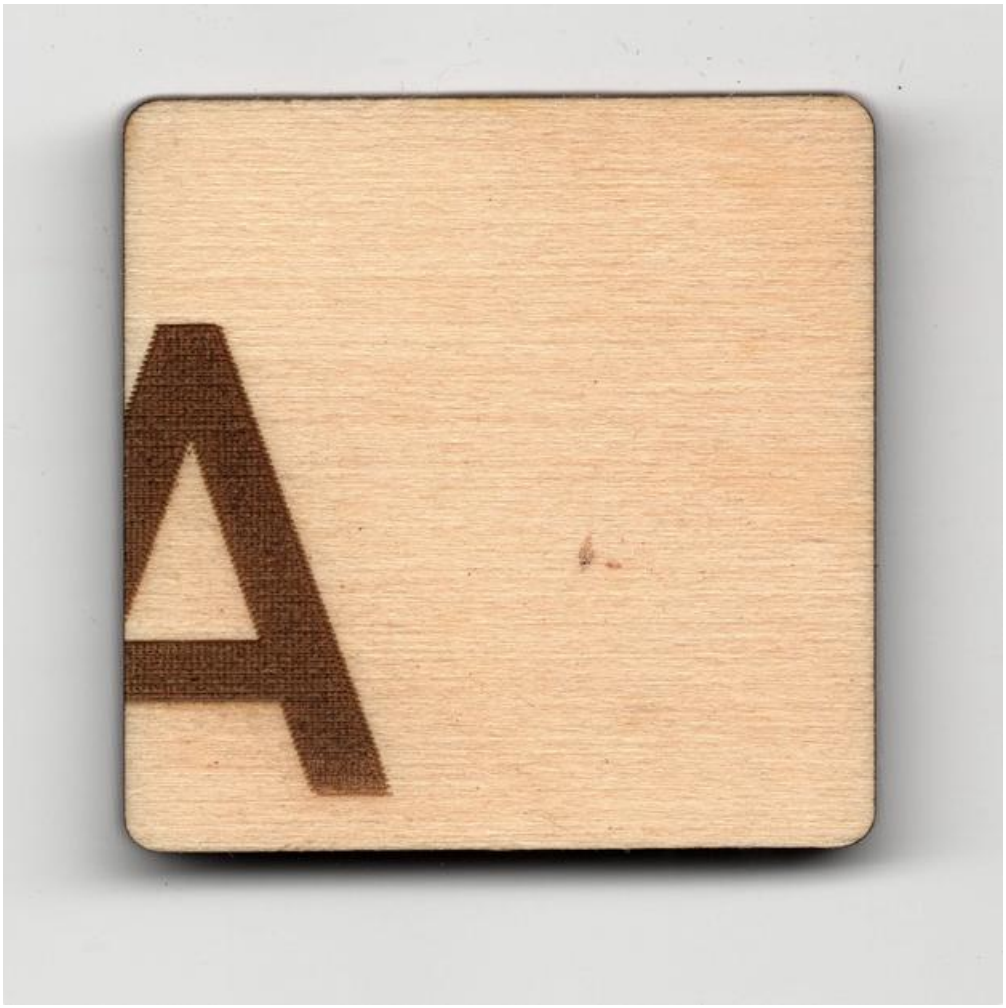
S-Value Max Set Incorrectly

If you've already lowered your speed and are still seeing lower than expected power output, the **S-value max** setting in the **Device Settings** window could be incorrect. This number tells LightBurn how to represent 100% power for your machine. Current versions of GRBL default to **1000** for this, while older versions use **255**. The important thing is for the numbers to match. If your machine expects 1000 to mean full power and receives 255 from LightBurn, you'll get roughly ¼ of the expected power output.

Solution

- Go to Machine Settings or enter `$$` in the Console window to see the **$30, Max spindle speed** setting in your laser's firmware.
- Adjust the **S-Value max** setting in Device Settings to match.

Wrong Start Location



Description

Job is starting in an unexpected locaation.

Solution

- Check that you have selected the correct **Start From** mode and **Job Origin** to position output the way you expect. Read more about these settings here.
- In the Device Settings window, verify that your machine's **Working Size** is set correctly.

Warped or Disconnected Lines



Description

Circles and other closed shapes are not joining up.

Solution

- This is generally a result of inertia combined with some play or slop in your machine's mechanics. Check out our page on Mechanical Issues for more information.
- You can also try enabling Hide Backlash in **Optimization Settings**.
- If the lines are disconnected, but otherwise lining up properly, you may also want to try increasing your Cut Through or Overcut settings.

Laser Fires During Travel Moves



Description

Unexpected lines where they shouldn't be. Laser doesn't turn off when moving between cuts.

Solution

- If you have a GRBL device, and your laser is still on when moving between shapes, like the image above, it most likely means you are using GRBL 1.1f or later, but do not have the controller in Laser Mode. Fix this by typing `$32=1` in the Console window and pressing `Enter`.
- If you have a Ruida or other DSP controller and are seeing this happen, it is most likely a dying high-voltage power supply unit (PSU). The drive circuit that turns the laser power off and on tends to go first, before the part that controls the overall power level. Contact your laser's manufacturer for assistance on sourcing a replacement.

Incorrect Size



Description

Job is larger or smaller than the sizes shown in LightBurn

Solution

If only the Y (vertical axis) is the wrong size and you have used a rotary previously, ensure the Enable Rotary setting is disabled.

If the size is incorrect in both dimensions, use the Calibrate Axis tool to correct your machine's step settings.

Fuzzy, Blurry, or Overblown Edges



Description

Fine details are blown out, or edges of engravings are fuzzy or blocky.

Solution

If alternating lines of an engraving are offset, add or adjust the Scanning Offsets.

If the lines are correctly positioned but look blocky or staggered at the edges, consider:

- Lowering your Line Interval (increasing DPI)
- Using Sub Layers to add an outline to your engrave
- De-focusing your laser by a few millimeters so each scan line blends into the surrounding ones

Dark & Burned Edges



Description

When doing a **Fill** or **Image** engraving, the edges or slow areas are darker or more burned than the center or faster areas.

Solution

This is caused by **Overscanning** not being set or being set incorrectly in the Cut Settings Editor for either Fill or Image operations.

Wobbly Engraving



Description

Shaky, warped, or wiggly engravings on images and fills.

Solution

This is generally a result of inertia combined with some play or slop in your machine's mechanics. Check out our page on Mechanical Issues for more information.

Poor Image Quality



Description

The photo/ image engraved doesn't look good in some way or another.

Solution

Image engraving is a complex topic, learn more on the 5 Steps to better image engravings page.

- You may also find Image Cut Settings helpful for understanding cut settings related to working with images

Inconsistent Engraving Tone or Color



Description

Dark and light patches within your engraving.

Solution

Usually this is a product of inconsistencies within the material itself. Either it has natural variation, like the grain in wood, else the surface of the material might vary in distance to the laser so that it's moving in and out of focus. This is usually caused by warped materials, or materials with uneven thickness. Potentially this can also be a result of the machine's beam path not being properly aligned, or the gantry not being trammed (parallel) to the work surface.

Periodic (Repeated) defects in Engravings



Description

A repeated pattern visible in the output of engravings.

Solution

This typically indicates a defect or adhesion of some kind on a moving mechanical part such as a lead screw or the motors that drive the belts. For more information, see Troubleshooting: Mechanical Issues.

Jagged Lines



Description

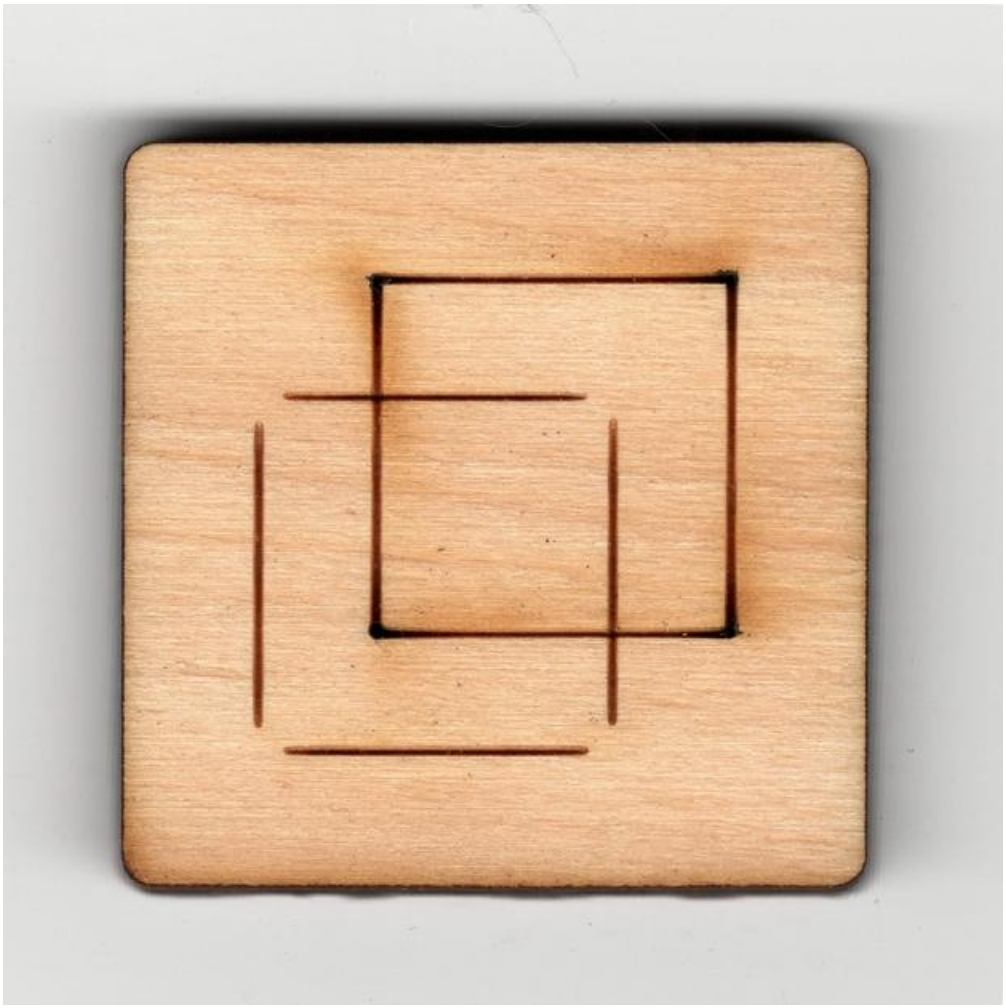
Your laser is outputting shaky, jittery, jagged, or sharp lines.

Solution

This is the result of a failed license integrity check. You may have downloaded LightBurn from an unauthorized source. Reverse any changes made to your system during the original installation, and then download an authorized copy of our software directly from our website. To activate LightBurn permanently, purchase a license from our website or from an authorized reseller.

If you're encountering this issue despite purchasing the software from our website, please email support for help. If you purchased from a reseller, reach out to them.

Corners are Too Dark, Too Light, or Missing



Description

Corners or any area where your laser changes directions while cutting are too dark, too light, or missing entirely.

Solution

This usually means you need to adjust your Min/Max Power settings.

DSP Controllers

If the Min Power setting is too high, the laser can leave scorch marks or wide kerfs at the slow areas of your jobs.

If your Min Power value is too low, you will see under-burned areas in slow spots.

If Min Power is so low that it falls below the firing threshold of your laser you will find completely unburned areas. If you are running a cut at very low speed, set your Min Power and Max Power to the same value to avoid this.

The firing threshold is usually around 10% of your laser's wattage. In other words, a laser with an 80W tube will usually fire around 8% power, but not below.

Start Speed in Machine Settings specifies the value where the controller begins ramping the power from Min to Max power. If the laser is moving at the 'Start Speed' or slower, the Min Power is used. The **Start Speed** is 10 mm/sec by default on most Ruida controllers, but some machines may have it set as high as 15 to 20 mm/sec, or may have it set lower.

If you are cutting at a low overall speed (for example, 10 to 20 mm/sec on a CO_2 machine) the laser might only ever use the 'Min Power' setting, so in that case it is best to use the same value for both. If you are doing 'surface marking' you ideally want Min Power to be just above the firing threshold of the laser, so corners still produce a burn, but without scorching.

GRBL Controllers

> **Warning**
>
> Setting the Max Acceleration too high can cause mechanical problems. Increase acceleration in small increments and test each time you adjust it.

With the exception of engraving Images in Grayscale mode, Min Power is not available on GRBL controllers. However, if the acceleration is set too low in the firmware, the laser may not speed up fast enough to fire at sufficient power when turning corners. Raising the X and Y Max Acceleration to 100-500 in Machine Settings may correct this.

You may also need to reduce your speed or adjust your **S-Value Max** setting. See more about GRBL Power Output problems.

Laser Lowing Power at Low Speeds or Not Reaching Full Power



Description

Laser doesn't fire when moving slowly or doesn't reach full power.

Solution

You most likely need to adjust your Min Power setting. If you are running a cut at a very low speed, set your Min Power and **Max Power** to match.

For DSP lasers, **Start Speed** in Machine Settings specifies the value where the controller begins ramping the power from Min to Max power. If the laser is moving at the Start Speed or slower, the Min Power is used. The Start Speed is 10 mm/sec by default on most Ruida controllers, but some machines may have it set as high as 15 to 20 mm/sec, or may have it set lower.

If Min Power is so low that it falls below the firing threshold of your laser, it will not fire at all when moving slower than the Start Speed. The firing threshold is usually around 10% of your laser's wattage. In other words, a laser with an 80W tube will fire around 8% power.

If you are cutting at a low overall speed (for example, 10 to 20 mm/sec on a CO_2 machine) the laser might only ever use the Min Power setting, so in that case it is best to use the same value for both. If you are doing surface marking you ideally want Min Power to be just above the firing threshold of the laser, so corners still produce a burn, but without scorching.

Slanted or Skewed Results



Description

Engraves are angled or skewed. Lines might be either correct or skewed.

Solution

Ruida-Specific Settings

If only the fill is slanted on a Ruida device, your motor pulse step polarity is probably reversed. To fix this, you can toggle the **PWM Rising Edge Valid** setting for one or both axes, depending on the angle of the skew or angle. You can find this setting by going to Laser Tools → Machine Settings and expanding **Vendor Settings**. You'll find the setting under the X and Y Axis sections.

> **More about this setting**

The controller sends commands to the motor drivers with a series of step pulses in which a signal is briefly switched from high to low or low to high. Motor drivers will look for either a transition from high to low (falling edge) or low to high (rising edge) to count as a step.

If the laser controller expects the motor driver to look for the leading edge signal, it will pulse the line and could change the direction line immediately after that. If the motor driver is waiting for the falling edge, it will see the direction change before the falling edge of the pulse, meaning that it will change direction one step too soon.

In the image below, the upper line of steps is interpreted as 4 steps in one direction followed by two in the other. The lower line is interpreted as 3 and 3, and the only difference is which side of the step signal the driver is looking for.

All Gantry-Style Machines

If both lines and engraves are slanted (not necessarily in the same way) and you have a gantry style machine, you likely have one of the following problems:

- Your frame and/or mirrors may be misaligned
- Your motors might be losing steps for some reason, such as overheating
- You may have other mechanical issues

Galvo-Specific Issues

If you're running a galvo machine, you may need to go through the Lens Correction process.

Engraving Shifted Relative to Cut Lines



Description

Engravings on small pieces aren't matching up with the cut lines around them.

Solution

This can happen when a piece is cut out and falls down or shifts before the engraving step.

Preview the path to check that the laser's intended output matches what you expect to see.

Change the order of operations in the Layers window, order of Sub-layers in the Cut Settings Editor, Cut Order Priorities, or Cut Optimization Settings ensure that engravings are performed before cuts.

If the misalignment persists, check on your machine's mechanical setup. For more information, see Troubleshooting: Mechanical Issues.

Layers Shift When Cutting Multiple Objects



Description

Cuts and engravings aren't lining up as expected when cutting multiple objects, but are correctly (or almost correctly) aligned for a single object.

Solution

This can happen when the laser's travel distance while jumping between shapes doesn't quite match the distance it should be traveling. The layer shift may not be visible on a single object if the difference in distances is small, but as the number of jumps between shapes increases, small shifts add up to become very visible toward the end of a job. See the image above for an example.

If the Preview window shows the layers aligned correctly, this is most likely a mechanical issue. See Troubleshooting: Mechanical Issues for more help.

Wiggly Lines at Corners



Description

A wavy "ringing" pattern in places where direction changes suddenly and at high speed. .

Solution

This is typically a result of inertia/speed combined with some play or slop in your machine's mechanics. Check out our page on Mechanical Issues for more info.

If you are using a Galvo laser, you may need to adjust your Jump settings:

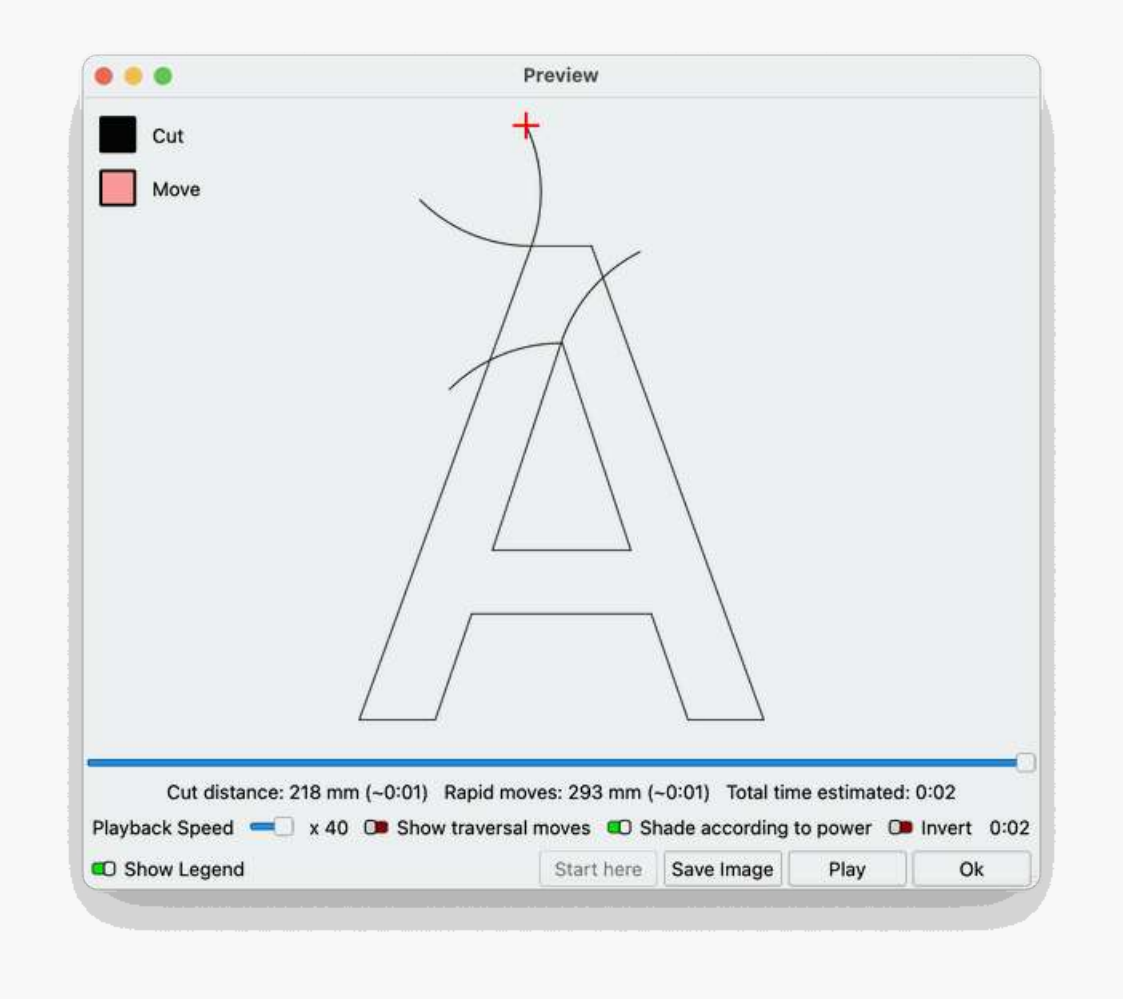
- If you did not import the markcfg7 file from the EZCAD configuration folder that came with your laser when you first set up your device profile, do so now.
- If you already imported the configuration file, try decreasing **Jump Speed** and increasing **Min Jump Delay** and **Max Jump Delay**.

Preview Window

We strongly recommend checking the **Preview** window before starting any job. If you see something that doesn't look right, but aren't sure what to do, check below for common issues and how to fix them.

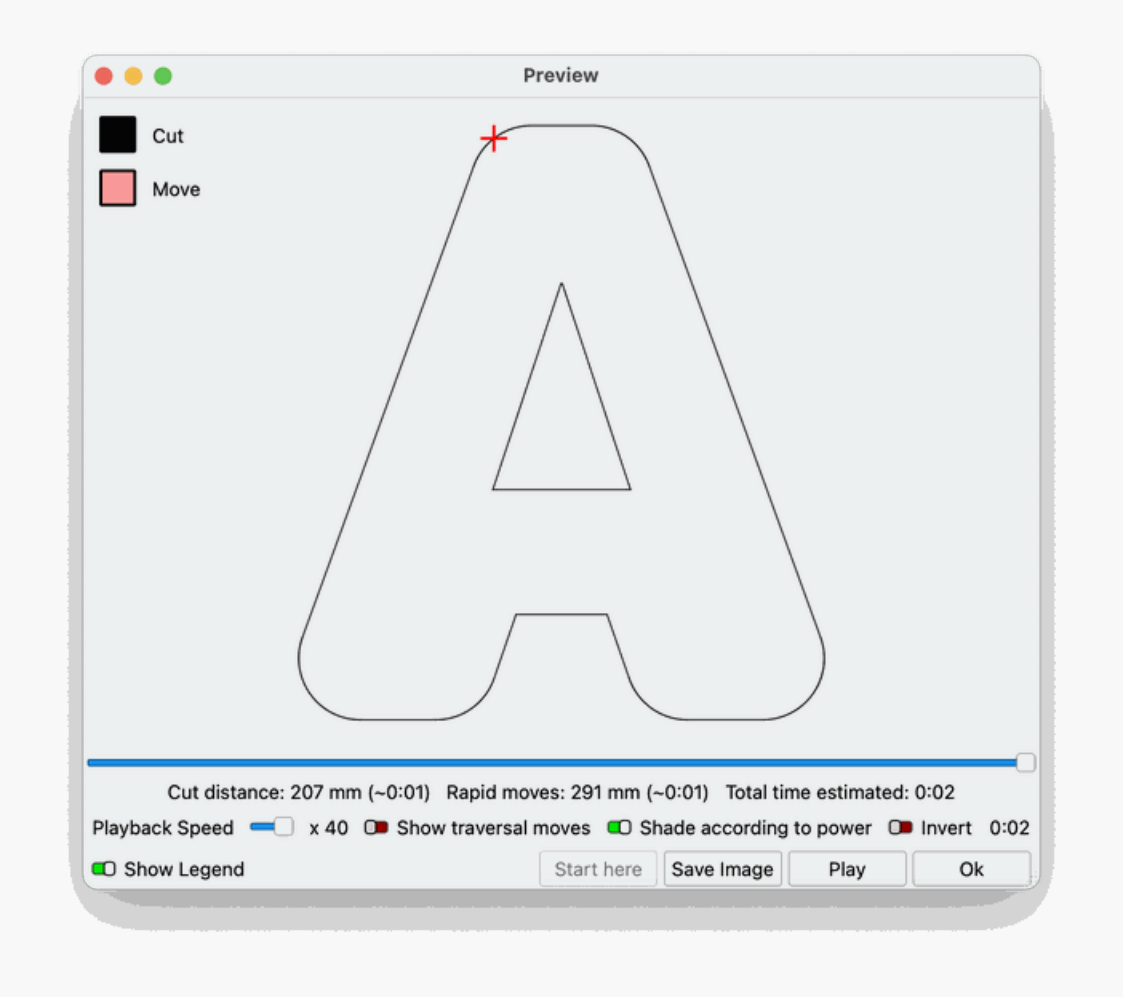
See Preview for information on interpreting and using the Preview window.

Added Lines



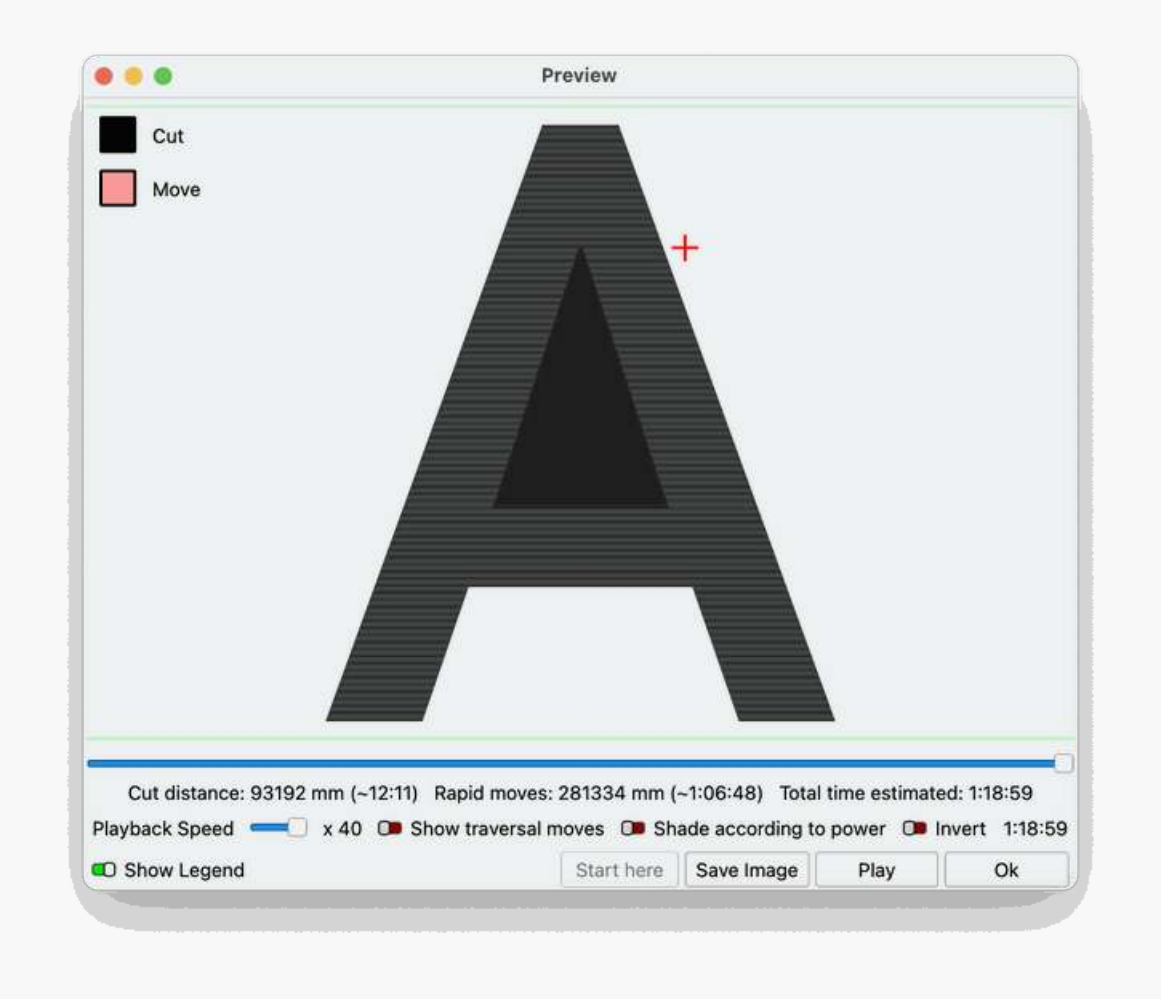
Added Lines

Ballooned or Incorrectly Sized Shapes



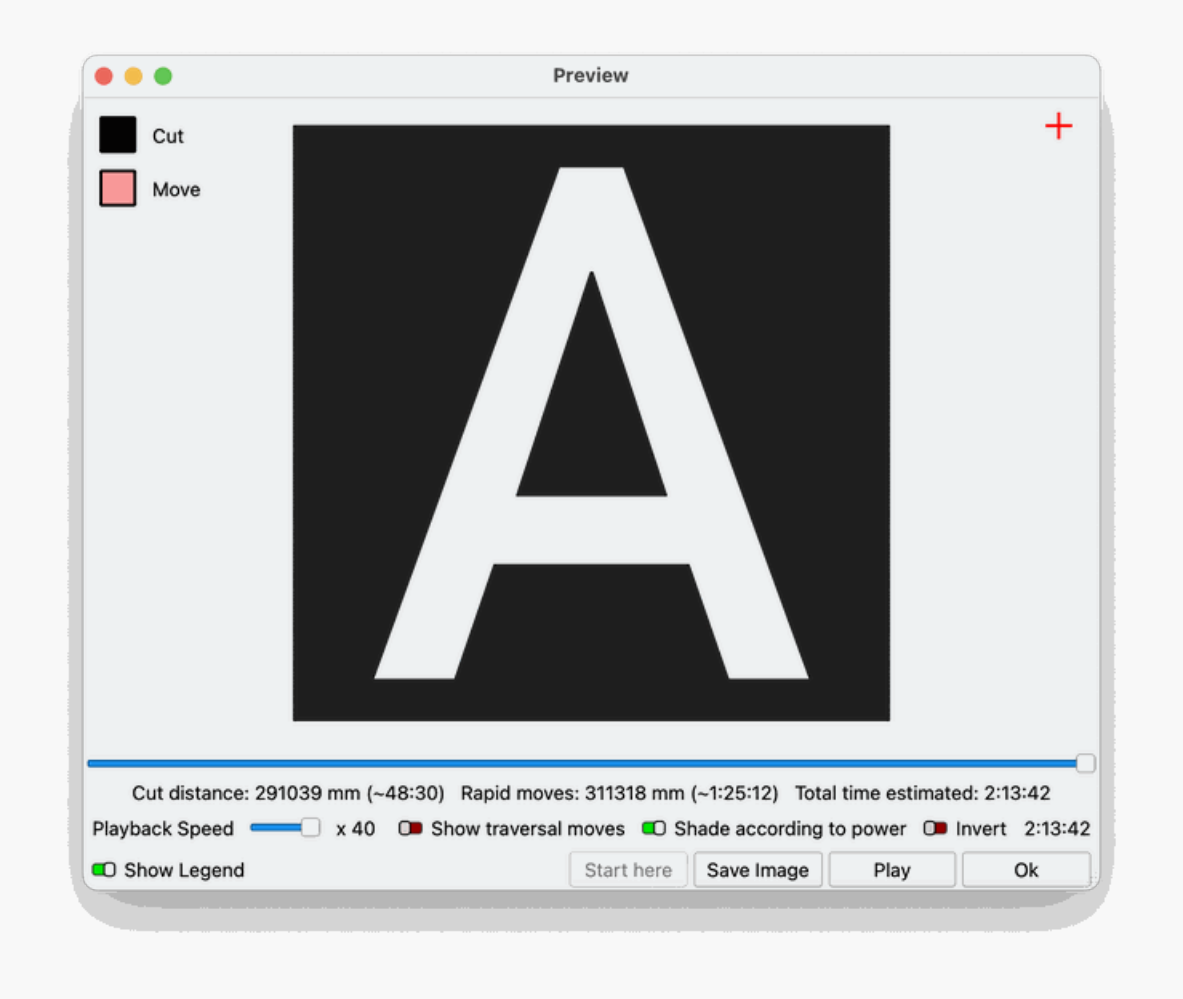
Ballooned or Incorrectly Sized Shapes

Double-Engraved Areas



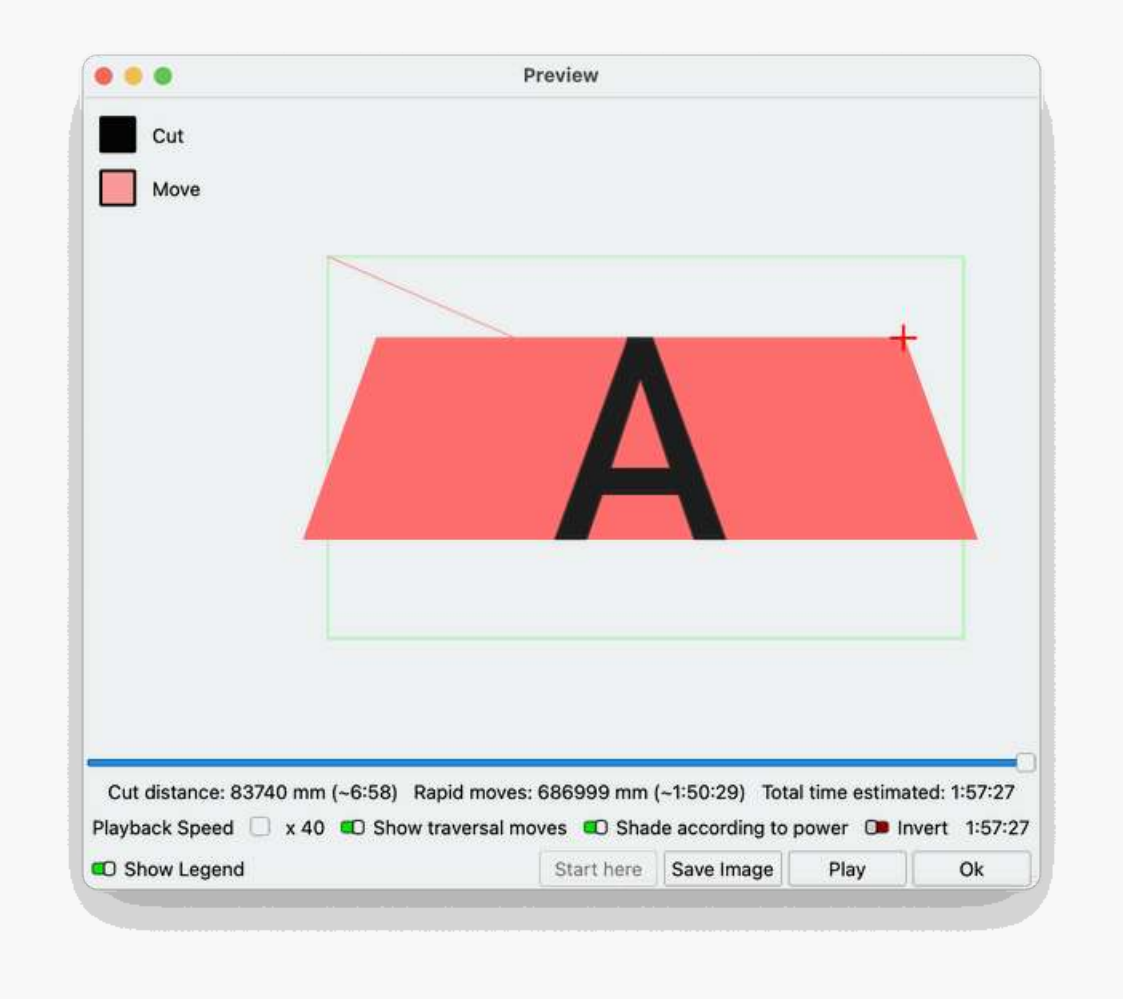
Double-Engraved Areas

Engraved Areas Opposite of Expectation



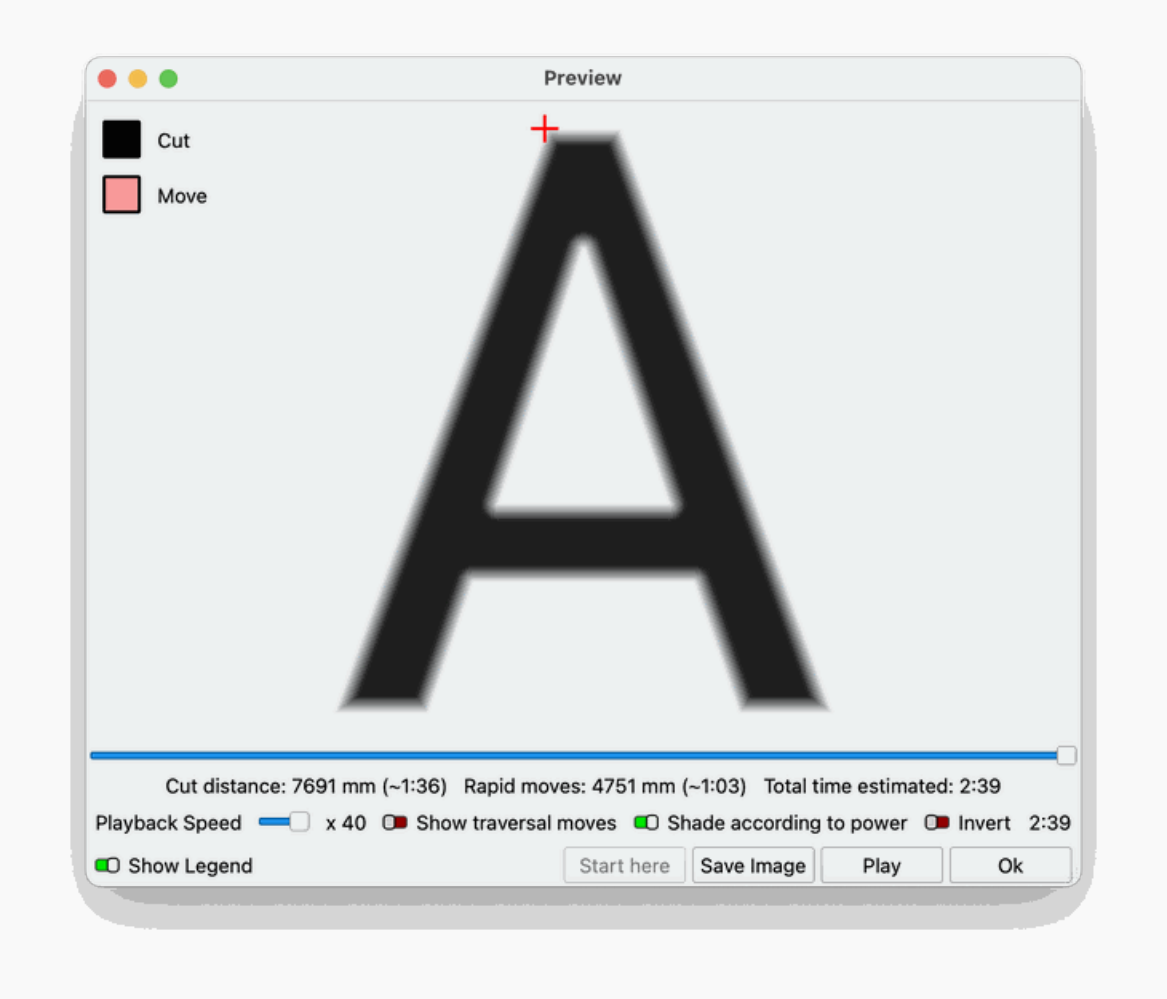
Engraved Areas Opposite of Expectation

"Excessive Overscanning"



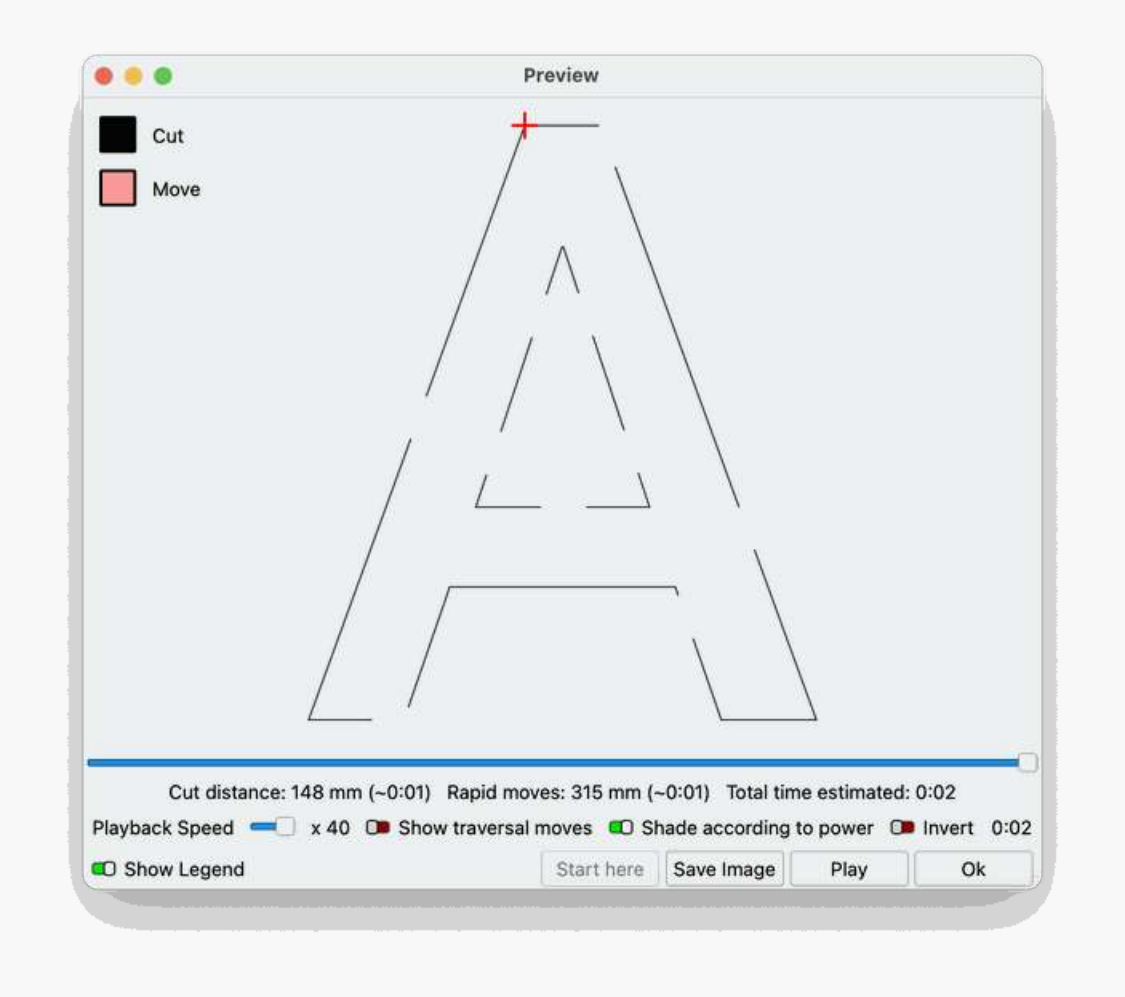
"Excessive Overscanning"

"Blurry Edges"



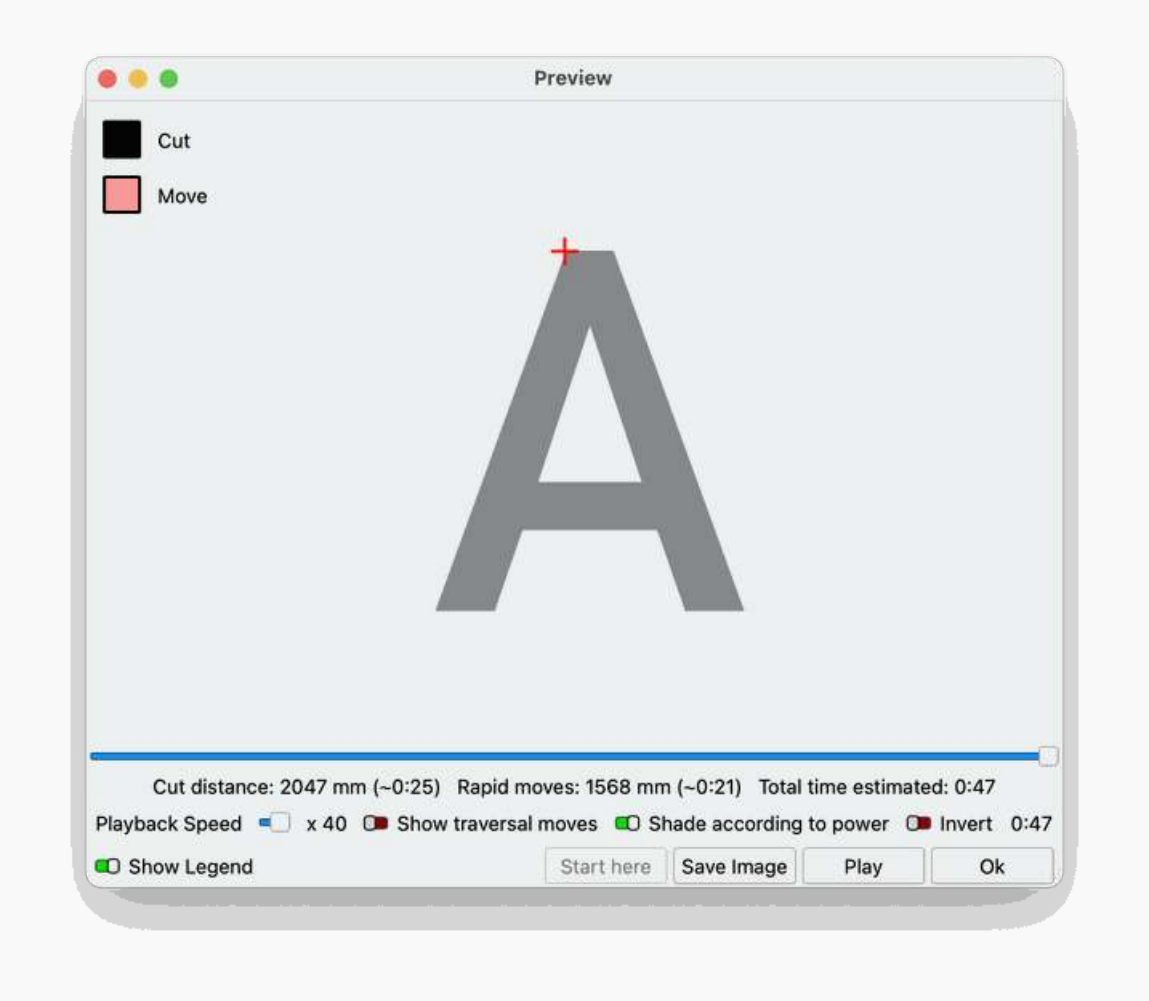
"Blurry Edges"

Gaps in Lines



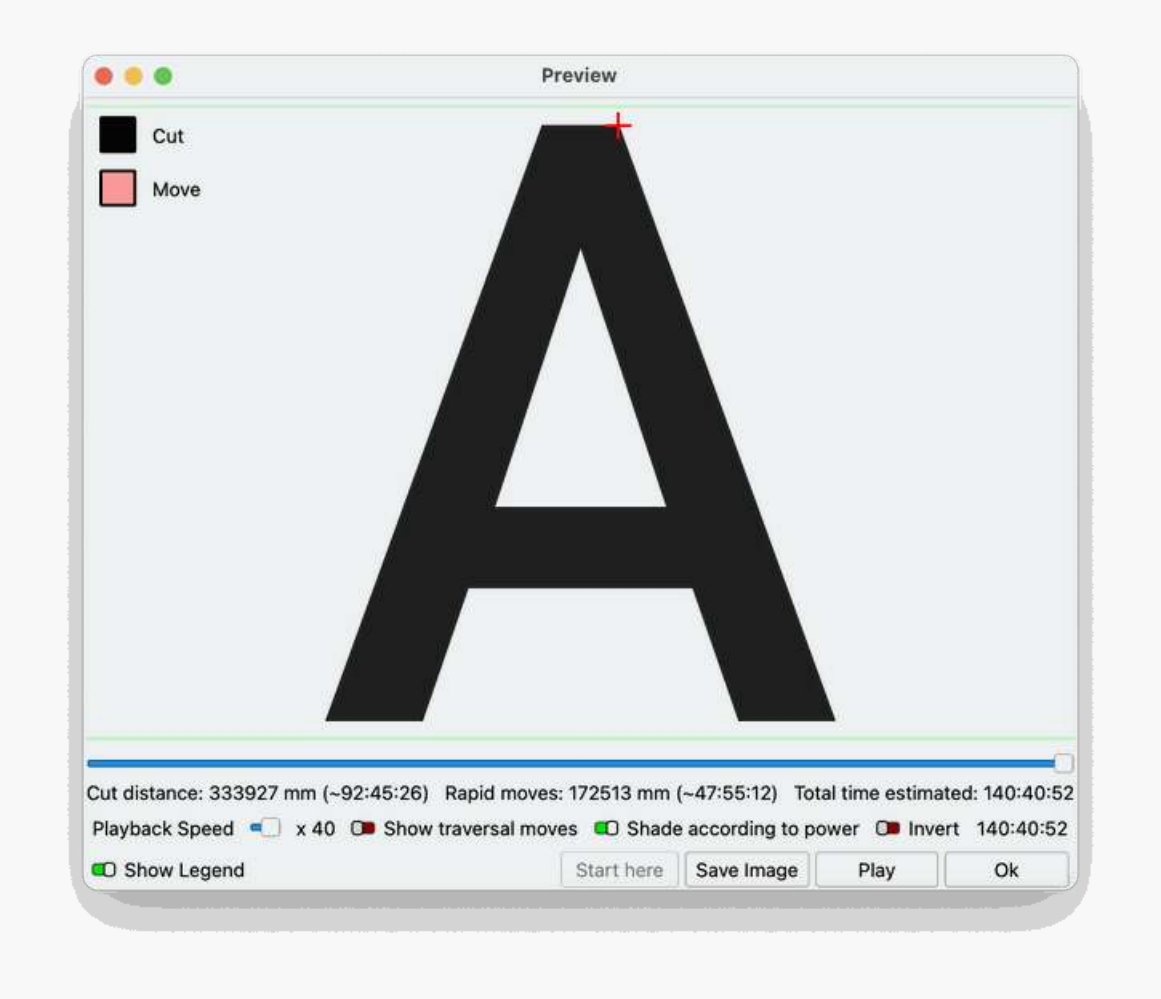
Gaps in Lines

Gray Shapes



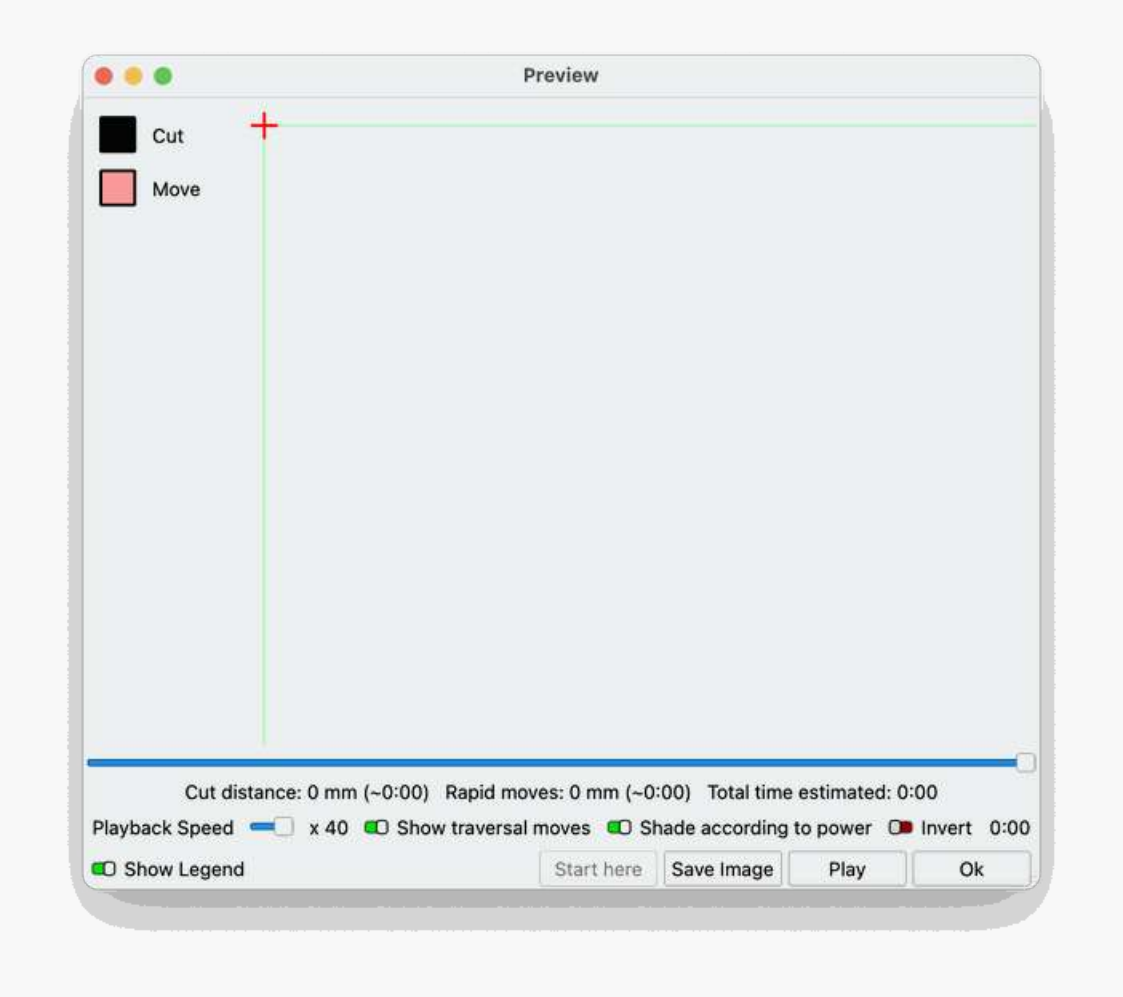
Gray Shapes

Inaccurate Time Estimates



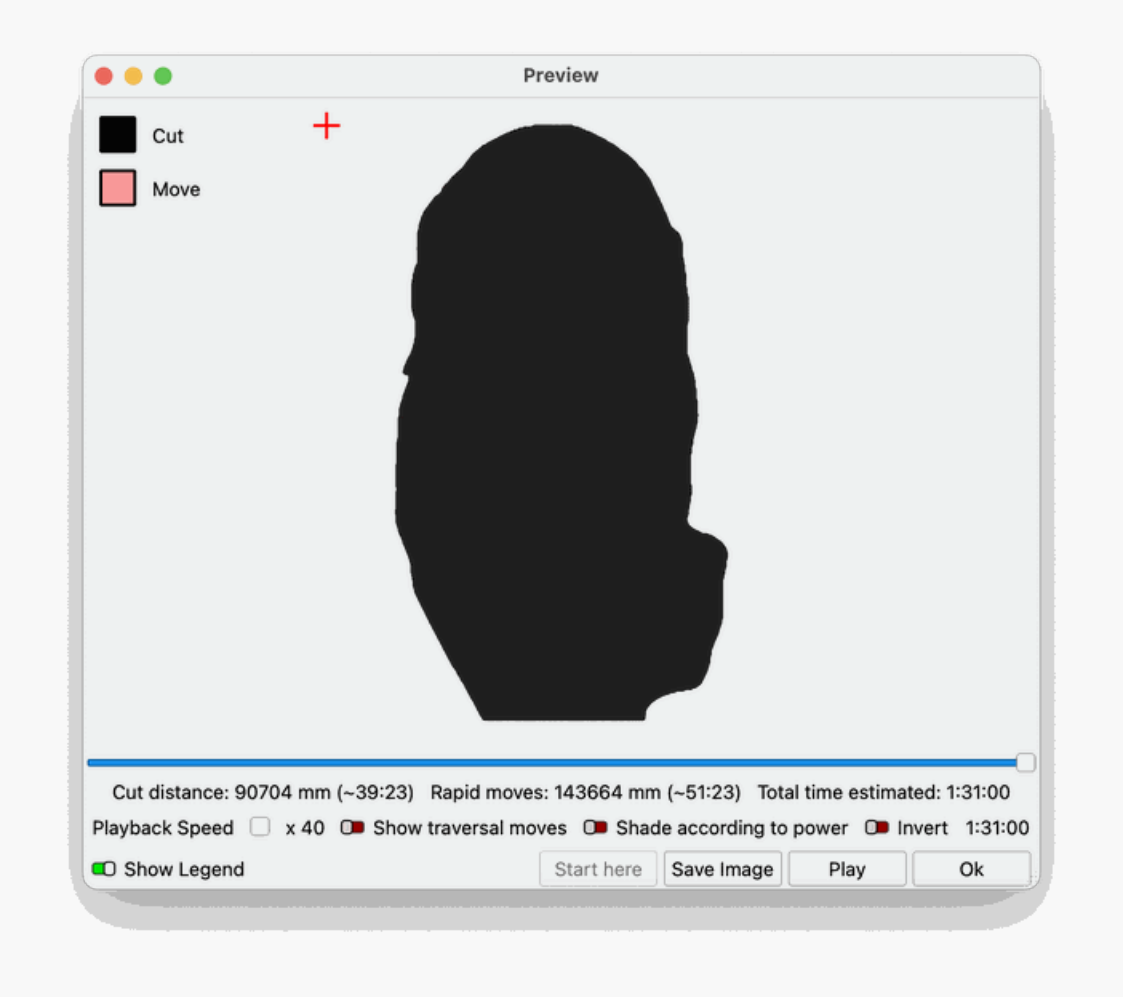
Inaccurate Time Estimates

Missing Shapes



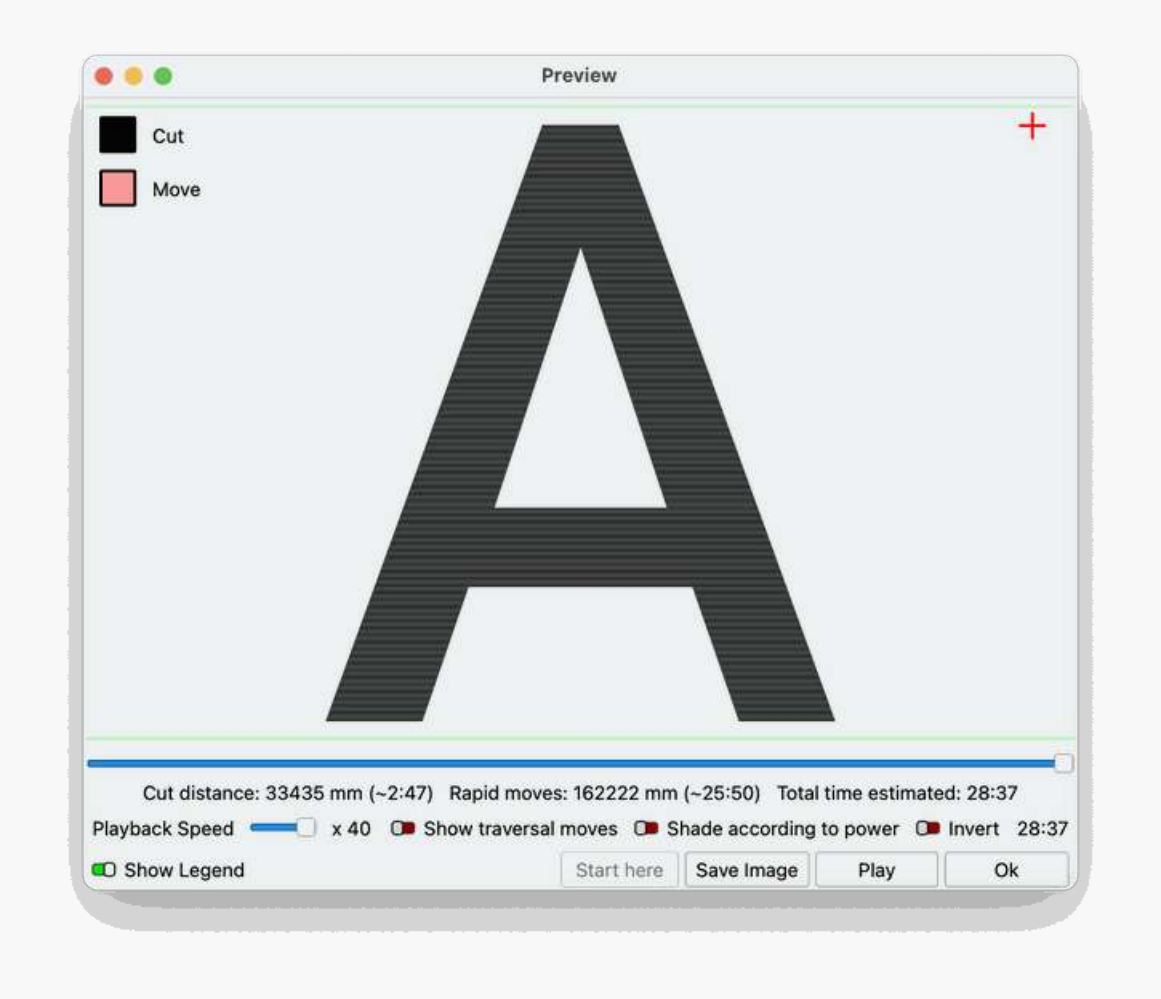
Missing Shapes

Solid Black Images



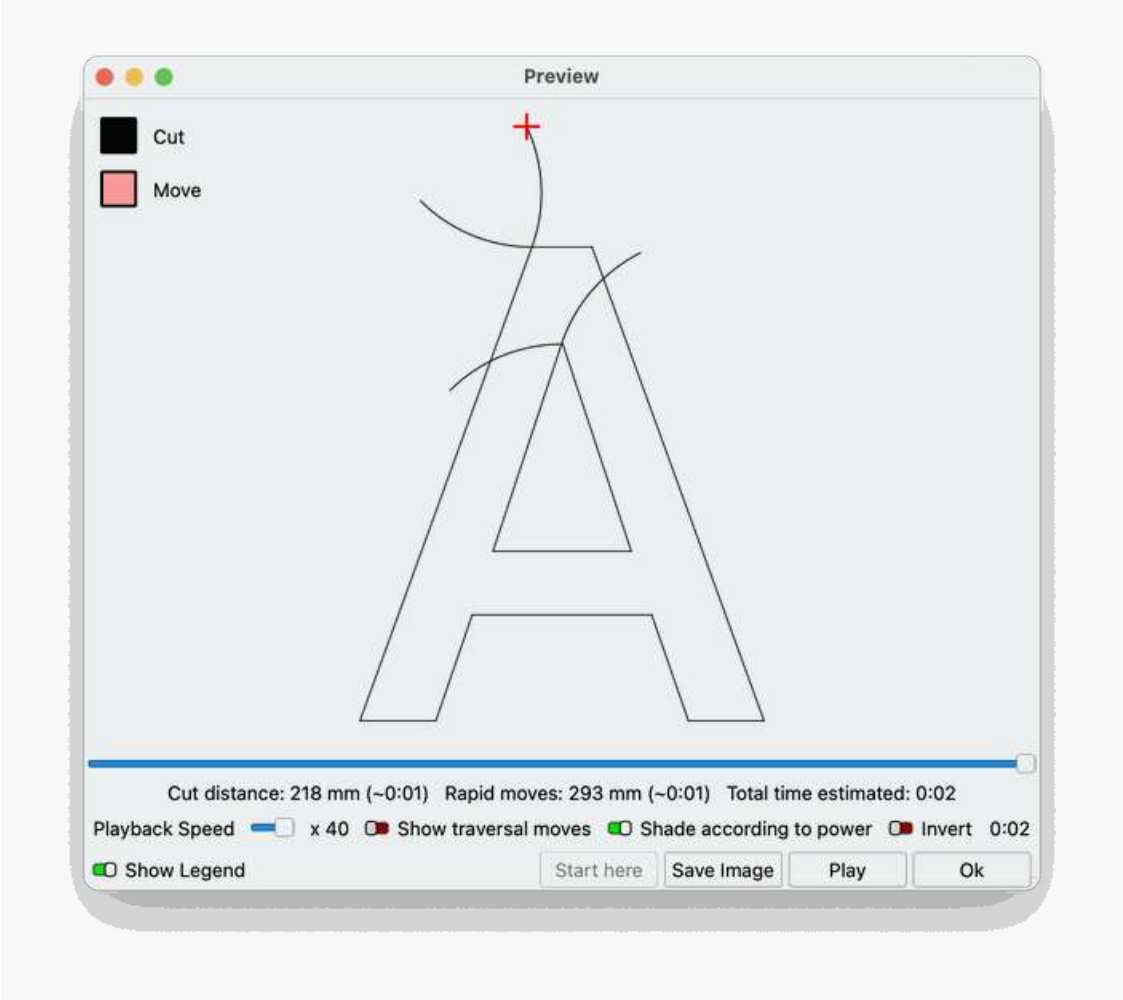
Solid Black Images

Strange Banding / Scan Lines Not Visible



Strange Banding / Scan Lines Not Visible

Added Lines



Description

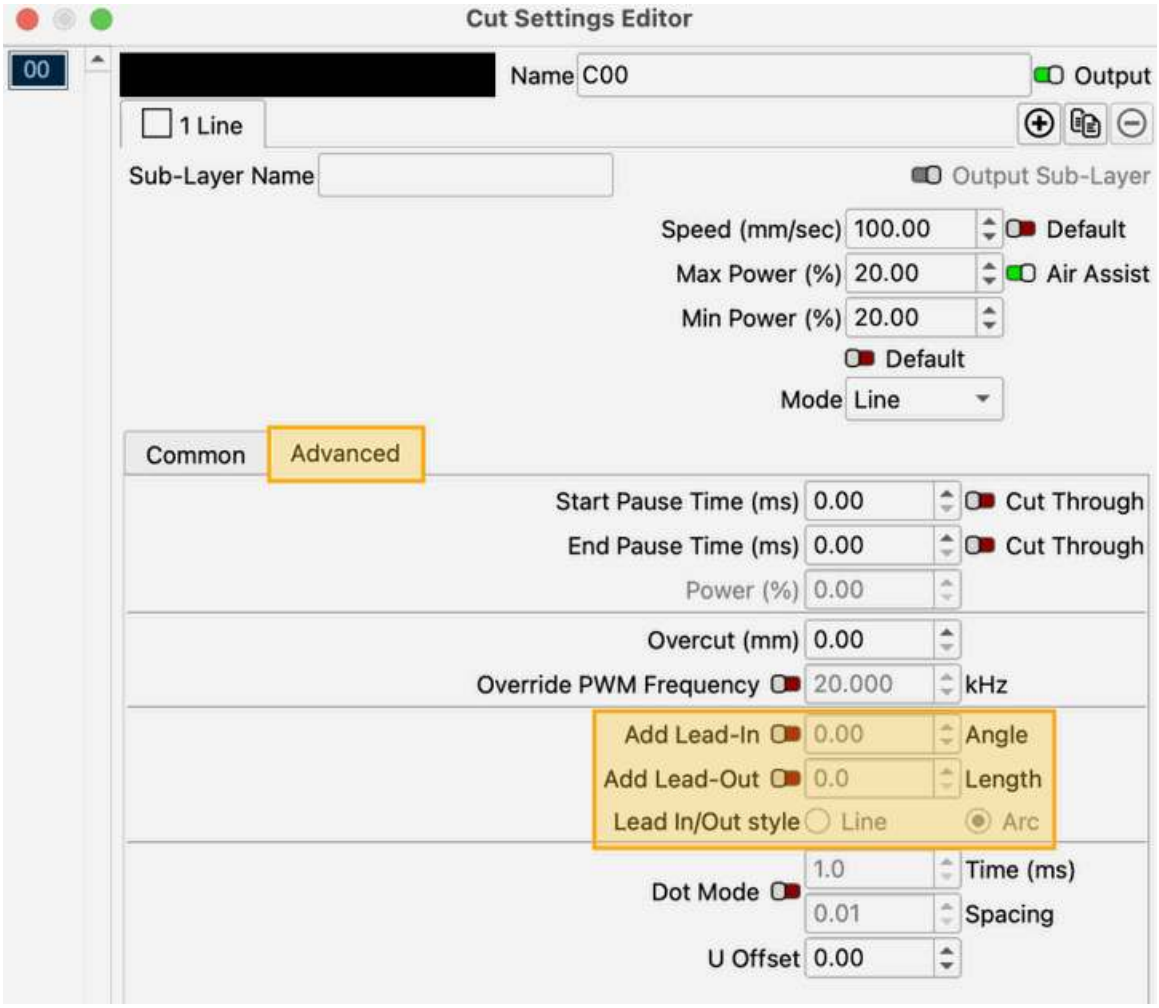
Extra lines appear at the starts and/or ends of closed cutting paths.

Solution

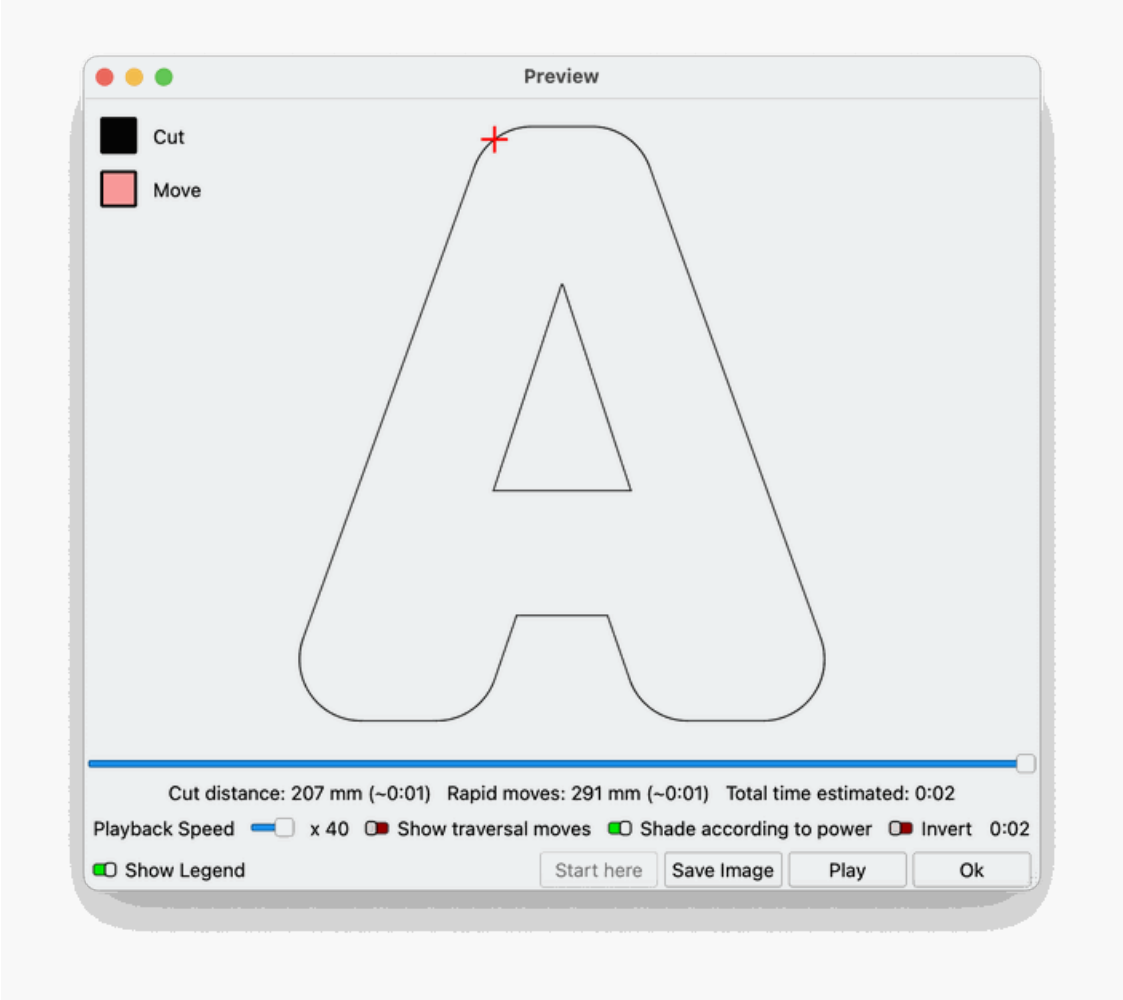
These lines are due to enabling Lead-In or Lead-Out in the Advanced tab of the Cut Settings Editor.

Lead-In / Lead-Out lines are intended to compensate for excessive burning at the starts or ends of cuts by extending the cut line, so the mark is left off of the cut part.

If you did not intend to enable this setting, open the Cut Settings Editor, switch to the Advanced tab, and toggle off **Add Lead-In** and/or **Add Lead-Out**.


Cut Settings Editor
00
Name C00
Output
1 Line
Sub-Layer Name
Output Sub-Layer
Speed (mm/sec) 100.00
Default
Max Power (%) 20.00
Air Assist
Min Power (%) 20.00
Default
Mode Line
Common
Advanced
Start Pause Time (ms) 0.00
Cut Through
End Pause Time (ms) 0.00
Cut Through
Power (%) 0.00
Overcut (mm) 0.00
Override PWM Frequency
20.000
kHz
Add Lead-In 0.00
Angle
Add Lead-Out 0.0
Length
Lead In/Out style Line Arc
1.0
Time (ms)
Dot Mode
0.01
Spacing
U Offset 0.00

Ballooned or Incorrectly Sized Shapes



Description

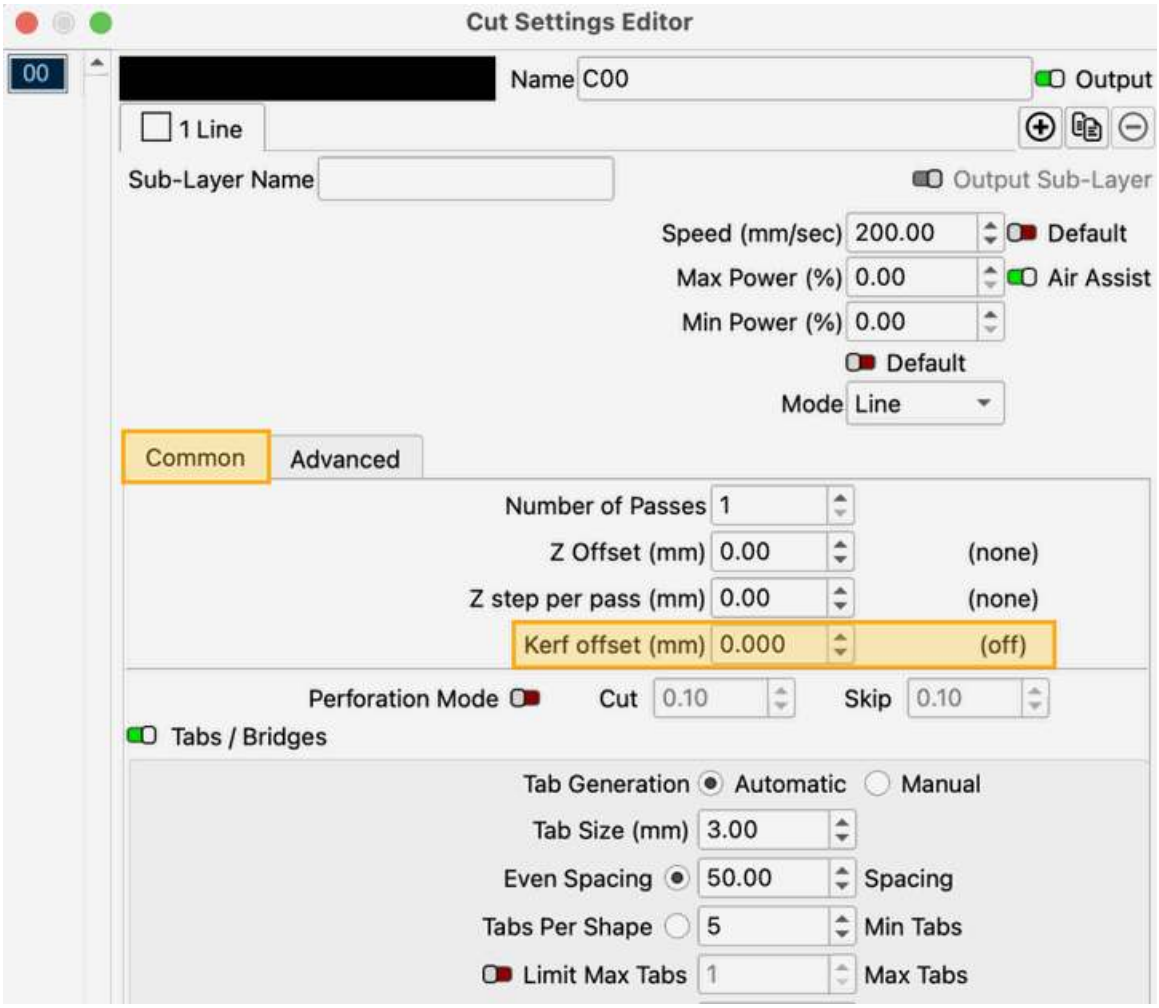
Shapes are sized incorrectly and rounded on some corners, in extreme cases appearing balloon-like.

Solution

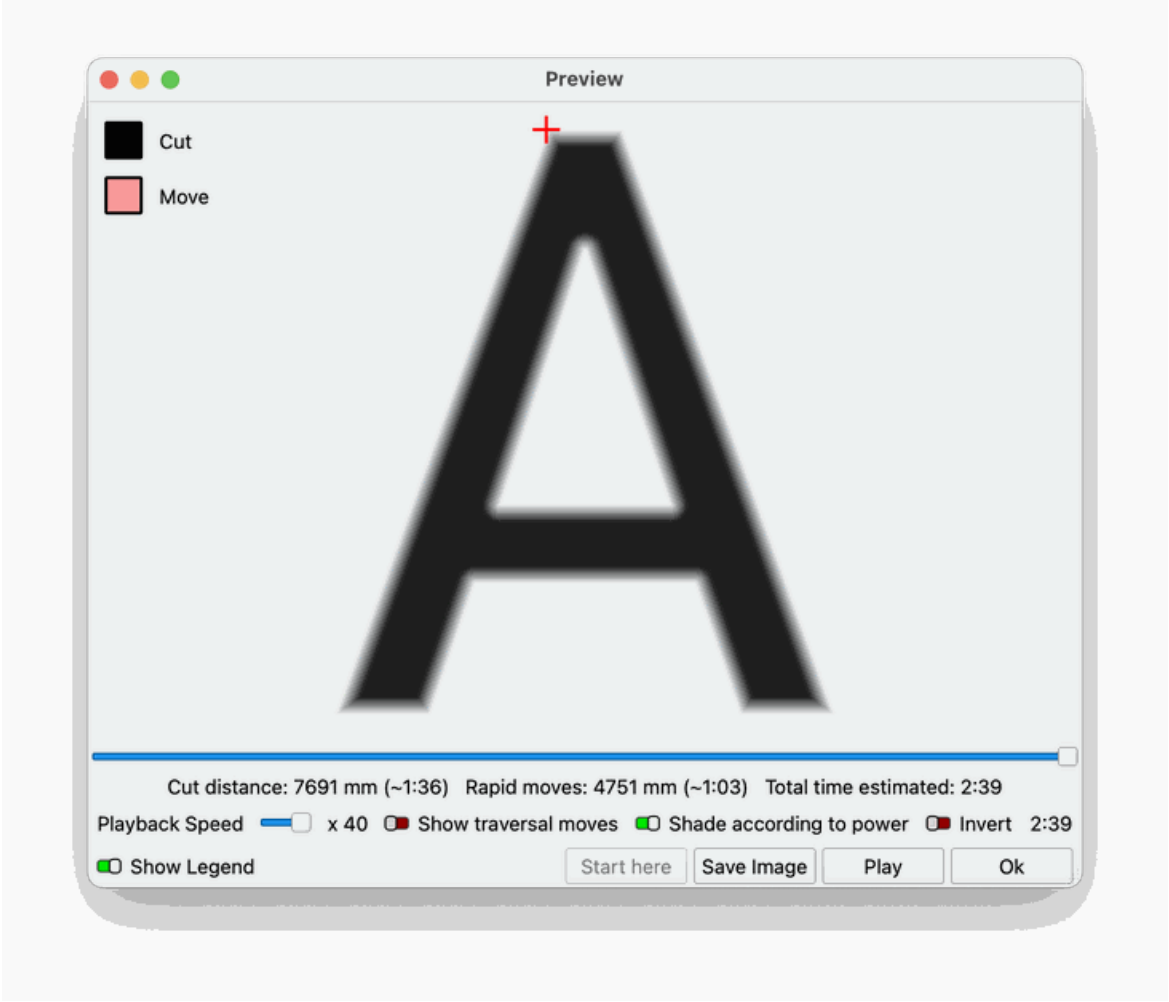
This effect is due to unintentionally enabling Kerf Offset, or setting the value too high.

Kerf refers to the thickness or width of a cut made by a cutting tool. Kerf Offset compensates for the extra material removed by offsetting the path of the laser to the outside of closed shapes (for positive values) or the inside (for negative values).

- If you did not intend to enable Kerf Offset, open the Cut Settings Editor and reduce the value to 0.00.
- If you intended to enable Kerf Offset, but the distortion is extreme, try reducing the value.


Cut Settings Editor
00
Name C00
Output
1 Line
Sub-Layer Name
Output Sub-Layer
Speed (mm/sec) 200.00
Default
Max Power (%) 0.00
Air Assist
Min Power (%) 0.00
Default
Mode Line
Common
Advanced
Number of Passes 1
Z Offset (mm) 0.00
(none)
Z step per pass (mm) 0.00
(none)
Kerf offset (mm) 0.000
(off)
Perforation Mode
Cut 0.10
Skip 0.10
Tabs / Bridges
Tab Generation
Automatic
Manual
Tab Size (mm) 3.00
Even Spacing
50.00
Spacing
Tabs Per Shape
5
Min Tabs
Limit Max Tabs
1
Max Tabs

Blurry Edges



Description

The edges of graphics set to Fill Mode are blurry.

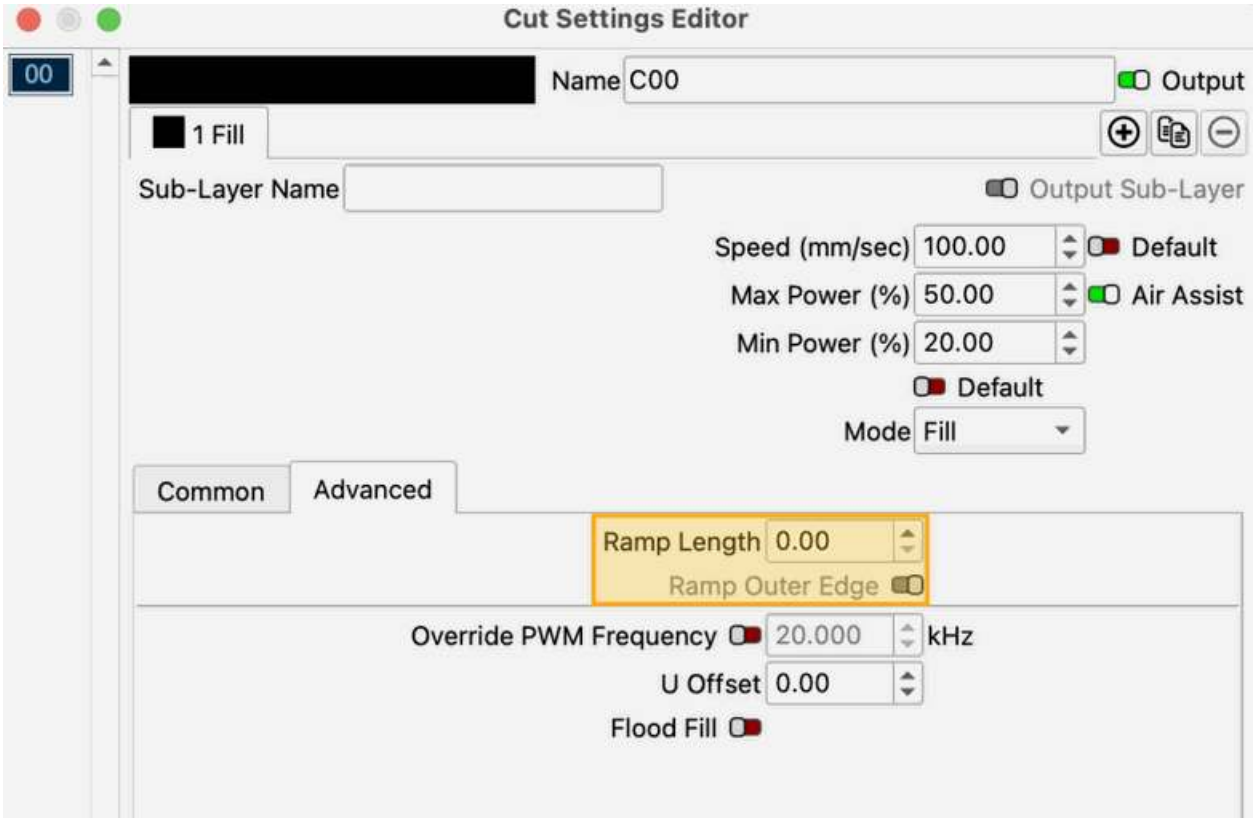
Solution

Blurred edges are due to applying a Ramp Length to layers set to **Fill Mode**.

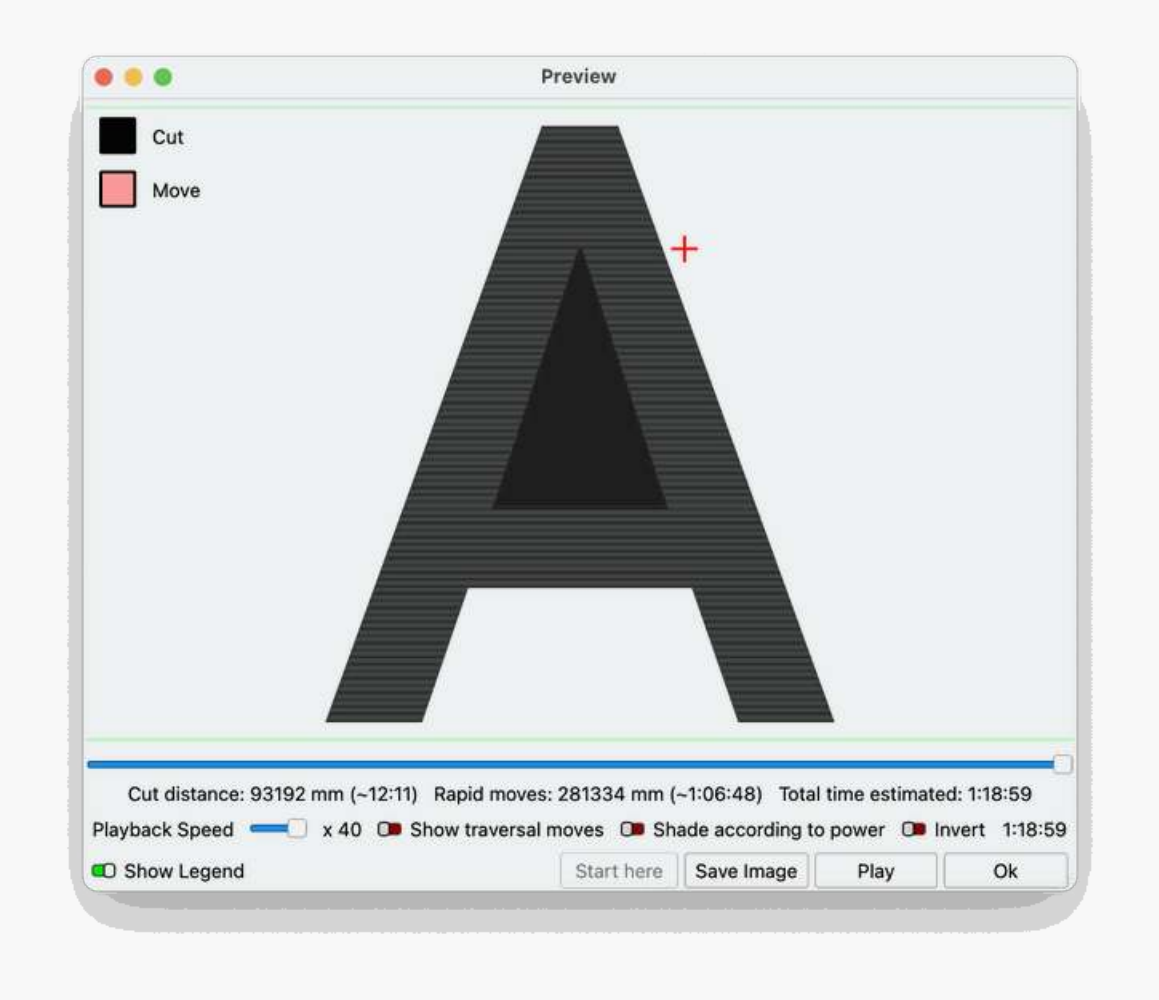
Ramp Length slopes the sides of an engraving by varying the laser power. The ramp length controls how far from the design the slope will extend. This setting is useful if you create rubber stamps, particularly if the designs have thin regions that need extra support.

The ramping effect is only visible in the **Preview** window When Shade according to power is toggled on.

To disable the ramping effect, open the Cut Settings Editor and set Ramp Length to 0.00.


Cut Settings Editor
00
Name C00
Output
1 Fill
Sub-Layer Name
Output Sub-Layer
Speed (mm/sec) 100.00
Default
Max Power (%) 50.00
Air Assist
Min Power (%) 20.00
Default
Mode Fill
Common
Advanced
Ramp Length 0.00
Ramp Outer Edge
Override PWM Frequency 20.000 kHz
U Offset 0.00
Flood Fill

Double-Engraved Areas



Description

Areas of your graphics that you intend to engrave only once — or not at all — are instead engraved twice.

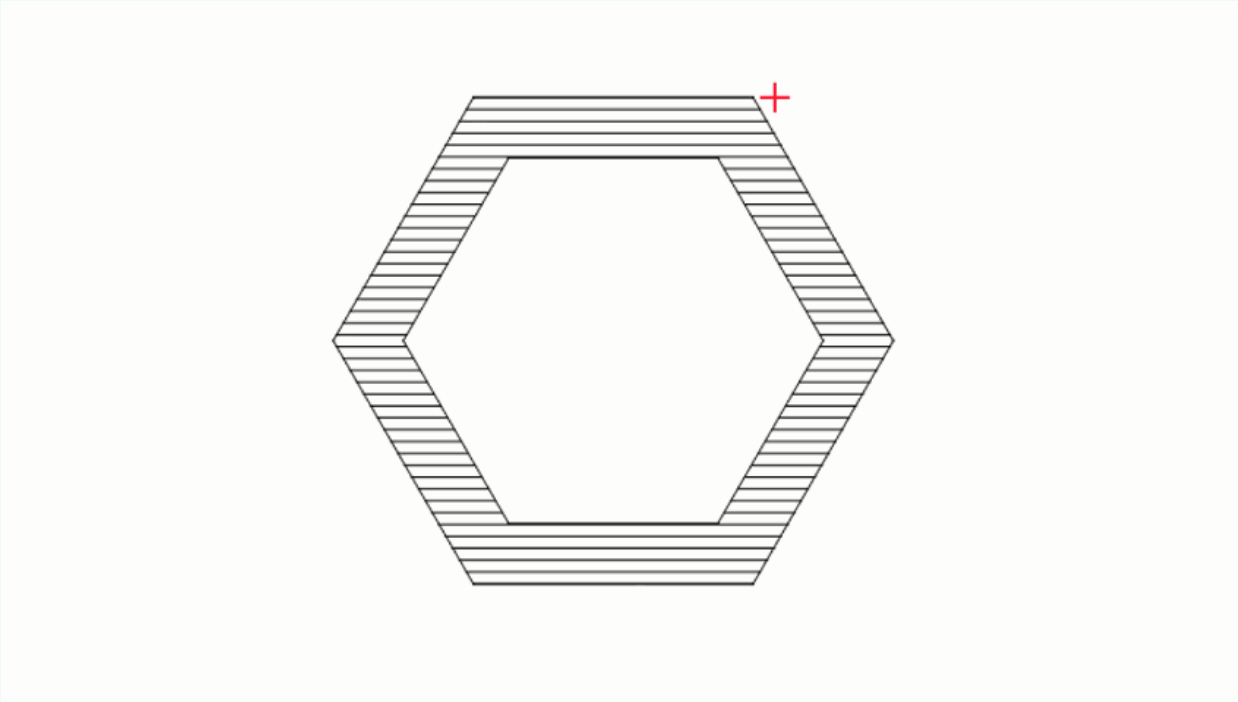
Solution

Graphics set to Fill Mode engrave differently depending on whether they are set to the same or different layers.

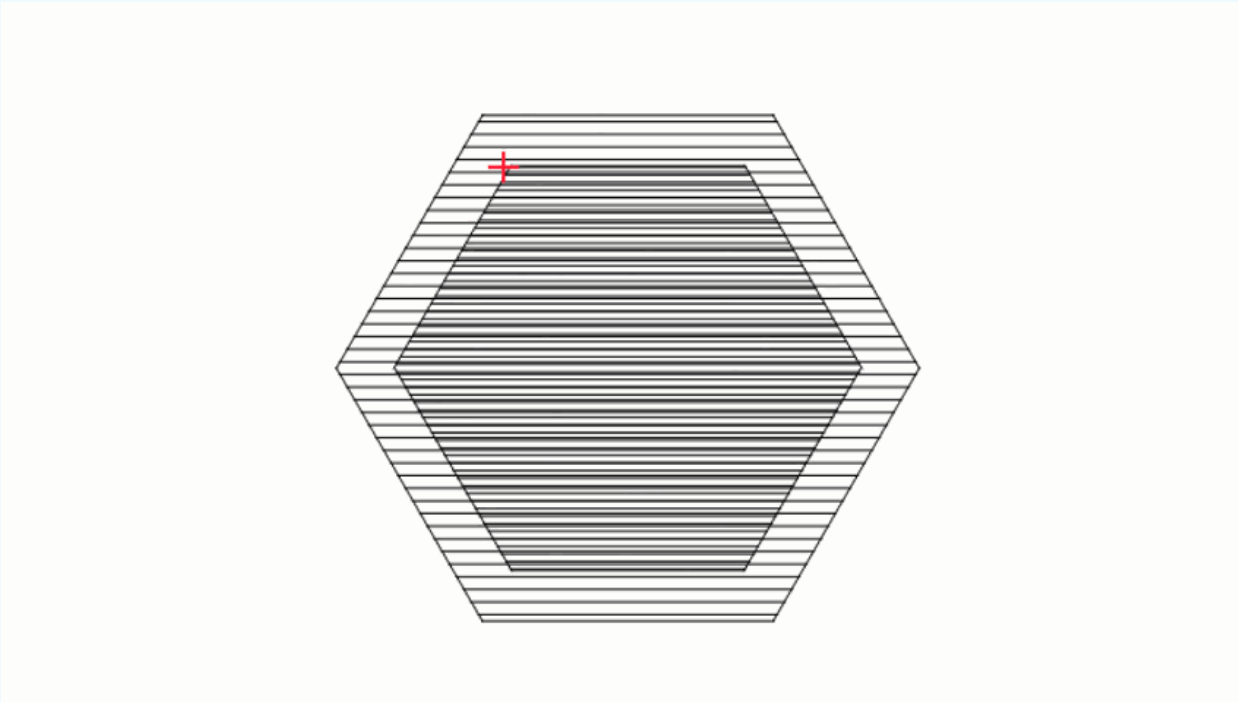
- If they are set to the same layer, and one shape is inside the other, the area between both outlines is engraved.
- If they are set to the same layer and they're partially overlapping, but neither contains the other, the area where they overlap is not engraved.
- If they are set to different layers, both shapes are engraved in their entirety, regardless of whether one contains the other, or they're overlapping.

- See Fill Mode and How to Invert a Vector Engraving for more information on these concepts.

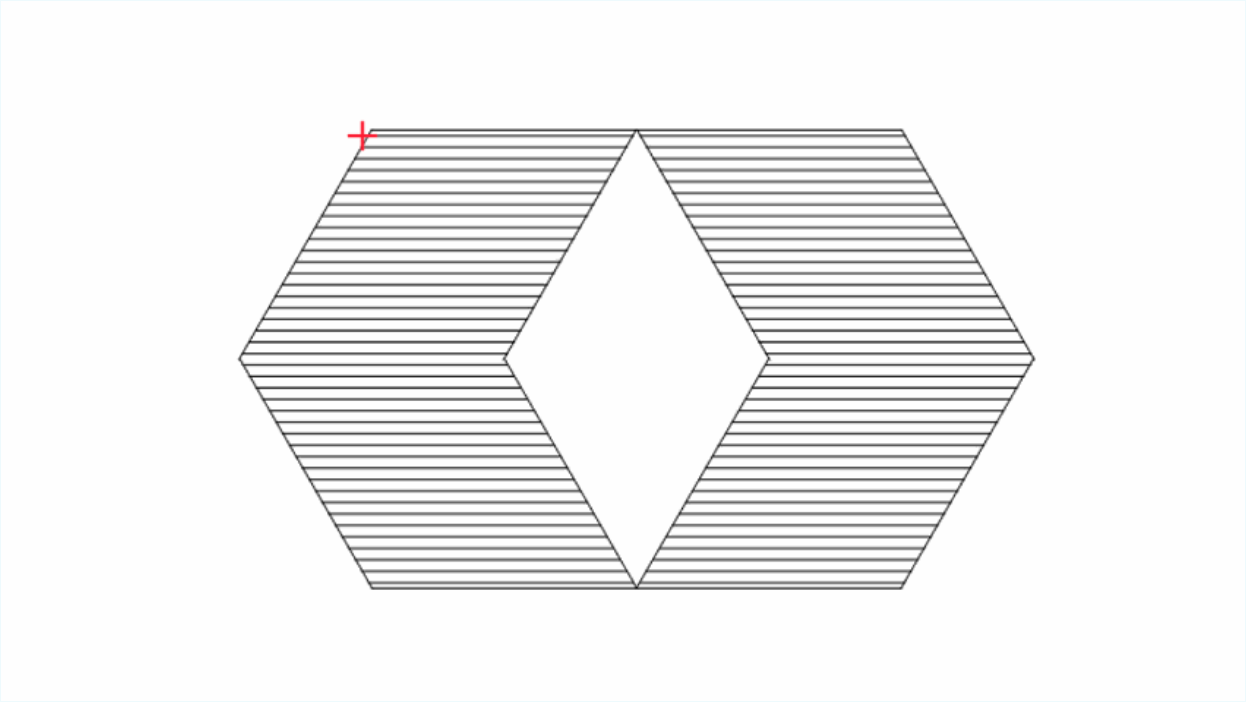
Click here for examples fully and partially overlapping shapes set to the same or different layers



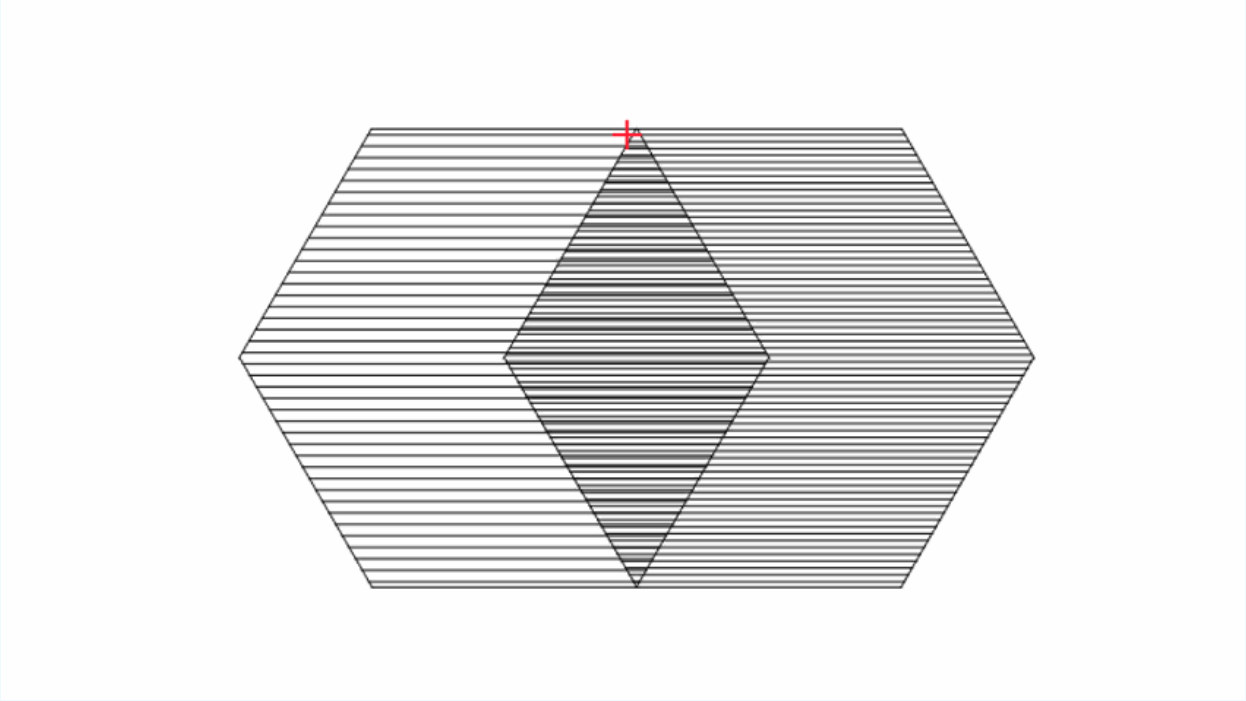
Fully overlapping shapes set to the same layer



Fully overlapping shapes set to different layers



Partially overlapping shapes set to the same layer

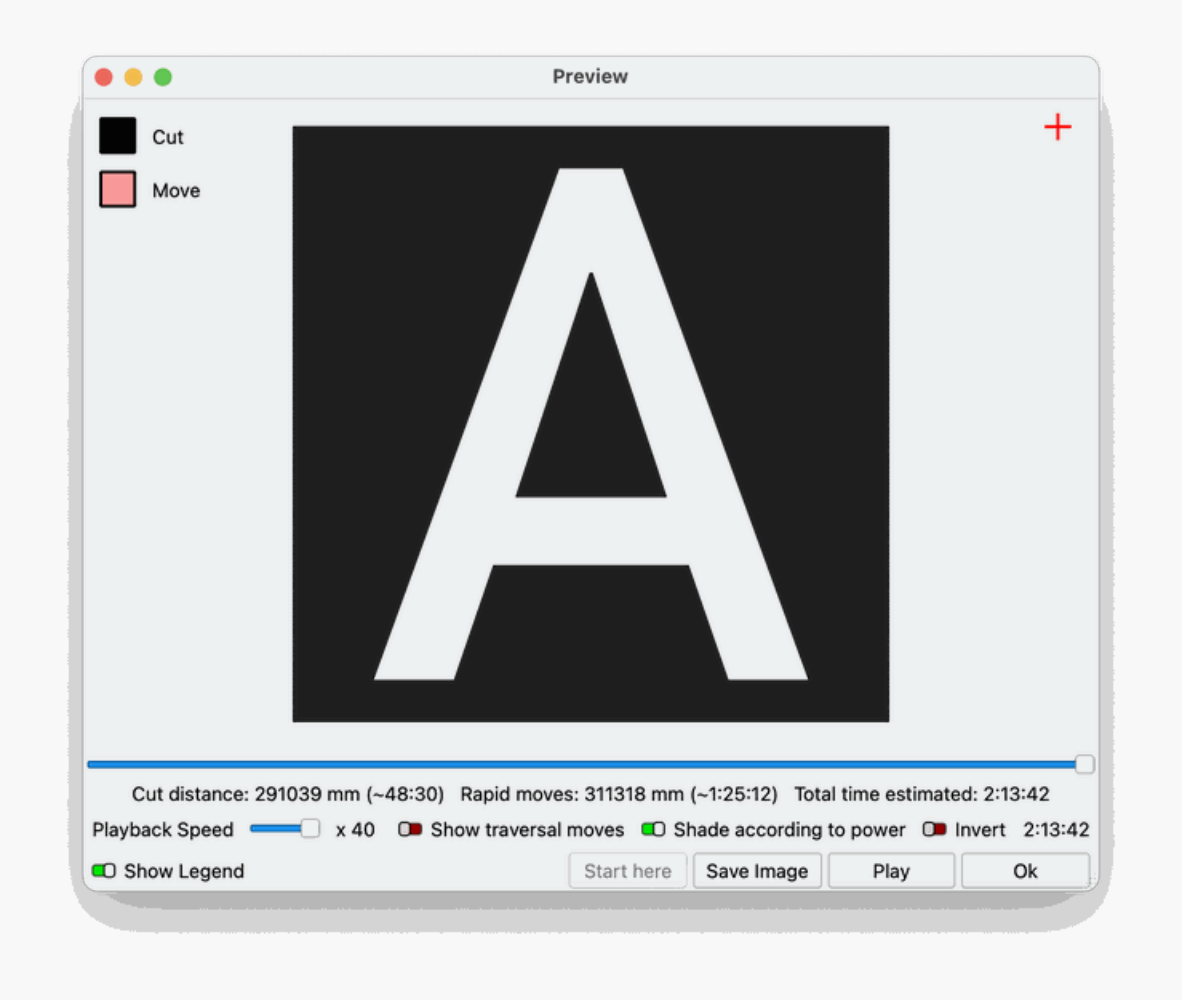


Partially overlapping shapes set to different layers

- Try setting overlapping shapes to the same layer to see if the new result in the Preview window matches your expectation.

- If you are using a Filled View Style, try switching to **Wireframe** mode to make sure there aren't any unwanted graphics hidden behind other graphics in your design. Delete them if necessary.

Engraved Areas Opposite of Expectation



Description

Areas of your graphics that you intend to engrave are not engraved, and areas you don't intend to engrave are.

Solution

This behavior may be due to having vector graphics set up incorrectly, having the **Negative Image** switch in the incorrect position for an image engraving, or a misunderstanding of the **Invert** option in the **Preview** window.

- To invert a vector engraving, so that the opposite of whatever area is currently set to be engraved is engraved instead, you need to add an outline around it, and make sure the outline is set to the same layer as the original vector graphic.

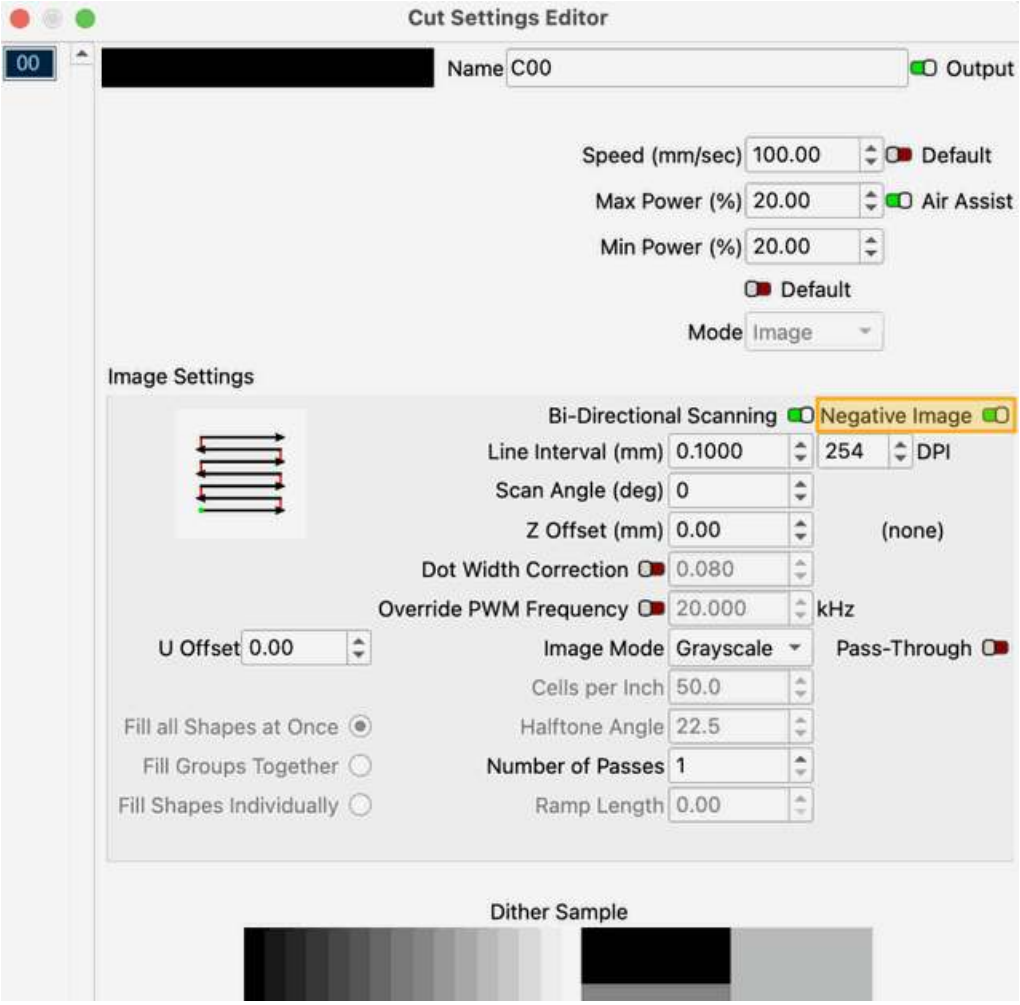
  See How to Invert a Vector Engraving for more information.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=EAlw371Ey0M

- To invert an image engraving, toggle on the **Negative Image** switch in the Cut Settings Editor or Adjust Image window.

  The Negative Image option is useful when you're engraving on slate or another material whose original surface is darker than the mark your laser will make on it.

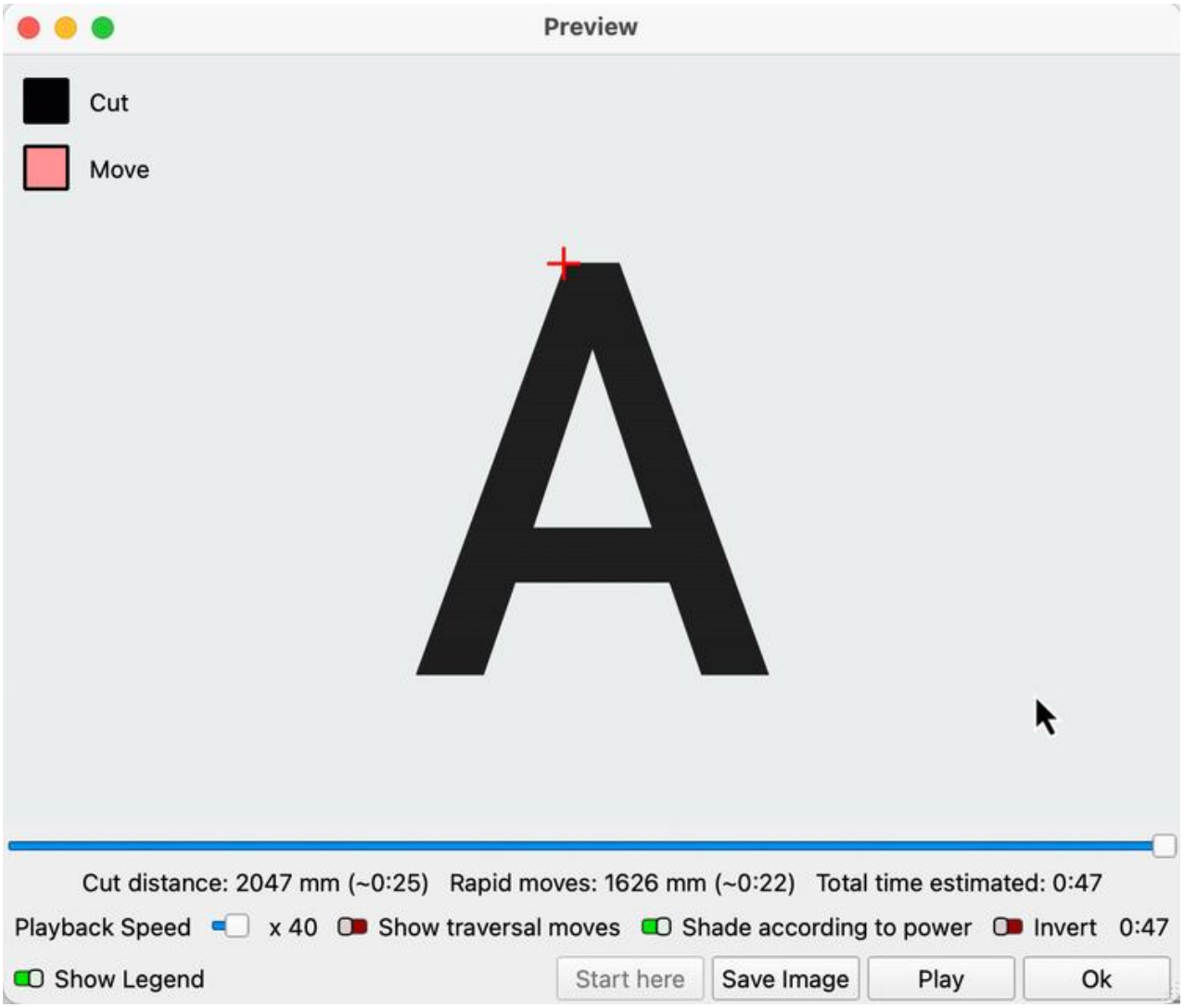
  If you are working with a light material, you should not enable this switch.



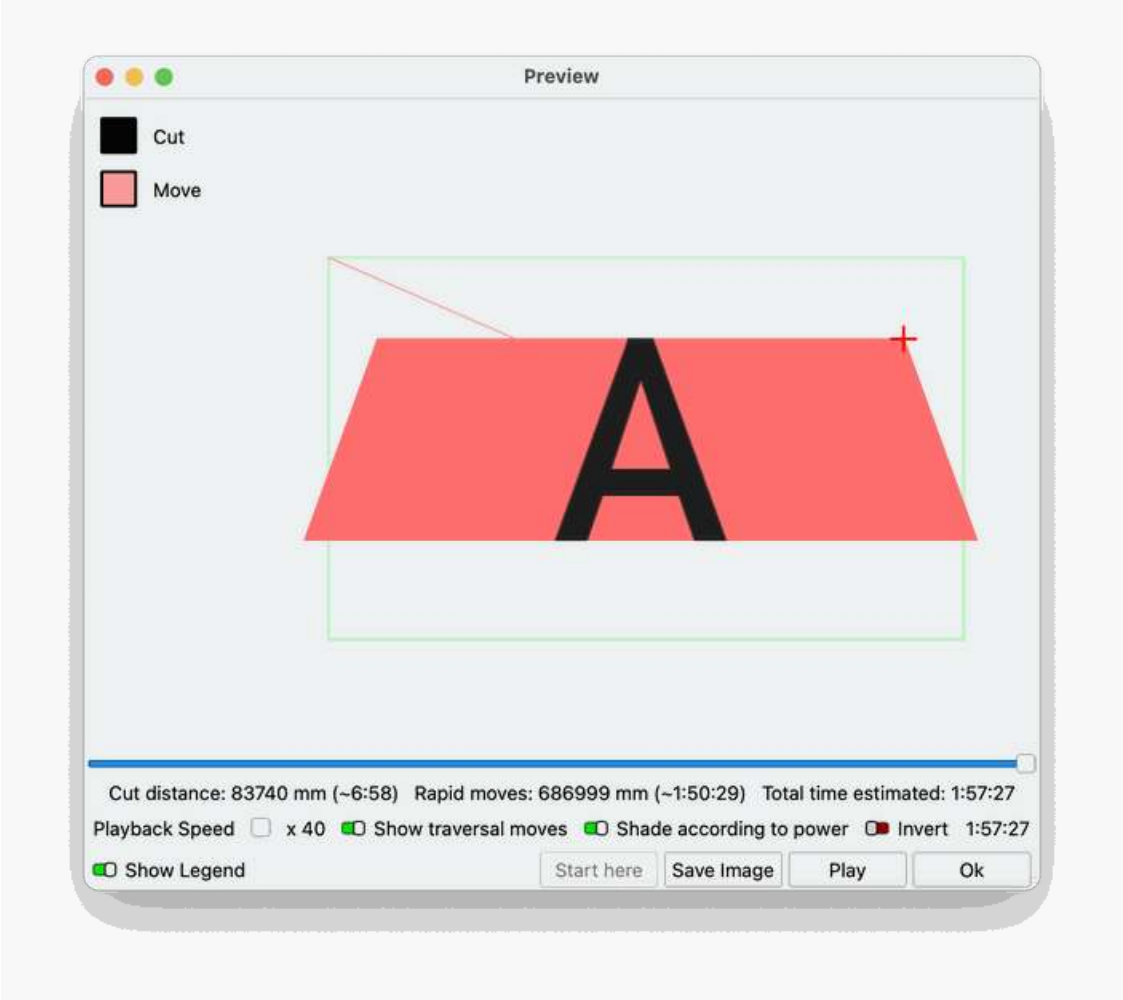
- If the Invert switch is toggled on in the **Preview** window, engraved areas will show in white, and unengraved areas in black. This is intended to represent how your artwork will look when it's engraved on slate, granite, or another dark material upon which the laser's mark will be lighter than the original surface finish.

  This switch only affects the representaton in the Preview window and makes no difference to how your laser will actually engrave your graphics.

  - If you are not engraving on a dark material, toggle off Invert to see a representation of your engraving on a light material.
  - If you are engraving on a dark material, everything is OK!


Preview
Cut
Move
Cut distance: 2047 mm (~0:25) Rapid moves: 1626 mm (~0:22) Total time estimated: 0:47
Playback Speed x 40 Show traversal moves Shade according to power Invert 0:47
Show Legend
Start here
Save Image
Play
Ok

Excessive Overscanning



Description

Red lines depicting traversal moves show the laser traveling a significant distance past the edges of graphics, or beyond the edge of the laser's work area, depicted as a green square/rectangle.

The red lines show Overscanning, additional moves added to the beginning and end of each line to give the laser space to speed up before firing, and slow down afterward, which in turn prevents the edges of engraved areas from burning darker than the centers.

Excessive Overscanning can lead to an unnecessary increase in overall job time, or out-of-bounds errors from your laser.

Solution

Error Messages

GCode-based lasers

If you enable Overscanning for a GCode-based laser and there isn't enough room at the edge of the job to accommodate the extra travel distance, you may see LightBurn's **Cut may be out of bounds due to overscan settings** error, or your laser's controller may return an ALARM:2 error.

DSP lasers

If a job on a DSP laser requires Overscanning and there isn't enough room at the edge of the job to accommodate it, your DSP controller may return the **Not Enough Extend Space** error.

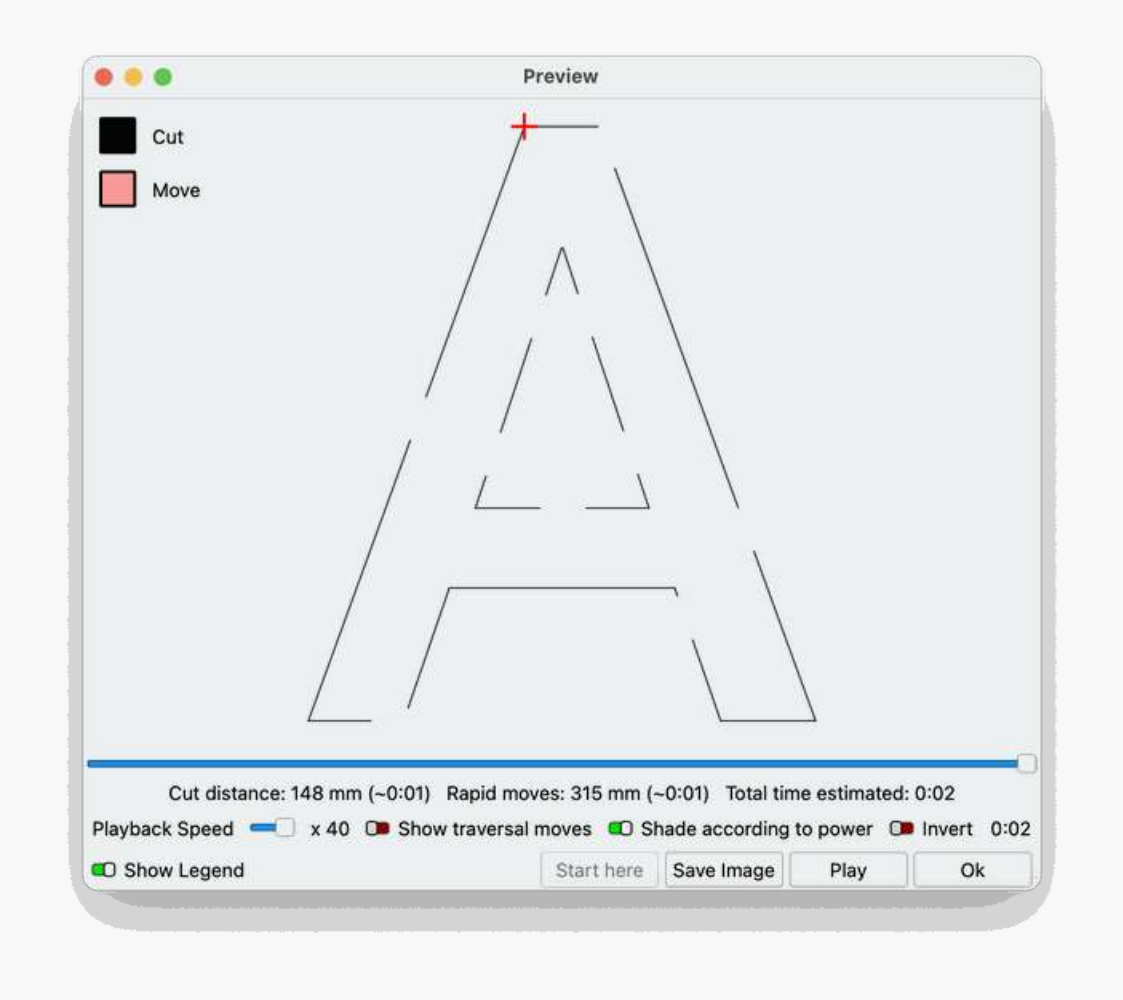
Because **Overscanning** requires your laser to travel extra distance outside the exact boundaries of your graphics, if your graphics aren't positioned properly, or you are too close to the edge of your machine when using Current Position or User Origin as your **Start From** mode, you may accidentally command your laser to travel outside of its physical limits, once the Overscanning distance is included.

To correct excessive Overscanning:

- Reduce Speed or Overscanning percentage (if you have a GCode-based laser).
- Move your graphics in toward the center of your Workspace, if you are using Absolute Coordinates positioning.
- If you are using Current Position or User Origin to set a custom Origin, set it closer to the center of your laser's work area.

See Overscanning for more information.

Gaps in Lines



Description

Some sections of graphics set to Line Mode are skipped.

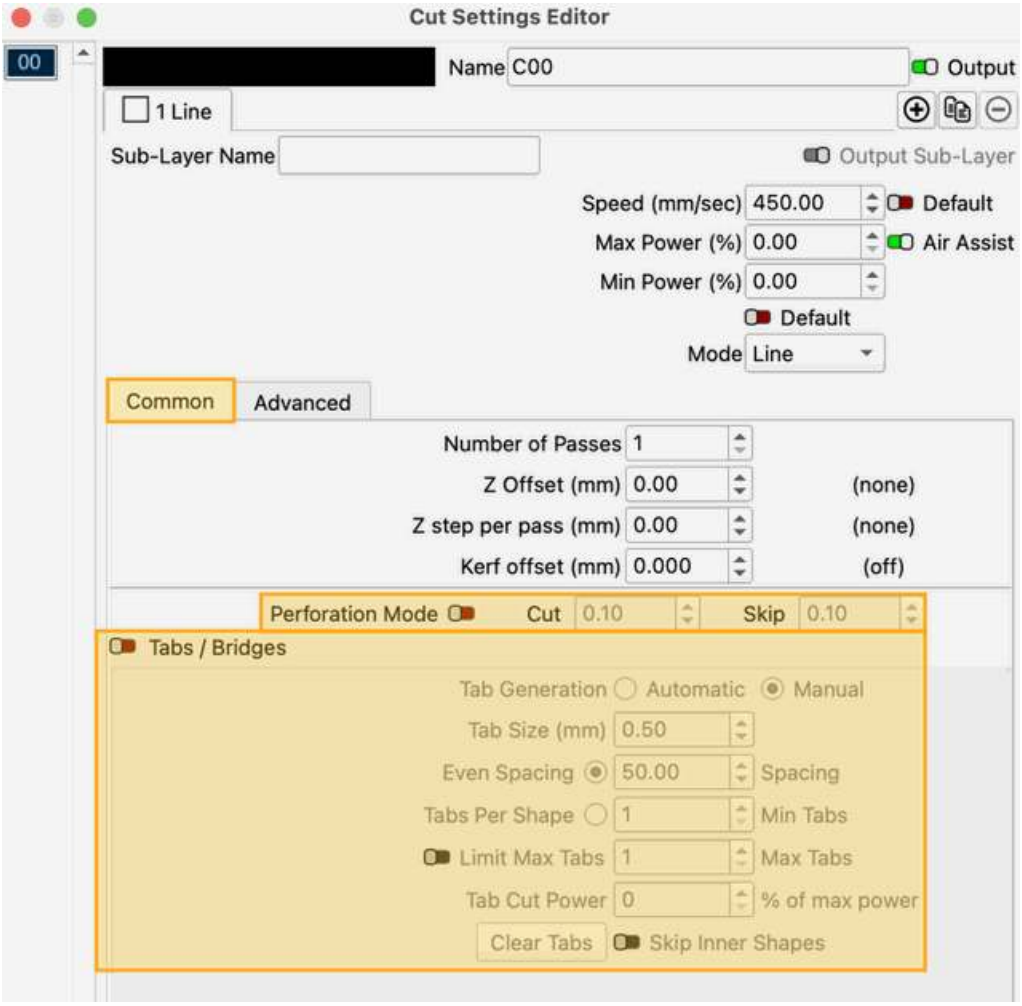
Solution

This behavior is due to enabling one or more of three special options in the Cut Settings Editor.

- Tabs / Bridges are small sections of cuts that the laser is commanded to skip, used to prevent objects that you are cutting from immediately falling out of the base material.

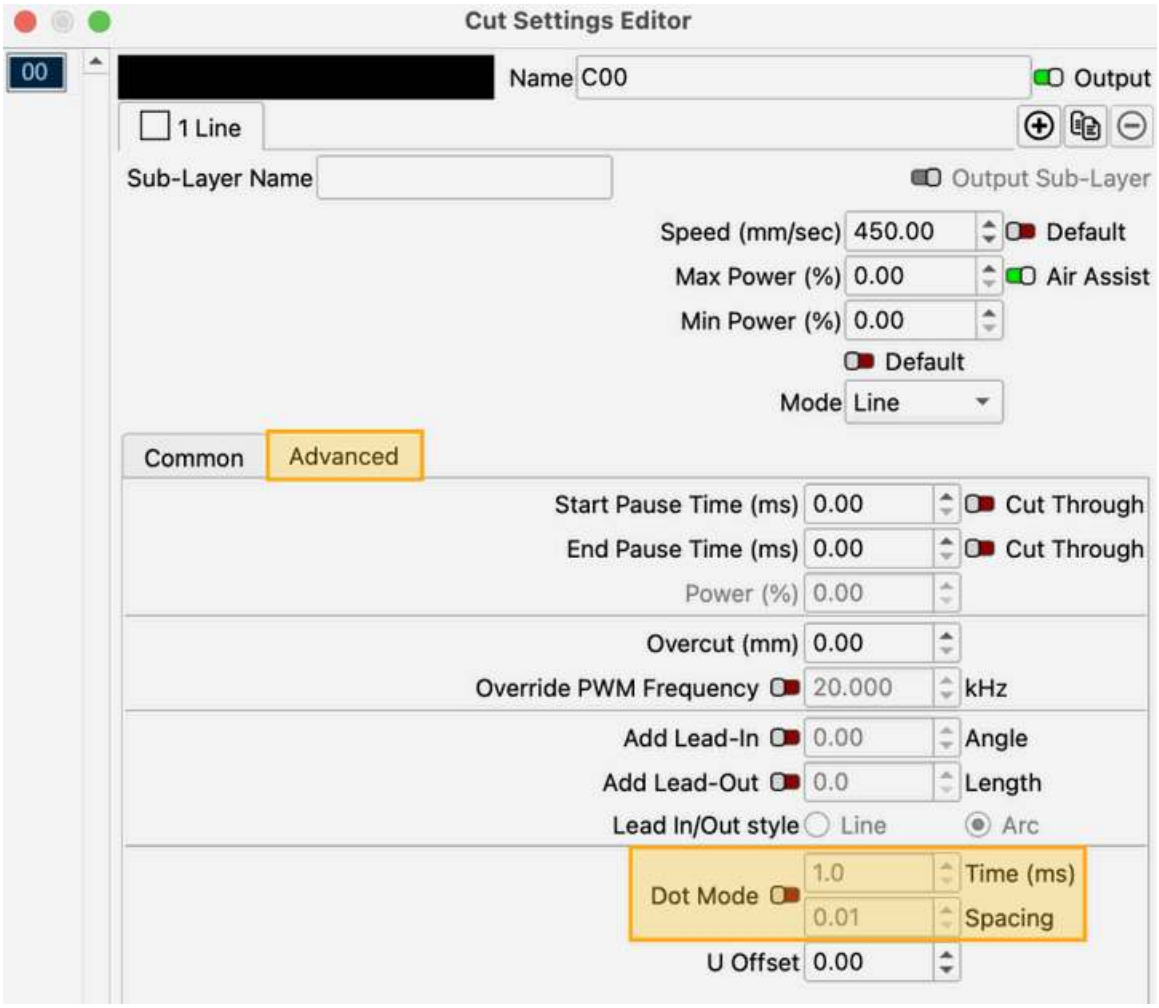
  If you did not intend to enable Tabs for this layer, toggle off the switch in the Common Settings tab of the Cut Settings Editor.
- Perforation Mode tells your laser to **Cut** and **Skip** sections of your graphics at specified intervals. Perforation mode is useful if you're trying to cut fold lines in card stock, stitch holes in leather, or make dashed lines.

To disable this setting, toggle off the Perforation Mode switch in the Common Settings tab Cut Settings Editor.

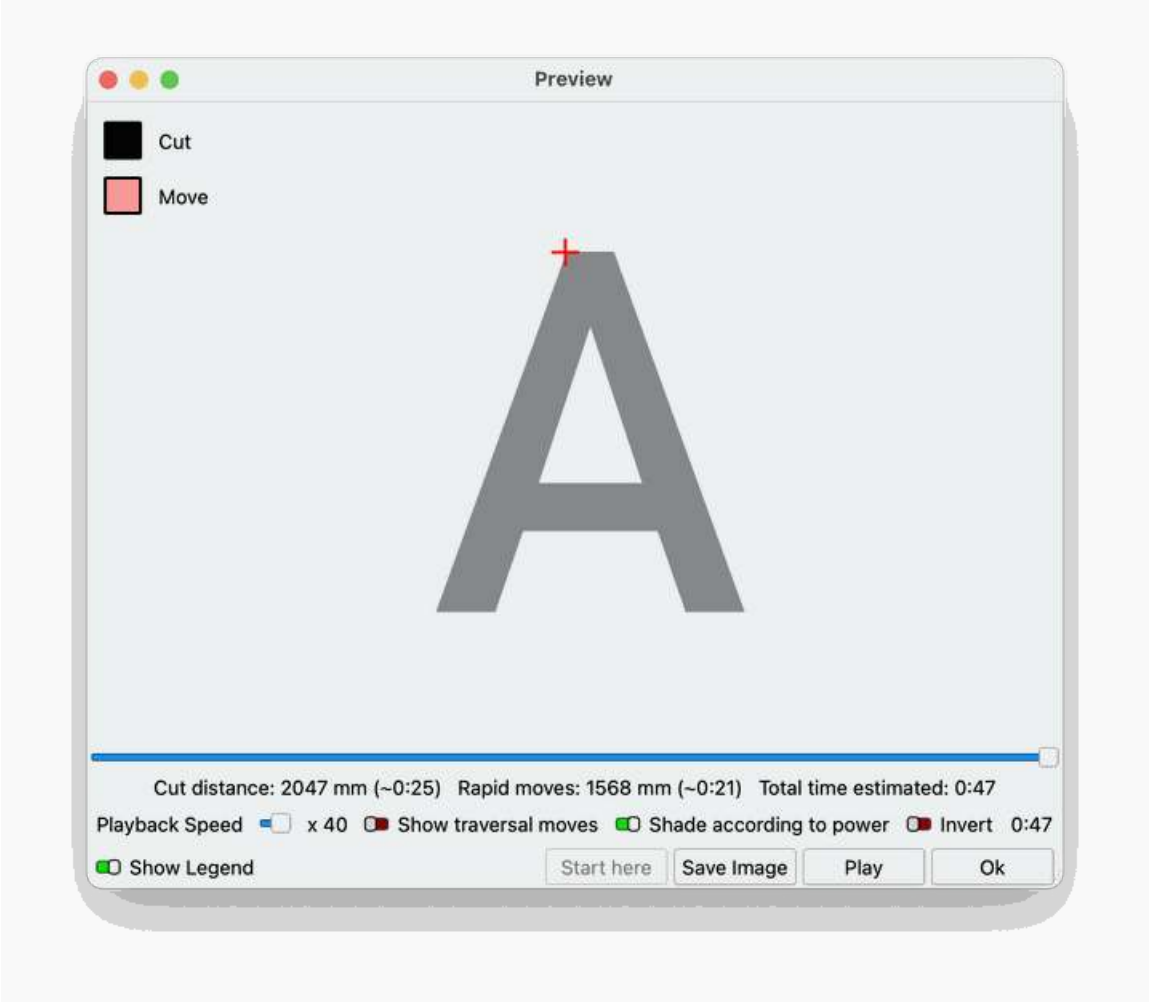


- Dot Mode tells the laser to pause and pulse at regular intervals along the path, instead of cutting continuously. This can be an effective way to do stitching holes, or cut very thin or delicate materials. This option is only available for lasers with Ruida controllers.

  Toggle off the Dot Mode switch in the Advanced Settings tab of the Cut Settings Editor to disable this setting.


Cut Settings Editor
00
Name C00
Output
1 Line
Sub-Layer Name
Output Sub-Layer
Speed (mm/sec) 450.00
Default
Max Power (%) 0.00
Air Assist
Min Power (%) 0.00
Default
Mode Line
Common
Advanced
Start Pause Time (ms) 0.00
Cut Through
End Pause Time (ms) 0.00
Cut Through
Power (%) 0.00
Overcut (mm) 0.00
Override PWM Frequency
20.000
kHz
Add Lead-In
0.00
Angle
Add Lead-Out
0.0
Length
Lead In/Out style
Line
Arc
Dot Mode
1.0
Time (ms)
0.01
Spacing
U Offset 0.00

Gray Shapes



Description

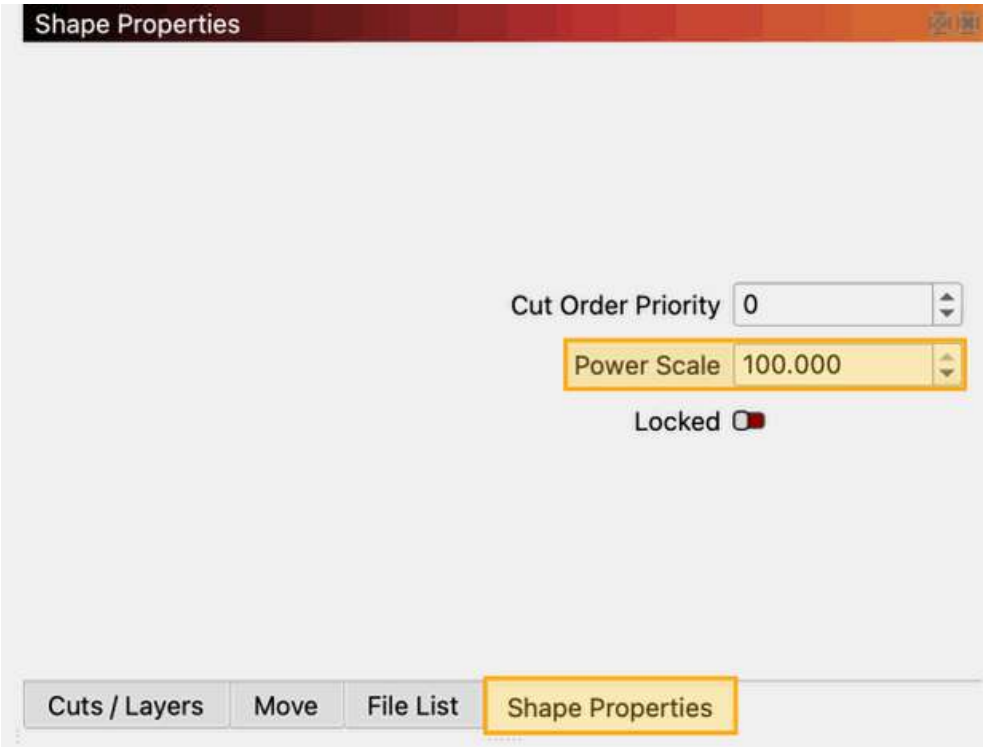
Graphics are rendered in a shade of gray rather than black.

Solution

When Shade according to power is toggled on in the **Preview** window, graphics with Power Scale values set below 100 are rendered in shades of gray. The lower the Power Scale, the lighter the shade of gray.

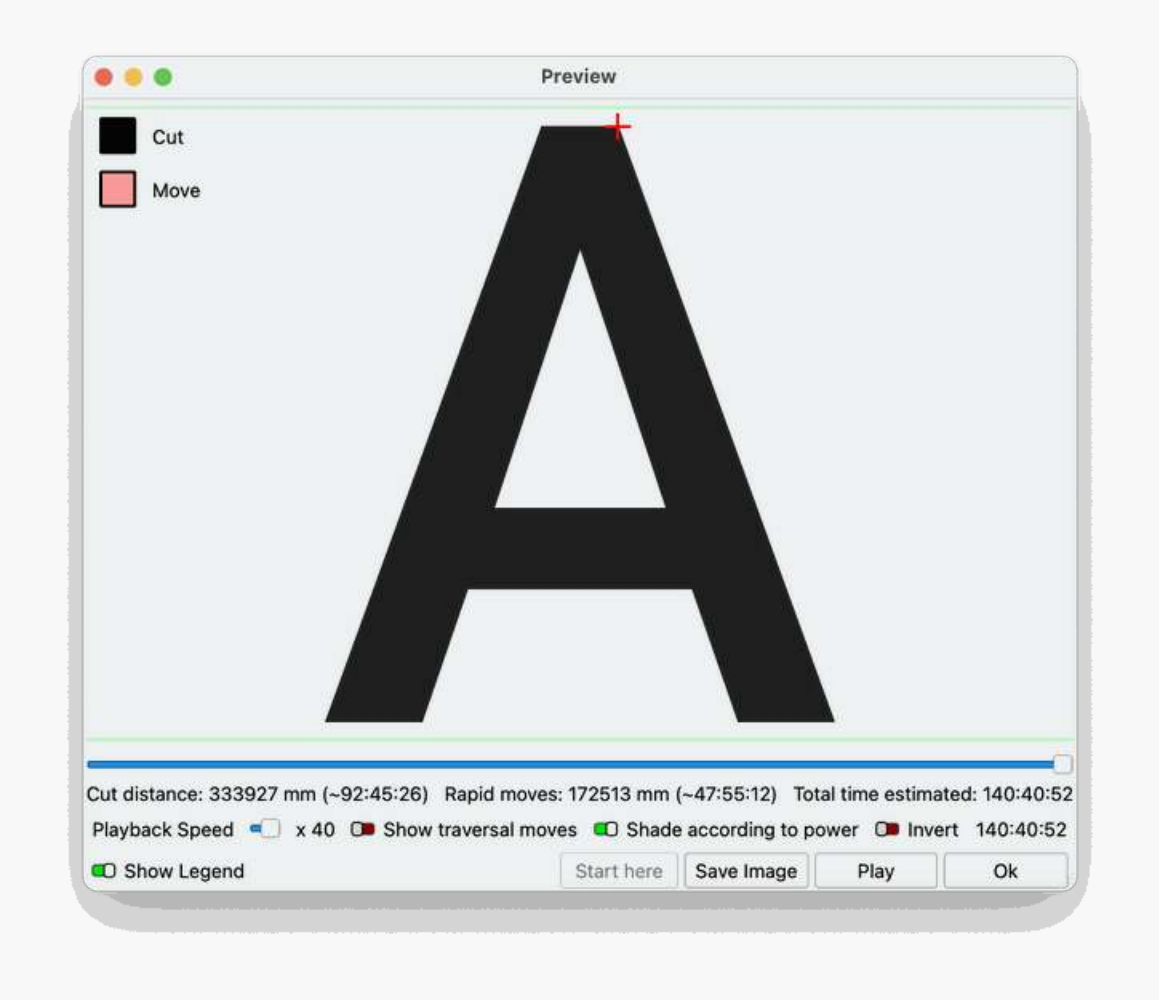
Power Scale reduces power output as a percentage between Min Power and Max Power. For devices that do not support the Min Power setting, power is scaled as a percentage between 0 and Max Power.

Increase Power Scale to 100 if you did not intend to reduce power output for the shapes depicted in gray.



▶ Watch the video on YouTube: https://www.youtube.com/watch?v=NC-UZokVvt4

Inaccurate Time Estimates



Description

The Total time estimated is very far off the actual run times of your projects.

Solution

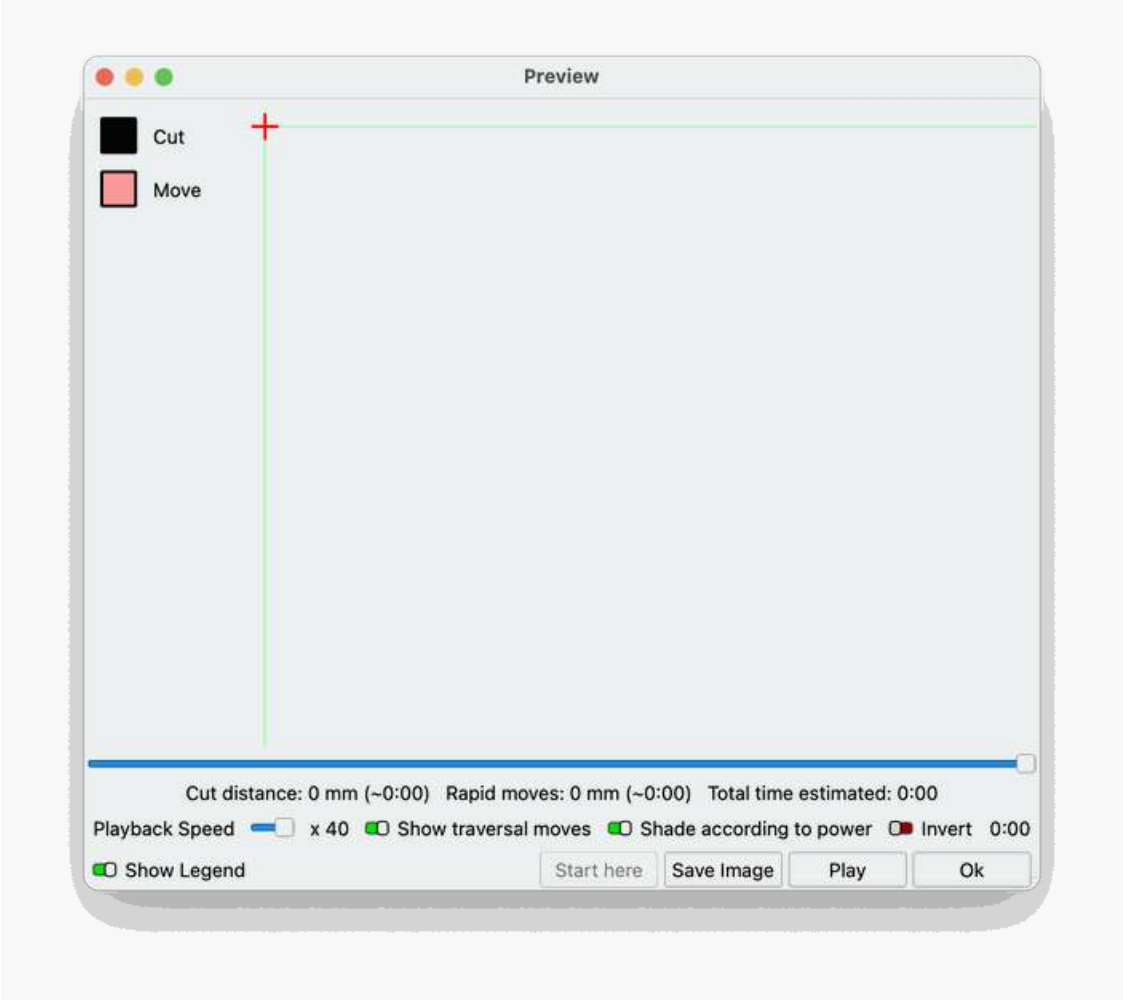
Time estimates in the **Preview** window are based off values in the Additional Settings tab of the Device Settings window.

Users with GRBL-based or Ruida controllers can use the **Read From Controller** button to sync these settings with your laser's internal settings, providing much more accurate time estimates without manual adjustment.

Otherwise, you can manually adjust the settings until they produce more accurate results.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=s1jCqB233N0

Missing Shapes

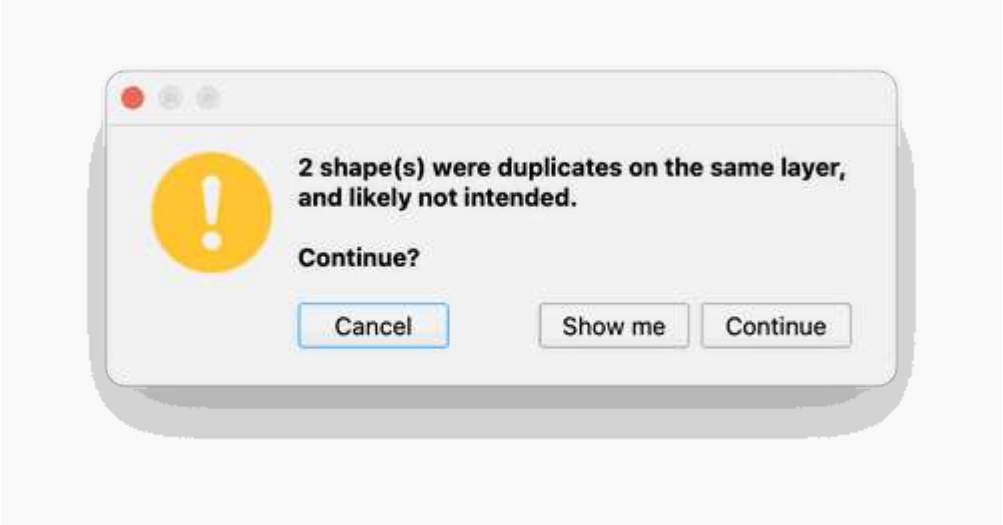


Description

Graphics you expect your laser to cut or engrave are not shown.

Solution

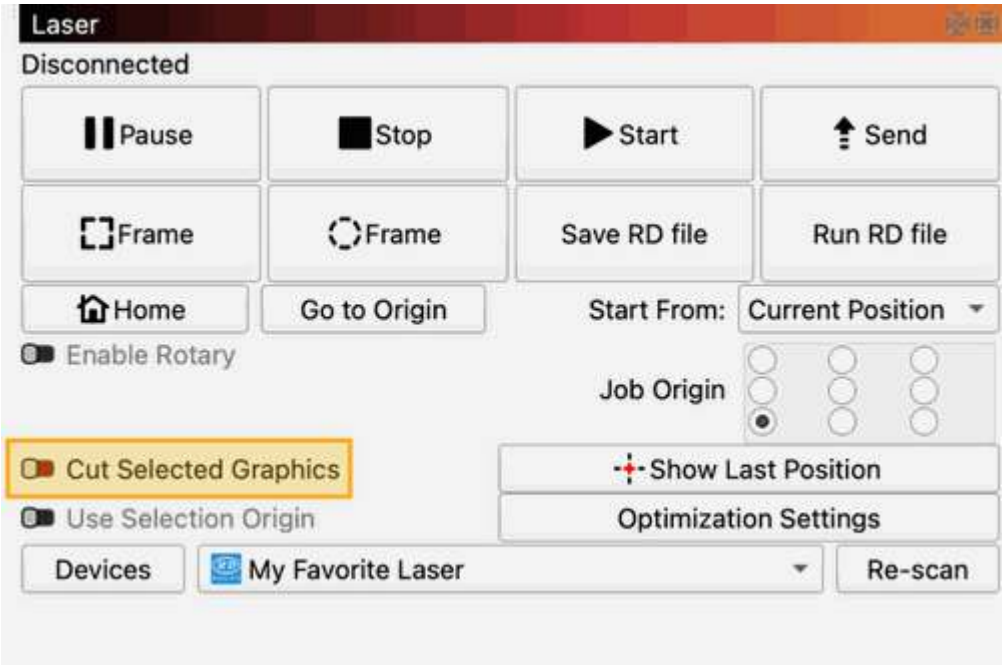
- If you have identical graphics directly overlapping one another, and both are assigned to the same Fill Mode layer, they will negate each other, and neither will output.
  - Before you attempt to **Preview** or engrave duplicate graphics positioned directly over one another, you will see a warning message. Click **Show Me** to select just one of each duplicate shape, then Delete them.



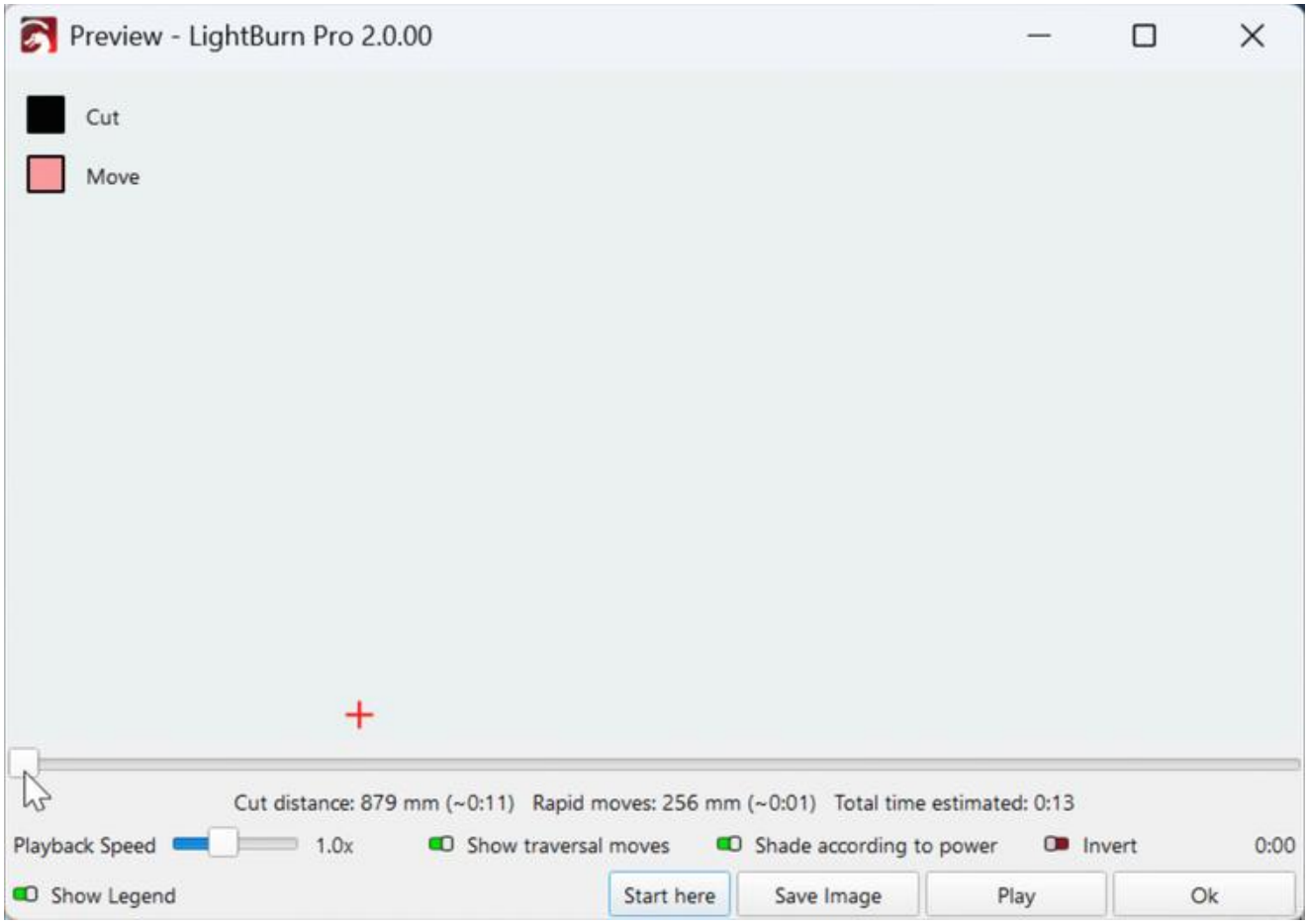
  - You can also go to Edit → Delete Duplicates to automatically identify and delete duplicate shapes.

- If you have enabled Cut Selected Graphics, make sure all graphics you want to cut or engrave are in your selection before you open the **Preview** window.

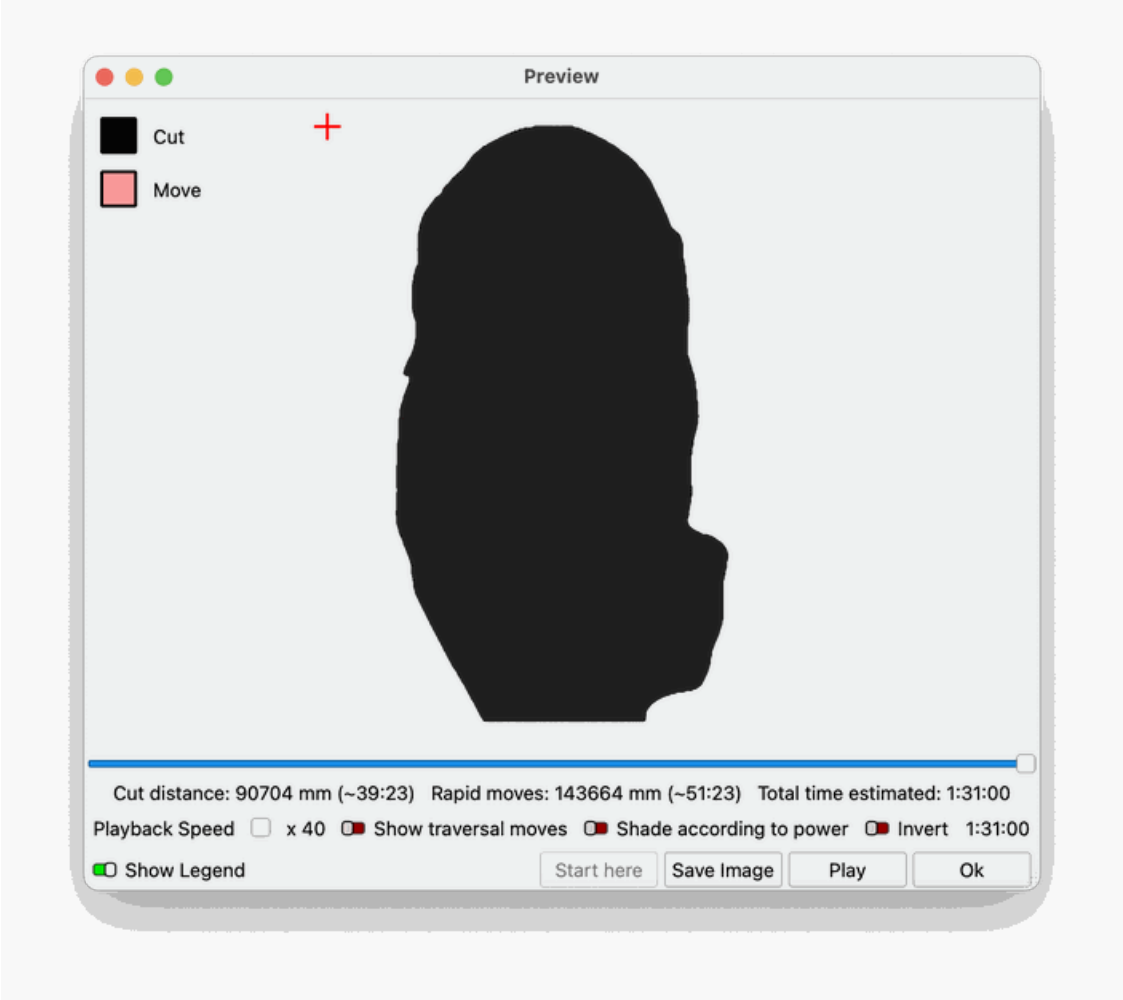
  To disable Cut Selected Graphics, toggle off the switch in the Laser Window.



- Make sure the Time Slider is not positioned at the very beginning of the simulation. Click Play to run the simulation, or click and drag the Time Slider to manually adjust its position.


Preview - LightBurn Pro 2.0.00
Cut
Move
Cut distance: 879 mm (~0:11) Rapid moves: 256 mm (~0:01) Total time estimated: 0:13
Playback Speed
1.0x
Show traversal moves
Shade according to power
Invert
0:00
Show Legend
Start here
Save Image
Play
Ok

Solid Black Images

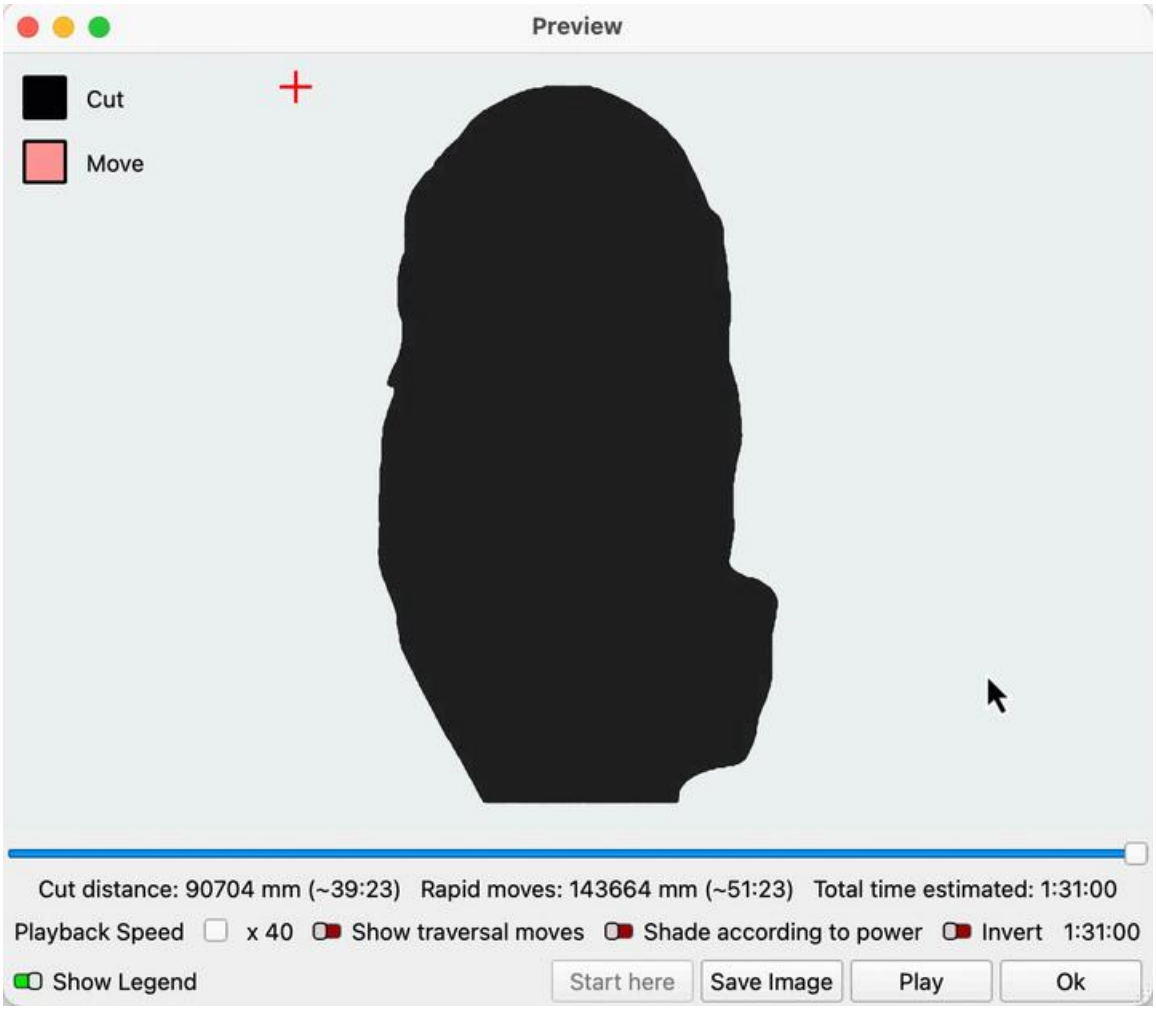


Description

Image engravings appear as solid black shapes, with no detail.

Solution

- If you are using Grayscale mode, make sure Shade according to power is enabled, or you will be unable to see any detail.

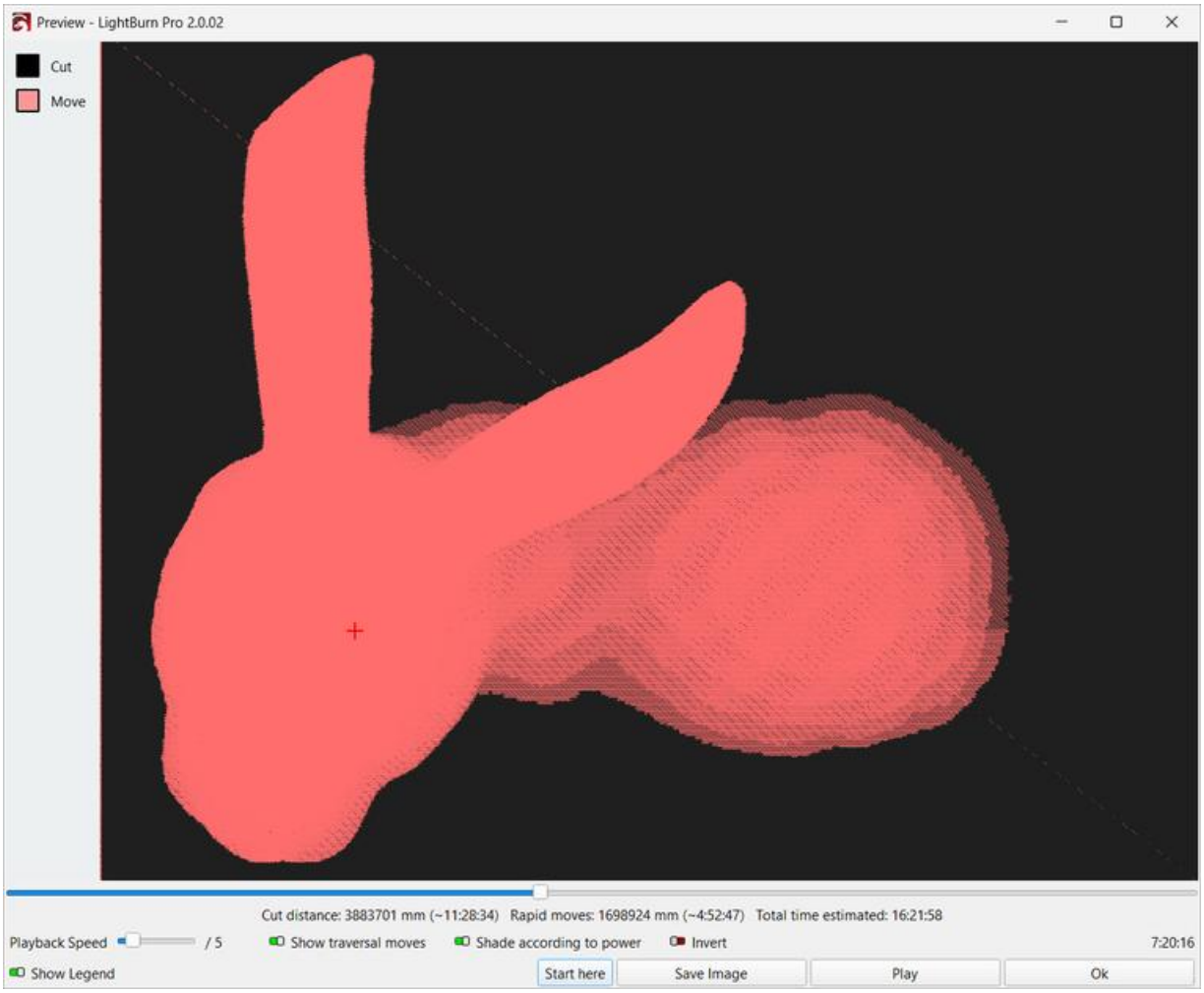


- If you are using a dithered **Image Mode** such as Stucki or Jarvis, you may need to zoom in to see detail in the engraving.

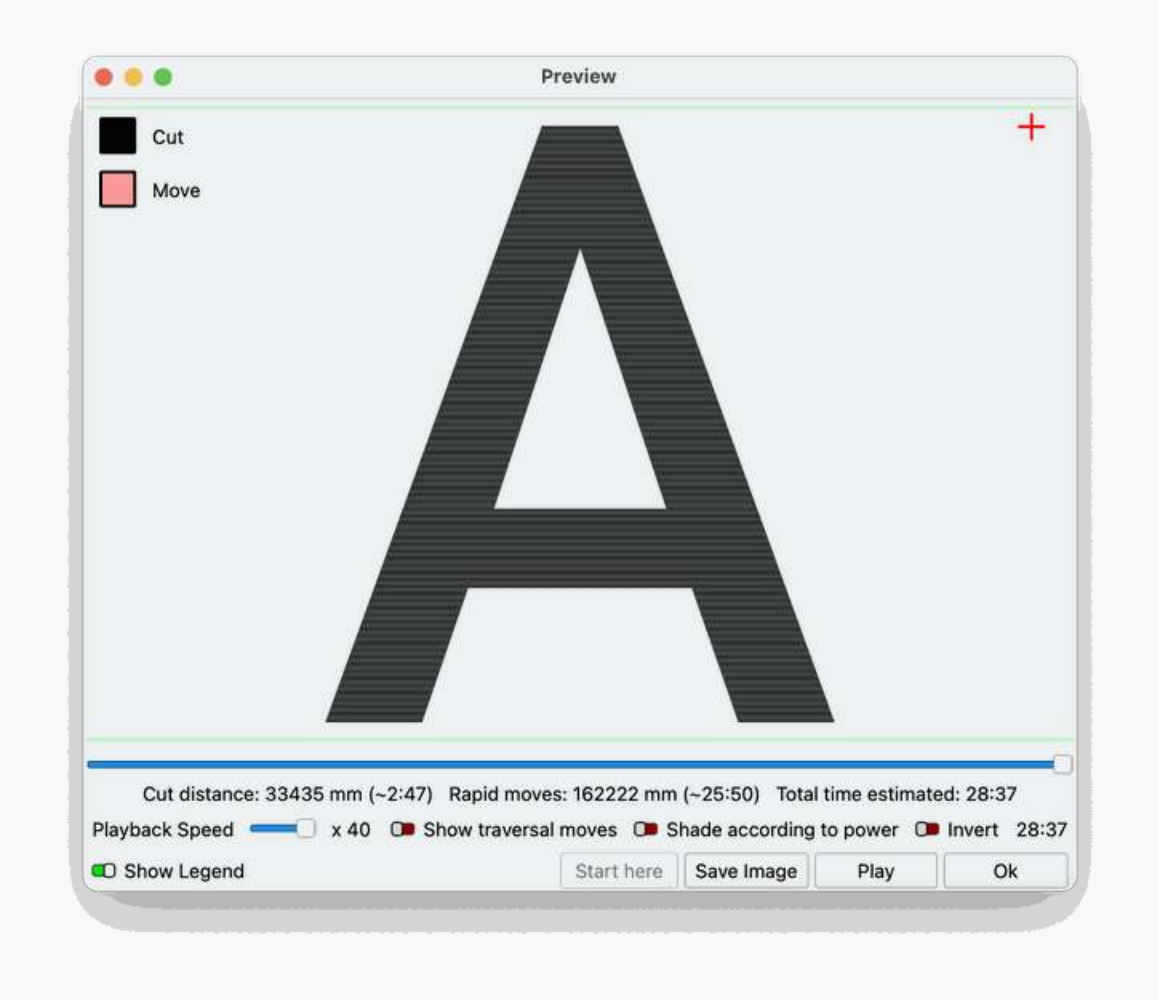


- If you are using 3D Sliced mode, make sure you have set the Image layer to engrave multiple Passes, and, for best results, that you've set an Angle Increment.

  When set up properly, with Show traversal moves enabled you should see the red areas of the Preview expand with each pass, as the laser travels over rather than engraving progressively larger (and lighter) areas of the image.


Preview - LightBurn Pro 2.0.02
Cut
Move
Cut distance: 3883701 mm (~11:28:34) Rapid moves: 1698924 mm (~4:52:47) Total time estimated: 16:21:58
Playback Speed / 5
Show traversal moves
Shade according to power
Invert
7:20:16
Show Legend
Start here
Save Image
Play
Ok

Strange Banding / Scan Lines Not Visible

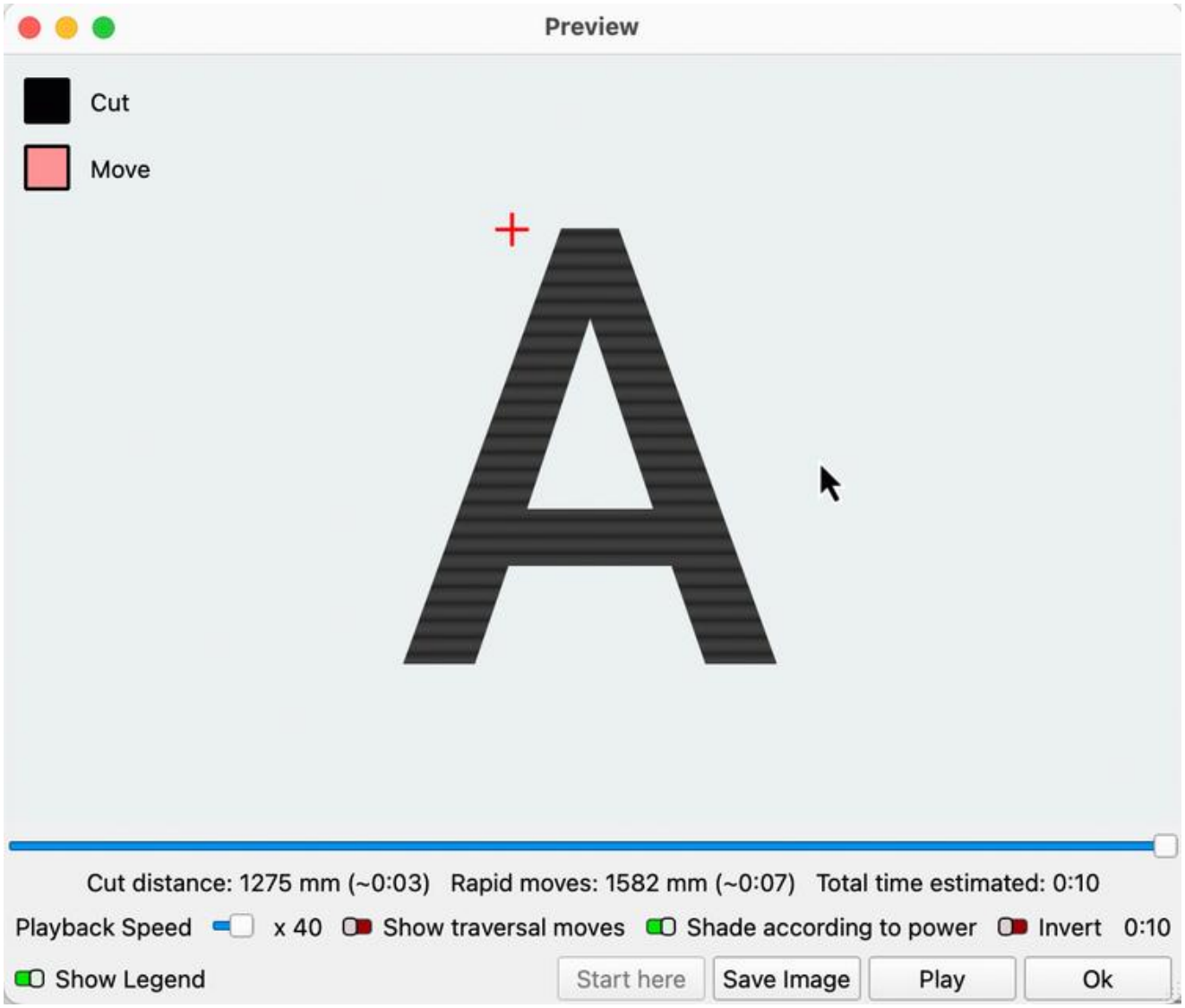


Description

Alternating black and relatively lighter gray lines appear, or individual scan lines are not visible.

Solution

- Banding sometimes occurs as an artifact of rendering the Preview at certain zoom levels. Zoom in or out to eliminate banding.
- Especially with relatively low Line Interval values, individual scan lines will not be visible at distant zoom levels. Zoom in to see individual scan lines.


Preview
Cut
Move
Cut distance: 1275 mm (~0:03) Rapid moves: 1582 mm (~0:07) Total time estimated: 0:10
Playback Speed x 40 Show traversal moves Shade according to power Invert 0:10
Show Legend
Start here
Save Image
Play
Ok

Connection Troubleshooting

Can't Connect Laser to Computer

First steps:

- Make sure your laser is turned on
- If your laser has an Emergency Stop button, make sure it's released, typically accomplished by twisting the button
- If you're connecting with a cable, make sure it is plugged in securely on both ends
- Close any other control software that might be communicating with your laser
- Make sure you've selected the correct device in LightBurn, and that the controller type and connection method are correct

If you still can't connect to the laser:

- Check USB Cables
- Check Drivers
- Check Serial Port

Laser Doesn't Stay Connected To Computer

If your laser connects to the computer but loses its connection, you'll need to find what's causing the lost connection. You may see the following error message.



Connection lost meants that your USB connection was dropped, either by your controller or your computer. This is relatively common and there are a number of things that can cause it. Would you like to see help on troubleshooting this issue?

Please try the following if you're losing your connection to the laser:

- Check USB Cables
- Check For Electrical Issues
- Windows-Specific Issues

Laser Behaves Unexpectedly

If your device is restarting when you try to use it, you are most likely experiencing either communication problems or brief losses of power. This is often visible when a device set to home on startup homes unexpectedly. Please try the following:

- Check for Electrical Issues
- GRBL Communication Problems

If you're experiencing unexpected behavior from a device not originally intended as a laser, such as a diode module mounted to a 3018 CNC router, you may need to adjust the settings:

- Configuring CNC-Based Lasers

Laser Is Connected, But Not Responding

If your laser appears to be connected to the computer but is not responding, try the following:

- If your device has a screen, check for error messages or alarms
  - Some enclosed lasers have sensors to detect if the lid is open, and will not run until the door is closed and the error is cleared
  - Some lasers with water cooling have sensors to detect the flow of water, and will not run if they don't detect water flow
- Check to see if the correct serial port is selected, if relevant
- For GRBL devices, check the Console Window for Errors and Alarms

Transfer Failed

If LightBurn is unable to transfer the file to your laser, you'll see the following error message:

> There was a problem sending data to the laser. The machine may be busy or paused.

- If you're using a device that requires selecting a COM port in the Laser Window, make sure the correct COM port is selected. Other devices connected to your computer may also show up as COM ports, but LightBurn won't be able to connect to them. You can confirm that the COM port selected is the correct one by disconnecting or turning off your laser - if the COM port disappears when you do so, it's the right one.

For DSP (Ruida, Trocen, TopWisdom)

DSP lasers have some additional things to check.

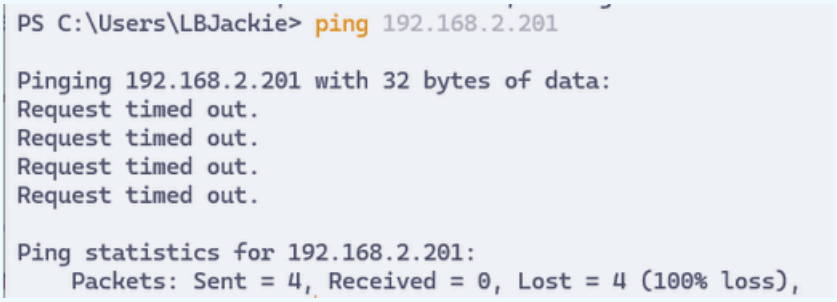
- If you're connected over USB and have the ability to connect over Ethernet, try an Ethernet connection and see if it makes a difference. If it does, you're having USB connection issues and may need to replace your USB cable.

- If you're using an ethernet connection, it's possible that the IP address you configured in LightBurn no longer points at the the laser. To test this, you can ping the laser. Click the blue box below for directions. If your computer can't communicate with the laser over the network, LightBurn won't be able to either.

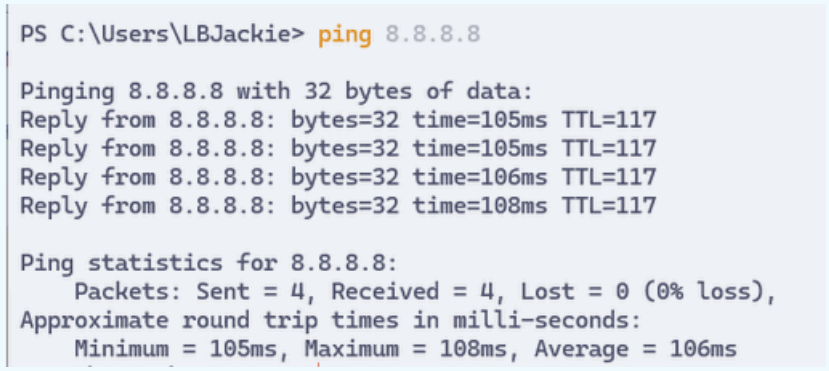
Testing network connections

I'll use the IP address `192.168.2.201` as a placeholder, but you should replace it with your own laser's IP address.

1. Go to Device Settings - Basic Settings and find the IP address you used for your laser.
2. Open the command prompt. You can do this by opening Power Shell on Windows or Terminal on Mac.
3. With the laser on, run the following command: `ping <IP_ADDRESS>`, so in my example it would be `ping 192.168.2.201`.



A ping that failed to connect. No devices are present at the specified address.



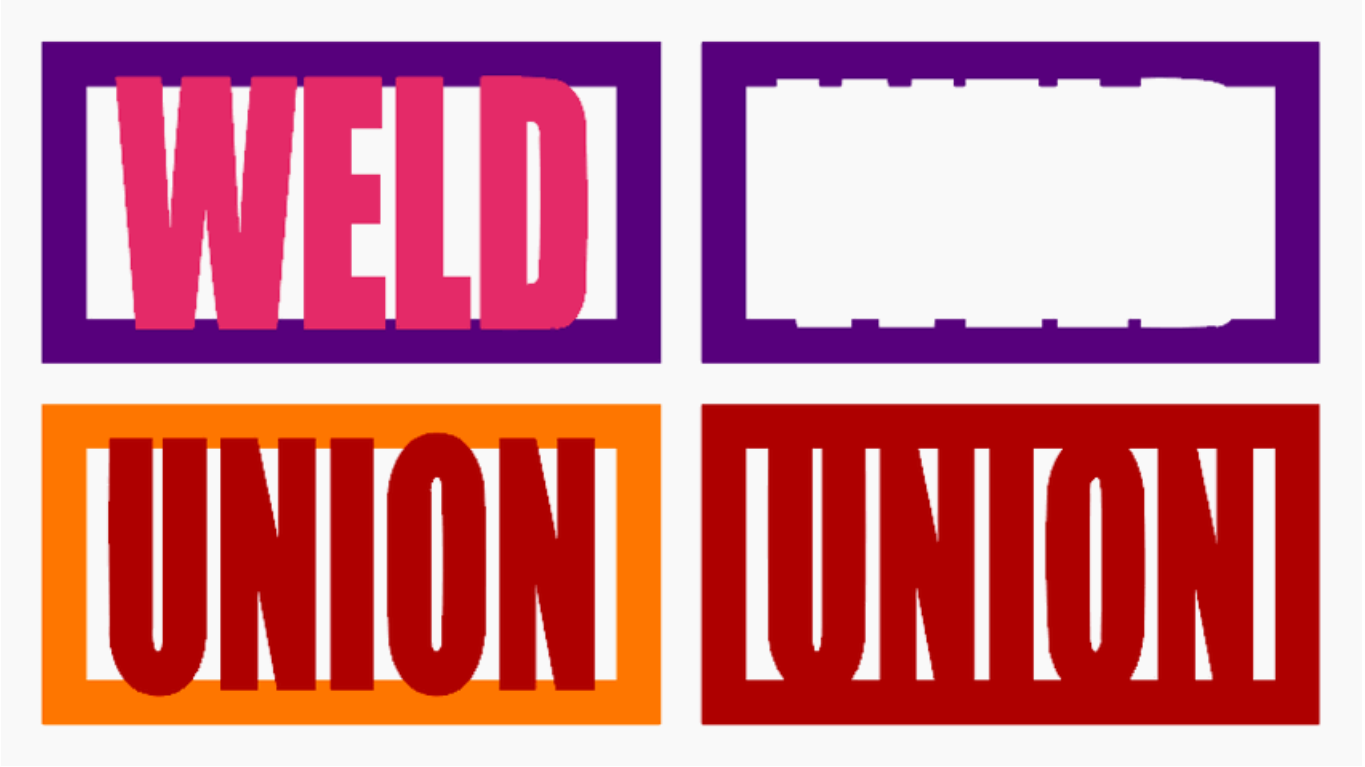
A successful ping. There is a device at this address.

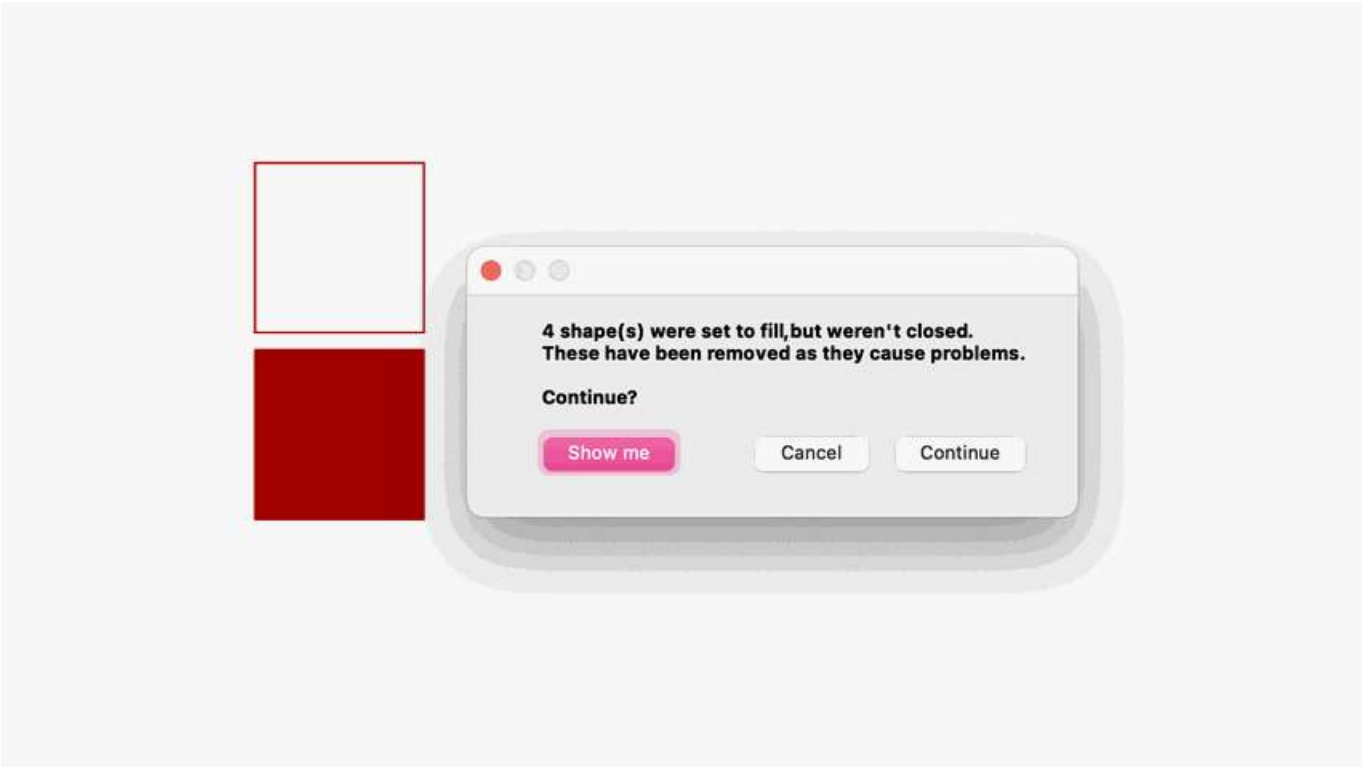
4. If you had an unsuccessful ping, your computer can't see your laser on the network. Make sure the IP address matches the one set on the controller.
5. If you had a successful ping, it's possible that another device on your network is using the IP address and is interfering with communications. Turn off your laser and try again - if you still get a successful ping, it's coming from something other than the laser. Change your IP address on both the laser and in Device Settings.

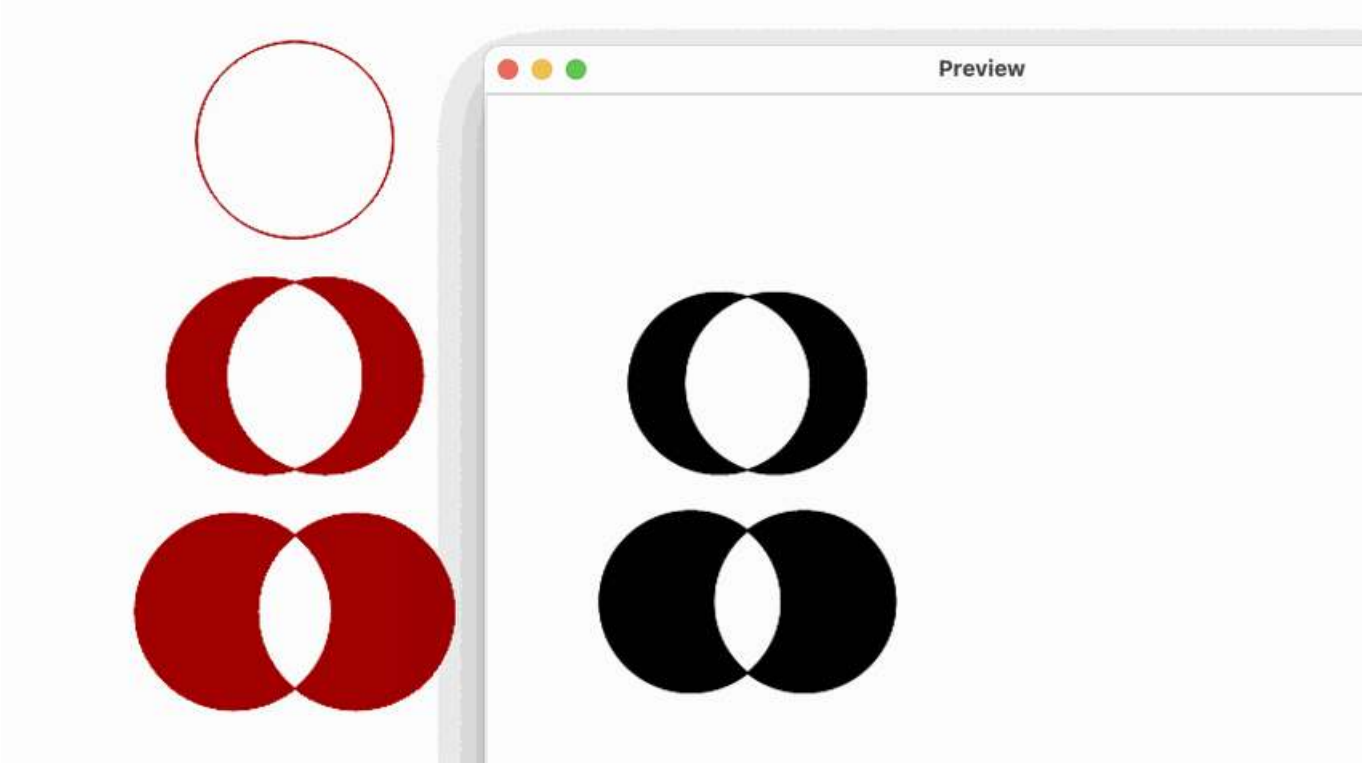
- It's possible that your laser is running out of storage space, preventing LightBurn from sending the file. Try clearing existing files off the controller. Controllers vary, but typically you'll want to find **File → Delete All** in the on-screen menu. You should be able to find the exact method in the user manual for your laser or controller.
- Extremely high resolution engravings are sometimes rejected by the controller. Resolutions above 254 DPI (equivalent to Line Intervals below .1mm) have diminishing returns in terms of image quality. Try reducing your resolution and see if the problem is resolved.

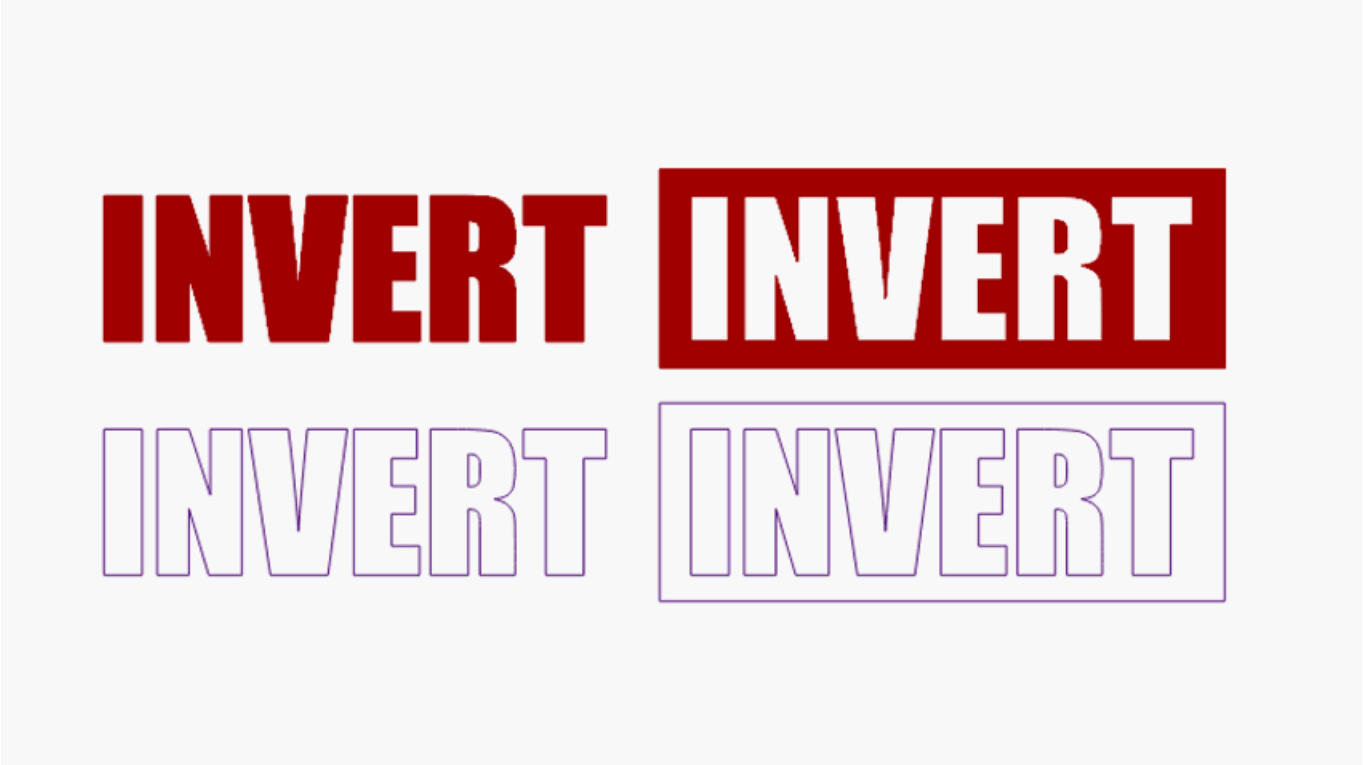
LightBurn Editor Help

This page contains help for common pitfalls, errors, and frequently asked questions that users have encountered while using LightBurn. If you're looking for help figuring out why your laser's output doesn't look like you expected, please visit our page on job quality and output issues.

Example	Summa
	Text or content of a fra disappe when using **Weld**

Example	Summa
	Filled shapes look like outlines "Shapes weren't closed" error when preview

Example	Summa
	Filled shapes look lik outlines

Example	Summa
	Need to invert which parts of vector shape a engrave

GRBL Errors

GRBL devices have a number of different alarms and error codes you may encounter. These will be visible in the Console Window.

This page contains information about the errors and alarms LightBurn users are most likely to encounter, as well as some other frequently encountered problems with GRBL devices. For additional information, we've found the following resources helpful:

Official GRBL Documentation

Diode Laser Wiki - GRBL Errors and Alarms

GRBL Alarm Codes

Alarms are emergency messages, and are used when something has gone very wrong. These are shown in the Console Window as `ALARM:#`, with a number to tell you what caused the alarm.

ALARM:1

A limit switch was triggered while the machine was moving. The machine position is probably lost, which causes the machine to hit the edges of its workspace.

Fix this alarm by rehoming the machine. If this continues to happen:

- Review your **Start From** and **Job Origin** selection, and make sure you are not commanding the laser to move outside of its physical bounds. See Coordinates and Job Origin.
- Make sure your machine is homing correctly by comparing the position shown in the Move window in LightBurn to the machine's actual position within its work area.
- Some limit switches are triggered by reading resistance to stepper motors. Make sure the wires aren't snagging and causing a false trigger, and check your machine for mechanical problems.

ALARM:2

You're trying to send the machine outside of its known working area, and GRBL has prevented this to prevent damage to the machine. This can happen for several reasons:

- You're trying to run a job that exceeds the size of the machine work area.
- You're trying to run a job in **Current Position** or **User Origin** mode and haven't properly set the **Job Origin** or User Origin settings. See Coordinates and Job Origin.
- You're working with a Y Axis rotary and the distance you are commanding the rotary to travel exceeds the Y Axis max travel limit set in your laser's firmware. Temporarily disable **Soft Limits** in the Machine Settings window, or by entering `$20=0` in the Console Window. Be sure to re-enable Soft Limits in the Machine Settings window or by entering `$20=1` in the Console Window when returning to flat engraving.
- You have your machine misconfigured, possibly trying to run a negative workspace machine without a workspace offset. See GRBL Configuration.

ALARM:3

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=zzQ9-K-7UqU

The laser halted abruptly, sometimes due to another Alarm or Error, or, often, the computer losing connection to the machine. Rehome the machine to ensure the machine reports its coordinates correctly. See Connection Troubleshooting for help if this happens repeatedly.

ALARM:8

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=SCSRDWzOWu8

The machine failed to pull off one or both limit switches after homing.

- Inspect your laser's limit switches to make sure they are not stuck in the closed position.
- Check your machine's belt tension and overall mechanical setup to make sure it is capable of moving, homing properly, and clearing the limit switches. See Troubleshooting: Mechanical Issues for more information.
- Make sure your power supply is securely plugged in and functional. A USB connection can sometimes be enough to power a laser's control board, but it will not provide enough power for the stepper motors to move off the limit switches.

ALARM:9

The machine's attempt to home failed because it didn't locate a limit switch. GRBL will stop homing to prevent potential damage to the machine if it doesn't encounter a limit switch within 1.5x the maximum travel distance.

- Make sure your machine has limit switches. If you installed them yourself, make sure they're wired correctly and the connectors are snug.
- Check your machine's belt tension and overall mechanical setup to make sure it is capable of moving and homing properly. See Troubleshooting: Mechanical Issues for more information.
- Make sure your power supply is securely plugged in and functional. A USB connection can sometimes be enough to power a laser's control board, but it will not provide enough power for the stepper motors to respond to the homing command.

GRBL Error Codes

These error codes are less severe than alarm codes, and are used when your machine receives a command it can't act on. Like alarms, these are shown in the Console window as `error:#`, with a number to tell you what caused the error.

Error:1

The machine received an invalid GCode command. This error code is usually the result of enabling **GCode Clustering** for a device that does not support it, or combining multiple commands into one line.

Make sure GCode Clustering is disabled in Device Settings.

If you are using Custom GCode, check your settings for accuracy. If you're typing commands in by hand in the **Start** or **End GCode** fields under GCode tab of the Device Settings window, check for typos.

Error:3

The machine received the `$` system command but does not recognize or support it.

Error:5

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=QpgvmTKU1cE

LightBurn sent the command to the controller to home it, but the controller does not have homing enabled or configured.

Typically, this occurs when Auto-Home On Startup is enabled in Device Settings for a machine without homing switches. Disable this setting if your laser does not have homing switches.

If your laser does have homing switches, enter $22=1 in the Console Window to enable homing in your machines firmware. Consult your laser's manufacturer if you are not sure.

Error:8

A command was sent to your laser while it was still processing another command. If your laser is homing, wait and try again.

This error can also be caused by communications corruptions or electrical interference.

- In the Device Settings window, try changing **Transfer Mode** to **Synchronous**.
- Try a new, shielded USB cable.
- See Troubleshooting: Electrical Issues for more information.

Error:9

LightBurn tried to send a command to your device, but the machine is locked. GRBL locks devices while jogging (moving the gantry) or when the machine has an active GRBL alarm.

- If the machine is moving, wait for it to finish moving.
- Enter $X in the Console window to unlock the machine.
- If there is an active alarm, identify the alarm and address the cause.

Error:20

The machine received an unsupported or invalid GCode command. This is usually the result of the wrong device type being selected.

> **Tip**

> To check what version of GRBL your laser is running, enter `$I` into the **Console** window and press `Enter`. The messages shown in response should include the GRBL version in a message starting with `VER`.

- Make sure your device type is set correctly. In most cases, your device should be set to **GRBL**. If your laser is running GRBL 1.1e or earlier, change your device type to **GRBL-M3**. To check or change your device type:
  - Click the **Devices** button in the **Laser** window
  - Double-click your device
  - Change the device type if needed
  - Click **Next** and then **Finish** to complete the setup prompts and save the changes
- If your laser is running GRBL 1.1f or later, and the Error:20 message is accompanied by `MSG: M4 requires laser mode or a reversible spindle`, enter `$32=1` in the **Console** window and press `Enter` to enable Laser Mode.

Error:24

The machine received an invalid GCode command, combining two GCode commands that both require X/Y/Z Axis parameters. This is usually the result of communications corruption.

- Try a new, shielded USB cable.
- See Troubleshooting: Electrical Issues for more information.

Other Common Problems for GRBL Machines

Busy

The **Busy** bar in the Laser Window is not necessarily an error — it shows that the machine is busy doing something. However, if you are seeing **Busy** but the machine is not moving, it may not be connected to the computer.

- You may need to select the correct communication port.
- If you don't have an appropriate port available, you may need to install drivers.

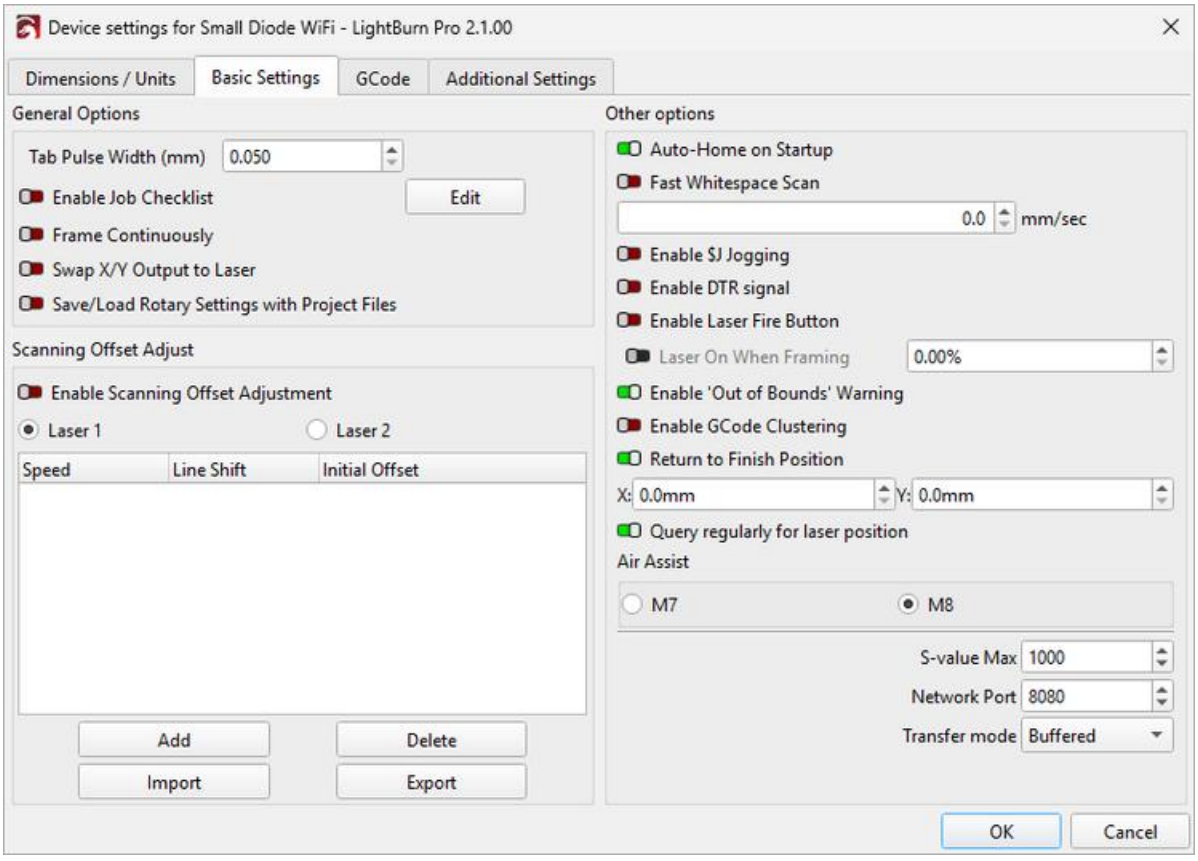
Laser Not Connecting

If you see **Disconnected** in the Laser Window, your computer is unable to communicate with the laser. Make sure it's plugged in and turned on. If it still does not connect:

- You may need to select the correct communication port.
- If you don't have an appropriate port available, you may need to install drivers.
- Check Connection Troubleshooting for more help troubleshooting your connection to the laser.

GRBL Communication

Check Transfer Mode



LightBurn defaults to the **Buffered** transfer mode for communication with GRBL devices. This is faster and more reliable than the **Synchronous** transfer mode. However, there are a few devices that are incompatible with buffered transfer mode. These devices will stutter, stop, or even restart when buffered communication is selected. If you notice this happening, try switching the transfer mode to **Synchronous**.

DTR Signal

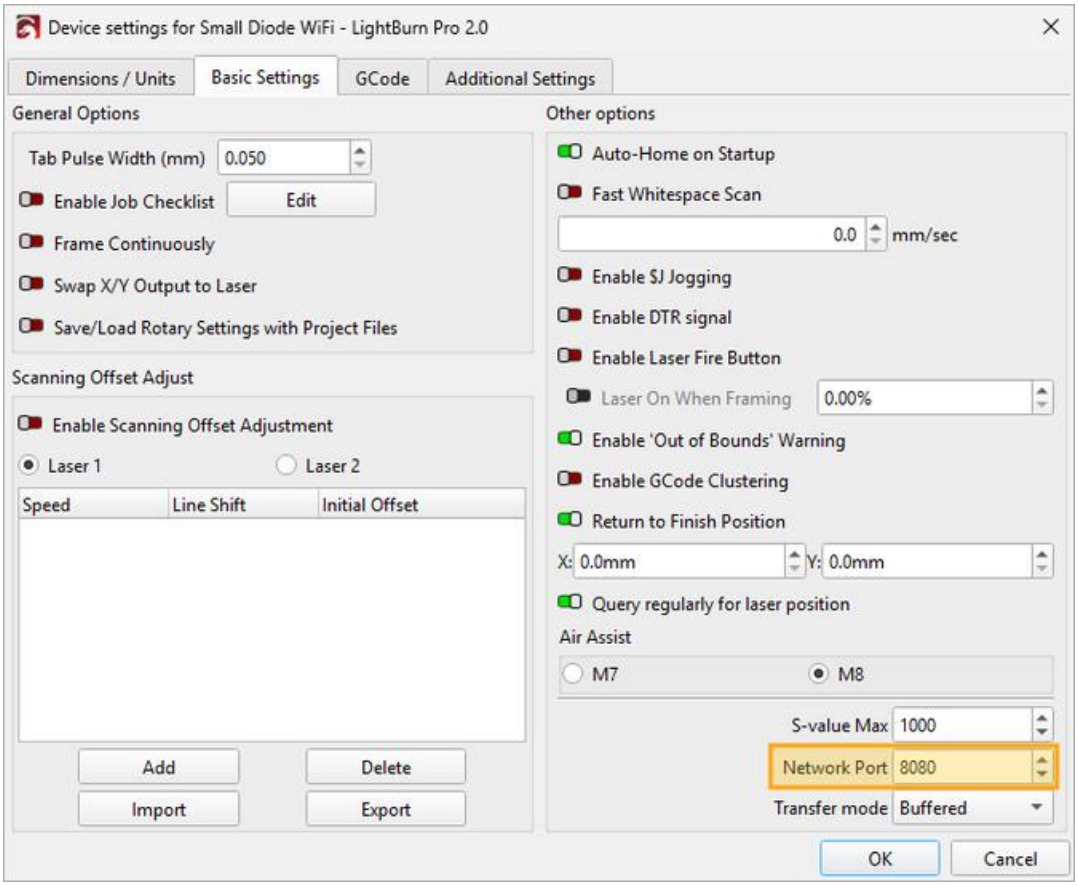
Standard serial ports have a pin (Data Terminal Ready, or DTR) that the host enables to tell modems that software is ready to receive data, and some devices require this signal to begin communication. Many programmable hobby-level controllers, like Arduino based systems, use the DTR pin to reset the controller. LightBurn usually sets this value for you, but if you find your GCode controller won't communicate, toggling this setting in Device Settings might help.

Electrical Noise

Route your USB cable so that it avoids the stepper motors or anything else that could cause electrical interference.

Network Connection Port

If you're connecting to the laser over a network instead of with a USB cable, you may need to change the network port setting in Device Settings.



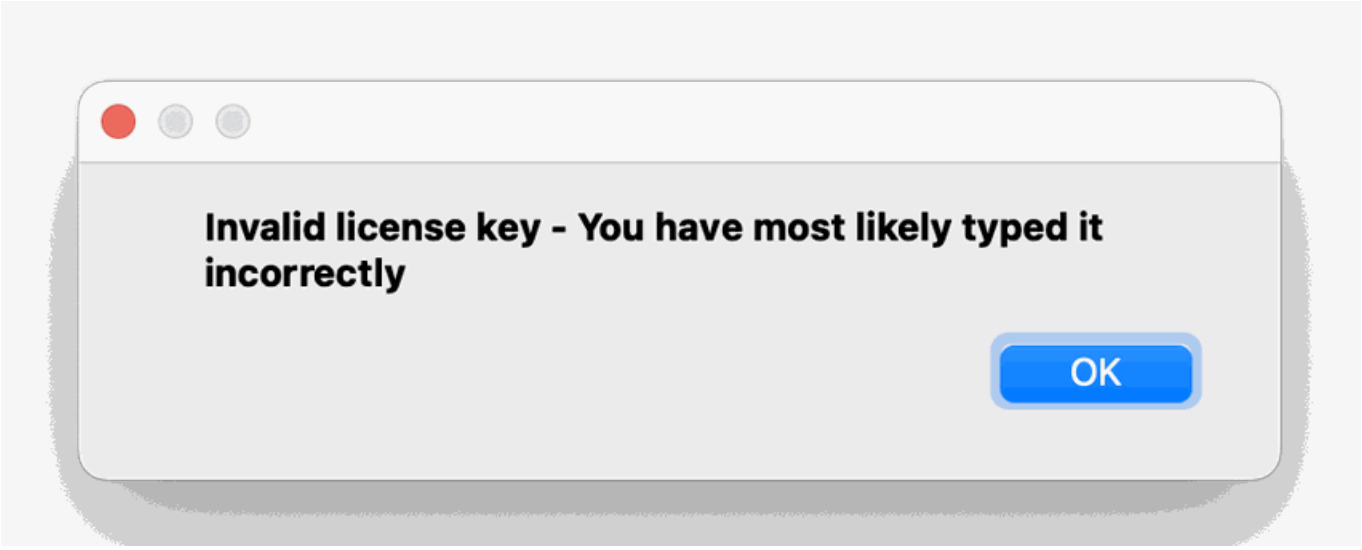
The **Network Port** setting is in the bottom right of the Basic Settings tab of the **Device Settings** window. The default setting is **23**, but some devices use a different port. Try setting the port to **8080**, and if it still doesn't work, check your laser's user manual for more information.

License Troubleshooting

This page contains solutions to common License Activation errors, and other License Management issues.

See our License Management Guide for full instructions on activating your license and How the LightBurn License Works for details on licensing terms.

Invalid License Key

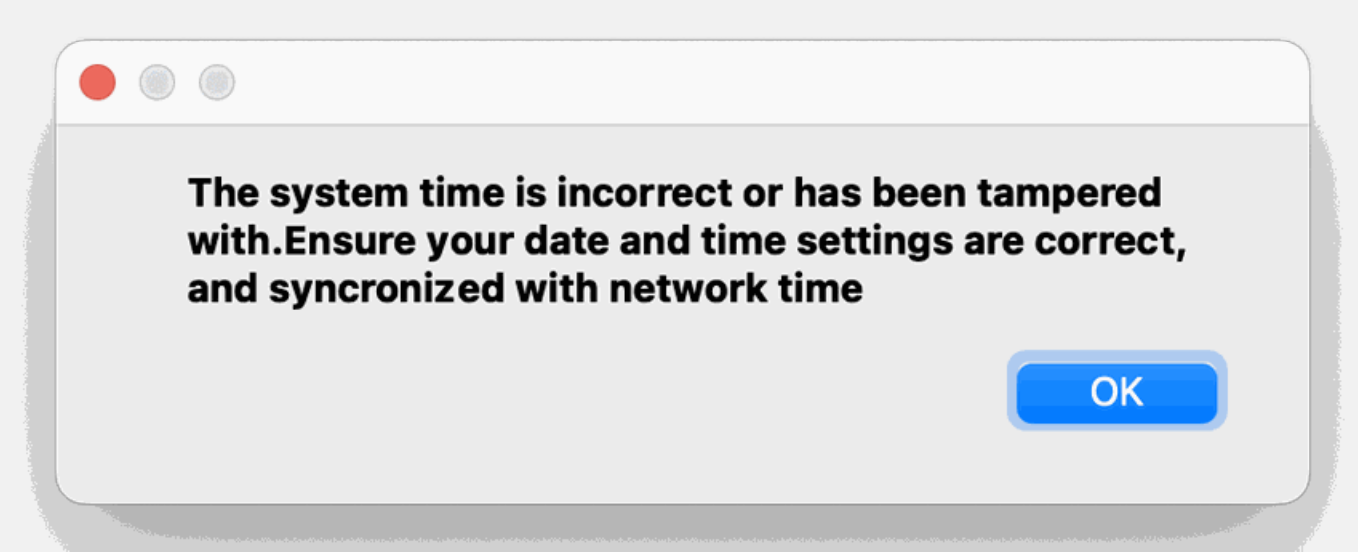


Your license key has been entered into LightBurn incorrectly. To avoid typos when entering your key into LightBurn, we strongly recommend copying and pasting the key directly from your original order email.

Highlight the key and press `Ctrl`/`Cmd` + `C` to copy it, then click in the **License Key** field in the License Management window in LightBurn, and press `Ctrl`/`Cmd` + `V` to paste it.

If you must type your key manually, take care to follow the exact formatting of the original key, including dashes. The letter O is never used in a LightBurn license — enter the number 0 instead.

System Time Incorrect

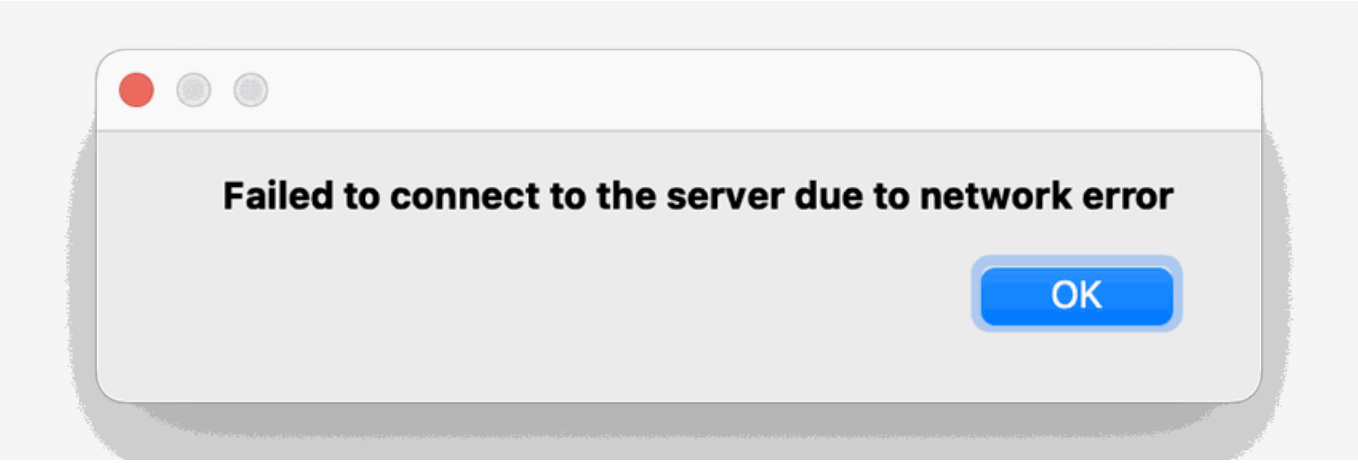


Your computer's system time is inaccurate, probably because your computer isn't set to sync automatically.

If you're using Windows, right-click on the clock on your toolbar, click **Adjust date / time**, then make sure **Set time automatically** is enabled. If it is, click the **Sync now** button as well.

If you're using a Mac, go to **System Settings** → **Date and Time** and enable **Set time and date automatically**.

Failed to Connect to the Server



LightBurn was unable to connect to our license server to verify your key. Make sure your computer is connected to the internet.

If you are certain your computer is connected to the internet but you are receiving this error message, your network may be restricting LightBurn's ability to connect to our license server.

If you are using a Firewall or have any other restrictions in place, try temporarily disabling them. LightBurn only needs to connect once in order to activate a license.

To permit access without disabling restrictions, whitelist **api.cryptlex.com** for port **443**.

It is also possible to configure LightBurn to access the network through a proxy. You can append the proxy connection address using `-p ProxyString`.

The proxy format is:

```
[protocol://][username:password@]machine[:port]
```

Examples of valid proxy strings:

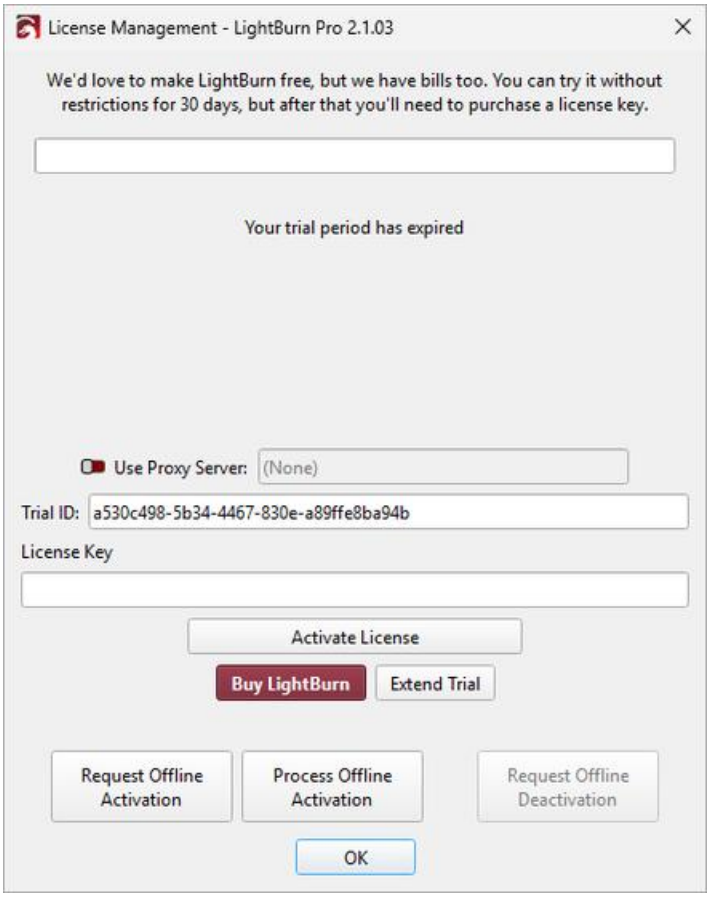
```
http://127.0.0.1:8000/
http://user:pass@127.0.0.1:8000/
socks5://127.0.0.1:8000/
```

If you are still unable to connect, use our **Offline Activation** process instead. For full instructions, see License Management: Offline Activation.

> **Note**
>
> Offline activation does not work with Trial Keys. LightBurn must connect to the internet once in order to activate the Trial Key, after which you are free to take your computer offline again.

Expired Trial or Need to Enter License Key, Despite a Valid License

If you know you have a valid LightBurn license, but LightBurn is warning you about an expired trial and asking you to purchase a license key to activate LightBurn, don't panic. This can happen for a few reasons, and is easy to fix.



How to Fix

1. Go to the License Management Portal and copy your LightBurn license key. You can also get your license key from your purchase confirmation email, if you still have it.
2. Paste the license key into the **License Key** box and click **Activate License**.

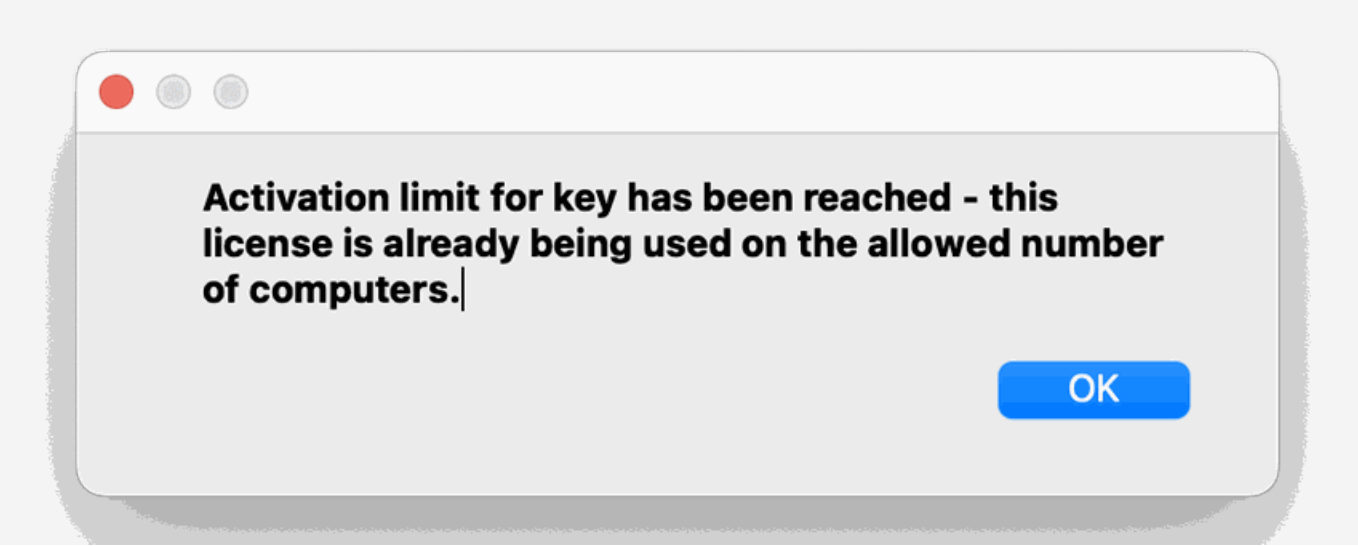
If you are still having problems, please contact support at support@lightburnsoftware.com for help.

Why it Happens

LightBurn doesn't require you to maintain a constant internet connection or log into a user account to use the software, but there are some situations that can cause your LightBurn installation to think you don't have a valid license.

- If you've purchased a LightBurn license and then go to use a copy of LightBurn you've already installed and used on a trial, LightBurn doesn't know you've purchased a license until you activate that installation with your license key. Even if you've activated on another computer, you'll need to activate it on each computer you use.
- In some cases, updates to your operating system or hardware changes/upgrades (especially replacing a hard drive) can change your computer's system enough that LightBurn mistakenly recognizes your computer as a new computer. Reactivating with your existing license key solves the problem. You can use our License Management Portal to deactivate the previous installation and free that seat/installation back up for future use.

Activation Limit



You have used up the maximum amount of simultaneously allowed activations for your key. Deactivate LightBurn from an unused computer by opening LightBurn on that computer and clicking **Deactivate License** in the **License Management** window, or log in to our License Management Portal to manage your activations there. For full instructions on using the License Management Portal, see License Management.

License Portal

A LightBurn license comes with 3 allowed activations (seats) by default. If you'd like to activate LightBurn on additional computers simultaneously, you can purchase additional seats for your key here.

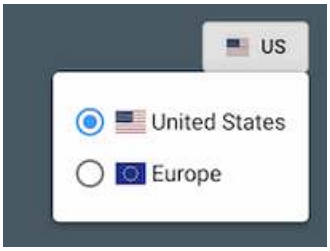
Deactivation Limit

LightBurn limits the amount of times a license key can be deactivated, to discourage sharing or other misuses of the key. If you reach this limit, contact us at support@lightburnsoftware.com to request an increase to your limit.

Unable to Access the License Portal

See License Management for full instructions on accessing the License Portal.

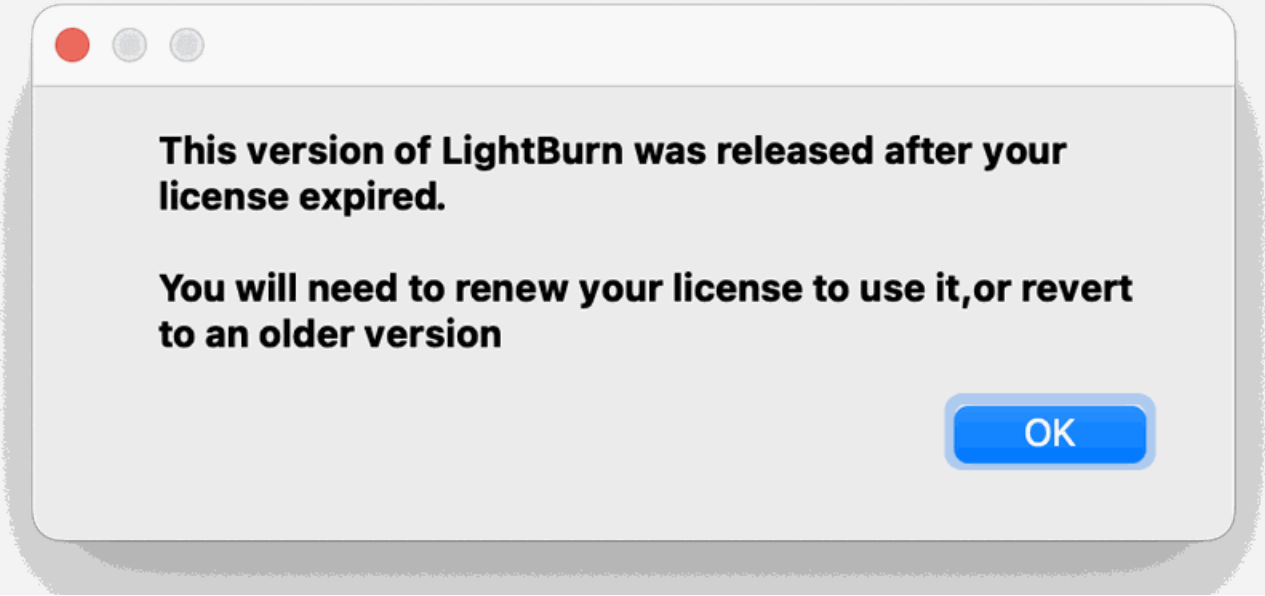
Select **United States** as your region in the upper-right corner of the License Portal login page, regardless of your location, or our License Management server may fail to send the password reset email.



If you purchased your key through our web store, make sure you are attempting to access the portal using the email address you used to purchase your key. If you purchased your key through a 3rd party vendor, you'll need to contact us support@lightburnsoftware.com so we can assign the key to your email address and grant you access to the portal.

If you do not see an email from Cryptlex when requesting a password reset, check your Promotions or Spam folders — the password reset email sometimes ends up there. If you still cannot find it, your email provider might be blocking the email from ever reaching you. Contact us to have us reassign your key to an address under a different provider. Gmail accounts work well.

License Expired



A LightBurn license is perpetual and never actually expires, but its update period expires one year after you first activate it. You will need to renew your key in order to continue receiving updates for another year. If you renew your key before your license expires, you'll receive an additional two months added to the one-year renewal.

To download and install a version of LightBurn compatible with your key, visit our release repository. You can find a table showing LightBurn versions and their operating system compatibility here.

You can renew your existing key here.

License Shows as Expired But Shouldn't

If you get the error about your license being expired but your license's update period is current (or valid for the version you're installing), it's possible that LightBurn isn't seeing your updated license.

To Fix:

1. Go to the License Management Portal and copy your LightBurn license key. You can also get your license key from your purchase confirmation email, if you still have it.
2. Paste the license key into the **License Key** box and click **Activate License**.

If you are still having problems, please contact support at support@lightburnsoftware.com for help.

Application is Being Run Inside a Virtual Machine

LightBurn keys are, by default, restricted from running in virtual machines. If you receive this error message, contact us at support@lightburnsoftware.com to request an exemption from this restriction.

Lost License Key

If you do not know your license key, try accessing our License Portal using the email address you used to purchase the key. Your key is visible in the License Portal.

License Portal

If you did not purchase your key through our website, you will be unable to access the License Portal. Search your email inbox for the original purchase email from the vendor you purchased your key through.

If you are still unable to find your key, contact us at support@lightburnsoftware.com.

Have to Enter Key on Each Computer Login

Follow our instructions on setting up a System Locked license key to activate the same key for all users on a given computer.

Camera Troubleshooting

Don't forget to calibrate and align your camera

If you haven't already done so, make sure you've gone through the lens calibration and alignment process for your camera.

Can't Connect to Camera, or No Camera Image

If LightBurn can't find your camera, or its capture is only a black screen:

- Make sure you've closed all other programs that may be using your camera.
- Make sure your computer is able to connect to the camera. If your computer can't access the camera feed, LightBurn won't be able to access it either. To do this, use your computer's built-in camera software (**Camera** for Windows and **Photo Booth** for macOS) to view the camera feed. Directions for both operating systems are below.

  I can see the camera and video feed

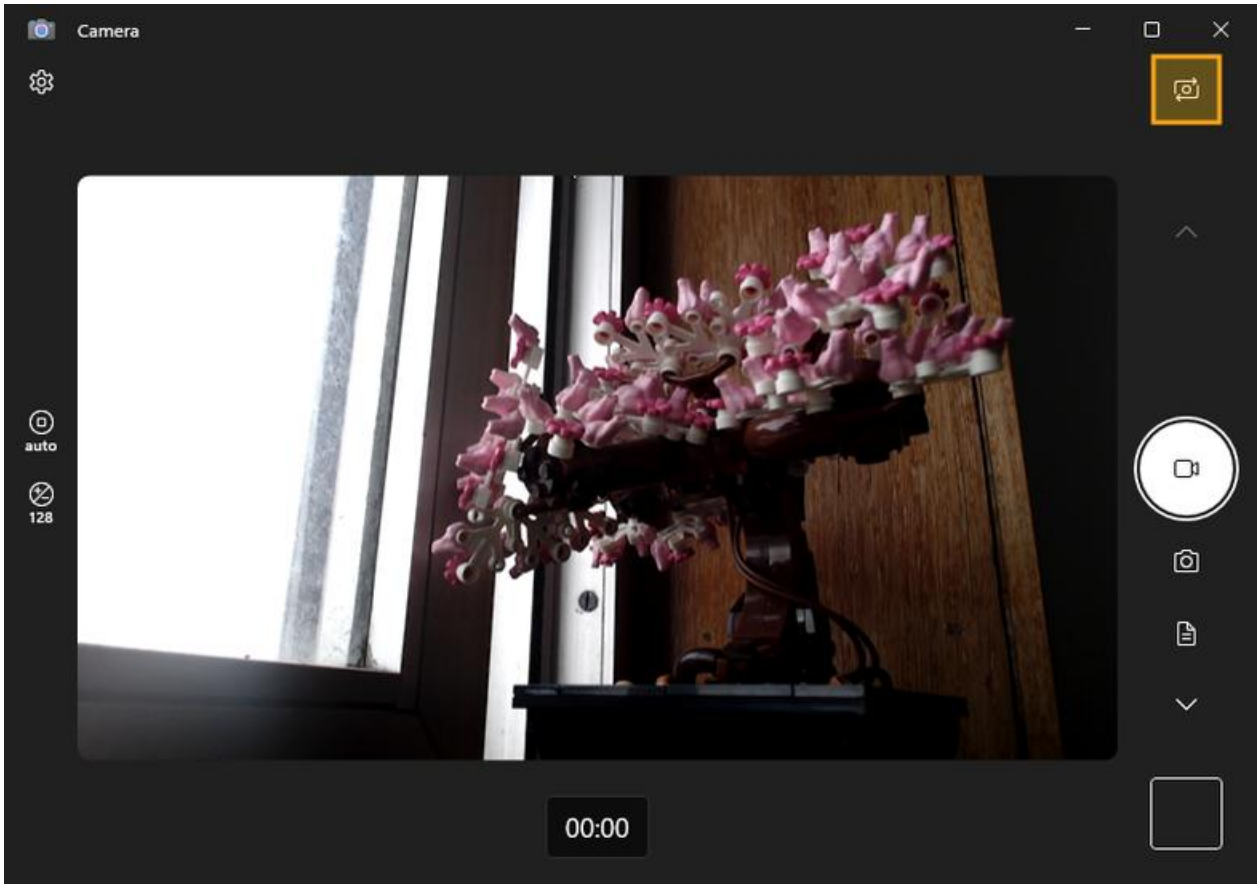
  If your built-in camera app sees your camera and gets a video feed, your computer can see the camera. Skip checking the USB cable and jump to making sure you've granted LightBurn permission to access your camera.

  I don't see the camera or there's no video feed

  If you can't connect to your camera and get a video feed, you'll need to get your computer communicating with the camera before worrying about LightBurn. Try checking your USB cable as described below.

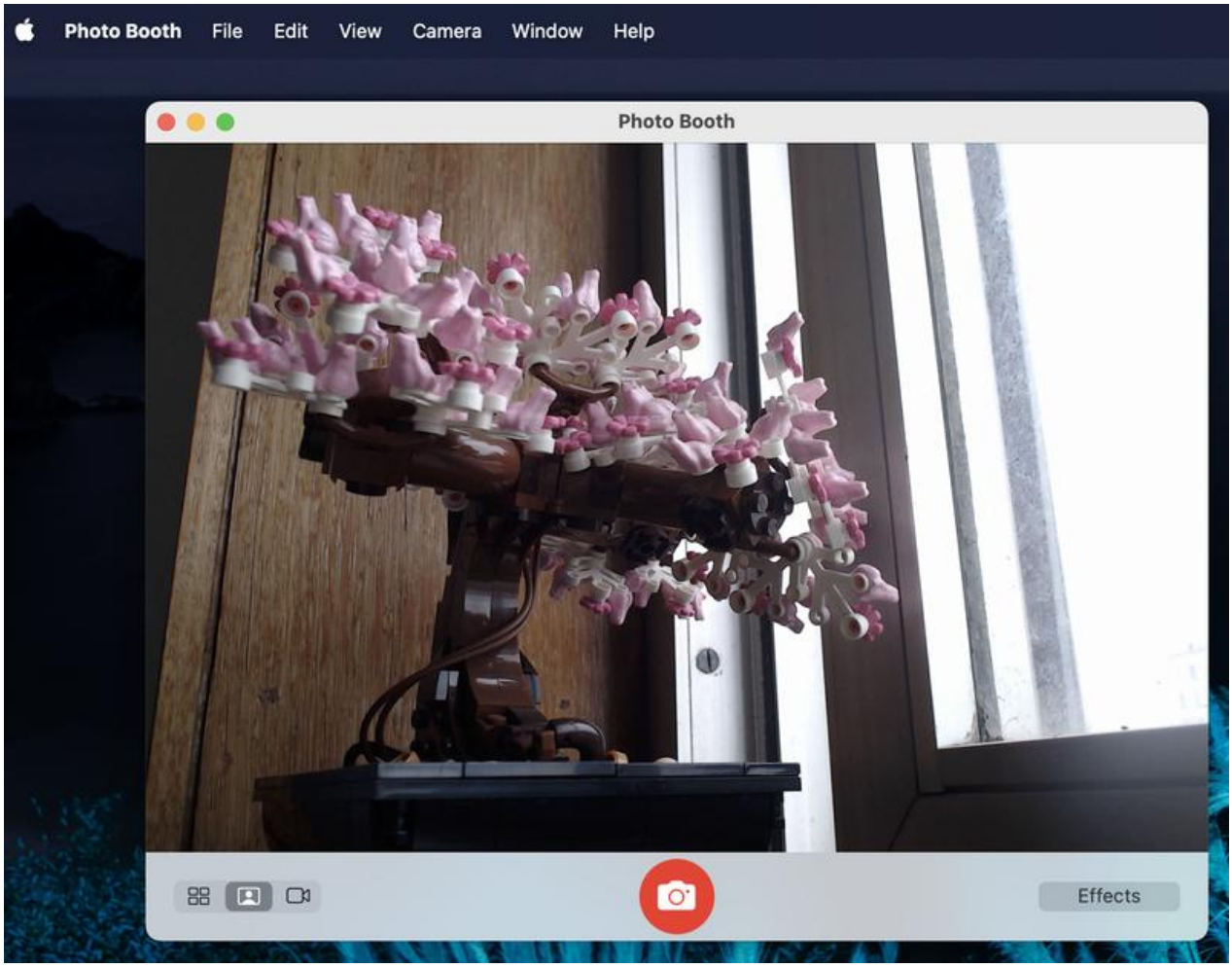
  For Windows

  Open the Windows **Camera** app and select the camera you want to use in LightBurn. The highlighted button in the top right corner cycles through all of your connected cameras.



For Mac

Open the built-in **Photo Booth** app, and use the **Camera** menu to select the camera you want to use in LightBurn.



USB cameras only

These directions are for USB cameras that connect with a USB cable. If you're using a network camera (wireless, or connected with an Ethernet cable), directions are in Network Cameras.

- If you're using a USB extender, try taking it out of line and connecting the camera directly to your computer. USB cables are limited to 3 meters (16 feet) long — anything longer can cause transmission loss. To connect over a greater distance, you must use an active, externally powered extender.
- Make sure you have granted LightBurn permission to access your camera in your system settings:

For Windows

  - Click the **Start** menu, type “camera privacy” and click the **Camera Privacy Settings** entry.
  - Check that **Allow apps to access your camera** is enabled.
  - Scroll down below the list of Windows apps and check that **Allow desktop apps to access your camera** is enabled too.

- If LightBurn isn't in that list, it should be.
- You might need to reboot your machine and try again after the settings change.
- If LightBurn still can't access the camera, and you're using Windows, try changing the **Camera Capture System** from **Recommended** to **Fallback** in your Settings/Preferences.

For Mac

- Go to **System Preferences → Security & Privacy → Privacy → Camera**
- Make sure the box next to LightBurn is checked
- In some cases, Macs require active, externally powered USB-A to USB-C adapters/hubs in order to connect to cameras.

Network Cameras

Typically, the URL for the video feed should be provided by the hardware manufacturer, or within the software if software is included.

Most security-style IP cameras use RTSP to stream video, which is not supported in LightBurn.

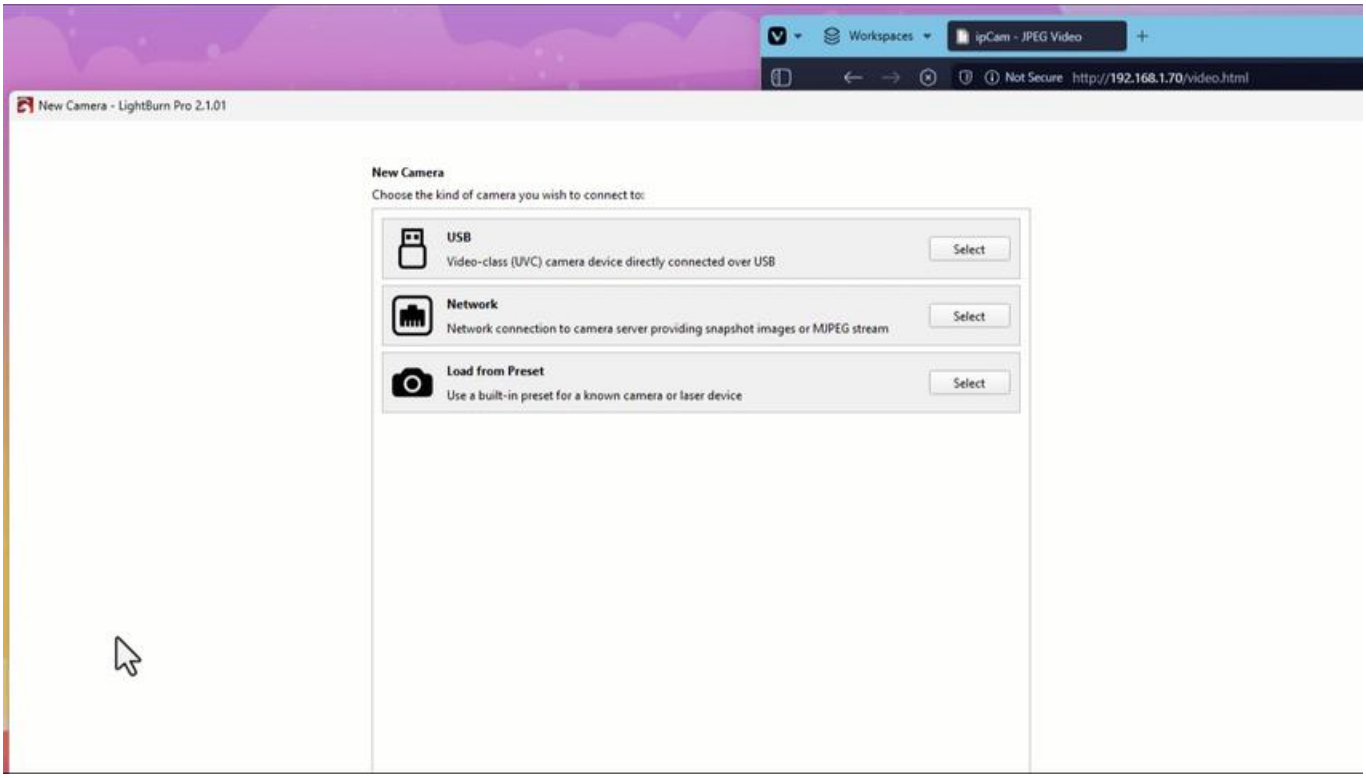
In general, if you can load your network/IP camera feed in a web browser and see a simple video feed — no UI for commands or controls — it should work in LightBurn. Any URL starting with `rtsp://` is using the RTSP protocol and is not compatible. Some RTSP cameras can provide both RTSP and MJPEG streams. See your camera's user guide for more information.

If you're looking to buy a new camera, look for one with MJPEG streaming.

Getting the Correct Feed URL

The web URL provided by many camera manufacturers is to a page that displays the camera stream, not the camera stream itself. If you can see the camera preview in your browser but get an error saying "URL seems to be invalid. Ensure that you are connected to the same network as your device", this is often the reason.

In the GIF below, using the URL provided for my example camera (`http://192.168.1.70/video.html`) results in this exact error.



To fix the error, try the following, as shown in the GIF:

1. Right-click on the camera feed in the browser.
2. In the context menu that comes up, choose **Open Image**, **Open Image in New Tab**, or **Open Image in New Window**. Depending on your computer's operating system and your preferred browser, these options may be named slightly differently.
3. You should now have a page displaying a single image. Copy the URL from this page, and paste it into the **Network URL** bar. If this URL is valid, your camera feed should load when you click out of the box.

> **Why not copy the entire URL?**
>
> If you see a `?` followed by a bunch of characters, it may or may not be a necessary part of the URL. This particular camera works fine with or without it.

If this method didn't work:

If right-clicking on the video feed doesn't offer you the option to **Open Image**, **Show Image**, **Load Image**, or something similar, you may not be able to get a valid video feed, or you may need to get the feed another way. Please reach out to support for help, and include what camera you're using.

Lens Calibration

If you're having trouble getting valid captures, or a good final calibration:

- Make sure you're using the newer AprilTag calibration pattern, which is linked from the **Calibration** wizard and from our Camera Calibration page. For your convenience, you can also

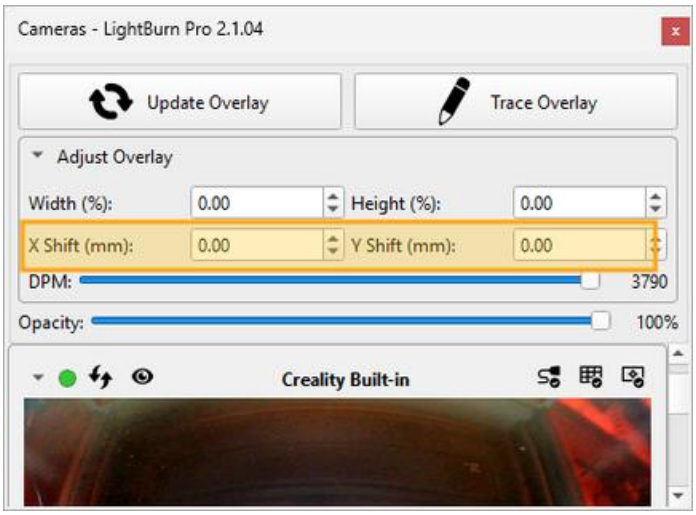
download it here. Some hardware manufacturers link to or ship a card with the older circle pattern, which doesn't work with our updated calibration method.

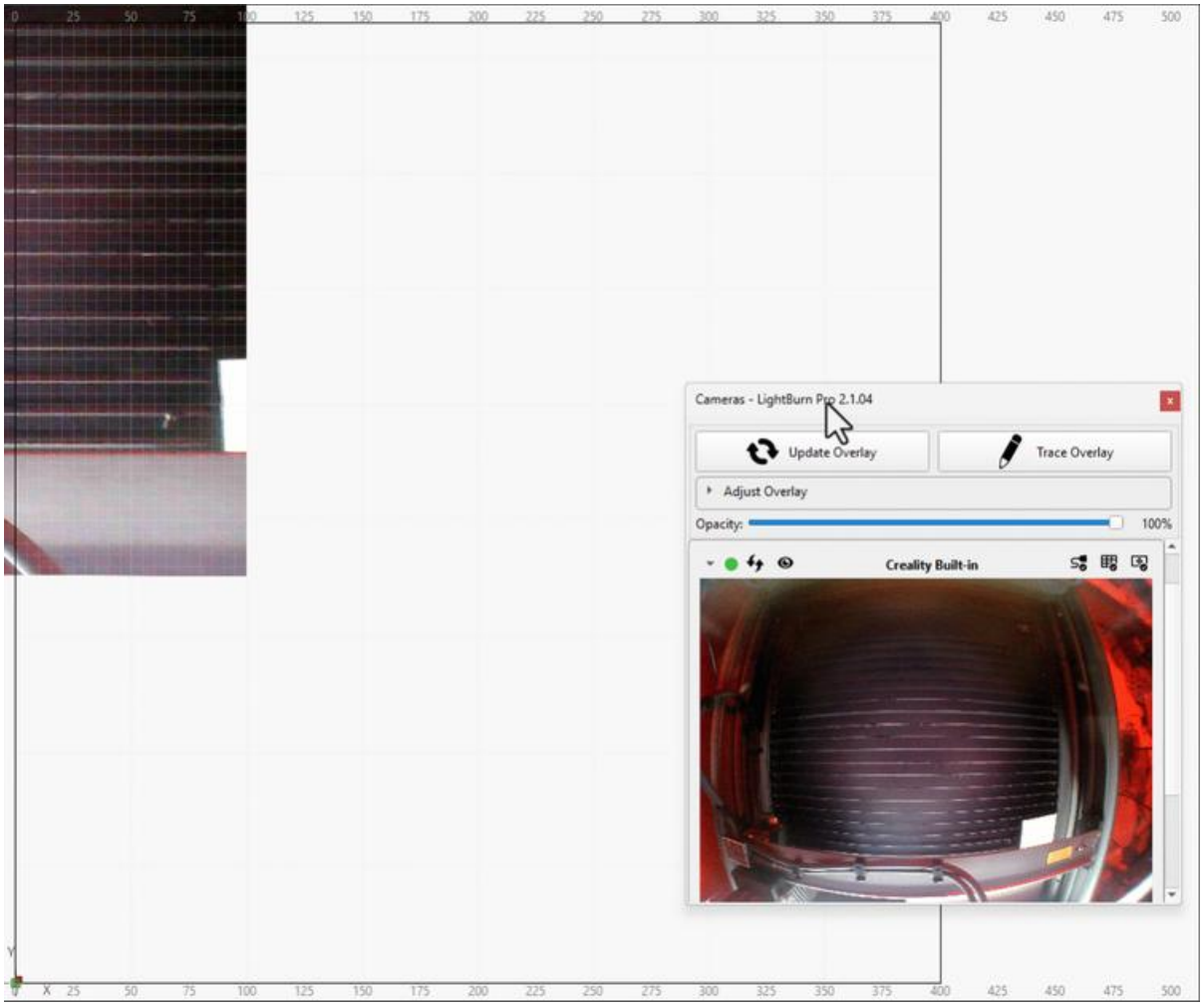
- Keep the calibration pattern as flat as possible by gluing it to wood, foam board, or any other perfectly flat material. If it isn't flat, the curvature will be considered part of your lens distortion.
- Make sure the pattern is in focus, and not shaking. If you're unable to keep your hand steady while holding it, place it on something instead.
- The calibration card should be facing directly at the camera for a fisheye lens, or perfectly flat and parallel with the lens for a non-fisheye lens.
- The pattern should fill roughly $1/9^{th}$ of the view of the camera for each captures. If it's much too small or too large, you won't get good results. You can rescale the pattern or move the pattern or camera closer or farther from one another to adjust the relative size of the pattern in the camera's field of view.
- The camera doesn't need to be on the machine for the lens calibration — it's only the lens and pattern that matter for this part.

Overlay Misaligned

Any time your Camera Overlay is misaligned, please check the following:

- Make sure your **Start From** mode is set to Absolute Coords.
- Check the Adjust Overlay to make sure **X-Shift** and **Y-Shift** haven't gotten changed. For most users, those values should be 0.





If these checks don't fix the problem, we have suggestions below for different situations.

Misaligned on First Setup

If your Camera Overlay is misaligned immediately after your first setup:

- Re-run the camera alignment process, taking care to tag the target markers as precisely as possible.
- To improve accuracy, zoom in on the target markers using the mouse scroll wheel or two-finger dragging.
- If possible, increase the size of the target marker engraving using **Pattern Width** and **Pattern Height**. The farther apart the target markers are from one another, the more accurate the alignment will be.
- If the alignment is close, but still a bit off, use the **Width**, **Height**, **X Shift**, and **Y Shift** fields in the Adjust Overlay section of the **Cameras** window to fine-tune placement and scale. You can see this window in action above.

Misaligned Between Uses

If the overlay becomes misaligned between uses, it means the position of your camera relative to your laser's work area has changed from when you did your initial alignment.

- Make sure your camera is securely mounted, with no room to wiggle.
- If it's mounted on a lid, make sure the lid is returning to exact same position it was in when you completed the alignment process.
- Make sure there is no side-to-side play in the lid's hinges.
- You can use a cord or rod to ensure the lid is always at the same position when opened.
- If you have tabletop engraver, make sure it cannot move at all — the motion of the laser head during engraving can sometimes jostle the entire machine if it is not secured down.

You'll need to address these issues, and then re-run the alignment process.

Misaligned When Switching to Thicker/Thinner Materials

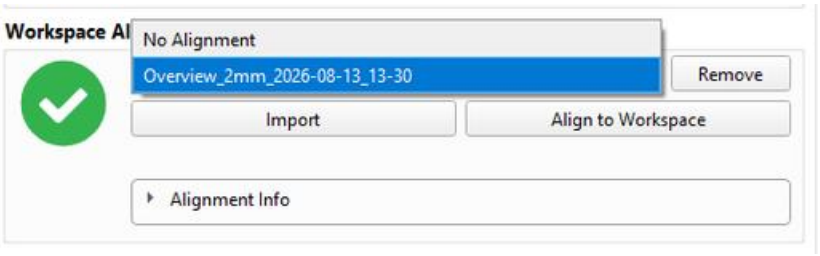
> **Improvements in LightBurn 2.1**
>
> LightBurn 2.1 made improvements to how LightBurn's camera system handles camera alignment for different material thicknesses so you don't need to go through the export/import process described in older versions. You can read more about it here.
>
> If you're using a version older than LightBurn 2.1, please use the older version of this page.

If you switch to a thicker or thinner material than the one you used to complete the alignment process, and your laser does not have an adjustable height bed, that will cause a misalignment — the height between camera and work area will have effectively changed.

The amount of misalignment will depend on the lens angle of the camera, the mounting angle, and how different the height is from the aligned height. If you're using a top-down view with a narrow angle lens, the difference will be much less than if your camera is wide angle, or viewing from an angle.

Fixing the Problem

- Make sure your **Material Height** is set correctly in the Cuts / Layers Window.
- If you've already aligned your camera for multiple heights, open the Edit Camera Window and select the alignment for your current **Material Height** in the Alignment Dropdown. The Material Height for each saved alignment is included in the name.
- If you haven't gone through the alignment process at the current **Material Height**, you'll need to either align for the current height or try using the **Interpolate Alignment Height** option described there.


Workspace Al
No Alignment
Overview_2mm_2026-08-13_13-30
Remove
Import
Align to Workspace
Alignment Info

CNC-Based Lasers

Some of our users have set up CNC routers – particularly the inexpensive and widely available 3018 machines – as lasers. Although these are GRBL devices similar to most diode lasers, they are pre-configured for use as CNC machines and you will need to modify settings to account for several important differences.

To see your device's current settings, type `$$` into the Console Window and press `Enter`, or go to Laser Tools → Machine Settings. Both show the same values, but the Machine Settings view may be a little easier to use.

Although we can't account for every possible variation in machines, you will most likely need to do the following:

- Use GRBL Device Profile
- Enable Laser Mode
- Change Status Reporting Mode
- Set Max Speed
- Enable Overscanning
- Homing

Use GRBL Device Profile

What To Do

When you add your machine to LightBurn, make sure you select the **GRBL** device profile, not **GRBL-M3**. If you've already added your machine, click the **Devices** button in the Laser Window, select your machine, and click the **Edit** button to check the device profile and reassign it if needed.

Explanation

Machines set up with the **GRBL-M3** profile don't support **Variable Power** mode.

Enable Laser Mode

What To Do

Type `$32=1` into the Console Window and press `Enter` or set **Laser mode enable ($32)** to **True** in Machine Settings.

Explanation

Laser Mode will do three things:

- Remove the pause when power output changes, since there's no need to wait for a rotating spindle to speed up or slow down

- Set the power output to 0 to turn off the laser when moving between cuts
- Enable **Variable Power** mode to adjust the laser's power output from 0% when the laser is stopped, to 100% of the power output you've set, once the laser reaches the full speed you've requested the job run at.



Change Status Reporting Mode

What To Do

Type `$10=0` in to the Console Window and press `Enter` or set **Status: Position reporting ($10)** to **Workspace Position** in the dropdown menu.

Explanation

GRBL-based CNC routers typically have their origin in the rear right corner and use a negative coordinate system, while GRBL-based lasers typically have their origin in the front left corner. This is specific to using a machine intended as a CNC router as a laser, and is typically not needed for other lasers.

Set Max Speed

What To Do

Increase the maximum speed settings to account for the higher speeds needed for a laser compared to a CNC router.

Find your machine's steps/mm values:

In the console, type `$100` and press `Enter` to find the X steps/mm value, and repeat the process with `$101` for Y. If you prefer to use the Machine Settings window, look in **Outputs setup** for **X Steps per mm (\(100)** and **Y Steps per mm (\)101)**. These values will typically be the same.

Set your max speed based on steps/mm

Check the table below for the recommended max speed:

Steps/mm Value	Recommended Max Speed
800	2250
1600	1125

Set the max speed for the X Axis by typing `$110=SPEED` (with SPEED replaced by the actual number) and pressing `Enter`. Repeat with `$111` for the Y Axis.

If you prefer to use the Machine Settings window, look for **X Max rate (mm/min) (\(110)** and **Y Max rate (mm/min) (\)111)** and set the appropriate speeds there.

Explanation

Most 3018 and similar CNC routers have their maximum speed set to about 500 mm/minute (8.3 mm/second), which is really slow for a laser.

Torque is more important than speed for a CNC routers, so these machines typically have their motor stepping rates set very high (either 800 or 1600 steps/mm). An 8-bit GRBL controller will only output about 30,000 steps per second, so you're limited to a maximum of 2250 mm/minute (37.5 mm/second) or 1125 mm/minute (18.75 mm/second).

With variable power mode enabled, if you ask the controller to go faster than it's allowed to go, it will reduce the output power to compensate. If you ask for 100% power at 50 mm/second, but the controller can only go 25 mm/second, the laser will only move at half the speed you asked for. Without variable power mode, the power will remain at 100%, resulting in a much darker mark than expected. With variable power mode, the power will be reduced to 50% to match the reduced speed.

More information about configuring GRBL can be found at the GRBL documentation.

Enable Overscanning

What To Do

Enable overscanning in the **Cut Settings Editor** for Fill or Image layers.

If your machine accelerates slowly, you may need to set the overscanning value significantly higher than the default.

Explanation

The **Overscanning** setting prevents engravings from appearing darker at the edges and lighter in the middle as a result of the laser slowing down to change directions.

With Overscanning disabled, the laser slows down and changes directions while within the bounds of the engraving. The reduced speed near the edges make the engraving darker. With Overscanning enabled, LightBurn runs the tool head a bit past the boundaries of the engraving with the laser off to change directions.

Homing

What To Do

Homing your machine allows you to identify a consistent, repeatable starting point.

If your device has limit switches to enable automatic homing, you can use the **Home** button in the **Laser Window** combined with a workspace offset for automated and repeatable positioning. The linked directions include information on saving the offset command to a macro button.

Unfortunately, the 3018 and other similar machines typically don't come with homing switches, so the zero point is reset to the head's current location every time you turn it on. To set the zero point properly, do one of the following:

- Make sure the head is in the front left corner before turning on the machine
- Jog the head over to the front left corner with the arrow buttons in the **Move** window, type `G92 X0 Y0` into the console window, and press `Enter`

Either way, don't forget to make sure your machine is set to Workspace Position mode

Explanation

The `G92 X0 Y0` command tells the machine's controller "Your current location is X=0 Y=0".

Workspace Position mode tells the controller to report its position relative to the origin you've set, not the machine's zero position. With this set, you can use the machine in **Absolute Coords** mode, which is recommended. This and other **Start From** modes are described in Coordinates and Job Origin.

Drivers

If you have trouble connecting to your laser, you may need to install or update the drivers that allow your computer to communicate with the laser's controller software.

We've done our best to list the driver requirements for a variety of devices and operating systems, along with information on how to install the drivers. If you have a device with a driver that isn't listed, we'd appreciate you letting us know by clicking the envelope icon next to the header below to send an email to our support and documentation teams.

Some lasers that come with their own software may require you to install that software to install the drivers.

Driver List

Laser Type	OS	Notes
Diode lasers	All	If not specified, probably requires the CH340 driver. The latest version is available from the WCH website for Windows and Mac. The Windows CH341 driver has worked for our team on ARM-based computers as well.
Ruida or Top Wisdom	Windows	FTDI driver required. If you're not sure you've installed it, you can rerun the LightBurn installer and check the box to install the FTDI driver. You can also manually install the driver by going to `C:\Program Files\LightBurn\` and double-clicking `FTDI_Driver.exe` to run the installer. Starting with the installer for LightBurn 2.2, the included driver is compatible with ARM / Snapdragon computers as well.
Trocen	Windows	Might need to install LaserCad to get Trocen's driver
Galvo lasers	Windows	See Galvo Installation for directions.

Windows on Arm Drivers

With previous versions of LightBurn, if you were using a Windows computer with an Arm processor, you needed to download and install different drivers compatible with your computer.

In LightBurn 2.2, the drivers included in the installer are compatible with Arm processors, so no special driver installation is needed.

Electrical Problems

Shared Power Circuits

Other devices sharing the same power circuit as your laser, such as chillers and external air assists, can cause a small power drop that causes your laser to brown out and temporarily disconnect or reset. Try isolating the laser to its own breaker if possible.

> **Info**
>
> You can check if other outlets are on the same breaker by turning off the breaker controlling your laser's power outlet and seeing if other outlets also turn off.

Poor Bonding/Grounding

Bonding is tying the various metal parts of a device together, so that if one part builds up a charge, all the other parts build up the same charge. If your laser isn't bonded correctly, static can build up on the laser head or rails. If the laser is not grounded (earthed), that static build-up can eventually discharge through the USB connection to the computer, causing communication errors. This is especially common with lasers driven by belts that interact with plastic wheels. You can prevent this by tying the laser's frame elements together electrically and making sure your laser controller is grounded correctly.

Electrical Noise

Route your USB cable so that it avoids the stepper motors or anything else that could cause electrical interference.

Galvo Laser Troubleshooting

Below are some common problems and frequently asked questions related to using a Galvo laser with LightBurn.

Can I import an EZCAD project/parameter/file?

LightBurn does not have the ability to import or open EZCAD project files or parameters. Due to the file format not being documented, and the sheer volume of effort to produce a useful, cohesive importer, it isn't possible at this time.

If the grid in EZCAD is turned off temporarily, LightBurn can trace screenshots of EZCAD to import placement in many situations.

Why won't LightBurn recognize fiber laser plugged in with USB on Linux?

> **Warning**
>
> LightBurn version 1.7 is the last version compatible with Linux, versions of Windows prior to 10, and versions of macOS prior to 11. Revert to version 1.7 and see our 1.7 documentation for information and installation instructions.

This is an issue we haven't generally encountered in our own testing in Ubuntu and Fedora, the two distros we support. The laser is treated as a libusb device and should just work. However, this is occasionally an issue with USB permissions. Please try the following, as discussed on our forum.

- Unplug your laser from the computer.
- Run `sudo gedit /etc/udev/rules.d/usb.rules` (you can use any text editor you prefer) and enter the following contents:

  `SUBSYSTEM=="usb", MODE="0666"`
- Save the file and close.
- Run: `sudo udevadm control --reload-rules`
- Now plug your galvo USB connection back into the machine.
- Run LightBurn without sudo and you should be able to automatically find and connect to the device.

Where are the hatch settings?

Hatch in EZCAD is the equivalent of Fill Mode in LightBurn. See our EZCAD Hatch Quick Translation page for help matching the settings you're used to.

Why won't LightBurn connect to my laser after moving USB ports?

Some lasers require you to fully reboot the device after moving USB ports. Try these steps:

1. Turn the laser fully off
2. Move the USB cable to the correct port.

3. Turn the laser back on.

My Galvo is trying to engrave my rotary, what's wrong?

LightBurn allows you to shift the center position of your rotary output to compensate for the center of the rotary not lining up precisely with the center of the field.

If your offset value is set incorrectly, output may be shifted far to the left, right, top, or bottom of your laser's work area. See Rotary Mode (Galvo) — Output Center for more information.

How to Ask For Help

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=cpm9hSia6Gk

If you're having trouble using your laser with LightBurn, you're in the right place — the Troubleshooting section of our official documentation.

But if you can't find answers here, you might need additional assistance. You can reach us by email at Support@LightBurnSoftware.com, or join our Support Forum, where you can search to find solutions to your issue, or make a new post asking for help.

So that we can help you as quickly and effectively as possible, please keep in mind the following things, and provide these important details. Scroll down for more information on each list item, plus a few additional things we might request.

Provide Setup Information:

- The specific model of laser you're using
- The type of controller it uses, and what firmware it's running.
- A direct link to your laser's product page
- What operating system you're running LightBurn on (Windows or macOS)
- The version of LightBurn you're using
- Method of connection (USB, Ethernet, WiFi, or LightBurn Bridge)

Explain the Issue:

- Step-by-step explanation of what you're doing and seeing
- What you expected to happen, and what happened instead
- Context — is this your first time using your laser? Is this issue new, or has it happened before?
- Please use complete sentences and explain the issue as thoroughly as you can
- Include screenshots — not cell phone photos — of your entire LightBurn window
- Share your LightBurn project file (.lbrn or .lbrn2)
- Send Support Data

Please be kind to our Support Agents

We know it's frustrating when things don't work the way you expect them to, especially when you've got orders to fill and deadlines to meet. But please remember there's a human on the other side of the screen. Our Support Agents want nothing more than to help resolve your issue as quickly as possible.

Setup Information

Laser Model

We support hundreds of machines, and they all behave differently depending on their exact type. Some lines of machines even have features that are unique to them alone.

Please tell us the name of your laser's manufacturer, and its exact model. A direct link to the product page where you purchased the laser is a big help.

Controller / Firmware

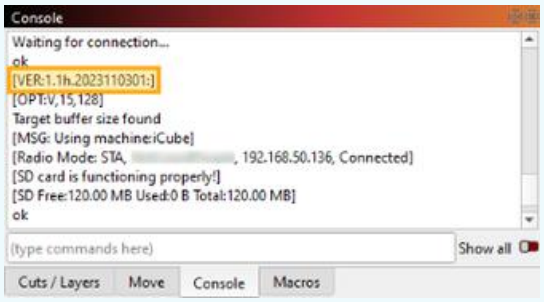
> **Check for Firmware Updates**
>
> It's a good practice to regularly check for firmware updates on your laser manufacturer's website. Sometimes firmware updates are necessary to fix an issue your laser is having, or to ensure compatibility with LightBurn.

A controller is essentially a computer within your laser, and the firmware it runs determines the type of instructions it can translate into signals it delivers to the motion system.

Because of the differences in controllers/firmware between different lasers, we might need to know what controller/firmware runs your laser in order to assist you with your issue.

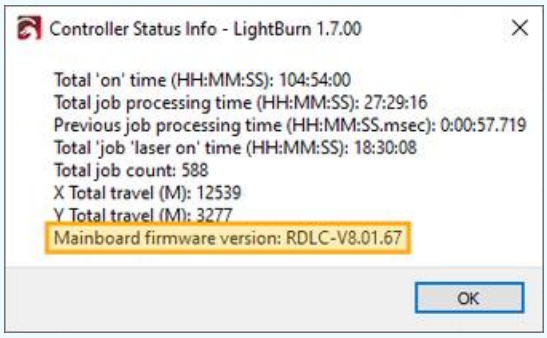
If you're not sure, see Identify Your Laser. If you're still not sure, a direct link to a product page for your machine is usually enough for us to figure it out on our own.

> **Identifying exact firmware version for GCode-based lasers**
>
> Most entry-level diode lasers use GCode-based controllers.
>
> When you connect to a GCode-based laser, you'll see a start up message in the Console Window, sent by the controller to LightBurn. This message contains the name of your laser's firmware, and its version.



> **Identifying exact firmware version for Ruida controllers**
>
> Ruida controllers are common in more industrial-grade machines. If your machine is a CO2 laser in a metal case and has an LCD display, it likely has Ruida controller.

The firmware version will usually appear on the controller's display when you start it up, but when you've connected to it using LightBurn, you can also go to Laser Tools → Get Controller Info to see it.



Operating System

A handful of processes are different depending on whether you're running LightBurn on **Windows** or **macOS**, and we occasionally see software bugs that affect only one or the other operating system.

Sometimes knowing the exact version of your operating system helps too.

Here's how to check your specific version:

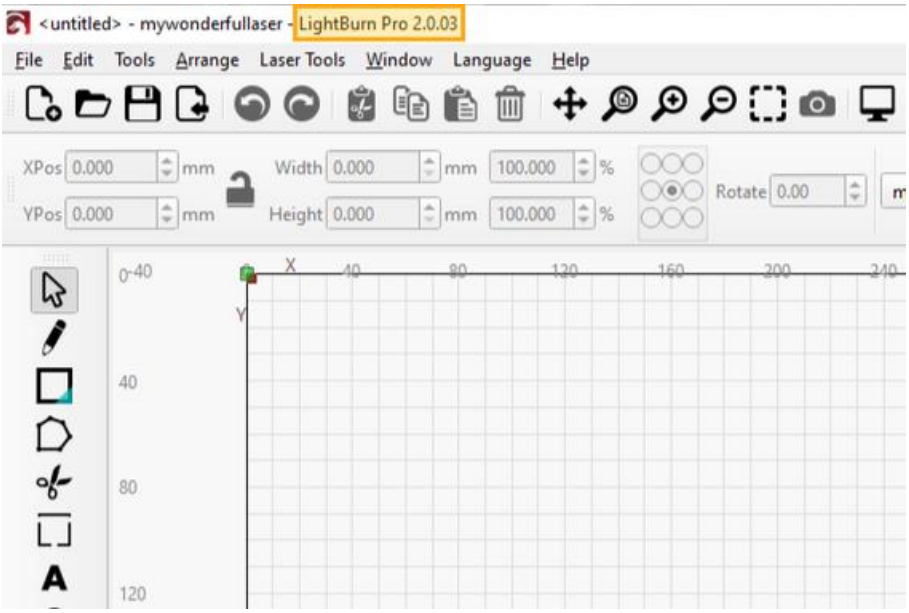
- Windows: go to **Settings** → **System** → **About**
- Mac: go to **Apple** → **About this Mac**

LightBurn Version

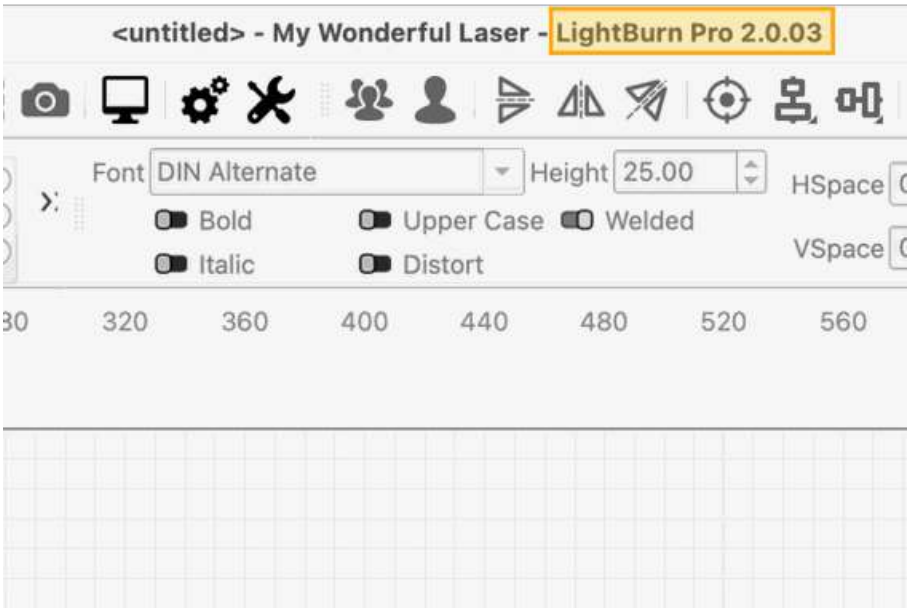
Because LightBurn has changed over the years, we need to know the exact version of LightBurn you're running to know what behaviors you should or shouldn't expect to see. Some tools work differently than they used to, or didn't exist at all in a past version.

If you're using the latest version, we want to know that too, in case you've uncovered a bug we need to investigate.

Your LightBurn version is always visible in the topbar of the LightBurn window. It's off to the left on Windows, and centered on macOS.



The location of the LightBurn version on Windows.



The location of the LightBurn version on macOS.

Note

Older releases of LightBurn did not contain the version information in the topbar. If you do not see the version number there, go to Help → Quick Help and Notes → **About** instead.

Connection Method

> **Tip**
>
> Be sure to check out our Connection Troubleshooting page before contacting support.

If you're having trouble connecting to your laser, the exact method you're using makes a big difference to the troubleshooting steps we'll recommend.

While this information is mostly relevant for connection issues, it can also be an important piece of the puzzle if you're seeing unexpected results from your laser.

Please tell us whether you're connecting by:

- USB
- Ethernet
- WiFi
- LightBurn Bridge

Explaining Your Issue

Step-by-Step Description

Whatever issue you're having, we need details to help you resolve it. Lots of them.

Tell us what you were doing step-by-step, what you expected to see happen, and what did or didn't happen instead.

Especially if what you're doing is too complicated to fit in a few sentences, sending us a numbered list of the actions you're taking is a great way to help us understand what you're doing and seeing.

Follow this format:

1. I did ____
2. then ____
3. then ____
4. I expected ____ to happen,
5. but instead ____ happened / did not happen.

Please use complete sentences, proper grammar, and punctuation. This isn't an English exam, and you won't be graded, but sentence fragments and run-on sentences are not easy for us to understand.

If you are a non-English speaker, please feel free to write in your native language — we'll use Google translate.

Context

Some additional context helps too:

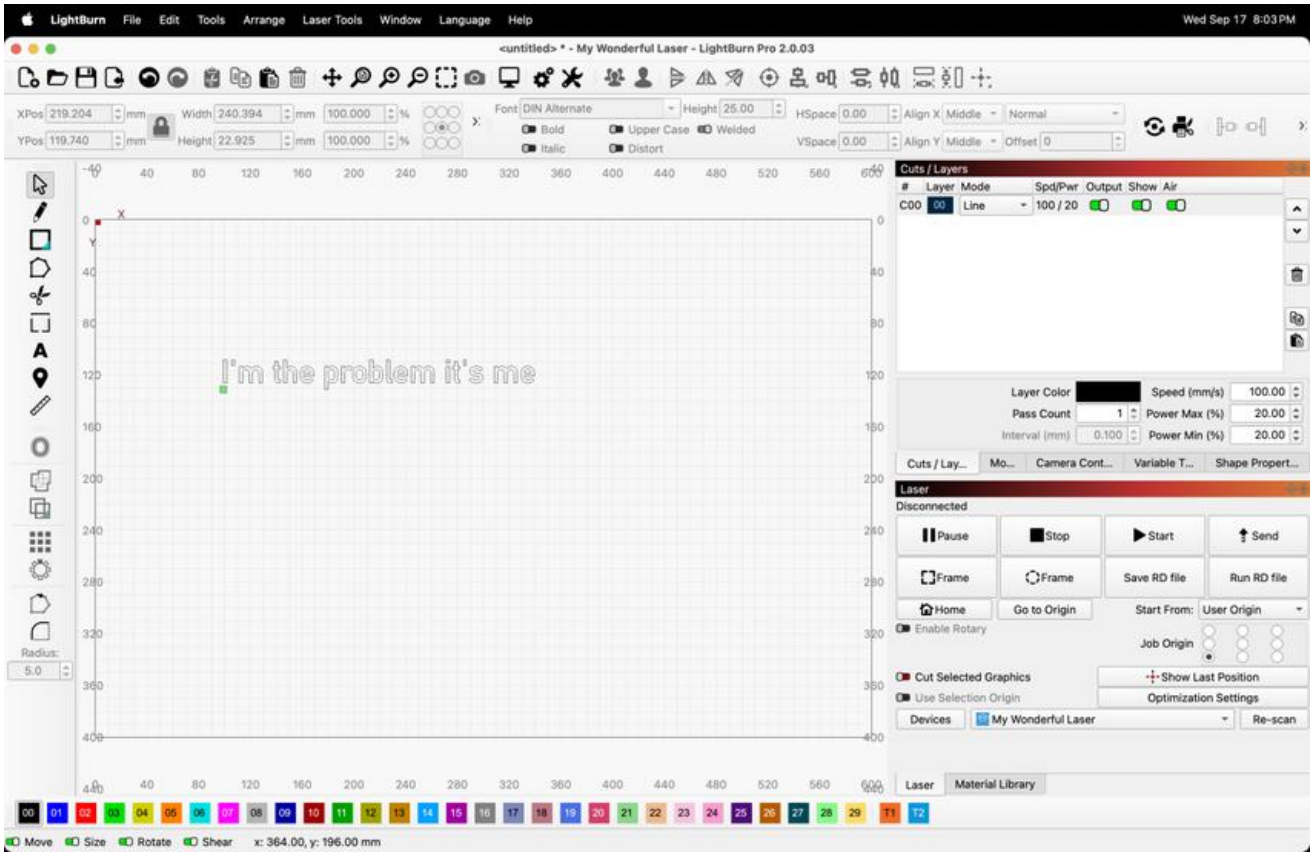
- Is this your first time setting up your laser or using LightBurn?
- Did what you're trying to do work before, and then stop working, or has it never worked for you at all?
- What have you tried doing to resolve the issue so far?

Screenshots / Videos

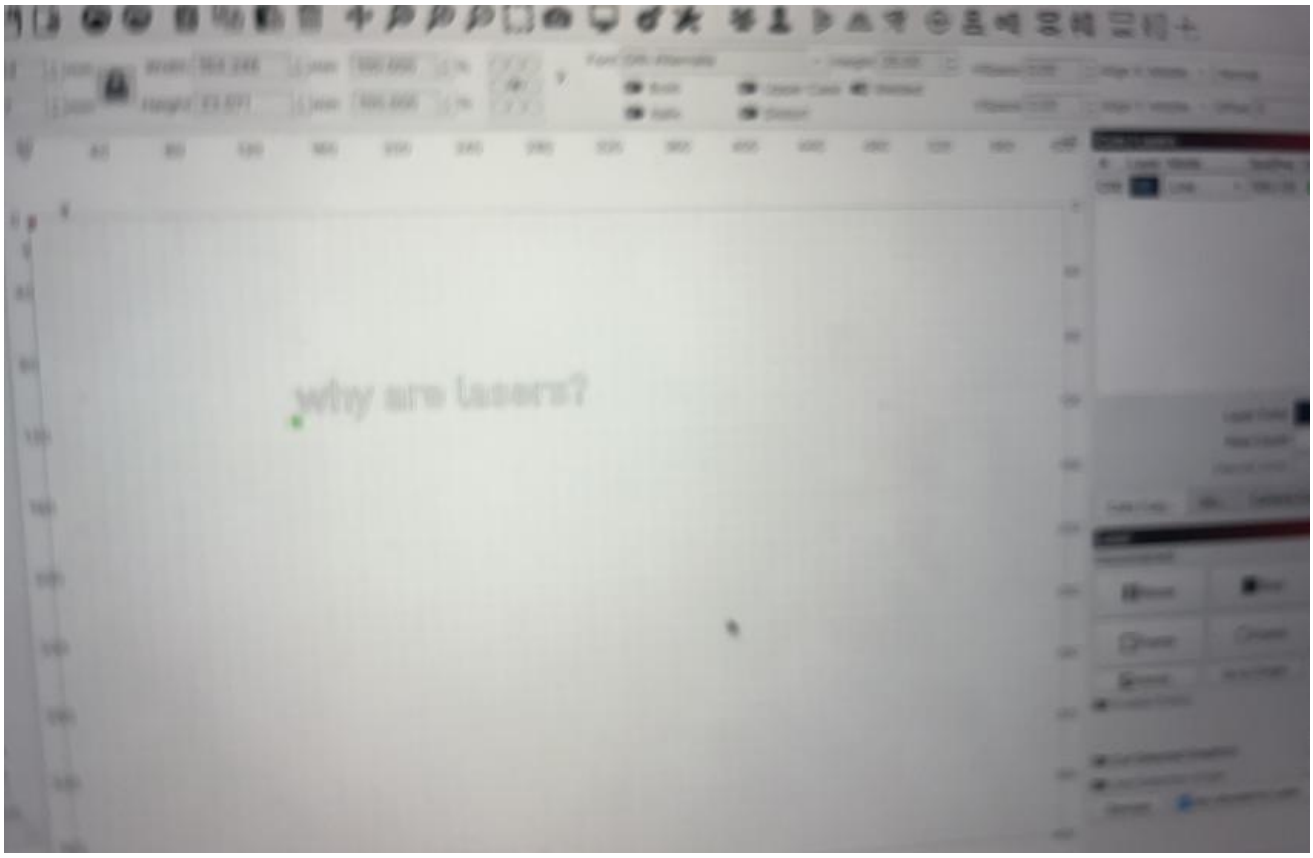
Even with all that information, it's hard to know exactly what's happening without visuals. Please send us full screenshots of your entire LightBurn window, with everything in view.

When we say screenshot, we mean your computer's built-in ability to capture an image of its own screen. Low quality cell phone photos are extremely difficult for us to see.

If you don't know how to take a screenshot, visit Take-A-Screenshot.org for instructions.



A high quality screenshot of the entire LightBurn window, containing lots of useful information to help us help you.



A low quality cell phone photo, lacking almost any useful information. This will make your Support Agent sad.

Videos can sometimes be helpful as well, especially when you need to show us the behavior of your machine. Please do your best to keep the camera steady, and focused on the relevant activity.

Please do not take videos of your computer's screen using a cell phone, and use your computer's built-in screen recording feature instead.

Even if you're sending us screenshots or videos, remember that we still need the written step-by-step explanation so we can make sense of what you're showing us.

Project Files

Sending us the project file you're working with in addition to screenshots is even better. We can look over all your settings, and directly test out any edits you're trying to make.

Go to File → Save (or Save As) to save your project file in `.lbrn` or `.lbrn2` format, then attach that file to an email to us.

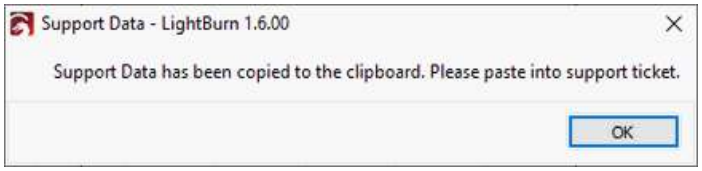
Please don't send us the original source `.svg`, `.dxf`, or `.ai` file, unless the issue you're having is specifically with the importation process.

Support Data

Sometimes, we might ask you to send us your Support Data.

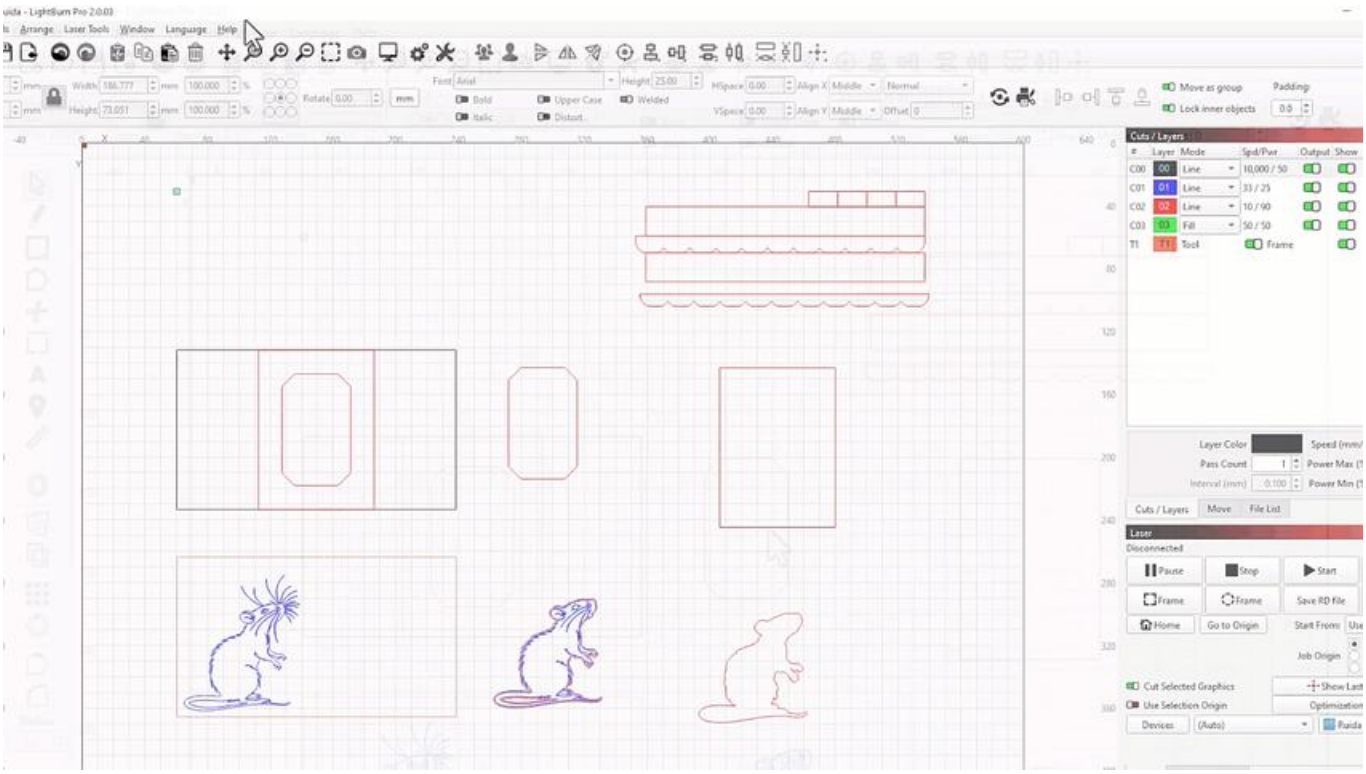
Go to Help → Generate Support Data to copy an automatically-generated block of encoded characters to your clipboard. The clipboard is the temporary storage location your computer uses when you copy and paste items.

LightBurn will return a message indicating that the data has been successfully copied.



Then, in an email to us, press `Ctrl`/`Cmd` + `V` to paste it. It might look like a random string of characters, but don't worry, that's what we need.

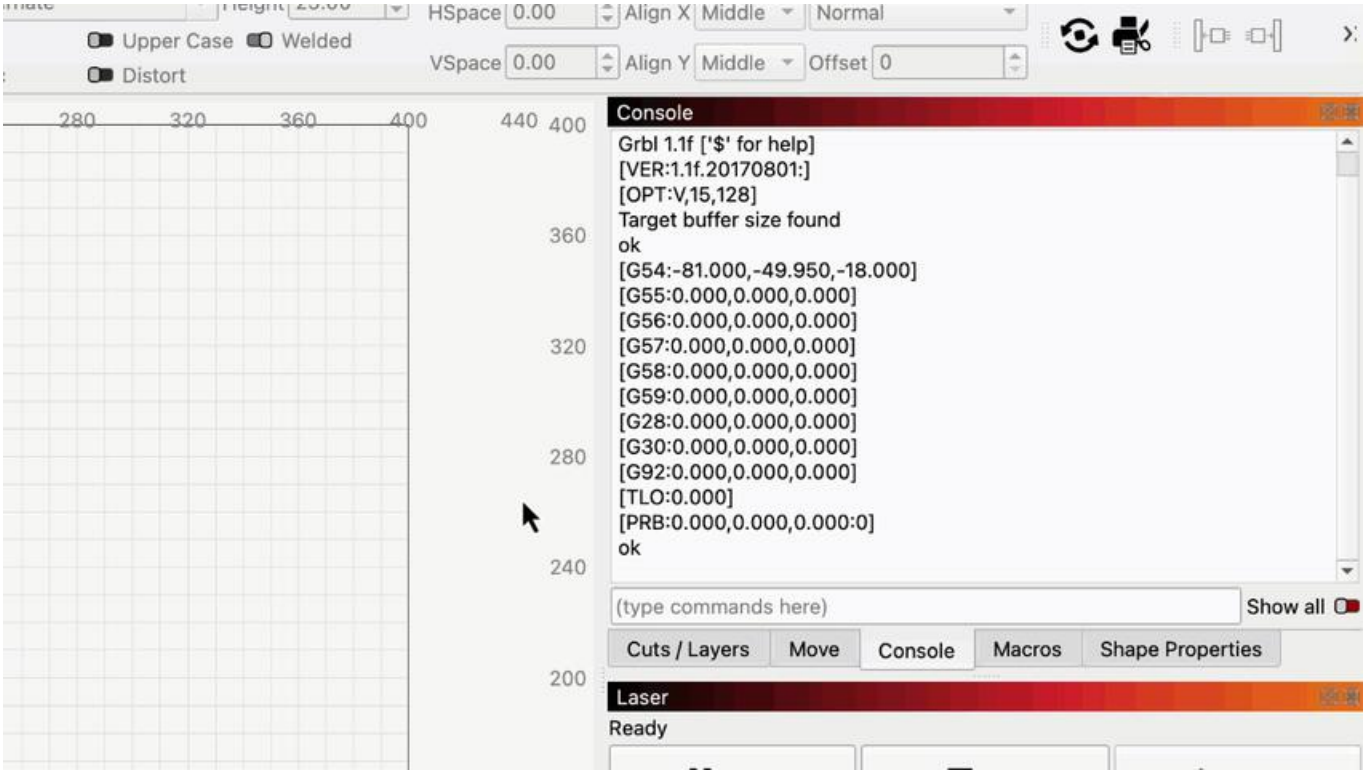
> **Warning**
>
> Never shared your Support Data publicly.



Console Information

If you're using a GCode-based laser, we might ask you to send us information from the Console Window.

You can select and copy that text (`Ctrl`/`Cmd` + `C`), then paste it (`Ctrl`/`Cmd` + `V`) directly into a message to us.

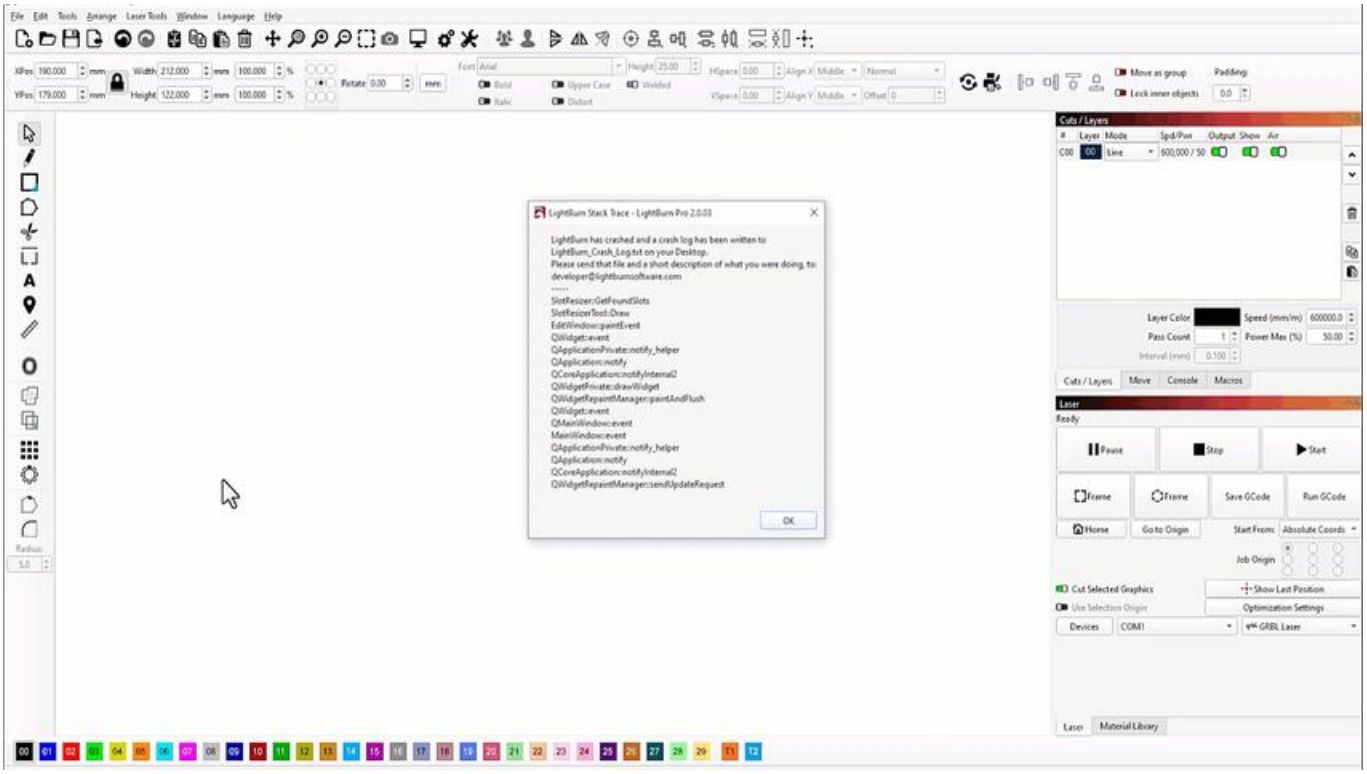


Crash Logs

If LightBurn crashes, we want to know about it. You might see a **LightBurn Stack Trace** message, and a `LightBurn_Crash_Log.txt` file will be saved to your computer's desktop.

Sending us this file is the best way for us to identify exactly what caused the crash, and fix it.

Please remember that we still need the explanation of what you were doing, your LightBurn project file, and that we may need to follow-up to ask for more details.



Feature Requests

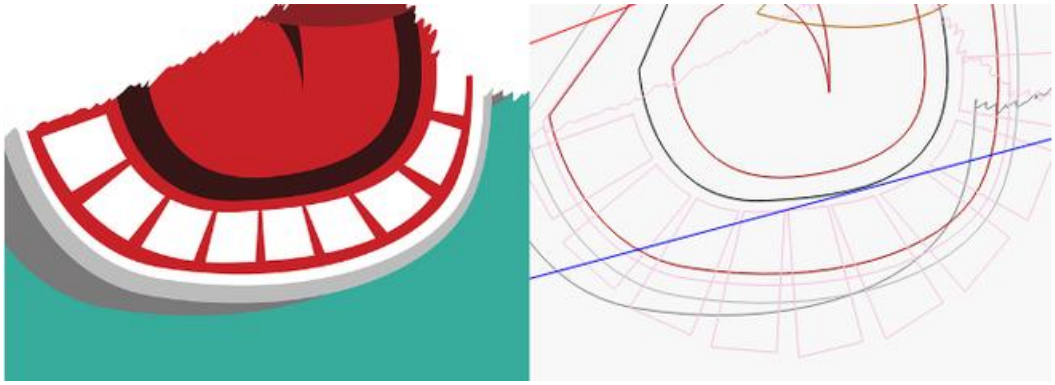
If you've got an idea for a new feature, we're all ears. But the best way to share it with us isn't to contact support.

We provide a dedicated site for users to vote for new features — LightBurn.Fider.io.

You can search to see if this has been requested, and vote it up, or add it so others can vote their support as well. This helps us prioritize which features to direct development time toward.

Vector Artwork Imports With Extra Lines

When importing vector artwork into LightBurn, sometimes files open with a large number of extra lines and shapes that aren't visible in the output from other graphics editing software, as shown in the example below.



This happens when artwork is created by stacking shapes on top of each other and/or using masks to hide parts of shapes. This method works fine for digital artwork, but leads to extra shapes like those shown above when loaded into LightBurn, which is designed to work with all the paths in your file.

Fixing Extra Lines and Shapes

Depending on the artwork and what you want to do with it, there are multiple approaches to correcting these files.

Rasterize for Engravings

If you're engraving the design and don't need the actual paths, you may want to save the design as a raster image, such as a `.png` file, import the new file, and engrave it as an image.

Rasterize and Trace

For some artwork, you may be able to use LightBurn's Image Tracing to recreate simplified paths that you can cut along.

1. Save the design as a raster image, such as a `.png` file.
2. Import the new image file into LightBurn.
3. Trace the image to recreate the paths.

Clean Up Original Paths

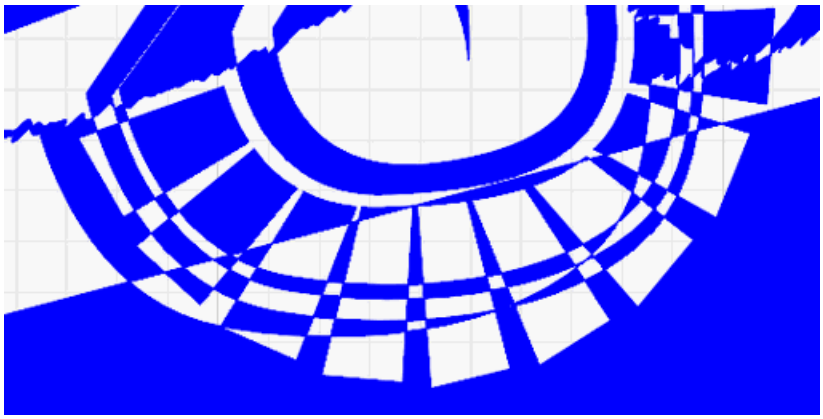
You may be best served by cleaning up the original paths. This can be time-consuming, but gives you the most control over your finished result. The exact method will vary from file to file, but in general:

1. Select the shapes you want to keep and move them onto a layer of your choice.

2. If shapes extend further than wanted — as with the teeth in the sample image above — you may need to use LightBurn's editing tools to modify the shapes to produce the end result you want.
   - Use Boolean and Weld tools to combine, cut, or intersect shapes.
   - Use Cut Shapes to divide shapes using another closed shape as a tool.
   - Use Trim Shapes to trim line segments back to their nearest intersections.
   - Use Edit Nodes to directly edit vector shapes.
3. Delete or turn off the Show and Output switches for layers containing unwanted shapes.

Other Problems

When you move objects onto the same layer, you may find that graphics set to Fill Mode don't appear filled, or you may end up with results like below, where shapes flip back and forth between filled and unfilled. In both cases, the likely culprit is duplicated or overlapping shapes.



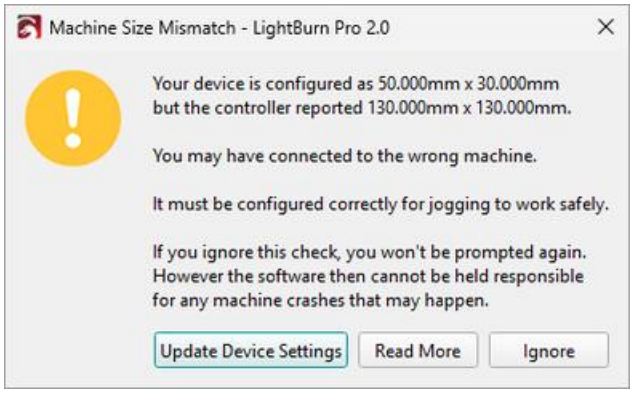
- Duplicated shapes negate each other and can cause closed shapes set to Fill Mode to appear unfilled and fail to engrave, or to cut/engrave multiple times in Line Mode. Use Delete Duplicates to eliminate shapes that are perfectly identical in size, shape, and position.

  > **Filled vs. Wireframe View Style**
  >
  > Graphics set to Fill Mode are only rendered as solid objects in your Workspace when you are using a Filled View Style mode. You can change your View Style in the Window Menu, or by pressing `Alt`/`Option`+`Shift`+`W`.
  >
  > See View Style for more information.

- The sample image above shows overlapping shapes on the same layer that aren't exact duplicates. The checkerboard effect comes from the shapes negating each other where they intersect. See Fill Mode and How to Invert a Vector Engraving for more information on these concepts.

Machine Size Mismatch

A machine size mismatch occurs when the machine size LightBurn reads from your device's firmware doesn't match the machine size saved in Device Settings.



Machine Size Mismatch

The error message reads:

"Your machine is configured as width x length but the controller reported firmware width x firmware length.

You may have connected to the wrong machine.

It must be configured correctly for jogging to work safely.

If you ignore this check, you won't be prompted again. However the software then cannot be held responsible for any machine crashes that may happen."

This message occurs when the **Working Size** in LightBurn's device settings is different from the device's firmware machine size values. If this happens, one of the following is likely the cause:

- The **Working Size** in LightBurn's Device Settings were entered incorrectly, either during manual setup or when modifying the device settings.
- The device's firmware has the wrong values for machine size, which may be an error from the manufacturer.
- If you have multiple device profiles, the device you've selected may not be the one you've connected to.

It's important for your machine to be set to the correct size. If the working size is set to be larger than the machine, the laser head can crash into the edges of its work area. If it's smaller than the actual size, you will be unable to use the full area of the laser.

Resolving the Error Message

The error message provides you with three options.

Update Device Settings

Click this button to automatically adjust the Device Settings to match the device's firmware. Check your device's user manual or contact your machine's manufacturer to verify the correct dimensions and make sure they match the firmware before choosing this option.

Read More

This button brings you to this troubleshooting page for more information, and does not make any changes in LightBurn. You can still make changes manually.

Ignore

Ignore the error for this machine. You won't be prompted again, but the software cannot be held responsible for any machine crashes that may happen.

Correct the Mismatch

If you're certain the **Working Size** value is correct, you most likely either have the wrong device selected or have firmware with incorrect size values.

- If you have multiple device profiles, you may be connecting to a machine other than the one you intended. To check, use the dropdown menus in the Laser Window to switch to the correct profile, or select the correct communications port to connect to a different machine.
- In some rare cases, your machine's firmware may be set incorrectly. If this is the case, you will need to update the size in firmware using Machine Settings. This should be done with caution and only after verifying the correct size in your device's user manual.

Related Topics

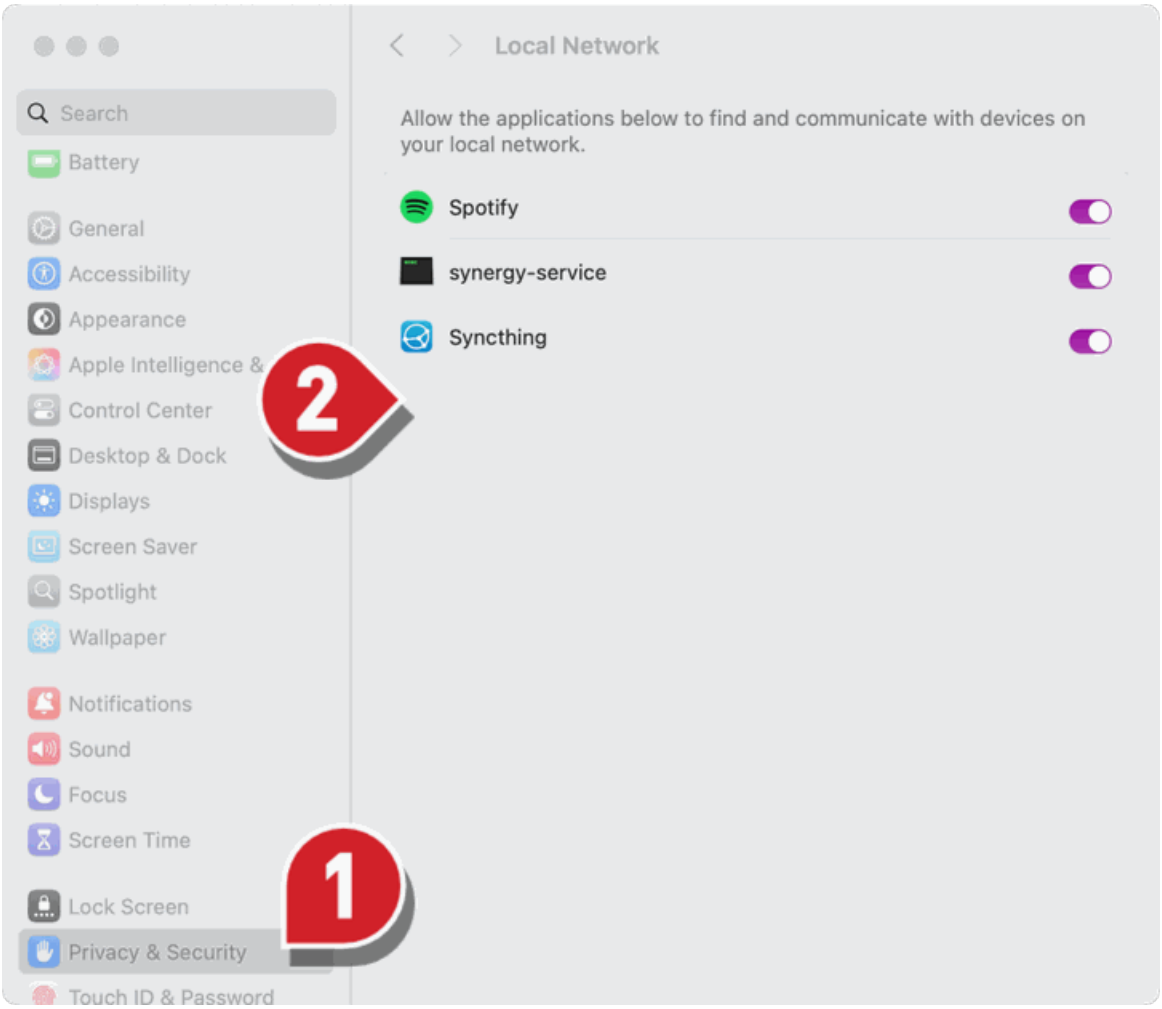
- Device Settings
- Machine Settings

macOS-Specific Problems

Unable to Connect over Ethernet

Starting with macOS Sequoia (macOS 15), connecting to a laser over Ethernet requires granting LightBurn access to the local network. If you're using macOS 15 or higher, you'll need to enable this in your computer's settings. To do this:

1. Go to the **Privacy & Security** section of the settings.
2. Find LightBurn in the list (not pictured in this screenshot) and enable it.



Windows-Specific Problems

Power Management Causes Dropped Connection

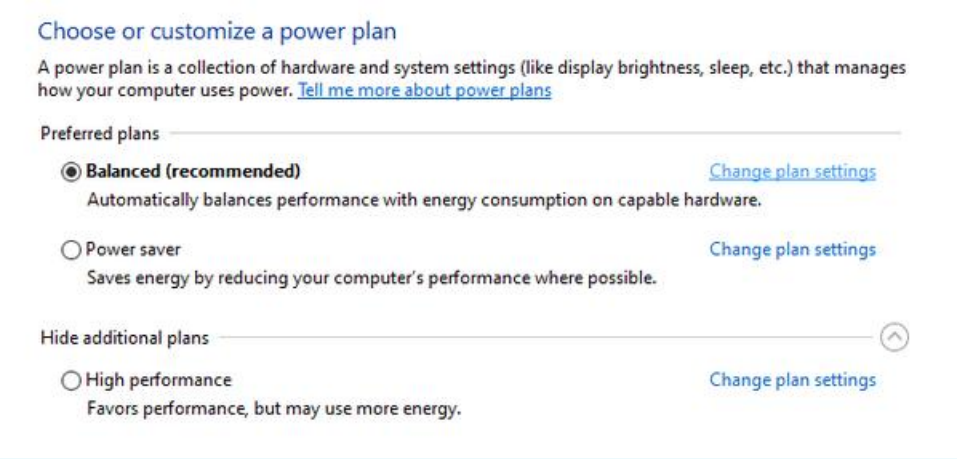
If you're on a Windows computer and can connect to your laser, but tend to lose your connection after a period of time, you may need to change your power management settings. This is most common with laptops, but can happen with desktop computers as well.

When Windows thinks a USB connection is idle, it can put the port into power-save mode. This reduces power usage, but can also cause communication problems for your device. The following directions will help you disable this feature.

Directions for Windows 10

If you're on a laptop, you'll have separate power plans for **On Battery** and **Plugged In**, so make sure to repeat this process for both.

- Open the **Control Panel**
- Click on **Hardware and Sound**
- Click on **Power Options**
- Under "Preferred Plan", click on **Change plan settings** for the active power plan



- Click on **Change advanced power settings**
- Expand **USB settings**
- Expand **USB selective suspend settings** and select **Disabled** from the drop down menu
- Click **Apply** and then **OK** to close the window.

Directions for Windows 7

Part 1

- Open the **Control Panel**
- Find **Power Options**
- Click on **Change Power Plan/Settings**

- Click on **Advanced Power Settings**
- Uncheck both boxes in **USB Settings**
- **Save** settings and close the window

Part 2

- Go to **Device Manager**
- Double click on the first listed **USB Root Hub** and go to the **Power Management** tab
- Uncheck both boxes
- Repeat this process for each USB Root Hub listed in Device Manager

When you're done, restart the computer to make sure these settings are applied

LightBurn Crashes on Windows N Editions

Windows N editions are special editions of Window 8, Windows 10, and Windows 11 that come without Windows Media Player and related features preinstalled. LightBurn may crash on these editions.

To solve the problem, install the Media Feature Pack. Directions from Microsoft are located here.

Mechanical Issues

If there is play/ slop/ backlash somewhere in the machine's mechanics you will see wobbly engraves, closed shapes that don't match up at the end, wiggly bits on corners of lines and a range of other issues.

The most likely culprits are loose belts or debris on the rails, but it could also be an unsecured lens, mirror, toolhead, axis, leadscrew, frame, etc.

- Follow your laser manufacturer's directions for cleaning the rails and any other motion components in your laser.
- Identify the issue by attempting, gently, and with the machine off and unplugged, to wiggle mechanical parts of the machine. See if anything is loose and tighten as necessary. Belts should be taut but not overtightened. They should be able to be plucked like a guitar string. See your device's user manual for more information on maintaining your specific device.
- If you can’t find anything loose it could be that your table is wobbly, and the inertia is carrying into your machine or moving your material. If it is possible to do so, try putting your laser on the ground in order to identify this issue. To correct wobbliness, consider adding cross-bracing or a backer board to stiffen and square up your table.

If it’s still occurring after these checks, you are likely running speeds that are too fast for your machine. It is not always possible to preserve quality when running at or near to your machine's theoretical maximum speed. Try a lower speed.

Reset to Default Settings

Follow these steps to restore LightBurn to the same settings as when you first installed — be aware that you will also lose any device profiles you've created in this process.

> **Warning**
>
> While this process can be helpful in cases where you have a great many incorrect settings and want to start over, we recommend trying Load Prefs Backup or Reset to Default Layout before a full reset.
>
> We also recommend making a User Bundle that can be reimported into LightBurn if needed.

1. Open LightBurn.
2. Go to File → **Preferences** → Open Prefs Folder. You should see a pair of files called `prefs.ini` and `prefs.old` along with a folder called `backup`, along with a few other files.
3. Close LightBurn. If you leave LightBurn open, the settings you're trying to delete will automatically be restored when you eventually do close the program.
4. Delete all the files and folders in the prefs folder. If you leave the `backup` folder, LightBurn will automatically restore the settings you're trying to erase from the backups.
5. Re-open LightBurn. All settings should be reset to default, and all devices will be removed.

That’s it. When you re-run you’ll be asked to set up your laser again, and everything will be as new.

Restoring Lost or Changed Settings

If you've opened up LightBurn and settings and devices are missing or not as you expect, don't panic! We have information to help you get things back to how you expect them.

Upgrading LightBurn

Upgrading your LightBurn installation on the same computer shouldn't affect your LightBurn configuration, but it's still always a good idea to back up your LightBurn configuration with a User Bundle and wait until you finish any critical, time-sensitive projects before upgrading.

Your project files aren't managed by LightBurn and aren't touched by the upgrade process, but consider this your reminder to back them up as well.

> **Installing Release Candidates**
>
> If you want to try a Release Candidate alongside the current live version of LightBurn, don't worry! The Release Candidate will install in a separate directory, so you can run it alongside the current version.

Restoring Lost Settings

Do you have a Release Candidate installed?

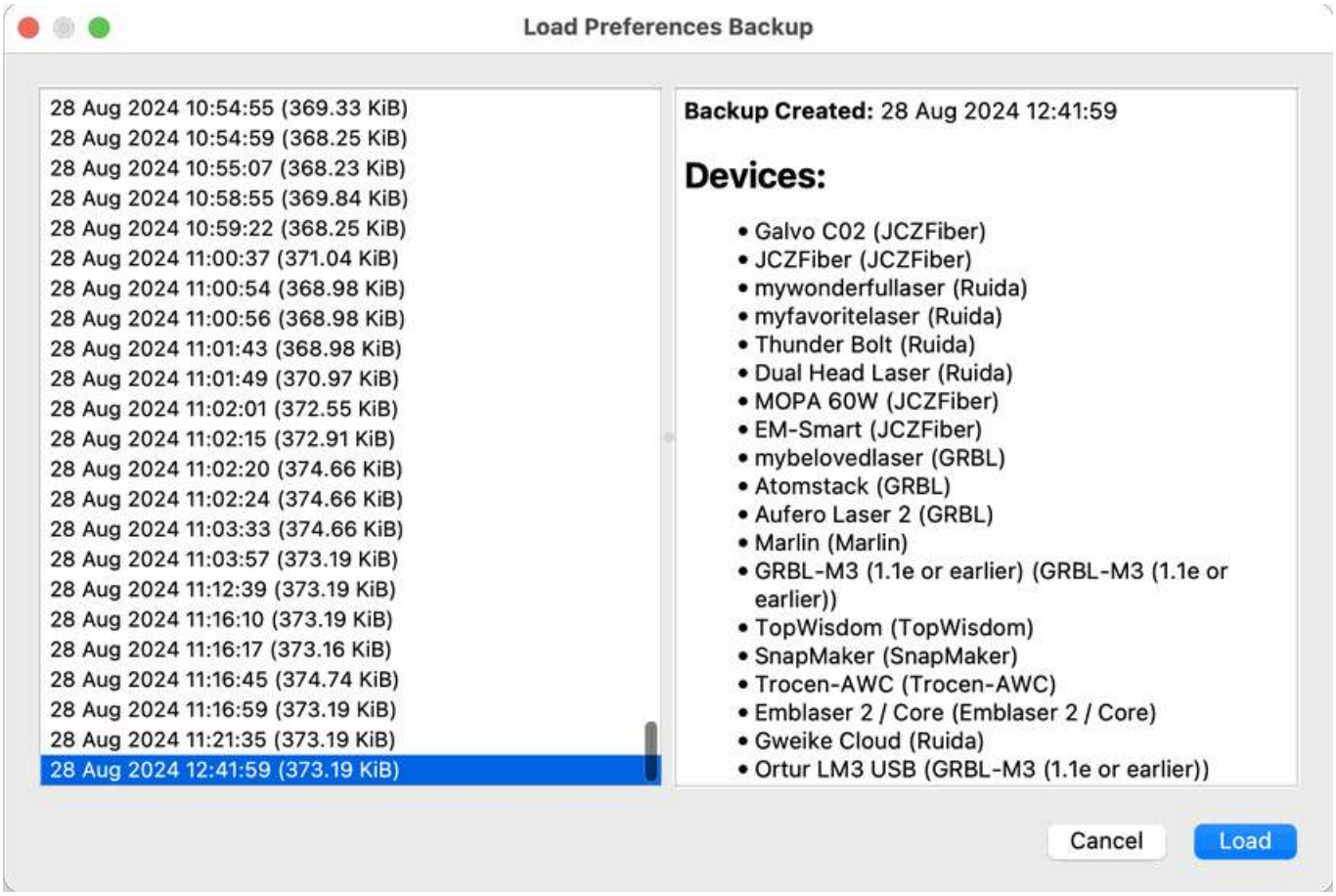
Release Candidates, which act as previews for adventurous users to try new releases ahead of time, install alongside the main release and have seperate preferences. If you have multiple installations, check to make sure the settings you're looking for aren't in the other version.

If you've got a recent User Bundle saved, you can use that to restore devices, settings, libraries, and more.

If you don't already have a User Bundle saved, all is still not lost. LightBurn also creates automatic backups, which you can access by going to **Files → Preferences → Load Prefs Backup**.

The **Load Prefs Backup** command allows you to import preferences from a timestamped list of automatic backups. Click a backup in the list to see the list of Devices available in that backup, on the right side of the window. Use the list of devices and the timestamp showing when the backup was made to find and select the backup you want to use.

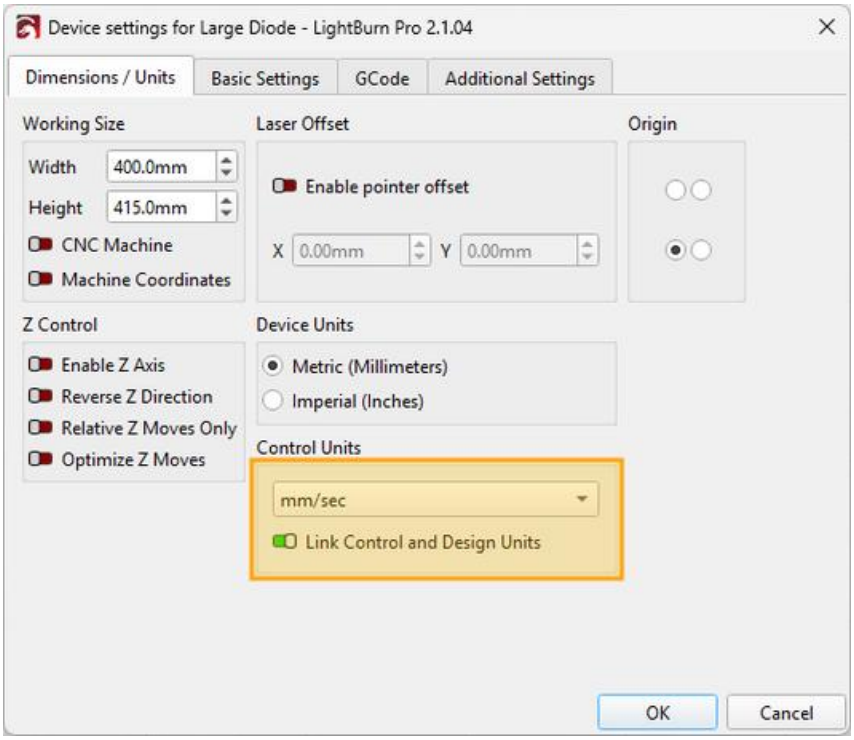
Click **Load** to restore the settings from the selected backup, or **Cancel** to close the window and maintain your current settings.



Cut Settings are Different

Sometimes, we hear from users that their cut settings changed. What's more likely is that your cut settings have stayed the same, but the units they're displayed in have changed.

If this happens to you, check your Control Units and make sure they're set to what you've been using.



Why It Happens

The **Speed** you cut at can display differently if your Control Units accidentally get changed. This happened to some users who used different units for designing and controlling when they installed LightBurn 2.1. If your Control Units change, your cut settings will be displayed in the new units.

As an example, let's say you usually cut plywood at 50 mm/sec. If your Control Units change, this could now display as 1.97 in/sec, or even .032 in/min, but the actual speed hasn't changed. This is easy to miss if you don't change your units often, beause it's easy to get used to thinking of speed as just a number.

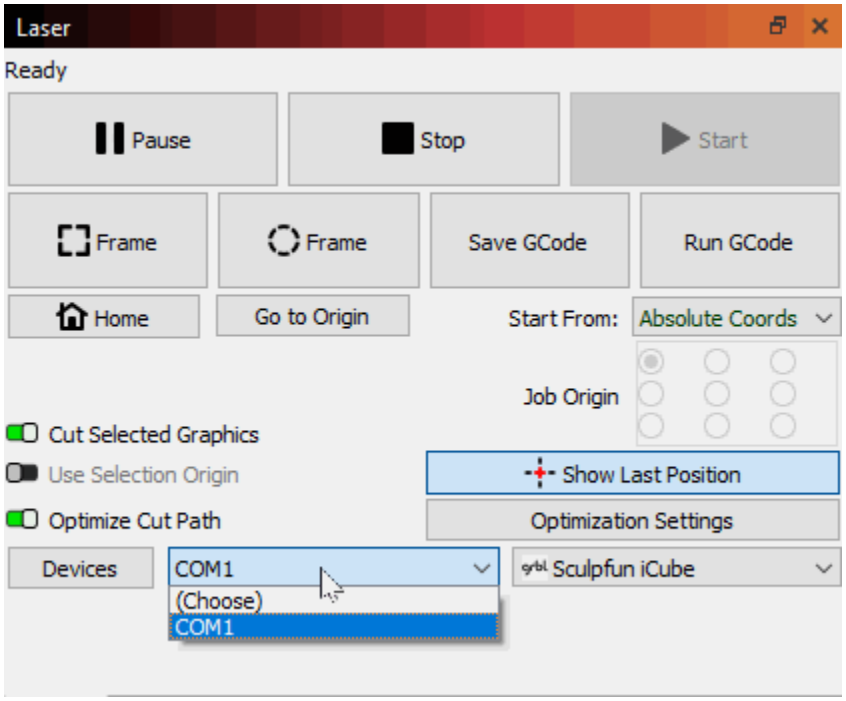
You can read more about this on our blog.

Related

- Reset to Default Settings
- Customizing the LightBurn Window
- Reset Layout to Default

Serial Port Problems

Depending on the type of device you have and the connection method you're using, you may have a dropdown in the **Laser** window to choose the serial port. The screenshot below highlights its location between **Devices** and the name of the active device.



LightBurn will attempt to automatically select the correct serial port, but you may need to manually select it from this dropdown. When connected, the **Disconnected** message in the top left corner of the **Laser** window will change to **Ready**. If you are using a GCode-based laser, you will see a startup message in the Consolewindow.

You may need to right-click the **Devices** button in the Laser window to refresh the list of available ports.

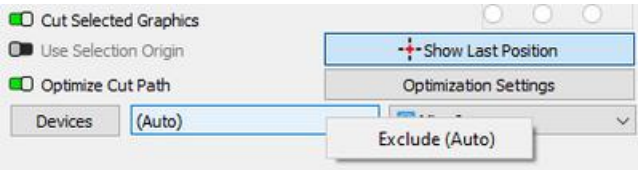
CU.BLTH or cu.bluetooth-incoming-port

Ports with these or similar names are for Bluetooth connections, and are not the correct ports for a USB connection. If you select one of these ports, you may see a **Ready** message in the **Laser** window, but your laser will be unresponsive.

If you see these ports but not a USB connection, you probably need to install the CH340 driver, or refresh the list of available ports.

How To Ignore Unwanted Ports

This will be unnecessary for most people, but if LightBurn is attempting to connect to serial ports for devices other than your laser and it becomes a problem, you can specify ports for LightBurn to ignore.



To exclude a port:

1. Select the port in the dropdown.
2. Right-click the dropdown and choose **Exclude <portname>**.

To include an excluded port:

1. Right-click the dropdown and choose **Show Excluded**.
2. Select the port you want to include and choose **Include <portname>**.

Method Two: `PortExclude.txt`

1. Go to **File** → **Preferences** → **Open Prefs Folder** to open LightBurn's prefs directory.
2. Create a file in this folder called `PortExclude.txt` – use this name exactly, including capitalization.
3. In the file you just created, list the ports you want to exclude, exactly as they show up in the port selection drop down in LightBurn. Each port should be on its own line.
4. Save the file.

LightBurn will now ignore the ports you've told it to filter out.

USB Cables

If you're connecting over USB, cable problems can be a frustrating problem to solve. The steps below may help you identify and solve problems. If you're not sure what type of USB port or connector you have, please compare the shape to the diagram below.



Common USB connector styles used on computers and laser cutters

Check Your USB Cable

- Try upgrading to a higher quality USB cable, especially if you're still using the cable that came with your laser. The shielding in a good USB cable prevents electrical interference with the signals between your laser and your computer. Low cost USB cables, including many of the ones supplied with laser cutters, often have inadequate shielding
- USB cables are limited to 3 meters (16 feet) long. If you need a longer cable, look into active USB extenders or alternate methods of connecting to your laser. For example:
  - For Ruida devices with Ethernet ports, use LightBurn Bridge or Ethernet connections
  - For GRBL devices with WiFi avaible, use our guide to GRBL Network Connections to set up a WiFi connection in LightBurn

USB C Cable Problems

> **More on USB C Cable Problems**
>
> The USB C connectivity of some lasers isn't built to industry standards, which can cause connection issues when trying to connect directly with a USB C to USB C cable. These devices **require** a USB A to USB C cable. If you don't have a USB A port on your computer, you can use a USB A to C adapter/dongle or a USB hub to connect a USB A cable to the computer.

If your device came with a USB A to USB C cable and you're having connection problems with a USB C to C cable, try a USB A to USB C cable instead. This is due to a hardware limitation of some lasers. There's a more detailed explanation of the problem in the info box to the right.

Lasers known to have USB C connectivity problems:

- Sculpfun iCube
- Creality Falcon
- Ortur LM3

Try A Different USB Port

USB ports can occasionally fail, even without physical damage. If your computer has multiple USB ports, try plugging into a different USB port and see if that makes a difference.

Windows: Check Power Management Settings

If you're on a Windows computer and find that you can connect over USB but the connection fails after a while, you may need to adjust your power management settings

Reference

Reference

Our reference pages are here to help you use all of LightBurn's tools and features.

If you know what tool you're looking for, you can search for it or use the navigation list on the left side of the page. If you know what you want to do but not what tool you need, try looking through the categories below.

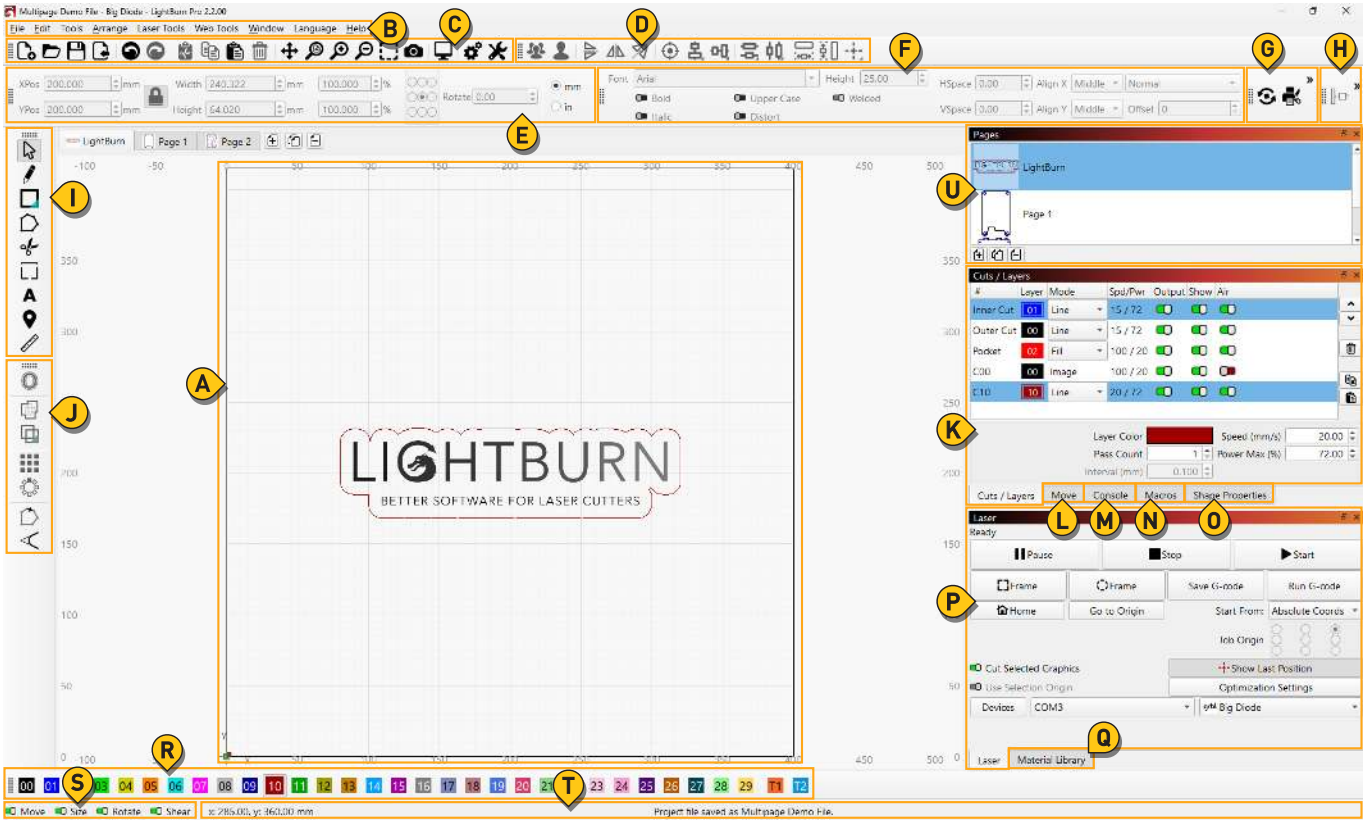
Main Window

Below is the default layout of the full LightBurn window, for a GCode-based device. This layout will be slightly different depending on the type of laser you're working with.

For information on rearranging the layout, see Customizing the LightBurn Window.

For a full list of available windows and toolbars, see Window Menu.

Click any UI element in the image below to go directly to the relevant page for that element, or scroll down for a list of elements.



A: Workspace / Edit Window

B: Menus

C: Main Toolbar

D: Arrange Toolbar

E: Numeric Edits Toolbar

F: Text Options Toolbar

G: Modes Toolbar

H: Docking Toolbar

I: Creation Toolbar

J: Modifiers Toolbar

K: Cuts / Layers Window

L: Move Window

M: Console Window

N: Macros Window

O: Shape Properties Window

P: Laser Window

Q: Material Library Window

R: Color Palette

S: Transform Control Toggles

T: Status Bar

U: Pages

Workspace / Edit Window

The area where you create, adjust, modify, and arrange artwork in LightBurn.

See Workspace for more information.

Menus

Contains all of LightBurn's top-level menus. Individual menus contain a wide range of tools and features.

See Menu Toolbar for a full list of top-level menus.

Main Toolbar

Contains functions for opening or importing files, saving, using the clipboard (copy and paste), moving or zooming the view.

See Main Toolbar for more information.

Arrange Toolbar

Contains tools for positioning objects, both on an individual basis, and in relation to each other.

See Arrange Toolbar for more information.

Numeric Edits Toolbar

Contains options for adjusting the size, position, and orientation of shapes or groups of shapes in your project.

See Numeric Edits Toolbar for more information.

Text Options Toolbar

Contains options for adjusting attributes of Text Objects, including, **Font**, size (**Height**), spacing (**HSpace** and **VSpace**), and more.

See Create and Edit Text — Text Options Toolbar for more information.

Modes Toolbar

Contains buttons to open dialog windows or wizards for advanced lasering processes.

See Modes Toolbar for more information.

Docking Toolbar

Contains tools to move selected objects left, right, up, or down, until they meet the edge of the Workspace or another object.

See Docking for more information.

Creation Toolbar

Contains tools for selecting objects in your Workspace, or creating Primary Shapes, text, lines, and more.

See Creation Toolbar for a full list of tools.

Modifiers Toolbar

Contains tools for combining, altering, and producing arrayed duplicates of graphics.

See Modifiers Toolbar for a full list of tools.

Cuts / Layers Window

Shows the layers currently assigned to graphics in your design, and lets you alter common Cut Settings, or double-click to access additional settings.

See Cuts / Layers Window for more information.

Move Window

Provides buttons to jog and position your laser, along with additional options for users with GCode-based lasers.

See Move Window for more information.

Console Window

> **Note**
>
> Available for GCode-based lasers only.

Displays messages from your laser's controller and commands sent to it by LightBurn. You can also input direct commands to your laser through this window.

See Console Window for more information.

Macros Window

Allows you to create buttons to store frequently used commands for GCode-based lasers. These functions will only work when LightBurn is directly connected to a GCode-based laser.

See Macros Window for more information.

Shape Properties Window

Displays editable properties of selected objects.

See Shape Properties Window for more information.

Laser Window

The hub for setting up and selecting a laser, Framing and Starting jobs, determining job positioning, and more.

See Laser Window for more information.

Material Library

Provides a place to store, organize, and apply preset Cut Settings for different operations and materials.

See Material Library for more information.

Color Palette

Contains the colors/layers that you can assign to individual graphics in LightBurn. Use these colors/layers to assign different kinds of operations and settings to the shapes in your design.

See Color Palette for more information.

Transform Control Toggles

Switches that enable and disable specific types of editing across an entire project, in order to prevent accidental edits, such as inadvertently scaling a design that contains strict sizing, or moving tightly nested parts.

See Transform Control Toggles for more information.

Status Bar

Provides contextual information related to the active tool or process being performed.

See Status Bar for more information.

Pages

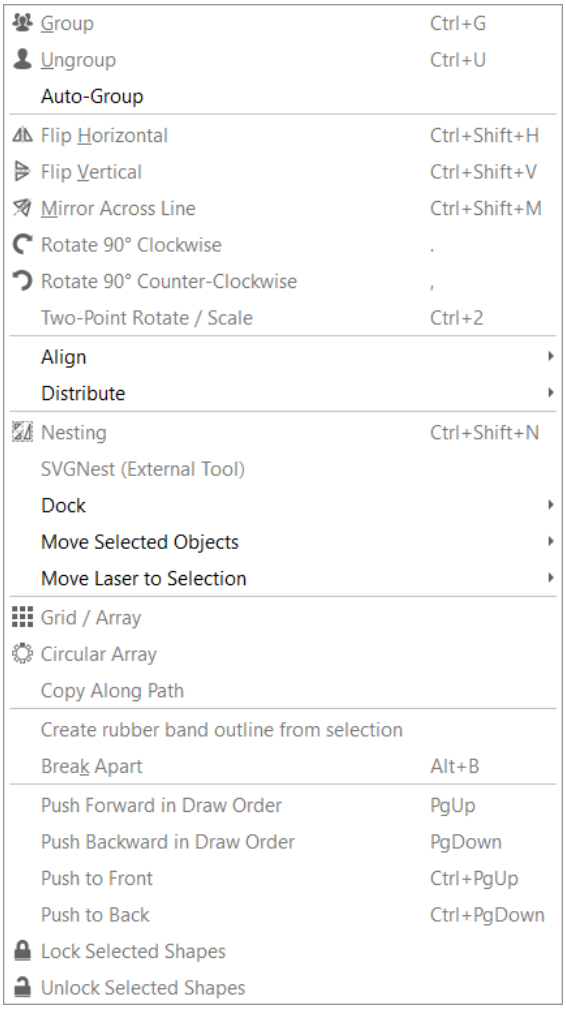
Create, copy, remove, and preview pages for multisheet designs.

See Pages for more information.

Arrange Menu

The **Arrange Menu** contains tools for positioning objects, both on an individual basis, and in relation to each other. It also contains tools for moving the physical laser in relation to objects within your project.

Click any option in the image below to go directly to the relevant page for that tool, or scroll down for a list of options and descriptions.



Group

`Ctrl` / `Cmd` + `G`

Combines multiple objects into a single entity, allowing you to move, resize, and manipulate them as though they were one object.

See Grouping for more information.

Ungroup

`Ctrl` / `Cmd` + `U`

Splits **Groups** back into the individual objects that make them up.

See Grouping for more information.

Auto-Group

Groups objects that fall within the boundaries of a closed shape together with the outermost shape.

See Auto-Group for more information.

Flip Horizontal / Vertical

- **Flip Horizontal** — Mirrors objects along the X Axis.
  `Shift` + `Ctrl` / `Cmd` + `H`
- **Flip Vertical** — Mirrors objects along the Y Axis.
  `Shift` + `Ctrl` / `Cmd` + `V`

See Mirror Tools for more information.

Mirror Across Line

`Shift` + `Ctrl` / `Cmd` + `M`

Flips objects along a selected line.

See Mirror Tools — Mirror Across Line for more information.

Rotate 90° Clockwise / Counter-Clockwise

- **Rotate 90° Clockwise** — Rotates the object to the right by 90°.
  `.`
- **Rotate 90° Counter-Clockwise** — Rotates the object to the left (anti-clockwise) by 90°.
  `,`

See Transform Controls — Rotate for more information.

Two-Point Rotate / Scale

`Ctrl` / `Cmd` + `2`

Rotates and resizes objects around custom pivot points in your Workspace.

See Two-Point Rotate / Scale for more information.

Align

Repositions objects according to their edges or centers.

See Align for more information.

Distribute

Evenly spaces objects either vertically or horizontally, spaced from their centers or edges.

See Distribute for more information.

Distribute → Move Together

Arranges objects so that their edges abut along horizontal or vertical planes.

See Distribute — Move Together for more information.

Nesting

Opens the **Nesting** dialog to use True Nest and Quick Nest.

See Nesting for more information.

SVGNest (External Tool)

Arranges objects to minimize material waste using an online service.

See SVGNest (External Tool) for more information.

Dock

Moves selected objects left, right, up, or down, until they meet the edge of the Workspace or another object.

See Docking for more information.

Move Selected Objects

Moves objects to a corner, midpoint, or center of your Workspace. If you have a Gantry style laser, it can also move selected objects to the last reported position of your laser head.

See Move Selected Objects for more information.

Move Laser to Selection

Moves your laser in its physical work area to a location that corresponds to a position on your current selection in your LightBurn Workspace.

See Move Laser to Selection for more information.

Move Laser To Selection → Jog Laser

Moves the laser by the amount specified in the Move Window, in a chosen direction.

See Move Window — Jogging for more information.

Grid Array

Creates copies of an object (or objects) in regularly spaced rows and columns, and includes options to adjust spacing, mirror the shapes, randomize orientation, and more.

See Grid Array for more information.

Circular Array

Arranges multiple copies of an object — or multiple objects — in a radial pattern around either a point in space, or a central object.

See Circular Array for more information.

Copy Along Path

Creates duplicates of an object that follow the contour of a path.

See Copy Along Path for more information.

Create Rubber-Band Outline from Selection

Makes a new shape outlining the graphics in your current selection. The new shape is defined as the shortest possible path that can contain all graphics in your selection, as if a rubber band were stretched around them.

See Create Rubber-Band Outline for more information.

Break Apart

`Alt` / `Option` + `B`

Separates paths into their constituent lines and curves.

See Break Apart for more information.

Push in Draw Order

Changes the order of objects in the draw order, which affects the visual appearance of graphics in your Workspace, when using a Wireframe View Style.

See Push in Draw Order for more information.

Lock Selected Shapes / Unlock Selected Shapes

- **Lock Selected Shapes** prevents editing of selected objects.
- **Unlock** allows them to be edited once again.

See Lock Shapes for more information.

Arrange Toolbars

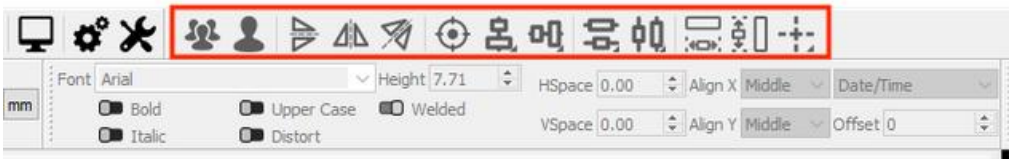
The **Arrange Toolbars** each contain a selection of tools found in the Arrange Menu, that are broadly used for positioning objects.

The toolbars come in a long version, for high-resolution screens, and a shorter one, with sub-menus to fit on smaller screens.



Accessing the Arrange Toolbars

The **Arrange Toolbar** is located near the center of the top toolbar in LightBurn by default, to the right of the Main Toolbar.



If you have closed the Arrange Toolbar, go to Window → **Arrange** to re-enable it.

To restore it and all other toolbars and windows to their default positions, go to **Window** → Reset to Default Layout.

To enable the **Arrange (long) Toolbar**, go to **Window** → **Arrange (long)**.

> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Arrange Toolbar Options

Click any button in the images below to jump to information on that tool, or scroll down for a list of options and descriptions.

Group and Ungroup 

Combines multiple objects into a single entity, allowing you to move, resize, and manipulate them as though they were one object.

See Grouping for more information.

Flip or Mirror 

Flips objects across their vertical or horizontal centers, or a custom-selected line.

See Mirror Tools for more information.

Align 

Repositions multiple objects in relation to the last object selected, according to their edges or centers.

See Align for more information.

Make Same Width or Height 

Resizes the selected object to match the width or height of the last object selected.

See Make Same Width or Height for more information.

Distribute 

Evenly spaces objects either vertically or horizontally, spaced from their centers or edges.

See Distribute for more information.

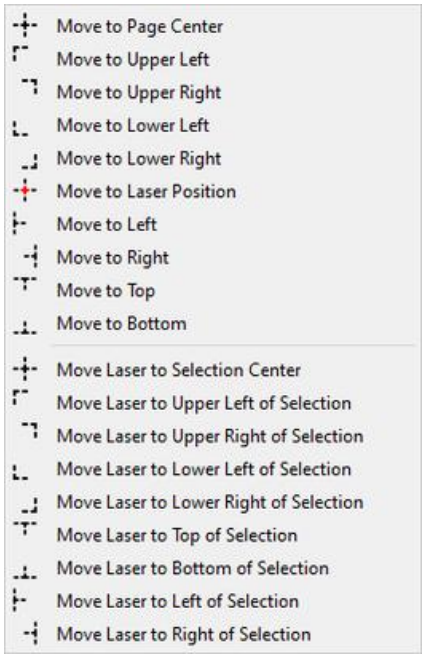
Move Selected Objects 

Moves objects to a corner, midpoint, or center of your Workspace. If you have a Gantry style laser, it can also move selected objects to the last reported position of your laser head.

See Move Selected Objects for more information.

Move Selected Objects / Move Laser to Selection Dropdown 

This button opens a dropdown menu containing all the Move Selected Objects and Move Laser to Selection options.


Move to Page Center
Move to Upper Left
Move to Upper Right
Move to Lower Left
Move to Lower Right
Move to Laser Position
Move to Left
Move to Right
Move to Top
Move to Bottom
Move Laser to Selection Center
Move Laser to Upper Left of Selection
Move Laser to Upper Right of Selection
Move Laser to Lower Left of Selection
Move Laser to Lower Right of Selection
Move Laser to Top of Selection
Move Laser to Bottom of Selection
Move Laser to Left of Selection
Move Laser to Right of Selection

Color Palette

In LightBurn, different colors indicate different layers. These colors don't represent the final product's colors but instead differentiate each operation needed to complete the job.

Use the **Color Palette** to assign colors/layers to graphics in your Workspace. A common convention is to use bright red for cuts and black for engravings, but how you use the colors/layers is up to you.

After assigning colors to your graphics, you can adjust their layer settings in two places:

- Cuts / Layers Window — where you can re-order layers and change critical settings.
- Cuts Settings Editor — which contains the full suite of settings available for adjustment.

Accessing the Color Palette



The Color Palette is located along the bottom of the Main Window by default, though a common alternative is docking it along the left side of the Workspace.

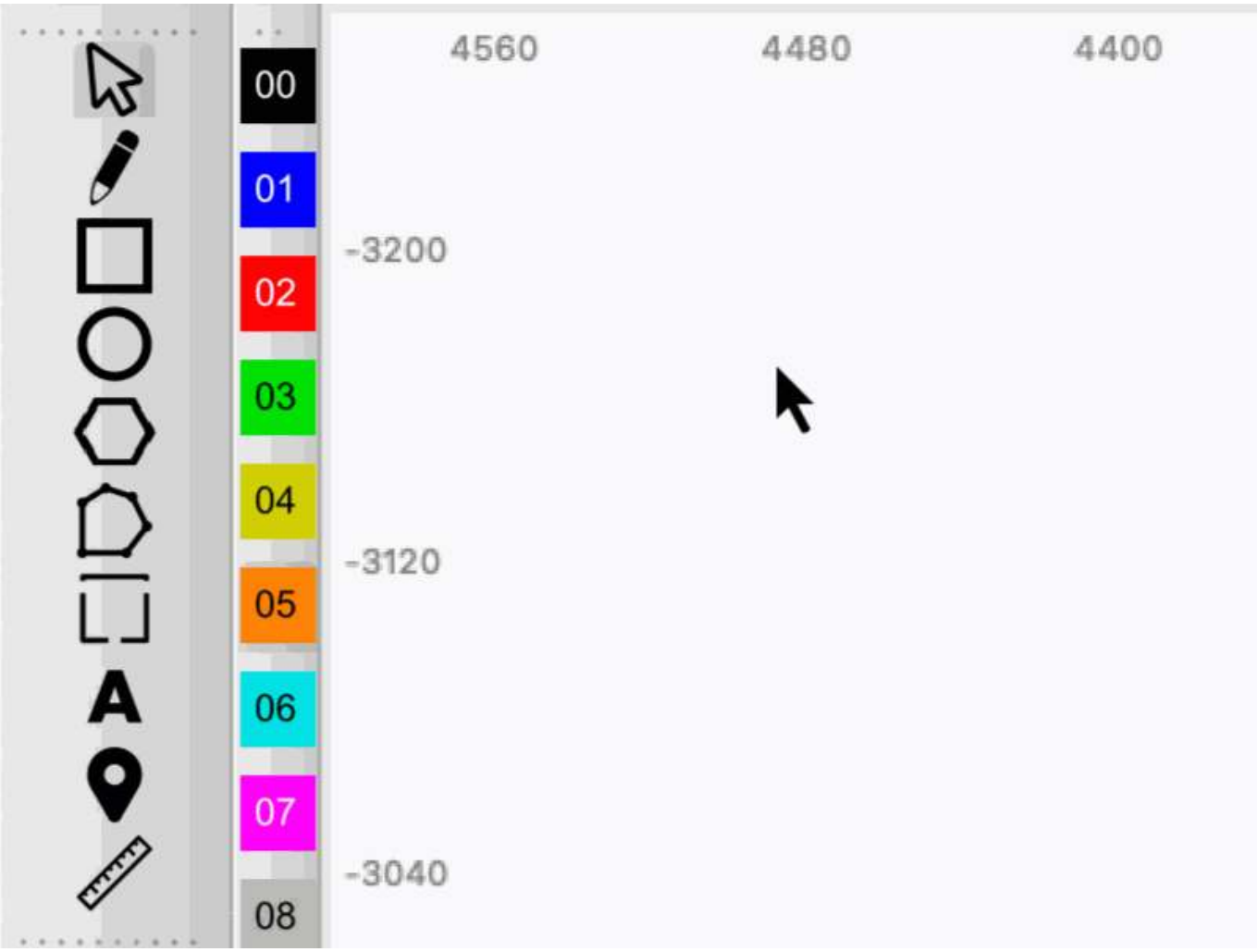
The colors currently used in your design will also appear as entries in the Cuts / Layers Window, where you can choose the operations that each color represents.

If you have closed the Color Palette, go to Window → **Color Palette** to re-enable it.

To restore it and all other toolbars and windows to their default positions, go to **Window** → Reset to Default Layout.

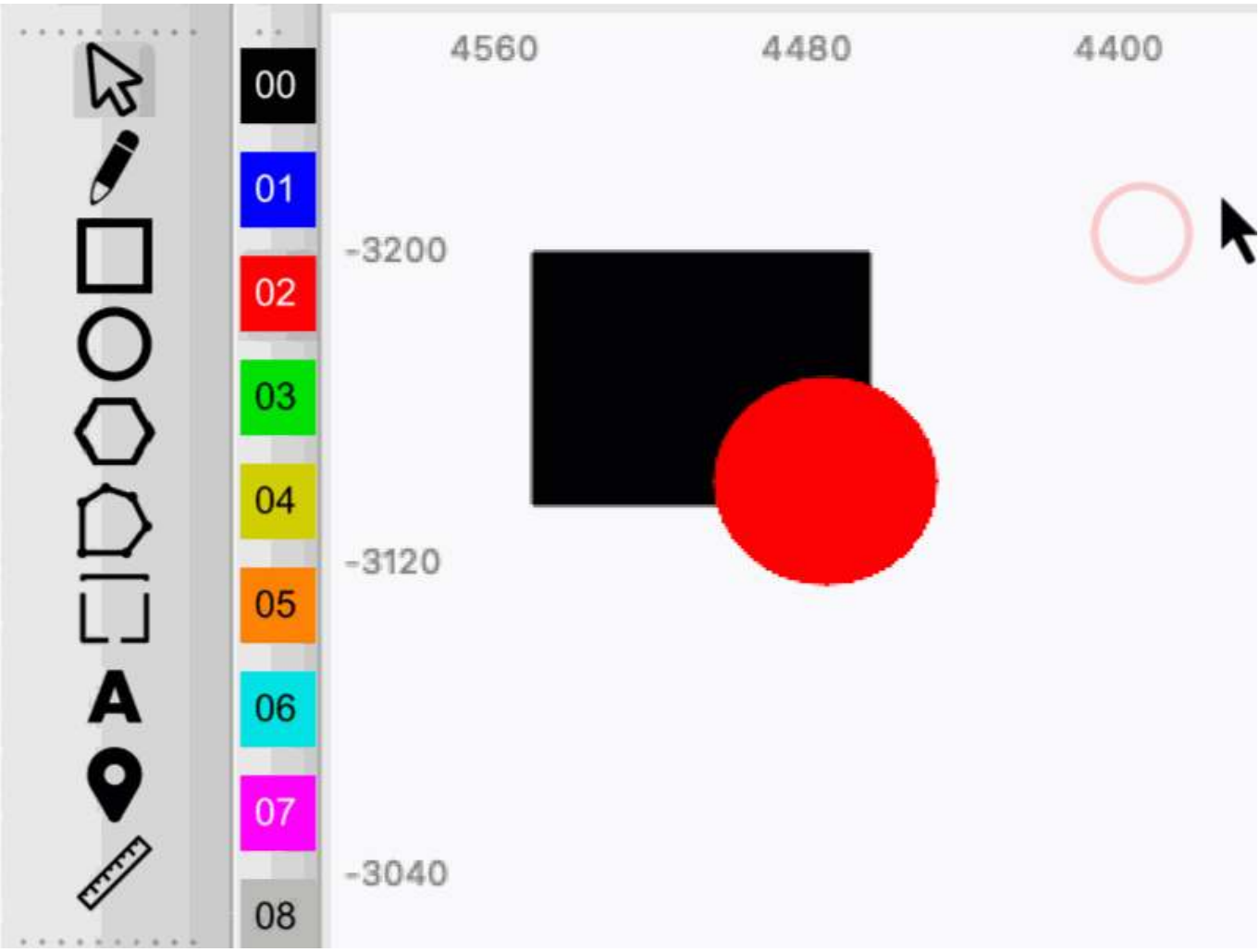
> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Change the Color for New Objects



With nothing selected in your Workspace, click a color entry. Subsequent new shapes will be created in that color, and assigned to the corresponding layer in the Cuts / Layers Window.

Change the Color of Existing Objects



With the objects already selected, click a color entry to switch them to that color, and assign them to the corresponding layer in the Cuts / Layers Window.

Tool Layers

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=FgCRrQC16xs

There are two colors/layers at the end of the palette labeled **T1** and **T2**. These refer to **Tool** layers — special layers that have no cut parameters and will never be output to the laser.

Use Tool layers to create non-output shapes in your designs. For example you can use them to:

- Describe the boundary of your material
- Create guidelines for aligning shapes
- Apply a Path to Text without the path being output
- Apply a Mask to an Image

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=tnISpDQkq74

Importing Graphics from Other Software

When importing graphics, LightBurn will attempt to match the colors used in that file to the default LightBurn **Color Palette**. Use exact matches for the colors used by LightBurn to ensure layers are assigned properly.

Downloadable Color Palettes

We have color palettes available for Inkscape, CorelDRAW, Affinity Designer, and Adobe Illustrator. The `.ase` file provided for Adobe and Affinity products can be imported into many other programs as well.

Download .gpl for Inkscape/GIMP

Download .xml for Corel Products

Download .ase for Adobe and Affinity

Please see your preferred program's documentation for directions on installing color palettes.

RGB and Hex Codes for the LightBurn Color Palette

The table below contains RGB and hex values for each of LightBurn's layer colors. You can use it to assign the right color for any software that doesn't support the above files.

Layer	RGB Values	Hex Values
00	rgb(0, 0, 0)	#000000
01	rgb(0, 0, 255)	#0000FF
02	rgb(255, 0, 0)	#FF0000
03	rgb(0, 224, 0)	#00E000
04	rgb(208, 208, 0)	#D0D000
05	rgb(255, 128, 0)	#FF8000
06	rgb(0, 224, 224)	#00E0E0
07	rgb(255, 0, 255)	#FF00FF
08	rgb(180, 180, 180)	#B4B4B4
09	rgb(0, 0, 160)	#0000A0
10	rgb(160, 0, 0)	#A00000
11	rgb(0, 160, 0)	#00A000
12	rgb(160, 160, 0)	#A0A000
13	rgb(192, 128, 0)	#C08000

14	rgb(0, 160, 255)	#00A0FF
15	rgb(160, 0, 160)	#A000A0
16	rgb(128, 128, 128)	#808080
17	rgb(125, 135, 185)	#7D87B9
18	rgb(187, 119, 132)	#BB7784
19	rgb(74, 111, 227)	#4A6FE3
20	rgb(211, 63, 106)	#D33F6A
21	rgb(140, 215, 140)	#8CD78C
22	rgb(240, 185, 141)	#F0B98D
23	rgb(246, 196, 225)	#F6C4E1
24	rgb(250, 158, 212)	#FA9ED4
25	rgb(80, 10, 120)	#500A78
26	rgb(180, 90, 0)	#B45A00
27	rgb(0, 71, 84)	#004754
28	rgb(134, 250, 136)	#86FA88
29	rgb(255, 219, 102)	#FFDB66
T1	rgb(243, 105, 38)	#F36926
T2	rgb(12,150, 217)	#0C96D9

Related Topics

- Cuts Settings Editor
- Cuts / Layers Window

Creation Toolbar

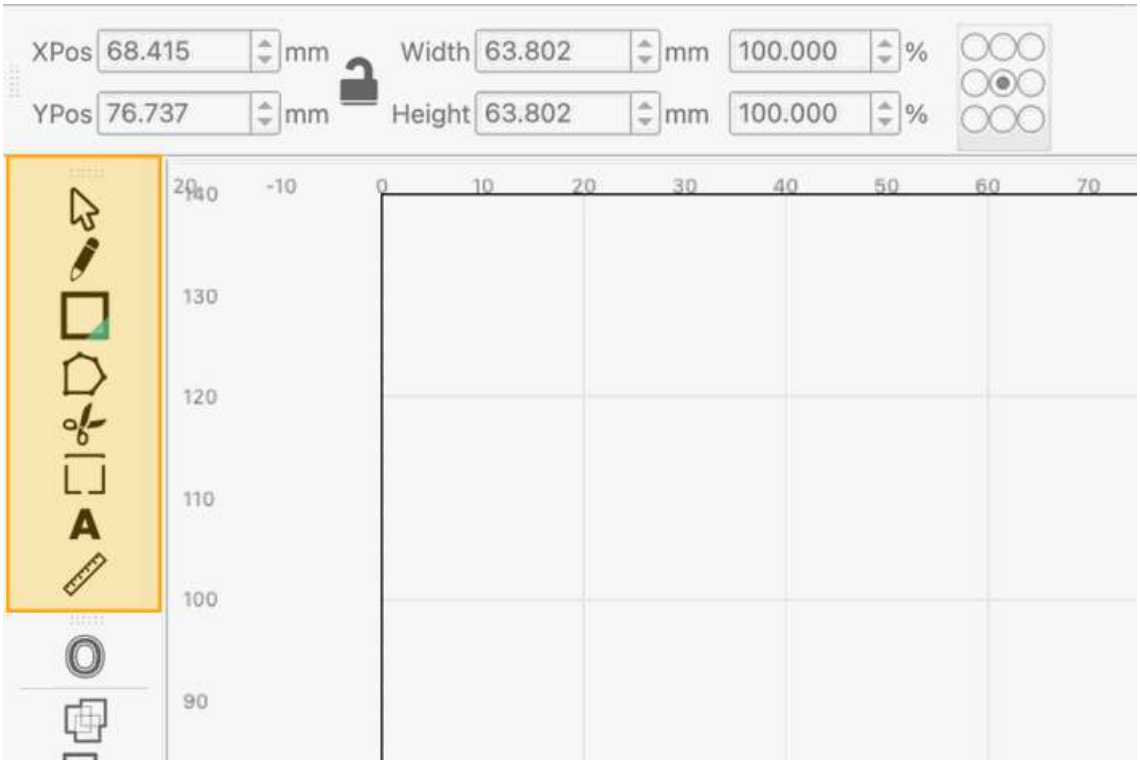
The **Creation Toolbar** contains tools for selecting, creating, and editing objects.

Accessing the Creation Toolbar

By default, the **Creation Toolbar** is the upper toolbar on the left-hand side of the screen.

If you have closed the Creation Toolbar, select Window → **Creation Tools** to re-enable it.

To restore it and all other toolbars and windows to their default positions, go to **Window** → Reset to Default Layout.



Tip

The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Creation Toolbar Options

Click any button in the image below to jump to information on that tool, or scroll down for a list of options and descriptions.



Tip

> The Creation Toolbar is normally vertical, but can also be removed from the left-side toolbar and oriented horizontally, as presented above. See Customizing the LightBurn Window for more information.

Select

 `Esc`

Objects in LightBurn must be selected before they can be modified. With the **Select** tool active, you can click on objects to select them, or click, hold, and drag to select objects contained or crossed by a selection box.

See Selection for more information.

Draw Lines

 `Ctrl`/`Cmd`+`L`

Creates vector paths out of straight and curved lines. With the tool active, click to place points.

See Draw Lines for more information.

Draw Shapes



Several **Primary Shapes** are available in the **Draw Shapes** submenu of the **Creation Toolbar**.

Click and hold the Draw Shapes button, or click the arrow in the lower right corner, to select a shape from the menu, then click and drag in your Workspace to create that shape.

See Draw Shapes for more information.

Edit Nodes

 `Ctrl`/`Cmd`+`` ` ``

Use the **Edit Nodes** tool to edit the nodes, or points, that make up vector paths. This only works on Ungrouped vector paths.

See Edit Nodes for more information.

Trim Shapes

 `Ctrl`/`Cmd`+`K`

Use the **Trim Shapes** tool to remove sections of vector graphics that lie between intersections.

See Trim Shapes for more information.

Add Tabs

 Ctrl/Cmd+Tab

Use **Add Tabs** to add tabs – small sections of material deliberately left uncut – to your project. Tabs prevent cut objects from immediately falling out of the base material.

See Add Tabs for more information.

Create / Edit Text

 Ctrl/Cmd+T

Use **Create / Edit Text** to create or change Text Objects in your design.

See Create and Edit Text for more information.

Position Laser

 Alt/Option+L

With **Position Laser** active, click the Workspace to move the laser to the corresponding location in its physical work area. Make sure nothing is in the path of your laser before using this tool.

See Position Laser for more information.

Measure

 Alt/Option+M

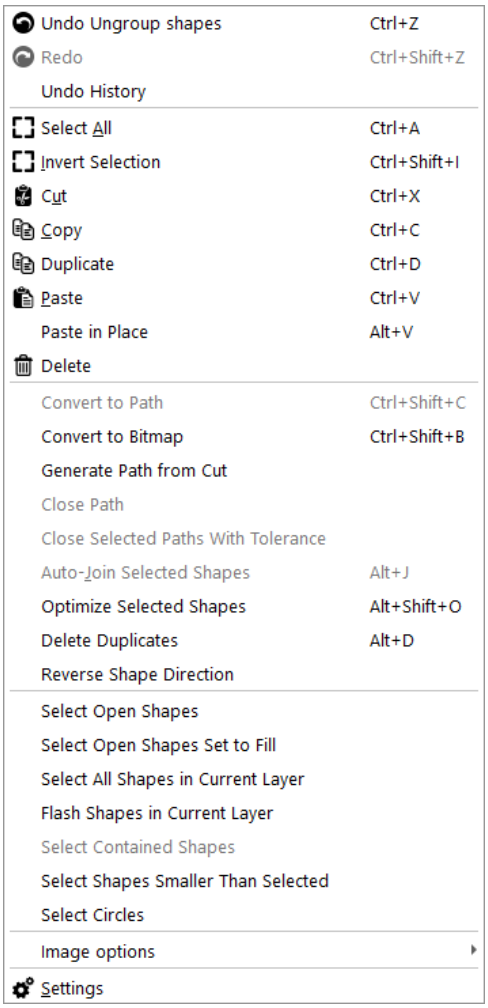
With the **Measure** tool active, hover over an object to see measurements and node information, or click and drag to measure the distance between any two points.

See Measure for more information.

Edit Menu

The **Edit Menu** contains tools for selecting and manipulating objects in addition to LightBurn settings.

Click any option in the image below to go directly to the relevant page for that tool, or scroll down for a list of options and descriptions.



Undo

`Ctrl` / `Cmd` + `Z`

Reverses previous actions taken, starting with the most recent action taken, and proceeding to previous actions with successive Undos.

See Undo/Redo for more information.

Redo

`Ctrl` / `Cmd` + `Shift` + `Z`

Restores actions previously reversed using Undo, starting with the most recent action Undone, and proceeding to previous reversed actions with successive **Redos**.

See Undo/Redo for more information.

Select All

`Ctrl`/`Cmd`+`A`

Selects all objects in your Workspace.

See Selection Tools for more information.

Invert Selection

`Ctrl`/`Cmd`+`Shift`+`I`

Deselects all objects that are currently selected and selects all objects that are not currently selected.

See Selection Tools for more information.

Cut

`Ctrl`/`Cmd`+`X`

Deletes selected objects while copying them to clipboard at the same time.

See Clipboard Tools for more information.

Copy

`Ctrl`/`Cmd`+`C`

Saves a copy of selected objects to your computer's clipboard (temporary storage location).

See Clipboard Tools for more information.

Duplicate

`Ctrl`/`Cmd`+`D`

Create duplicates of selected objects directly on top of the existing objects.

See Clipboard Tools for more information.

Paste

`Ctrl`/`Cmd`+`V`

Inserts objects that are copied to your clipboard (temporary storage location) into your. Pasted objects are centered at the location of the cursor at the time of pasting.

See Clipboard Tools for more information.

Paste in Place

Inserts objects copied to your clipboard (temporary storage location) into your Workspace, and positions them according to their location in the project they were copied from.

See Clipboard Tools for more information.

Delete

`Del`

Deletes selected objects, removing the from your Workspace.

See Clipboard Tools for more information.

Convert to Path

`Ctrl`/`Cmd`+`Shift`+`C`

Converts squares, ellipses, polygons, text, or QR codes created using LightBurn's Creation Tools into editable paths made up of lines, curves, and nodes.

See Convert to Path for more information.

Convert to Bitmap

`Ctrl`/`Cmd`+`Shift`+`C`

Converts selected vector graphics into a bitmap image.

See Convert to Bitmap for more information.

Generate Path From Cut

Generates new paths from your selected objects that include properties added through the Cut Settings Editor, like tabs, kerf offsets, or perforation settings.

See Generate Path From Cut for more information.

Close Path

Connects a shape with one start and one end node by creating a new line between them. The existing start and end nodes must be within .5 mm of one another.

See Close Path for more information.

Close Selected Paths With Tolerance

Opens a dialog window to close selected paths with one start and one end node within a specified distance apart from one another, and to move their ends together or join them with lines.

See Close Selected Paths with Tolerance for more information.

Auto-Join Selected Shapes

`Alt`/`Option`+`J`

Joins many disconnected line segments together at once, if the segments' end points are almost touching (within .05 mm of one another).

See Auto-Join Selected Shapes for more information.

Optimize Selected Shapes

`Alt`/`Option`+`Shift`+`O`

Smooths selected shapes or fits them to arcs or lines. The dialog box allows you to set your tolerance for how much deviation from the original shape is acceptable.

See Optimize Selected Shapes for more information.

Delete Duplicates

`Alt`/`Option`+`D`

Deletes duplicated objects (objects with identical shapes, sizes, and positions) in your Workspace.

See Delete Duplicates for more information.

Reverse Shape Direction

Reverses the direction the path is drawn from the start point. This determines the direction the laser will move in when cutting lines.

See Set Start Point for more information.

Select Open Shapes

Selects all shapes in the Workspace that are open, meaning they are not paths that form a closed, continuous loop but, instead, have start and end points that are different from one another.

See Selection Tools for more information.

Select Open Shapes Set to Fill

Selects all shapes assigned to a layer set to Fill or Offset Fill mode that are open, meaning they are not closed, continuous loops but, instead, have start and end points that are different from one another.

See Selection Tools for more information.

Select All Shapes in Current Layer

Selects all shapes in the active layer in the Cuts / Layers window.

See Selection Tools for more information.

Select Contained Shapes

Selects all shapes contained by the currently selected shape. The currently selected shape remains in the resulting selection.

See Selection Tools for more information.

Select Shapes Smaller Than Selected

Selects all shapes that are smaller than the currently selected shape. Currently selected shape remains in the resulting selection.

See Selection Tools for more information.

Image Options

Provides a set of tools for updating and changing imported images.

See Image Options for more information.

Refresh Image

Reloads the selected image file. Useful if an imported image file has been updated.

Replace Image

Replaces the selected image with a different image file.

Replace Image to Fit

Replaces the selected image with a different image file, adjusting the size of the new image to fit in the same space as the original image.

Settings

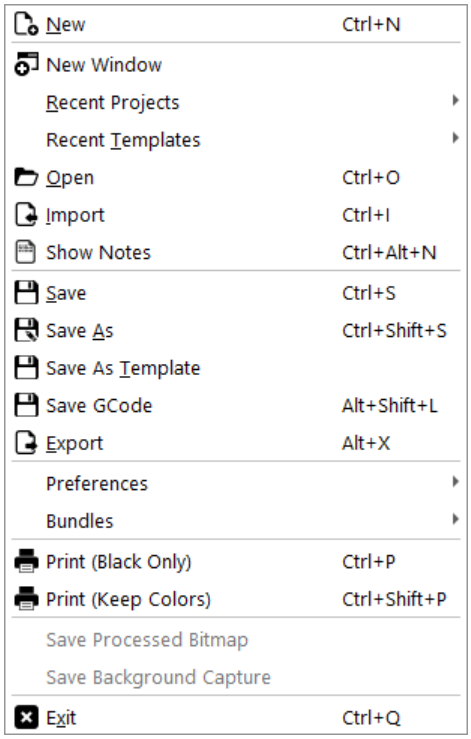
View and edit general, device-independent LightBurn settings.

See Settings/Preferences for more information.

File Menu

The **File Menu** contains tools for creating, opening, and saving projects; importing designs and images; and changing, saving, exporting, or importing your LightBurn preferences.

Click any option in the image below to go directly to the relevant page for that tool, or scroll down for a list of options and descriptions.



New

`Ctrl` / `Cmd` + `N`

Clears your current LightBurn project and starts a new, blank project.

See File Management — New for more information.

New Window

Opens a separate instance of LightBurn in a new window, allowing you to edit multiple projects at once, or edit one file while cutting a different one.

See New Window for more information.

> **Note**
>
> LightBurn will automatically save the current preferences and settings of the most recently closed window — overriding any changes made in other windows.

> See Multiple Independent LightBurn Instances for more information on running two completely independent LightBurn instances.

Recent Projects

View a running list of up to 24 projects recently opened in LightBurn.

See File Management — Recent Projects for more information.

Open

`Ctrl` / `Cmd`+`O`

Opens LightBurn project files saved in .lbrn or .lbrn2 format.

Click on the dropdown in your system's file browser to show all supported file types, including supported image and vector file types.

> **Warning**
>
> Selecting a supported image or vector file instead of a project file will create a new LightBurn project, replacing the currently loaded project. Use Import to bring artwork files or designs into an existing project.

See File Management — Open for more information.

Import

`Ctrl` / `Cmd`+`I`

Adds additional artwork, designs, and objects stored in a range of file formats to an existing, open project.

File Type	Files
LightBurn Project Files	.lbrn2, .lbrn
MillMage Project Files	.mage
Image Files	.bmp, .jpg, .pg, .gif, .tif, .tiff, .tga, .webp
Vector Files	.ai, .pdf, .dxf, .nc, .hpgl, .svg
Additional Files	.gc, .gcode, .plt, .rd, .sc, .scpro2

See File Management — Import for more information.

Show Notes

`Ctrl` / `Cmd` + `Alt`/`Option`+`N`

Store and view project-specific notes and comments.

See Show Notes for more information.

Save

`Ctrl`+`S`

Saves the current project for later editing and production. Saving a file saves it under the current file name. Saving a file without an existing name will open a Save Project window.

See File Management — Save for more information.

Save As

`Ctrl` / `Cmd`+`Shift`+`S`

Saves the current project under a different file name.

See File Management — Save As for more information.

Save Machine Files

`Alt`/`Option`+`Shift`+`L`

This option will change depending on the type of laser you're using — it may say Save GCode (for GRBL, Smoothieware, or Marlin controllers), Save RD File (Ruida), Save OUT File (TopWisdom), or Save UD5 File (Trocen).

All options save your current project as a machine-ready file, in the appropriate format for your type of laser. You can transfer this file to your machine to run it without ever having to directly connect using LightBurn.

See Laser Window — Save Machine Files for more information.

Export

`Alt`/`Option`+`X`

Exports currently selected graphics in `.ai`, `.svg`, or `.dxf` format. If there is nothing in your current selection, the entire contents of your current project will be Exported.

See File Management — Export for more information.

Preferences

See Preferences Submenu for more information.

Import Prefs

Import previously copied or exported `.lbprefs` LightBurn preference files.

> **Note**
>
> A new project is required to import `.lbprefs` LightBurn preference files.

Export Prefs

Export the current LightBurn preferences.

Open Prefs Folder

Opens the folder on your computer containing the LightBurn preferences files, created automatically at program close or when settings are changed. Only the latest 50 preference backup files are preserved.

Windows LightBurn Preferences Folder

`%localappdata%/LightBurn` `C:\Users\username\AppData\Local\LightBurn`

Load Prefs Backup

Import a built-in LightBurn preferences back up file.

Windows LightBurn Preferences Backup Folder

`%localappdata%/LightBurn/backup` `C:\Users\username\AppData\Local\LightBurn/backup`

Edit Hotkeys

Create and change the keyboard shortcuts of many LightBurn functions and commands in the **Hotkey Editor**.

See Hotkey Editor for more information.

Bundles

Bundles simplify the process of migrating LightBurn by bundling everything that needs to be moved into a single `.lbzip` file.

Bundles can include:

- LightBurn Settings
- Custom Hotkeys
- Image Presets
- Devices
- Material Libraries
- Art Libraries

See the User Bundles for more information.

Import Bundles

Load previously exported Bundles into the current instance of LightBurn. Any component contained within the bundle that your computer detects is already present will be grayed out and unavailable on import. Devices can be imported even if already present, and will create a new copy of the device. Bundles can also be imported by dragging the `.lbzip` file into LightBurn.

Export Bundle

Create a Bundle of LightBurn settings, laser devices, Material and Art libraries, and more.

Print (black only)

`Ctrl` / `Cmd` + `P`

Output a black and white document to a printer or built-in "Print to PDF" tool. This document is saved exclusively in black and white.

See Print for more information.

Print (keep colors)

`Ctrl` / `Cmd` + `Shift` + `P`

Output a document to a printer or built-in "Print to PDF" tool. This document is saved with the assigned colors in the current Color Palette.

See Print for more information.

Saved Processed Bitmap

Exports a copy of an adjusted bitmap.

See Save Processed Bitmap for more information.

Save Background Capture

Saves a bitmap image of a Workspace Overlay produced by a camera affixed over your laser's work area.

See Save Background Capture for more information.

Exit

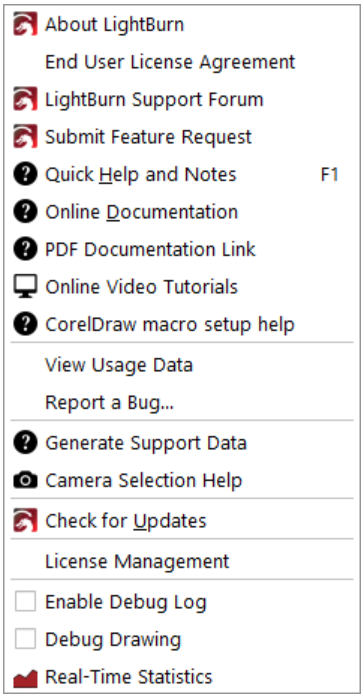
`Ctrl` / `Cmd` + `Q`

Completely close and exit LightBurn. You will be prompted to save the current project if there are any unsaved changes, discard any changes and close without saving them, or cancel the process of closing LightBurn.

Help Menu

The **Help Menu** contains links to documentation and instructional videos; information on your LightBurn license and version; and tools for troubleshooting software issues.

Click any option in the image below to jump to more information, or scroll down for a list of options and descriptions.



About LightBurn

Opens the **Help and Notes** window, where additional information on keyboard shortcuts, a few general LightBurn tips, and important program build/version data is listed.

See Help and Notes for more information.

End User License Agreement

Loads LightBurn's End User License Agreement (EULA) for you to read or save to your computer for future reference.

See LightBurn EULA to view the EULA here.

LightBurn Support Forum

Get help with LightBurn from the developers, support staff, and other users. Join the community to share projects, discuss hardware and software settings, and learn how to get the most out of LightBurn.

Access the LightBurn forum at: Forum.LightBurnSoftware.com.

Submit Feature Request

Go directly to LightBurn's feature request page, to make your own requests and suggestions, and vote on suggestions from others.

Go to Fider.

Quick Help and Notes

F1

Opens the **Help and Notes** window, directly to the **Hotkeys** tab, which features a small partial list of mouse gestures, keyboard shortcuts, and other hotkey tips.

See Help and Notes for more information.

Online Documentation

Find guides and detailed information on getting the most out of LightBurn. If you're reading this, you're already here!

PDF Documentation Link

View the current printable PDF of the LightBurn documentation.

This menu entry will always point to the latest copy of the LightBurn documentation PDF.

Online Video Tutorials

Access a curated list of tutorial videos from a library of over a hundred helpful guides at LightBurnSoftware.com.

View the full library of LightBurn tutorial videos on YouTube.

CorelDRAW Macro Setup Help

Simplify and speed up the process of sending work out of CorelDRAW to LightBurn on Windows-based computers.

See CorelDRAW Macro Setup for more information.

View Usage Data

Enable and disable the collection of anonymous usage data, clear collected data, and view and copy the data collected.

See View Usage Data for more information.

Report a Bug

Opens up a contact form to report bugs directly to LightBurn from within the software. If you've found what appears to be a bug, please let us know so we can figure out what's wrong and make improvements.

You can also report bugs to us directly by email.

Generate Support Data

Generates encoded data that helps the LightBurn support team answer your questions.

See Generate Support Data for more information.

Camera Selection Help

Opens a window to help select an appropriate camera for your laser based on machine size and mounting height.

See Camera Selection Help for more information.

Check for Updates

Checks whether there is a more recent version of LightBurn available than the version currently running, and whether it was released within your license's valid update period.

See Check for Updates for more information.

License Management

Opens the **License Page**, where you can activate or deactive your license, and see the number of days remaining in your license's update period.

See License Management for more information.

Enable Debug Log

Begins recording a log saved to the **My Documents** folder on Windows or **Documents** on macOS, to aide in troubleshooting and bug fixing when communicating with LightBurn support.

See Enable Debug Log for more information.

Debug Drawing

Activates the internal tool that shows the bounds of shapes being drawn in the Workspace, to aid LightBurn developers in troubleshooting and debugging.

See Debug Drawing for more information.

Real-Time Statistics

Shows an overview of bandwidth used by Network Cameras.

Language Menu

Change the LightBurn program interface language by selecting between 25 supported translations. The currently selected language is indicated by a dot next to the language in the list. English (United States) is enabled by default.

See Create and Edit Text — Text Settings for information on additional text settings. The Right-To-Left setting will automatically be enabled when the current program language is set to Arabic or Hebrew.

The Right-To-Left text setting will automatically be enabled in your Settings when the current program language is set to Arabic or Hebrew.

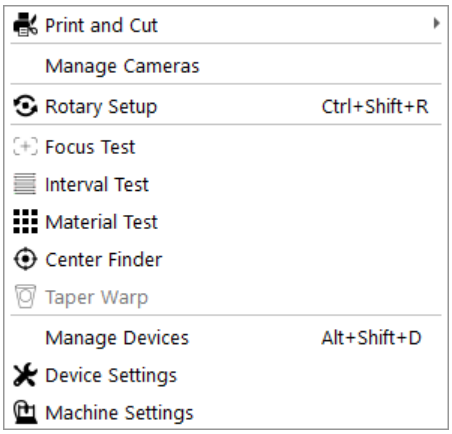


Laser Tools Menu

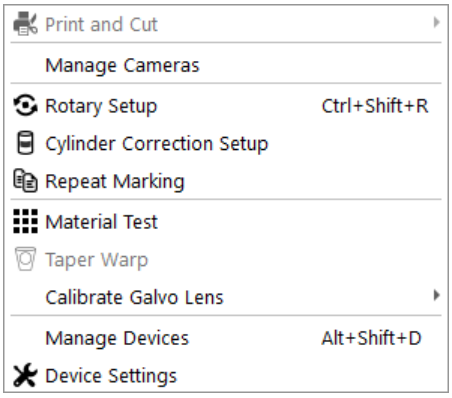
The **Laser Tools** menu contains tools, tests, and step-by-step wizards for performing advanced lasering operations, dialing in settings to optimize product quality, and setting up accessory hardware.

You'll see different menu options depending on the type of laser you have selected.

Click any option in the images below to go directly to the relevant page for that tool, or scroll down for a list of options and descriptions.



The Laser Tools Menu for Gantry style lasers (GCode and DSP)



The Laser Tools Menu for Galvo style lasers

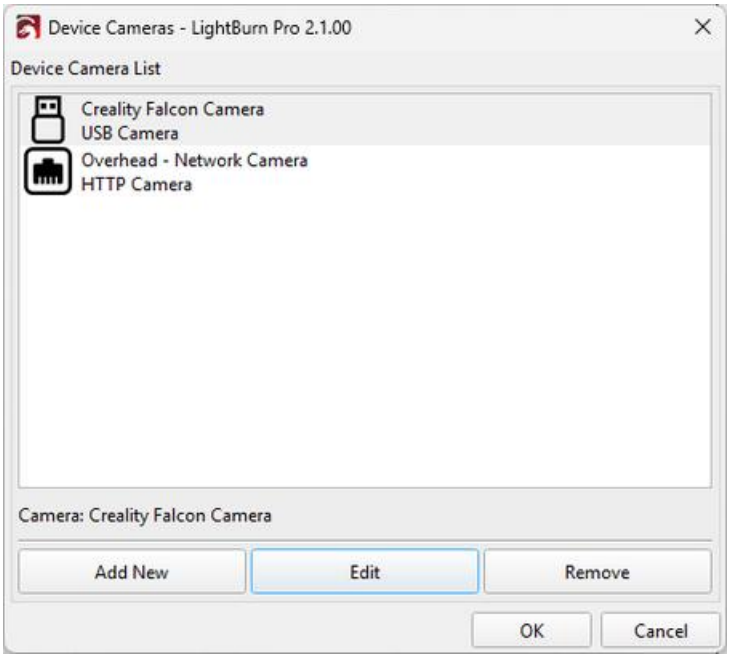
Print and Cut

Aligns your current LightBurn project to target locations in a previously output design. Using Print and Cut requires a properly homed Gantry style laser with an accurate motion system.

See Print and Cut for more information.

Manage Cameras

Opens the **Device Cameras** window, which shows a list of cameras associated with your current device. Use the buttons on the bottom to Add, Edit, or **Remove** a camera.



See Add a Camera and Edit a Camera for more information.

Rotary Setup

`Ctrl`/`Cmd`+`Shift`+`R`

Rotaries are used to turn cylindrical objects so that their surfaces can be marked by a laser. In order to use your laser with a rotary, you must first establish several important parameters in the **Rotary Setup** window that control the rotary's motion and adjust the output sent to your laser.

For more information on using a rotary with your type of laser, see:

- Rotary Mode (DSP)
- Rotary Mode (GCode)
- Rotary Mode (Galvo)

Cylinder Correction Setup

Warps the output of your design to correct for the expansion that happens when you mark on a cylindrical object, like a cup or tumbler, allowing you to mark cylindrical objects with a Galvo style laser, without using a rotary.

See Cylinder Correction for more information.

Repeat Marking

Enables the use of a **Rotating** or **Linear** axis table with Galvo lasers, to move parts past a laser while running repetitions of identical jobs.

See Repeat Marking for more information

Focus Test

Use the Focus Test setup dialog to set parameters and generate a test pattern to identify optimal focal height. Requires a machine with a mechanized, controllable Z Axis.

See Focus Test for more information.

Interval Test

Generates a test pattern to identify the optimal **Line Interval** for engraving at a given combination of **Speed** and **Power**.

See Interval Test for more information.

Material Test

Generates a test grid of varying combinations of two selected variables. Testable variables are device-dependent, and include **Speed**, **Power**, **Interval**, **Passes**, **Frequency**, and **Q-Pulse**.

See Material Test for more information.

Center Finder

Finds the center of a circular object in the laser's physical work area, by aligning the laser to three points on the object. Can also be used with any object whose shape can be described with a circle diameter.

See Center Finder for more information.

Taper Warp

Adjusts a design to compensate for the difference in diameter between the top and bottom of a cylindrical object, when engraving using Rotary Mode or Cylinder Correction.

See Taper Warp for more information.

Calibrate Galvo Lens

Marks a test pattern that is then measured and input into LightBurn, to compensate for distortion effects from a new or uncalibrated Galvo lens.

See Calibrate Galvo Lens for more information.

Manage Devices

Opens the **Devices** window, where you can create and edit profiles for your lasers.

See Devices for more information.

Device Settings

View and edit device-specific settings for the currently active device. The contents of this window vary depending on the device in use.

See Device Settings for more information.

Machine Settings

View and edit device parameters for lasers with supported controllers.

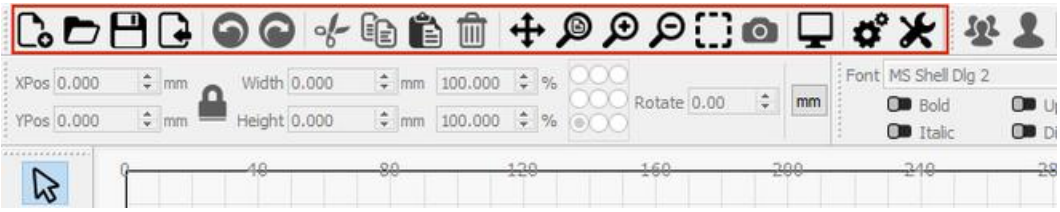
See Machine Settings for more information.

Main Toolbar

The **Main Toolbar** contains tools for creating, opening, and saving projects; copying, pasting, and deleting graphics; accessing your LightBurn preferences and Device Settings; and more.

Accessing the Main Toolbar

The **Main Toolbar** is located on the left side of the top toolbar in LightBurn by default, to the left of the Arrange Toolbar.



If you have closed the Main Toolbar, go to Window → **Main** to re-enable it.

To restore it and all other toolbars and windows to their default positions, go to **Window** → Reset to Default Layout.

> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Main Toolbar Options

Click any option in the image below to go directly to the relevant page for that tool, or scroll down for a list of options and descriptions.

File Management

See File Management for more information on **New**, **Open**, **Save**, and **Import**.

New

`Ctrl`/`Cmd`+`N`

Creates a new file. A new, unnamed project will replace any existing projects loaded in the LightBurn workspace.

Open

`Ctrl`/`Cmd`+`O`

Opens a file. The dialog box filters for `.lbrn` and `.lbrn2` LightBurn files by default. Click on the dropdown to show all supported file types, including supported image and vector file types.

> **Warning**
>
> Selecting a supported image or vector file instead of a project file will create a **new LightBurn project**, replacing the currently loaded project. Use Import to bring artwork files or designs into an existing project.

Save

`Ctrl`/`Cmd`+`S`

Saves your current project in `.lbrn` or `.lbrn2` format.

Import

`Ctrl`/`Cmd`+`I`

Loads graphics from a variety of file formats into LightBurn.

Undo and Redo

Reverse or reapply past actions and design changes in LightBurn.

See Undo/Redo for more information.

Clipboard Tools

See Clipboard Tools for more information on **Copy**, **Cut**, **Paste**, and **Delete**.

Copy

`Ctrl`/`Cmd`+`C`

Saves a copy of selected objects to your clipboard.

Cut

`Ctrl`/`Cmd`+`X`

Deletes selected objects in your Workspace while saving a copy of them to your clipboard at the same time.

Paste

`Ctrl`/`Cmd`+`V`

Inserts objects copied to your clipboard into your Workspace. Pasted objects are centered at the location of your cursor at the time of pasting.

Delete

`Del`

Deletes selected objects.

Pan / Drag View

Click to drag the Workspace around. Mouse users can click the mouse wheel or third button and drag.

See Panning for more information.

Zoom Tools

See Zoom for more information on **Zoom to Page**, **Zoom In**, **Zoom Out**, and **Zoom to Frame Selection**.

Zoom to Page 

Adjusts the view in your Workspace to center and show the grid as large as possible.

Zoom In and Zoom Out 

Adjusts the view in your Workspace in to get a closer view, or out to see more at once.

Zoom to Frame Selection

Zooms in or out to show all selected objects as large as possible, centered within your view. If no objects are selected, it will instead zoom to show all objects in the Workspace.

Update the Background From the Camera 

Captures a new image to use as an Overlay for your Workspace. This button is only selectable when a camera is in use.

See Cameras / Overlays Window for more information.

Preview 

Opens the **Preview** window, which displays the simulated movements and lasering operations for the project in your Workspace.

See Preview for more information.

Settings 

Opens the **Settings** window, which contains global user preferences for LightBurn — these settings apply regardless of what lasers you have set up, and which one you currently have selected.

See Settings/Preferences for more information.

Device Settings 

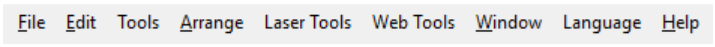
Opens the **Device Settings** window, which contains device-specific settings for your laser. The settings available change based on the device selected.

See Device Settings for more information.

Menu Toolbar

The **Menu Bar** contains all of LightBurn's top-level menus.

Click any menu in the image below to go directly to the relevant page for that menu, or scroll down for a list of menus and descriptions.



> **Note**
>
> On macOS, the menus will be shown at the top of the screen, rather than as part of the LightBurn window.

File

Contains options for saving, opening, importing, and exporting files, as well as accessing and exporting your LightBurn settings.

See File Menu for a full list of options in the File Menu.

Edit

Contains tools for selecting and manipulating objects in addition to general LightBurn settings.

See Edit Menu for a full list of options in the Edit Menu.

Arrange

Contains tools for positioning objects, both on an individual basis, and in relation to each other. It also contains tools for moving the physical laser in relation to objects within your project.

See Arrange Menu for a full list of options in the Arrange Menu.

Laser Tools

Contains tools, tests, and step-by-step wizards for performing advanced lasering operations, dialing in settings to optimize product quality, and setting up accessory hardware.

See Laser Tools Menu for a full list of options in the Laser Tools Menu.

Web Tools

LightBurn 2.1 has a new Web Tools Menu, where you'll find tools* for discovering, generating, or customizing projects for your laser.

All Web Tools open directly in a browser built into LightBurn, and you can import your results straight into your Workspace. From there you can utilize our laser control tools to apply Cut Settings, position your output, and send the job to your laser.

Some tools may require payment, while others will be completely free. In cases where a service requires a payment or subscription, LightBurn does not take a cut — our only aim in providing this menu is to share tools we're fans of, and facilitate smoother workflows our users.

*Ok, as of now, you'll only find Cuttle there. But we'll be exploring more of these types of integration down the road, so keep an eye out for new entries in our Web Tools Menu going forward.

Window

Use the **Window Menu** to control and customize your Workspace view, enable and disable Windows and Toolbars, or reset LightBurn's layout to its original state.

See Window Menu for a full list of options in the Laser Tools Menu.

Language

Use **Language Menu** to switch the LightBurn interface into 1 of 24 different languages.

See Language Menu for a full list of languages.

Help

Contains links to LightBurn resources, access to the **License Management** window, and a handful of diagnostic tools.

See Help Menu for a full list of options in the Help Menu.

Modes Toolbar

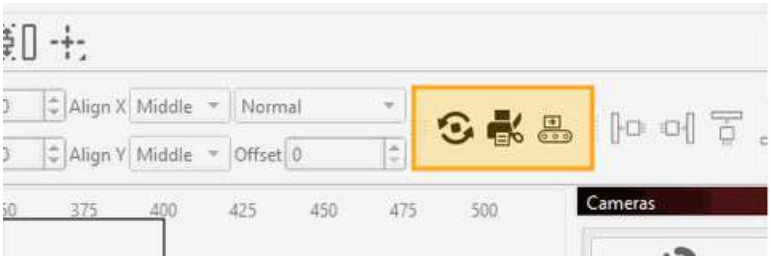
The **Modes Toolbar** contains buttons to open dialog windows or wizards for advanced lasering processes.

The Modes Toolbar appears differently depending on the type of laser you have selected.

- For Gantry style lasers, you'll see buttons for Rotary Mode, Print and Cut, and Split Marking.
- For Galvo style lasers, you'll see buttons for Rotary Mode, Cylinder Correction, and Split Marking.

Accessing the Modes Toolbar

The **Modes Toolbar** is located near the top right of the top toolbar in LightBurn by default, between the Text Options Toolbar and the Docking Toolbar.



If you have closed the Modes Toolbar, go to Window → **Modes** to re-enable it.

To restore it and all other toolbars and windows to their default positions, go to **Window** → Reset to Default Layout.

> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Modes Toolbar Options

Click any button in the images below to jump to information on that option, or scroll down for a list of options and descriptions.



The Modes Toolbar for Gantry style lasers (GCode and DSP)



The Modes Toolbar for Galvo style lasers

Rotary Mode

Rotaries are used to turn cylindrical objects so that their surfaces can be marked by a laser. In order to use your laser with a rotary, you must first establish several important parameters in the **Rotary Setup** window that control the rotary's motion and adjust the output sent to your laser.

For more information on using a rotary with your type of laser, see:

- Rotary Mode (DSP)
- Rotary Mode (GCode)
- Rotary Mode (Galvo)

Print and Cut

Print and Cut aligns a current project to existing **Target Positions** — also called registration marks — in a previously output design.

Print and Cut works by measuring the difference in location, orientation, and scale between two Target Positions in your laser's physical work area and the positions of two corresponding graphics in your LightBurn Workspace.

See Print and Cut for more information.

> **Note**
>
> This option is available for Gantry style lasers only.

Cylinder Correction

Cylinder Correction (called ProjectMark in EZCAD) warps the output of your design to correct for the expansion that happens when you mark on a cylindrical object, like a cup or tumbler, without using a rotary.

See Cylinder Correction for more information.

Split Marking

Split Marking enables the ability to mark designs larger than a Galvo laser's work area using a **Rotating** or **Linear** axis table.

See Split Marking for more information.

Modifiers Toolbar

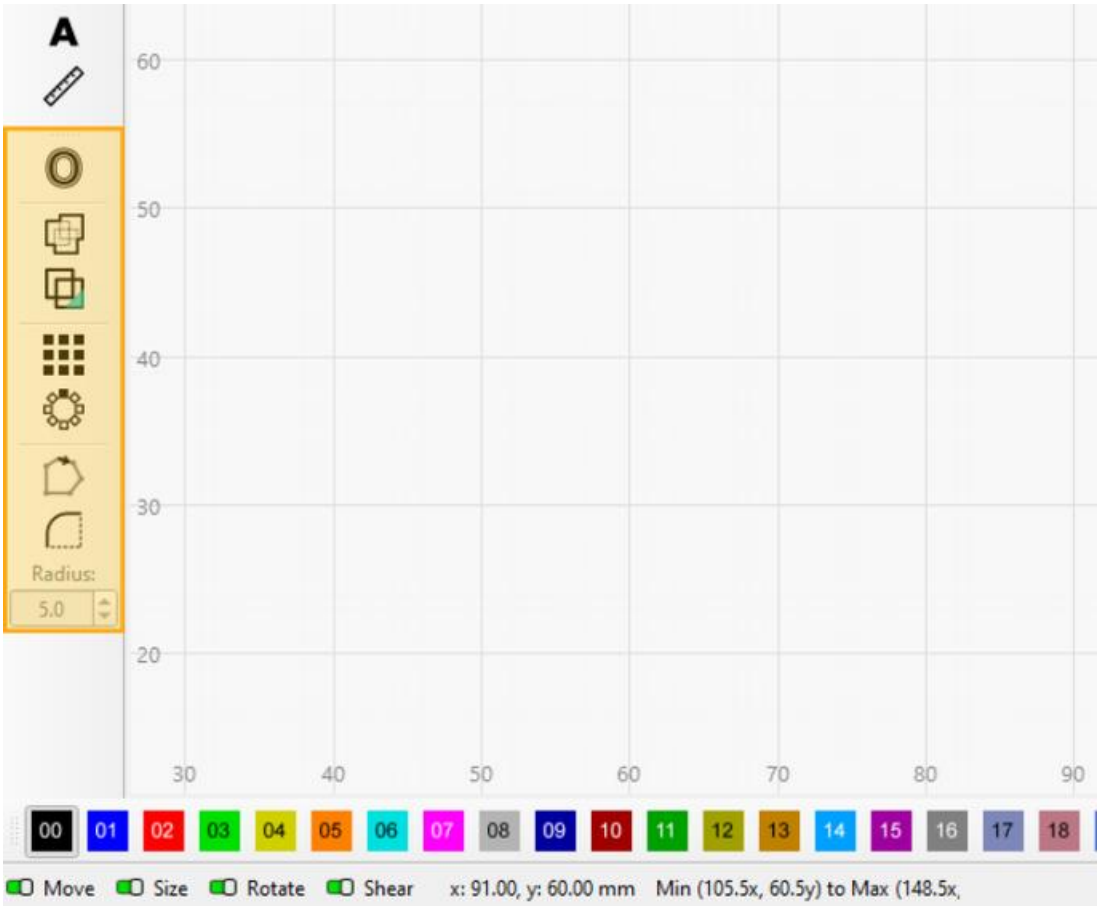
The **Modifiers Toolbar** contains tools for combining, altering, and producing arrayed duplicates of graphics.

Accessing the Modifiers Toolbar

By default, the **Modifiers Toolbar** is located in the bottom half of the left-side toolbar in the main LightBurn window.

If you have closed the Modifiers Toolbar, go to Window → **Modifiers** to re-enable it.

To restore it and all other toolbars and windows to their default positions, go to **Window** → Reset to Default Layout.



Tip

The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Modifiers Toolbar Options

Click any button in the image below to jump to information on that tool, or scroll down for a list of options and descriptions.



> **Note**
>
> The Modifiers Toolbar is normally vertical, but can also be removed from the left-side toolbar and oriented horizontally, as presented below. See Customizing the LightBurn Window for more information.

Offset Shapes

`Alt`/`Option`+`O`

Creates a new shape by outlining the shapes in your selection at a distance you specify, either inward, outward, or in both directions at once.

See Offset Shapes for more information.

Boolean Tools

Modify two overlapping shapes (or two Groups of shapes) to create new shapes.

- **Boolean Union** merges two shapes.
- **Boolean Subtract** removes one shape from another.
- **Boolean Intersection** leaves only the area covered by both shapes.
- **Weld** can merge more than two shapes at once.

See Boolean Tools for more information.

Grid Array

Creates copies of an object (or objects) in regularly spaced rows and columns, and includes options to adjust spacing, mirror the shapes, randomize orientation, and more.

See Grid Array for more information.

Circular Array

Creates copies of an object (or objects) in a regularly spaced circular (radial) pattern, with options to adjust spacing, rotate copies, and more.

See Circular Array for more information.

Set Start Point

Allows you to specify where on a shape your laser will begin cutting, and in which direction it will cut.

See Set Start Point for more information.

Radius 

Fillets (rounds over) sharp corners, takes a round bite out of a corner (using a negative radius value), or turns a radius back into a sharp corner (removes a radius).

See Radius/Fillet for more information.

Status Bar

The **Status Bar** provides contextual information related to the active tool or process being performed. Check the Status Bar for related information, instructions on the next steps, or options available.

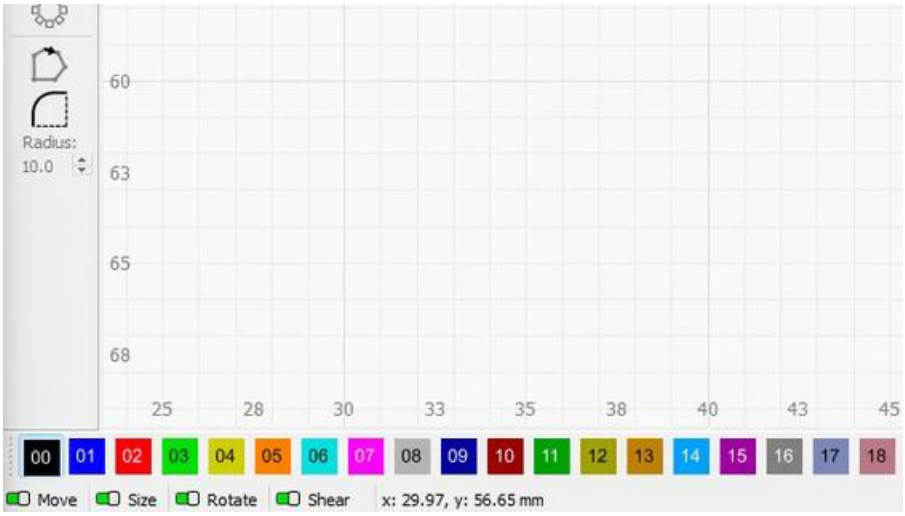
Locating the Status Bar

The Status Bar is located in the bottom left-hand portion of the LightBurn window directly to the right of the Transform Control Toggles, and shows the precise location of the mouse cursor/pointer; the dimensions and status of items selected; the Notification Area where important tool, program, and laser messages are displayed; and the special Modes indicator.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=u0jkbYheUJ8



Cursor Coordinates



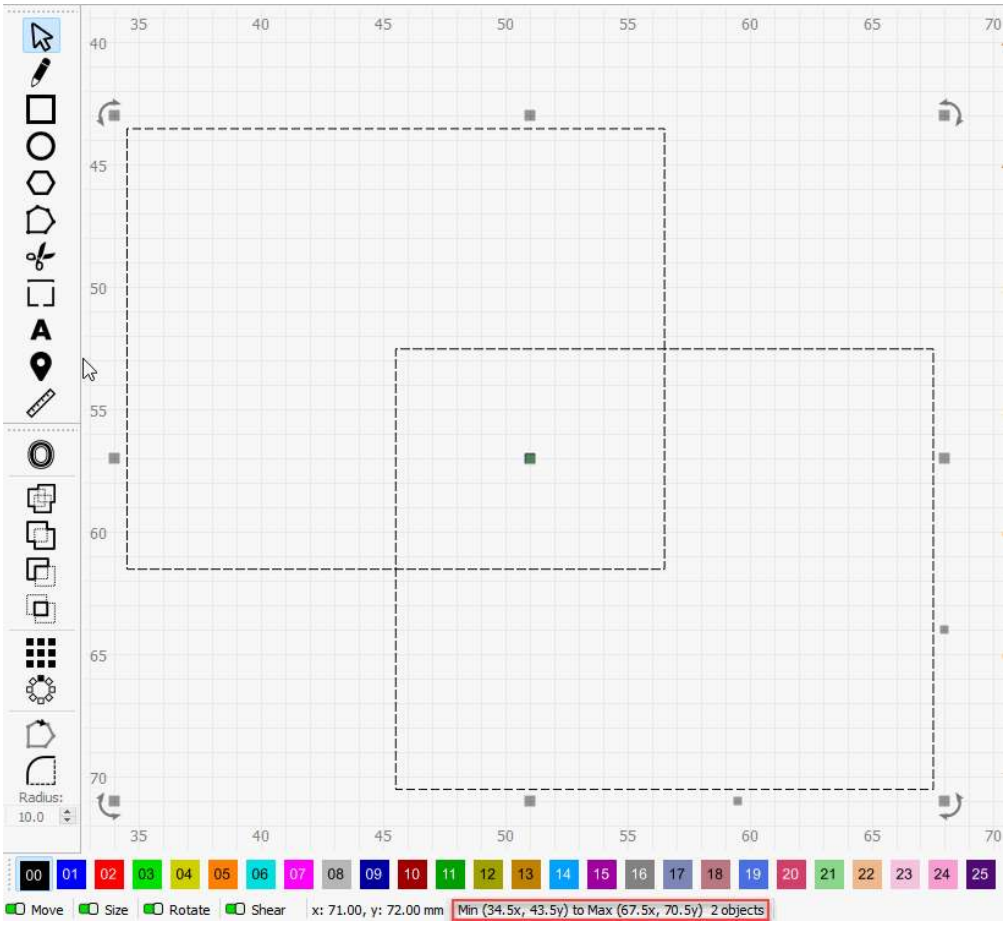
View the live and precise X and Y Axis coordinates of the mouse cursor within and around the LightBurn Workspace on the left side of the **Status Bar**. The displayed coordinates will stop updating when the mouse cursor leaves the area around the Workspace — for example when choosing tools, working within windows and toolbars, or interacting with menus.

Tool Status

The **Tool Status** section to the right of **Cursor Coordinates** displays information on the currently selected object(s) or tool.

Example Tool Statuses

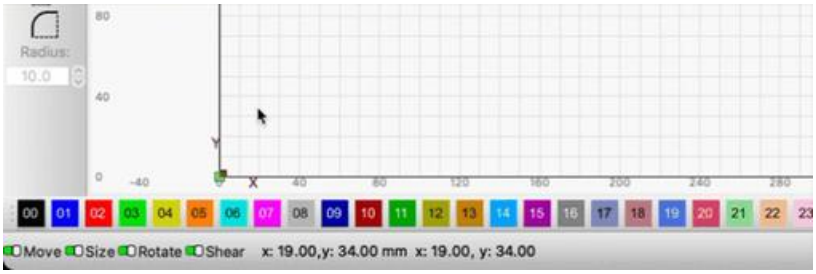
Selection Tool



Message: Min (34.5x, 43.5y) to Max (67.5x, 70.5y) 2 objects

Meaning: The rectangular selection bounding box has the corner closest to the origin at (34.5, 43.5) and the corner the farthest away from the origin at (67.5, 70.5), with two objects included and selected.

Draw Lines Tool

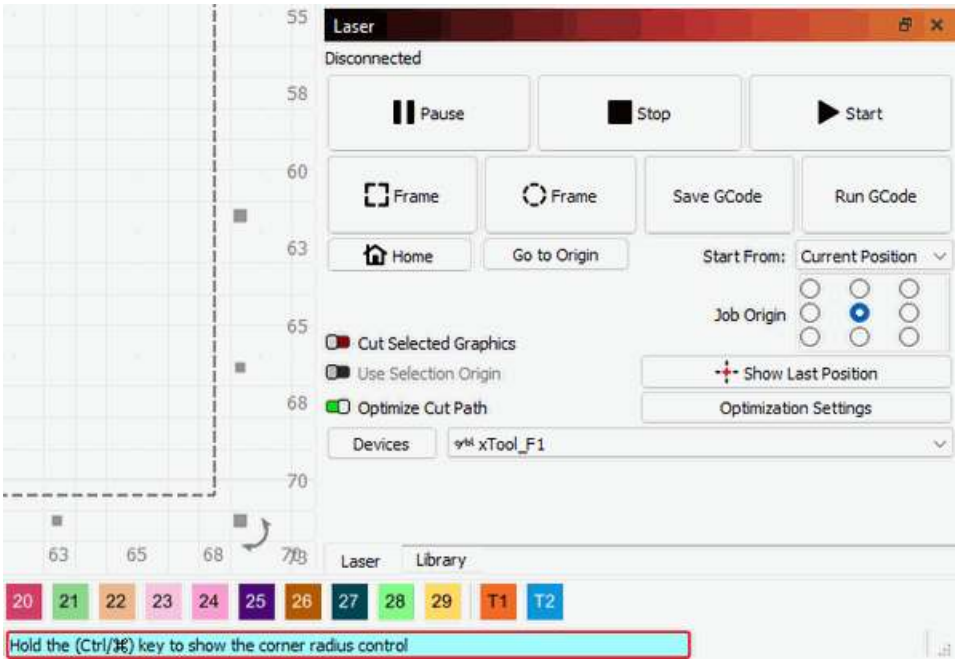


Message: x: 258.00, y: 66.00 (dx: 239.00, dy: 32.00, len: 241.13)

Meaning: Current cursor coordinates (difference in travel from the start of the line in the X and Y Axis, length of line)

> **Note**
>
> The Cursor Coordinates are always displayed to the left of Tool Status, and are displayed twice with this tool.

Notification Area



The **Notification Area** on to the right-hand side of the **Status Bar** displays tool-specific messages, in addition to important information from the program. The exact message that's displayed will change depending on the tool you're currently using, or the last action you took.

Connection Status

If you've set up and selected a DSP or Galvo device profile, the connection status of your device will be shown in the **Notification Area**.

DSP

If you've selected a DSP profile, and your laser is not connected, you'll see a **No device found** message.

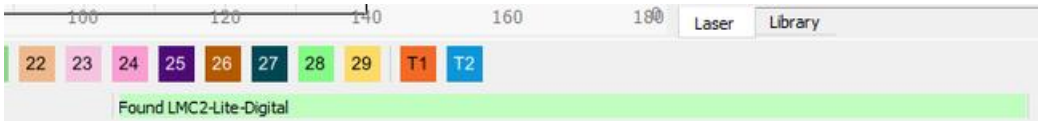
If it is found and connected, you'll see a message identifying the laser's controller, and indicating that the device has been found.



Galvo

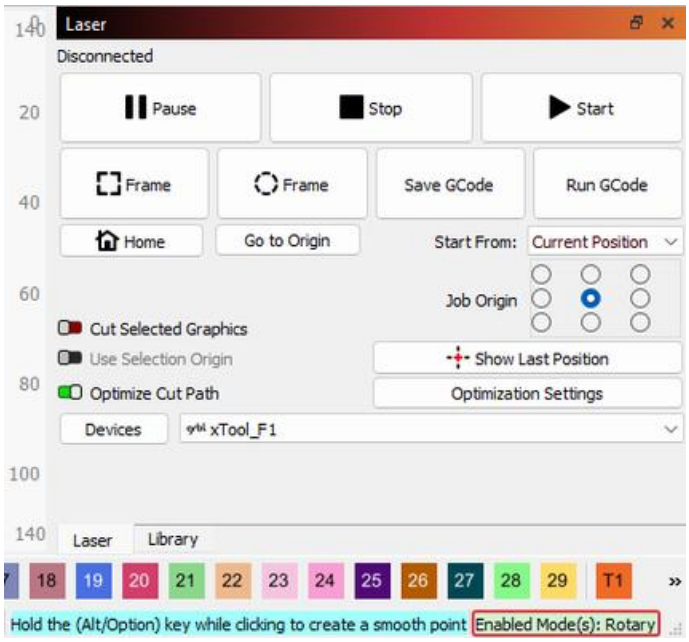
If you've selected a Galvo profile, and your laser is not connected, you'll see a **Waiting for connection...** message.

If the connection is established, you'll see message identifying the laser's controller, and indicating that it is connected.



If you are unable to connect to your laser, see Troubleshooting: Connection Problems.

Modes Indicator



The last section of the Status Bar displays the name of any active special Modes, including Rotary Mode, Print and Cut, and Cylinder Correction.

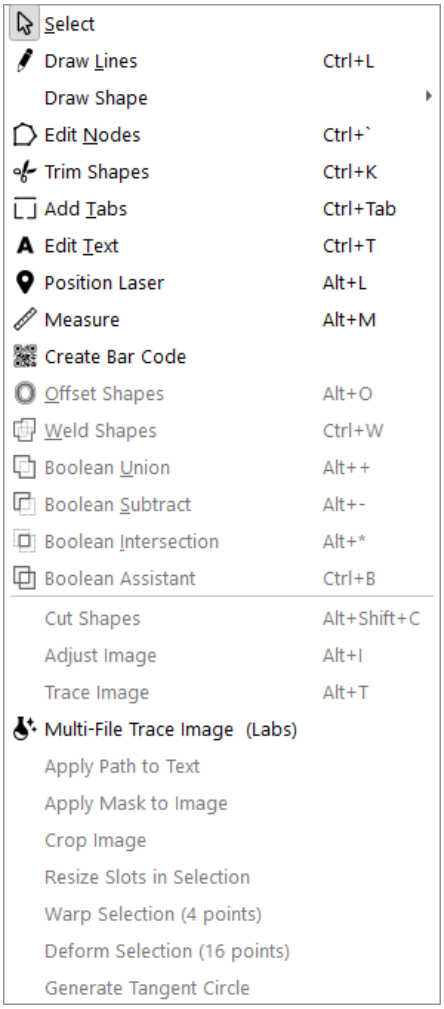
Related Topics

- Workspace / Edit Window
- Transform Control Toggles
- Color Palette
- Rotary Mode
- Print and Cut
- Cylinder Correction

Tools Menu

The **Tools Menu** contains an array of options for selecting, creating, editing, and modifying graphics.

Click any option in the image below to go directly to the relevant page for that tool, or scroll down for a list of options and descriptions.



Select

`Esc`

The primary means of selecting and deselecting objects.

See Selection Tools for more information

Draw Lines

`Ctrl`/`Cmd`+`L`

Allows you to create custom shapes by placing points, with lines or curves between them.

See Draw Lines for more information.

Draw Shape

Opens a submenu for the various Primary Shapes.

Rectangle

`Ctrl`/`Cmd`+`R`

Creates rectangles or squares.

See Primary Shapes for more information.

Ellipse

`Ctrl`/`Cmd`+`E`

Creates ovals or circles.

See Primary Shapes for more information.

Triangle

Creates triangles.

See Primary Shapes for more information.

Pentagon

Creates pentagons. For polygons with an adjustable number of sides, see Polygon.

See Primary Shapes for more information.

Polygon

Creates regular polygons — hexagons by default — with an adjustable number of sides.

See Primary Shapes for more information.

Octagon

Creates octagons. For polygons with an adjustable number of sides, see Polygon.

See Primary Shapes for more information.

Star

Creates stars with an adjustable number of sides.

See Primary Shapes for more information.

Dual Star

Creates dual stars — two overlapping stars — with an adjustable number of sides. Each star can have a separate radius.

See Primary Shapes for more information.

Edit Nodes

`Ctrl`/`Cmd`+`` ` ``

Allows you to edit vector graphics by adding, deleting, or moving their points, or altering the shape of the lines/curves between.

See Edit Nodes for more information.

Trim Shapes

`Ctrl`/`Cmd`+`K`

Cuts lines back to the next point at which they intersect with another line.

See Trim Shapes for more information.

Add Tabs

`Ctrl`/`Cmd`+`Tab`

Specifies small sections of cuts that the laser is commanded to skip, used to prevent objects that you are cutting from immediately falling out of the base material.

See Add Tabs for more information.

Edit Text

`Ctrl`/`Cmd`+`T`

Activates the **Create / Edit Text** tool, allowing you to add, edit, and format text.

See Create and Edit Text for more information.

Position Laser

`Ctrl`/`Cmd`+`L`

Commands your laser to jog to a location in its physical work area, by clicking in your LightBurn Workspace.

See Position Laser for more information.

Measure

`Ctrl`/`Cmd`+`M`

Allows you to hover over any shape in your Workspace and display an array of useful information about it, including perimeter and overall dimensions.

See Measure for more information.

Create Bar Code

Generates a Bar Code or QR Code which stores data that's visible when the Code is scanned.

See Bar Code for more information.

Offset Shapes

`Alt`/`Option`+`O`

Creates a new shape by outlining the shapes in your selection at a distance you specify, either inward, outward, or in both directions at once.

See Offset for more information.

Weld Shapes

`Ctrl`/`Cmd`+`W`

Combines two or more overlapping vector graphics into a single a shape.

See Boolean Tools for more information.

Boolean Union

`Alt`/`Option`+`+`

Combines two overlapping vector graphics into a single shape.

See Boolean Tools for more information.

Boolean Subract

`Alt`/`Option`+`-`

Removes one vector graphic from a second graphic where they overlap, creating a new shape from the area that remains of the second.

See Boolean Tools for more information.

Boolean Intersection

`Alt`/`Option`+`*`

Creates a new shape from the area where two vector graphics overlap.

See Boolean Tools for more information.

Boolean Assistant

`Ctrl`/`Cmd`+`B`

Opens a dialog window that shows a preview of what each of the Boolean operations will do to your selected shapes and gives you the option to click **OK** to confirm the change, or **Cancel** to discard it.

See Boolean Tools for more information.

Cut Shapes

`Alt`/`Option`+`Shift`+`C`

Cuts vector shapes — dividing them into new shapes — using a Closed Shape as the cutting tool.

See Cut Shapes for more information.

Adjust Image

`Alt`/`Option`+`I`

Opens a window where you can alter image properties such as Contrast, Brightness, and Gamma while also changing Layer Settings.

See Adjust Image for more information.

Trace Image

`Alt`/`Option`+`T`

Outlines the content of a bitmap image, turning the outlines into vector graphics.

See Trace Image for more information.

Multi-File Trace Image (Labs)

> **Labs Feature**
>
> Labs features are new tools in development that we couldn't wait to get into the hands of our users. Please experiment with caution, and report any unexpected results on our forum.

Automates image tracing for multiple files.

See Multi-File Trace Image for more information.

Apply Path to Text

Attaches text to a vector object, reshaping it to follow the contour of the chosen path.

See Apply Path to Text for more information.

Apply Mask to Image

Cuts a bitmap image using a closed vector graphic, leaving only the area of the image that is contained within the vector. Both the image and the mask can be edited independently until flattened.

See Apply Mask to Image for more information.

Crop Image

Applies a mask to an image and immediately flattens the result, making the cropping permanent. After this, the image and shape can no longer be edited independently.

See Flattening a Mask for more information.

Resize Slots in Selection

Adjusts the dimensions of slots or tabs in selected objects, based on user specifications.

See Resize Slots for more information.

Warp Selection (4 Points)

Allows you to modify text, shapes, or images by dragging 4 handles — one on each corner of your selection.

See Warp/Deform Selection for more information.

Deform Selection (16 Points)

Allows you to modify text, shapes, or images by dragging 16 handles spread out in a 4 x 4 grid across your selection.

See Warp/Deform Selection for more information.

Generate Tangent Circle

Create a circle tangent to three existing circles.

See Tangent Circle Generator for more information.

Tooltips and Topic-Aware Help

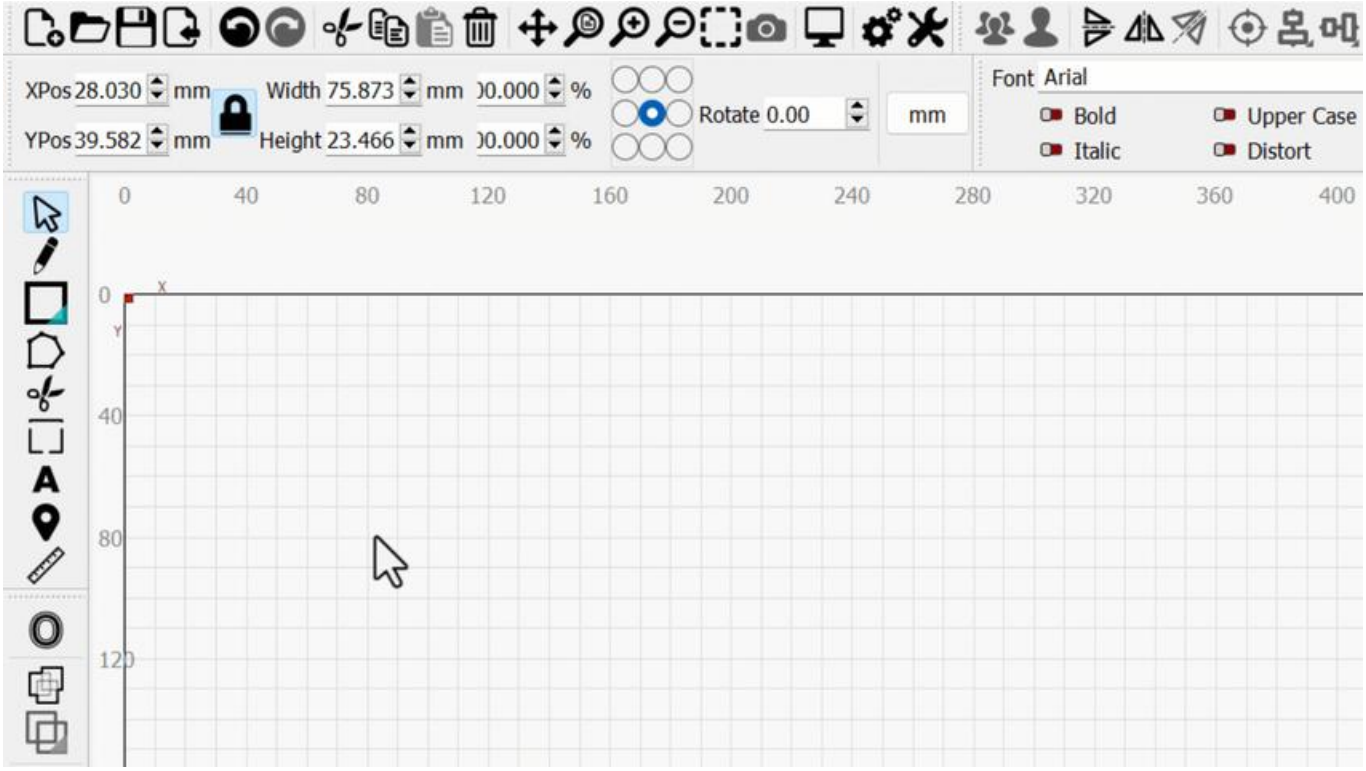
To help users navigate LightBurn's interface, we've included **Tooltips** and **Topic-Aware Help** for most features.

- Tooltips are brief descriptions of the action a tool performs. In some cases they also include keyboard shortcuts that tell you how to perform an action by pressing a combination of keys instead of clicking a button.
- When you press `F1` while hovering over a tool, LightBurn's Topic-Aware Help function launches the relevant documentation page in your default web browser.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=erKGi8LSu_g

Tooltips

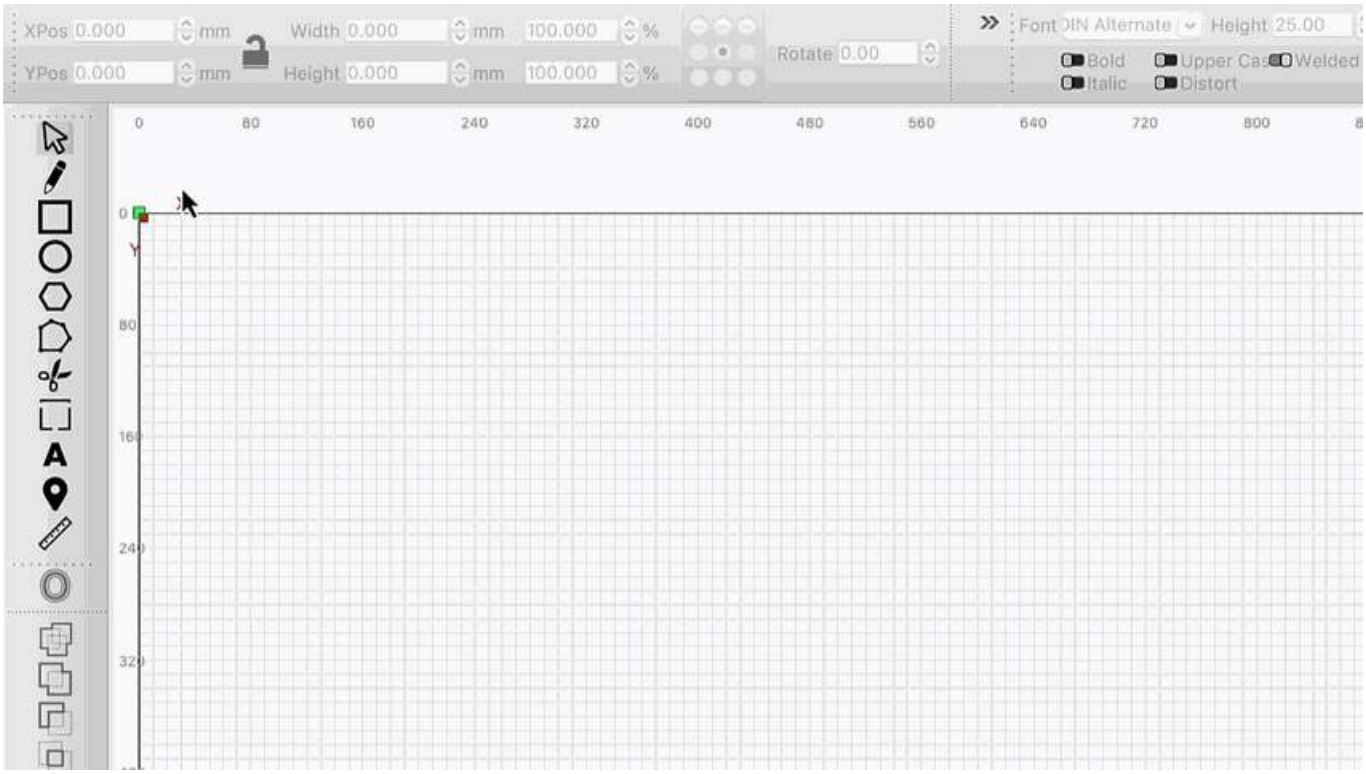
To reveal a **Tooltip**, briefly hover your cursor over anything you'd like to learn more about.



Topic-Aware Help

Press `F1` while hovering to open your default web browser to the relevant documentation page for the tool or feature you're hovering over.

> **Note**
>
> If you're using a Mac, hold the `Fn` key while pressing `F1`.

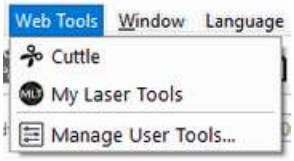

XPos 0.000 mm
YPos 0.000 mm
Width 0.000 mm 100.000 %
Height 0.000 mm 100.000 %
Rotate 0.00
Font DIN Alternate
Height 25.00
Bold
Italic
Upper Cas
Welded
Distort
0
80
160
240
320
400
480
560
640
720
800

Web Tools Menu

Use the **Web Tools Menu** to access LightBurn's built-in web tools and to add and manage your own custom web tools. These are all completely optional third-party services, and our built-in tools all have useful, relevant free options available, although they may have paid services as well.

About third-party Web Tools:

- Any memberships or accounts required by Web Tools are entirely separate from LightBurn.
- LightBurn does not have access to user accounts or memberships.
- LightBurn does not receive fees or compensation from third-party services.



Before you can use web tools, you'll need to acknowledge the third party service notice. The text of the notice is below for your convenience, bu the official and most up-to-date version is the one in the software itself.

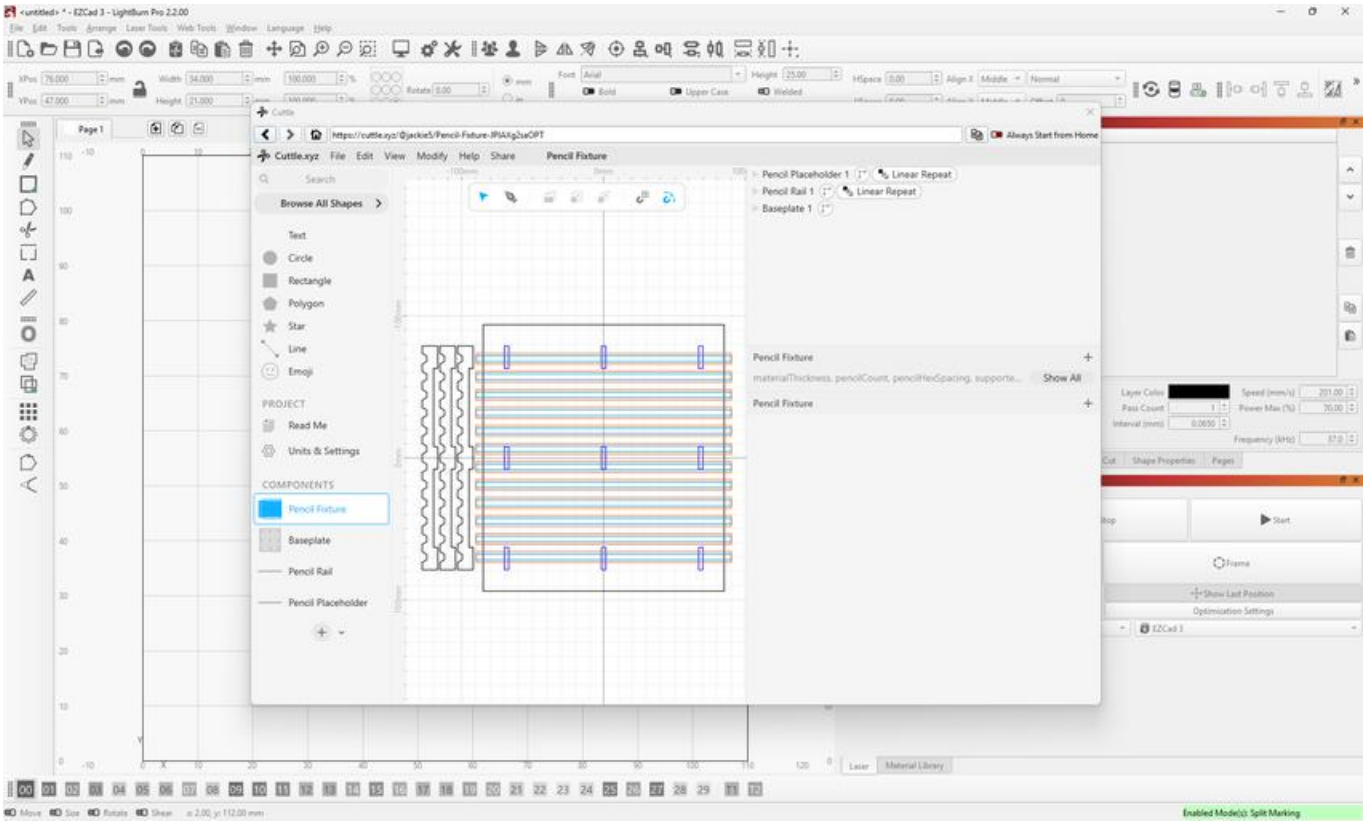
> **Third Party Service Notice**
>
> This feature may provide access to or display third-party websites or services operated independently from LightBurn. LightBurn does not control and is not responsible for the availability, content, functionality, security, privacy practices, transactions or any operations of such third-party websites.
>
> Your interaction with such services may be subject to separate third-party terms and privacy policies, and you are responsible for reviewing and complying with such terms.
>
> By continuing, you acknowledge that (a) this feature is governed by the third party's applicable terms and privacy policy; and (b) LightBurn is not responsible for any loss, damage, data breach, interruption, or other issue arising from your use of the third-party website or services.

Built-In Web Tools

Currently, LightBurn has two built-in web tools: Cuttle and My Laser Tools. Both are sites offering a wide range of project generators to help you create your own projects.

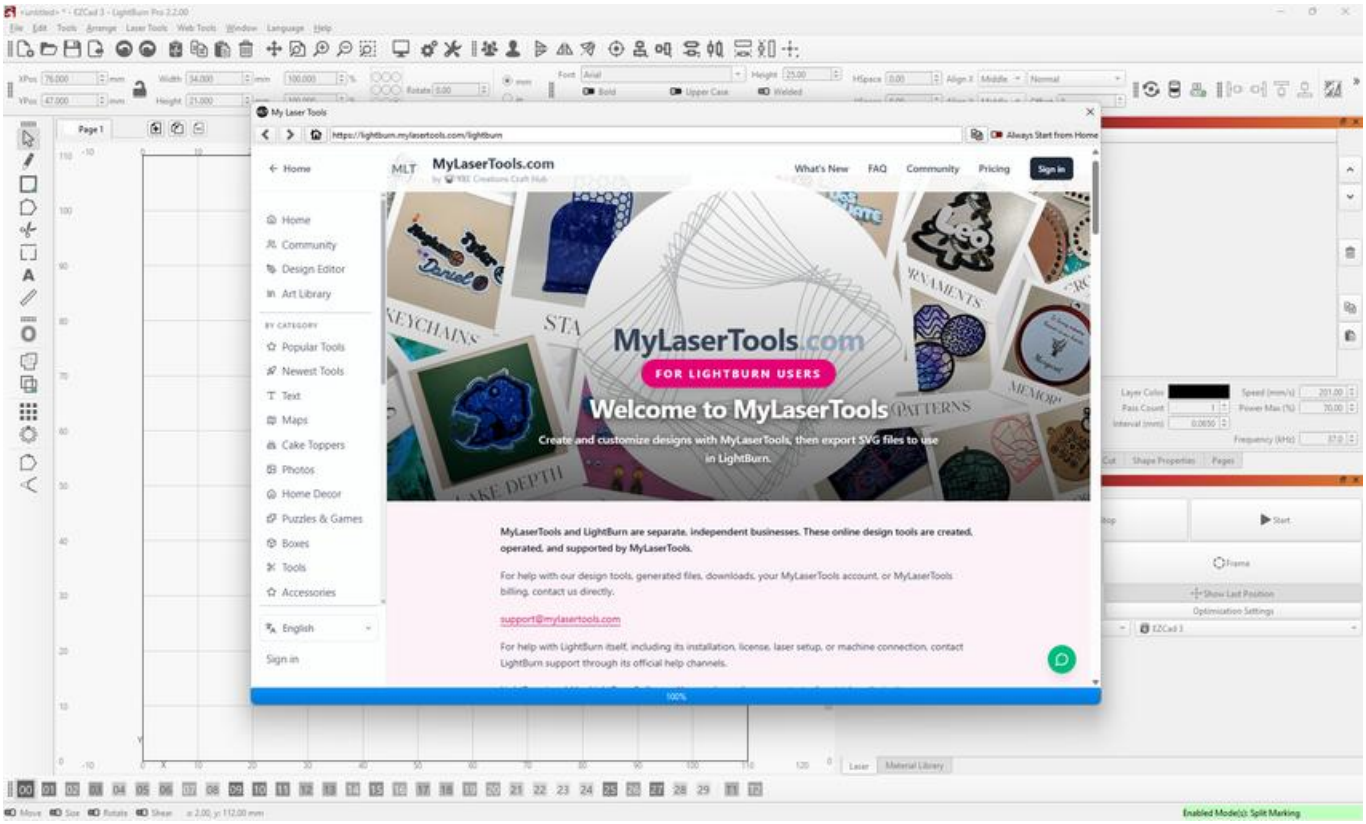
Cuttle

You can use our built-in Cuttle integration to access Cuttle without leaving LightBurn, and to import your designs directly into the workspace in LightBurn.



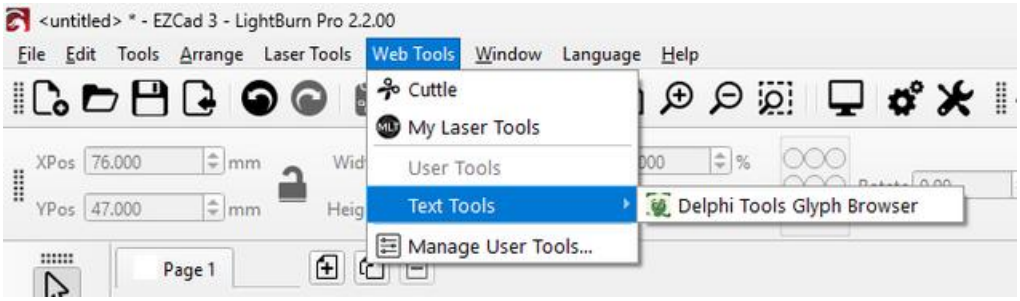
My Laser Tools

You can use our built-in My Laser Tools integration to access My Laser Tools without leaving LightBurn, and to import your designs directly into the workspace in LightBurn.



User Tools

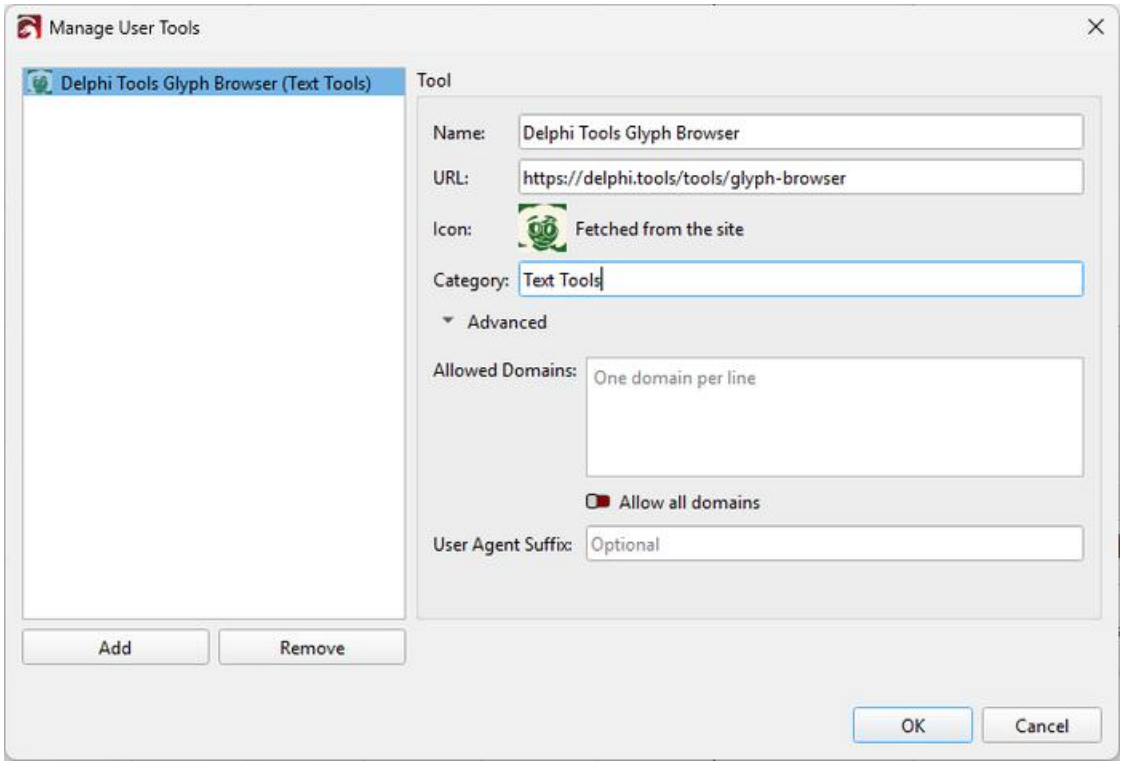
You can also add your own tools to the Web Tools and use them in much the same way as the built-in Web Tools. User Tools are only installed for your computer.



Once you've added at least one User Tool, you'll be able to find and access your User Tools from the Web Tools menu. User Tools will always be in the User tool section.

Adding User Tools

To add User Tools:



1. Open the Manage User Tools Window from the Web Tools menu.
2. Click the **Add** button.
3. Enter the required information. Click **Advanced** if you need to access advanced settings.
4. Click **OK** to confirm.

User Tool Settings

Setting	Required?	Description
Name	Required	The name to show in the menu.
URL	Required	The URL of the site to use for the web tool.
Icon	-	Automatically fetched from the site. Can't be changed by the user.
Category	Optional	If you enter a category, it will be used to group tools into submenus.
Allowed Domains (Advanced)	Optional	Additional domains the tool is allowed to open. Enabling **Allow All Domains** will allow the tool to open any website. Use with caution - if you don't understand this setting, don't use it.
User Agent Suffix (Advanced)	Optional	Text added to the end of the browser's user agent. Only needed if the site requires for it. Most tools will not need this.

Editing User Tools

To edit User Tools:

1. Open the Manage User Tools Window from the Web Tools menu.
2. Select the tool you want to edit.

3. Edit the information.
4. Click **OK** to confirm.

Removing User Tools

To remove User Tools:

1. Open the Manage User Tools window from the Web Tools menu.
2. Select the tool you want to remove
3. Click the **Remove** button.

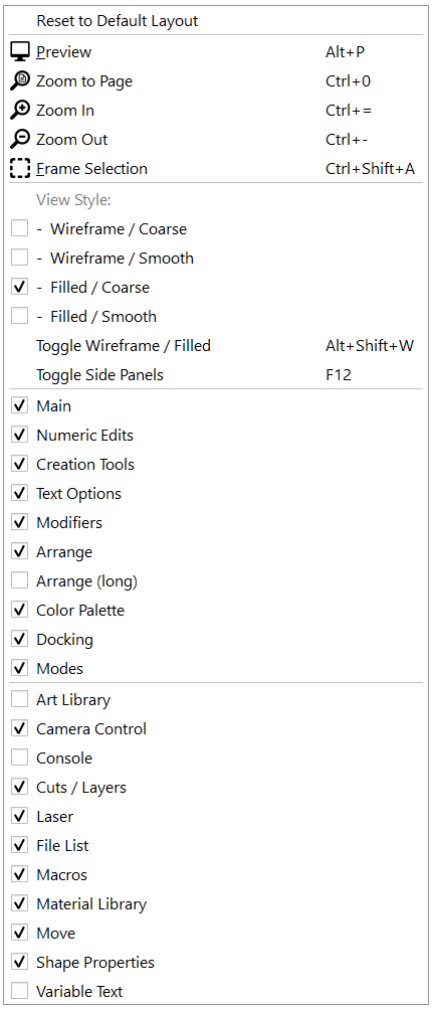
Window Menu

Use the **Window Menu** to control and customize your Workspace view, enable and disable windows and toolbars, or reset LightBurn's layout to its original state.

Any window or toolbar that is currently enabled will have a check mark next to it. Click unchecked windows or toolbars to enable them, and click again to disable them.

For more information on enabling, disabling, and customizing the position of windows and toolbars, see Customizing the LightBurn Window.

Click any option in the image below to go directly to the relevant page for that tool or window, or scroll down for a list of options, toolbars, windows, and descriptions.



Reset to Default Layout

Restores the layout of the full LightBurn window to the same format when first installed.

See Reset to Default Layout for more information.

Preview

`Alt`/`Option`+`P`

Displays a simulation of movements and lasering operations before a project is sent to the machine.

See Preview for more information.

Zoom

The Zoom options allow you to increase or decrease the visual size of your design on your screen, without changing the actual dimensions in the final output to the laser.

See Zoom and Pan for more information.

View Style

Changing the **View Style** alters the appearance of vector graphics in your Workspace, which can be useful for spotting potential design issues and visualizing the final output from a project.

See View Style for more information.

Toggle Side Panels

`F12`

Toggle off to automatically disable all currently-enabled windows. Toggle back on to re-enable the same set of windows.

See Customizing the LightBurn Window for more information.

Windows and Toolbars

Art Library

Stores artwork, to enable quick access when designing.

See Art Library for more information.

Arrange and Arrange (Long)

Provide tools for positioning graphics, relative to other graphics or positions in your Workspace.

See Arrange Menu for more information.

Modifiers

Contains tools for adjusting, combining, and arraying graphics.

See Modifiers Toolbar for more information.

Camera Control

Provides options for capturing, adjusting, and displaying an overlay image of your laser's work area in the LightBurn Workspace.

See Cameras / Overlays Window for more information.

Console

Displays messages from your laser's controller and commands sent to it by LightBurn. You can also input direct commands to your laser through this window.

The Console window functions are only available when LightBurn is directly connected to a GCode-based laser.

See Console Window for more information.

Cuts / Layers

Shows the list of layers and operations in your design. Select an entry on this list to edit select settings for that layer directly in the **Cuts / Layers Window**, or double-click to open the Cut Settings Editor, where additional options are presented.

See Cuts / Layers Window for more information.

Color Palette

Contains colors representing twenty-nine different layers, and two tool layers, which you can select and assign to objects in your Workspace.

See Color Palette for more information.

Docking

Provides tools to move selected objects left, right, up, or down, until they meet the edge of the Workspace, or another object.

See Docking for more information.

File List

Lists the jobs and estimated run times of files physically hosted on DSP controllers.

See File List Window for more information.

Laser

Serves as the hub for setting up and selecting a laser, **framing** and **starting** jobs, determining job positioning, and more.

See Laser Window for more information.

Material Library

Stores preset cut settings for different operations and materials, so you can quickly reapply them later.

See Material Library for more information.

Main

Contains tools for **saving** and **opening** files, **importing** artwork, **copying** and **pasting** graphics, adjusting view, and more.

See Main Toolbar for more information.

Modes

Contains buttons to open dialog windows or wizards for advanced lasering processes.

See Modes Toolbar for more information.

Move

Provides tools to jog your laser, set a custom User Origin, or view, save, and recall machine coordinates.

See Move Window for more information.

Numeric Edits

Contains fields to adjust the scale, position, and rotation of selected objects.

See Numeric Edits Toolbar for more information.

Shape Properties

Displays editable properties of selected objects. Properties vary based on the type of object currently selected.

See Shape Properties Window for more information.

Text Options

Presents options to adjust attributes of **text objects**, including, **font**, size (**height**), spacing (**HSpace** and **VSpace**), and more.

See Create and Edit Text — Text Options Toolbar for more information.

Tools

Contains tools for creating and editing graphics.

See Creation Toolbar for more information.

Variable Text

Provides options for controlling and incrementing **Variable Text** output. Variable Text allows you to use special codes in text entries that will be substituted for something else when a project is sent to your laser.

See Variable Text for more information.

Workspace / Edit Window

The **Workspace**, or **Edit Window**, is the area where you create, adjust, modify, and arrange artwork in LightBurn.

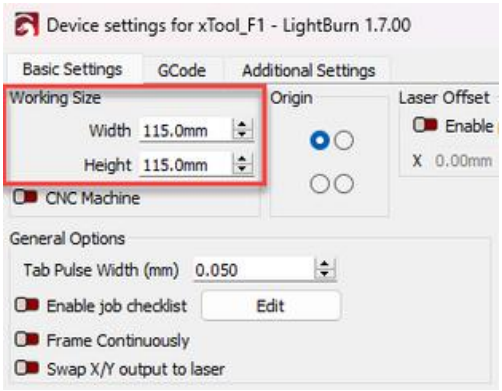
LightBurn sets the size of the border and grid within the Workspace to the machine-specific laser cutting and marking area. Machines equipped with a camera can capture the machine's working area and update the Workspace background image.

The objects arranged and displayed within the Workspace will be processed based on user-configured operations that are established for each artwork element using the Color Palette, Cuts / Layers Window, and Cut Settings Editor.

Customizing the Workspace Appearance

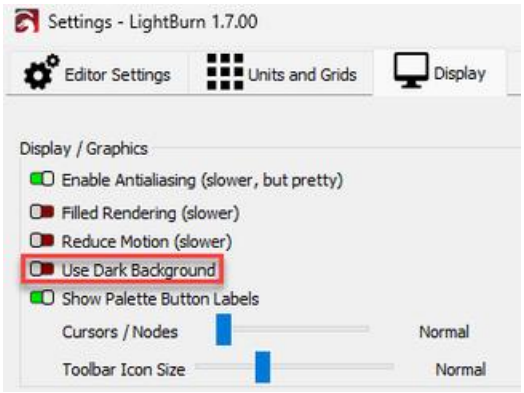
Changing the Size of the Workspace

Change the size of the Workspace in the Device Settings window by clicking on the  icon in the Main Toolbar or going to Edit → Device Settings → **Dimensions / Units** → **Working Size**. Adjust the Working Size **Width** and **Height** entries to match the manufacturer-provided values.



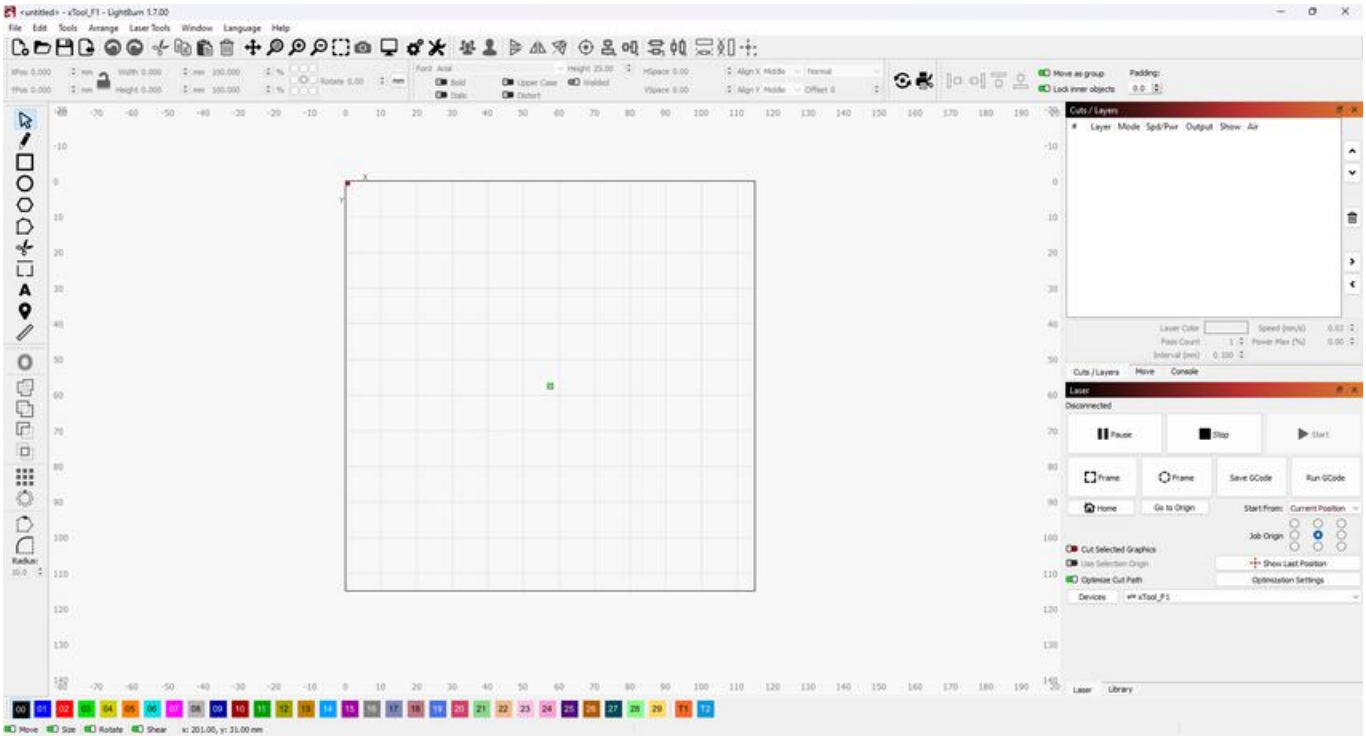
Enabling Dark Background

Switch from the default light background to a dark background in the LightBurn Settings window by clicking on the icon in the Main Toolbar or by going to Edit → Settings → **Display** → **Display / Graphics** → **Use Dark Background**.



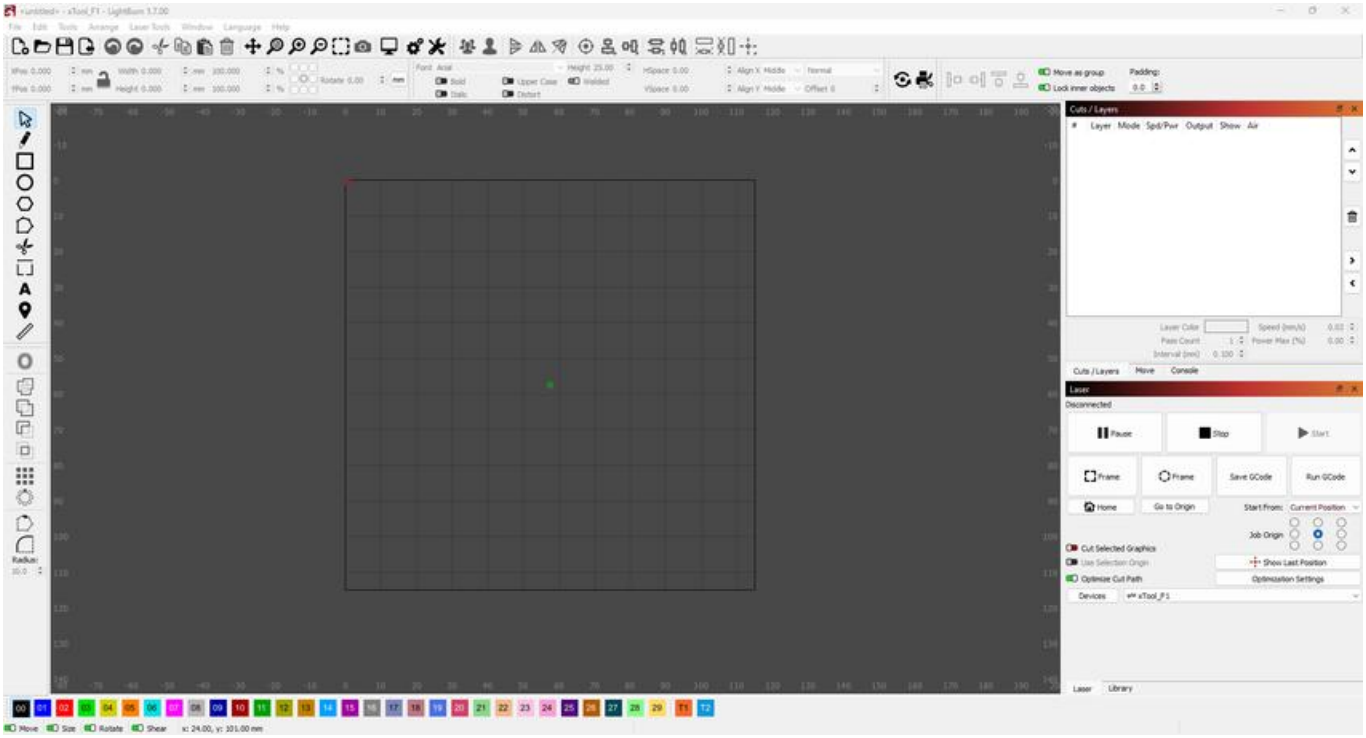
Background Examples

Default Light Color Background



Type	Color
Hex	#F8F8F8
RGB	248,248,248
CMYK	0,0,0,2.7
HSV	0,0,97

Dark Color Background



Type	Color
Hex	#484848
RGB	72,72,72
CMYK	0,0,0,71.8
HSV	0,0,28

Customizing the Workspace Background

LightBurn Camera Setup

Machines equipped with a camera can capture an image of the work area and display that behind the Workspace grid to aid in artwork positioning.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=ONnmCaoUkLQ

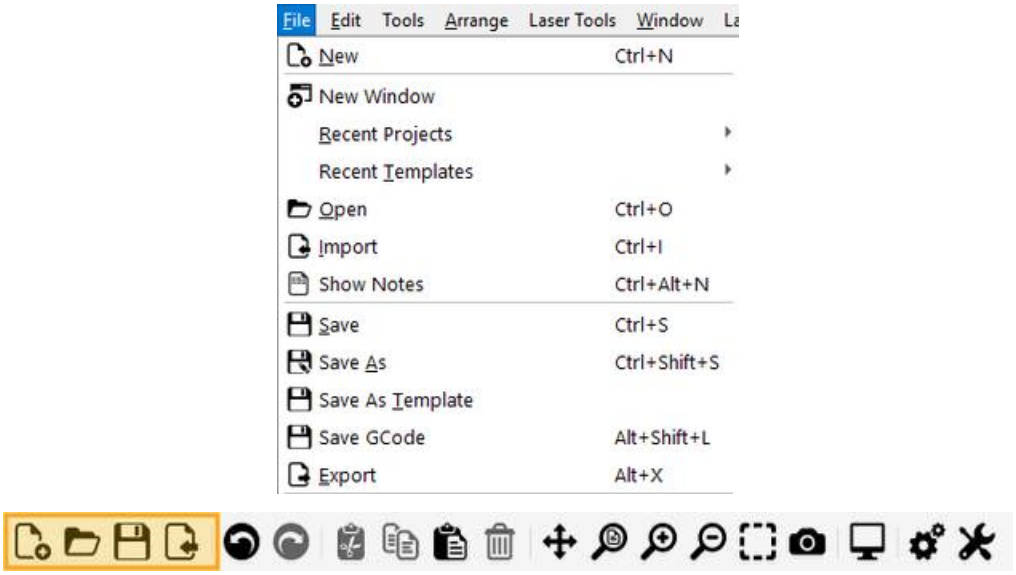
Alternatively, the material being cut or marked can be drawn within the Workspace as a Tool layer with the available LightBurn design Creation tools for similar element alignment and laser framing operations.

Essential Functions

File Management

Use the file management tools to start **New** projects, **Save** projects you're working on, **Open** previously saved projects, and **Import** or **Export** artwork in alternative file formats.

Most file management options are available in the Main Toolbar, with a few additional options available in the File Menu.



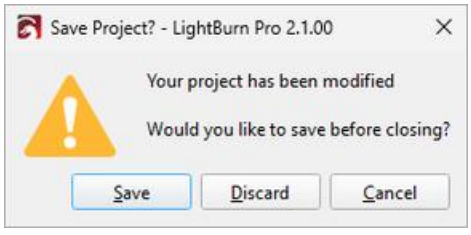
New

`Ctrl`/`Cmd`+`N`

Clears your current LightBurn project and starts a new, blank project.

Opening a new project does not change any Settings, window customizations, and special modes active from the last opened project.

If you have an existing project open with unsaved changes, LightBurn will ask you if you'd like to Save before starting the new project, click **No** to discard the changes, or **Cancel** to abort the process.

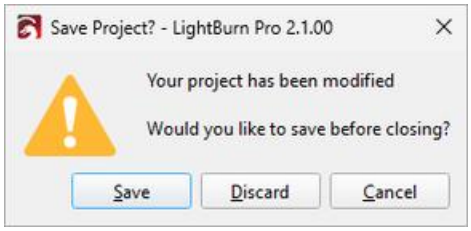


Open

`Ctrl`/`Cmd`+`O`

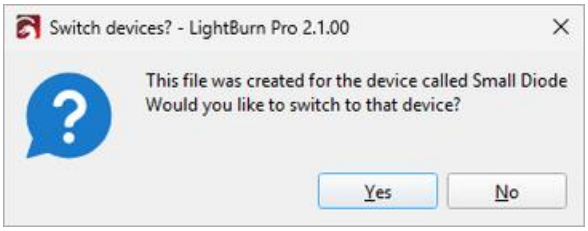
Opens a `.lbrn` or `.lbrn2` project file previously stored on your computer.

If you have an existing project open with unsaved changes, LightBurn will ask you if you'd like to Save before Opening the project, click **No** to discard the changes, or **Cancel** to abort the process.



If you have multiple device profiles, and the device you were using when you Saved the file is not currently selected, LightBurn will ask if you'd like to automatically switch to that device profile.

Click **Yes** to switch or **No** to open the file with your current device.



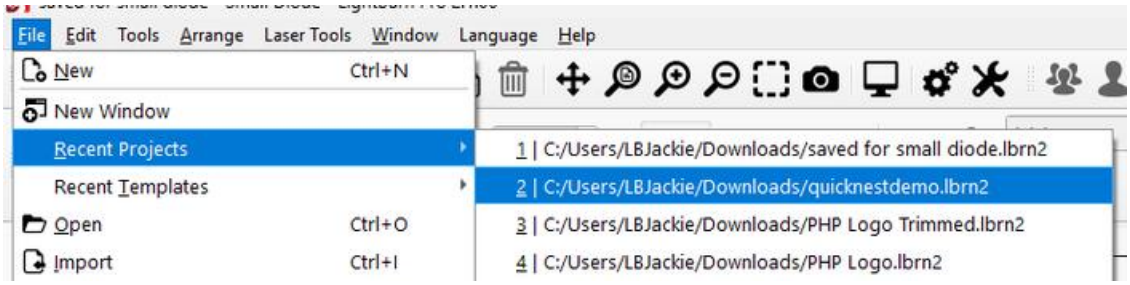
Click on the dropdown in your system's file browser to show all supported file types, including supported image and vector file types.

> **Warning**
>
> Selecting a supported image or vector file instead of a project file will create a new LightBurn project, replacing the currently loaded project. Use Import to bring artwork files or designs into an existing project.

Recent Projects

Hover over **Recent Projects** in the File Menu to open a submenu containing a list of projects you've recently worked on and **Saved**.

Click any file in the list to **Open** it.

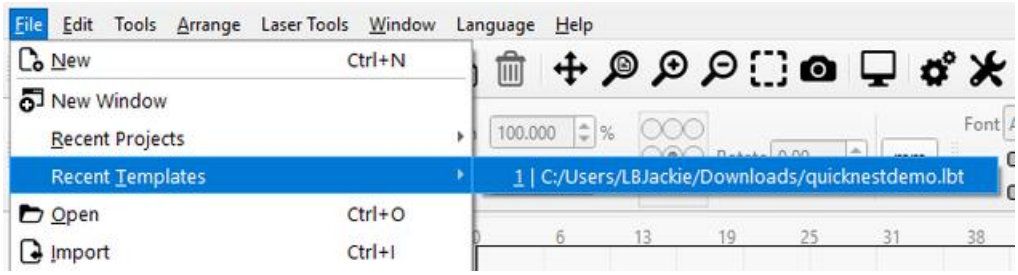


Recent Templates

Hover over **Recent Templates** in the File Menu to open a submenu containing a list of Template Files you've recently worked on and **Saved**.

Click any file in the list to **Open** it.

Read More



Save

`Ctrl`/`Cmd`+`S`

> **.lbrn and .lbrn2**
>
> The **Save** and **Save As** commands offer the option of saving as `.lbrn2` (default) or `.lbrn` (LightBurn Legacy Project).
>
> Unless you need to maintain compatibility with extremely old versions of LightBurn or need an easily human-readable file, we recommend leaving the file type set to `.lbrn2`.

Saves your current project in `.lbrn` or `.lbrn2` format.

- If you haven't Saved it previously, a dialog window will open where you can name it and choose the location to Save it.
- If you have already Saved a copy, the existing copy will be immediately overwritten with any new changes you've made.

Network and Cloud Storage

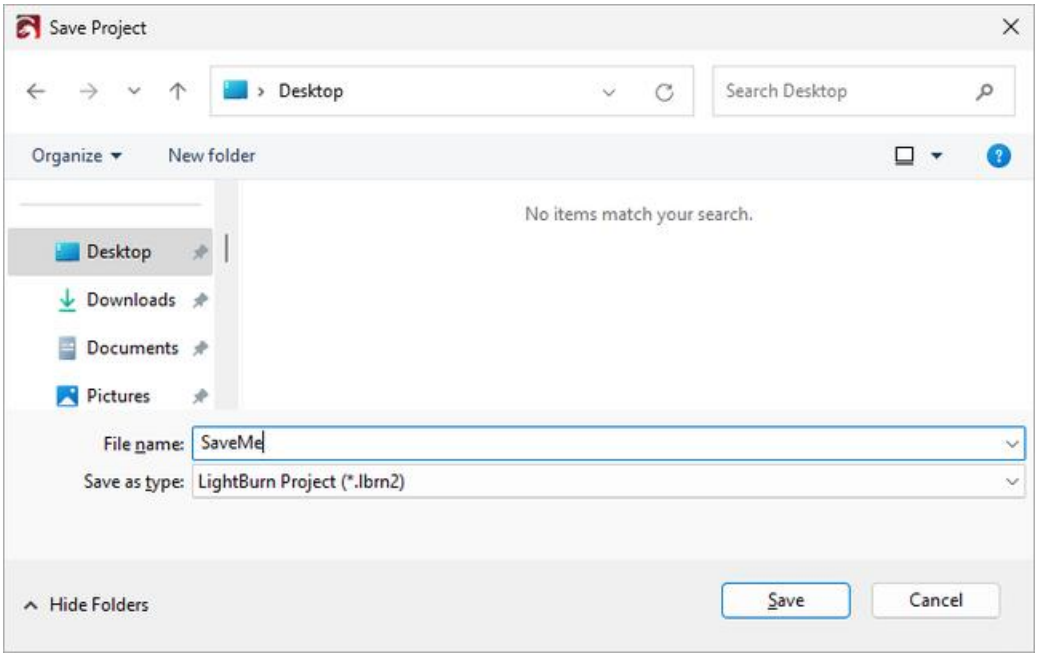
Saving your files to cloud storage drives with active syncing services like Dropbox, Google Drive, iCloud, and OneDrive can potentially cause problems. If these services happen to sync while a file is in active use, there is a chance they will corrupt the file. This is more of an issue with configuration files (such as settings, Material Libraries, and Art Libraries) than with your project files. You can use these cloud storage drives with your LightBurn files, but be aware that there's potential for problems.

Network drives, such as a shared folder on a different computer on your network, don't have the same risk of file corruption and don't cause problems.

Save As

`Ctrl`/`Cmd`+`Shift`+`S`

Rather than immediately overwriting your existing file, **Save As** opens a dialog window where you can save a renamed copy to a new file, leaving the current file unchanged.



Save As Template

Save As Template opens a dialog window where you can save a copy of your file as a **LightBurn Template**. Template files were introduced in LightBurn 2.1 and are intended to make it easier to reuse and customize designs.

Unlike ordinary LightBurn project files, a LightBurn template file saves all of your cut settings, not just the ones used in that project. This can be useful for creating pre-configured template files to share with others, with layers already set up with particular cut settings.

Template files open as new, unsaved projects, so when you save them, the template file isn't overwritten.

Read More

Save Machine File

This option will change depending on the type of laser you're using — it may say **Save GCode** (for GRBL, Smoothieware, or Marlin controllers), **Save RD File** (Ruida), **Save OUT File** (TopWisdom), or **Save UD5 File** (Trocen).

All options save your current project as a machine-ready file, in the appropriate format for your type of laser. You can transfer this file to your machine to run it without ever having to directly connect using LightBurn.

See Laser Window — Save Machine Files for more information.

Import

`Ctrl`/`Cmd`+`I`

Loads graphics from a variety of file formats into LightBurn. Click on the dropdown in your system's file browser to filter for supported image and vector file types.

LightBurn can **Import** the following formats:

Compatible Vector / Mixed Formats

- `.ai` — Adobe Illustrator
- `.svg` — Scalable Vector Graphics
- `.dxf` — AutoCAD Drawing Exchange Format
- `.pdf` — Adobe Portable Document Format
- `.plt` / `.hpgl` — Plotter / Hewlett-Packard Graphics Language
- `.gc` / `.gcode` / `.nc` - GCode files
- `.lbrn` / `.lbrn2` / `.lbt` - LightBurn files (imports paths and images, but leaves out other data such as cut settings)
- `.mage` - MillMage files (imports paths and images, but leaves out other data such as operations)
- `.rd` / `rld` - RDWorks files
- `.sc` - SignCut files
- `.scpro2` - SignCut Pro 2 files

Importing Text

If you import vector files not made in LightBurn that contain text, make sure that any fonts used in the file are installed on your computer, or that the text has been converted to paths using whatever tools are available in the other program. LightBurn is not able to import embedded fonts. If LightBurn is unable to import anything from your file, the status bar will display **Nothing Imported**.

Different programs use different terms for this process, but terms such as welding, convert to outline, and convert to path are common.

Importing GCode Files

Although LightBurn is able to import GCode files, it will only import the geometry saved within the file. Speed and power settings will not be imported, and filled shapes will be imported as the fill lines, not the outline of the shape.

Use caution with AI-generated files

If you use AI/LLM tools to generate files, please be aware that the files generated may not conform to standards for the file type you're importing and you may experience failed imports or unexpected results.

Compatible Image Formats

- `.png` — Portable Network Graphics
- `.jpg` / `.jpeg` — Joint Photographic Experts Group format
- `.bmp` — Windows Bitmap
- `.gif` — Graphics Interchange Format
- `.tif` / `.tiff` — Tagged Image File Format
- `.webp` - WebP

Quick Tip: Drag and drop directly into LightBurn

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=oQQK_ES4Xgo

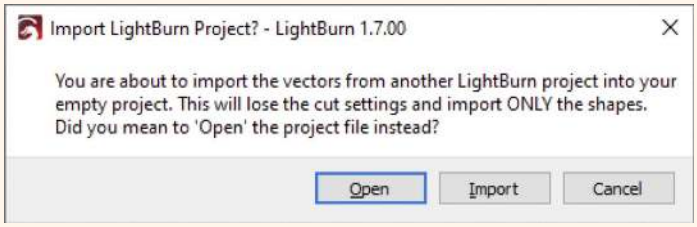
Importing `.lbrn` or `.lbrn2` files

It is also possible to **Import** `.lbrn` or `.lbrn2` files — however, Importing these types of files does not load the cut settings or any other project-specific information saved to the file. Only the graphics will be loaded.

In order to load graphics along with all other settings, you must use **Open** instead.

LightBurn will warn you if you attempt to Import a `.lbrn` or `.lbrn2` file.

Click **Open** to load the graphics and all other settings, **Import** to load only the graphics, or **Cancel** to abort the process.



Compatible PCB File Formats

Starting with LightBurn 2.2, LightBurn can import Gerber files for working with PCBs.

Compatible file extensions include:

- `.gbr`
- `.grb`
- `.gbx`
- `.ger`
- `.pho`
- `.gtl`
- `.gbl`
- `.gts`

- `.gbs`
- `.gto`
- `.gbo`
- `.gtp`
- `.gbp`
- `.gko`
- `.gm1`
- `.g1`
- `.g2`
- `.g3`
- `.g4`
- `.g5`
- `.g6`
- `.g7`
- `.g8`
- `.g9`
- `.g1l`
- `.g2l`
- `.g3l`
- `.g4l`
- `.g5l`
- `.g6l`
- `.g7l`
- `.g8l`
- `.g9l`
- `.drl`
- `.xln`
- `.exc`
- `.xnc`
- `.gbrjob`

Compatible 3D Model File Formats

Starting with LightBurn 2.2, you can import 3D models to create depth maps for use with LightBurn's 3D Sliced image engraving mode as well as MillMage's relief carving.

Supported 3D model file formats:

- `.stl`
- `.3mf`
- `.amf`

- `.obj`
- `.off`
- `.ply`

Export

`Alt`/`Option`+`X`

Exports currently selected graphics to your selected file format. If there is nothing in your current selection, the entire contents of your current project will be Exported.

Vector/Mixed Formats

The following formats are always available when exporting graphics:

- `.ai` — Adobe Illustrator
- `.svg` — Scalable Vector Graphics
- `.dxf` — AutoCAD Drawing Exchange Format

When exporting an SVG file, any layers that have Output turned off will be included but hidden.

Image Formats

The following formats are available only when you have a single image selected:

- `.png` — Portable Network Graphics
- `.jpg` / `.jpeg` — Joint Photographic Experts Group format
- `.bmp` — Windows Bitmap

Exporting in one of these formats does not save cut settings or other project-related settings.

Exporting an image does not include any image adjustments or image processing selections. To export an image including these adjustments, see Save Processed Bitmap.

About Template Files

Template files are LightBurn files with a different file format (`.lbt`) and a few special features.

- Opening a LightBurn template file creates a new, unsaved file, but otherwise opens like any other LightBurn file — all of your shapes, images, cut settings, notes, and other settings that would ordinarily be saved and loaded as part of a LightBurn file are included. This prevents you from accidentally saving over your template.
- When you save a template, all of your cut settings are saved as part of the file, not just the ones that are used. This can be really useful if you want to set up a file with pre-configured settings. To do this, create a shape on each layer you want to pre-configure, adjust the cut settings for those layers, and delete the shapes if you don't want them included in the template.
- To update a template:
  1. Open the template file.

2. Make your modifications.
3. Choose Save as Template.

We'll have a guide with more information on using templates out soon, but some of our staff and users have been using them to save common rotary configurations or to create files for workshops with layers ore-configured for common operations.

Related Topics

- Clipboard Tools
- Undo/Redo
- File Menu

Selection Tools

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=mTpR42AsOfk

In order to modify graphics, you must first select them. The **Select** tool is the primary means of selecting objects, but LightBurn also includes several additional options for making selections based on their type, size, layer, and more.

Making Selections

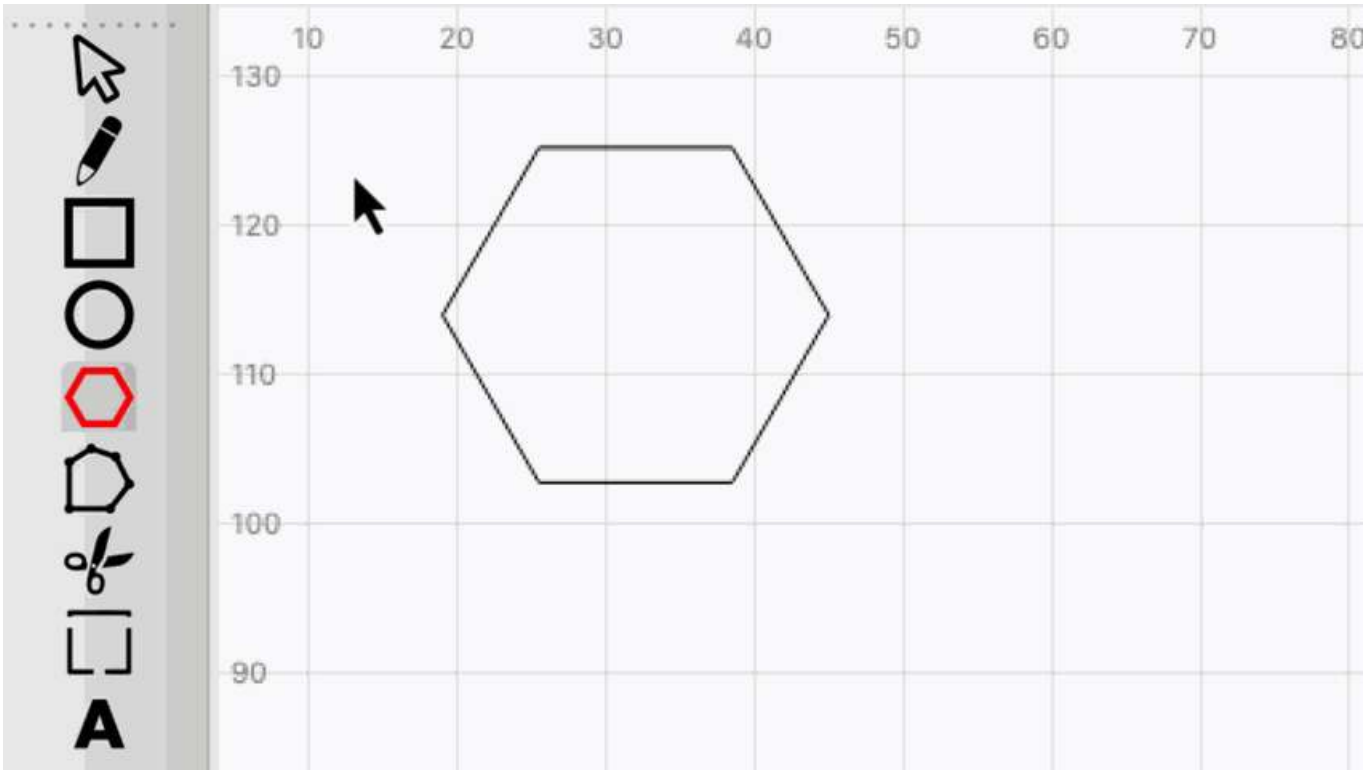
The **Select** tool is active by default. To reactivate it after using a different tool, click the cursor icon in the Creation Toolbar, or press the `Esc` key.



Click Selection

With the **Select** tool enabled, click any object in your Workspace to select it.

> **Tip**
>
> To increase or decrease the distance your cursor must be from an object in order to select it when you click, adjust Click-Selection Tolerance in your Settings/Preferences.

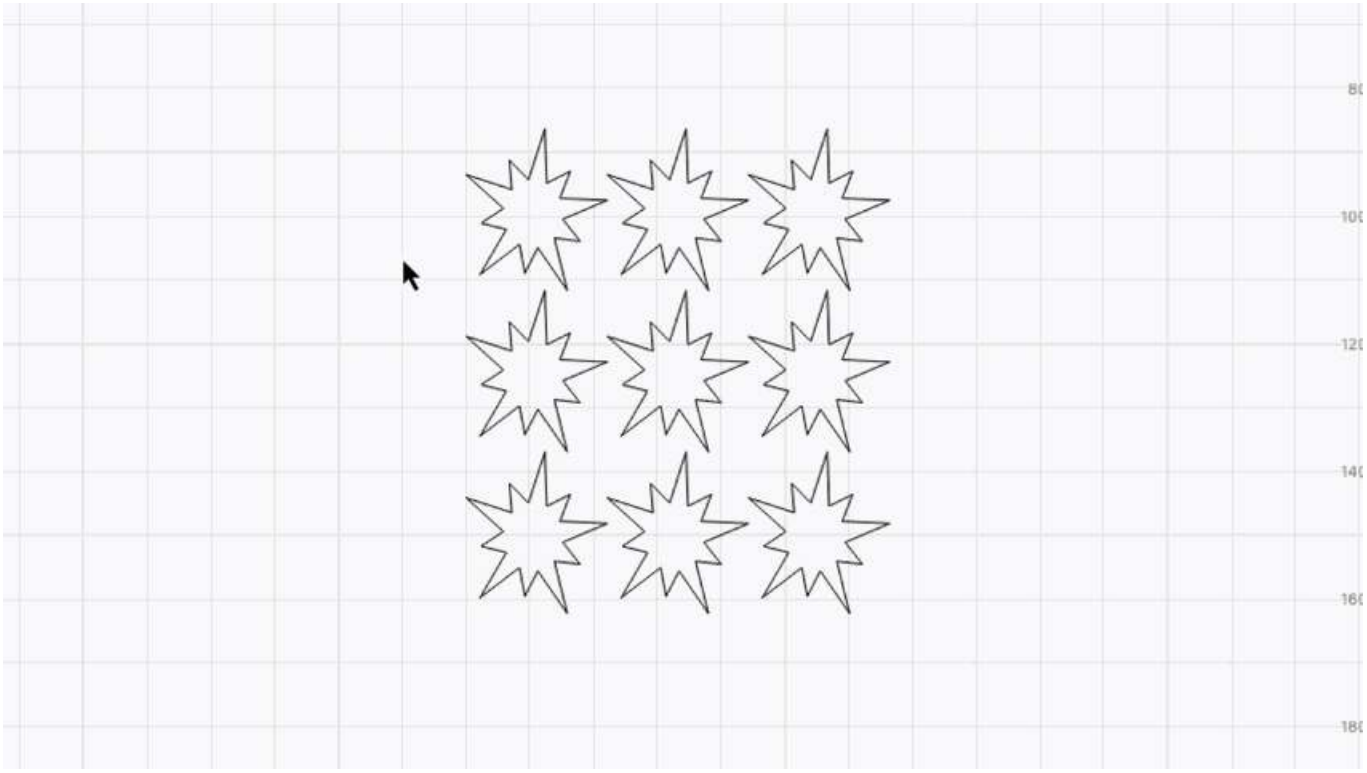


Box Selection

Select several shapes at once using one of two box selection options.

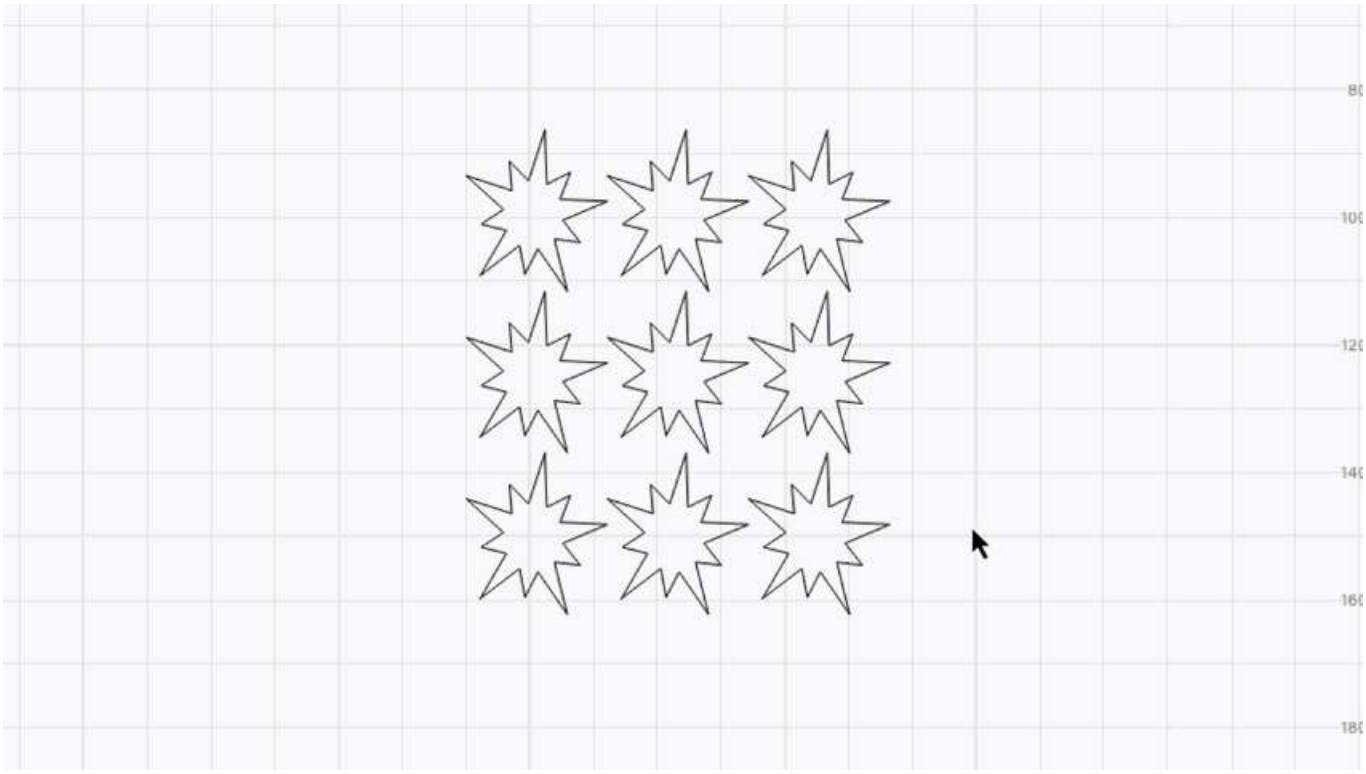
Enclosing Selection

With the **Select** tool enabled, click, hold, and drag to the right to create a red **Enclosing Selection** box. Release to select all objects that are fully contained within the box.



Crossing Selection

Click, hold, and drag to the left to create a green **Crossing Selection** box. Release to select all objects that are fully contained or crossed by the lines of the box.



▶ Watch the video on YouTube: https://www.youtube.com/watch?v=Fcya6slPnUc

Selection Modifiers

To supplement both click and box selection, LightBurn supports the following modifier keys.

While clicking or box-dragging:

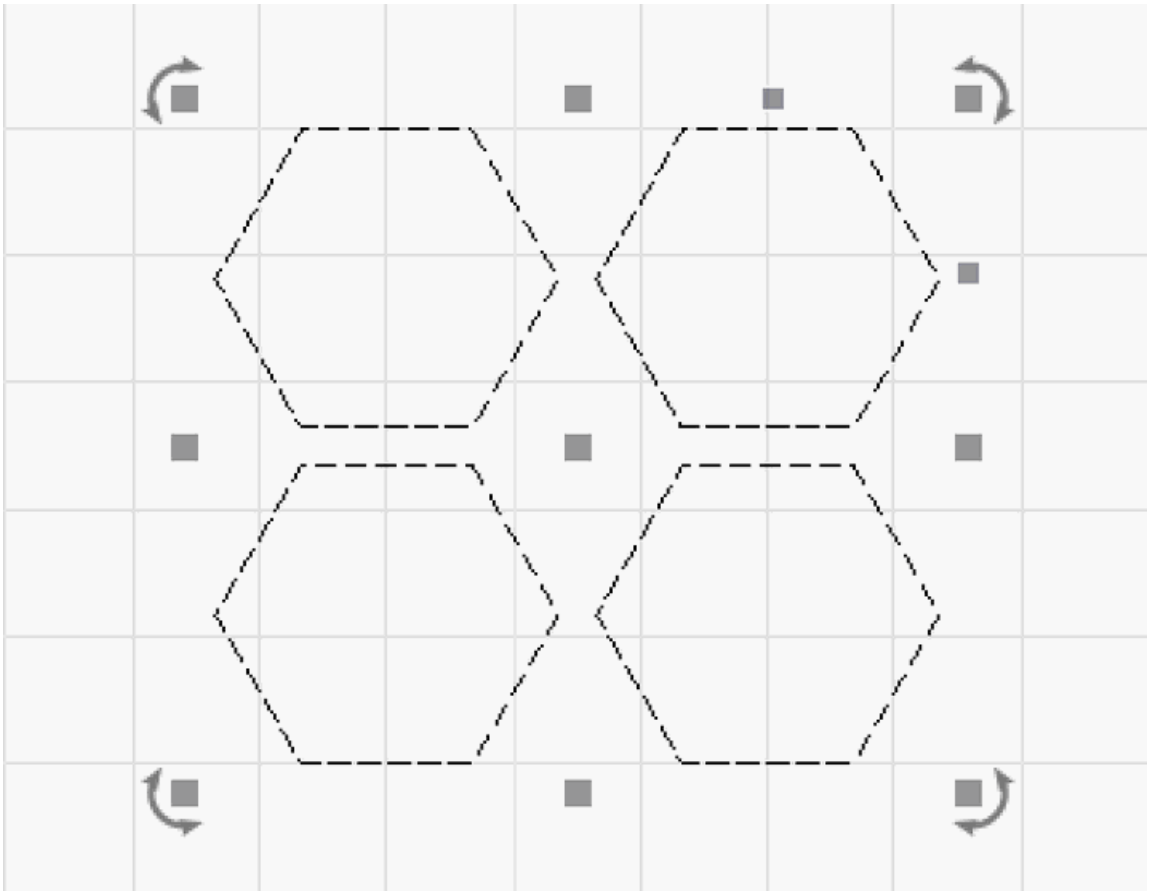
- Hold `Shift` to add a new object or objects to your current selection.
- Hold `Ctrl` / `Cmd` to toggle the selection status of an object or objects, deselecting selected objects, or selecting unselected objects.
- Hold both `Ctrl` / `Cmd` and `Shift` to remove an object or obects from your current selection.

Additional keyboard shortcuts:

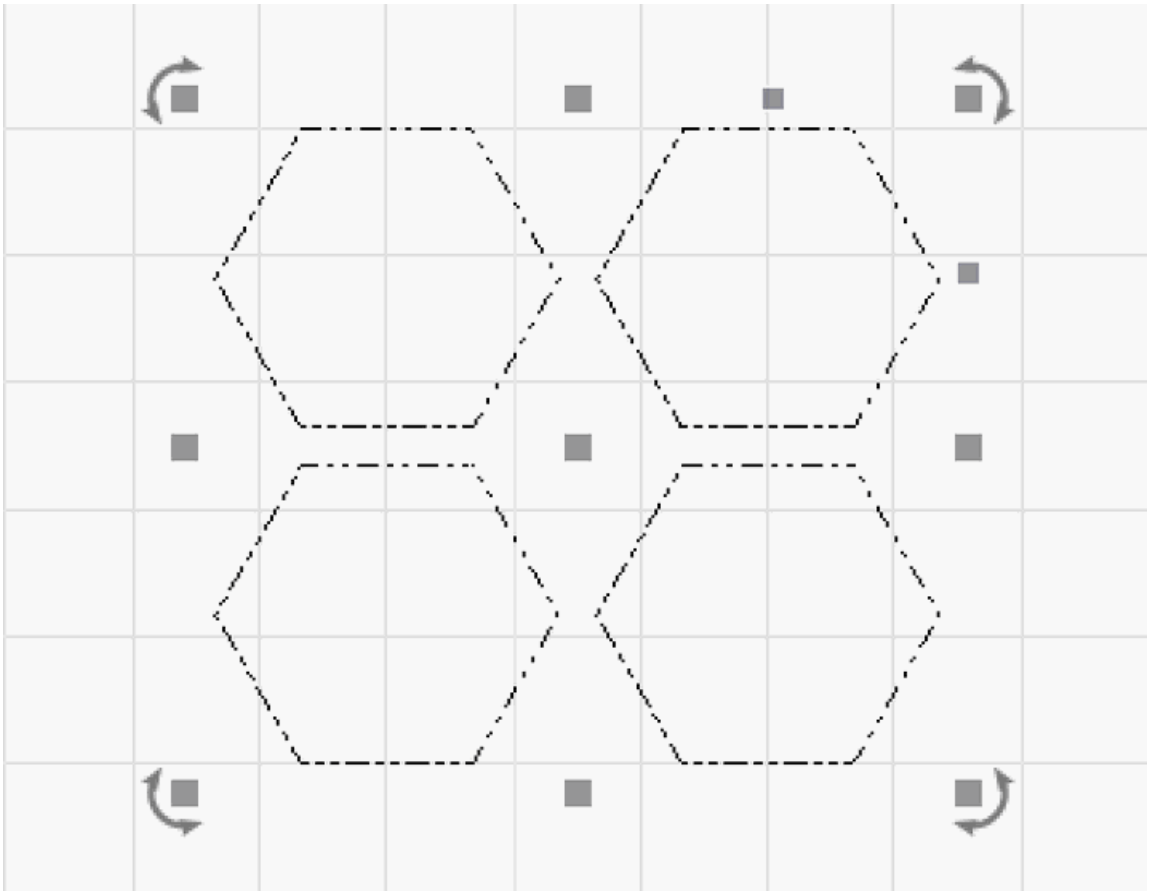
- Press the `Esc` key, or click in any open area in your Workspace to remove all objects from your selection.
- Press `Tab` to cycle your selection sequentially through objects or Grouped objects in your Workspace. Sequence is determined by the order in which the objects were created.
- Press both `Tab` and `Shift` to cycle your selection sequentially through objects or Grouped objects in your Workspace, in reverse order.

Selection Indicators

Selected objects are drawn with an animated dash pattern instead of solid lines. If you select Grouped objects, you'll see a dot-dot-dash pattern instead.



Ungrouped Objects Selected



Grouped Objects Selected

Several options for modifying or manipulating objects are available once they are selected:

- The Transform Control handles for Sizing, Moving, Skewing, or Rotating the selection appear.
- The size and position of your selection is shown in the Numeric Edits Toolbar.
- Other controls are enabled depending on the type of object you've selected:
  - If you select a Text Object, the Text Options Toolbar becomes active.
  - If you select a shape made by the Rectangle, Ellipse, or Polygon tools, shape-specific adjustment options are presented in the Shape Properties Window.
  - If you select an Image, image adjustment options are presented in the Shape Properties Window.

Additional Selection Tools

Several other selection options are available in the Edit Menu.

Select All

`Ctrl` / `Cmd` + `A`

Selects all graphics in your project.

Invert Selection

`Ctrl` / `Cmd` + `Shift` + `I`

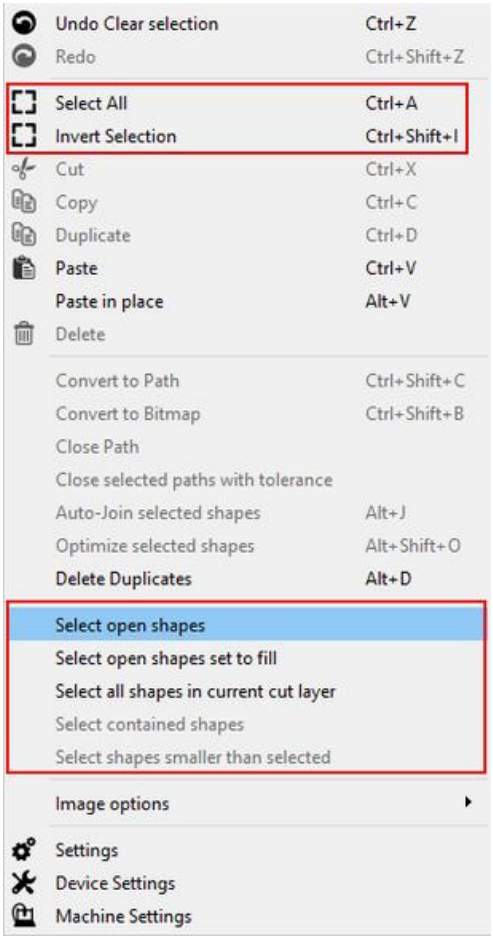
Deselects all objects that are currently selected and selects all objects that are not currently selected.

Select Open Shapes

Selects all shapes in your project that are open, meaning they are not paths that form a closed, continuous loop. See Open vs. Closed Shapes for more information on open shapes.

Select Open Shapes Set to Fill

Selects all shapes in your project that are open, and assigned to a layer set to Fill or Offset Fill Mode. LightBurn is unable to fill open shapes — this tool is a useful option for identifying them.



Select All Shapes in Current Layer

Selects all shapes assigned to the active layer in the Cuts / Layers Window.

You can perform the same action by holding `Shift` while clicking on a layer in the Cuts / Layers Window.

Select Contained Shapes

Selects all shapes contained by the currently selected shape. To use this tool, the selected shape must be closed — a path that forms a continuous loop. The containing object remains in the resulting selection.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=0xpDJJiPvvo

Select Shapes Smaller Than Selected

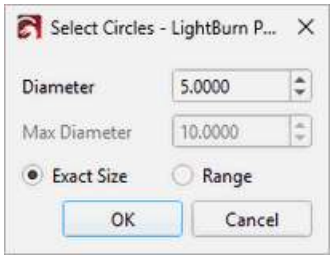
Selects all shapes that are smaller than the currently selected object. The currently selected object remains in the resulting selection.

Select Circles

Selects all circles of a specific size or within a size range. Select **Exact Size** and specify a size, or select **Range** and choose a minimum and maximum size.

This can be particularly useful for parts with a lot of holes for screws or other fasteners.

> **Using Select Circles with Groups**
>
> The **Select Circles** tool works on circles that are part of groups, but please be aware that the circles selected will no longer be grouped.



Troubleshooting

- Grouped objects are treated as single objects when using click or box selection, and any selection modifiers. All objects in a Group must be entirely contained by an **Enclosing Selection** in order to be selected, and if any object in a Group is crossed by a **Crossing Selection**, the entire group will be selected.
- There are a handful of exceptions to the behavior of Grouped objects with the additional selection tools:
  - **Select contained shapes** does not select any contained shapes that are Grouped.
  - When using **Select shapes smaller than selected**, any Grouped objects that are smaller than the selected shape will be selected, and automatically removed from their Group. The objects they were Grouped with will not be selected.
  - When using **Select open shapes** or **Select open shapes set to Fill**, any Grouped objects that are open will be selected, and automatically removed from their Group. The objects they were Grouped with will not be selected.

- If you find that you're getting odd flashing instead of the expected pattern when selecting objects, the most likely problem is overlapping geometry. This is often the result of importing the same file twice, or other design software exporting paths improperly.

  To solve this problem, try deleting duplicate shapes using Edit → Delete Duplicates (`Alt` + `D`).

  For more manual control, you can also try Ungrouping(`Ctrl`/`Cmd`+`U`) and deleting individual shapes.

Related Topics

- Group/Ungroup
- Workspace
- Zoom
- View Style
- Open vs. Closed Shapes

Zooming and Panning

Use the **Zooming** and **Panning** functions to adjust the visible area in your Workspace while designing and making edits.

Zooming In can make fine details easier to work on, and **Zooming Out** can provide a better view of an overall composition.

Panning allows you to move the area your screen is focused over, sliding the field of view between locations in your Workspace.

Zooming

Zoom In and Out

The most common method to quickly navigate a design is by zooming with the scroll wheel on a mouse, followed by two-finger dragging if you're using a trackpad. By default, rolling it backwards will **Zoom Out**, and rolling it forward will **Zoom In**, toward whatever your cursor is pointing.

These directions can be swapped by going to File → Settings and toggling on **Invert Mouse Wheel Zoom Direction**.

However, there are many ways to Zoom In and Out:

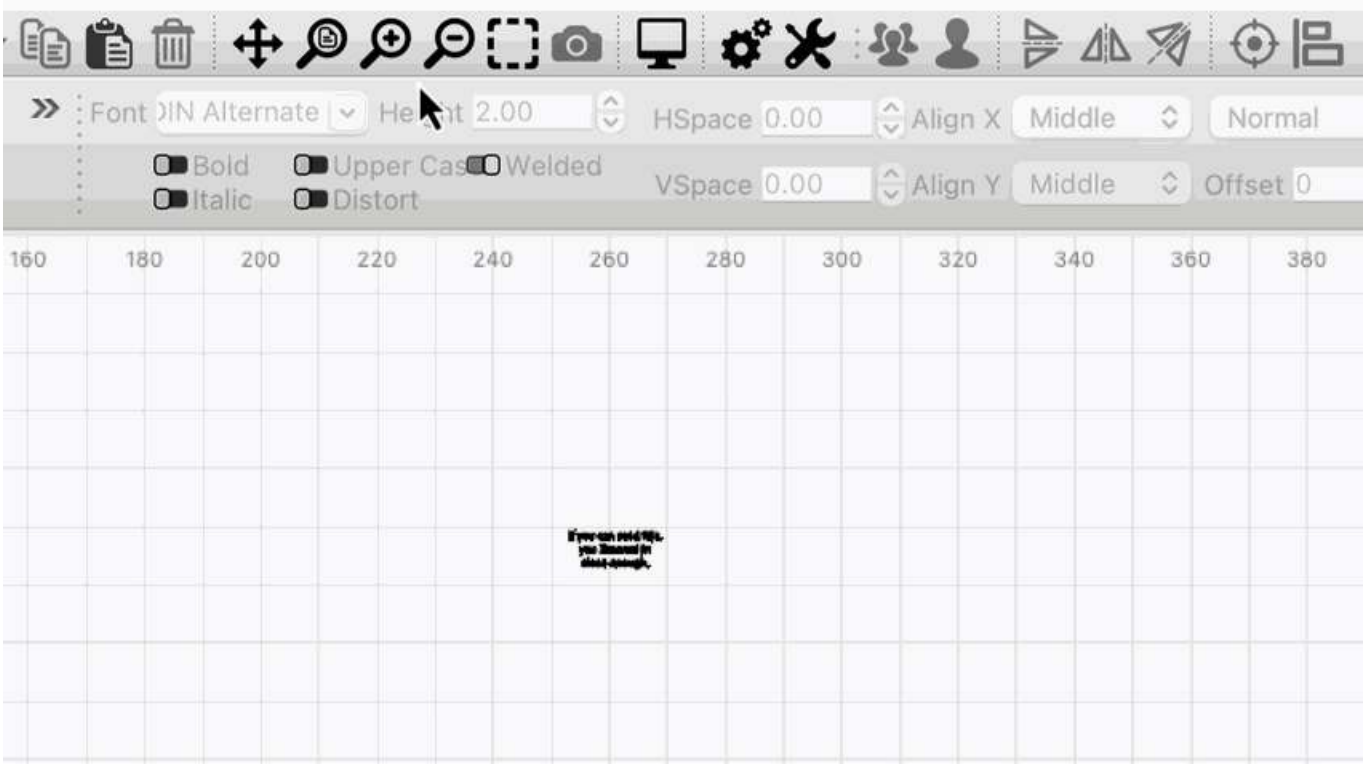
Method	Zoom In	Zoom Out
Roll the scroll wheel on a mouse	Forward	Backward
Slide two fingers on most touchpads	Backward	Forward
Window Menu	Zoom In	Zoom Out
Keyboard shortcuts (Edit Window in focus)	+	-
Keyboard shortcuts (Edit Window not in focus)	Ctrl / Cmd + +	Ctrl / Cmd + -
Main Toolbar		

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=0oaU6Iq5YeA

What does it mean for a window to be in focus?

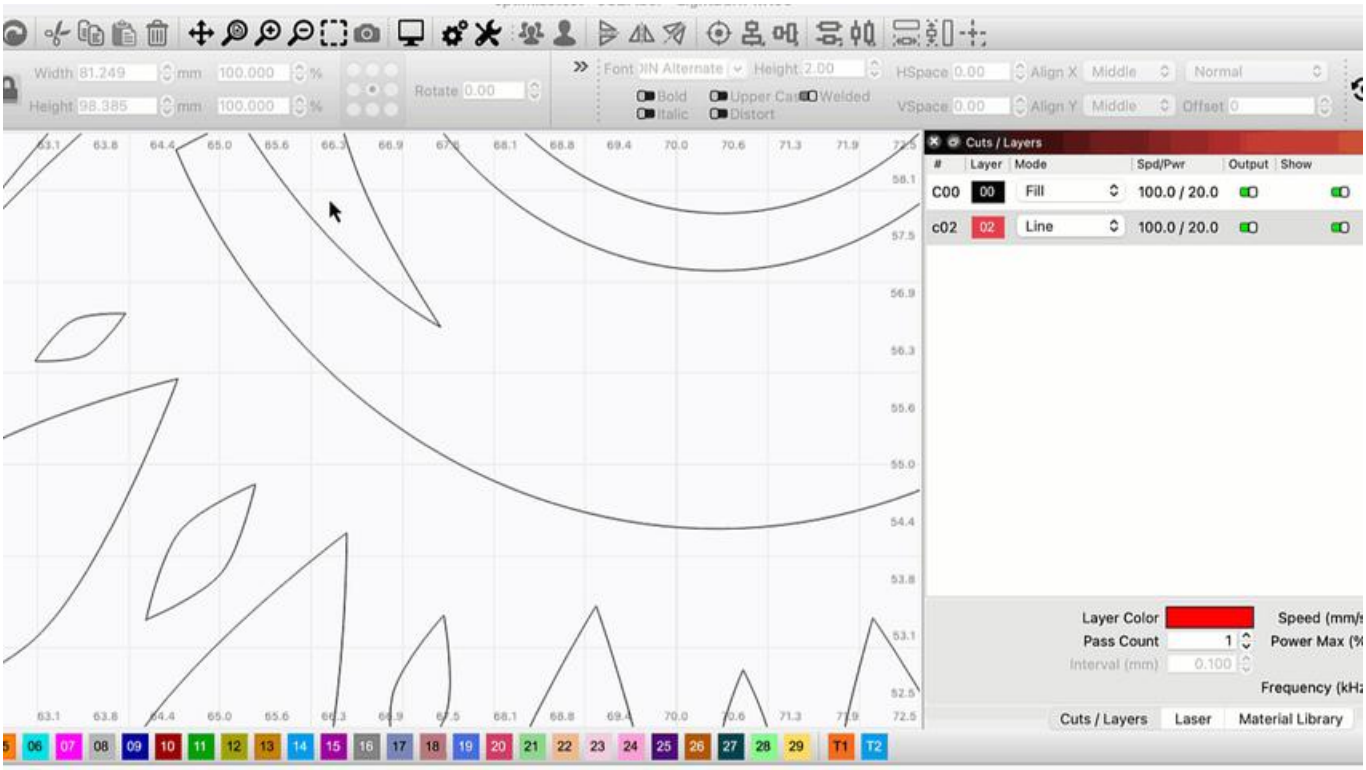
There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.

If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus. Selecting objects in the Workspace will also bring it into focus.



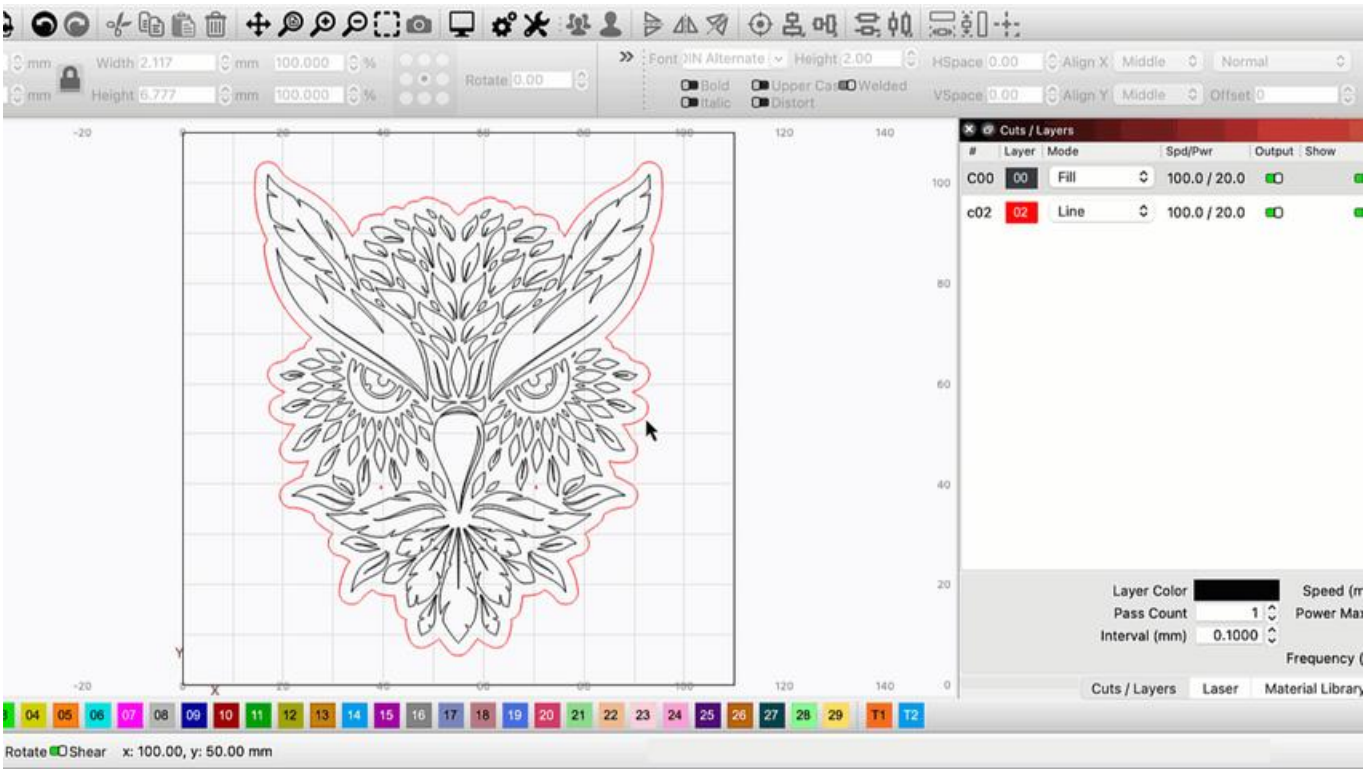
Zoom to Page

To **Zoom** your field of view to show the entire Workspace, go to Window → **Zoom to Page**, press `Ctrl` / `Cmd`+`0`, or click the Zoom to Page button  in the Main Toolbar.



Frame Selection

To **Zoom** your field of view to frame your current selection, go to Window → **Frame Selection**, press `Ctrl` / `Cmd`+`Shift`+`A`, or click the Frame Selection button in the Main Toolbar.



Panning

Panning slides your field of view around the Workspace. There are a few options for enabling Panning mode:

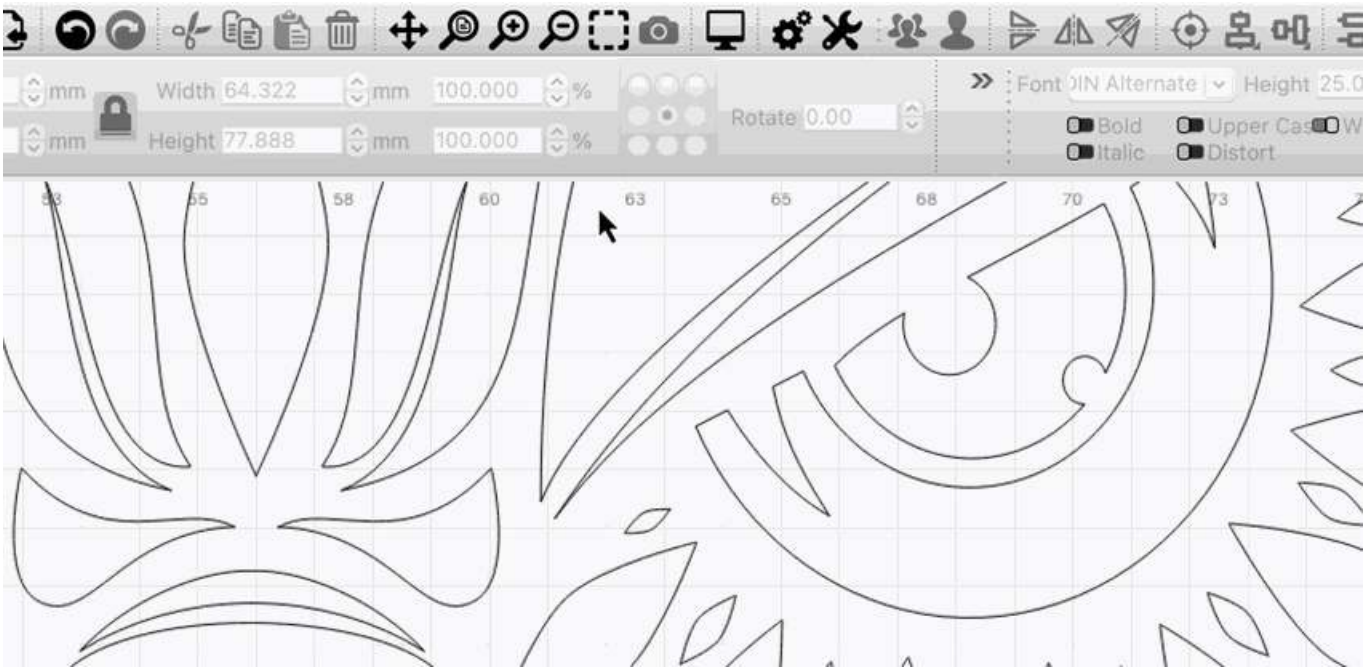
- Click the **Pan / Drag View** icon in the Main Toolbar.
- Press and hold the middle mouse button and move your mouse.
- Hold down `Space` on your keyboard.

When you've enabled Panning mode using any of the above methods, you'll see the mouse cursor change to a hand



, and then you can click and drag the view with the left mouse button.

To exit Panning mode, press the `Esc` key, or enable any other tool.



Related Topics

- Selection
- Main Toolbar
- Window Menu

Undo & Redo

Undo and **Redo** reverse or reapply changes you've made to your project, allowing you to backtrack if you've made edits you regret, or restore the edits you reversed, if that regret was only temporary.

Undo and Redo in LightBurn are unlimited — the list of actions that can be reversed and reapplied is only reset when closing out of the program or opening a new project.

The Undo History window added in LightBurn 2.1 makes LightBurn's unlimited undo list even easier to use.

Using Undo and Redo

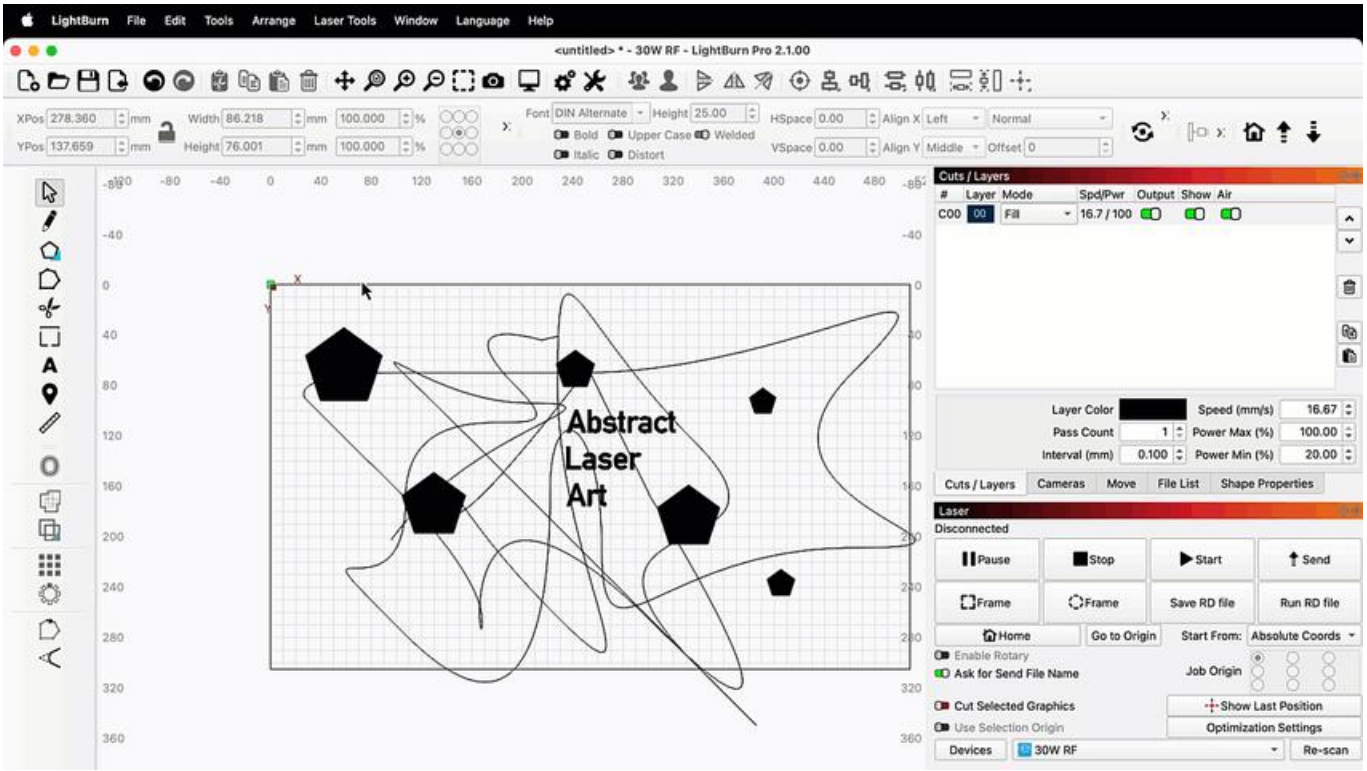
Undo reverses previous actions taken, starting with the most recent action, and proceeding to previous actions with successive Undos.

To Undo, press `Ctrl`/`Cmd`+`Z`, click the Undo icon in the Main Toolbar, or go to the Edit Menu and select Undo.

Redo restores actions previously reversed using Undo, starting with the most recent action Undone, and proceeding to previous reversed actions with successive Redos.

To Redo, press `Ctrl`/`Cmd`+`Shift`+`Z`, click the Redo icon in the Main Toolbar, or go to the Edit Menu and select Redo.

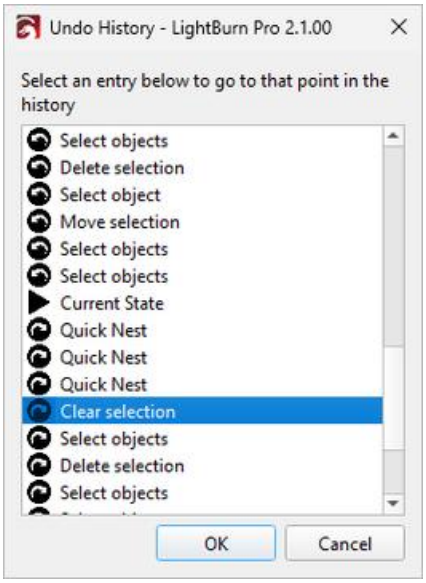
Undo History Window



LightBurn's new **Undo History** feature makes it easier to navigate a long undo list.

Go to Edit → **Undo History** to open the Undo History window. You can scroll through the list and select actions to preview your project as it was at that point in the project's history.

After selecting an action, click **Ok** to jump back to that action, undoing everything you've done since that point.

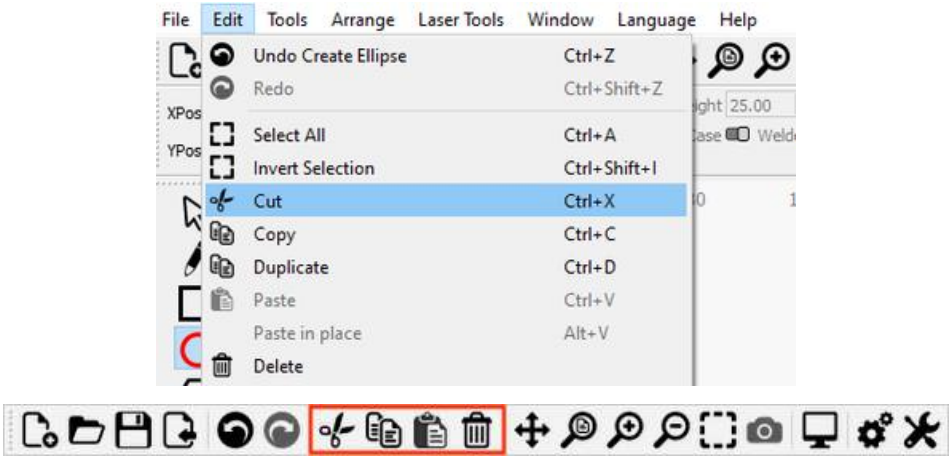


Clipboard Tools

The clipboard commands are tools for copying, inserting, removing, or duplicating objects in your Workspace, and transferring graphics into and out of LightBurn.

The clipboard commands are accessible in the Edit Menu, Main Toolbar, by hotkey, or by right-clicking in your Workspace to open a context menu.

> **Info**
>
> Your computer's clipboard is a temporary storage location used to save items that you copy, so you can paste them to another location shortly after.



Copy

`Ctrl`/`Cmd`+`C`

Saves a copy of selected objects to your clipboard.

Cut

`Ctrl`/`Cmd`+`X`

Deletes selected objects in your Workspace while saving a copy of them to your clipboard at the same time.

Duplicate

`Ctrl`/`Cmd`+`D`

Creates copies of selected objects, directly on top of the existing objects.

Paste

`Ctrl`/`Cmd`+`V`

Inserts objects copied to your clipboard into your Workspace. Pasted objects are centered at the location of your cursor at the time of pasting.

Copy and paste from Adobe Illustrator

Some users have had trouble copying and pasting vector shapes from Adobe Illustrator. If vector shapes copied from Illustrator are being pasted into LightBurn as images rather than vectors, you may need to update your Illustrator settings.

In Illustrator, go to **Illustrator → Settings → Clipboard Handling** and make sure that **On Copy: Include SVG Code** is turned on. You can also export an SVG file from Illustrator and import it into LightBurn.

Drag and drop directly into LightBurn

You can copy text, vector graphics, and images from other design software, and paste them directly into your LightBurn Workspace.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=oQQK_ES4Xgo

Paste in Place

`Alt`/`Option`+`V`

Inserts objects copied to your clipboard into your Workspace and positions them according to their location in the project they were copied from.

Delete 

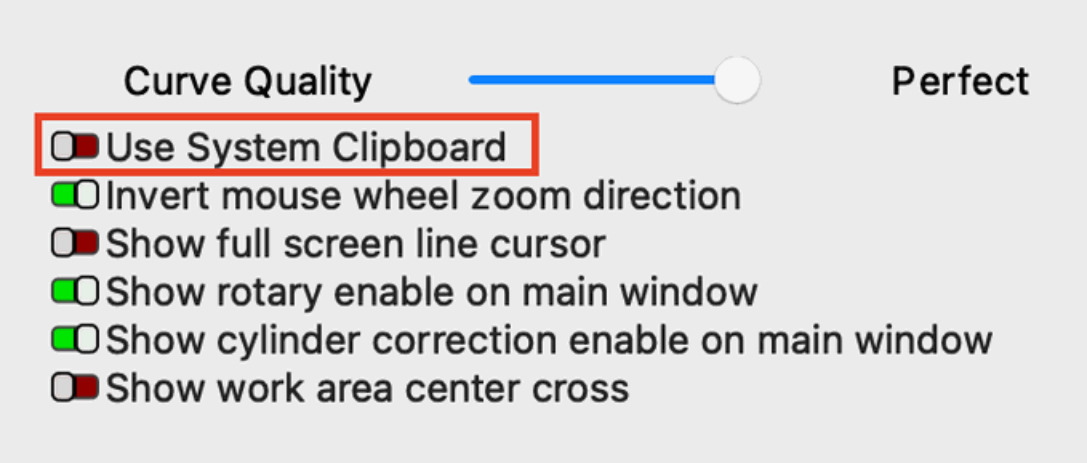
`Del`

Deletes selected objects.

Troubleshooting

- By default, LightBurn uses your system's clipboard. If you're experiencing slow performance when using the clipboard commands, disable **Use System Clipboard** in the Settings/Preferences window.

  With this setting disabled, you'll no longer be able to copy and paste from other programs into LightBurn, or between LightBurn windows.



Related Topics

- Selection Tools
- Undo/Redo
- Zoom
- Edit Menu
- Main Toolbar

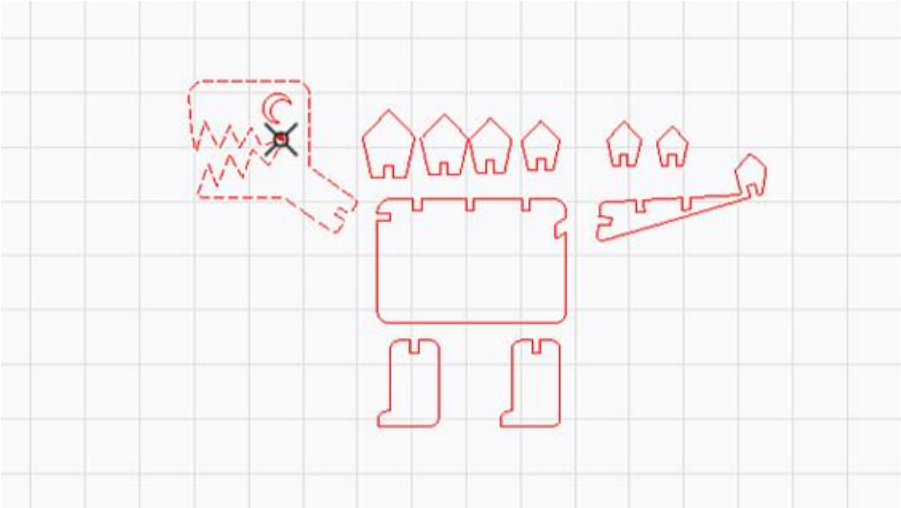
Transform Controls

The **Transform Controls** are tools for directly editing selections in your Workspace by clicking and dragging control handles that alter their position, dimensions, orientation, or shape.

Control Handles

Control handles appear on and around the selection area of any object or objects you select.

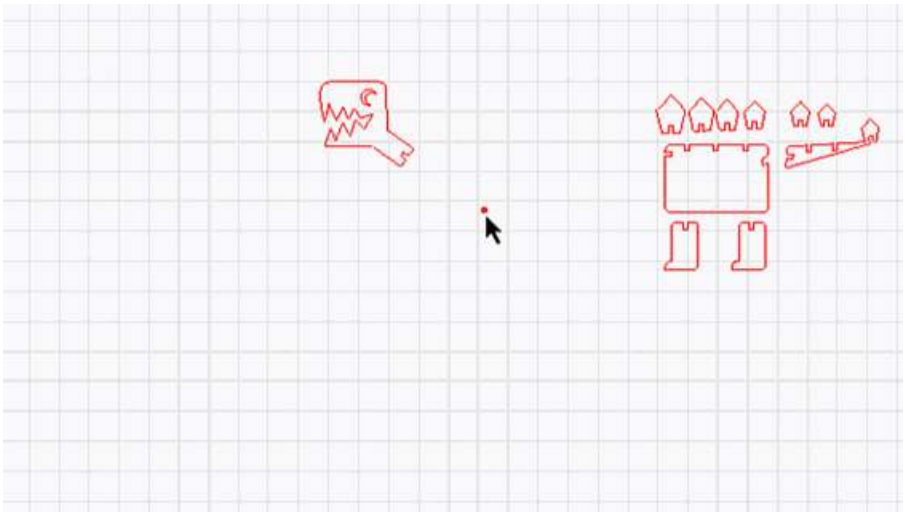
Imagine the selection area as the smallest possible rectangle that can contain every object in your selection. The edges and center of a selection are the same as the edges and center of that imaginary rectangle.



Move

The **Move** handle is a square located in the center of your selection area. When you hover over the Move handle, the four-arrow icon will appear.

- Click, hold, and drag the Move handle to move the selection around your Workspace.
- Release to place the selection in its new position.
- Press Esc before releasing to deselect and return the selection its original position after releasing.

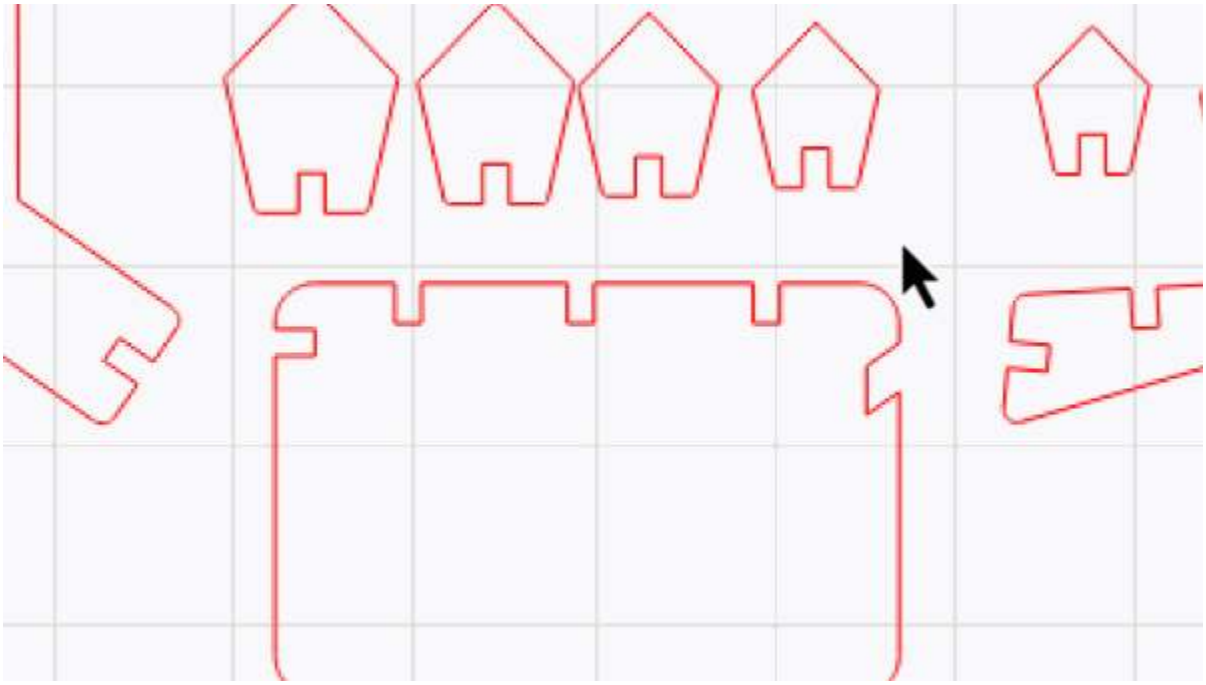


> **Tip**
>
> To precisely adjust a selection's position, use the Numeric Edits Toolbar.

Moving From Snapping Points

Hover over a line, corner, intersection, midpoint, or other node on any shape in your selection, then click, hold, and drag to move all objects in the selection from that point, and snap it to another point on another shape in your Workspace.

See Snapping for more information.



Modifier Keys

- Hold `Shift` while dragging to lock a selection's movement to a 90° or 45° angle from its current position.

- Hold `Alt`/`Option` to show temporary Alignment Guides for selection snapping.
- Hold `Ctrl`/`Cmd` to disable snapping while moving selections.

Move Increment Hotkeys

You can use your keyboard's arrow keys to move objects in your Workspace by set increments.

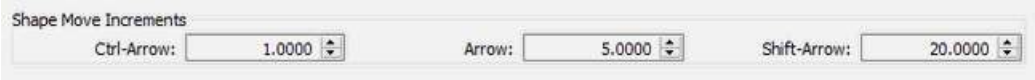
These hotkeys only work when you have objects selected and the **Edit Window** (Workspace) — the area where you create and edit graphics — is in focus.

> **What does it mean for a window to be in focus?**
>
> There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.
>
> If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus. Selecting objects in the Workspace will also bring it into focus.

- The `Left`, `Right`, `Up`, and `Down` arrow keys nudge your selection by **5 mm**, by default.
- Hold `Ctrl`/`Cmd` while pressing an arrow key to nudge your selection **1 mm**, by default.
- Hold `Shift` while pressing an arrow key to nudge your selection **20 mm**, by default.
- Hold `Ctrl`/`Cmd`+`Shift` while pressing an arrow key to nudge your selection **0.1mm**, by default. This value is always 1/10th of the `Ctrl`/`Cmd` nudge increment.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=7T5fuJX6fhI

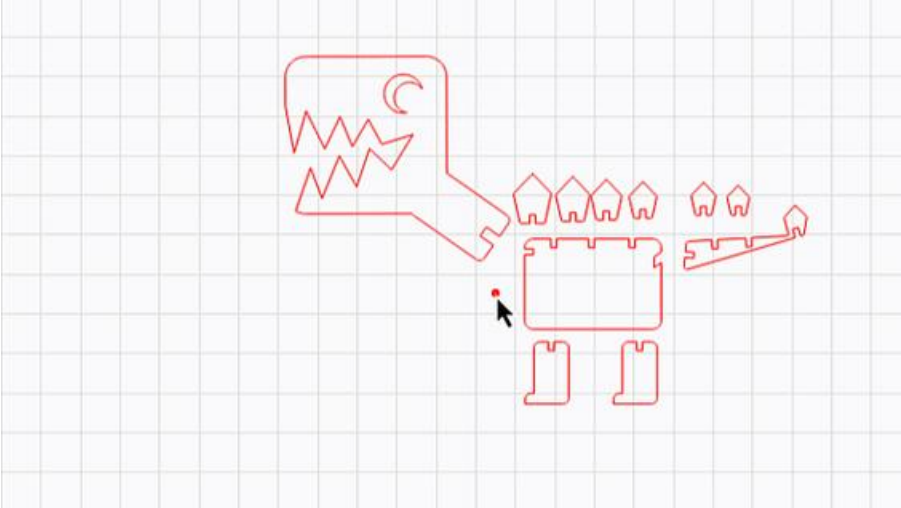
You can adjust the nudge distance of each arrow key and modifier in the **Units and Grids** tab of the Settings window, under Shape Move Increments.



Size

The **Size** handles are squares located around the perimeters of your selection area — at each corner, and the midpoints of each side. When you hover over a Size handle, the two-arrow icon will appear.

- Use the corner handles to rescale both dimensions of your selection while maintaining its aspect ratio.
- Use the midpoint handles to rescale a single dimension at once.
- When dragging any of the Size handles, the behavior is asymmetric — the other side of the object acts like an anchor and stays in place.
- Click, hold, and drag a Size handle to rescale your selection. Drag outward to make it larger, or inward to make it smaller.
- Release to set the selection to its new scale.
- Press `Esc` before releasing to deselect and return the selection its original scale after releasing.



Modifier Keys

- Hold `Shift` while dragging from a corner handle to rescale both dimensions at once, while not maintaining the original aspect ratio. Left-right movement will adjust the width, while up-down movement will adjust the height.
- Hold `Ctrl`/`Cmd` while dragging from any **Size** handle to rescale the selection symmetrically from its center, rather than from the chosen handle.

> **Tip**
>
> To precisely adjust a selection's dimensions, use the Numeric Edits Toolbar.

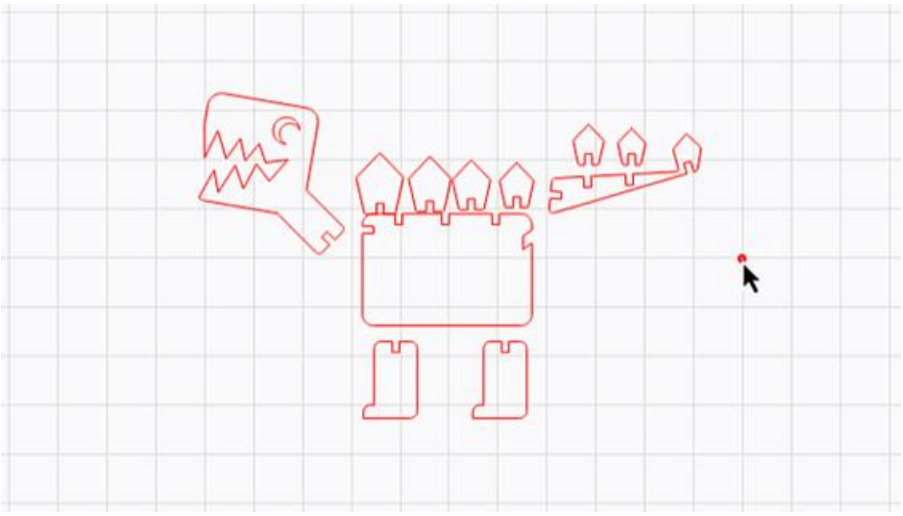
Rotate

There are four **Rotate** handles — they're the curved two-arrow icons, one on each corner of your selection. When you hover over a Rotate handle, your cursor will change to the curved two-arrow icon as well.

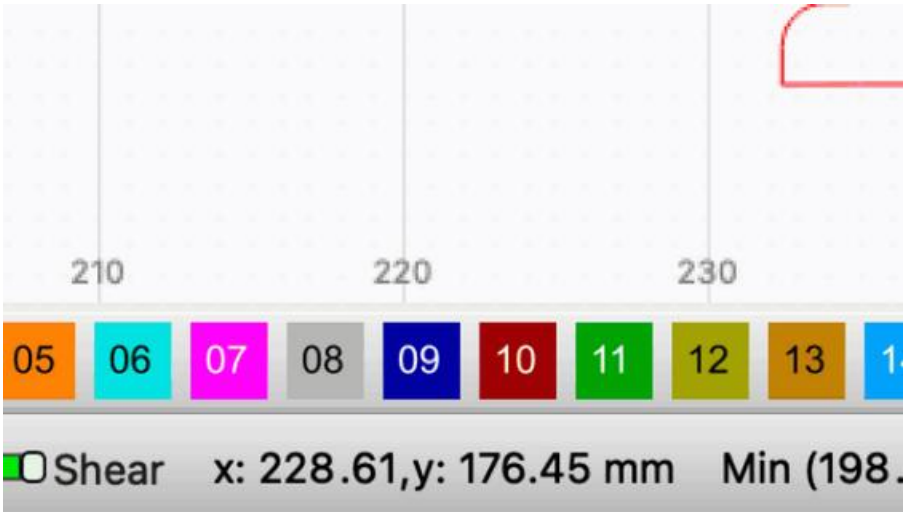
- Click, hold, and drag a Rotate handle clockwise or counterclockwise to turn the selection around its center point.
- Release to leave the selection in its new orientation.
- Press `Esc` before releasing to deselect and return the selection its original orientation after releasing.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=bcRy5-V-r4E

> **Use Two-Point Rotate / Scale to turn your selection from a custom pivot point instead of a Rotate handle.**
>
> ▶ Watch the video on YouTube: https://www.youtube.com/watch?v=axO2M5asBbQ



As you Rotate your selection, the Status Bar will actively show the change in degree of rotation.



Modifier Keys

- Hold `Shift` while dragging to lock rotation to 15° increments.
- Hold `Ctrl`/`Cmd` to lock to 5° increments.

Rotate Hotkeys

You can rotate objects by set increments using hotkeys.

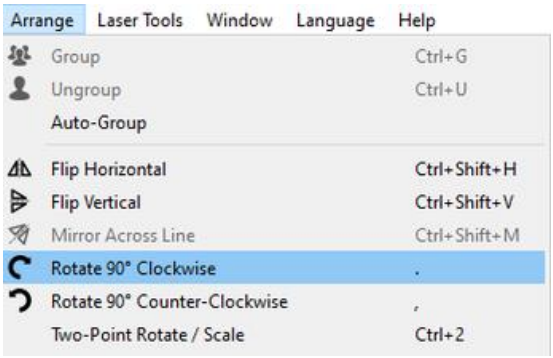
These hotkeys only work when you have objects selected and the **Edit Window** (Workspace) — the area where you create and edit graphics — is in focus.

> **What does it mean for a window to be in focus?**
>
> There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.
>
> If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus. Selecting objects in the Workspace will also bring it into focus.

Action	Windows	macOS
Rotate 90° clockwise	.	.
Rotate 90° counterclockwise	,	,
Rotate 45° clockwise	Shift + .	Shift + .
Rotate 45° counterlockwise	Shift + ,	Shift + ,
Rotate 15° clockwise	Ctrl + .	Cmd + .
Rotate 15° counterclockwise	Ctrl + ,	Cmd + ,
Rotate 1.5° clockwise	Ctrl + Shift + .	Cmd + Shift + .
Rotate 1.5° counterclockwise	Ctrl + Shift + ,	Cmd + Shift + ,

Rotate in the Arrange Menu

You can also select **Rotate 90° Clockwise** or **Rotate 90° Counter-Clockwise** from the Arrange Menu.



Rotate in the Numeric Edits Toolbar

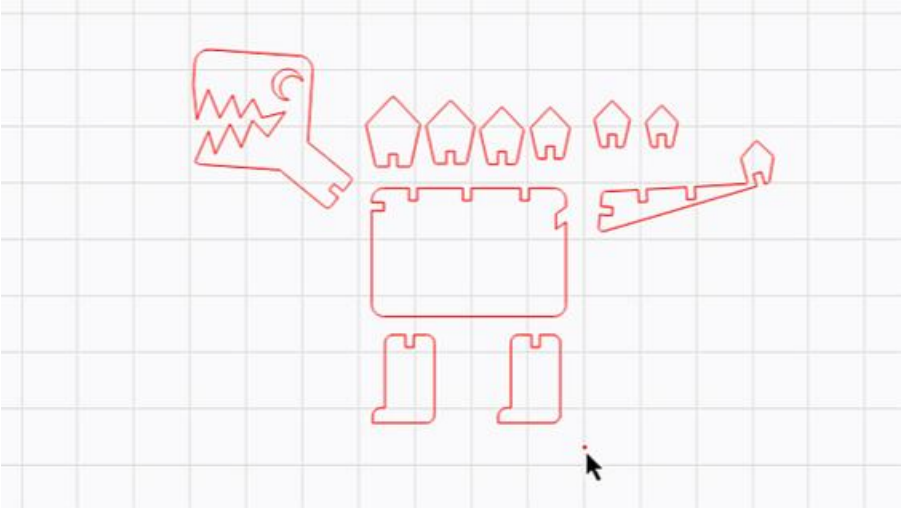
Enter an angle of rotation in the **Rotate** field of the Numeric Edits Toolbar to Rotate objects at any angle. This field accepts negative values.



Shear

The two **Shear** handles are small squares located just above or below and just to the left or right of a corner of your selection.

- The handle above or below skews your selection vertically, while the handle to the left or right skews it horizontally.
- Click, hold, and drag a Shear handle to skew the selection.
- Release to apply the new skew to the selection.
- Press Esc before releasing to deselect and return the selection to its original skew.



> **Tip**
>
> Use Warp/Deform Selection to skew selections in any direction.

Transform Control Toggles

Use the **Transform Control Toggles** in the bottom left of the LightBurn window to enable and disable specific types of editing across an entire project, and prevent accidental edits, such as inadvertently scaling a design that contains strict sizing, or moving tightly nested parts.

> **Note**
>
> This feature is disabled when using Beginner Mode.



▶ Watch the video on YouTube: https://www.youtube.com/watch?v=4DdNVzh6FRY

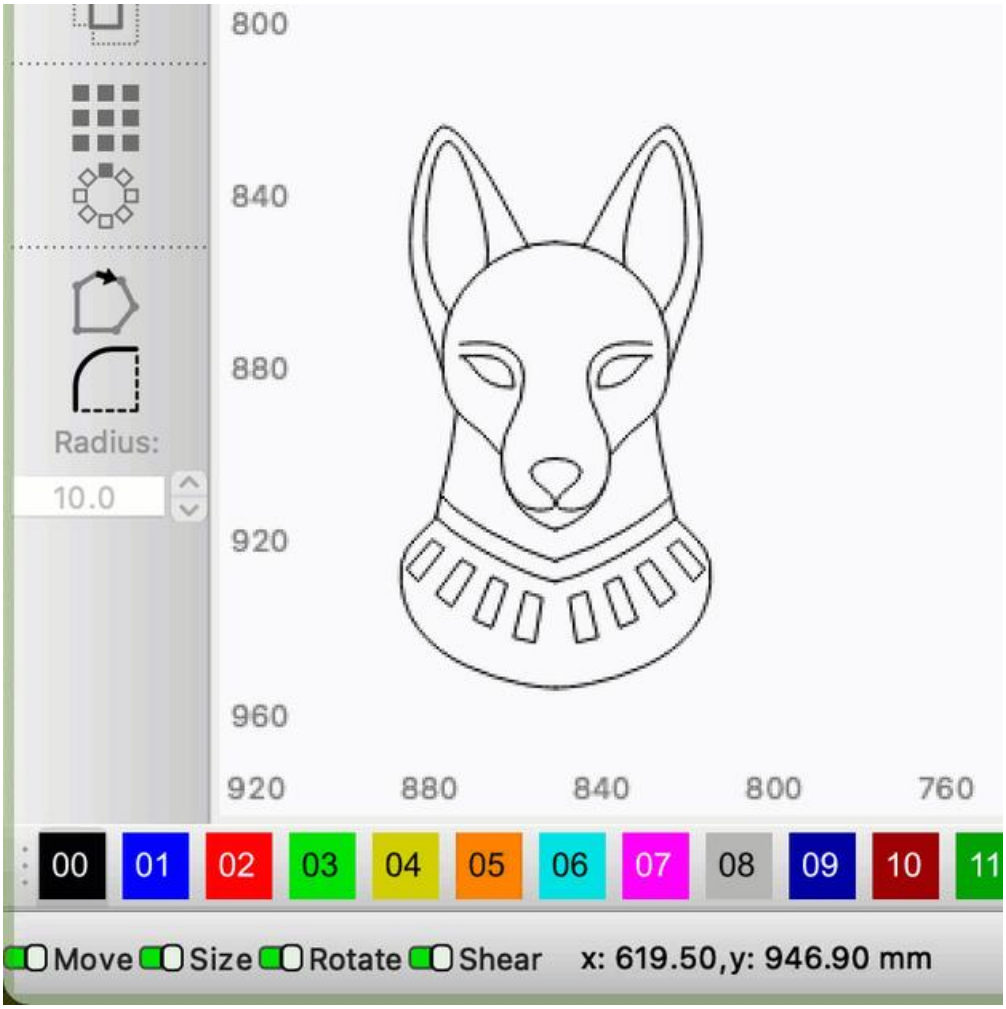
The Transform Control Toggles are all enabled by default. Each toggle enables or disables a different type of editing:

- **Move**: shifting, repositioning, relocating, dragging, or rearranging objects.
- **Size**: scaling, resizing, enlarging, or shrinking objects.
- **Rotate**: spinning, turning, tilting, or adjusting the angle of objects.
- **Shear**: skewing or slanting objects.

When a toggle is enabled, the corresponding Transform Controls appear around selected objects. As toggles are disabled the controls disappear, restricting that type of editing.

Tip

Use Lock Shapes to lock editing of individual objects in a design.



Related Topics

- Lock Shapes
- Selection Tools
- Snapping
- Select
- Numeric Edits Toolbar
- Warp/Deform Selection
- Status Bar

Grouping & Ungrouping

The **Group** tool combines collections of objects into single units, and the **Ungroup** tool splits Groups back into the individual objects that make them up.

Normally, all objects are independent of one another, and can be manipulated individually. But it's sometimes helpful to treat a collection of things as a single unit, to make sure they keep their relative position and size when you're moving them around.

LightBurn also provides options in Optimization Settings to tell your laser to engrave Grouped objects together or to cut and engrave multi-layer Groups in order, completing each layer in each Group before moving on to the next one.

Both Group and Ungroup are available in the Arrange Menu, Arrange Toolbar, and by hotkey.



▶ Watch the video on YouTube: https://www.youtube.com/watch?v=w15iADDWuyA

Grouping and Ungrouping

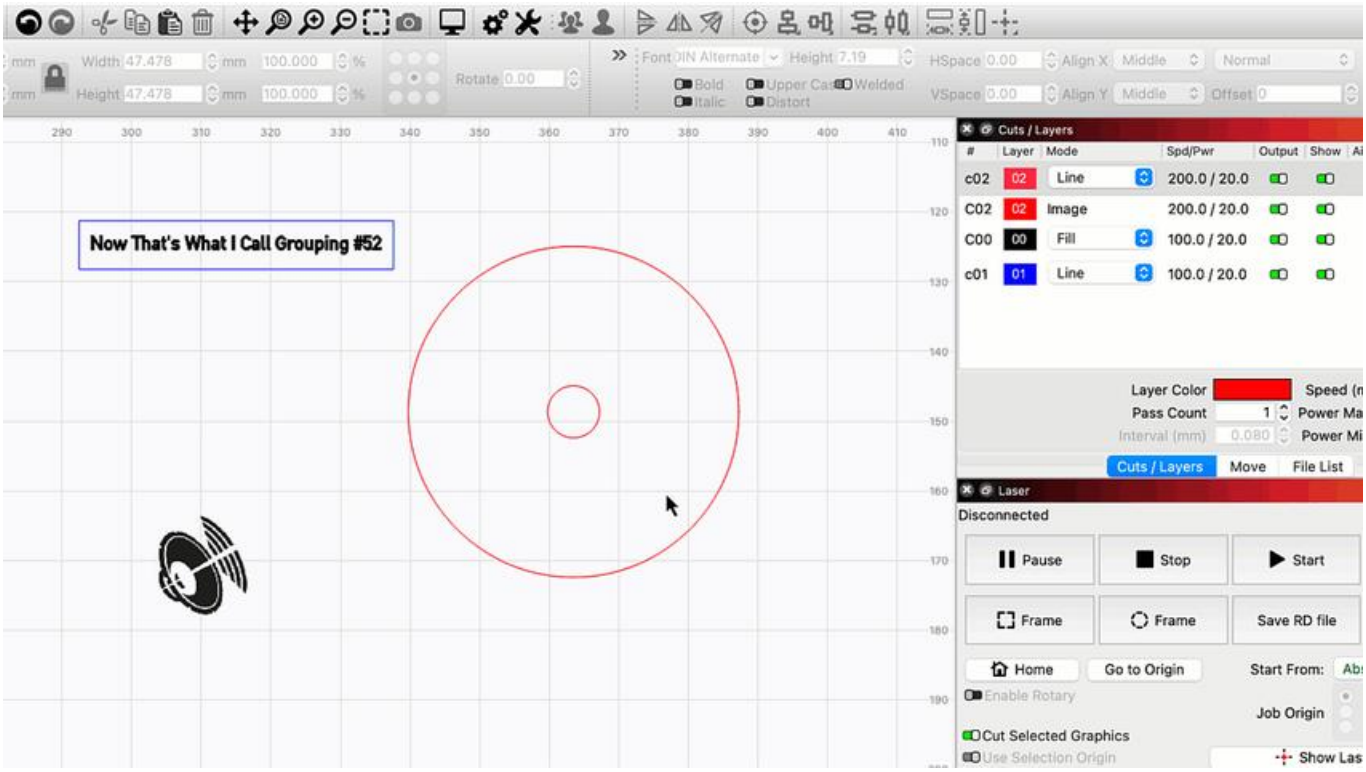
Group

`Ctrl`/`Cmd` + `G`

Group selected objects to treat them as a single unit when they are:

- Selected
- Moved
- Resized
- Used with the Boolean tools

You can create Groups out of any objects, including objects assigned to different layers, and multiple types of objects such as lines, shapes, text, and images. You can even combine objects that have already been Grouped into other Groups.

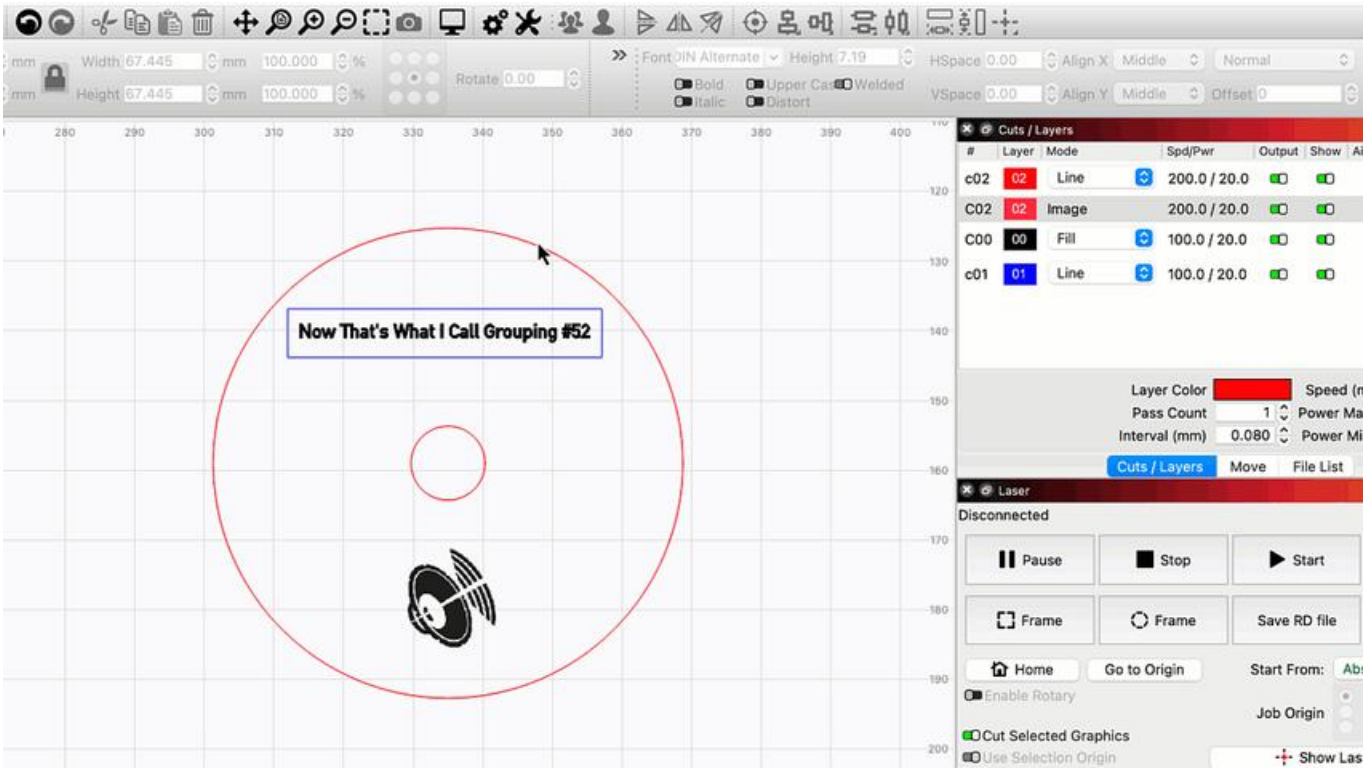


Ungroup

`Ctrl`/`Cmd` + `U`

Ungroup selected (previously Grouped) objects to treat them as discrete, independently adjustable units again.

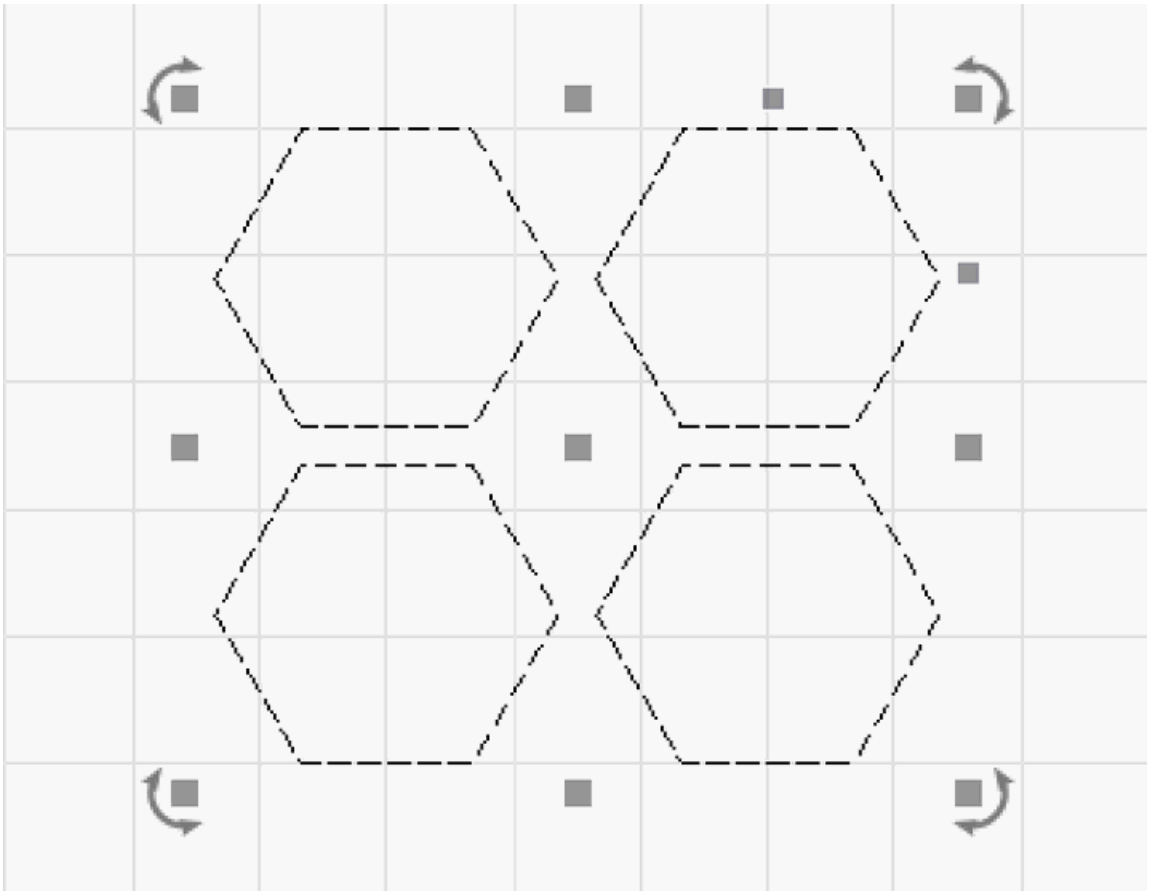
If you have a Group made up of other Groups, you'll need to Ungroup multiple times to separate all objects into individual units again.



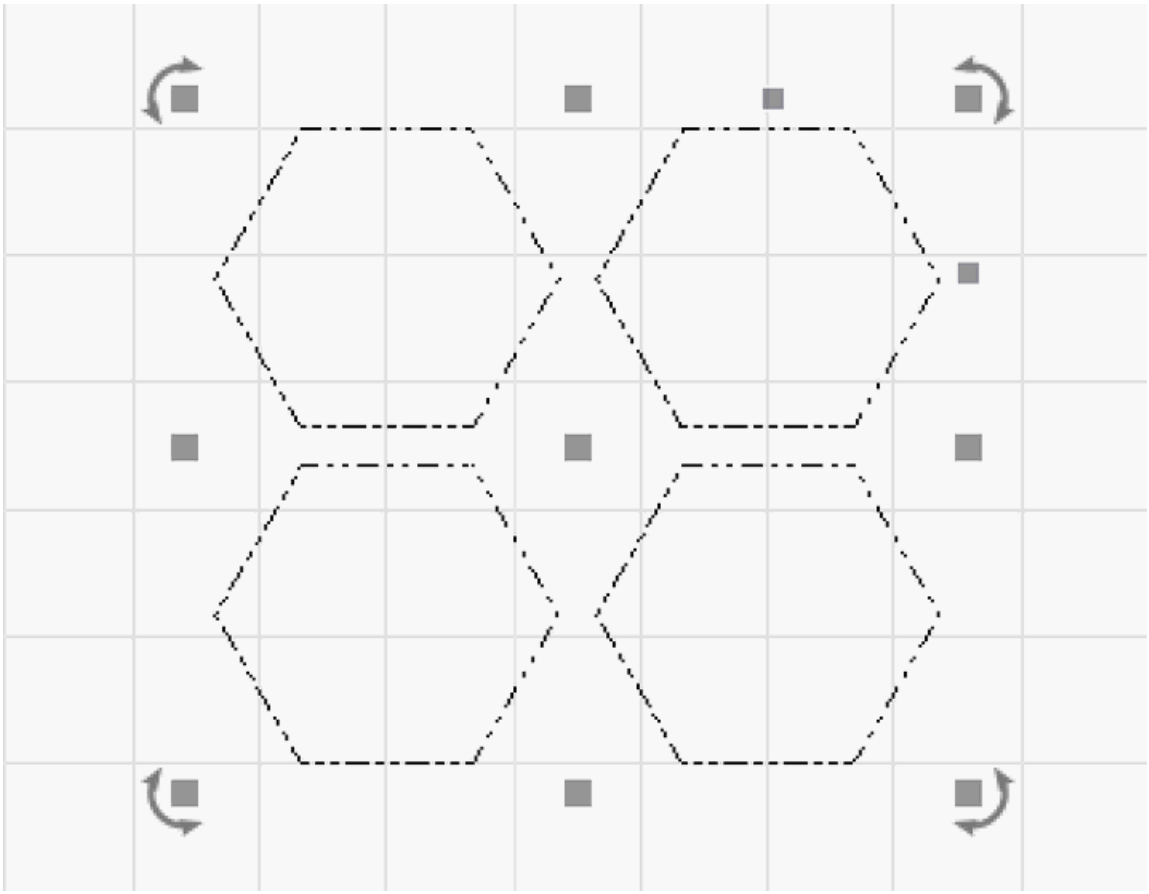
Selection Appearance of Grouped Objects

You can tell whether an object is **Grouped** based on its appearance when you select it.

Ungrouped objects are drawn with an animated dash pattern instead of solid lines, while Grouped objects are drawn with a dot-dot-dash pattern.



Ungrouped Objects Selected

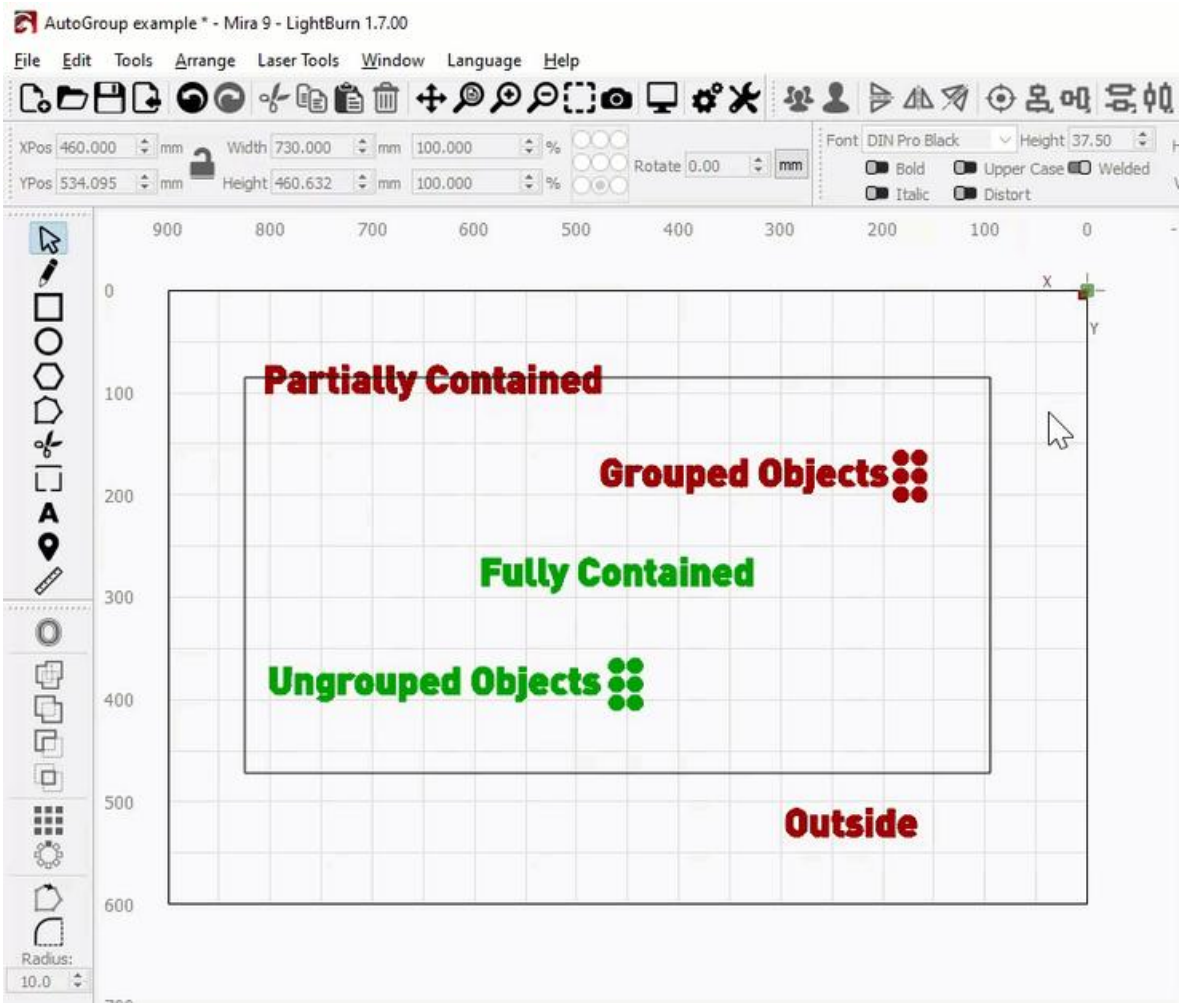


Grouped Objects Selected

Auto-Grouping

Auto-Group identifies shapes that are fully contained within another shape and Groups them together with the outer shape.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=mYv92lztU3k



In the animation above, the objects shown in red are unaffected by Auto-Group, either because they're not fully contained within the rectangle or because they are already Grouped. The green objects are automatically Grouped with the rectangle.

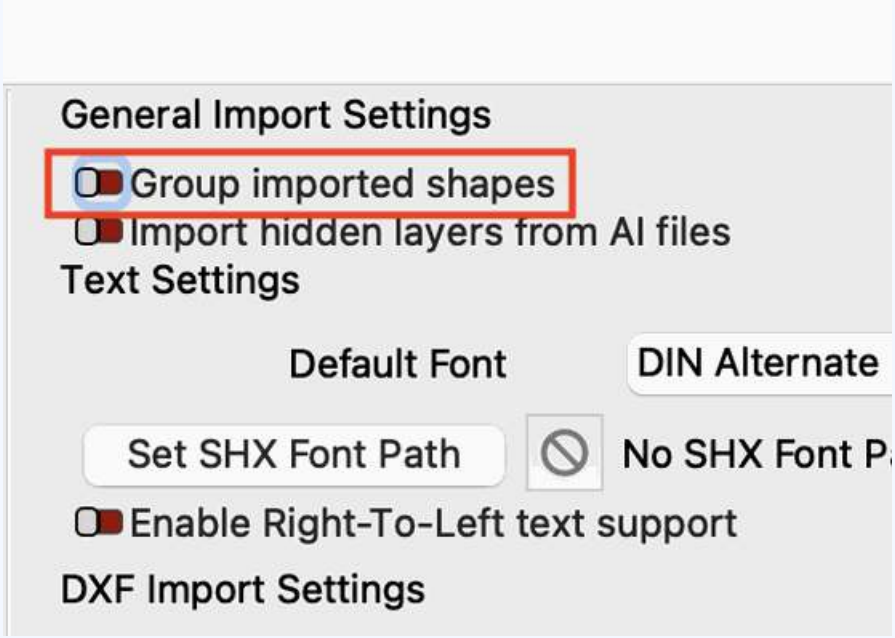
To use Auto-Group:

1. Select the shapes you want to use Auto-Group on.
2. Make sure the shapes are not Grouped using Arrange → **Ungroup** or `Ctrl` + `U`.
3. Use Arrange → Auto-Group to automatically group all fully contained shapes inside the outermost shape.

Troubleshooting

> **Note**

Imported shapes are Grouped by default. Disable **Group imported shapes** in your Settings to turn off automatic Grouping upon importation.



- The following tools cannot be used with **Grouped** objects:
  - Edit Nodes
  - Trim Shapes
  - Resize Slots
  - Convert to Path
  - Auto-Join Selected Shapes
  - Close Path
  - Close Selected Paths with Tolerance
  - Break Apart
  - Optimize Selected Shapes
  - Select contained shapes
- Grouped objects are treated as single units when working with the Boolean tools. If any object in a Group is an open shape, the entire Group cannot be used with the Boolean tools.
- Using Select open shapes, Select open shapes set to Fill, and Select shapes smaller than selected will select applicable Grouped shapes, and remove them from their Group.
- **Auto-Group** only works on shapes that are fully contained within the outer shape. If any portion of a shape extends outside the outer shape, it will be left out of the Group.
- Auto-Group only works on Ungrouped shapes. Use Ungroup to remove shapes from an existing Group.

Related Topics

- Fill Groups Together

- Order by Group
- Selection Tools
- Numeric Edits Toolbar
- Align
- Docking
- Grid Array
- Boolean Tools

Preview

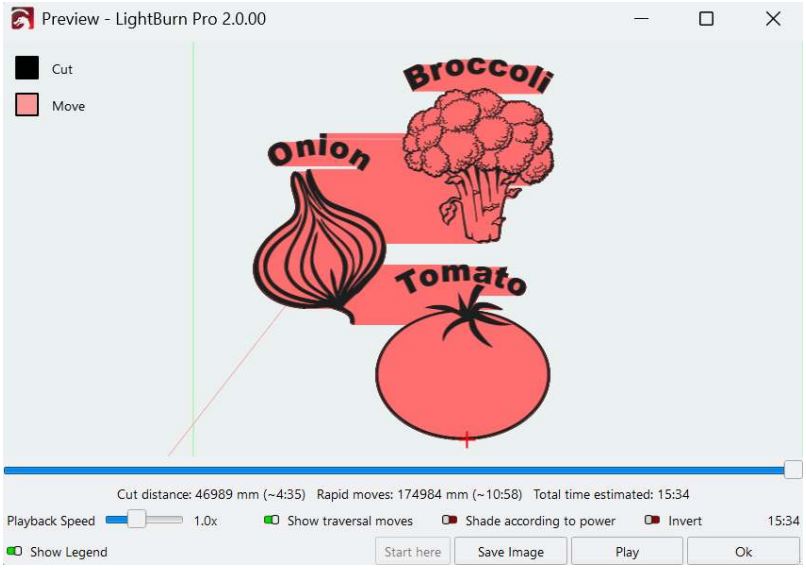
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=o1_FKGhJoek

The **Preview** window is an accurate representation of what LightBurn will send to the laser. Black lines represent cut moves, while red lines are travel moves between cuts.

It's a good practice to check the Preview window before starting any job, to make sure the simulated output matches your expectations, or to identify any problems before it's too late.

To open the Preview window, use one of the following methods:

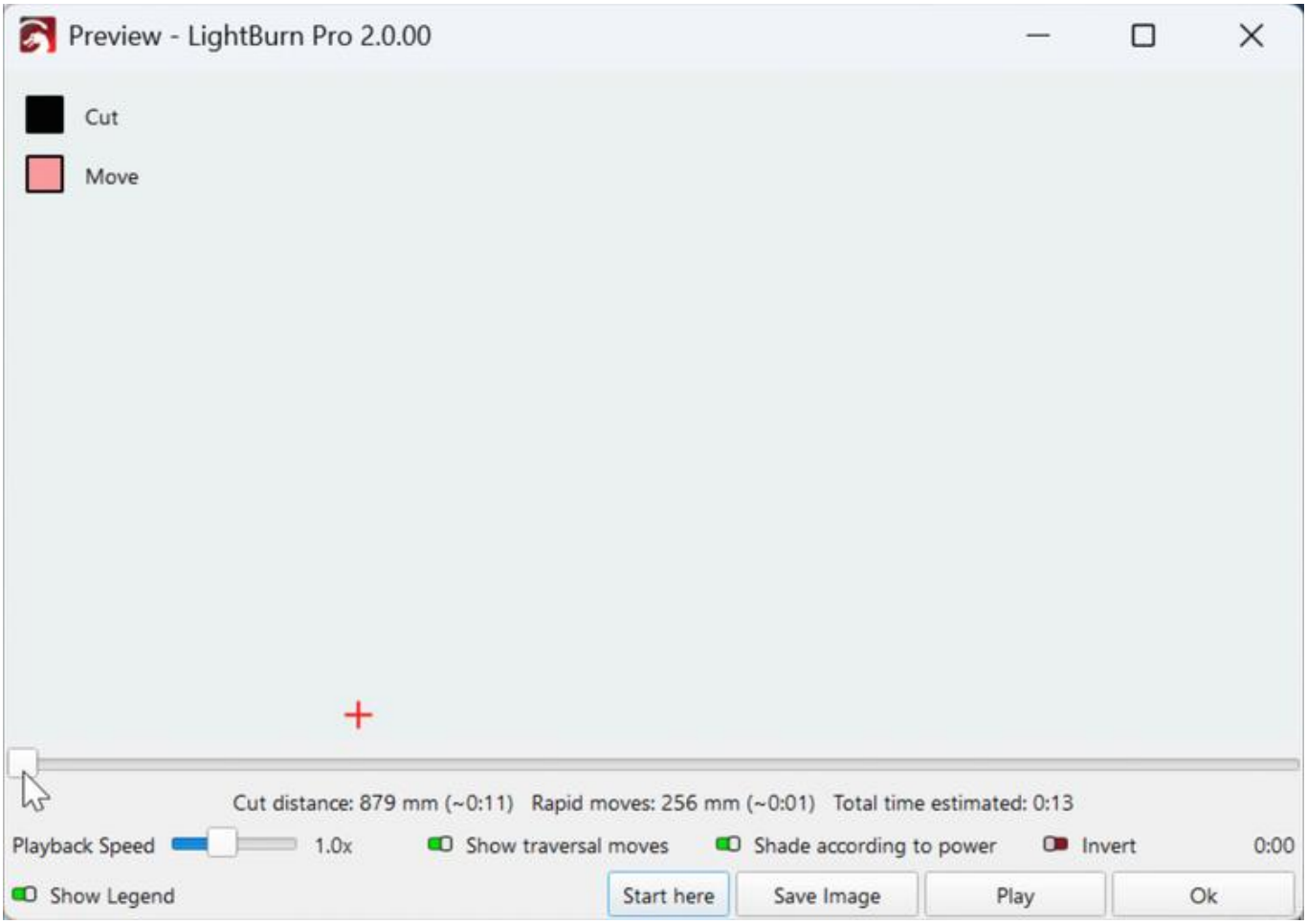
- Go to Window → **Preview**
- Click the Preview icon in the Main Toolbar
- Press `Alt`/`Option`+`P`
- Right-click in your Workspace and select Preview from the context menu



Using the Preview Window

Time Slider

The **Time Slider** allows you to slide quickly through the project and see the cut order.

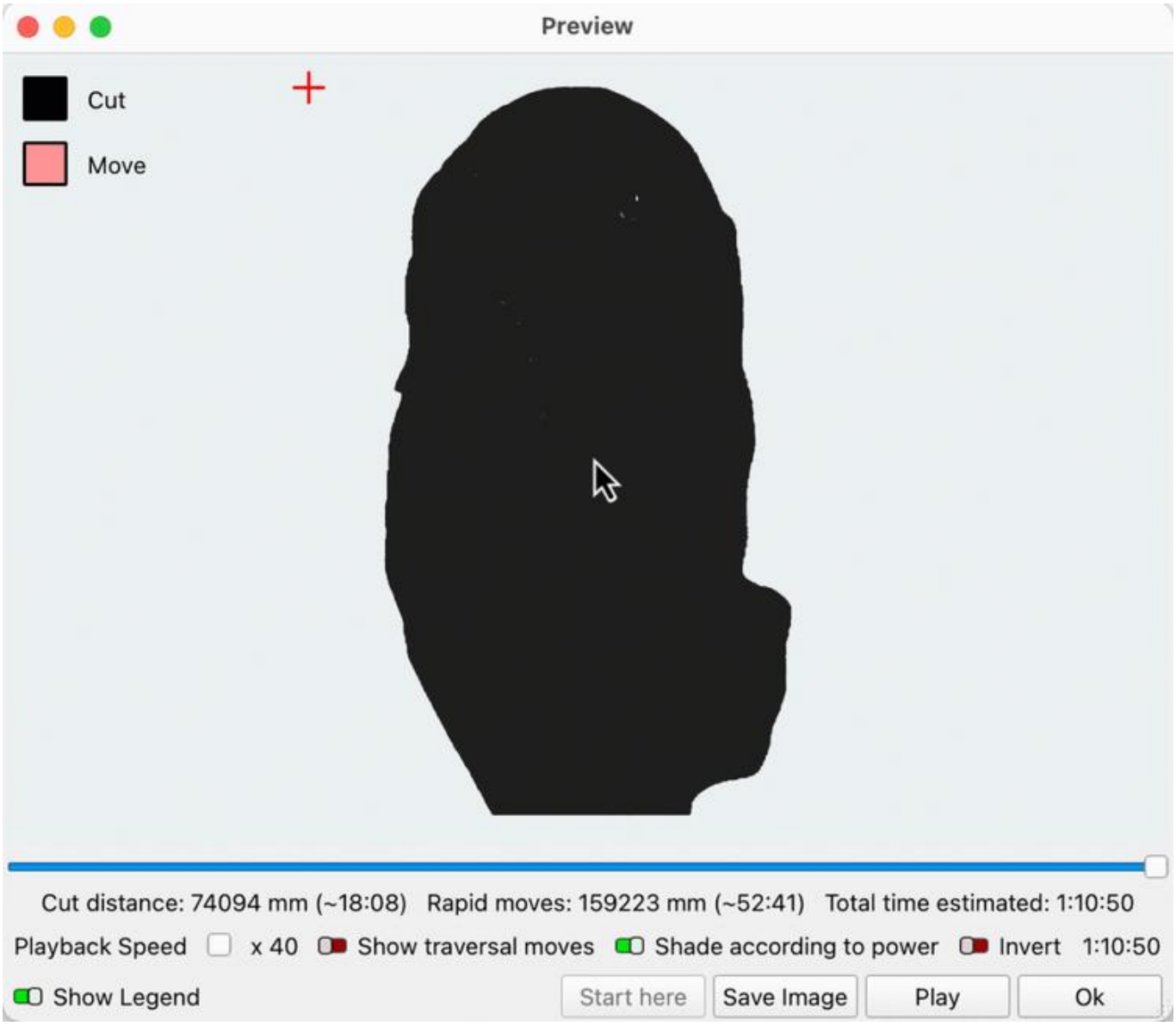


Zooming and Panning

Zoom in and out using your mouse's scrollwheel, two-finger dragging if you're using a trackpad, or by pressing the + or - keys on your keyboard.

Pan by holding your middle mouse button and dragging, or by pressing Space, clicking, holding, and dragging.

You can check the placement of your artwork relative to your laser's workspace by zooming out. The outer edge of your laser's cutting area is shown as a green border in the Preview window.



Job Time Statistics

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=s1jCqB233N0

Below the **Time Slider** are statistics about the job and the estimated run time. The Additional Settings tab in **Device Settings** can help you get more accurate time estimates – see the video above for help.

- **Cut distance** shows the distance the laser will travel while on, and the estimated time it will take the laser to complete those movements in parentheses.
- **Rapid moves** shows the distance the laser will travel while off, between cutting and engraving, and the estimated time it will take the laser to complete those movements in parentheses.
- **Total estimated time** shows the combined time it will take the laser to complete all Cut and Rapid moves.

Preview Settings

The settings here only affect the **Preview** window – they don't change anything about LightBurn's output to the laser.

Setting	Description
Playback Speed	Sets the speed of playback with the **Play** button – ranges from 1/10 of actual speed to 40 times the actual speed.
Show Traversal Moves	If enabled, shows travel moves in red. If disabled, travel moves are hidden.
Shade According to Power	If enabled, uses grayscale shading to indicate higher and lower power cut moves. This only works with the Grayscale Image Mode, Power Scaling, or Ramp Length.
Invert	If enabled, sets cut moves to white on a black background. Useful when engraving on dark surfaces that engrave lighter, such as slate.
Show Legend	If enable, shows a legend in the top left indicating which colors represent Cuts and Moves.



Start Here

Use the **Start Here** button to recover interrupted jobs by starting the laser at a precise time partway through the operation. Move the Time Slider to the point at which you want to start, then click **Start Here**.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=WmOGoHiYQu4

Save Image

Saves the current **Preview** as an image file.

Play

Animates the simulation to show the job from start to finish at the speed set in **Playback Speed**.

Troubleshooting

- If the time estimates are inaccurate, update the **Simulation Settings** in Device Settings — Additional Settings.
- If the output of the laser doesn't match the **Preview**, see Troubleshooting: Job Quality.
- See Troubleshooting: Preview if you see something in the Preview window that doesn't look right, but aren't sure how to fix it.

Related Topics

- Print and Cut
- Device Settings — Additional Settings
- Cut Settings
- Optimization Settings

New Window

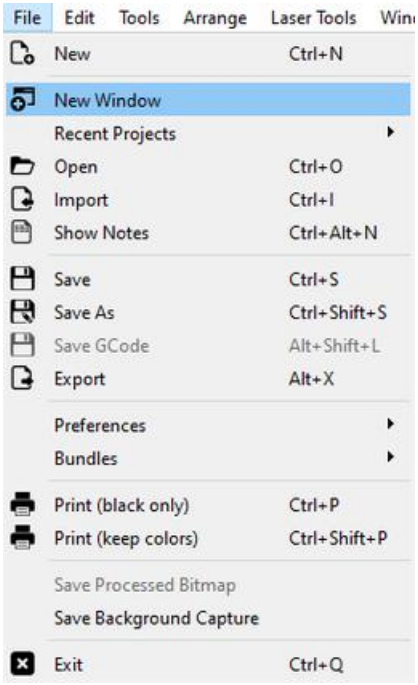
New Window opens a separate instance of LightBurn in a new window, allowing you to edit multiple projects at once, or edit one file while cutting a different one.

Working With Multiple Windows

Go to File → **New Window** to open a separate instance of LightBurn in a new window.

All open windows share the same settings and preferences. The last window closed after changing LightBurn settings will save those changes to the preferences file and will overwrite any changes made previously in other open instances.

> **Note**
>
> Windows users can also double-click the LightBurn application or shortcut to open a new instance.



Related Topics

- For an alternate method that creates fully independent instances of LightBurn, refer to the Running Multiple Independent Instances of LightBurn guide.

View Style

Changing the **View Style** alters the appearance of vector graphics in your Workspace, which can be useful for spotting potential design issues and visualizing the final output from a project.

These options only change the rendering of the objects in your Workspace and do not affect the final work produced by the laser.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=6IuXmIK-zkc

View Style Types



Filled

Shapes that are assigned to layers set to Fill Mode will appear as solid shapes, blocking out shapes behind them on other layers. This is handy for getting a better understanding of the overall design, but may hide duplicate or unwanted shapes on different layers. Filled View Styles may also be trickier to edit, as some objects are hidden behind others and not easily selected. Render times may be longer on older or less-powerful computers.

Wireframe

All vectors in the design appear as outlines (strokes). This style renders quickly and makes issues in a design more obvious, but does not provide a visual indication of which graphics are assigned to layers set to Fill Mode. Use Preview to see a rendering of your finished result instead.

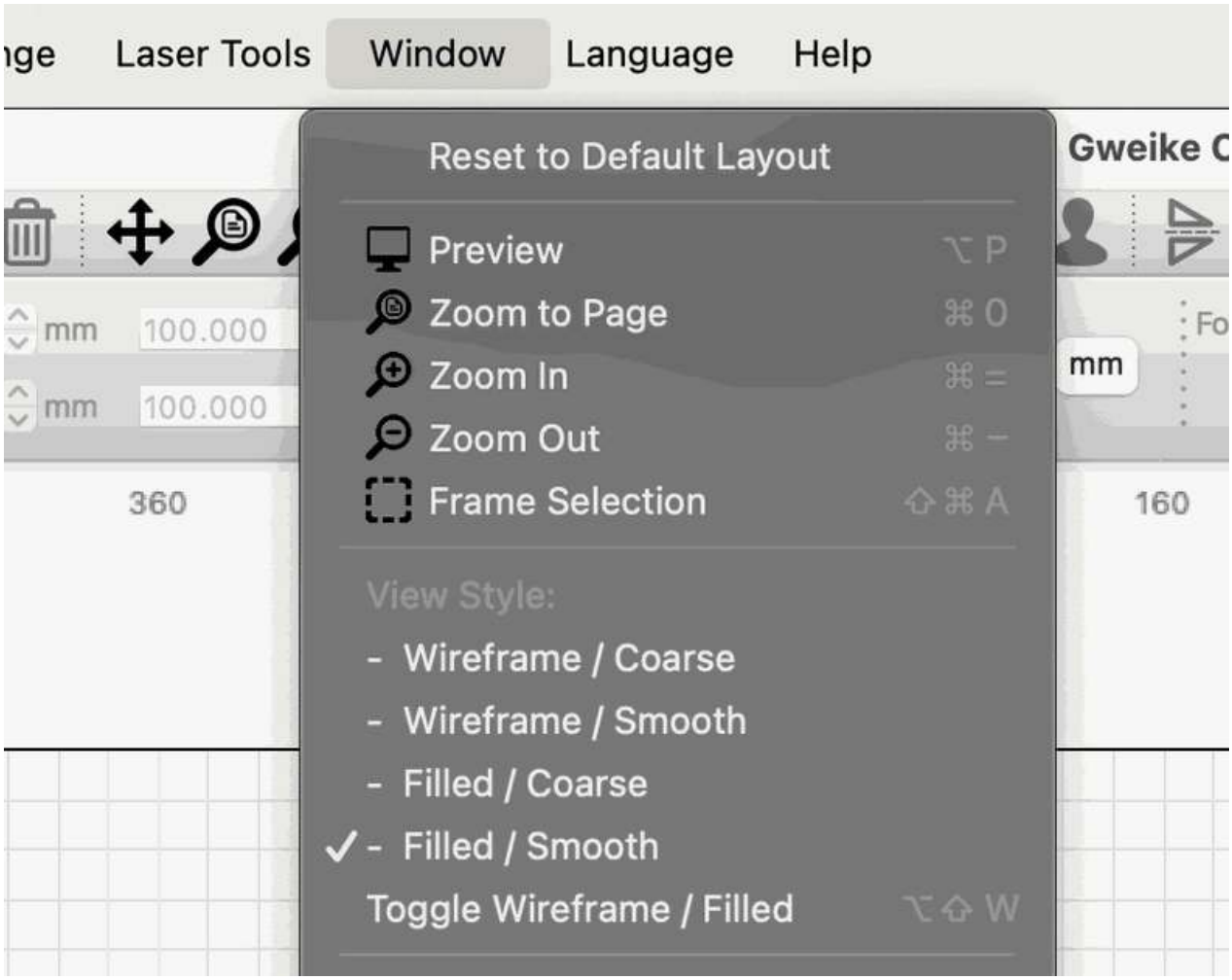
Edge Rendering Quality

- **Smooth**: High detail, slower rendering.
- **Course**: Lower detail, faster rendering.

Changing View Style

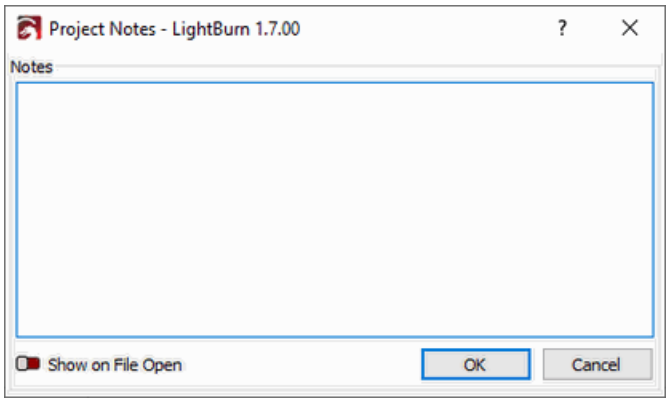
Select your desired **View Style** from the Window Menu.

Toggle between **Wireframe** and **Filled** mode at any time by pressing `Option` / `Alt`+`W`.



Show Notes

Show Notes allows you to add, edit, and view notes for a project. This is often useful for files that will be shared with others, or to store related information for working with a project in the future. Go to **File** → **Show Notes** or press `Ctrl` + `Alt` + `N`.



Edit the **Notes** by typing into the text box and clicking **OK** to save. Press **Cancel** to discard any changes and close the window.

Enable the **Show on File Open** toggle to show the notes automatically when the file is opened in LightBurn.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=QGOIwYQYL7I

Related Topics

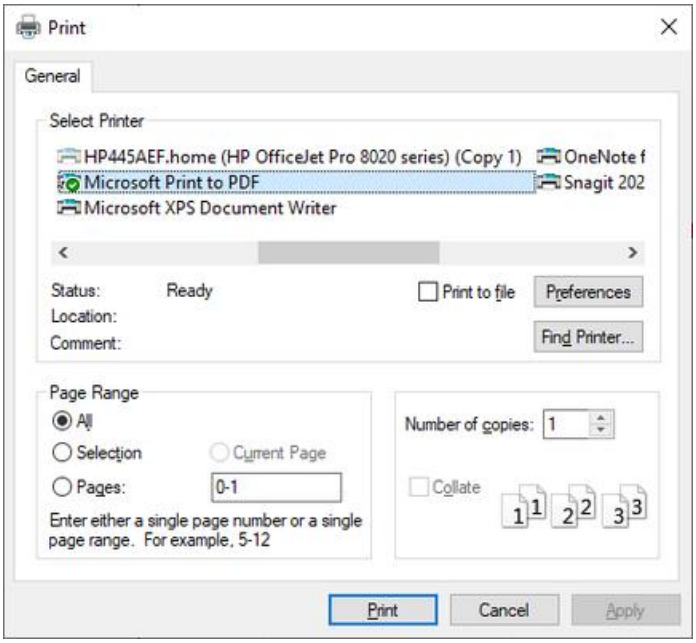
- Job Checklist
- File Management

Print

Use **File → Print (Black Only)** or **File → Print (Keep Colors)** to print the current contents of the work area using your computer's print dialog. Most computers have built-in PDF creation tools, allowing you to export a PDF using these tools as well.

Starting with LightBurn 2.1, the line width when printing from LightBurn is controlled by Print Settings - Line Width. If you need a specific line width, for instance to print to a laser that uses a print driver, you can now specify the line width you need.

> **Page Size Warning**
>
> Be aware that the page size is controlled by the print dialog, not by LightBurn. If your work area is larger than the paper selected for printing, your design will be scaled to fit on the paper.



Windows 10 print dialog showing options including sending to a printer and Print to PDF

Troubleshooting

If your design is printing too small, check the paper size you're printing on.

Beyond that, printing problems are likely to be related to your computer and/or printer, rather than LightBurn. Please see the relevant customer service resources for more help.

Related Topics

- Save Processed Bitmap
- Export

Layout and Design

Creation Tools

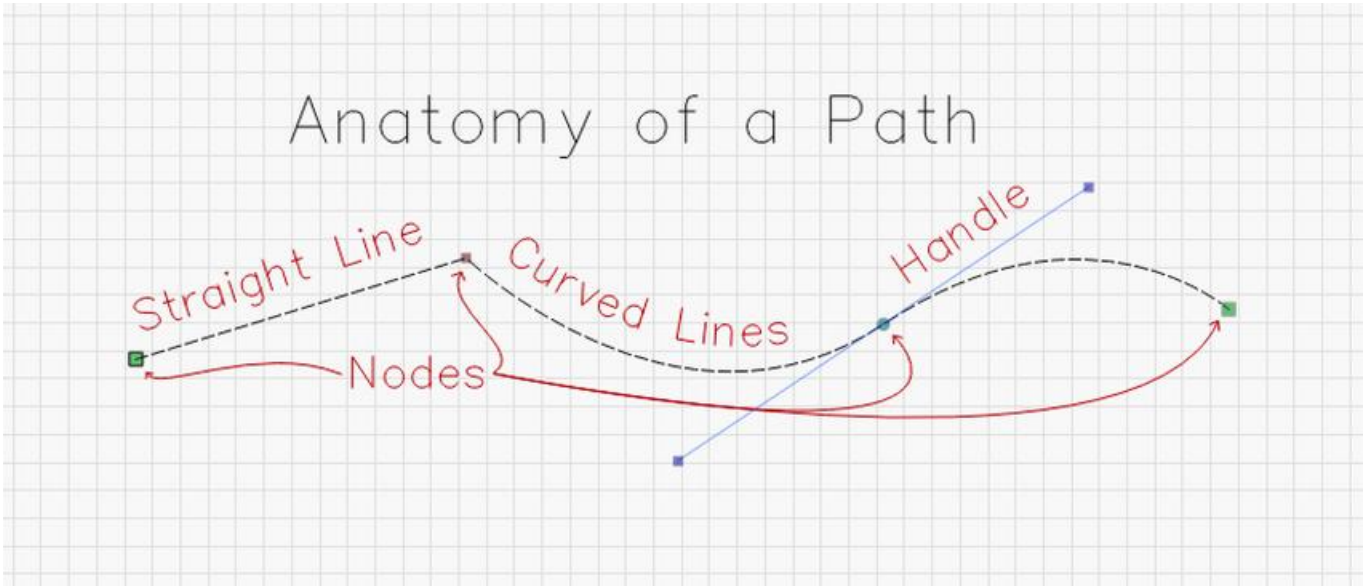
Draw Lines

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=RURJuZhn_IA

Draw Lines is the fundamental tool for creating vector paths out of straight lines and curves. Use it to create designs from scratch, modify existing graphics, or to manually trace outlines around images (see Trace Image for automatic tracing).

> **Tip**
>
> It's often helpful to refine designs with the Edit Nodes tool after roughing them out with the Draw Lines tool.

Draw Lines allows you to place **nodes** (points in space, sometimes called anchor points) that are linked by straight **lines** or **curves** (Béziers). Curved lines are described by the length and direction of **handles** (tangents) that appear at one or both of the nodes at either end of a curve.



Using the Draw Lines Tool

Activate the **Draw Lines** tool by clicking the pencil icon in the Creation Toolbar or pressing `Ctrl`/`Cmd`+`L`.



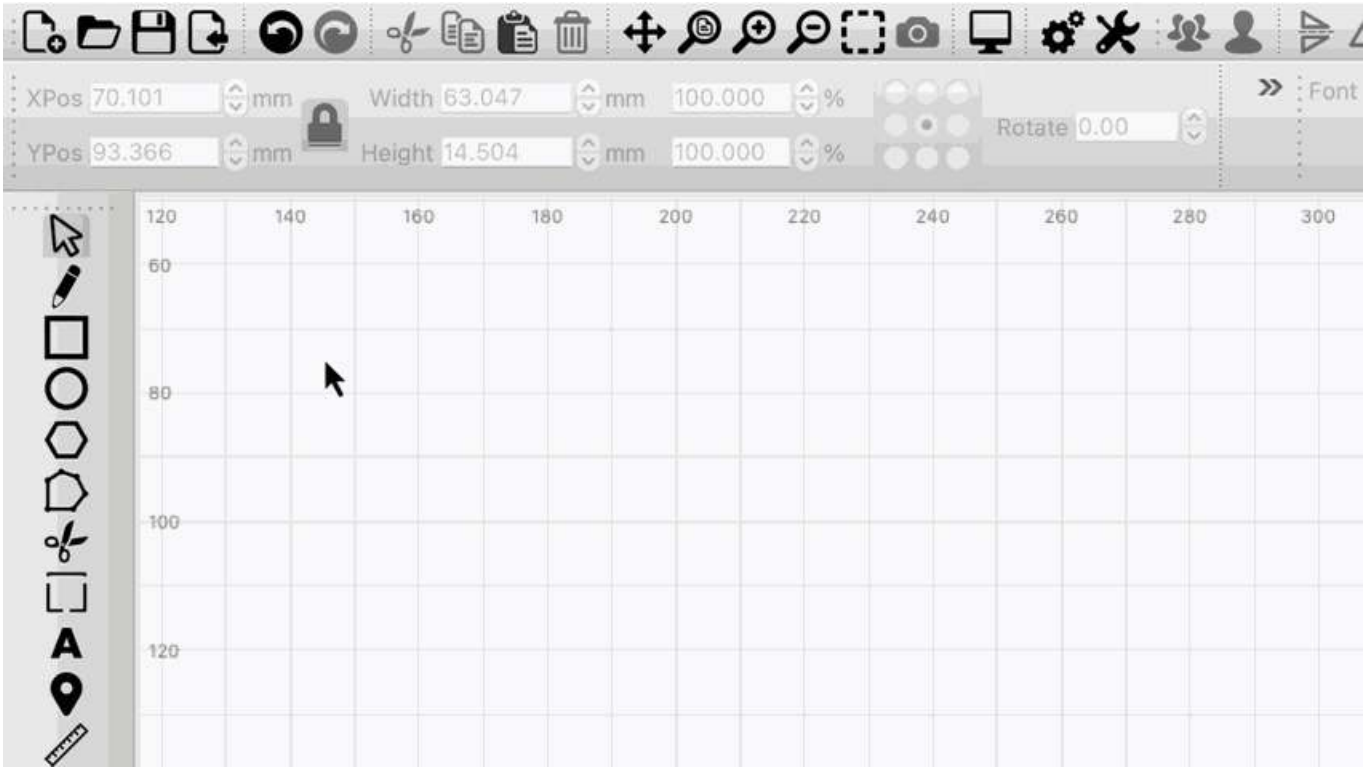
Once the tool is active, the cursor will change to indicate when you are placing corner or smooth nodes.

Lines 

Placing corner nodes and creating lines is the default behavior of the **Draw Lines** tool.

Click anywhere in the Workspace to place a node at the start of a new line, then move your cursor and click again in a different location to place another node with a line in between.

Draw Lines will continue adding nodes with line segments between them as you click, until you either click back at the starting point to close a shape, right-click your mouse to stop, or press the Esc key.



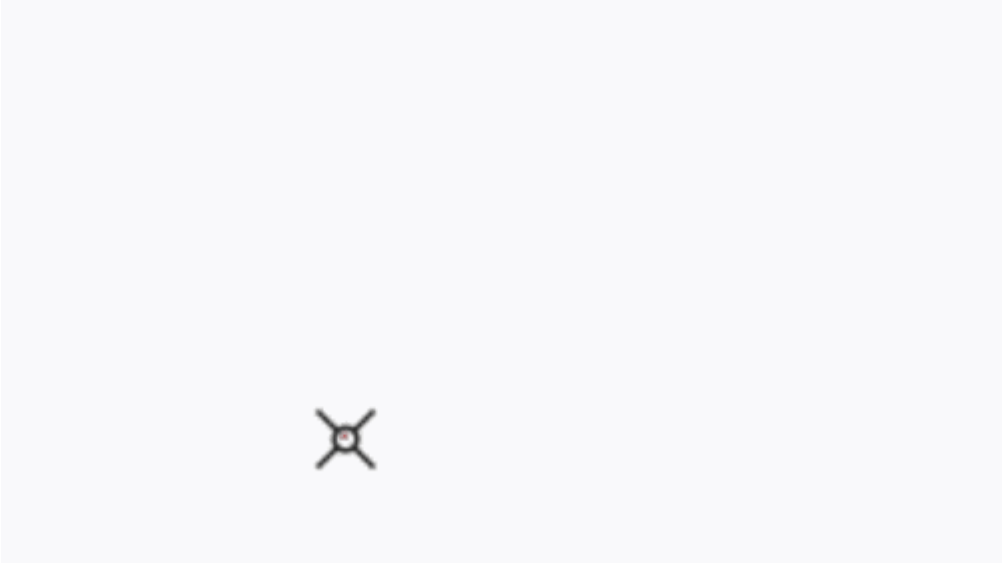
Curves

There are two options for placing smooth nodes and creating curves:

- **Auto-Smoothed Mode**: press S to toggle between creating corner and smooth nodes (the cursor will change to indicate which mode you're in). After toggling, click to place a smooth node. The next line you create after placing a smooth node will be curved, and its direction and intensity will be determined by the placement of the next node.

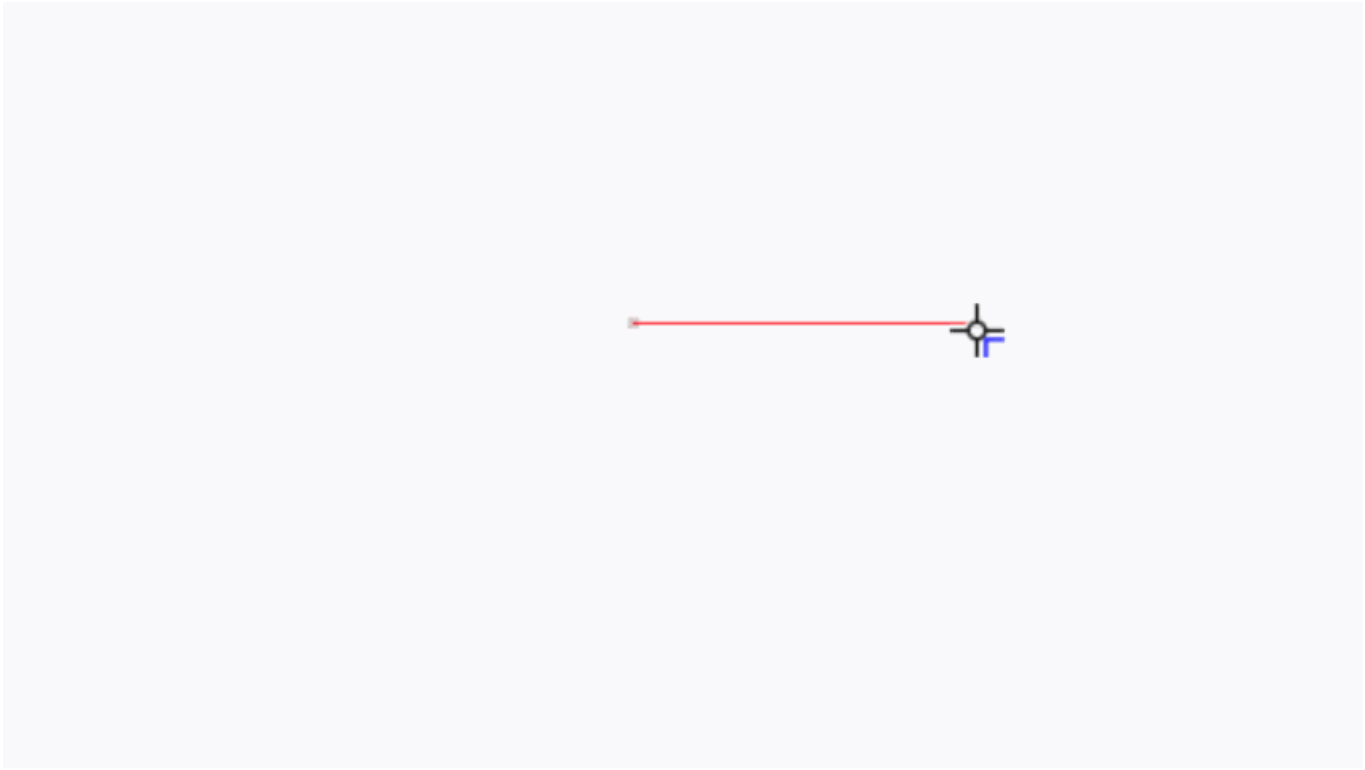


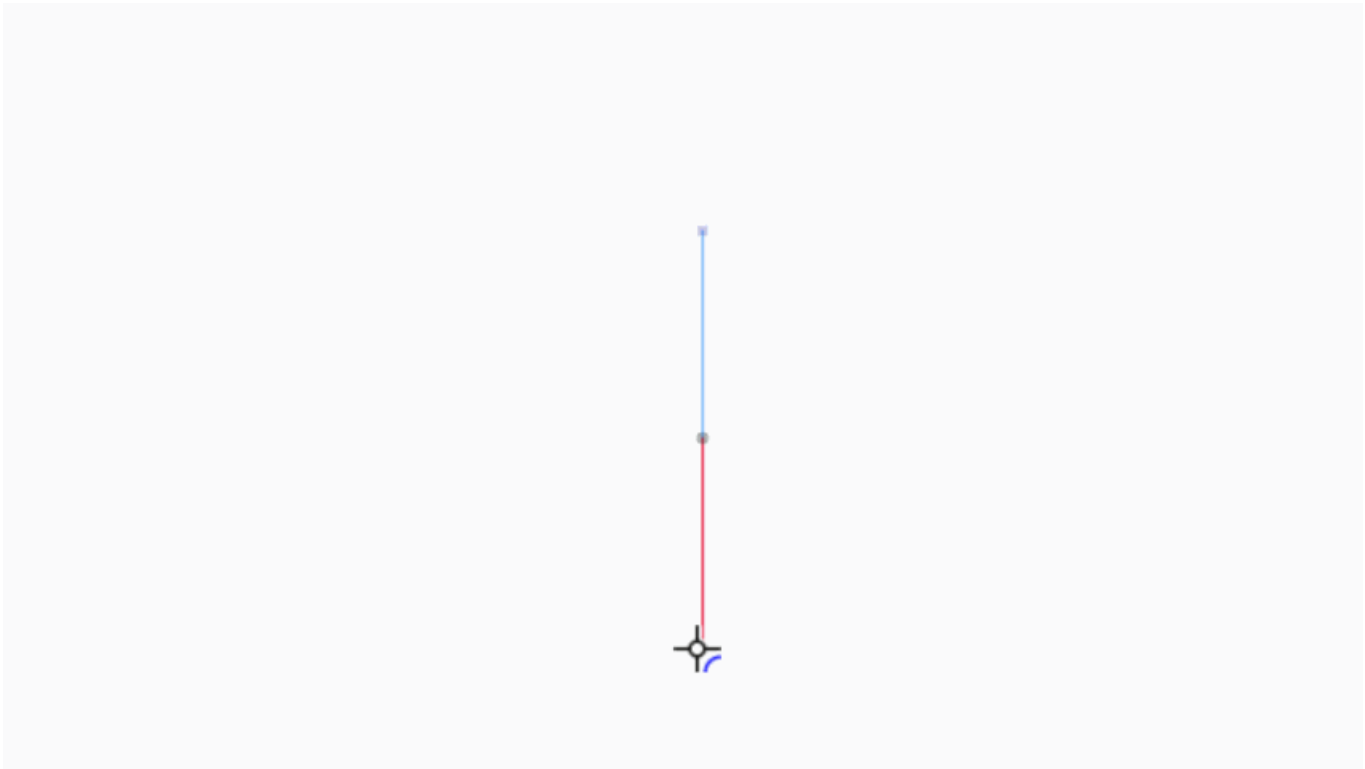
- **Controlled Mode**: click, hold, and drag while placing a node to create a smooth node with control handles — move your cursor while holding the click to control the direction and intensity of the curve handle. Release to place the handle, then click to place a node.



Constrain Placement Angle

Hold `Shift` while moving your cursor to constrain node and handle placement to a horizontal, vertical, or 45 degree angle from the last node placed.



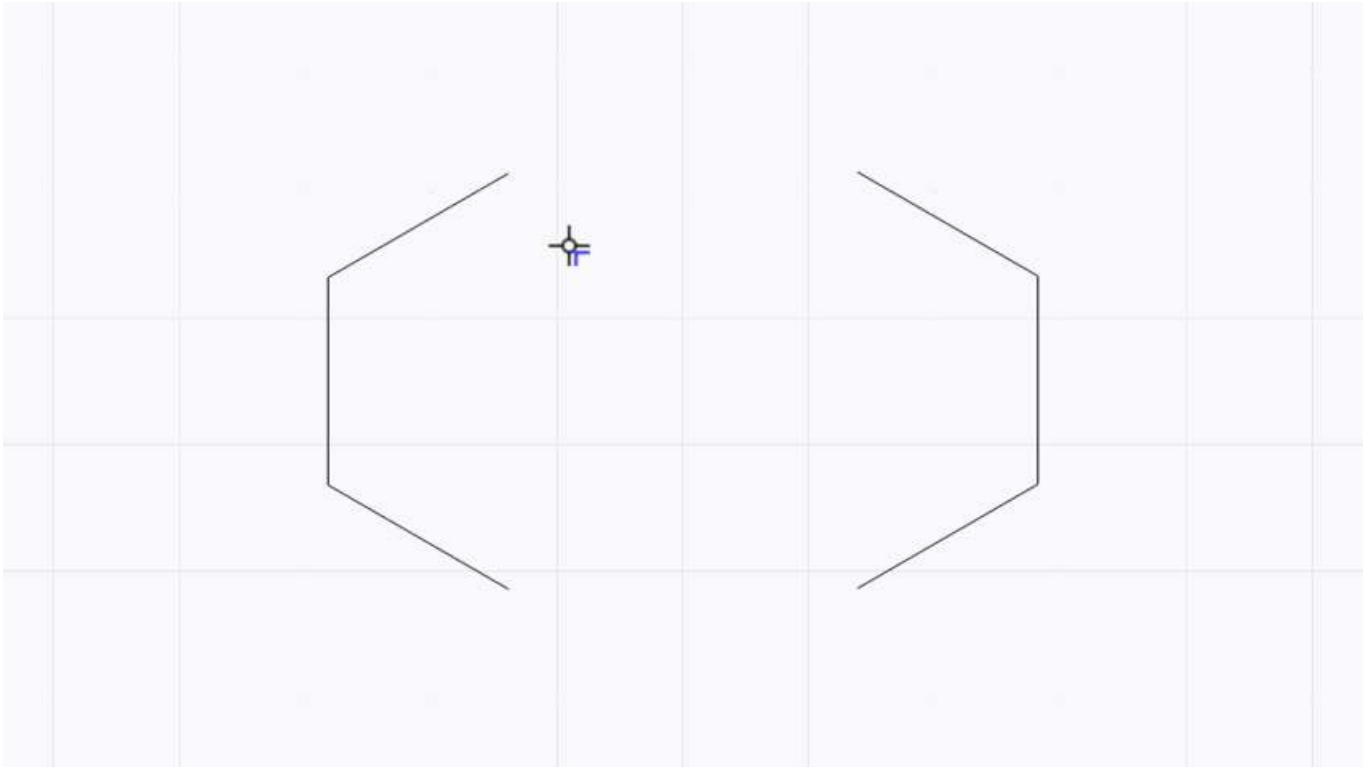


Continue an Existing Drawing

Click on the start or end point of an open path to continue an existing drawing — new lines started from these points are automatically connected to the pre-existing path.

If you create a line or curve that connects the start or end points of two existing open shapes, LightBurn will Auto-Join them, and the new, combined path will be indicated by the animated dash selection pattern.

Hold `Ctrl`/`Cmd` while placing nodes to disable the Auto-Join behavior.

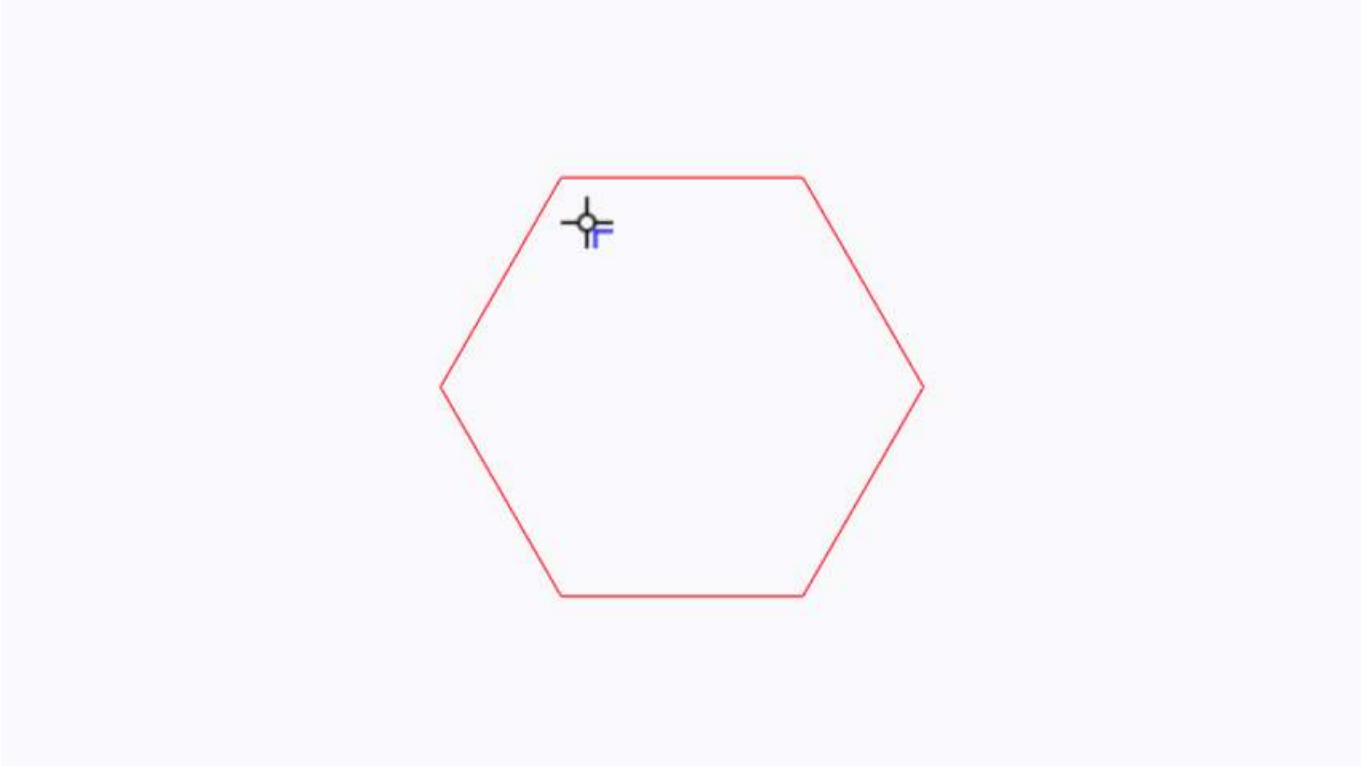


Snapping

In either placement mode, you can add nodes by Snapping to points on existing geometry: lines, corners, midpoints of lines, intersections of lines, and other nodes.

Your cursor will change to indicate the type of Snapping point you're hovering over, and when you move your cursor close enough to a valid point, it will Snap into position — click to place a node perfectly aligned to that point.

Hold `Option`/`Alt` while moving your cursor to Snap to points from farther away.

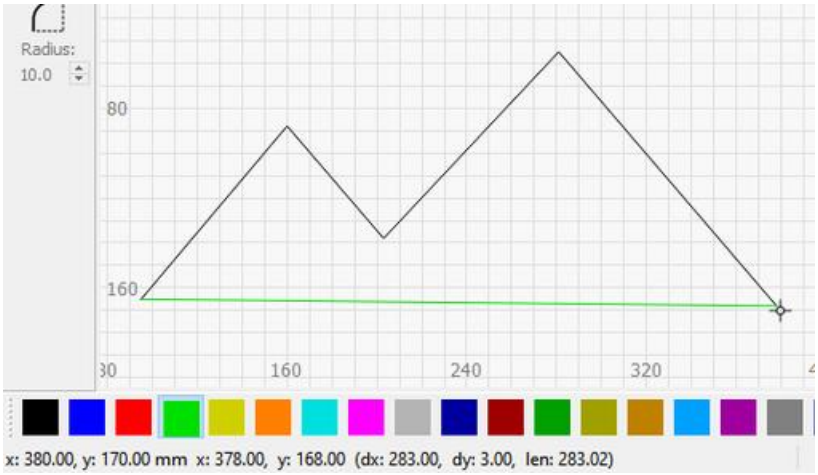


Measuring

To measure while drawing a line, check the Status Bar.

It shows:

- **dx:** the placement of the cursor along the X (horizontal) axis
- **dy:** the placement of the cursor along the Y (vertical) axis
- **len:** the length of the line segment that is currently being placed
- The **angle** between successive line segments as you create them



How to Practice

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=O7kzRu1f3Xs

Related Topics

- Edit Nodes
- Snapping
- Selection
- Images vs. Vectors
- Open vs. Closed Shapes
- Trace Image

Draw Shapes

This page covers the creation and editing of various **Primary Shapes**:

- **Rectangles** and squares
- **Ellipses** (ovals) and circles
- **Triangles**
- **Pentagons**
- **Polygons** with an adjustable number of sides (hexagons, by default)
- **Octagons**
- **Stars**
- **Dual Radius Stars**

The products of these tools are often used to create the foundation of a design, either on their own, or combined with other objects to build up complexity.

These shapes are a special kind of vector object called **Built-in Shapes** that retain editable properties such as number of **Sides** for Polygons, or **Corner Radius** for Rectangles.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=HQL7CEt83Rg

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=Mg6kHnkg9cs

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=CO8ZXHOjxnQ

Shape Selection

Each **Primary Shape** is available in the **Draw Shapes** submenu of the Creation Toolbar or Tools Menu.

In the Creation Toolbar, click the arrow in the lower right corner of the icon, or click and hold anywhere on the icon, to open a submenu with the indivual tools.

Click one of the **Primary Shape** tools from the submenu to enable it.

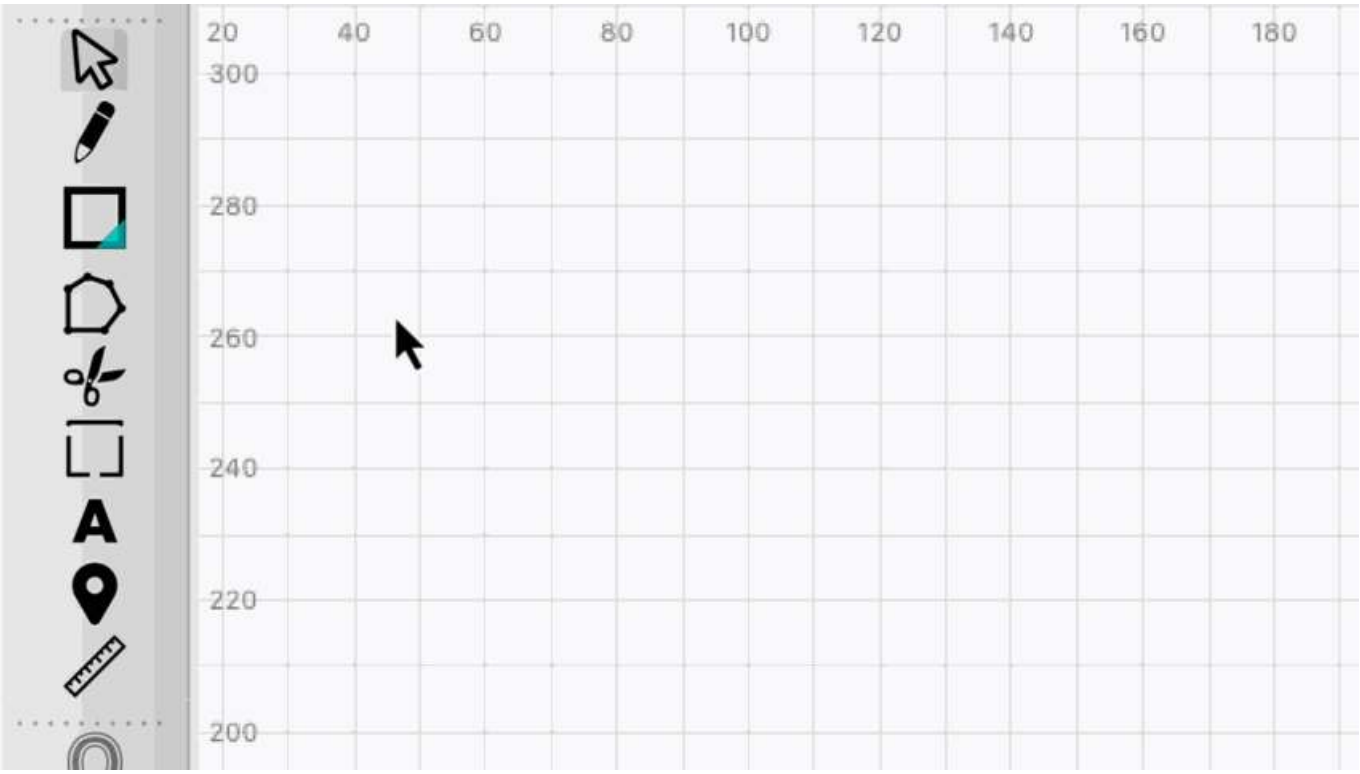
By default, the Draw Shapes icon is set to the **Rectangle** tool, and clicking once anywhere on the icon (other than the arrow in the bottom right) without holding will enable Rectangle creation.

Once you've selected a different shape from the submenu, its icon will appear, and you'll be ready to create it. If you exit the Draw Shapes tool, you can click once to re-enable creation of the last-selected type of shape.

You can also enable Rectangle or Ellipse creation by hotkey:

- **Rectangle/Square**: `Ctrl` / `Cmd` + `R`
- **Ellipse/Circle**: `Ctrl` / `Cmd` + `E`

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=e0-CDS4DxqE



Shape Creation

Click, hold, and drag in your Workspace to create the selected Primary Shape.

The behavior of shape creation changes if you use certain keyboard modifiers:

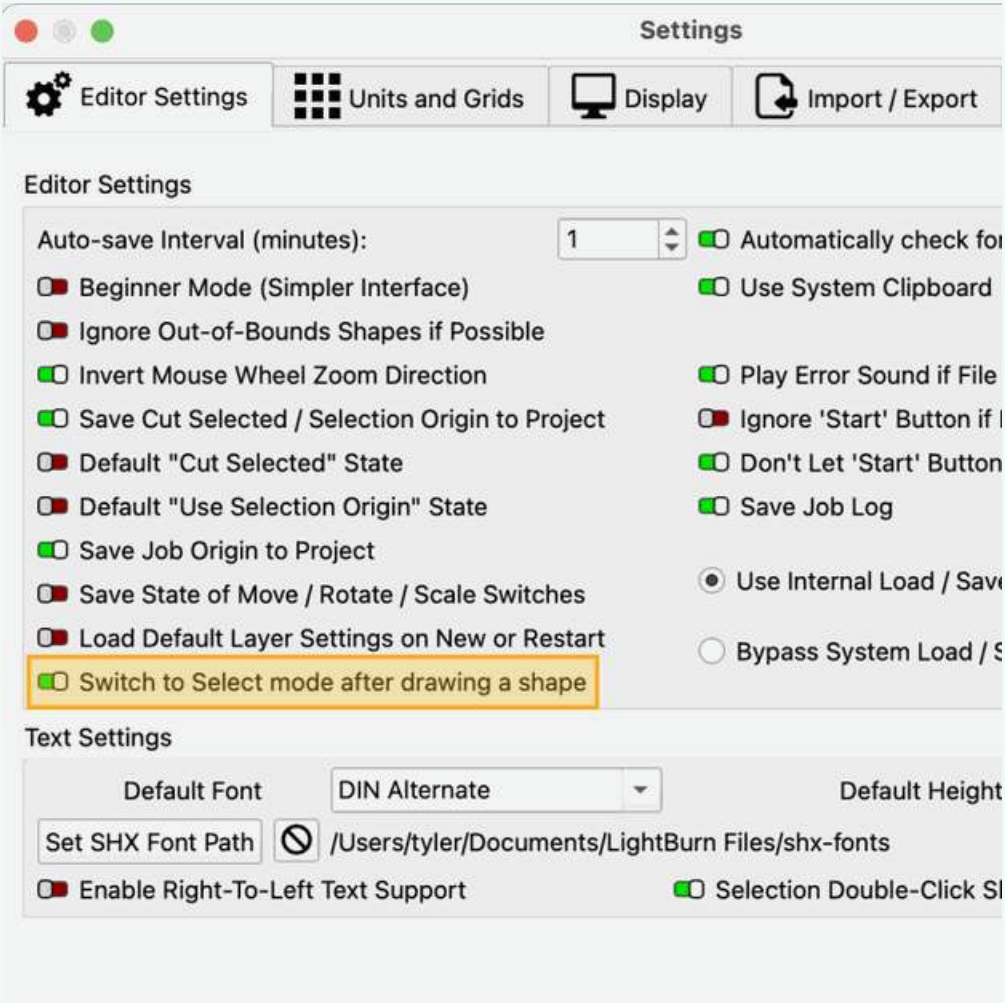
- By default, the shape will begin at the first point you click, and extend from that point in both directions as you drag.
- Holding `Ctrl` / `Cmd` will instead draw the shape from the center outward, which can be useful when aligning shapes to other parts of a design.
- Holding `Shift` while dragging will make the shape into its regular form — the **Rectangle** tool will make a square, the **Ellipse** a circle, any of the polygon tools will make an equal-sided polygon, and the **Stars** will have sides of equal lengths (in the case of **Dual Radius Stars**, there will be two types of sides with matching lengths).
- Hold both keys at once to combine their effects.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=jBFI82KjTtU

Tip

After creating a new shape, you can immediately use the handles to Move, Size, Rotate, or Shear it, while the **Draw Shapes** tool remains active.

Enable Switch to Select mode after drawing a shape in the **Editor Settings** tab of your Settings to immediately exit the Draw Shapes tool after creating a shape and enable Selection mode instead.



▶ Watch the video on YouTube: https://www.youtube.com/watch?v=7KCl0ijnW_k

Editing Primary Shapes

It's possible to adjust various aspects of an existing **Primary Shape** in the Shape Properties Window, or by using special control handles on the shape in your Workspace. The aspects vary between shape types but include dimensions, rotation, **Corner Radius**, and number of **Sides**. These control handles are available both immediately after creating the shape and when selecting a shape you've created previously.

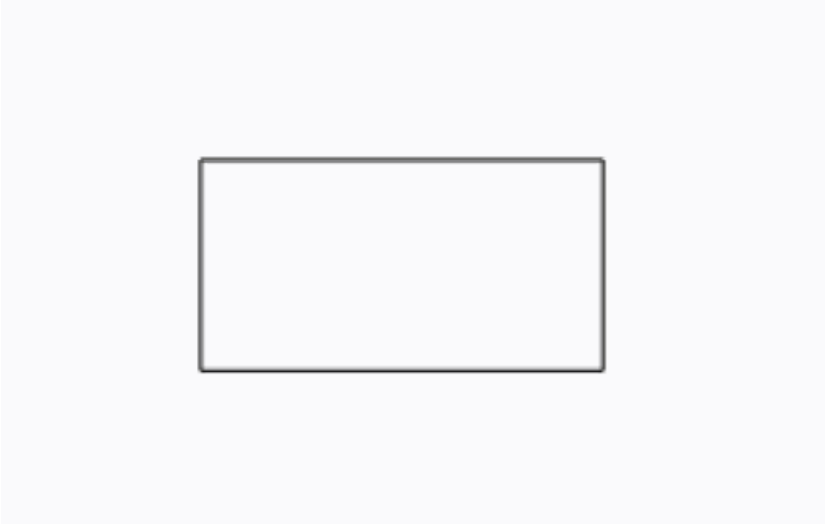
Dimensions

You can change **Width** and **Height** in the Shape Properties Window and the Numeric Edits Toolbar.

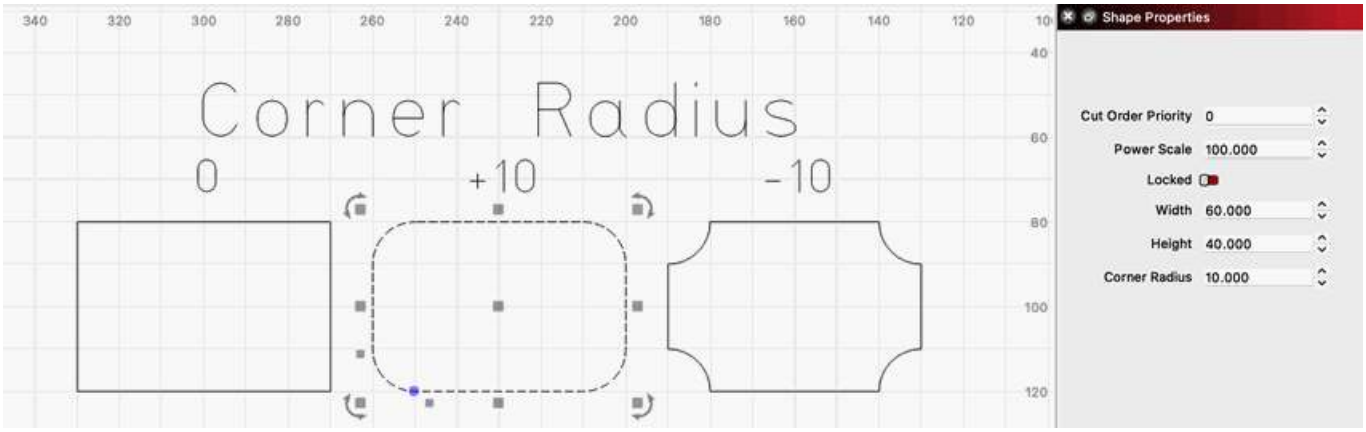
Corner Radius — Rectangle Only

Corner Radius rounds out the corners by a given amount, which remains consistent even if the dimensions of the rectangle are later changed.

With a **Rectangle** selected, hold `Ctrl` / `Cmd`, and a blue control handle will appear. Drag the blue handle away from the corner to increase the radius of all four corners, and drag back toward the corner to decrease it. Dragging vertically creates a reversed radius, with a bite taken out of the corner.



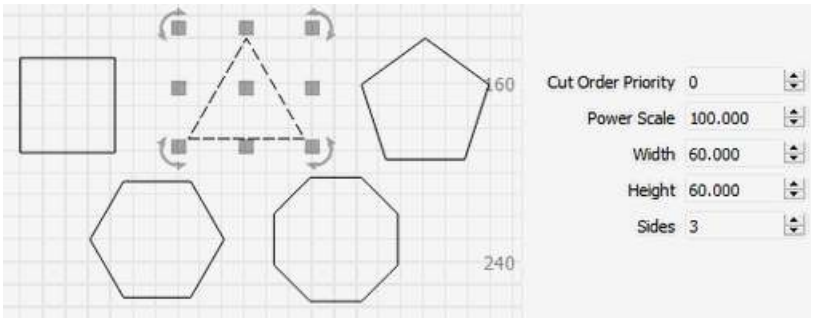
You can also edit Corner Radius numerically in the Shape Properties Window.



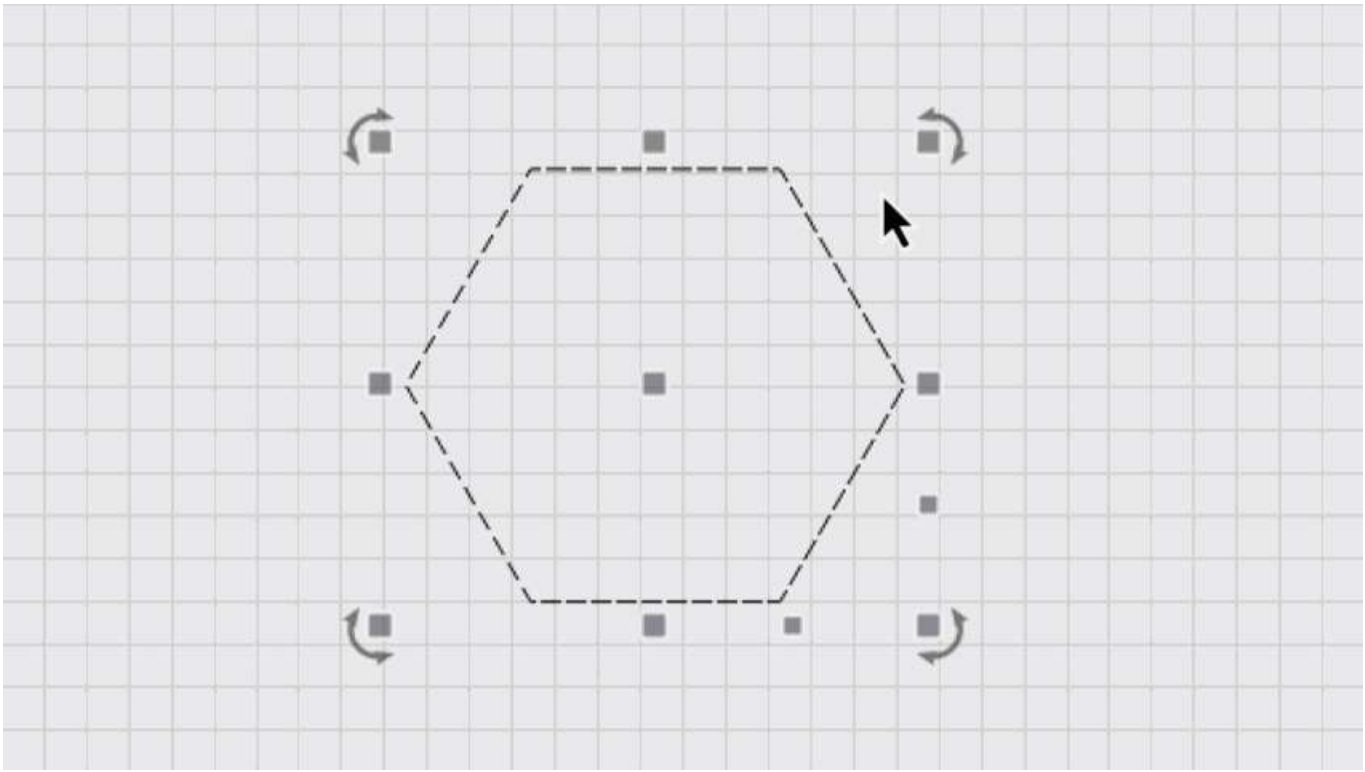
Sides — Polygons Only

Adjust the number of **Sides** of a **Polygon** in the Shape Properties Window to form different shapes — 3 will create a triangle, 6 will create a hexagon, 8 will create an octagon, and so on.

The **Triangle**, **Pentagon**, and **Octagon** tools all create polygons with the appropriate number of sides already set in the Shape Properties window, but each can be changed to a different type of Polygon after they've been created.



It's also possible to change the number of sides by holding `Ctrl` / `Cmd` and dragging the purple control handle that appears near the polygon.



Points, Bulge, and Ratio — Stars Only

Points

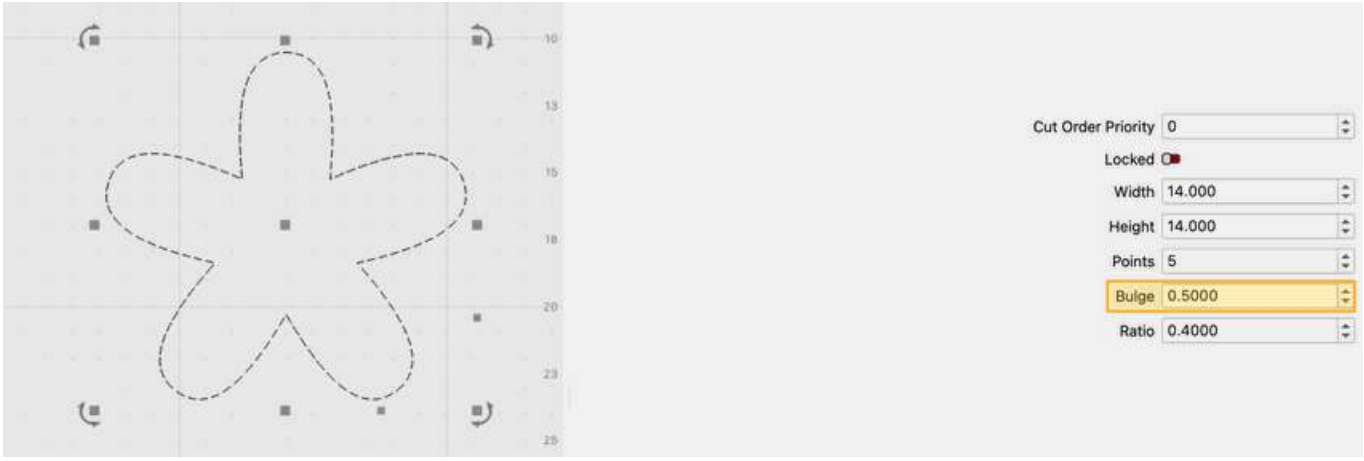
The **Points** field controls the number of points on a **Star**.

For **Dual Radius Stars**, two sets of points are created, the Points field controls the amount of each individual set — in other words, the total number of points is two times the value entered in the Points field.



Bulge

Applying a **Bulge** value creates an arc from the top of each point to its base. Accepted values range from 0 to 1.

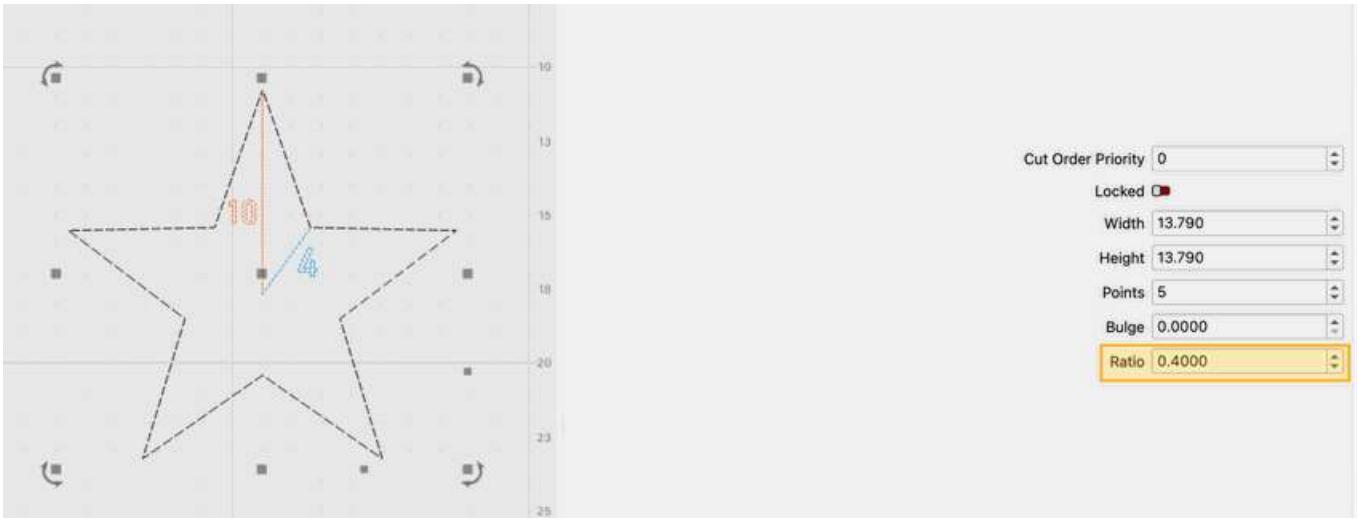


Ratio

Ratio defines the relationship between the distance from the **Star's** center to an outer point and to an inner corner, effectively determining the length and angle of star points.

For **Dual Radius Stars**, you can enter a separate value in Ratio and the **Ratio2** fields to adjust the two sets of points individually.

Accepted values range from 0 to 1.



Node-Editing a Primary Shape

Primary Shapes must be Converted to Paths to make editing their nodes possible. This process permanently removes the ability to edit a shape's dimensions, **Corner Radius**, or number of **Sides** in the Shape Properties Window.

Related Topics

- Create and Edit Text
- Draw Lines
- Shape Properties
- Convert to Path

Text

Use the **Create Text** tool to add text to your projects. Text created using the Create Text Tool is a special type of built-in shape called a **Text Object**.

There are several special settings you can apply to Text Objects through the Text Options Toolbar, Shape Properties Window, or by directly editing them in your Workspace.

Use these settings to adjust font and size, change spacing, bend or curve your text, create Variable Text, and more.

LightBurn 2.1 has a new Font Manager to make it easier to view, search for, favorite, and hide fonts.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=JMsQLfYoaGM

Go to Font Manager

Looking for single line fonts? Find them in the "Zip Files" section

Creating Text



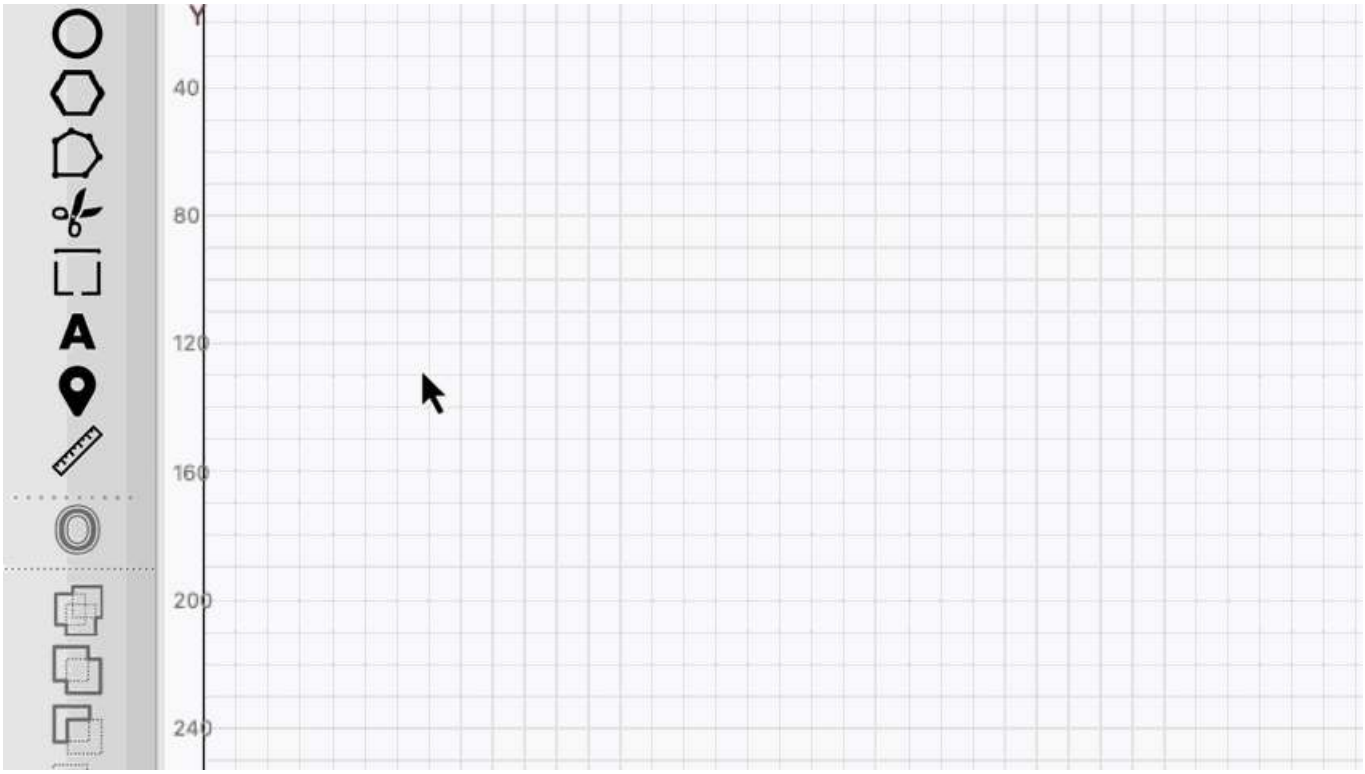
To add a **Text Object** to your project, click the **Create Text** icon in the left sidebar, go to Tools → **Edit Text**, or press `Ctrl`/`Cmd`+`T`.

Click anywhere in your Workspace and type in the text you'd like to add.

Press `Enter` while typing your text to jump to a new line and continue typing.

Press `Esc` to finish creating the Text Object.

> **Note**
>
> By default, the location you click will be the center of the Text Object you create, but you can adjust alignment in the **Text Options Toolbar**.



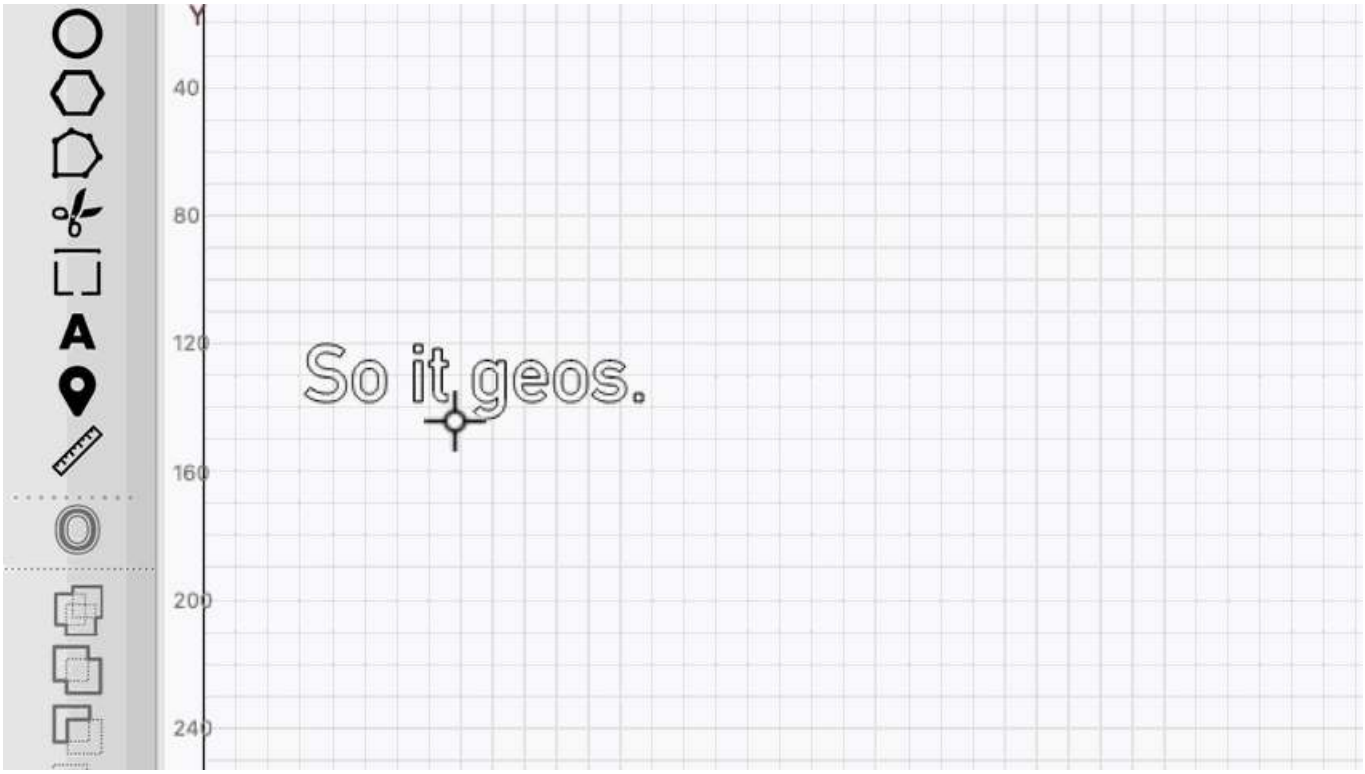
Editing Text

To edit a **Text Object** after creating it, enable the **Create Text Tool** and click on it again, or double-click it without the Create Text Tool enabled.

Right-click any Text Object and select **Edit Text Shape** to open a dialog window where you can directly edit and adjust the text's attributes.

Enable **Selection double-click shows Text Edit dialog** in the Settings window to open the Edit Text Shape window by double-clicking instead.

> **Quick Tip: Text Box Edits in LightBurn**
>
> ▶ Watch the video on YouTube: https://www.youtube.com/watch?v=tTA-3we9R6w



Bending Text

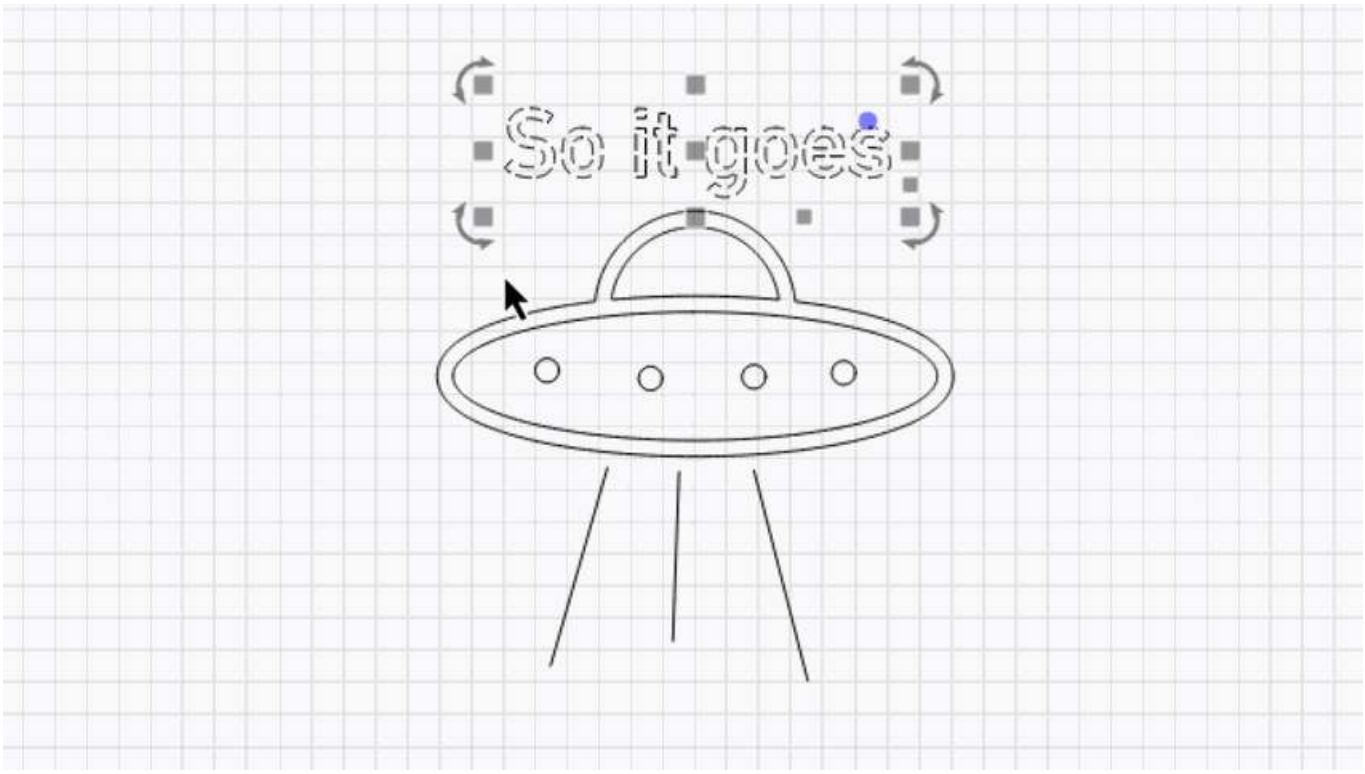
In addition to the Transform Control Handles used to **Move**, **Resize**, **Rotate**, and **Shear** objects, **Text Objects** have a special handle for **Bending**.

Select any Text Object to show a blue dot over the text, then hover your cursor over the dot until you see the Bend cursor

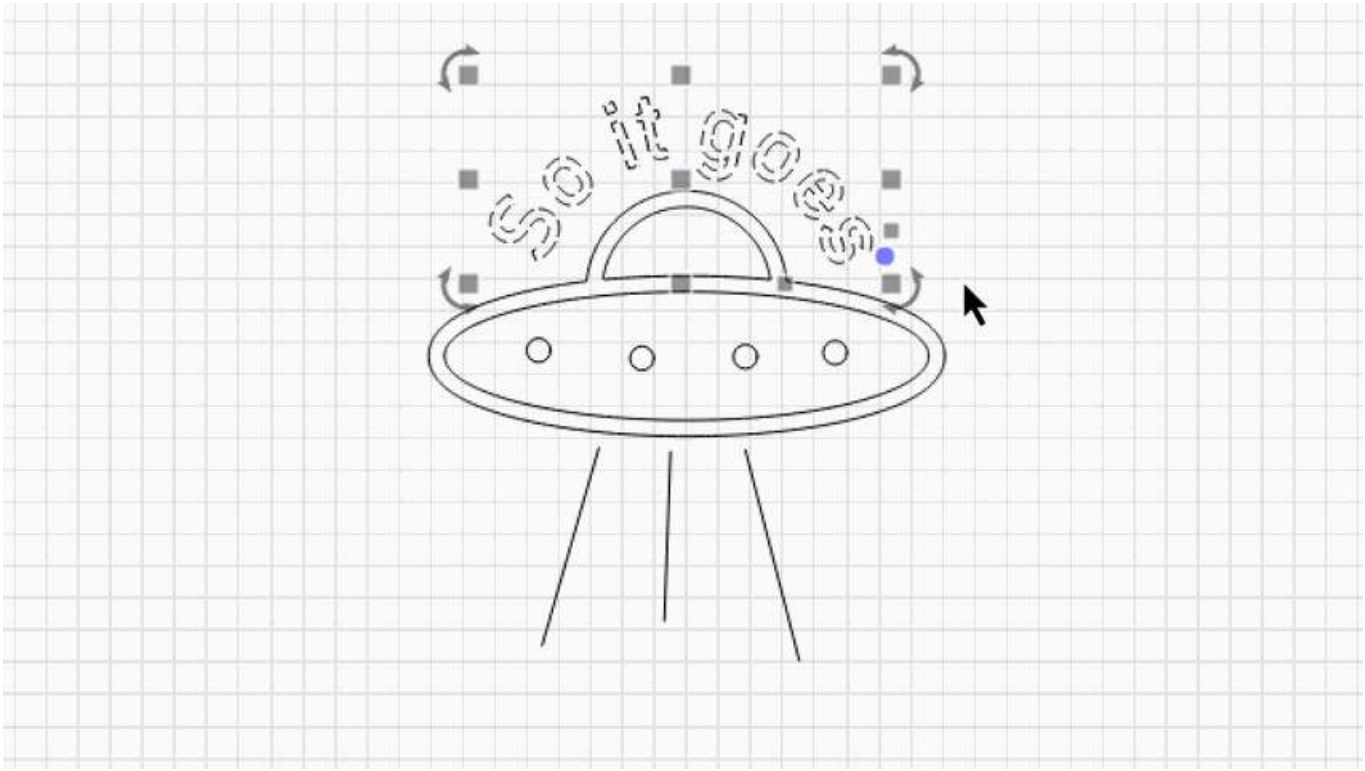


.

Click, hold, and drag the cursor up or down to curve the text around a circle.



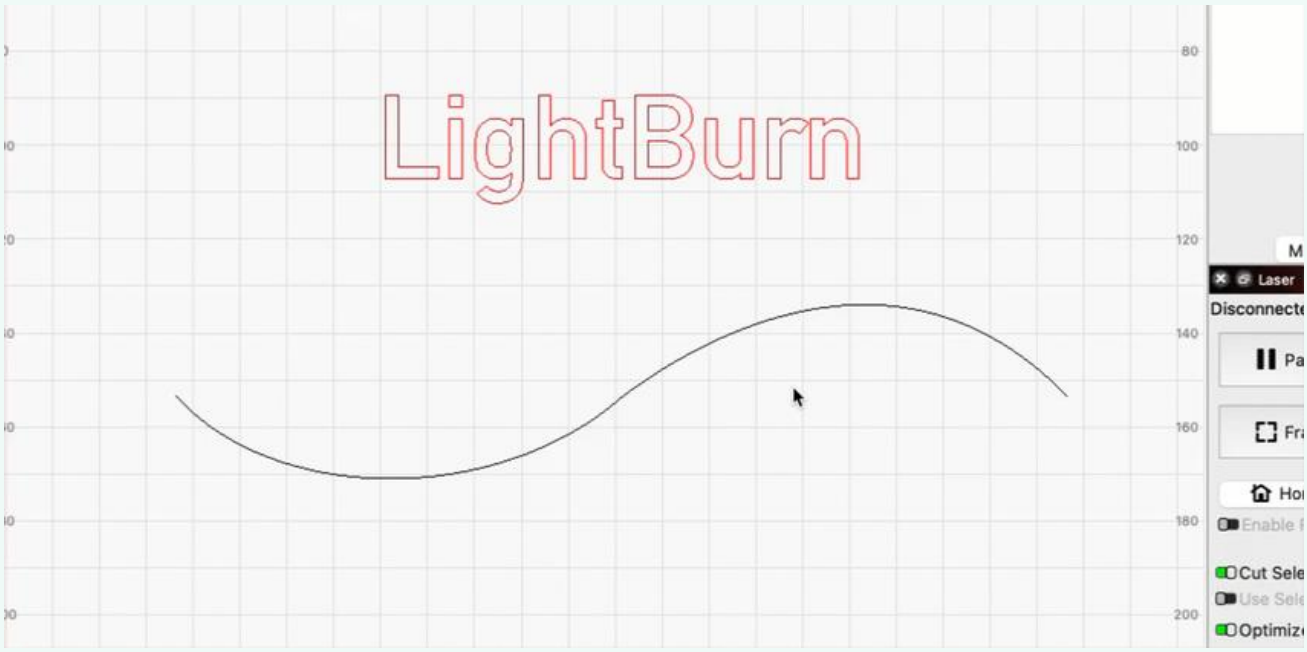
Double-click the dot to clear the curve and restore the text.



Enable or disable the **Distort** toggle in the **Text Options Toolbar** to choose whether to distort the text as part of the curve, or leave the individual characters unchanged.



To curve text around more complex paths, use Apply Path to Text.



.

Text Options Toolbar

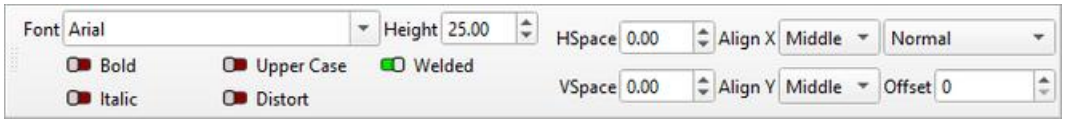
The **Text Options Toolbar** is where you adjust attributes of **Text Objects**, including, **Font**, size (**Height**), spacing (**HSpace** and **VSpace**), and more.

Accessing the Text Options Toolbar

The **Text Options Toolbar** is located in the top toolbar in LightBurn by default, to the right of the Numeric Edits Toolbar. The options in this toolbar are only available when you have a Text Object selected.

> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

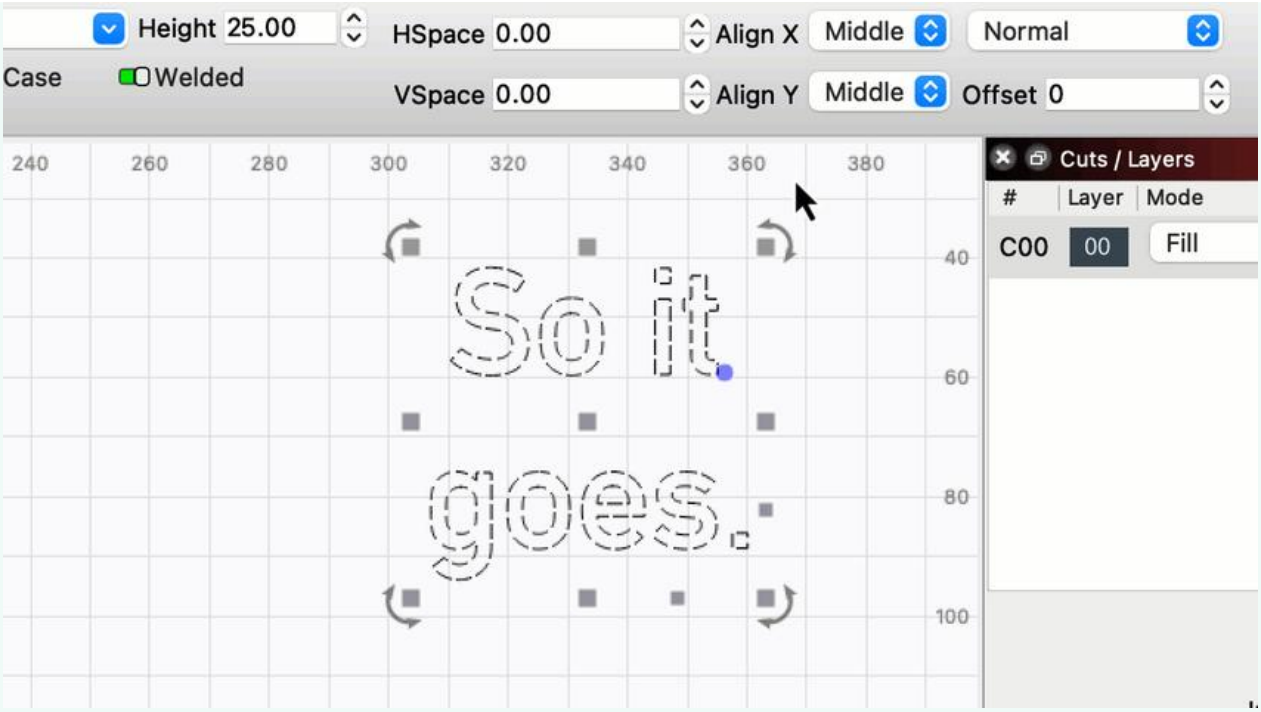
Options



- Use the **Font** dropdown menu to select a a font. See below for more information on fonts.
- **Height** sets the overall height of text. Height varies based on your selected font, and is not exact — it is generally the height of a capital letter X in the font, but every font has an internal size that is scaled by this Height, and the dimensions aren't required to be accurate.
- **HSpace** adjusts the horizontal spacing between individual characters, as a percentage of the font size. Positive numbers space the characters out more, and negative numbers move them closer together.
- **VSpace** adjusts the vertical line spacing as a percentage of the font height. Positive numbers increase the distance between lines, and negative numbers reduce it.

> **Note**
>
> In the case of text, horizontal always refers to the direction of the text as it is read, regardless of whether the Text Object is rotated. Vertical refers to the opposite direction, regardless of rotation.
>
> HSpace always controls distance between individual characters,and VSpace always controls distance between lines.

> **Adjusting HSpace and VSpace**



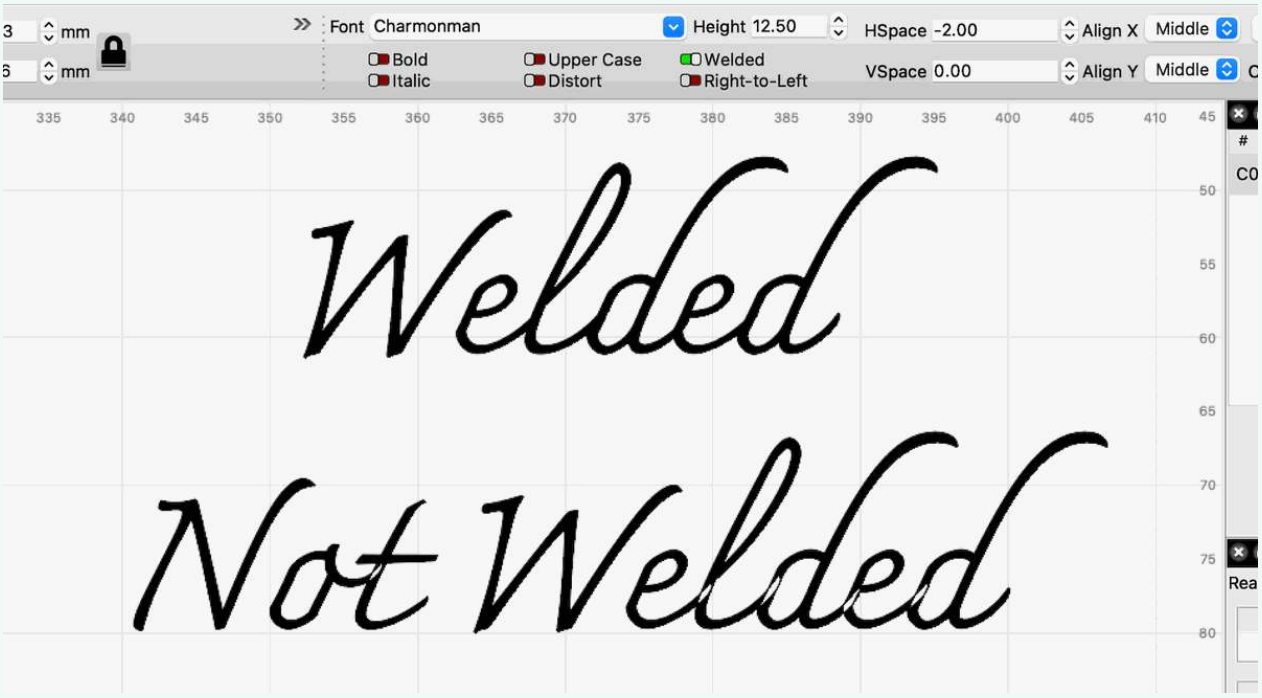
- **Align X** determines the horizontal anchor position of the text — **Left**, **Right**, or **Middle**.
- **Align Y** determines the vertical anchor position of the text — **Bottom**, **Top**, or **Middle**. Top aligns text to the top of capital letters, and Middle aligns to roughly the middle of the capital letters. This is most useful when using Apply Path to Text.

Using Align X and Align Y



- Enabling **Bold** displays the font in bold typeface, if available for the selected font.
- Enabling **Italic** displays the font in italics, if available.
- Enabling **Upper Case** forces all characters to be upper case.
- **Welded** enables automatic welding of characters. When characters touch or overlap, as is common with script fonts, enabling this option will automatically weld the overlaps together. Characters will also weld together if HSpace is reduced to a point where they overlap.

Welded vs. Unwelded comparison



- **Distort** controls whether individual characters in a Text Object will be warped when Bending them.
- Enabling **Right-to-Left** forces Right-to-Left text rendering. This option will not appear unless it's enabled in your Settings. Right-to-Left will automatically be enabled if your current input language is Right-to-Left.
- The Text Mode dropdown menu and Offset field are used when working with **Variable Text**. For information on these options, see Variable Text.

  Leave this value set to **Normal** if you are not using Variable Text

Fonts

The list of fonts in LightBurn is taken from your computer system. If you want to use a new font in LightBurn, install it on your computer, then re-start LightBurn. For single-line SHX fonts, see SHX Fonts.

Installing fonts

On Windows, after you have downloaded a font, right-click it and choose **Install**. On Mac, double-click the font and click **Install** in the font preview window that opens.

Font Manager

LightBurn 2.1 has a new **Font Manager** for working with fonts. If you're using an older version of LightBurn, please use the version dropdown at the top of this page to go to the correct version or go to LightBurn 2.0.

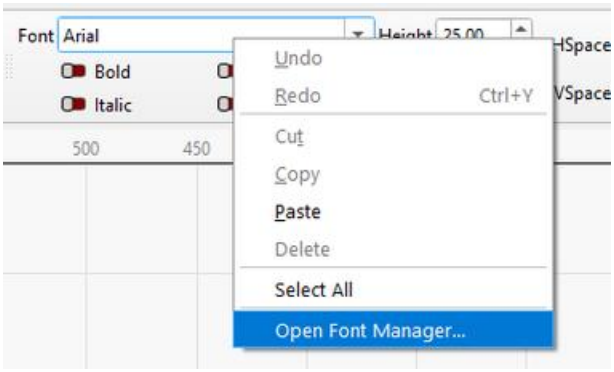
Quick Tip Video: Font Filtering

Our Quick Tip video on font filtering shows the older interface from LightBurn 2.0 and earlier, but is still a good overview of the usefulness of filtering and favoriting fonts. Information on the Font Manager is below.

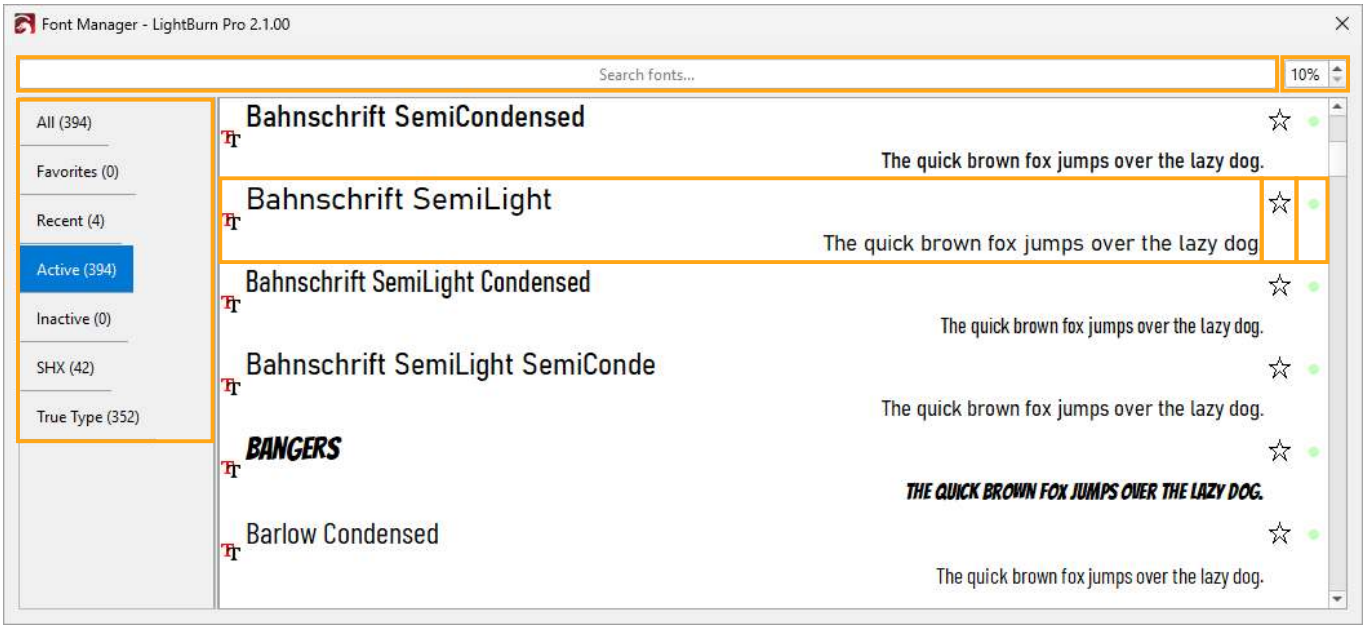
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=jmdEKwefBGs

Your favorite fonts will appear at the top of the font list, followed by your most recently used fonts.

For additional options, while the font list is active, right-click the font list and select **Open Font Manager...**.



The **Font Manager** allows you to search installed fonts, view both SHX and regular fonts, manage your favorite fonts, and hide and unhide fonts.



Search

Use the search bar in the Font Manager to search for fonts by name.

Zoom Level

Use the zoom level controls to set the size of the Font Manager's font previews.

Font Categories

The categories on the left side of the Font Manager help you find and manage your fonts. The **All** category shows every font installed on your computer. The other categories break your fonts down further:

- **Favorites** shows fonts marked as favorites.
- **Recent** shows recently used fonts.
- **Active** shows all fonts not marked as hidden.
- **Inactive** shows all fonts marked as hidden. Look here if LightBurn isn't showing a font you expect to see.
- **SHX** shows all SHX fonts in your SHX Font Directory. This is only available if you have set an SHX file path and have at least one SHX font located there. See below for more information on SHX Fonts.
- **True Type** shows all regular (non-SHX) fonts installed on your computer. This is only available if you have set an SHX file path and have at least one SHX font located there.

Font Preview

The Font Preview shows the font name and a sentence using it in the font itself.

Set Favorite

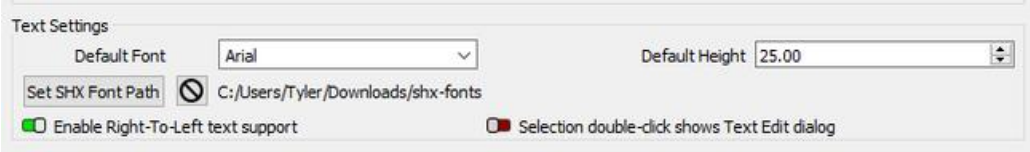
Click the star next to a font preview to set it as a favorite. Your favorite fonts have a filled in star next to them - click again to remove the font from favorites.

Hide Font

Click the green circle next to a font preview to set it hidden/inactive. To unhide a font, go into the **Inactive** category and click the outlined circle next to the preview.

Text Settings

There are several global font settings in the **Editor Settings** tab of the Settings window.



- **Default Font** changes the automatically selected font for new projects.
- **Default Height** changes the automatically applied font **Height** for new projects. If you change this value, you need to close and relaunch LightBurn before the change will be applied to new projects.
- **Enable Right-to-Left text support** allows you to turn on Right-to-Left font rendering for any text. This setting is automatically enabled on first run if a Right-to-Left language is the current system input language, but it can be forced on here later if necessary.
- Enable **Selection Double-Click Shows Text Edit Dialog** to open a text edit window when you double-click text in your Workspace.

> **Quick Tip: Text Box Edits in LightBurn**
>
> ▶ Watch the video on YouTube: https://www.youtube.com/watch?v=tTA-3we9R6w

SHX Fonts

Looking for single line fonts? Find them in the "Zip Files" section

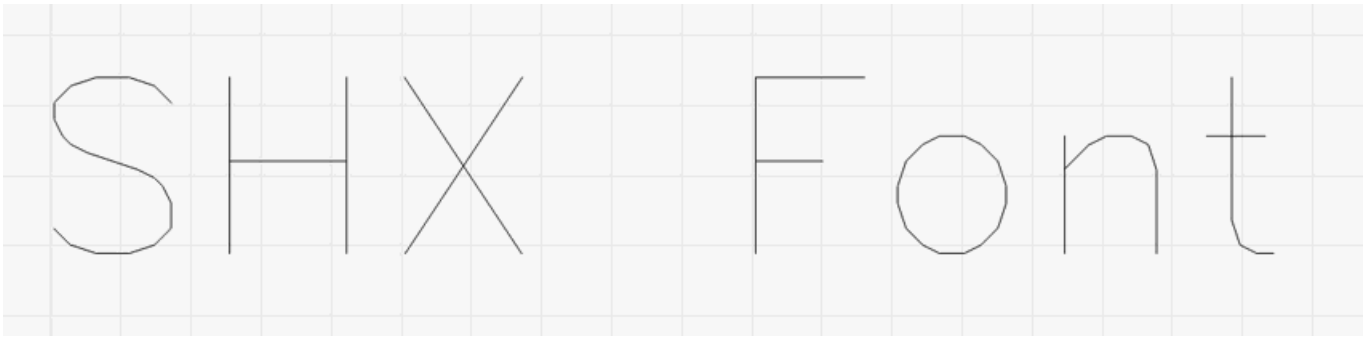
Set SHX Font Path sets the directory containing SHX font files — clicking this button will open up your system's file explorer window, where you can navigate to the location of the SHX font folder on your computer. Click the button next to Set SHX Font Path to clear the path.

Once LightBurn knows where to load the SHX fonts from, they will automatically appear in the **Font** dropdown menu in the **Text Options Toolbar**.

SHX Fonts are differentiated from normal fonts by the icon next to each entry:

- SHX denotes an SHX font.
- TT denotes a standard operating system font.

SHX fonts lack many of limitations of standard system fonts and have the ability to define single line fonts, such as this:



Moving LightBurn Files With Fonts

If you want to move designs containing text between computers and continue to edit the **Text Objects**, be sure to install all necessary fonts on all computers you're using.

When you save a copy of a file that contains text, LightBurn automatically includes a copy of the text as a path and embeds it into the file. If you open the file on another computer that has the same font, you will be able to edit the Text Objects as usual.

However, if you open the design on a computer without the font, LightBurn will warn you about the missing font(s) and load the path embedded in the file. This prevents the file from breaking when you move to another computer, but you won't be able to edit the text without the correct font.



If the font is installed at a later time, LightBurn will recognize it and the text will become fully editable again.

Right-click the imported text and select **Convert to Text** to change the font of the text, and regain the ability to fully edit it.

Right-click the imported text and select Convert to Path to change the text into a standard vector graphic that can be edited using the Edit Nodes tool. Note that this change is permanent, and you'll be unable to edit any of its text properties later.

Importing Other Files With Text

If you open files not made in LightBurn that contain text, make sure that any fonts used in the file are installed on your computer, or that the text has been converted to paths using whatever tools are available in the other program.

Different programs use different terms for this process, but terms such as welding, convert to outline, and convert to path are common.

Converting Text to Paths in LightBurn

In order to directly edit the lines and nodes of text in LightBurn, select all **Text Objects** you wish to convert, then go to Edit → **Convert to Path**, press `Ctrl`/`Cmd` + `Shift` + `C`, or right-click in your Workspace and select Convert to Path from the menu.

Read more about Convert to Path here.

Related Topics

- Variable Text
- Apply Path to Text
- Convert to Path
- Rectangles, Ellipses, and Polygons

Variable Text

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=5KONOAPuXV8

Use **Variable Text** to have LightBurn automatically substitute specially formatted expressions with alternate data when you output a project to your laser.

In addition to outputting alternate text when you Start or Send a project to your laser, the expressions will be replaced when you Preview, save a project in machine-specific format, or Export graphics in SVG, DXF, or AI format.

You can use Variable Text to replace special expressions with:

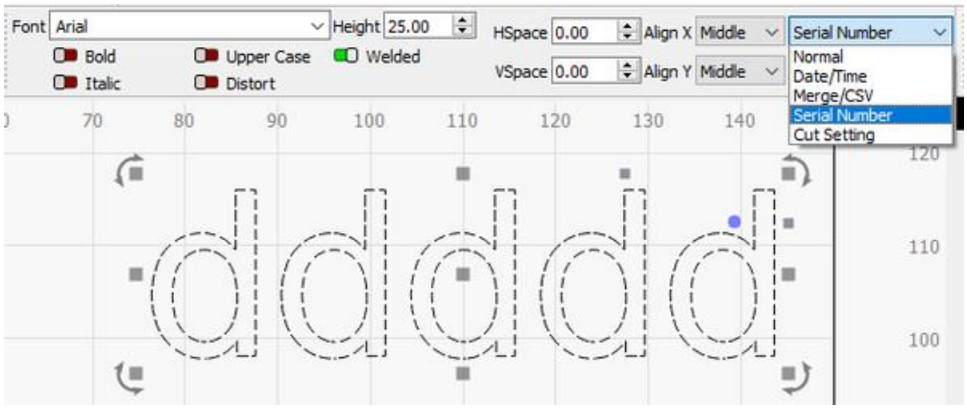
- Date or time stamps
- Entries in a CSV file
- Serial numbers
- Cut Settings

See Variable Text Formatting for full details on supported formatting expressions.

Selecting a Variable Text Mode

To create **Variable Text**, first use the Create Text tool to enter the appropriate expressions for the type data you'd like to output, then select the matching **Text Mode** from the dropdown menu in the Text Options Toolbar.

Select Date/Time (for date and time stamps), Merge/CSV (for entries in a CSV file), Serial Number, or Cut Setting.



If you select Merge/CSV or Serial Number formatting, you'll have additional controls available in the **Variable Text Window**.

Accessing the Variable Text Window

The **Variable Text Window** is not enabled by default. To enable it, go to Window → **Variable Text**.

The Variable Text Window will appear docked behind the Cuts / Layers Window on the upper right side of your screen. Click the Variable Text tab to bring the Variable Text Window to the front.

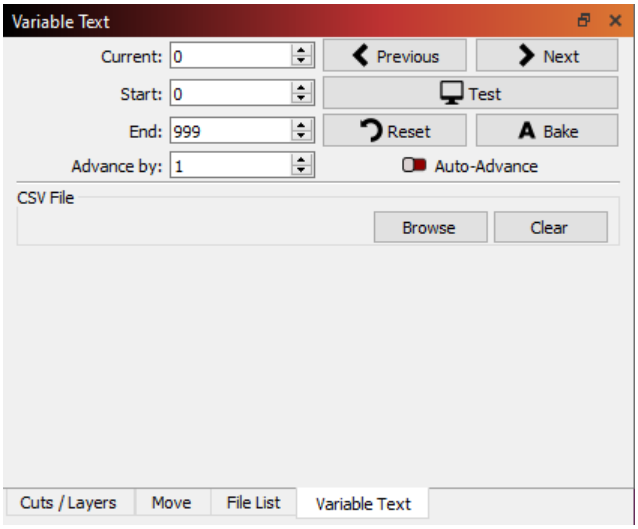


Tip

The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Variable Text Window Options

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Current

Determines the serial number or row from a CSV file that will be substituted in place of the formatted expressions the next time the project is output.

Note

The **Current** value advances between the **Start** and **End** values. In other words, the sequence returns to the Start value when increasing beyond End, or to the End value when decreasing beyond Start.

Start

Determines the first serial number or row from a CSV file to be output.

Warning

> For values other than 0, on the first run of a project you must Reset the **Current** value to the **Start** value, or the values will not match. The Current and Start values are only matched automatically when advancing forward from the **End** value, returning to the beginning of the sequence.

End

The last serial number or row from a CSV file to output, before advancing back to the **Start** value.

Advance By

Tells LightBurn how many entries to advance (or reverse) the **Current** value each time you click **Next** or **Previous**.

Auto-Advance

When enabled, tells LightBurn to advance the **Current** value by the amount entered in the **Advance by** field each time you Start, Send, or save a project in machine-specific format.

Previous

Decreases the **Current** value by the amount set in the **Advance by** field.

Next

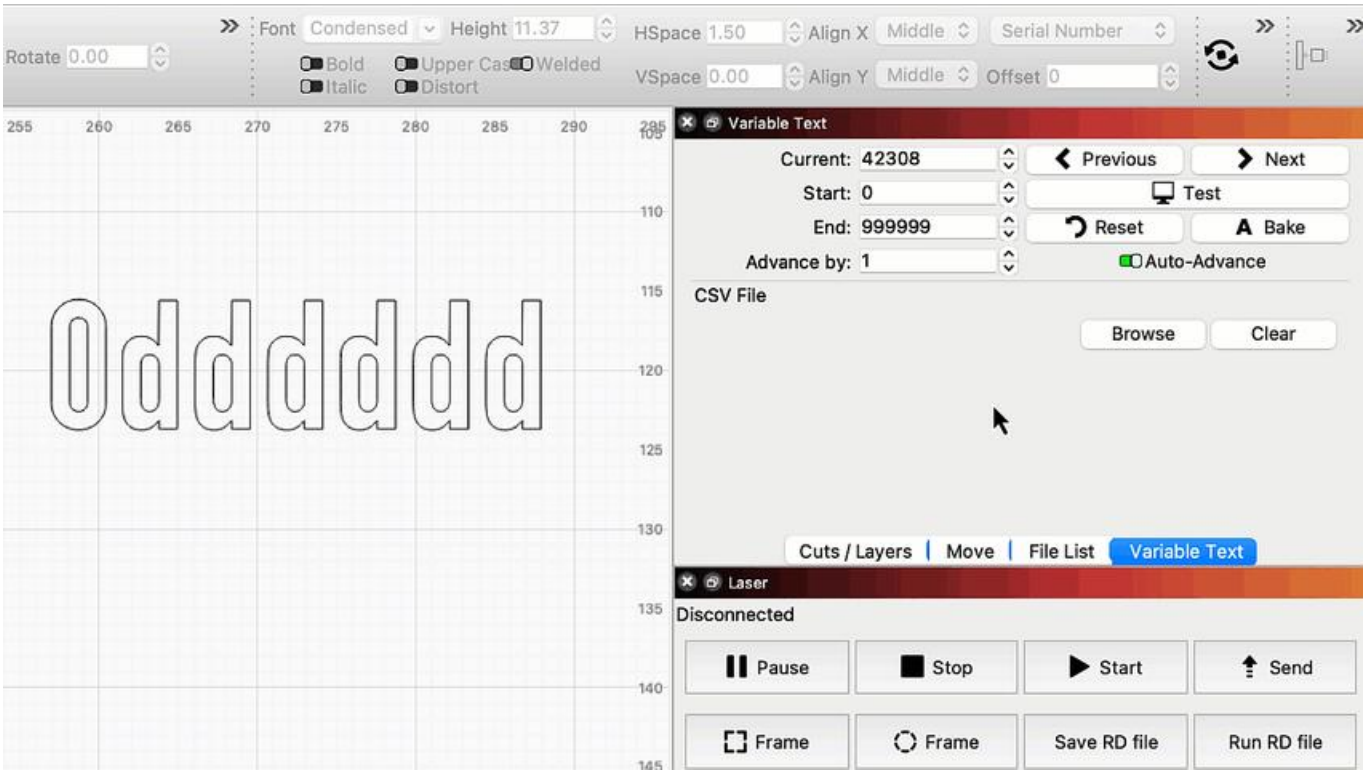
Decreases the **Current** value by the amount set in the **Advance by** field.

Reset

Resets the **Current** value to the **Start** value.

Test

Displays the text that will be output in place of the formatted expressions, for as long as the button is held.



Using Variable Text in Virtual Arrays

When using Variable Text in a **Virtual Array**, the Test button will not show the final output of the Variable Text in LightBurn's workspace. To see an accurate preview, you'll need to use the Preview window.

Bake

Converts the formatted expressions to whatever data would be output based on the **Current** value in the **Variable Text Window**, in your Workspace.

Hold down the `Shift` key while clicking the **Bake** button to bake only the selected variable text.

The Baked text will no longer increment as a Variable Text Object — the displayed text is exactly what will be output to your laser.

Browse

Opens your systems's file browser, where you can navigate to and load a CSV file saved to your computer. The name of the CSV file will display in the **Variable Text Window** after it has been loaded.

Special Characters Displaying Incorrectly

If special characters don't display correctly when using Merge/CSV, please make sure your file is encoded as UTF-8 or newer. Files encoded as ANSI may not display special characters correctly. See more in Variable Text Formatting.

Clear

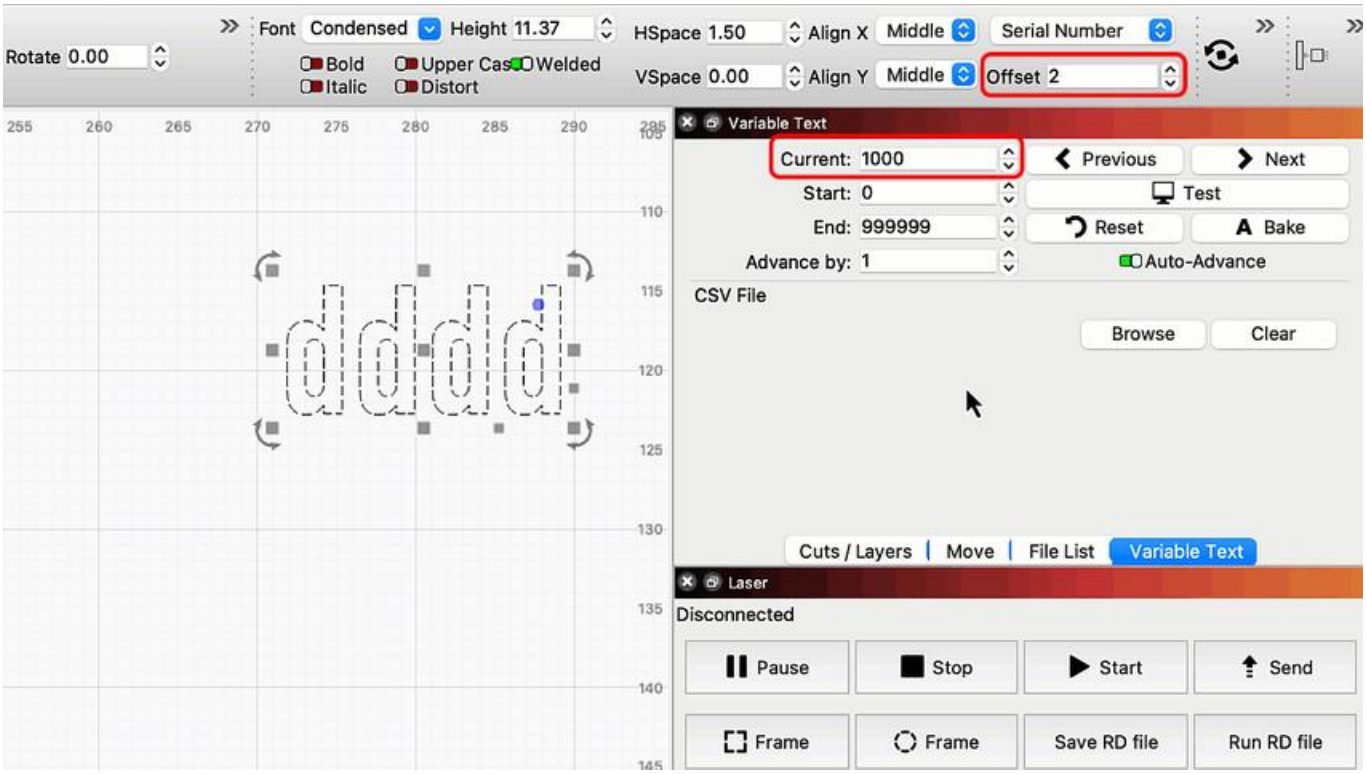
Unloads a previously loaded CSV file.

Variable Text Offsets

Text Objects in LightBurn have an adjustable property called **Offset**, available in the Text Options Toolbar. Using Offsets allows you to use the same formatting expression to output different data.

The Offset value is added to the **Current** value in the **Variable Text Window** when LightBurn evaluates the Text Object to determine the data to output in its place.

For example, if you applied an Offset of 2 to a Text Object formatted to output a serial number, with a Current value of 1000, the output serial number will be 1002.

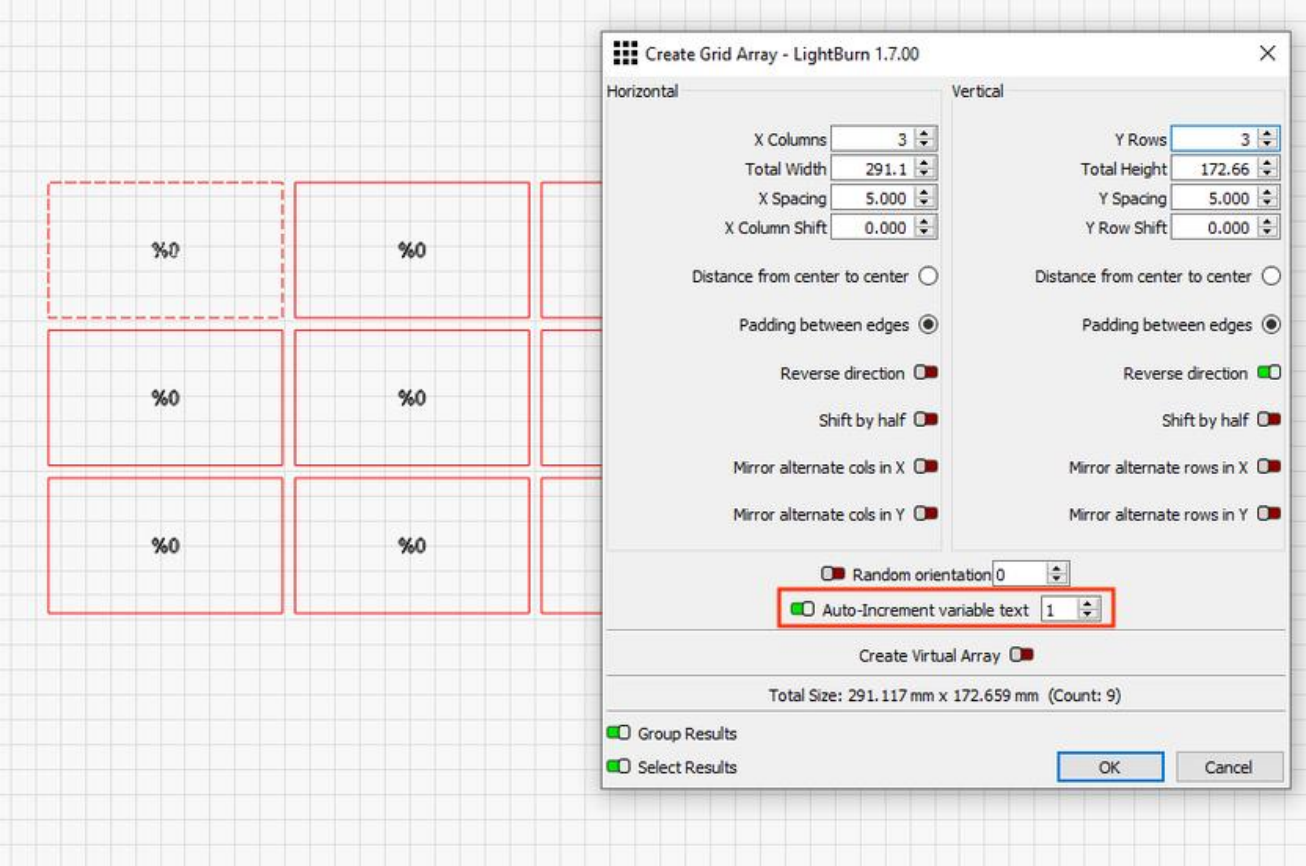


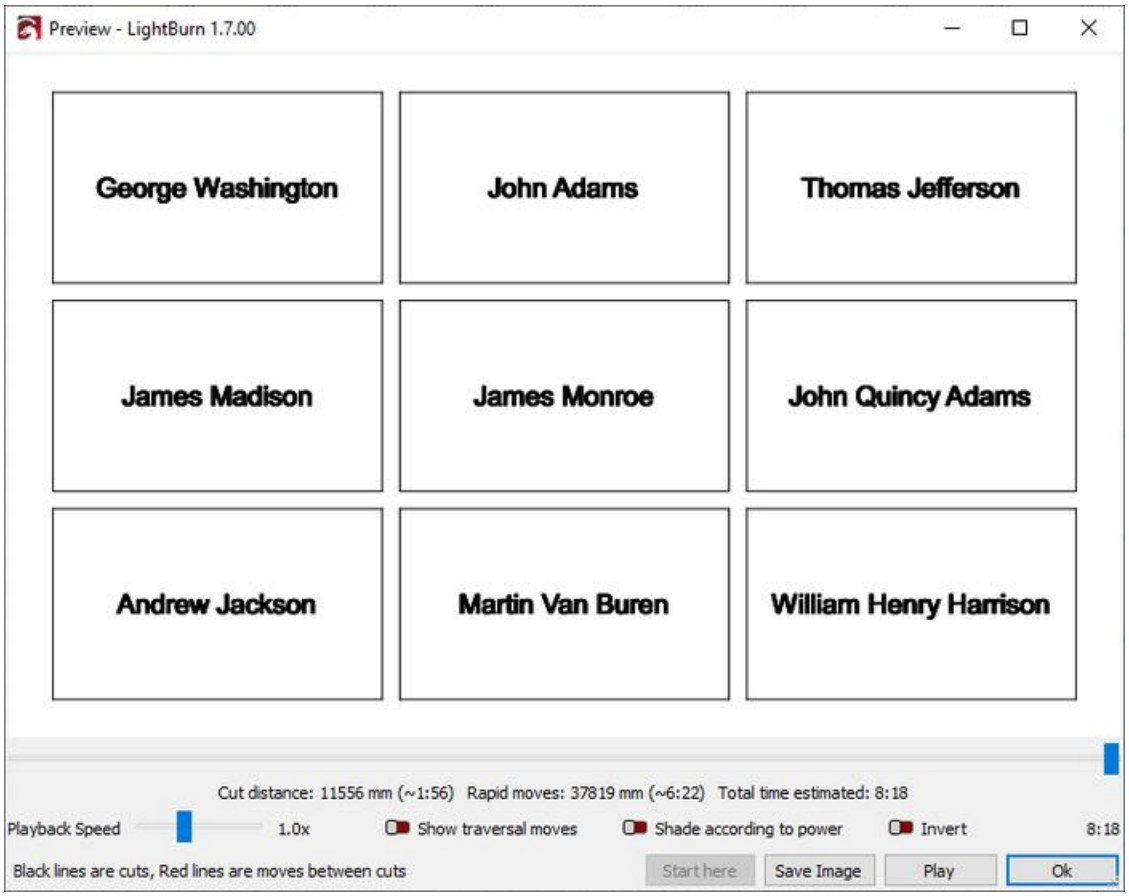
> **Tip**
>
> Use the **Offset** value in conjunction with the Advance by value to increment batches of the same design using **Variable Text**. For instance, if you created a design with 4 name tag labels on the page, you would set the Offset value for the four labels to 0, 1, 2, and 3, and set the **Current** value to Advance by 4 with each run.

Automatic Offsets

Some LightBurn tools can apply automatically advancing **Offset** values to duplicated objects.

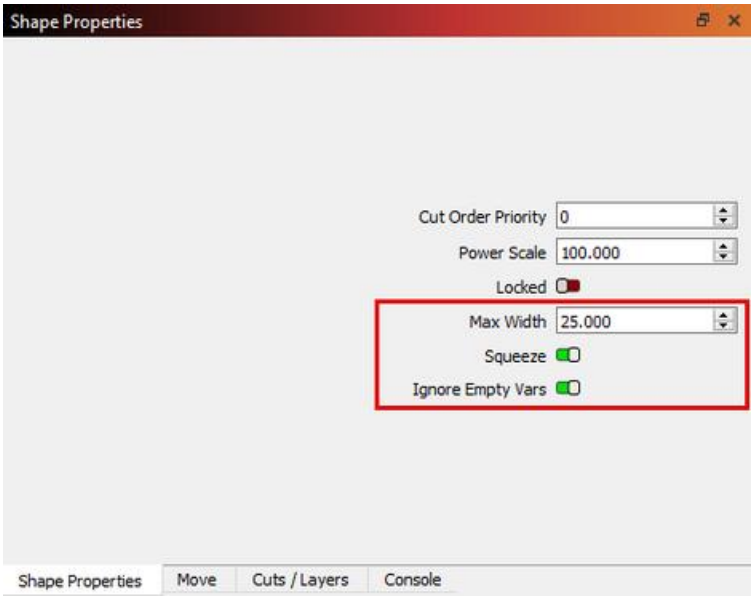
Use Grid Array, Circular Array, or Copy Along Path to lay out copies of designs while increasing the Offset of each copy by a specified amount.





Shape Properties and Variable Text

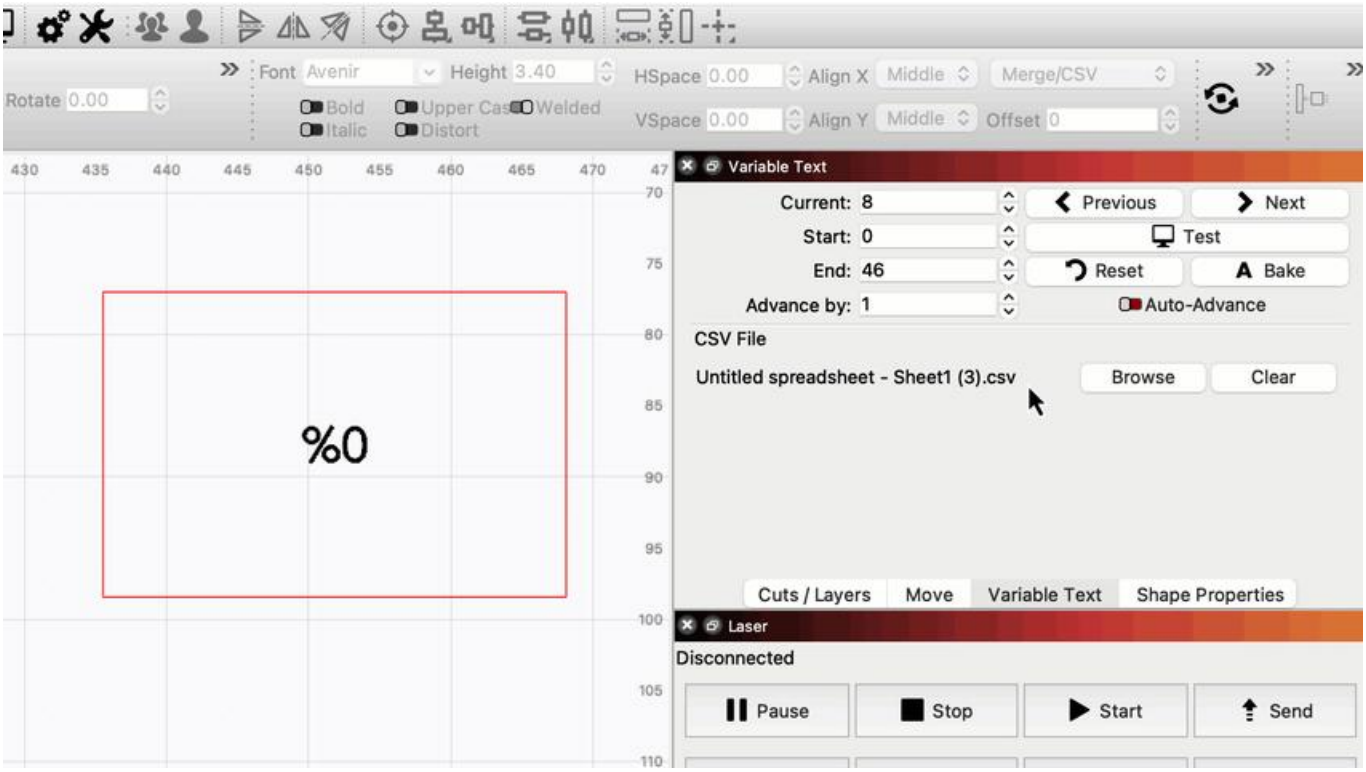
There are three settings in the Shape Properties Window that are useful when working with **Variable Text**.



- **Max Width** sets the maximum possible width of selected Text Objects — use this field to prevent Variable Text outputs from exceeding a specified width after they're converted from formatted expressions.
- The **Squeeze** toggle determines whether Text Objects will be rescaled in both directions in order to fit within the Max Width value.

  When enabled, text will be shrunk horizontally, if necessary to stay within the Max Width, but maintain the same height.

  When disabled, text will be shrunk proprotionally, both horizontally and vertically.
- **Ignore Empty Vars** instructs LightBurn to remove newlines preceding empty Variable Text fields when aligning text.



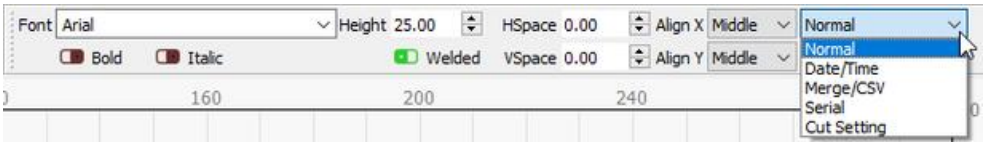
Related Topics

- Variable Text Formatting
- Create and Edit Text
- Repeat Marking
- Grid Array
- Circular Array
- Copy Along Path

Variable Text Formatting

Use the following formatting expressions with the appropriate Variable Text mode to have LightBurn replace the expressions with alternate data when you output a project to your laser.

In order for each type of formatting expression to be replaced with alternate data, you must also select the corresponding mode from the dropdown menu in the Text Options Toolbar.

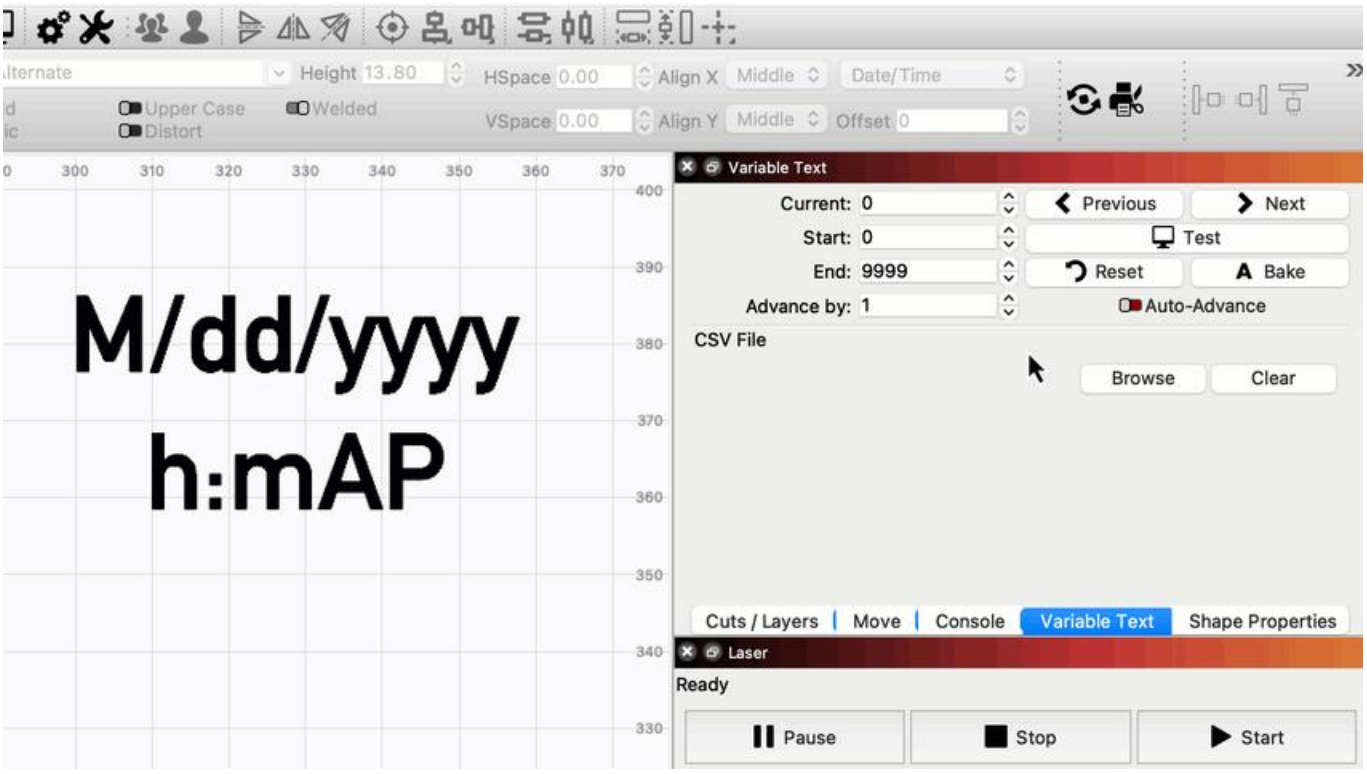


Tip

Use the Test button to verify that you have entered the formatting expressions correctly, before sending a project to your laser.

Date/Time

When using **Date/Time** formatting, LightBurn will automatically substitute special combinations of characters with values for the current local date and time, as determined by your system's clock.



Use the following expressions for dates:

Output	Expression
The day as number without a leading zero (1 to 31)	d

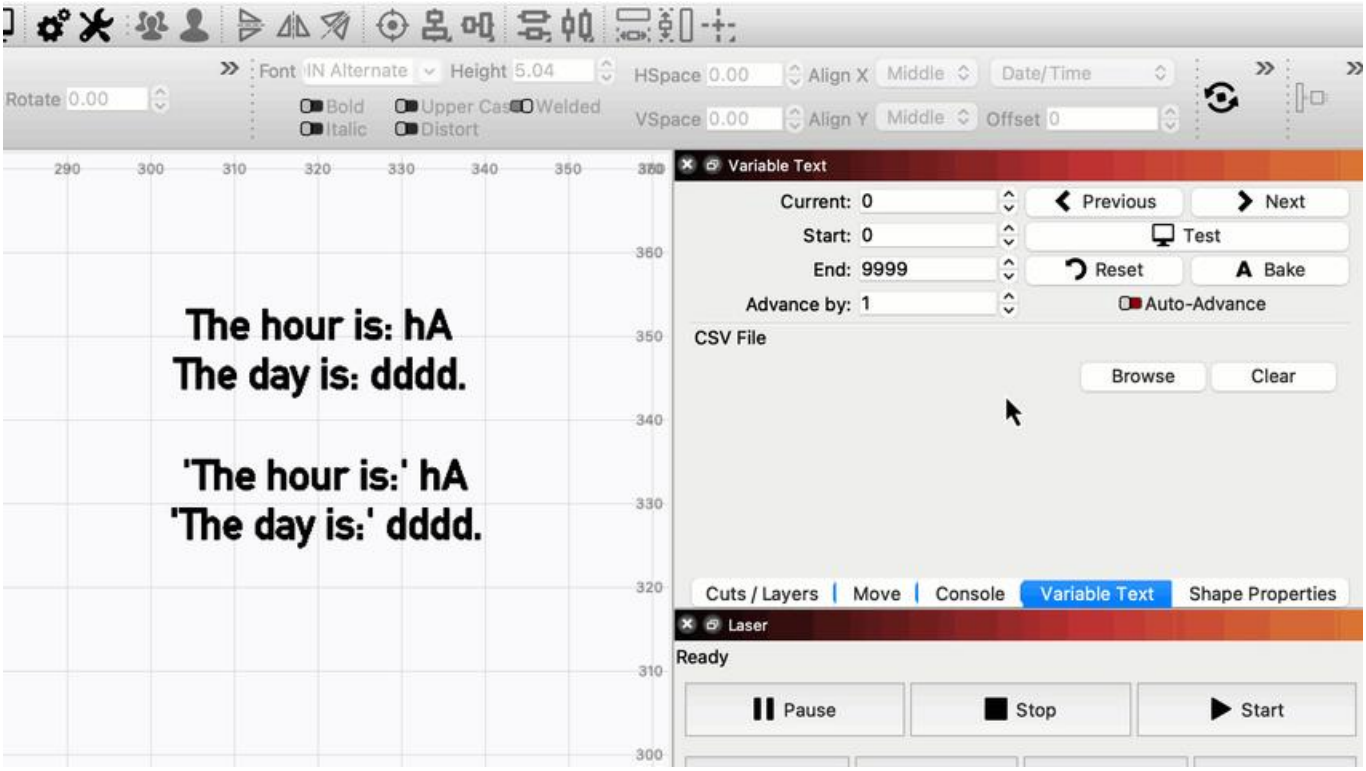
Output	Expression
The day as number with a leading zero (01 to 31)	dd
The abbreviated localized day name (e.g. "Mon" to "Sun"). Uses the system locale to localize the name.	ddd
The long localized day name (e.g. "Monday" to "Sunday"). Uses the system locale to localize the name.	dddd
The numeric week of the year in ISO-8061 format (e.g. 51)	w
The year/week of the year in ISO-8061 format (e.g. 2025/51)	W
The month as number without a leading zero (1-12)	M
The month as number with a leading zero (01-12)	MM
the abbreviated localized month name (e.g. "Jan" to "Dec"). Uses the system locale to localize the name.	MMM
the long localized month name (e.g. "January" to "December"). Uses the system locale to localize the name.	MMMM
the year as two digit number (00-99)	yy
the year as four digit number	yyyy

Use the following expressions for time:

Expression	Output
h	The hour without a leading zero (0 to 23 or 1 to 12 if AM/PM display)
hh	The hour with a leading zero (00 to 23 or 01 to 12 if AM/PM display)
H	The hour without a leading zero (0 to 23, even with AM/PM display)
HH	The hour with a leading zero (00 to 23, even with AM/PM display)
m	The minute without a leading zero (0 to 59)
mm	The minute with a leading zero (00 to 59)
s	The whole second without a leading zero (0 to 59)
ss	The whole second with a leading zero where applicable (00 to 59)
z	The fractional part of the second, to go after a decimal point, without trailing zeroes (0 to 999). Thus "`s.z`" reports the seconds to full available (millisecond) precision without trailing zeroes.
zzz	The fractional part of the second, to millisecond precision, including trailing zeroes where applicable (000 to 999).
AP or A	Use AM/PM display. A/AP will be replaced by either "AM" or "PM".
ap or a	Use am/pm display. a/ap will be replaced by either "am" or "pm".
t	The time zone (e.g. "CEST")

Additional Notes

- Any sequence of characters enclosed in single quotes will be included verbatim in the output string (stripped of the quotes), even if it contains formatting characters.
- Two consecutive single quotes ('') are replaced by a single quote in the output.
- All other non-formatting characters in the input string are included verbatim in the output.

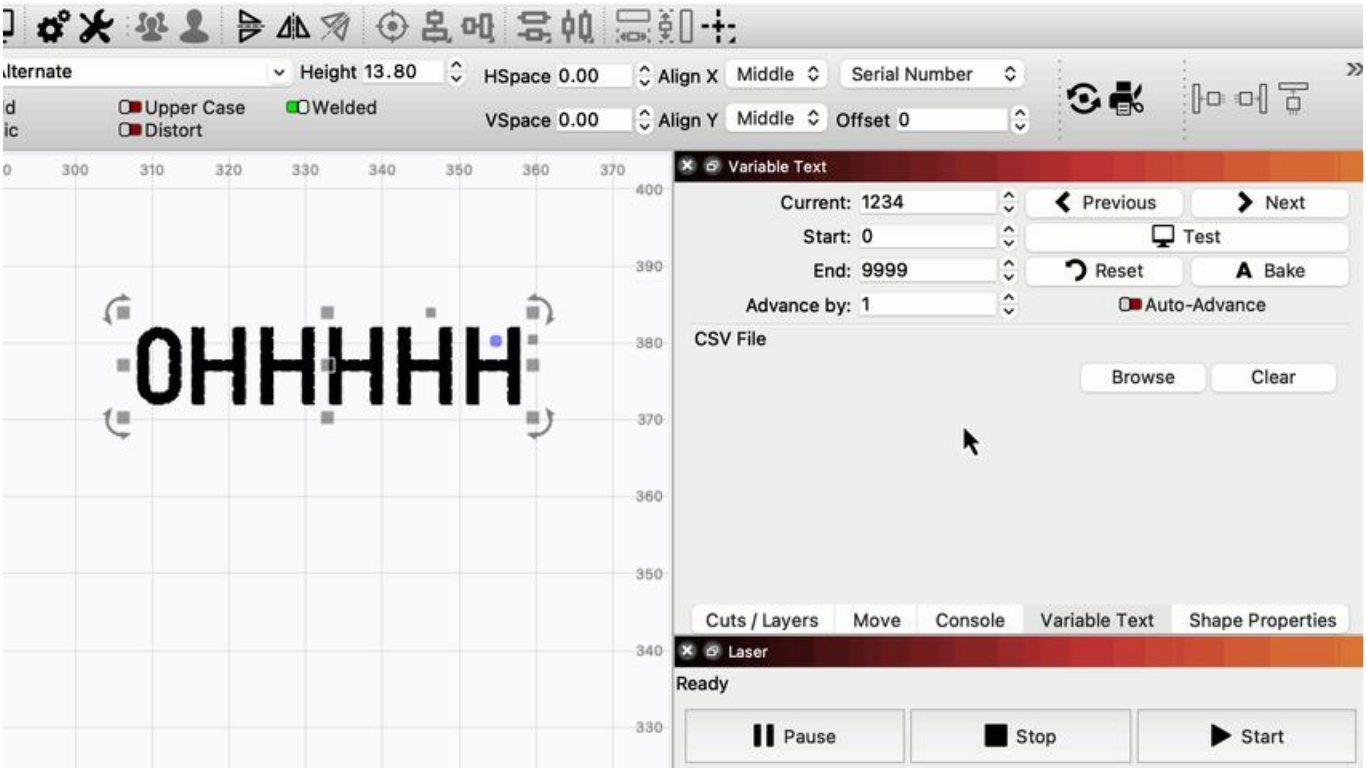


Tip

Formats without separators (e.g. "ddMM") are supported but must be used with care, as the resulting strings aren't always reliably readable (e.g. if "dM" produces "212" it could mean either the 2nd of December or the 21st of February).

Serial Number

When using **Serial Number** formatting, LightBurn will automatically substitute special combinations of characters with the Current serial number value.



Use the following expressions for serial numbers:

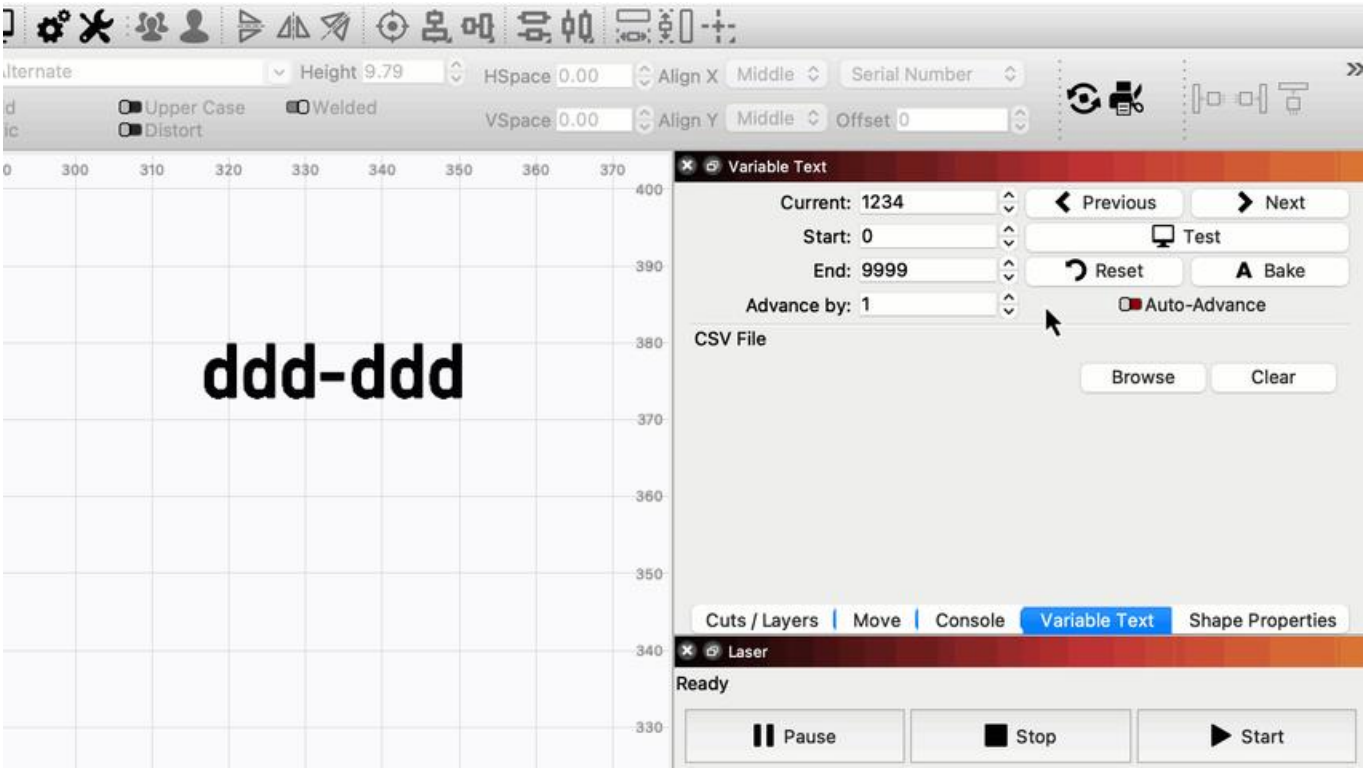
Output	Expression
The serial number as a decimal value	d
The serial number as a hexadecimal value, lower case	h
The serial number as a hexadecimal value, upper case	H
Tells LightBurn to pad the number with leading zeros	0

The number of characters controls how many digits will output. If the Current serial number is larger than the number of digits allowed, as many digits as will fit from the end of the number will be displayed. For example, if your Current serial number is 1234, the table below shows how that number would be formatted for each of the displayed formatting inputs:

Input	Output	Input	Output
d	4	0d	4
dd	34	0dd	34
ddd	234	0ddd	234
dddd	1234	0dddd	1234
ddddd	1234	0ddddd	01234
dddddd	1234	0dddddd	001234

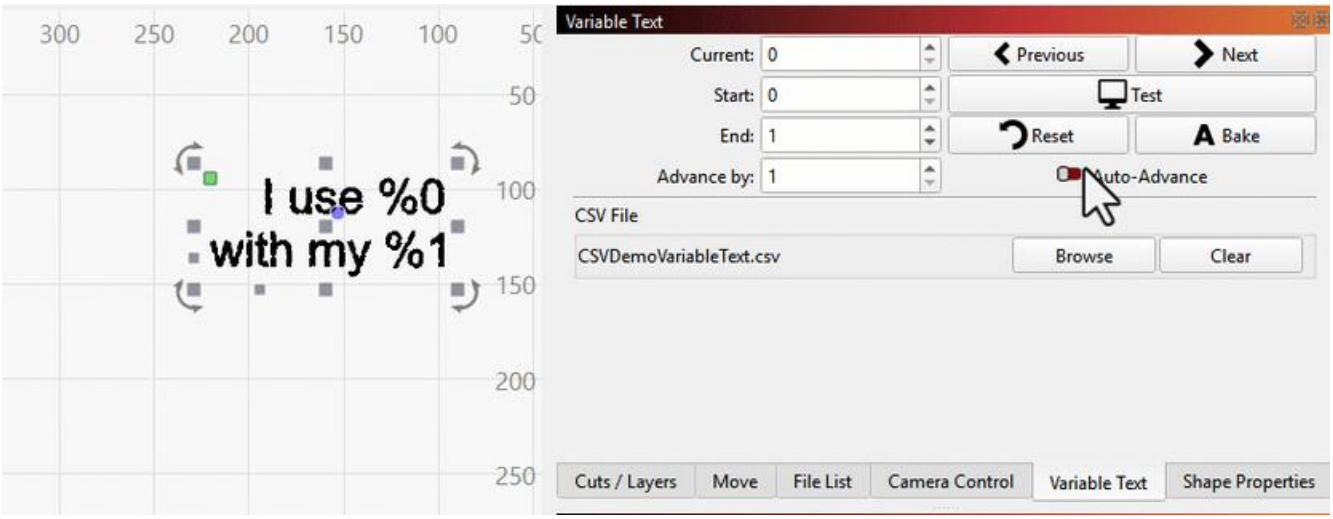
Additional Notes

- Any sequence of characters enclosed in single quotes will be included verbatim in the output string (stripped of the quotes), even if it contains formatting characters.
- All other non-formatting characters in the input string are included verbatim in the output.
- Use the Offset field in the Text Options Toolbar to enter multiple indentical formatting strings that output a different serial number.
- You cannot mix decimal and hexadecimal formatting in the same text entry.
- You cannot split a serial number with other characters. For example, because of the hyphen between the two groups of format characters, this string is not valid: ddd-ddd.
- Invalid strings will output as "bad serial format".



Merge/CSV

When using **Merge/CSV** formatting, LightBurn will automatically substitute special combinations of characters with an entry from a CSV file.



A CSV file is a Comma Separated Values file — a very simple text format that uses a line in a file as the row, and commas to separate columns. You can create a CSV file using a plain text editor, or by exporting from most common spreadsheet software.

In a Merge/CSV entry in LightBurn, the text you enter uses the percent sign followed by a number to look up a column in the current row of the CSV file.

- Columns and rows are numbered starting from 0.
- Refer to the first column in a CSV using the formatting "%0", the second using "%1", the third using "%2", and so on.
- A Current value of 0 refers to the first row, 1 to the second, 2 to the third, and so on.

For example, using a CSV file with the following content:

```
LightBurn,laser cutter
MillMage,CNC router
```

And entering this formatting:

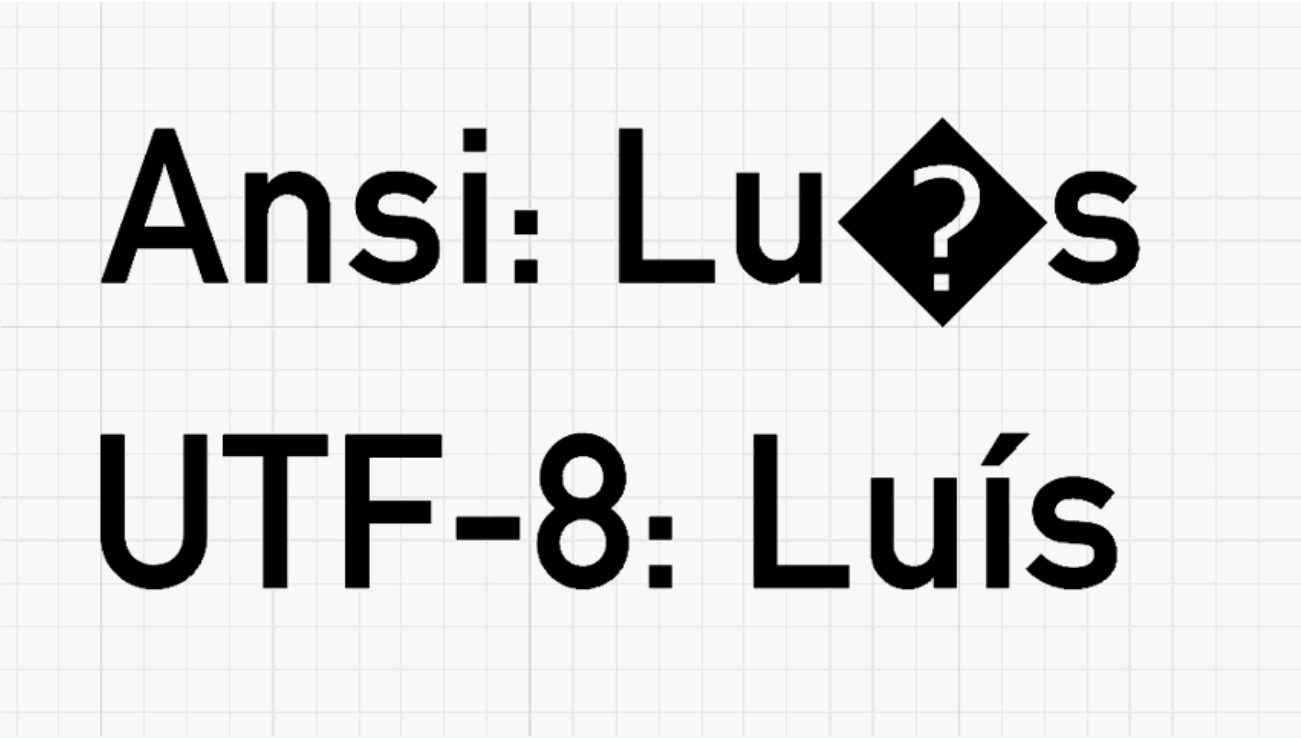
"I use %0 with my %1"

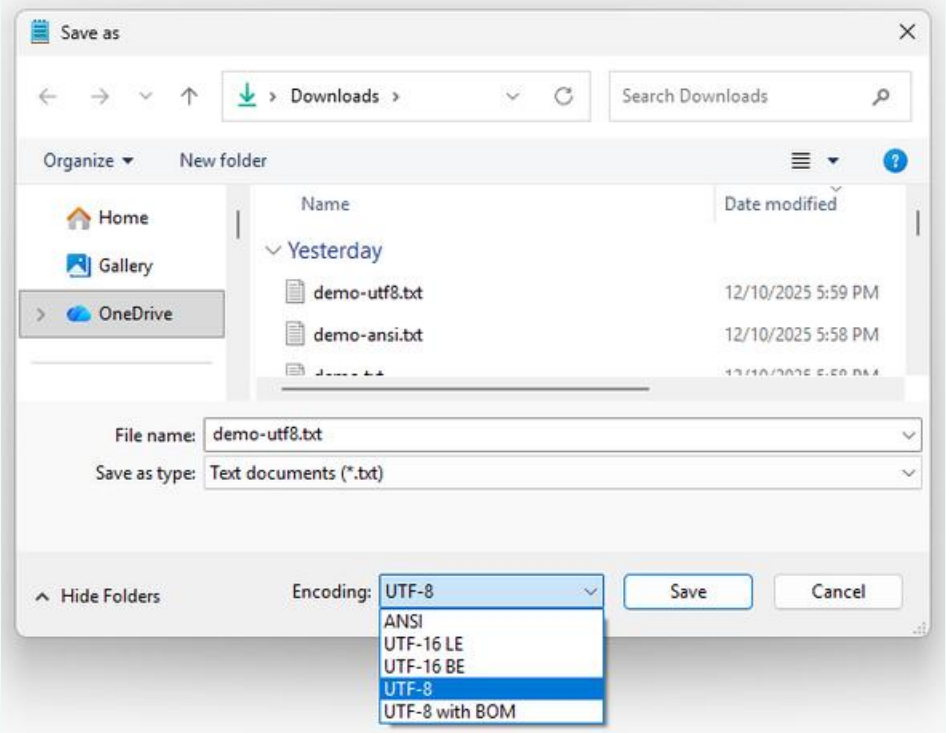
Would output:

"I use LightBurn with my laser cutter"

Special Characters Displaying Incorrectly

If special characters display wrong — typically as a question mark inside a diamond as shown in the image here — this is likely the result of the text encoding on the CSV/Text file being merged. Older programs may use the legacy ANSI format, which can cause problems with special symbols or accented characters.
To fix this problem, open the file in a modern text editor or spreadsheet program and save with UTF-8 encoding. Notepad on Windows and TextEdit on macOS both work well for this and come pre-installed with the operating system.



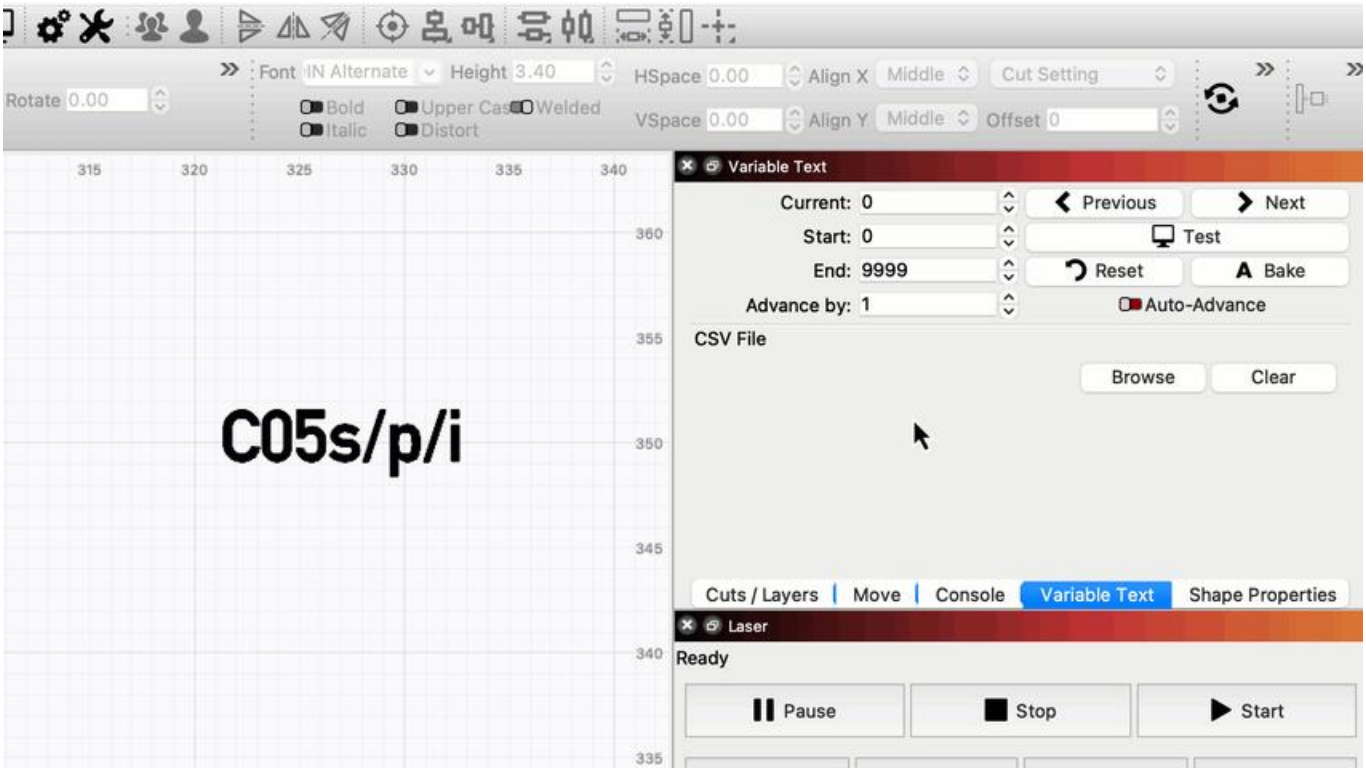


Additional Notes

- Use the Offset field in the Text Options Toolbar to enter multiple indentical formatting strings that output a different row from a CSV.

Cut Setting

When using **Cut Setting** formatting, LightBurn will automatically substitute certain characters with values from the Cut Settings applied to the text, or another specified layer.



Use the following expressions for Cut Settings:

Expression	Output
C	Followed by a number, pulls settings from the numbered cut layer (e.g. C03) for the remainder of this string
s	Speed, as a number in the current speed units
S	Speed, including the current units (like mm/sec)
p	Max Power, as a percentage
P	Max Power, including the percent sign
m	Min Power, as a percentage
M	Min Power , including the percent sign
n	Number of passes for the current layer
d / D	DPI, as a number, always dots per inch
i	Interval, in the current distance units
I	Interval, including the current distance units (like mm)

Expression	Output
L	Displays the name of the laser. Can optionally be followed by a character index to start displaying from, and optionally, a comma and a 2nd number for the number of characters to display. For example, if L displayed "Ruida 6442G", L6 would display "6442G", and L6,4 would display "6442" (without the quotes)
z	Z Offset for the current layer, in the current distance units
Z	Z Offset for the current layer including the units (eg, mm)
q	Q-Pulse width
Q	Q-Pulse width, followed by "ns" to indicate the units
k	Frequency in kHz
K	Frequency in kHz, followed by "kHz" to indicate the units

Additional Notes

- Any sequence of characters enclosed in single quotes will be included verbatim in the output string (stripped of the quotes), even if it contains formatting characters.
- All other non-formatting characters in the input string are included verbatim in the output.

Related Topics

- Variable Text
- Create Text
- Repeat Marking
- Grid Array
- Circular Array
- Copy Along Path

Create Bar Code

Create Bar Code generates a Bar Code which stores data that's visible when the Code is scanned. This tool offers a wide range of common Bar Code formats, including QR Codes.

Creating Bar Codes

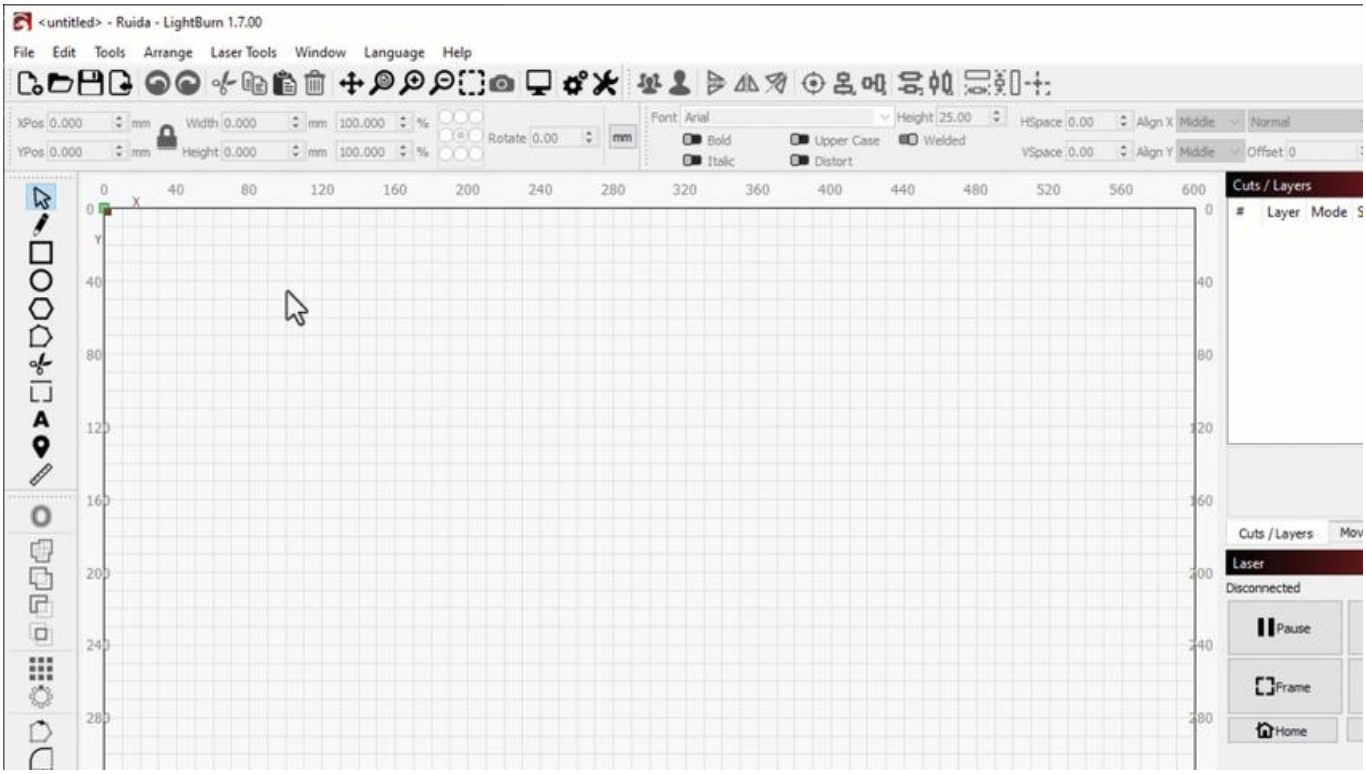
To activate **Create Bar Code**, go to Tools → **Create Bar Code**.

Click, hold, and drag in your Workspace to define the size of the Bar Code, then release to create it. You can resize the Bar Code later.

The **Bar Code Properties** window will automatically open, where you can select a Bar Code format, enter data, and adjust additional options.

The appearance of the Bar Code will live-update in your Workspace as you make changes.

Click **OK** to finish creating the Bar Code, or **Cancel** to delete it.

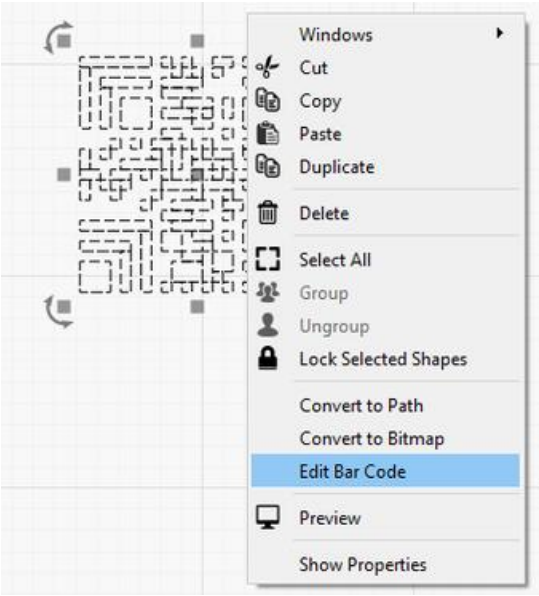


Editing Bar Codes

There are two ways to re-open the **Bar Code Properties** window after making a Bar Code:

- Double-click the Bar Code.
- First select it, then right-click in your Workspace, and select **Edit Bar Code** from the context menu.

> **Note**
>
> Edit Bar Code will only appear in the context menu if you have exactly one Bar Code — and nothing else — in your selection.



Bar Code Properties

Bar Code Type

Select a **Bar Code Type** from the dropdown menu at the top of the **Bar Code Properties** window.

The default Bar Code Type for all newly created Bar Codes is **QR Code**, or the last format you selected.

> **Full list of supported Bar Code Types**
>
> - 2 of 5
> - 2 of 5 IATA
> - 2 of 5 Industrial
> - AprilTag 16h5
> - AprilTag 25h9
> - AprilTag 36h10
> - AprilTag 36h11
> - ArUco MIP 36h11
> - ArUco OpenCV 7x7
> - Aztec
> - Chinese Sensible Code
> - CodaBar

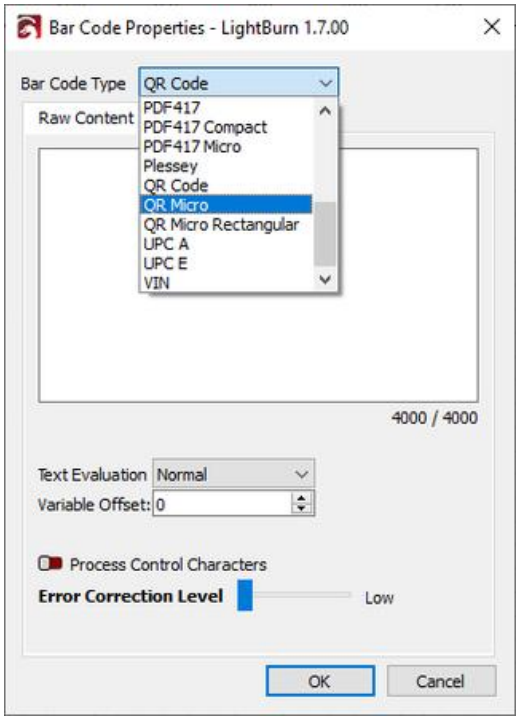
- Code 128
- Code 93
- Data Matrix
- EAN
- Grid Matrix
- ISBN
- PDF417
- PDF417 Compact
- PDF417 Micro
- Plessey
- QR Code
- QR Micro
- QR Micro Rectangular
- UPC A
- UPC E
- VIN

Data Entry Field

Enter data to store in the Bar Code in the **Raw Content** field. The type of data allowed depends on the type of Bar Code selected.

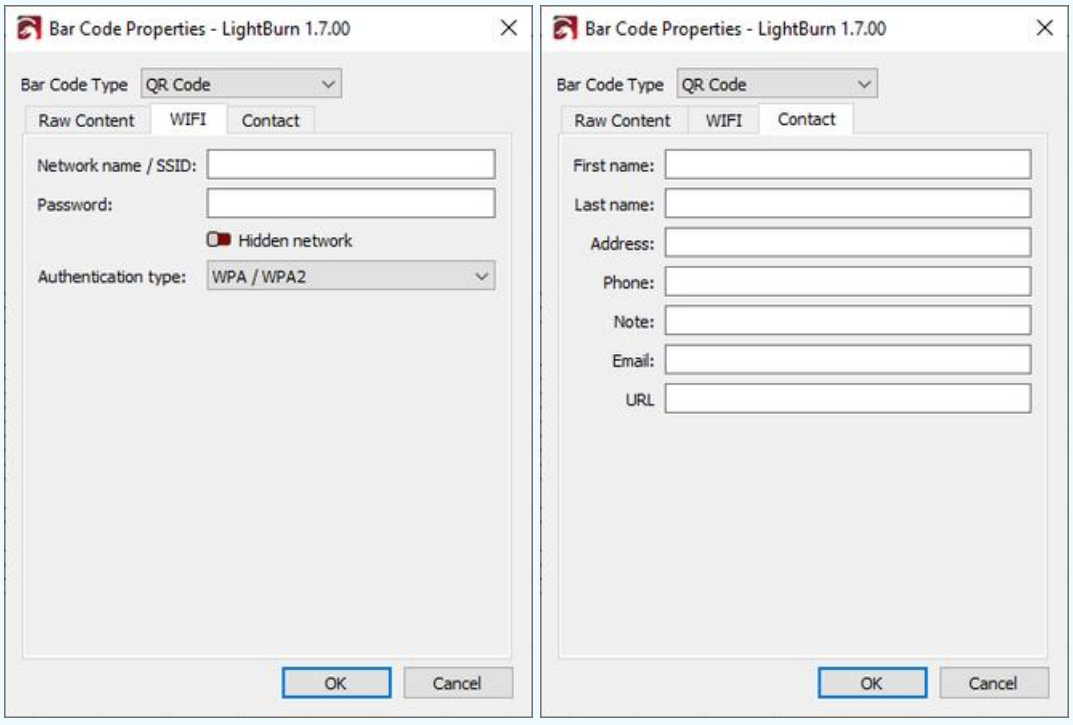
For **QR Codes**, you can choose to enter data as Raw Content, **WIFI** credentials, or **Contact** information.

- Any data entered in the WIFI or Contact tab will automatically populate the Raw Content tab.
- The last tab you enter data into overwrites data previously entered into another tab.



AprilTag and **ArUco** markers are used differently from typical Bar Codes. These markers are typically used as computer vision targets – we use AprilTags for Camera Calibration – and come from a list of pre-defined symbols. To get a specific marker, enter the identifier into the **Raw Content** field. This is a number from 0-1000, although not all types of AprilTag and ArUco markers use that full range.

The WIFI and Contact tabs for QR Codes



Character Limit

The character limit beneath the **Raw Content** field displays the amount of remaining characters supported by the **Bar Code Type** you have selected.



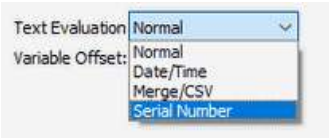
Text Evaluation

> **Info**
>
> See Variable Text and Variable Text Formatting for full details on working with Variable Text in LightBurn.

The **Text Evaluation** dropdown menu allows you to select a Variable Text format that will substitute different output in place of specially formatted codes.

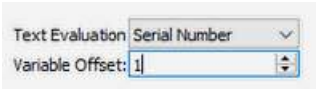
Normal is selected by default — the data stored in your Bar Code will match the **Raw Content** exactly, if you leave Normal selected.

You'll see the output that will result from your formatted codes in the **Evaluated Text** field.



Variable Offset

Variable Offset adds to the **Current** value in the Variable Text Window by the entered amount, allowing you to have multiple Bar Codes in your Workspace using the same Variable Text formatting, but displaying different **Serial Numbers**, or referring to different rows of a **CSV**.



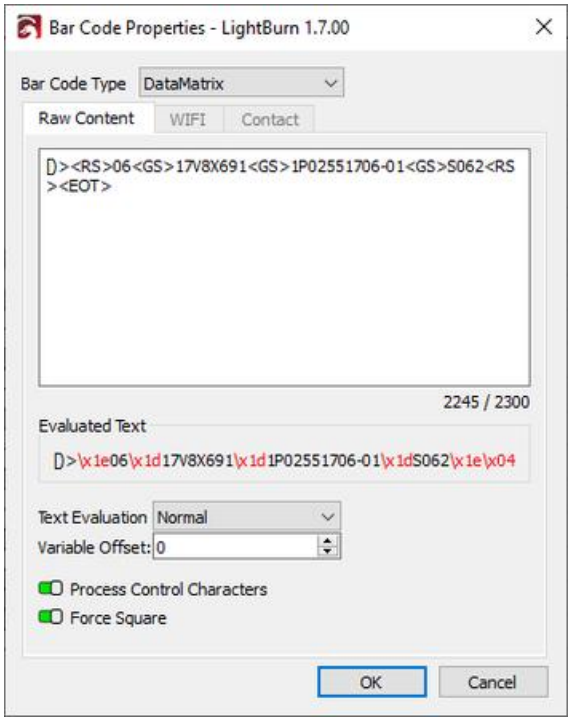
Process Control Characters

Enable this switch to convert specially formatted characters (`<RS>` , `<ACK>` , etc.) into non-printable characters used by GS1 compliant systems.

Control characters should be enclosed in <> brackets like this:

```
[)><RS>06<GS>17V8X691<GS>1P02551706-01<GS>S062<RS><EOT>
```

When **Process Control Characters** is enabled and valid characters are entered, you'll see converted characters in red in the **Evaluated Text** field.



Error Correction Level

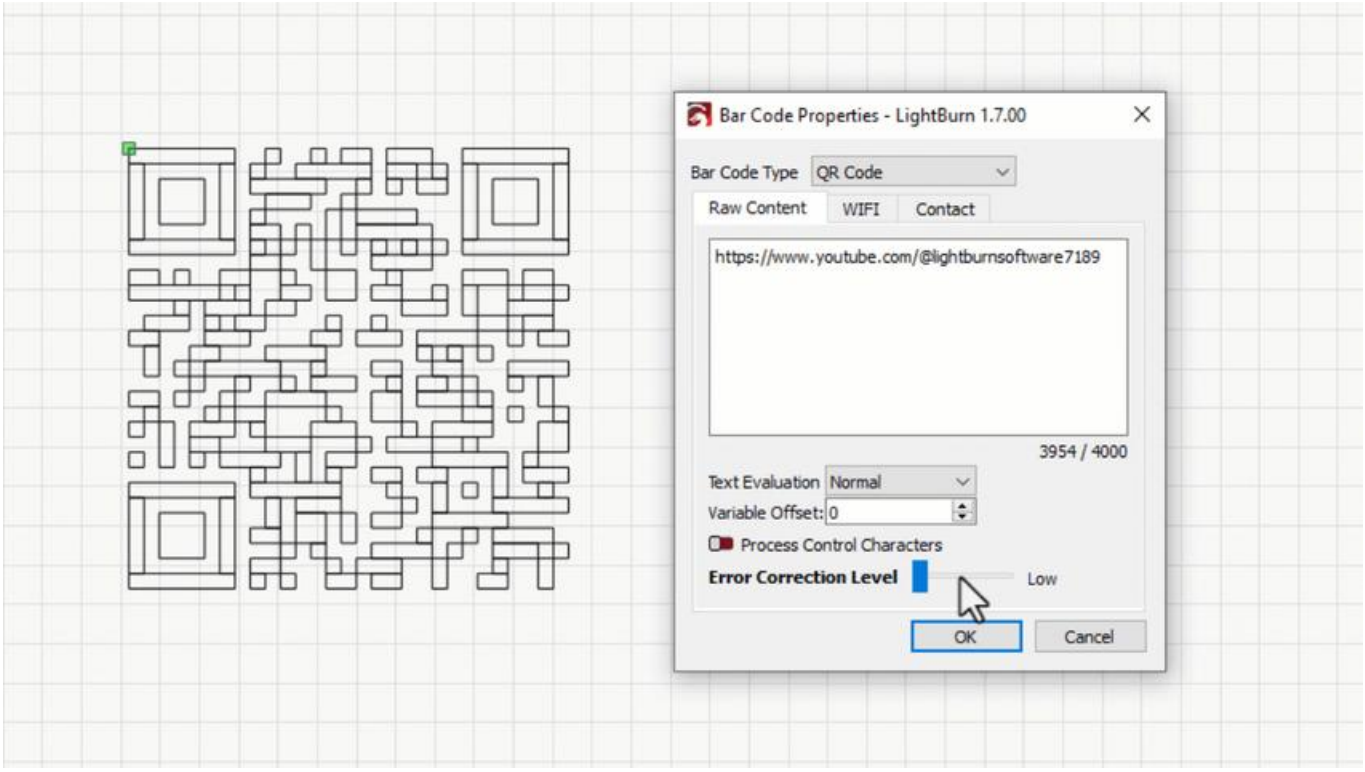
Error Correction helps to keep Bar Code data readable even if the Bar Code is partially damaged or obscured — use the Error Correction slider to adjust the amount of Error Correction in a Bar Code.

Different **Bar Code Types** provide different ranges of possible Error Correction — the highest Error Corrections produce Bar Codes that are more resilient to damage, but are visually denser and more complex.

Not all Bar Code Types support Error Correction.

Bar Code Types supporting Error Correction

- Aztec
- Chinese Sensible
- Grid Matrix
- PDF417
- PDF417 Compact
- PDF417 Micro
- QR Code
- QR Micro
- QR Micro Rectangular

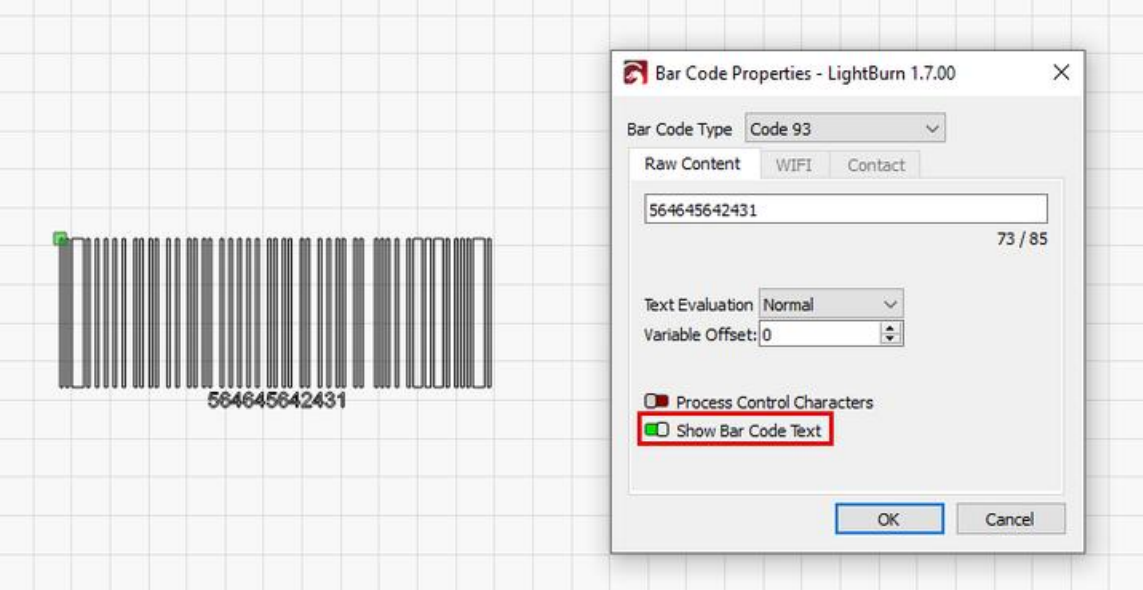


Show Bar Code Text

For supported **Bar Code Types**, tells LightBurn to output the text from the **Raw Content** field along with the Bar Code. This switch is enabled by default.

Bar Code Types supporting Barcode Text

- 2 of 5
- 2 of 5 IATA
- 2 of 5 Industrial
- CodaBar
- Code 128
- Code 93
- EAN
- ISBN
- Plessey
- UPC A
- VIN

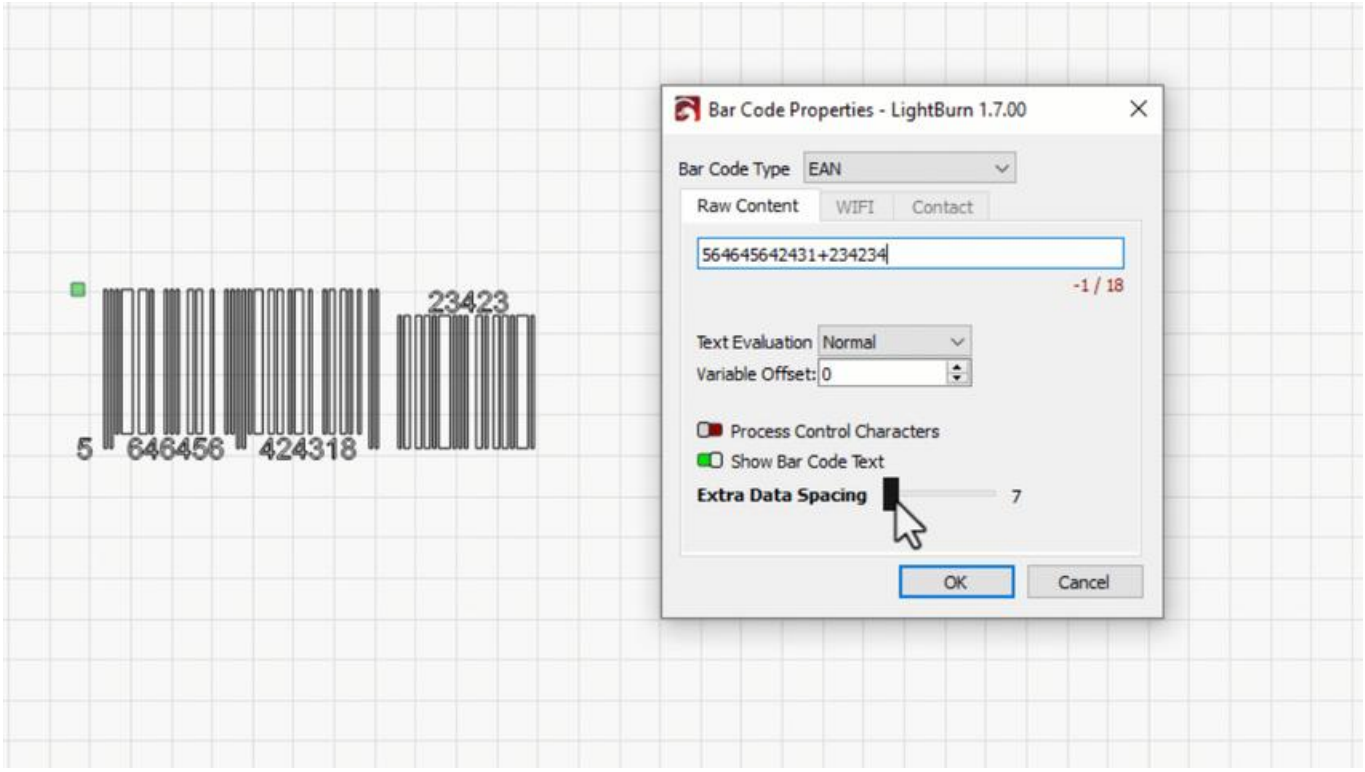


Extra Data Spacing

Some **Bar Code Types** support adding a secondary Bar Code based on extra data input after a "+" symbol. The **Extra Data Spacing** slider controls the distance between the primary and secondary Bar Codes.

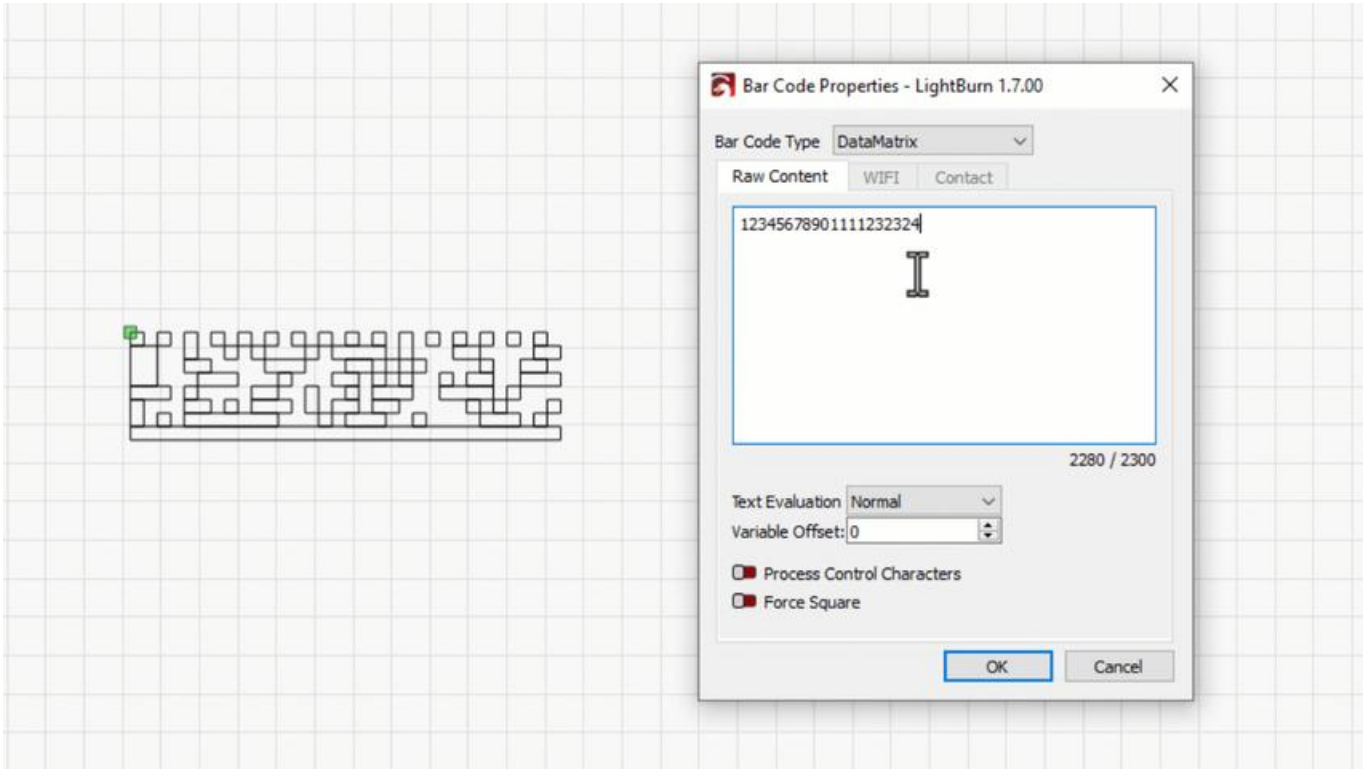
Bar Code Types supporting Extra Spacing

- EAN
- ISBN
- UPC A
- UPC E



Force Square

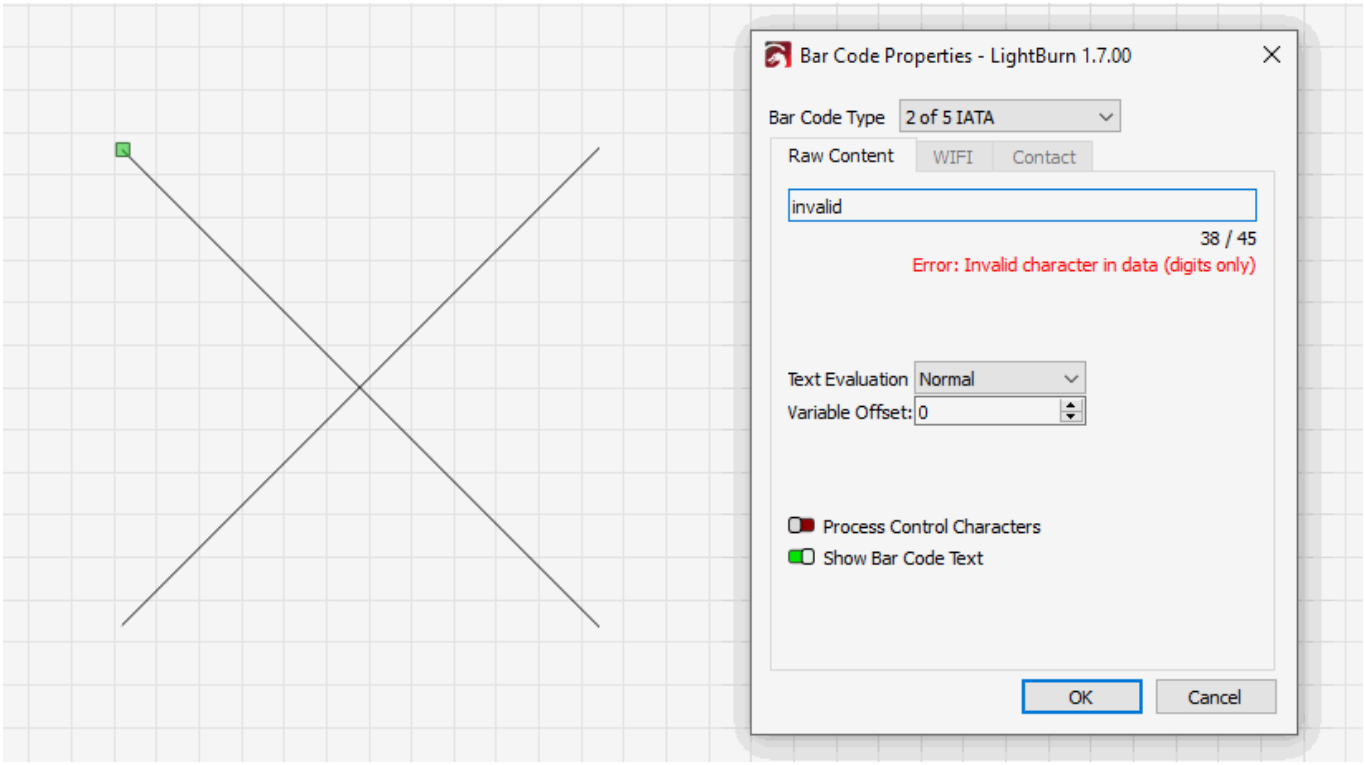
When **Data Matrix** is the selected format, some data inputs can result in a rectangular Bar Code. Enable **Force Square** to have the Data Matrix always remain square. This switch is enabled by default.



Error Messages

If the data you've entered is invalid for the type of Bar Code you've selected, you'll see an X in the Workspace and an error message in the **Bar Code Properties** window.

The error message will indicate which characters are invalid.



Related Topics

- Variable Text
- Repeat Marking
- Create and Edit Text

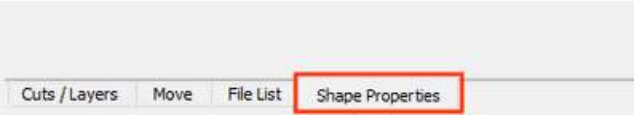
Shape Properties Window

The **Shape Properties Window** displays editable properties of selected objects.

Accessing the Shape Properties Window

The **Shape Properties Window** is not enabled by default. Select Window → **Shape Properties** to enable the window.

The Shape Properties Window will appear docked behind the Cuts / Layers Window on the upper right side of your screen. Click the Shape Properties tab to bring the Shape Properties Window to the front.



> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=uGWvIoNSPeA

Shape Properties Options

The options available in the **Shape Properties Window** depend on the type of object you have selected.

Three options are available for all types of objects, or when you have multiple types of objects selected at once.



- **Cut Order Priority**: sets the order this shape is cut by the laser. Lower values are cut first, starting with 0 as the highest priority. You must also select **Order by Priority** in the Optimization Settings for this setting to take effect.
- **Power Scale**: rescales the power used to cut this shape by this percentage, between Min Power and Max Power. For devices that do not support the Min Power setting, power is scaled as a percentage between 0 and Max Power.

Video: Fine Tune Laser Output With Power Scaling

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=NC-UZokVvt4

- **Locked**: prevents any edits to the selected shape(s). See Lock Shapes for more information.

When multiple shapes are selected, changing these values acts on all selected shapes at once.

See below for additional properties that are presented only when you've selected specific types of objects.

Image

These settings are presented for image graphics.

Gamma: adjusts mid-tones shades — Lower values lighten the mid-tones, and higher values darken them. **1.0** is normal.

Contrast: increases or decreases image contrast — higher values make light tones lighter and dark tones darker.

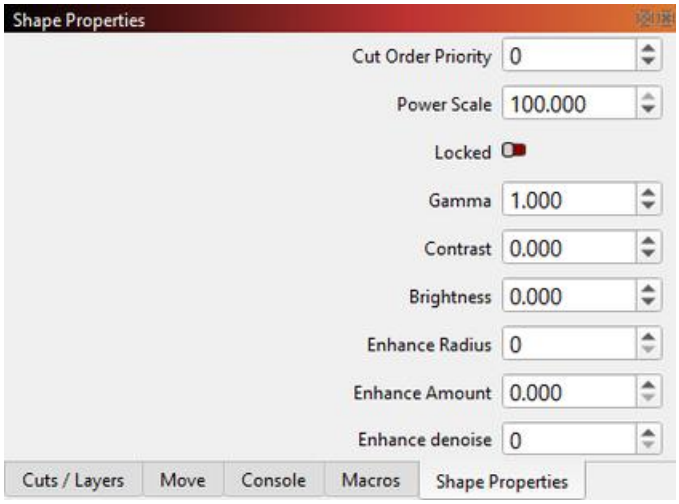
Brightness: increases or decreases image brightness, making both light tones and dark tones lighter or darker.

Enhance: increases the contrast of edges within the image - also known as Unsharp Masking and High-Pass Sharpening.

Enhance Radius: controls the spread of edge contrast — a larger radius spreads the Enhance effect further from the edges within an image.

Enhance Amount: controls the intensity of the edge contrast.

Enhance Denoise: reduces noise in smooth areas.



Ellipses

These settings are presented for shapes made using the Create Ellipse tool.

Width: the length of the ellipse in the horizontal (X Axis) dimension.

Height: the length of the ellipse in the vertical (Y Axis) dimension.



Polygons

These settings are presented for shapes made using the Create Polygon tool.

Width: the length of the polygon in the horizontal (X Axis) dimension.

Height: the length of the polygon in the vertical (Y Axis) dimension.

Sides: sets the number of sides of the shape — 3 = triangle, 6 = hexagon, 8 = octagon, etc.



Rectangles

These settings are presented for shapes made using the Create Rectangle tool.

Width: the length of the rectangle in the horizontal (X Axis) dimension.

Height: the length of the rectangle in the vertical (Y Axis) dimension.

Corner Radius: applies a radius to corners. Positive values create curved corners, negative values takes a bite out of the corner.



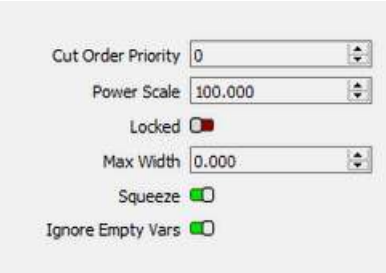
Text

These settings are presented for text made using the Create Text tool.

Max Width: text begins scaling if the length in the horizontal (X Axis) dimension exceeds this — use **0** for no limit.

Squeeze: when enabled, as the text exceeds the Max Width, the horizontal axis is scaled, but the vertical height remains the same, resulting in a squished appearance. When disabled, text will shrink proportionally, both horizontally and vertically.

Ignore Empty Vars: instructs LightBurn to remove newlines preceding empty Variable Text fields when aligning text.





Related Topics

- Adjust Image
- Primary Shapes
- Create and Edit Text

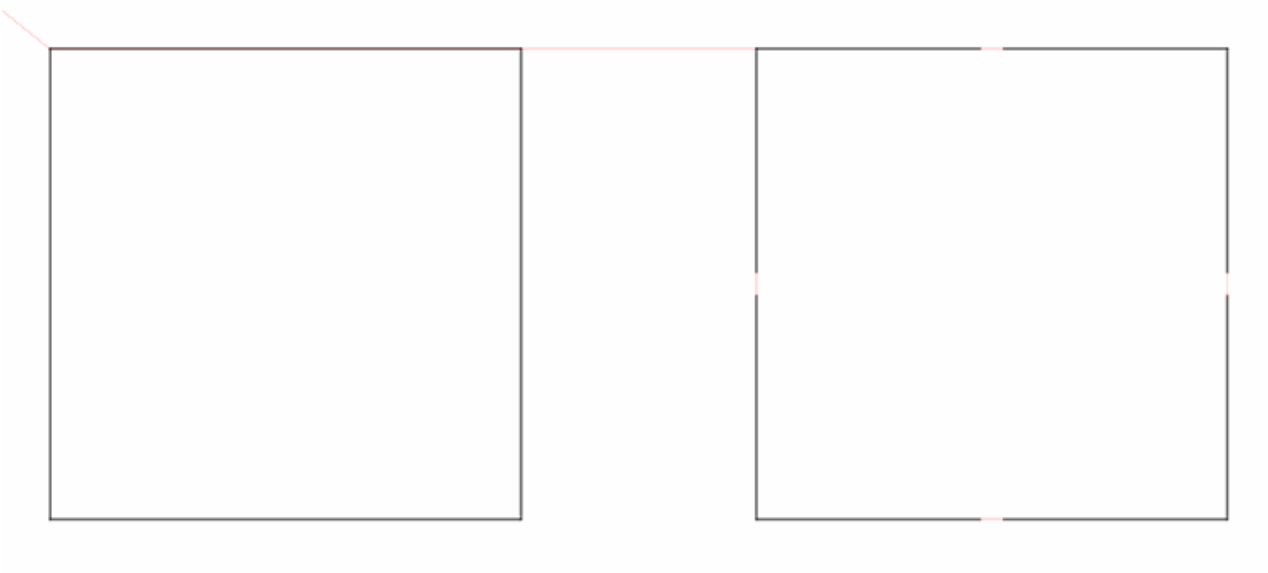
Add Tabs

Tabs are small sections of cuts that the laser is commanded to skip, used to prevent objects that you are cutting from immediately falling out of the base material. When sized appropriately, the tabs can be broken with minimal force and the objects removed from the material after the job is finished cutting.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=xn_CJT6IoOk

You can create tabs manually using the **Add Tabs** tool, or LightBurn can generate them automatically. Options controlling the size of **Manual** tabs, and spacing, number, and positioning of **Automatic** tabs, are available in the Cut Settings Editor.

In this example, the left square has no tabs applied, while the right has a single 1 mm wide tab on each side.

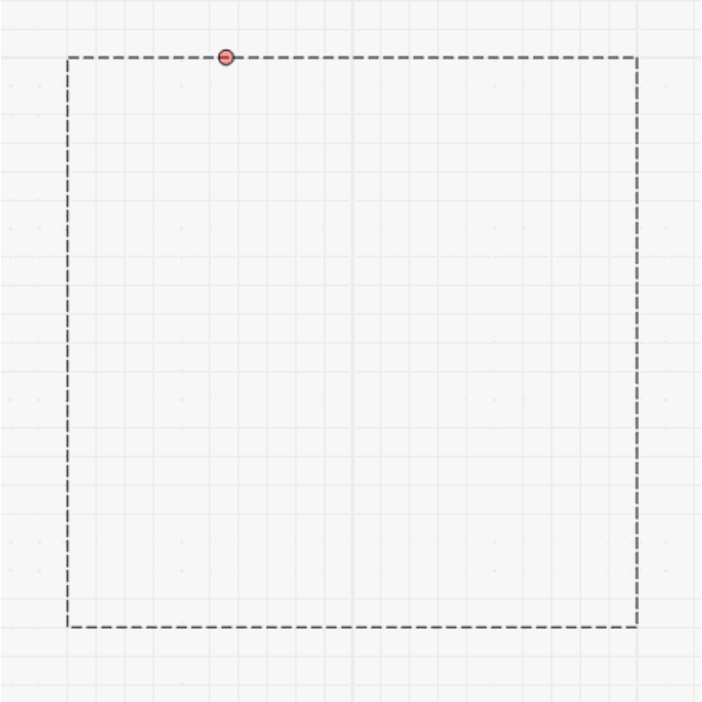


Adding Tabs Manually



To manually add tabs, select **Add Tabs** from the Creation Toolbar (or press `Ctrl` / `Cmd`+`Tab`), then hover over a shape in your Workspace that's assigned to a layer set to Line Mode.

If you are hovering over a valid shape, you'll see the **Insert Tab** cursor ( ). Click to insert a tab at that location. A red dot will appear on the shape, indicating that the tab has been placed.



Click and drag the red dot to move its location on the shape, or even to another valid shape in your Workspace.

The default width of manually-placed tabs is .5 mm. Tabs cannot be placed closer together than is allowed by their width. See the next section for information on adjusting tab width.

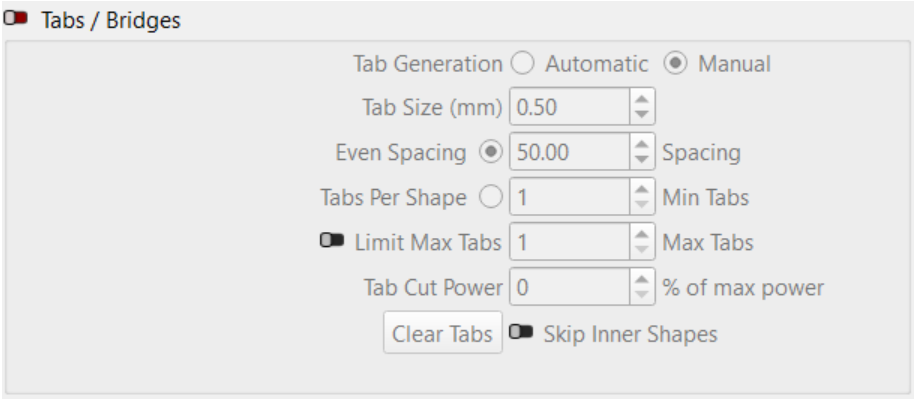
To remove an existing tab, double-click the red dot, or hold the `Shift` key while clicking it.

Automatic Tab Generation and Tab Settings

To adjust the settings for **Manual** tabs, or enable and adjust settings for **Automatic** tabs, double-click a layer set to Line Mode in the Cuts / Layers Window, to open the Cut Settings Editor.

Tab settings are available in the lower half of the Cut Settings Editor.

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Global Enable / Disable

The switch next to **Tabs / Bridges** controls whether tabs are enabled for the layer. If you have manually added tabs to a shape, this will automatically be switched on for the layer the shape is assigned to.

Switch Tabs / Bridges off to disable tabs. Tab settings and placement are not lost when this switch is turned off, and are restored exactly as they were if it is switched on again.

Tab Generation

- **Manual**: all tabs must be manually placed using the methods described above.
- **Automatic**: tabs will be placed on shapes assigned to the layer, based on the values of **Even Spacing**, **Tabs Per Shape**, and **Skip Inner Shapes**.

Tab Size

This setting affects both **Manual** and **Automatic** tabs.

Tab Size defines the total length of all tabs, in the currently configured system unit type (mm or inch). The laser will skip cutting for this distance, if possible.

Even Spacing

With **Even Spacing** selected, each contiguous path section will have at least one tab applied at the very start of the path and then again at the distance interval set in the **Spacing** field.

Spacing is set in the currently configured system unit type (mm or inch).

Limit Max Tabs

Enable **Limit Max Tabs** to set an upper bound on how many tabs can be automatically generated for each path, when using **Even Spacing**.

Tabs Per Shape

Select the **Tabs Per Shape** option to specify the number of tabs to create on each path assigned to the layer. Tabs will be automatically spaced evenly along the path.

> **Detailed placement info**
>
> To calculate automatic placement of tabs when using **Tabs Per Shape**, LightBurn does the following:
>
> 1. Calculates the total perimeter length.
> 2. Divides the perimeter into segments – one more segment than the number of tabs selected.
> 3. Places tabs where the segments meet, shifting the start point by half a segment so things are evenly placed on closed shapes.

Tab Cut Power

This setting affects both **Manual** and **Automatic** tabs.

If **Tab Cut Power** is set to any value other than 0%, the laser will make pulsed cuts across the width of the tabs instead of skipping them completely.

Increasing the percentage increases the amount the laser pulses on during tabs, resulting in tabs that are more weakly connected to the base material.

In the example below, the top tab is set to 25%, while the bottom tab is set to 75%. The black lines represent areas along the path where the laser is on, and the red lines represent areas where the laser is off.



Skip Inner Shapes

When enabled, any shapes that are completely contained by another shape in the same cut layer will not have **Automatic** tabs applied. This does not affect **Manual** tab placement.

> **Closed Shapes Only**
>
> This feature only works with shapes that are closed. A shape is closed when it is a complete, continuous loop whose start and end points are the same. When a shape's start and end points are different, it is open.
>
> See Open vs. Closed Shapes for more information.

Clear Tabs

Deletes all tabs (**Manual** or **Automatic**) from every shape on this cut layer.

Troubleshooting

- Tabs functions are not available if you are using Beginner Mode. You can disable Beginner Mode in Settings/Preferences.

Related Topics

- Cut Settings Editor
- Perforation Mode
- Kerf Offset

- Ramp Length

Measure

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=azPoxglNHk4

Use the **Measure** tool to hover over any shape in your Workspace and display an array of useful information about it, including perimeter, overall dimensions, whether it is Open or Closed, and the lengths of individual line or curve segments within it. You can also click and drag to create and measure a temporary line between two points.

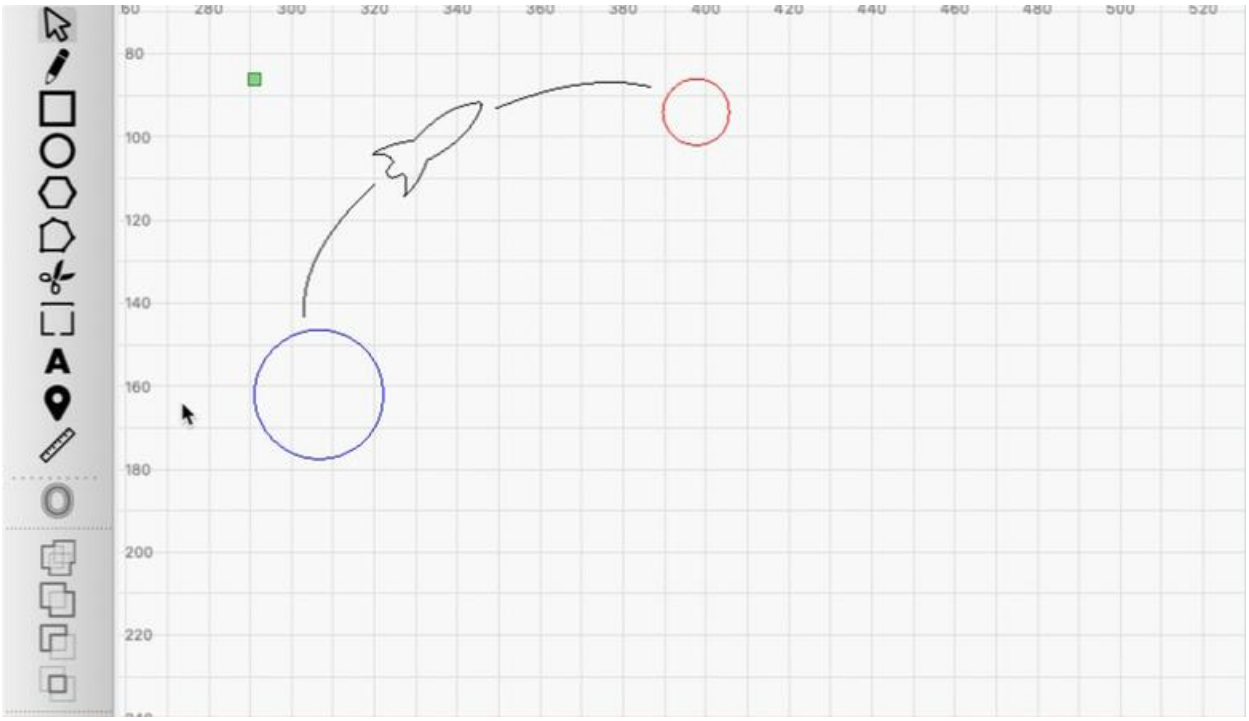
To enable the Measure tool, click on the ruler icon in the Creation Toolbar, or press `Alt`/`Option`+`M`.



Using the Measure Tool

Hover your cursor over any shape in your Workspace to see information on a given shape. Move your cursor over any individual line or curve segment in a shape to see information on that specific segment.

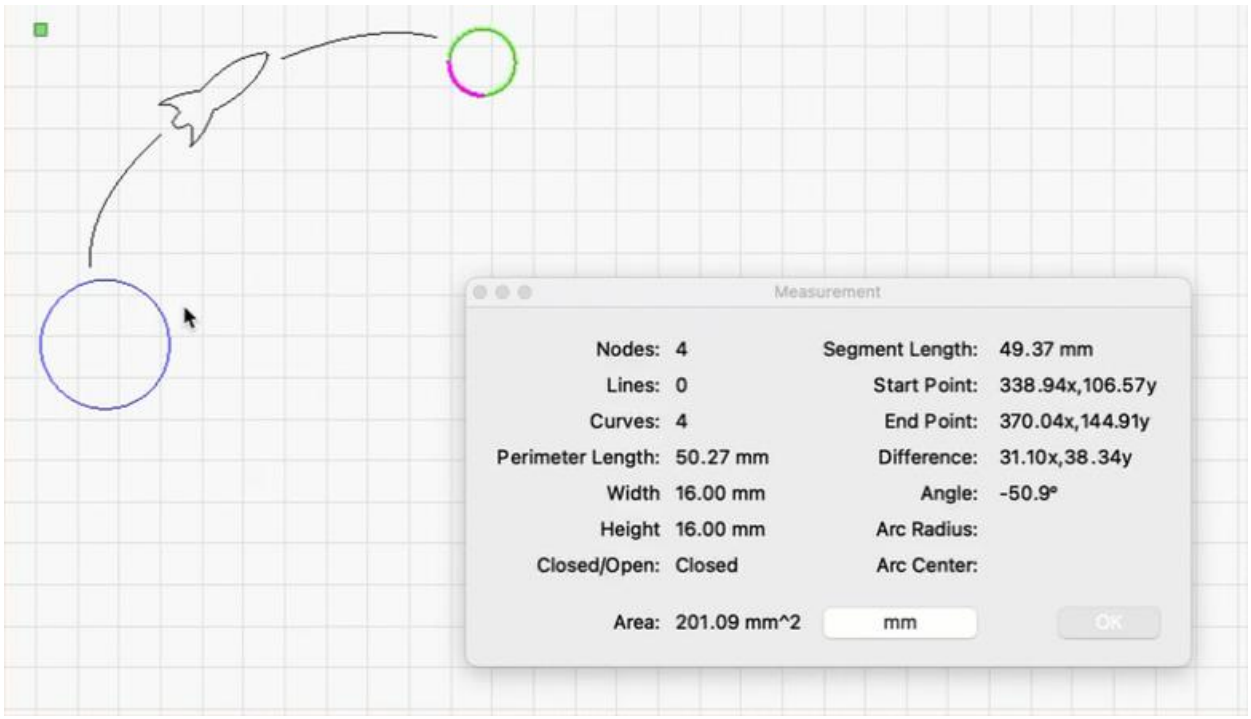
As you hover the cursor over shapes, individual shapes are highlighted in green, and the segment the cursor is currently hovering over is highlighted in purple, as shown below:



Drag Measure

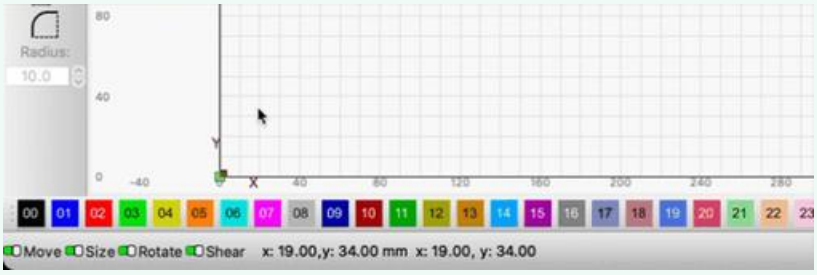
Click, hold, and drag between any two points in your Workspace to display information on a temporary line created between those points. You can snap to a shape or node on a shape, or click anywhere in your Workspace, even if no shape is present.

Hold `Shift` while dragging to lock the temporary line to horizontal, vertical, and 45 degree angles.

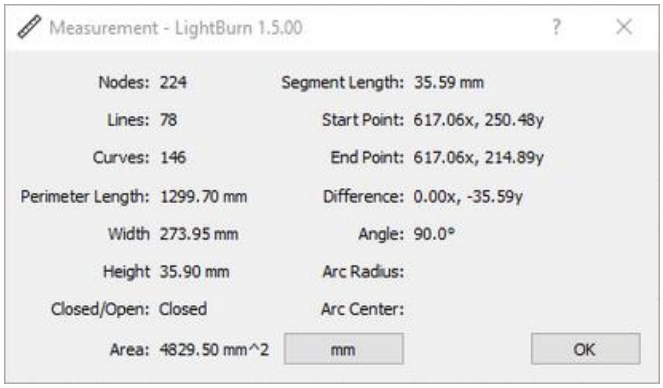


View shape dimensions as you create them

When using any of LightBurn's shape creation tools, you can also view the measurements of the shape you're creating in the Status Bar.



Measurement Window



The Measurement dialog window describes the highlighted shape and individual segment. You can click the button displaying **mm** or **in** to switch between either unit of measurement.

The dialog displays the following values:

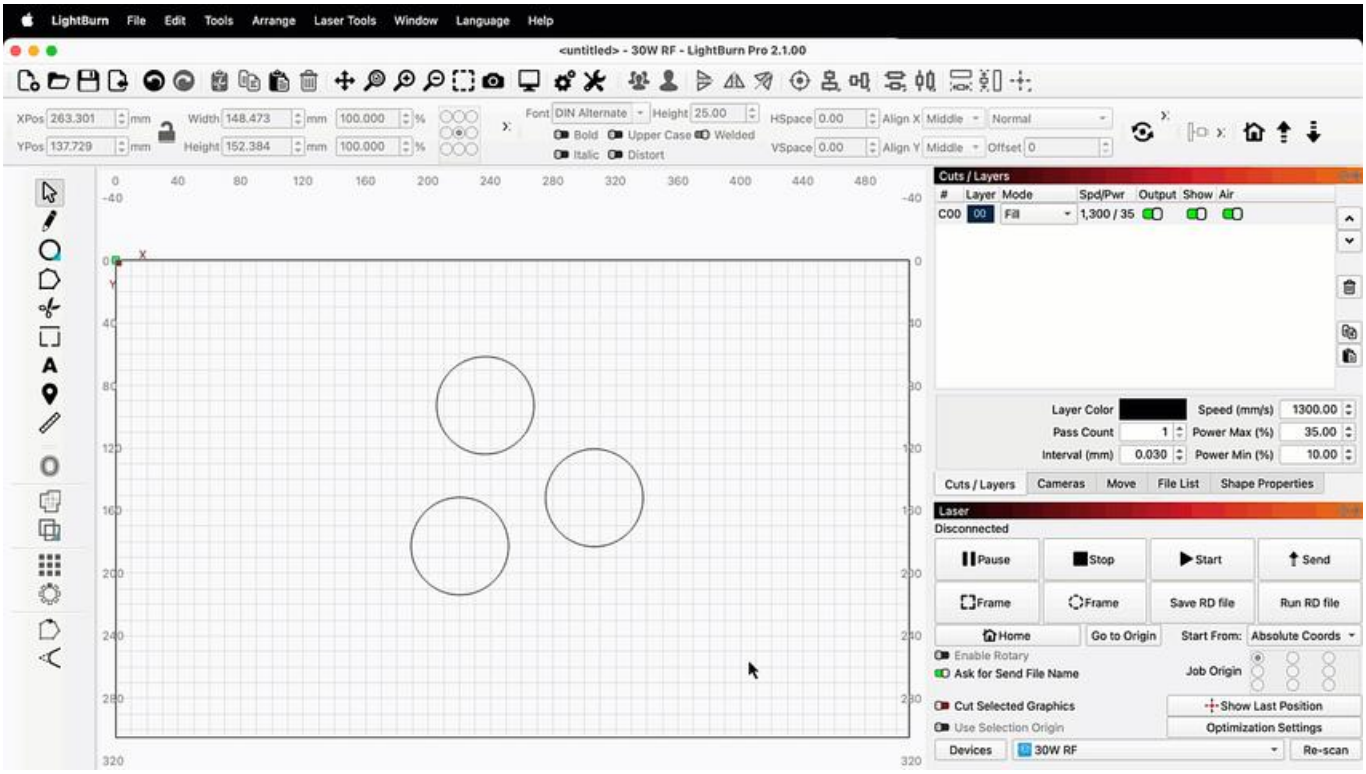
Value	Explanation
Nodes	Number of nodes (points at starts and ends line segments and curves)
Lines	Number of straight lines
Curves	Number of arcs or Bézier curves
Perimeter Length	Total length of all line/curve segments
Width	The total span of the object in the X dimension
Height	The total span of the object in the Y dimension
Closed/Open	Whether or not the shape is closed
Area	The total area of the shape. Empty for open shapes
Segment Length	Length of the highlighted line/curve segment
Start Point	X, Y coordinate of the start node of the highlighted segment
End Point	X, Y coordinate of the end node of the highlighted segment
Difference	Difference in X and Y between the start and end nodes
Angle	Angle of the line between the start and end points
Arc Radius	Radius of the arc, if the curve is a true circular arc. Empty otherwise
Arc Center	Center point of the arc, if the curve is a true circular arc. Empty otherwise

Related Topics

- Selection Tools
- Numeric Edits Toolbar
- Shape Properties
- Align Tools

Tangent Circle Generator

With the **Tangent Circle Generator**, you can select three circles and automatically create a circle tangent to those three circles - meaning that it touches each of the three circles at exactly one point.



To use the **Tangent Circle Generator**:

1. Select 3 circles in your Workspace. Each circle must be an actual circle, with equal width and height, in order for the Tangent Circle Generator to work.
2. Go to **Tools → Generate Tangent Circle**.
3. LightBurn will show you all possible tangent circles in light red outlines.
4. Hover over an outline to highlight it in bold and see it more clearly.
5. Click to select the circle you want to use. The circle will be created on your currently active layer.

> **Create multiple tangent circles**
>
> When clicking the light red preview circle in step 5, you can hold down `Shift` to select and create multiple circles.
>
> The circles created by the Tangent Circle Generator are recognized as shapes and can be used to create more tangent circles.

Troubleshooting

- Make sure your circles are actually circles, with equal height and width. You can draw circles with the ellipse tool by holding down `Shift` while drawing, or resize ellipses using the Numeric Edits Toolbar or Shape Properties Window.
- Make sure you've selected exactly three circles.

Related Topics

- Draw Shapes
- Circular Array

Art Library

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=30hYVXQ-4CA

An **Art Library** provides a way to store artwork so you can quickly access and reuse it in future projects.

You can load multiple Art Libraries at the same time, and separate Libraries are a great way to split artwork up by theme (holidays, birthdays, animals), or purpose (tests, tools, templates, boxes).

An entry in an Art Library can contain any combination of objects that would normally make up a project, including text, Primary Shapes, paths, and images.

> **Note**
>
> The Library only stores artwork, much like the contents of AI, SVG, DXF, or image files — it doesn't store any Cut Settings applied to the artwork. If you wish to store cut settings with your design, you must Save your work instead.

Accessing the Art Library Window

The **Art Library Window** is not enabled by default. To open it, go to Window → **Art Library**.

By default, the Art Library will appear docked on the left side of your Workspace.

> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

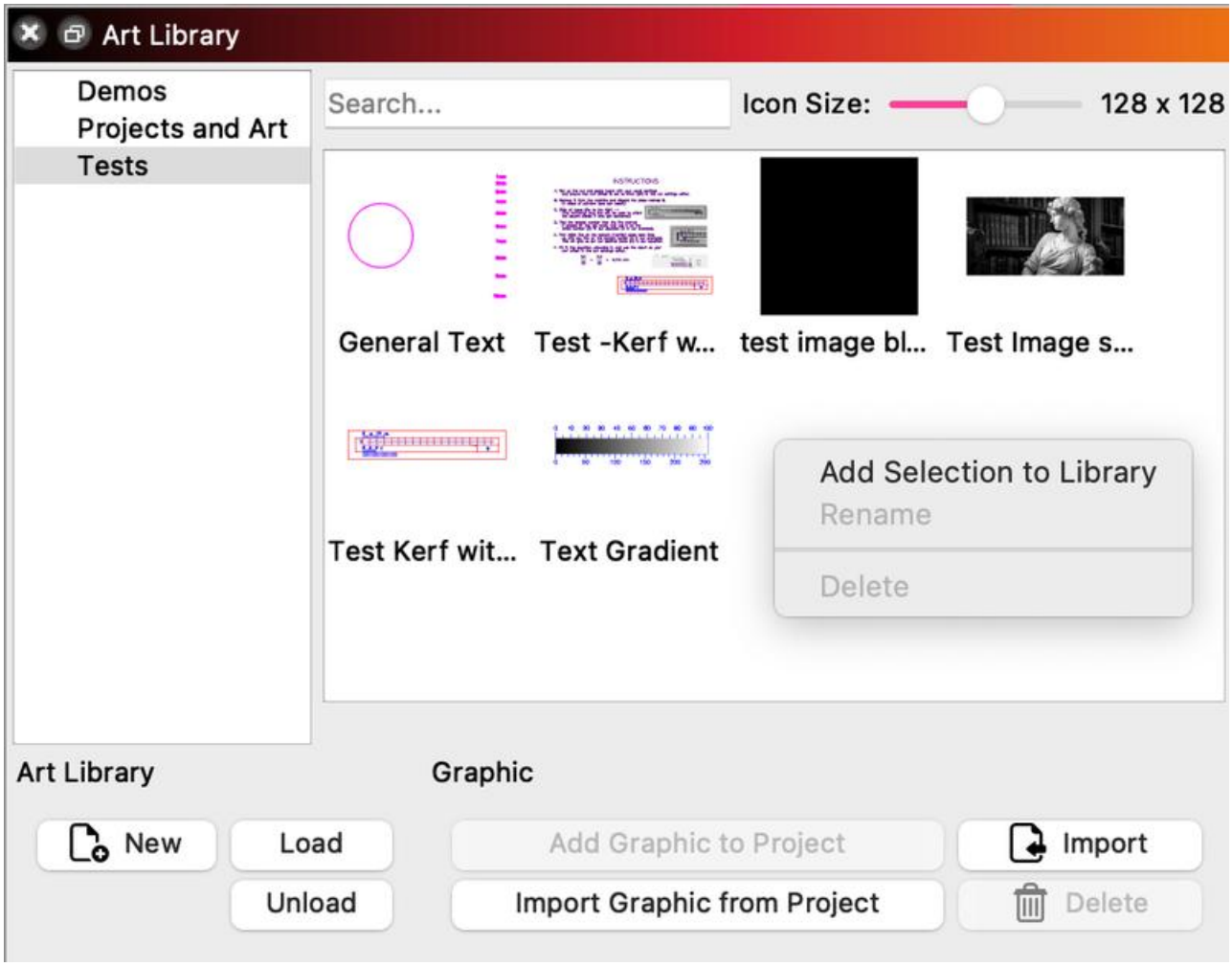
Using the Art Library

You'll see a list of Art Libraries on the left, and thumbnails of the artwork within the currently selected Library on the right.

> **Note**
>
> If there are no Libraries listed, a **New** Library must be created, or an existing one **Loaded**, in order to access artwork.

Creating a Library

To create an empty Library, click the **New** button, then choose a location and name for the `.lbart` file — this is where added artwork will later be stored, in compact form.



Loading and Unloading Libraries

To add a previously saved Library, click the **Load** button in the bottom of the **Art Library Window**. Saving, transferring, and Loading Libraries makes it possible to share artwork between computers and users.

To remove a Library from the list, click **Unload**. Unloading a Library removes it from view, but does not delete the file or its contents from the hard drive where it is saved.

Storing Art in a Library

Use the **Import** button to add artwork from saved files to your Library. You can import multiple files at once using this method, and each will become an individual entry in the current Library.

To add selected objects from your current project, do one of the following:

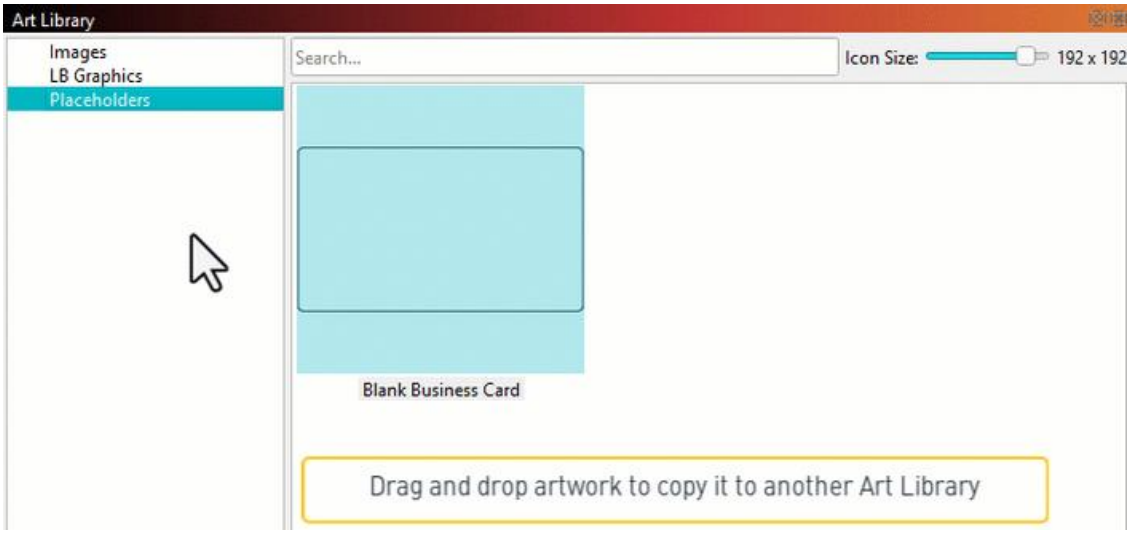
- Click **Import Graphic from Project** in the **Art Library** window.
- Right-click in the **Art Library Window** and choose **Add Selection to Library**.
- Right-click in the Workspace and select either **Add to Current Art Library** or **Add to New Art Library** from the context menu that pops up. Add to Current Art Library is only available if you've already loaded an Art Library.

Editing Art and Libraries

Right-clicking artwork in an **Art Library** brings up the options to **Delete** or **Rename** it.

To change the name of a Library, change the `.lbart` file name in your file browser, and then **Unload** and **Reload** the Library in LightBurn.

To copy artwork from one Library to another, drag and drop in the Art Library window. To move artwork, hold `Shift` while dragging the artwork.



Adding Art to a Project

Dragging a thumbnail from the **Art Library** into a project adds it to the project wherever the mouse is released.

Clicking the **Add Graphic to Project** button adds the selected artwork into the center of the current field-of-view in the Workspace.

Related Topics

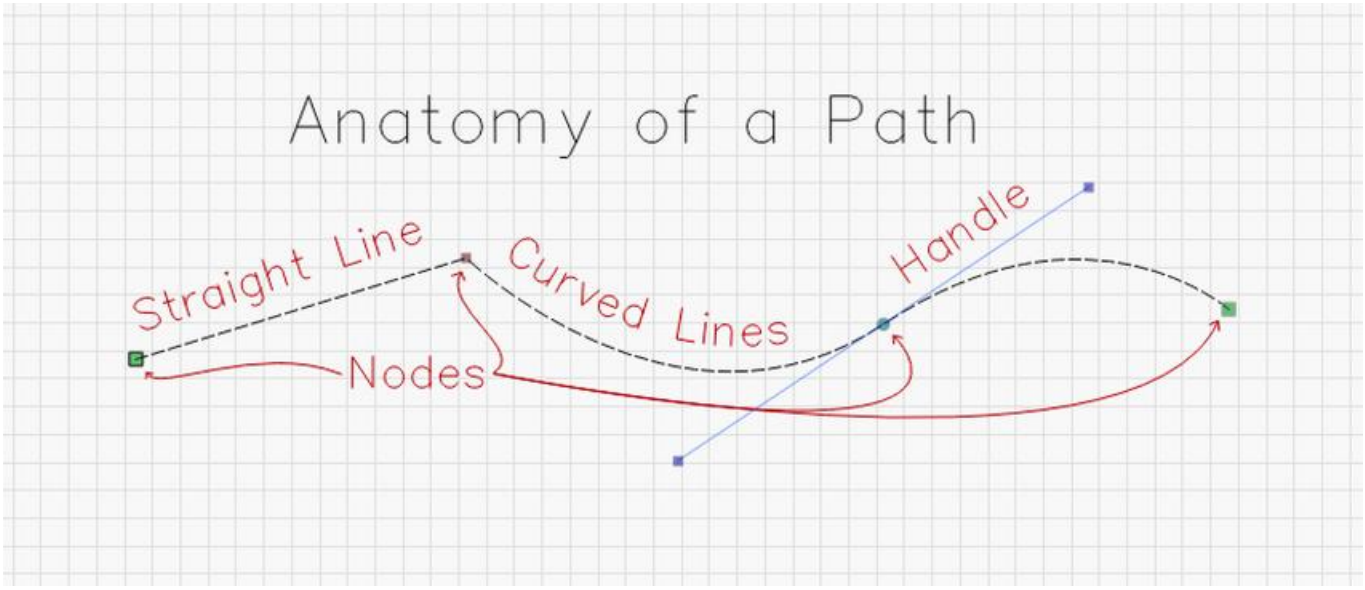
- Import
- Save
- Material Library

Editing

Edit Nodes

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=eFmLKBr5Xxg

Edit Nodes allows you to make precise, granular edits to vector paths, by controlling the placement and behavior of **nodes** (points in space, sometimes called anchor points) which are linked by straight **lines** or **curves**. The curved lines are described by the length and direction of **handles** (tangents) which appear on one or both nodes at either end of a curve.



Using Edit Nodes

To enable **Edit Nodes**, click on the Edit Nodes icon in the Creation Toolbar or press `Ctrl`/`Cmd` + `` ` `` on your keyboard.



If you already have a valid object (or objects) in your selection, its nodes will appear. If you do not have anything currently selected, click an object to view and edit its nodes.

> **What is a valid object?**
>
> The **Edit Nodes** tool works on vector paths. Paths must be ungrouped. Text Objects, Primary Shapes, and Barcodes must be converted to a path before using Edit Nodes.

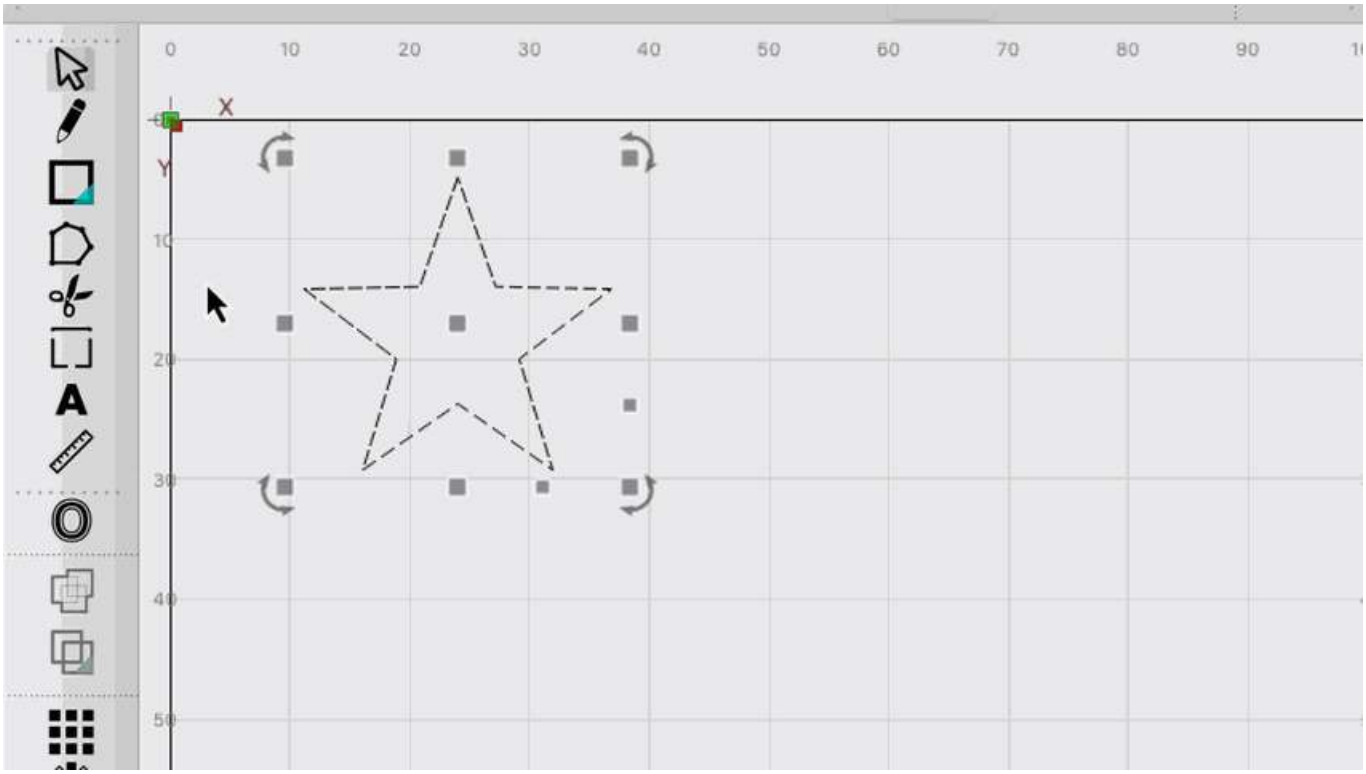
Edit Nodes Toolbar

When you enable the **Edit Nodes** tool, an additional toolbar will appear to the right of the Creation Toolbar.

Hover over any button in the **Edit Nodes Toolbar** to see a tooltip explaining the action it will perform.

Click on the desired node tool, then click on a valid node or line in your Workspace to perform that action.

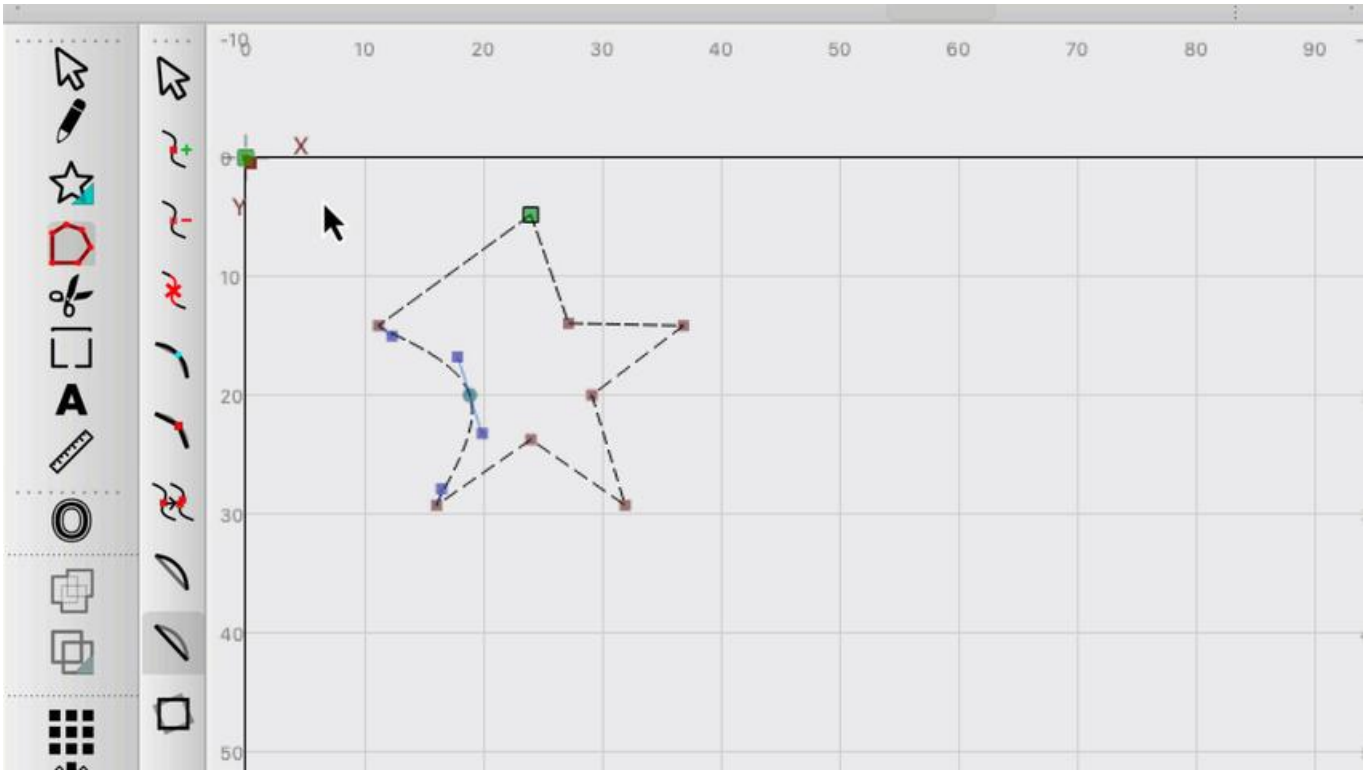
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=LpLdBrZAhYc



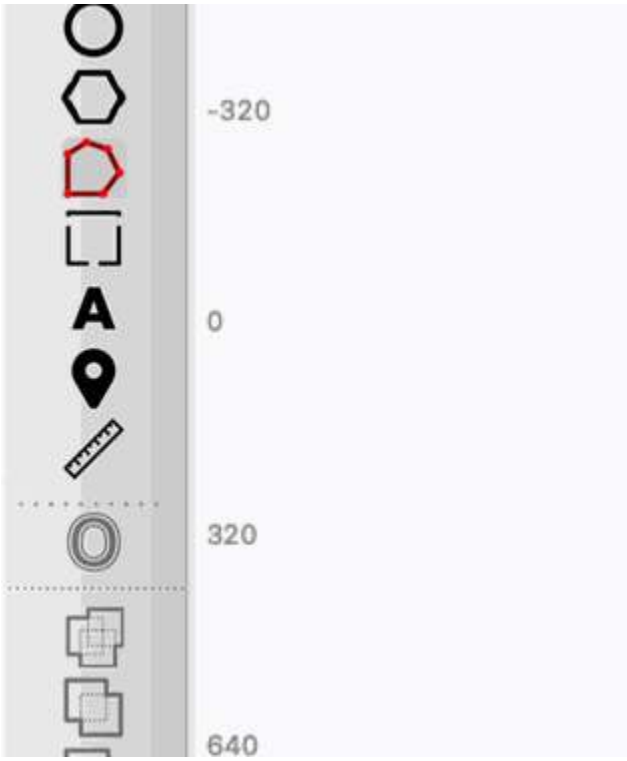
Mouse Actions

With the **Select** action enabled in the **Edit Nodes Toolbar**, you can select and adjust nodes and lines directly with your mouse.

- Click and and drag on nodes, handles, or curves to move them.
- Click any node or handle to select it, and then use the keyboard arrow keys and modifier keys to move it by the increments set in the Shape Move Increments fields in the Settings window.
- With a node or handle already selected, hold `Shift` while clicking on another node or handle to add it your selection, then drag or use the arrow keys to move each selected node or handle in unison.
- Click any straight line and drag to automatically turn it into a curve.
- Drag disconnected nodes on top of one another to automatically join them, turning the lines on either side into a single shared path.



Keyboard Actions



Each of the actions in the **Edit Nodes Toolbar** has a corresponding hotkey, listed below, that you can use to edit and manipulate lines, curves, and nodes.

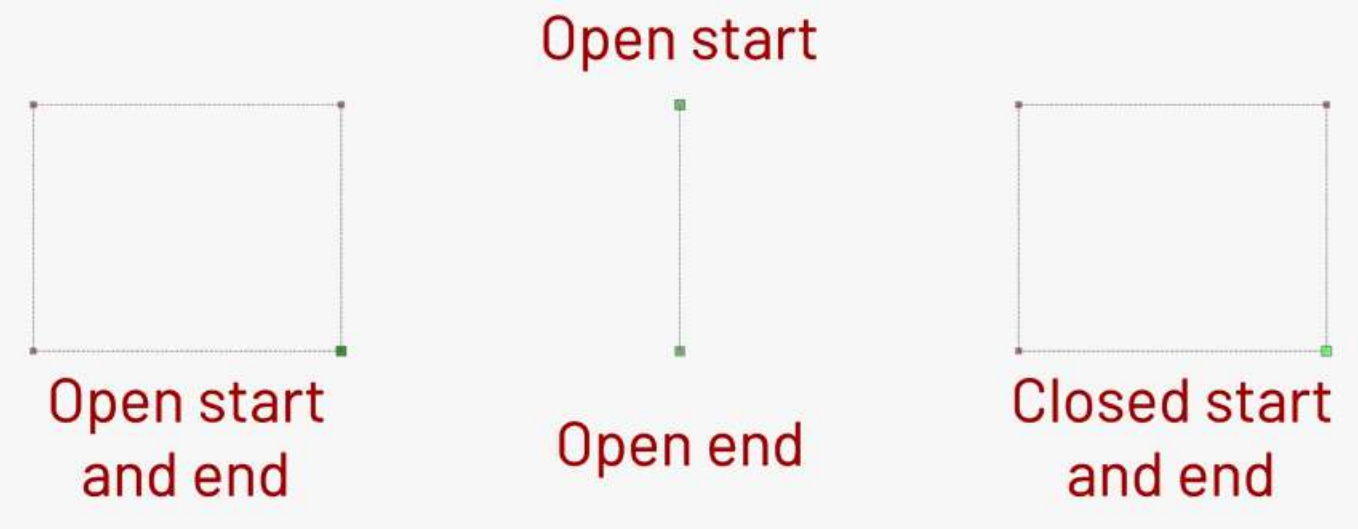
You can quickly reference the keyboard commands for each of these at any time by hovering over the **Edit Nodes** tool in the toolbar.

Action	Key	Description	Method
Delete node	D	Delete nodes and combine adjacent lines	Hover over a node (or select multiple nodes) and press key
Insert	I	Insert node anywhere on path	Hover over curve or line and press key
Insert node at midpoint	M	Insert node at the halfway point between two nodes	Hover over curve or line and press key
Convert smooth node to corner	C	Convert node to corner (curve handles will have independent angles)	Hover over smooth node and press key
Smooth corner node	S	Convert node to smooth (curve handles will have same angle)	Hover over corner node and press key
Break shape at point	B	Open path at node	Hover over a node and press key (move one of the nodes to reveal the gap)
Delete line	D	Delete line or curve (open or split path)	Hover over a line or curve and press key
Convert curve to line	L	Convert to line (straighten a curve and remove handles)	Hover over a curve and press key
Convert line to smooth curve	S	Convert to curve (add handles either side of line)	Hover over line and press key
Align Selected	A	Rotate selection by aligning segment to nearest horizontal, vertical, or 45° angle — all selected objects will rotate along with the segment	Hover over line and press key
Trim line	T	Cut line under mouse back to the next place it intersects with another line	Hover over curve or line and press key
Extend line	E	Extend line to nearest intersecting path	Hover over an open end point and press key

Node Types

The shape, size, and color of a node all give you important information about the node.

> **Having trouble seeing the nodes?**
>
> If you're having trouble seeing your nodes, go to the Display tab of Settings/Preferences and increase the size of your nodes with the **Cursors / Nodes** slider.



.

- Square nodes represent corner nodes.
- Round nodes represent smooth or bezier nodes, which have handles that can adjust the curves going in and out of them.
- Large nodes are start or end points of the shape.
- Large bright green nodes are closed nodes - they're the start node and end point of a closed shape. You can see an example in the shape all the way on the right in the screenshot above, labeled "Closed Start and End".
- Large dark green nodes with a black outline are the start node of an open shape. You can see an example in the line in the middle of the screenshot above, labeled "Open Start".
- Large dark green nodes with a light gray outline are the end node of an open shape. You can see an example in the line in the middle of the screenshot above, labeld "Open End".
- Large dark green nodes with a medium gray outline are the start node and end node of an open shape overlapping, which can result in a shape looking like it should be closed while actually being open. You can see an example in the shape all the way on t he left in the screenshot above, labeled "Open Start and End". Read more at Open Vs. Closed Shapes.

Requirements

- You can only use **Edit Nodes** with paths. You can create paths with Draw Lines or the Trace Image tools. Otherwise you can Import vector objects into LightBurn from another source.
- To use Edit Nodes with other, built-in vector objects, you need to turn them into standard paths. This requirement applies to:
  - Text Objects
  - Primary Shapes: Rectangles, Ellipses, and Polygons
  - Barcodes

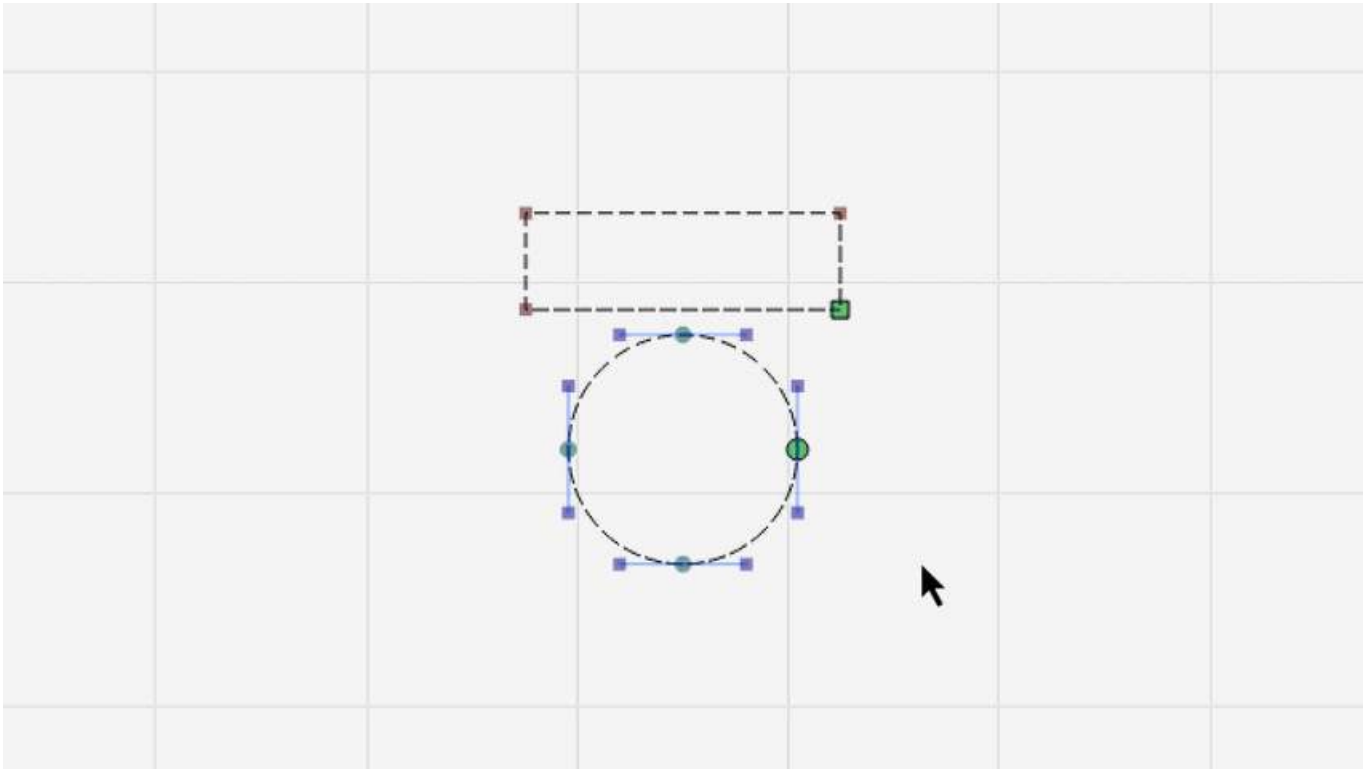
> **Converting objects to paths**

> There are two ways to do convert your object to a standard path.
>
> - With the Edit Nodes tool open, double-click on the shape to convert it to a path. If you're already in the Edit Nodes tool, hovering over a shape that needs to be converted will show a tool tip warning you "This is an object, like a rectangle or text. Double-click it to convert it to an editable path."
> - Go to Edit → Convert to Path to turn them into standard paths.

- You cannot use Edit Nodes with Grouped objects. Ungroup them first.

Snap to Existing Geometry

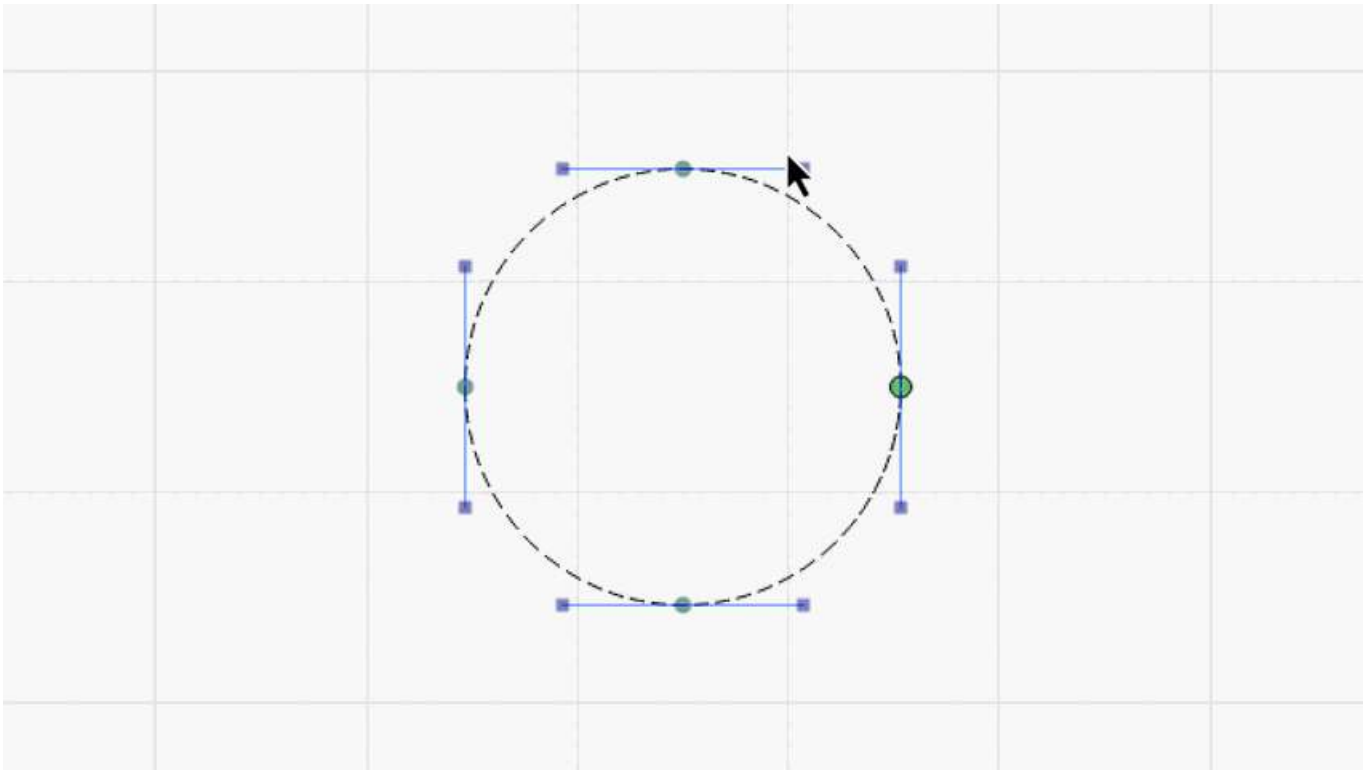
Snapping allows you to align your nodes or handles to specific points on existing objects.



Constrain Movement

Hold `Shift` while moving a node or handle to constrain movement to a 90° or 45° angle from its starting position, indicated by temporary guidelines in your Workspace.

If you select multiple nodes or handles at once, only the movement of the last-selected node or handle will be constrained, and the other nodes or handles will move along with it.



Trim and Extend Demo

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=1sbK-Dux214

There is also a separate tool, Trim Shapes that you can use to perform the **Trim Line** function.

Align to Angles Demo

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=ihMS5vhtomo

How to Practice

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=O7kzRu1f3Xs

Related Topics

- Draw Lines
- Trim Shapes
- Snapping
- Selection
- Images vs. Vectors

- Open vs. Closed Shapes
- Auto-Join Selected Shapes
- Break Apart
- Trace Image

Trim Shapes

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=DGkAJtQxhKM

Use the **Trim Shapes** tool to cut line segments back to the next intersection with another line.

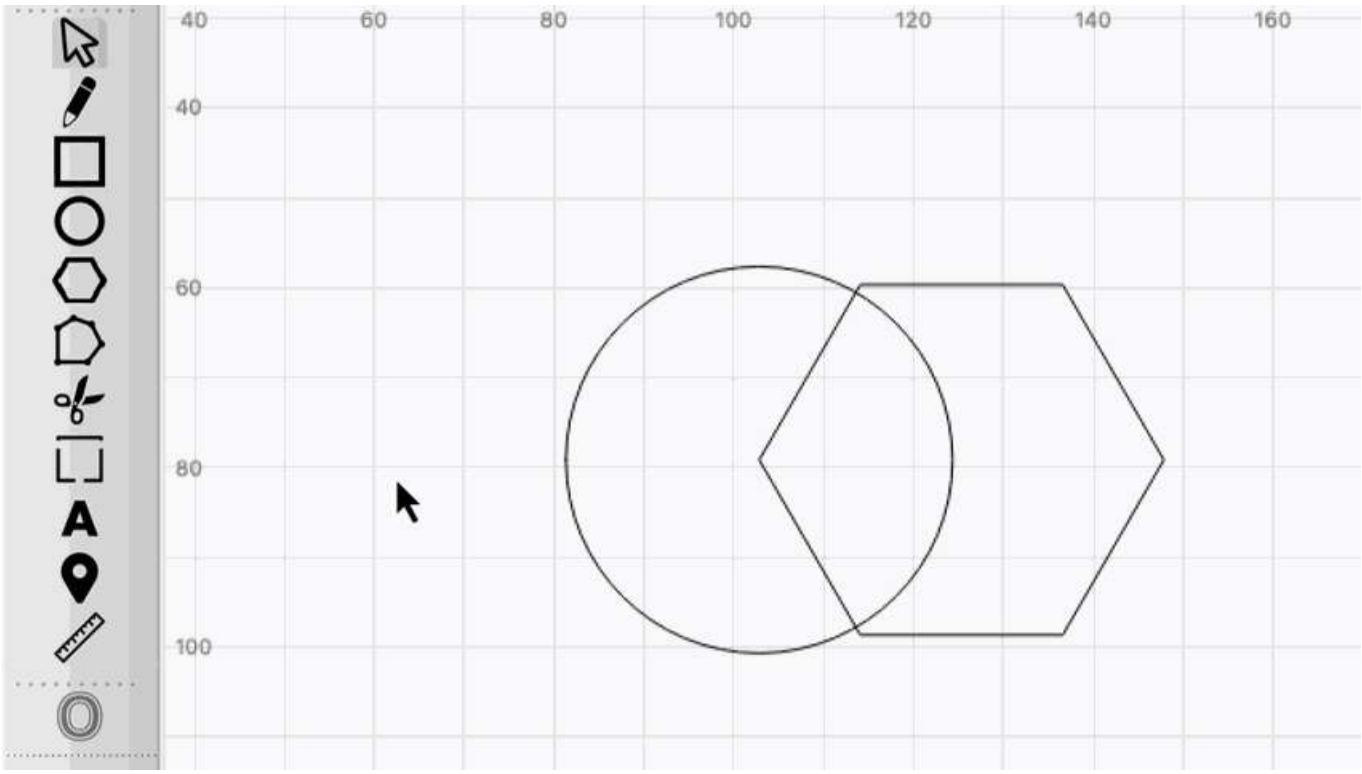
If a shape does not intersect with any other lines, Trim Shapes deletes the entire shape.

Using Trim Shapes

To enable **Trim Shapes**, click the scissor icon in the Creation Toolbar or press `Ctrl`/`Cmd`+`K`.



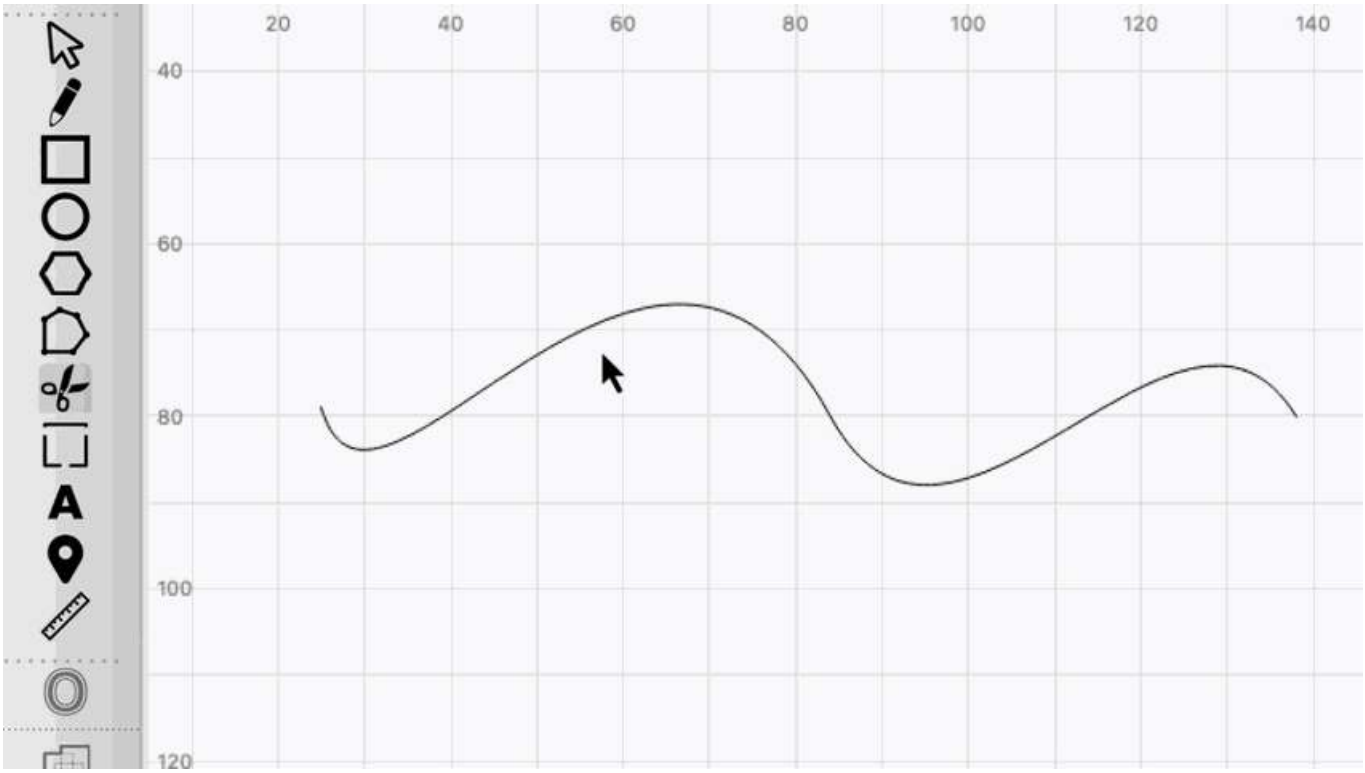
1. After activating the Trim Shapes tool, hover over any vector object in your Workspace to see a preview of the effect Trim Shapes will have on that object.
2. When you hover over a valid line segment, the cursor will change to the scissor icon, and the segment to be deleted will turn red.
3. Click to delete the segment.



Primary Shapes

This feature works with Primary Shapes, but will remove the ability to edit their special Shape Properties, as they are Converted to Paths in the process.

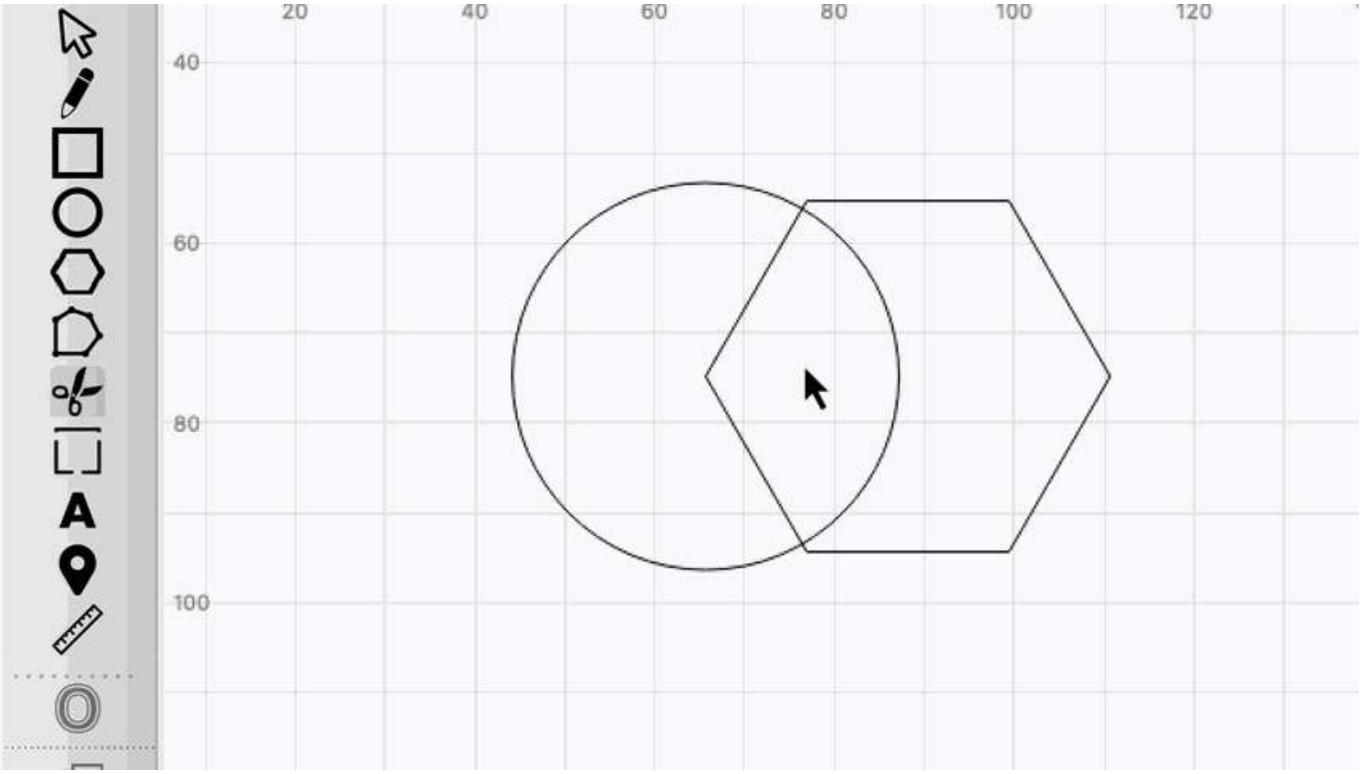
If you hover over a shape that does not intersect with any other lines, the entire shape will be highlighted in red, and deleted when you click.



Auto-Joining With Trim Shapes

When possible, **Trim Shapes** joins the new vector objects at the points where they used to intersect, combining them into a single path.

When a shape has been joined with another, it will automatically be selected, and you'll see the animated dash pattern that indicates selected objects.

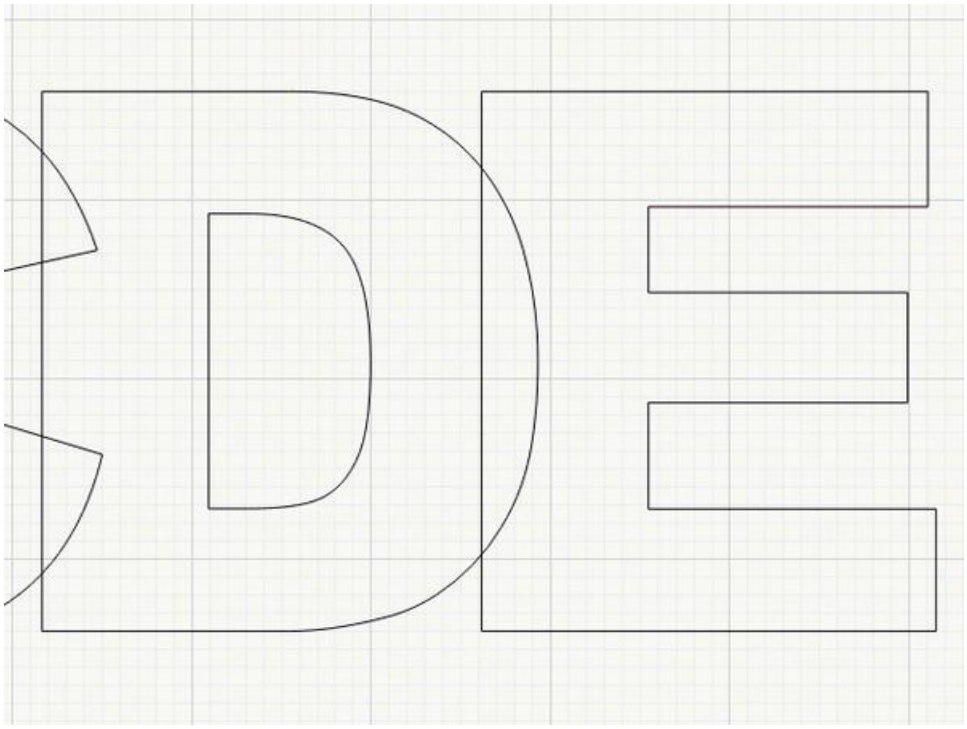


Note

Hold `Ctrl` / `Cmd` while clicking to disable automatic joining when using Trim Shapes.

Preserve Trimmed Line Segment

Sometimes, it's useful to preserve a copy of the trimmed line segment. Hold `Shift` while trimming to make a copy of the removed line segment and move it to the currently selected layer. LightBurn will warn you if this is the same layer the shapes being trimmed are on.



Troubleshooting

- **Trim Shapes** does not work on Grouped objects — you'll need to **Ungroup** them first.

Related Topics

- Edit Nodes
- Boolean Tools
- Cut Shapes

Numeric Edits Toolbar

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=Orf9v0bplpA

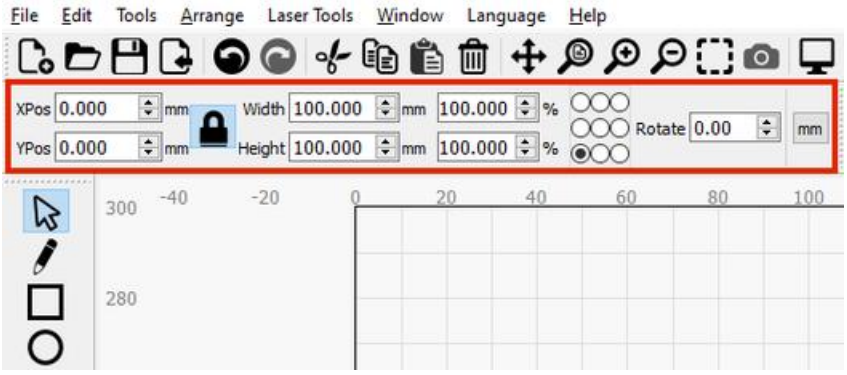
Use the **Numeric Edits Toolbar** to adjust the size, position, and orientation of shapes in your selection using precise numeric values.

When adjusting objects through the Numeric Edits Toolbar, all objects in your current selection are treated as a single unit, maintaining their relative positions and scale as you make changes.

Accessing the Numeric Edits Toolbar

The **Numeric Edits Toolbar** is located in the top toolbar in LightBurn by default, to the left of the Text Options Toolbar.

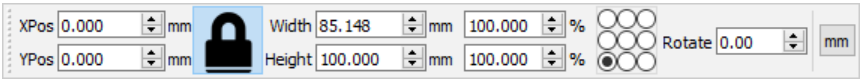
If you have closed the Numeric Edits Toolbar, go to Window → **Numeric Edits** to re-enable it. To restore it and all other windows to their default positions, go to **Window** → Reset to Default Layout



> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Numeric Edits Toolbar Options

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



9-Dot Control

> **Note**

When working with any of the Numeric Edits options, imagine the selection area as the smallest possible rectangle that can contain every object in your selection. Each position in the 9-Dot Control is a point around the perimeter of that rectangle, or in its center.

The **9-Dot Control** determines the location on the current selection that is referenced when displaying and adjusting selection coordinates; the anchor point which objects are rescaled in toward or away from; and which point to use as the center of rotation when rotating the selection.

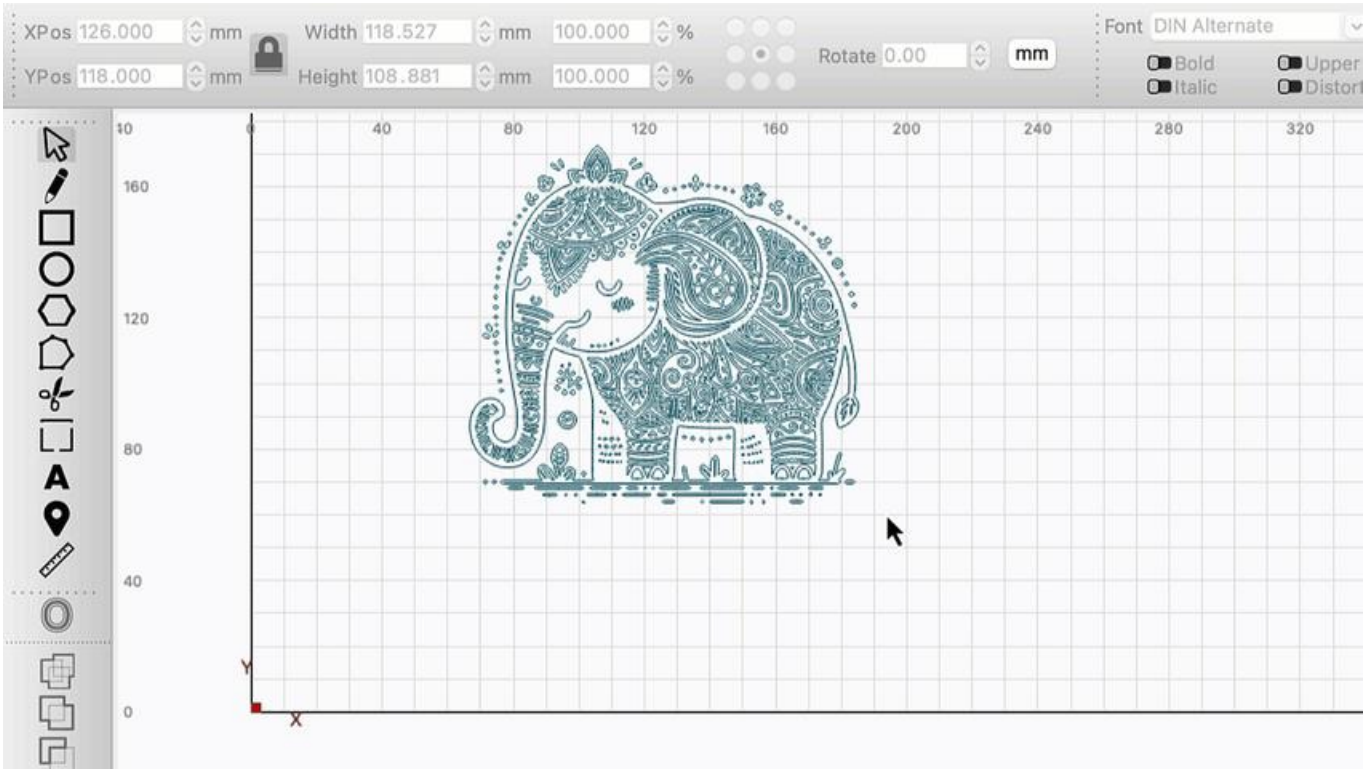
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=ZId8vhwgAOM

XPos and YPos

XPos and **YPos** display the current X (horizontal) and Y (vertical) positions of your selection, relative to the point in your selection indicated by the **9-Dot Control**, and your laser's Origin.

The laser's Origin is indicated by a small red square in a corner of the Workspace. Positive XPos and YPos values are always within the Workspace, and negative values outside of it.

Adjusting either field will move the indicated position on the current selection to the new location you enter, and all objects in the selection will move in unison.



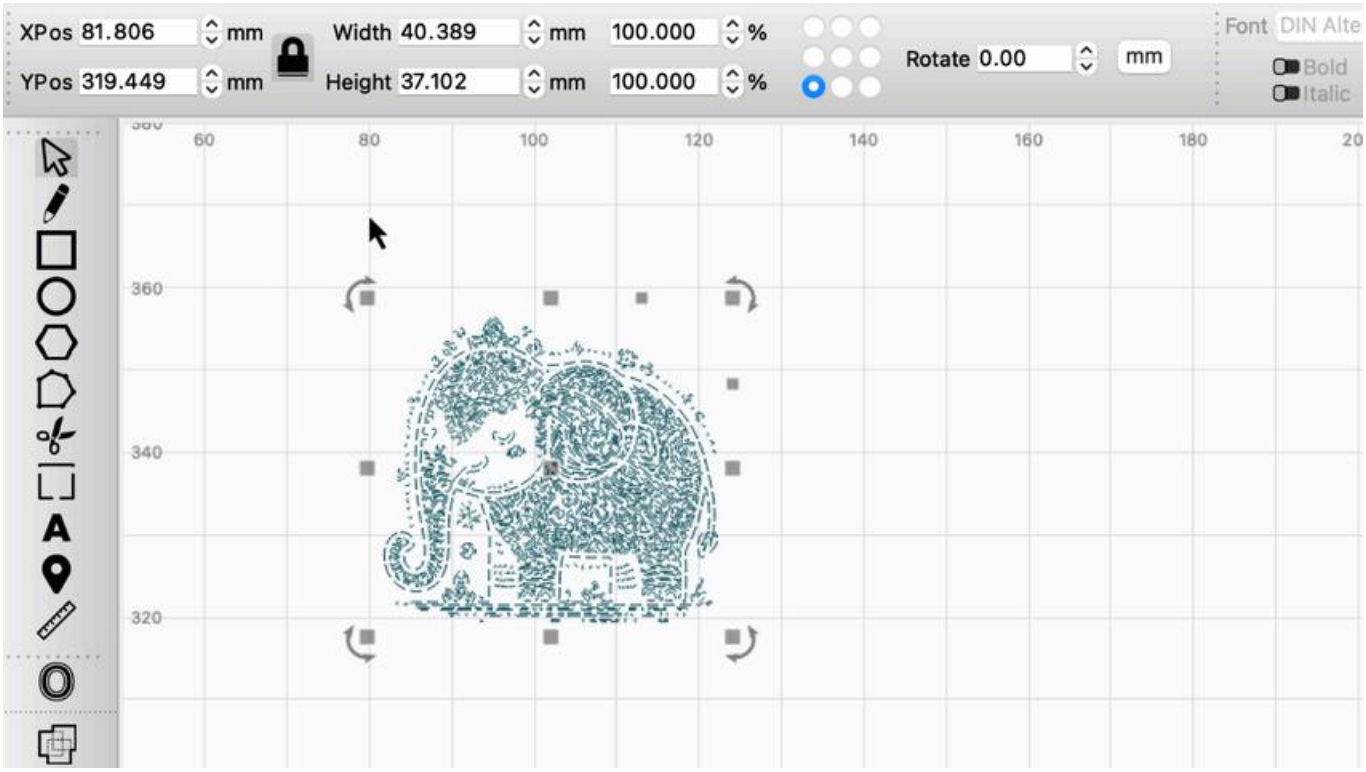
Tip

You can adjust position by imprecise values directly in your Workspace using the Move handle on your selection.

Width and Height

Width and **Height** display and determine the length of your selection in the X (Width) and Y(Height) dimensions. You can enter exact values in the field directly to the right of Width and Height, or change the size of either dimension by percentage in the fields to the right of those.

The chosen location in the 9-Dot Control determines an anchor point that objects will be rescaled in toward or away from.



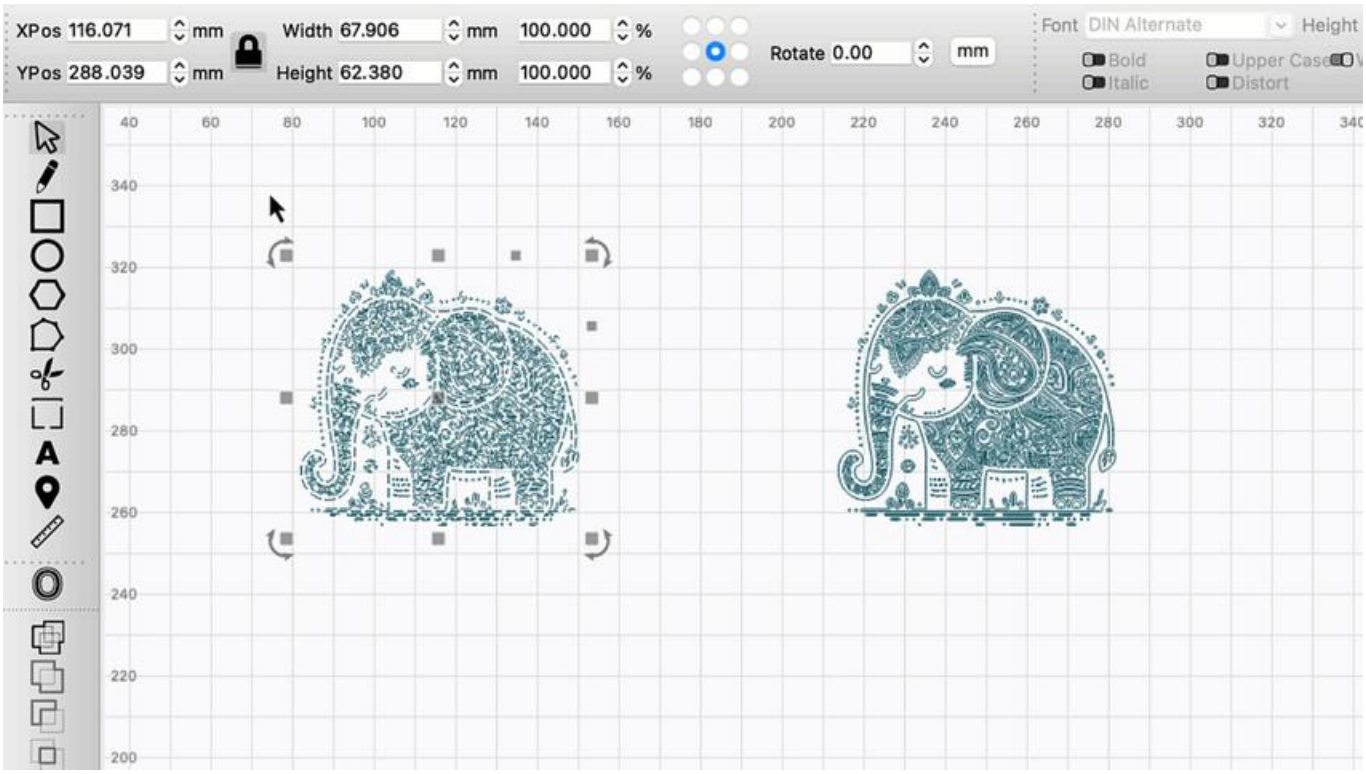
> **Tip**
>
> You can adjust Width and Height by imprecise values directly in your Workspace using the Size handles on your selection.

Lock

The **Lock** icon to the left of **Width** and **Height** determines whether the aspect ratio of the selection is maintained as objects are resized.

- In locked mode — indicated by a highlighted, closed lock icon — the proportions of your selection remain consistent as they're resized.

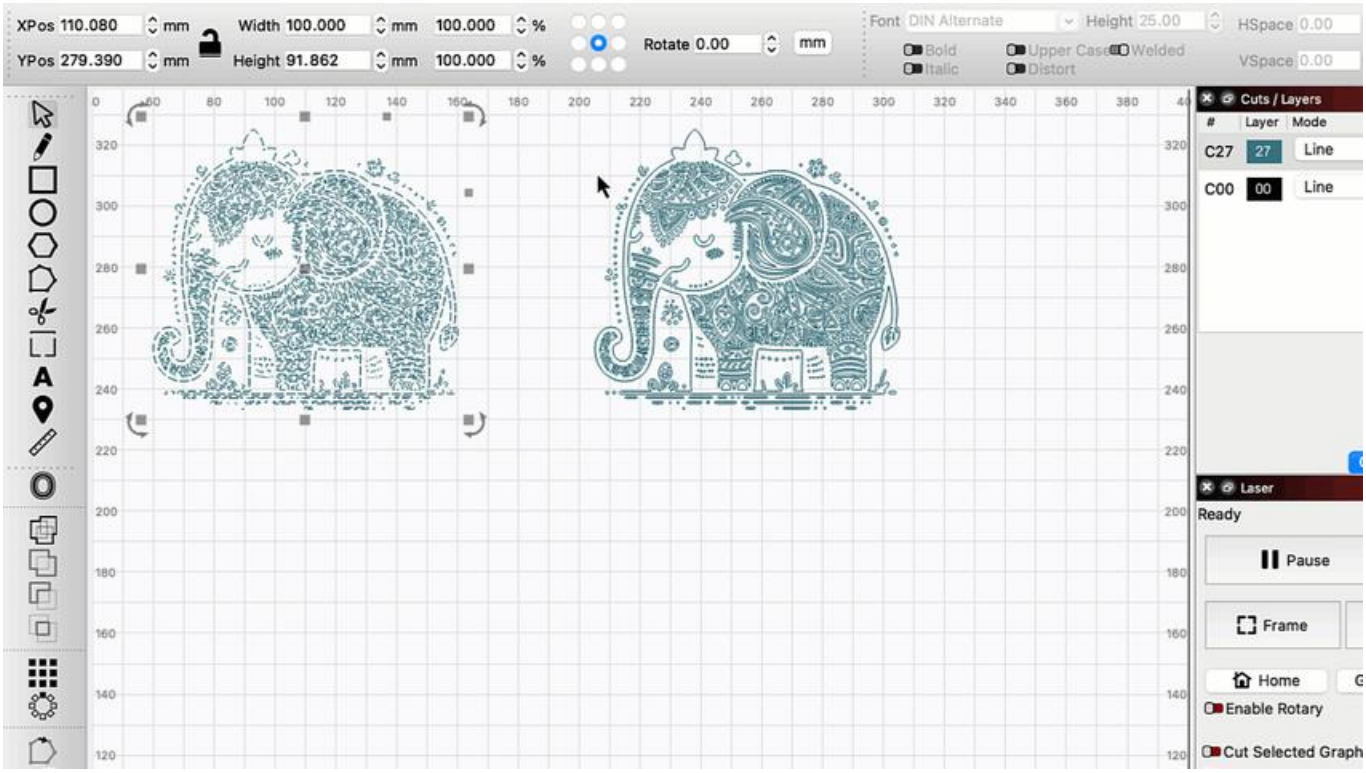
  As an example, a rectangle that is currently 50 mm wide and 25 mm tall is twice as wide as it is tall, for a 2:1 aspect ratio. If the aspect ratio is locked, changing the width to 80 mm will automatically change the height to 40 mm.
- In unlocked mode — indicated by an unhighlighted, open lock icon — the proportions of your selection will not remain consistent as they're resized, and will instead change independently.
- Click the icon to change modes.



Rotate

Use the **Rotate** field to turn the current selection by a precise number of degrees. Positive values rotate the selection clockwise, while negative values rotate it counterclockwise.

The chosen location in the 9-Dot Control determines the center of rotation.



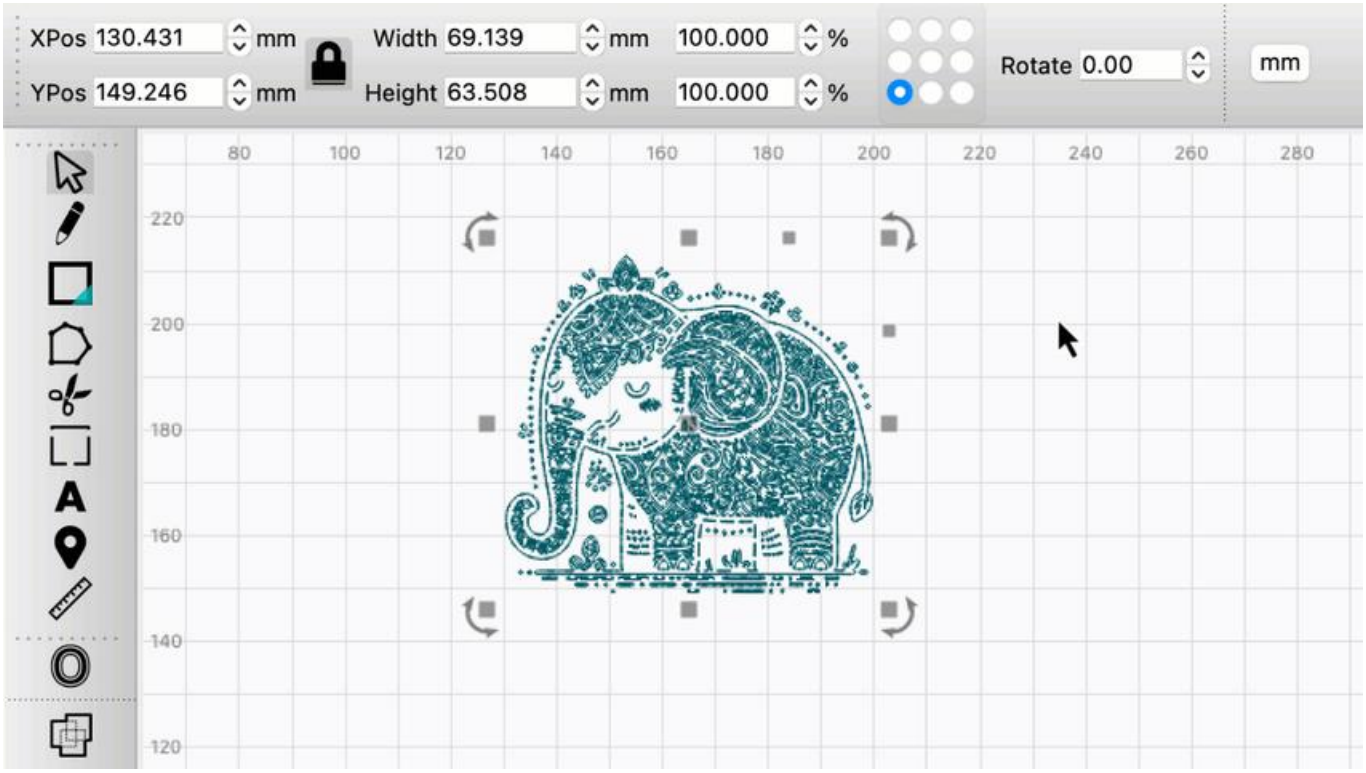
> **Tip**
>
> You can rotate your selection by imprecise values directly in your Workspace using the Rotate handles, or by precise values using hotkeys.

Units Toggle

Click the **Units Toggle** to switch your current design units between imperial and metric. You can also adjust units in the Settings window.

The current unit of measurement is displayed as **mm** for millimeters or **in** for inches.

> **Design and Control Units**
>
> In LightBurn 2.1, Design Units and Control Units are separate, but linked by default. See Link Control and Design Units for more information.



Equation Support

The **XPos**, **YPos**, **Width**, **Height**, and **Rotate** fields all accept equations.

Examples of valid equations are:

- 58 + 37
- 89 - 13
- 10.25 / 2
- (10+2) * 4 + 1

Press enter to process the equation and adjust the value.

You can also use the constants e and pi, and functions like sin, cos, tan, sqrt, abs, atan, log, pow, and more.

Automatic Unit Conversion

The **XPos**, **YPos**, **Width**, and **Height** fields accept units of measurement as well.

If you are working in millimeters, but you want to create a shape that is 5 inches wide, you can enter 5in or 5" into the width field and LightBurn will convert it to millimeters for you.

This works in the opposite direction too — if you're working in inches, you can enter mm or cm and LightBurn will convert the values to inches.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=tk9BUgZFWWo

Related Topics

- Transform Controls
- Text Options Toolbar
- Shape Properties

Convert to Path

Convert to Path transforms built-in shapes created in LightBurn into paths made up of lines, curves, and nodes, that can be edited using the Edit Nodes tool.

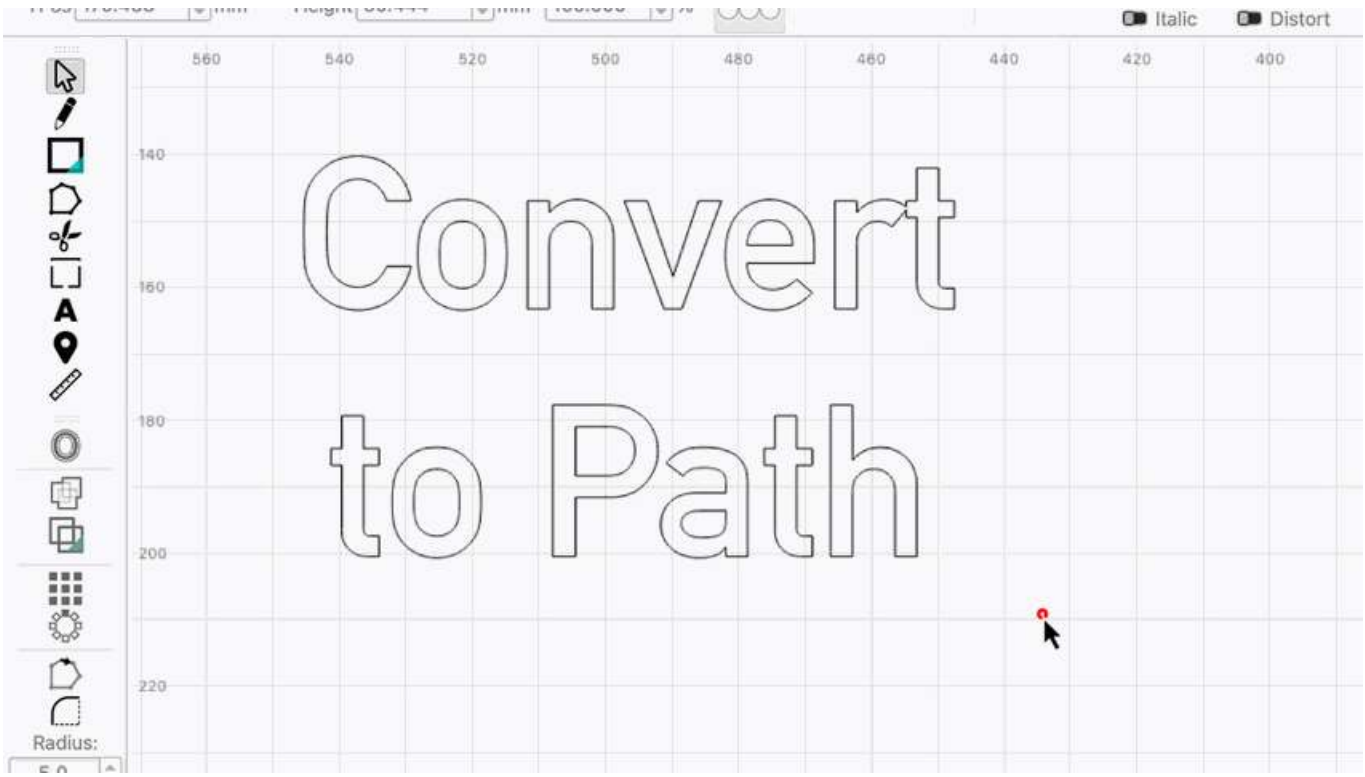
The following built-in shapes must be converted to paths in order to edit their lines, curves, and nodes:

- Primary Shapes
- Text Objects
- Bar Codes

Using Convert to Path

Select all shapes you wish to convert, then go to Edit → **Convert to Path**, press `Ctrl`/`Cmd` + `Shift` + `C`, or right-click in your Workspace and select Convert to Path from the context menu.

The objects become normal paths, and their lines, curves, and nodes can be adjusted using the Edit Nodes tool. The letters of Text Objects are separated into discrete shapes that can be moved and manipulated individually.



Troubleshooting

- Grouped objects cannot be converted to paths, and the option will be grayed out or unavailable if you have Grouped objects selected. **Ungroup** any object you'd like to convert.

- After converting an object to a path, you will lose the ability to adjust attributes associated with the type of shape it was converted from.
  - The following attributes can no longer be edited in the Shape Properties Window:
    - The **Width** and **Height** of Rectangles, Ellipses, and Polygons.
    - The **Corner Radius** of Rectangles.
    - The number of **Sides** of Polygons.
    - The **Max Width** of Text.
  - Text properties are no longer adjustable in the Text Options Toolbar.
  - Variable Text input will no longer be substituted for different output.
  - The content of a Bar Code can no longer be edited.

Related Topics

- Edit Nodes
- Create Rectangle
- Create Ellipse
- Create Polygon
- Create and Edit Text
- Create Bar Code

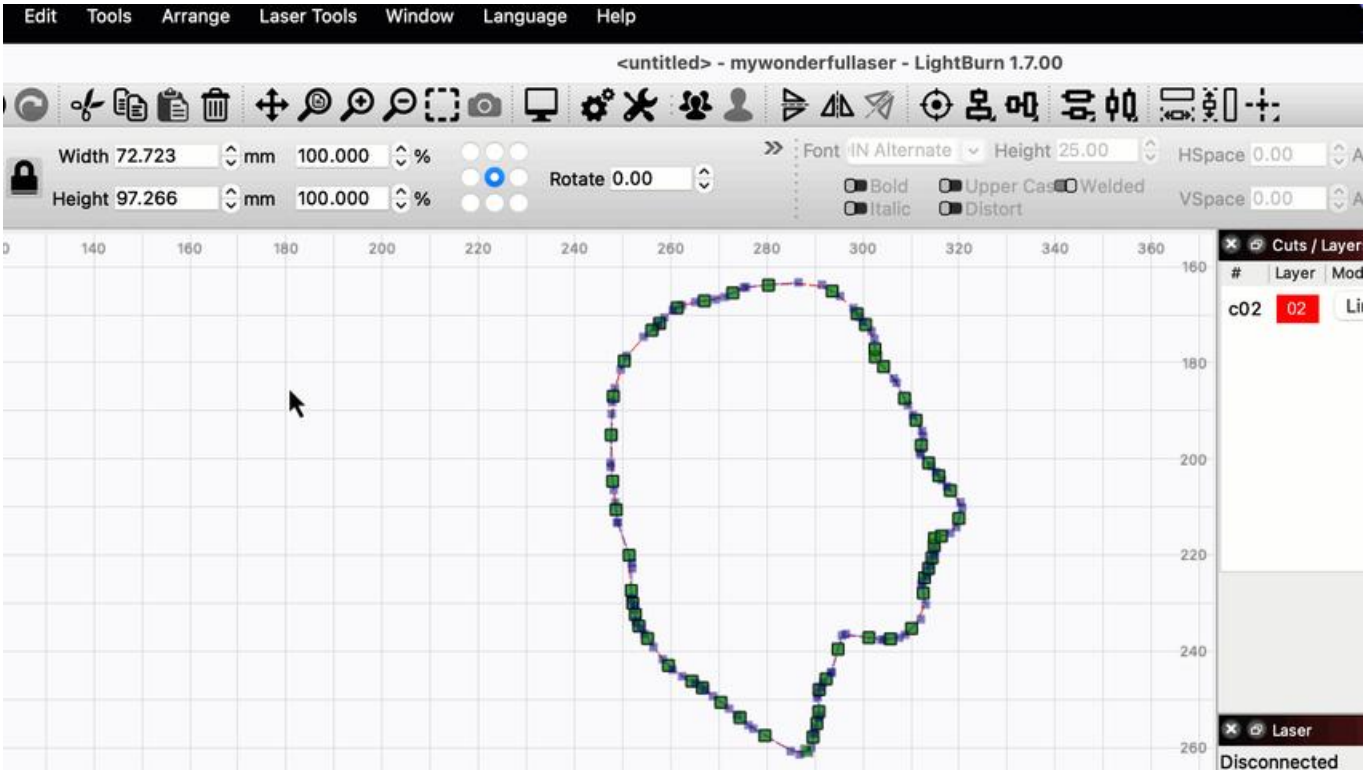
Auto-Join Selected Shapes

Auto-join selected shapes connects line or curve segments whose start and end nodes are directly on top of (or very near to) one another, but not technically connected. When the segments are connected, they are combined into a single path (a sequence of lines, curves, and nodes).

Auto-Join is most often helpful in cases where you've Imported graphics containing open shapes that you would like to use with a function that requires closed shapes.

To connect start and end nodes using Auto-Join, the nodes must be within .05 mm of one another.

To use Auto-Join, select all segments you want to join, then go to Edit → **Auto-join selected shapes**, or press `Alt`/`Option`+`J`.



Troubleshooting

In some cases where **Auto-Join** fails to join segments with start and end points within the necessary .05 mm tolerance, it can help to first go to the Arrange Menu and select Break Apart (`Alt`/`Option`+`B`), before again trying Auto-Join.

Related Topics

- Edit Nodes
- Close Path
- Close Selected Paths With Tolerance

- Break Apart
- Open vs. Closed Shapes

Close Path

Close Path closes shapes with their start and end points close together but disconnected by creating a new line between the points.

Several functions in LightBurn, including Fill Mode, require shapes that are closed loops, with start and end points that are the same. If the start and end points of a shape are very close, but not quite connected, use Close Path to connect them.

Using Close Path

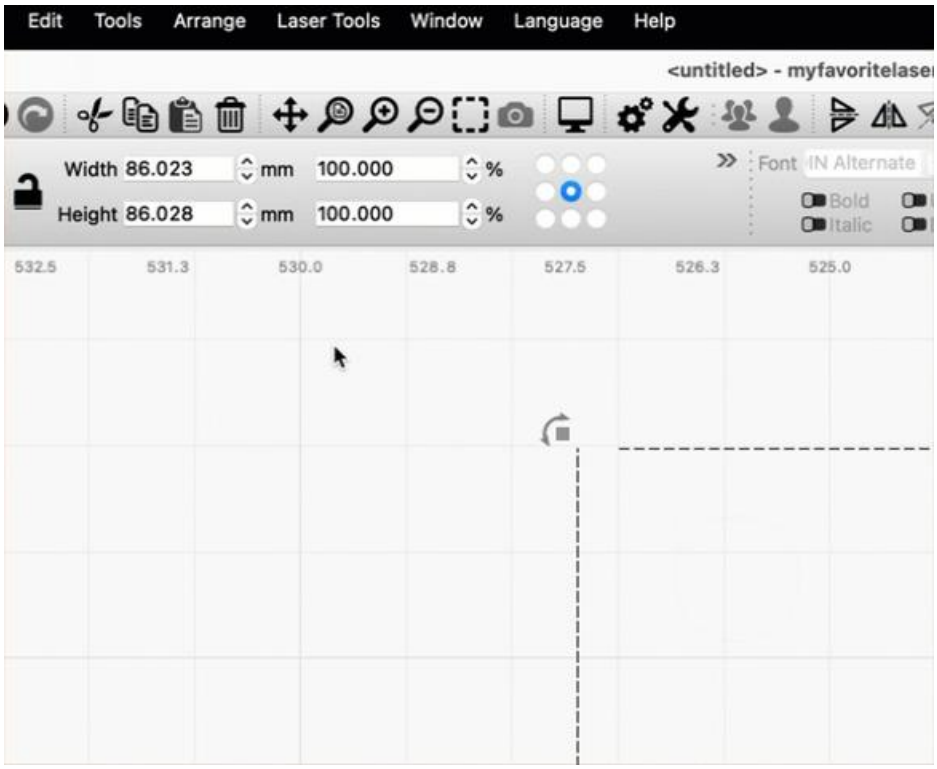
Select all shapes you wish to close, then go to Edit → **Close Path**.

You can select and close multiple paths at once, but, in order for a path to be connected by Close Path, it must already be completely connected except for its start and end nodes.

Close Path cannot connect fully separate paths, each with their own start and end nodes.

The start and end nodes of selected paths must be within .5 mm of one another in order for Close Path to connect them.

For gaps larger than .5 mm, use Close Selected Paths with Tolerance instead.



Related Topics

- Edit Nodes
- Open vs. Closed Shapes

- Close Selected Paths with Tolerance
- Auto-Join Selected Shapes

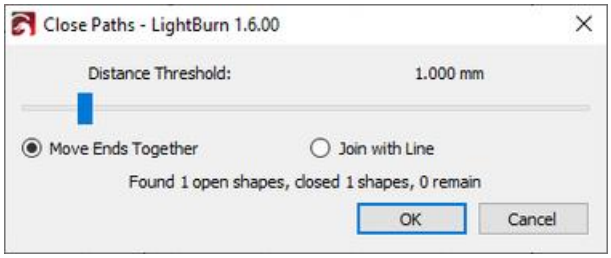
Close Selected Paths With Tolerances

Close selected paths with tolerance closes individual shapes with disconnected start and end points by creating a new line between the points or moving the two points together. By specifying a tolerance, you can determine how near to one another the start and end points must be in order to be closed.

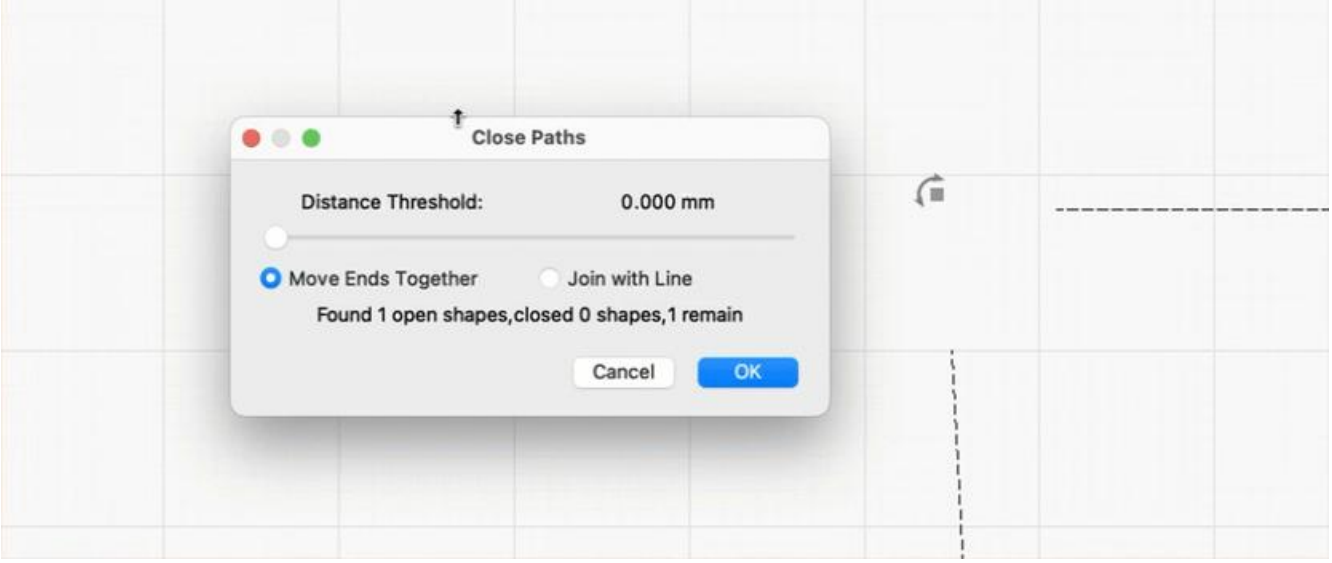
Several functions in LightBurn, including Fill Mode, require shapes that are closed loops, with start and end points that are the same. This option works similarly to Close Path to close shapes with start and end points that are not the same, while providing greater control over how far apart they must be in order to connect them, and how they will be connected.

Using Close Selected Paths With Tolerance

Select all paths you wish to close and go to Edit → **Close selected paths with tolerance** to open a dialog window.



- Use the slider to set the **Distance Threshold**, or tolerance, which determines the maximum distance between start and end points to be connected. Start and end points that are farther apart than this threshold will be ignored.
- Select **Move Ends Together** to connect the start and end points by bringing existing lines together.
- Select **Join with Line** to create a new line segment between start and end points.
- The readout at the bottom of the window shows how many open shapes were found, how many shapes have been closed, and how many open shapes remain in your selection.
- Click **OK** to confirm the changes and close the selected shapes, or **Cancel** to discard them and leave the shapes open.



Close selectected paths with tolerance can close muliple paths at once, but they must already be completely connected except for their start and end nodes. This tool cannot connect fully separate paths, each with their own start and end nodes.

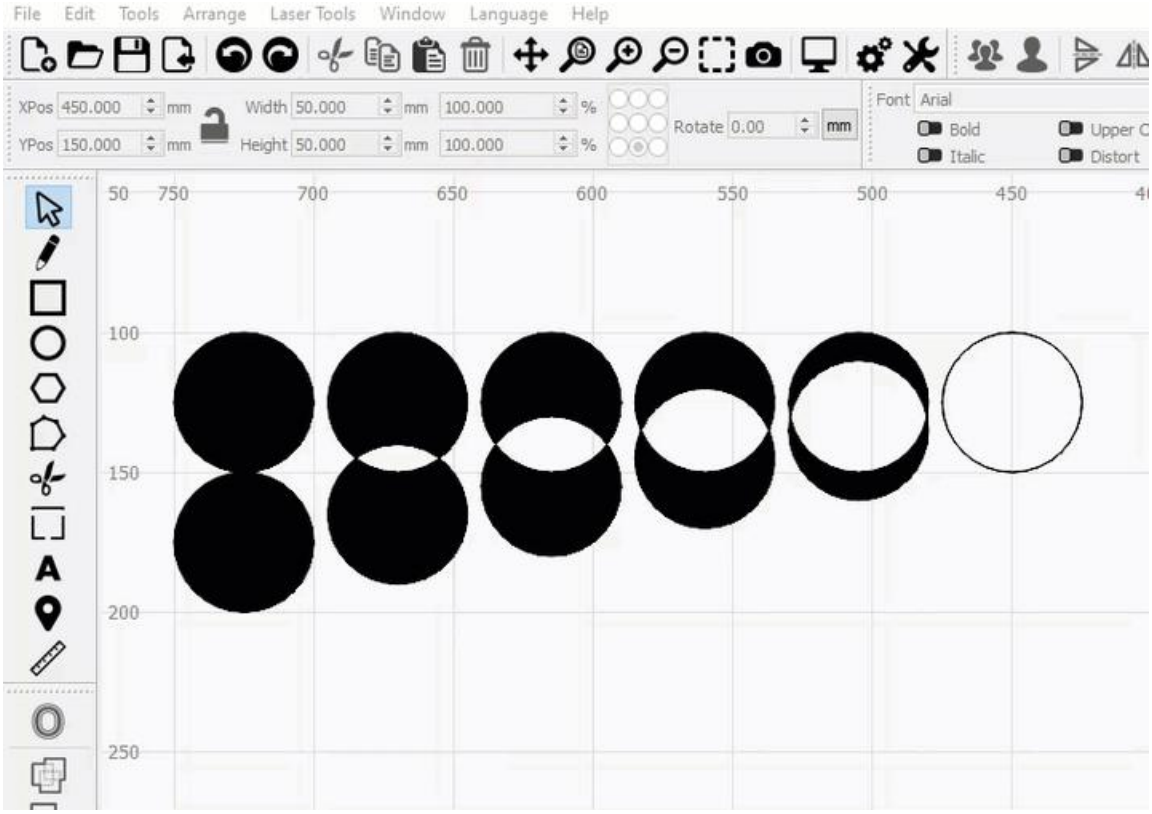
Related Topics

- Close Path
- Auto-Join Selected Shapes
- Edit Nodes
- Open vs. Closed Shapes

Delete Duplicates

Delete Duplicates finds and removes duplicated objects in your project. Objects don't need to be selected in order to be identified and deleted, but must be Ungrouped and identical in size, shape, and position.

To use Delete Duplicates, go to Edit → **Delete Duplicates** or press `Alt`/`Option`+`D`.



Delete Duplicates in use. The circle furthest to the right is duplicated.

Duplicated objects can cause filled objects to appear as outlines in the Workspace, even with View Style set to Filled. Those objects may also disappear in the Preview window. Duplicated objects can cause extra passes in Line Mode and failures to engrave in Fill Mode.

Troubleshooting

- Make sure objects are Ungrouped in order to be detected by **Delete Duplicates**.

Related Topics

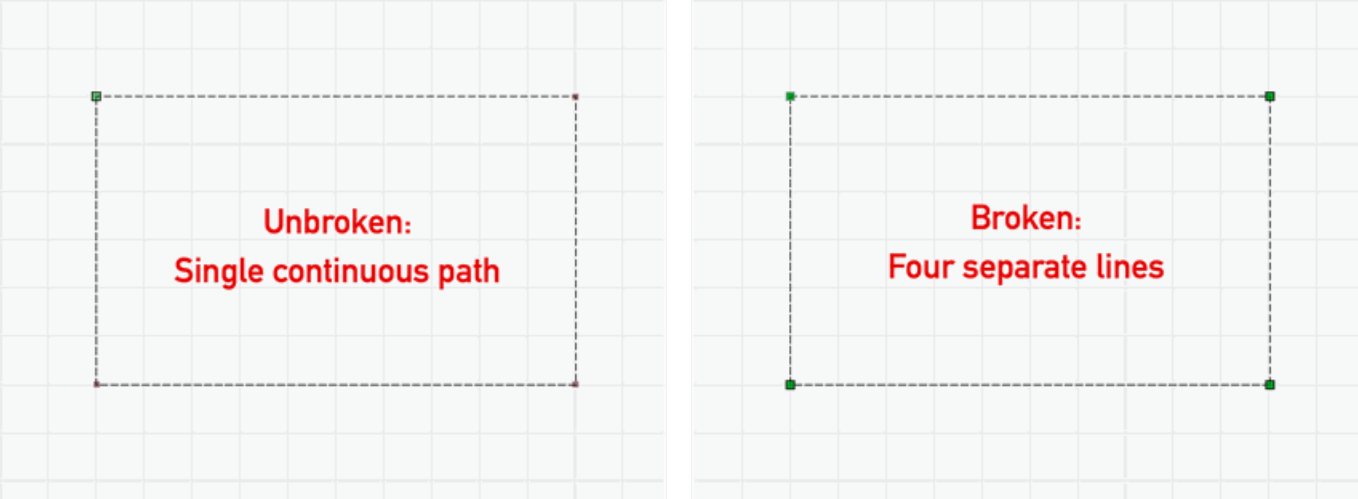
- Preview
- Additional Selection Tools

Break Apart

Break Apart separates the paths of selected vector graphics into the individual curves and lines that make them up. The individual segments become disconnected, allowing them to be selected and manipulated independently.

To use Break Apart, select all objects you wish to break, then go to Arrange → **Break Apart**, or press `Alt` / `Option` + `B`.

For example, a rectangular vector graphic is a path made of four straight lines connected by four nodes, with one node that is both the start and end of the path. Breaking Apart a rectangle results in four disconnected lines, each with their own start and end node.



Left: an unbroken rectangle with one start and end node in green Right: four separate lines with eight start and end nodes

Related Topics

- Edit Nodes
- Auto-Join Selected Shapes
- Open vs. Closed Shapes

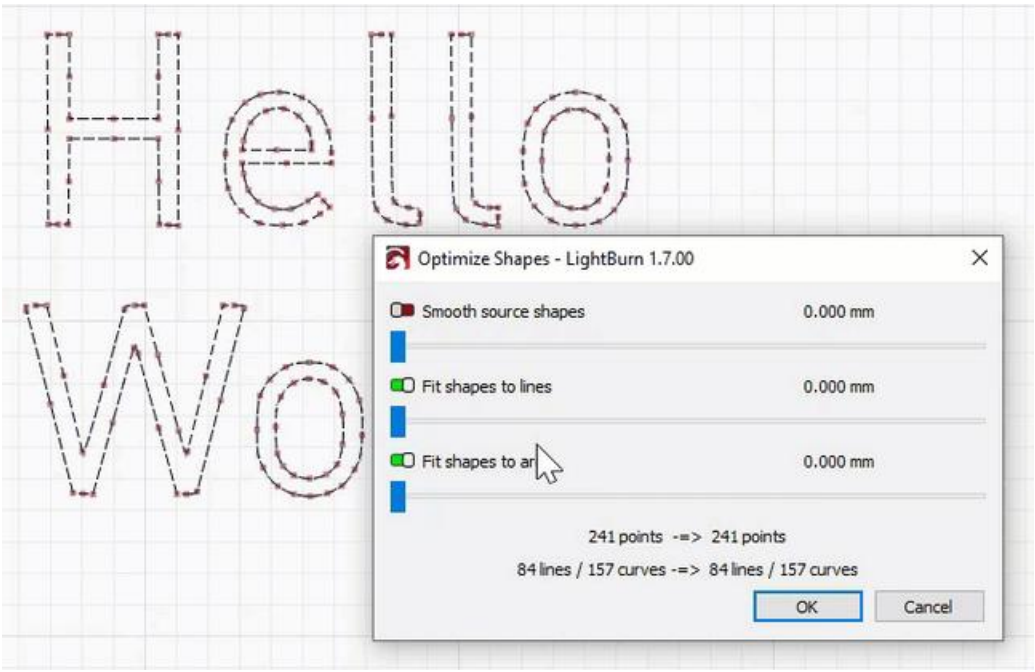
Optimize Selected Shapes

The **Optimize Shapes** tool simplifies and smooths selected shapes by reducing the node count or by fitting them to arcs or lines. This is often useful when importing art created in other software, or cleaning up traced artwork.

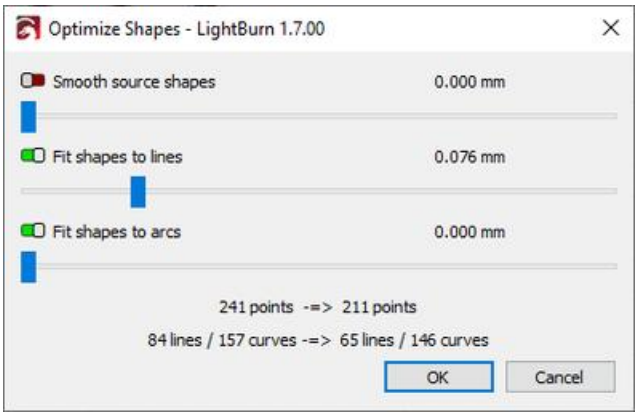
Using Optimize Shapes

To use **Optimize Shapes**, select the shapes you want to Optimize, and make sure they're Ungrouped. Go to **Edit → Optimize selected shapes** or use the shortcut `Alt` / `Option` + `Shift` + `O` to open the Optimize Shapes window.

Text objects or Primary Shapes must be Converted to Paths in order to Optimize them.



Optimize Shapes Window



The **Optimize Shapes** tool adjusts selected shapes within a specified error tolerance, which defines the maximum amount the new optimized shapes can deviate from the original source shapes.

Error tolerance is controlled by the sliders beneath each optimization option. Switches next to each option control whether they are enabled or disabled.

Optimizations are visible in your Workspace as you make changes — adjust the tolerance sliders or enable and disable options to see the result of different combinations of tolerances and optimizations.

Press **OK** to confirm changes and Optimize the source shapes, or **Cancel** to discard the changes and leave the shapes as they are.

Smooth source shapes

Smooth source shapes smooths out wavy or jagged lines. This is often useful for cleaning up vectors from Image Tracing. Increase the value to strengthen the smoothing effect, at the expense of potentially losing small details or rounding off sharp corners.

Fit shapes to lines

Fit shapes to lines simplifies shapes by turning sections that are close to being a line into lines. This can reduce node count or help clean up graphics with many small or overlapping lines.

Fit shapes to arcs

Fit shapes to arcs simplifies shapes by converting lines into arcs. This can reduce node count or recreate true arcs from software that exports them as many small line segments.

Points, Lines, and Curves

At the bottom of the window, LightBurn displays the number of **Points** (nodes), **Lines**, and **Curves** (arcs) in the original shapes and in the optimized versions of the shapes.

Troubleshooting

- If the **Optimize selected shapes** menu option is grayed out, or the shortcut key doesn't open the **Optimize Shapes** window, make sure you have at least one object selected.
- If the **Optimize Shapes** window shows 0 points, 0 lines, and 0 curves, make sure your shapes are Ungrouped and that any Primary Shapes and Text have been Converted to Paths.

Related Topics

- Trace Image
- Edit Nodes

Warp and Deform

Use **Warp Selection** and **Deform Selection** to stretch and shape objects in a non-linear fashion by dragging handles positioned around the perimeter of the selection or in a grid overlaid on the selection.

Using Warp or Deform

Select all objects you wish to edit, and choose either **Warp Selection** or **Deform Selection** from the Tools Menu.

Then click and drag the handles that appear on the object to make your changes.

You can modify the edit behavior in the following ways:

- Double-click on a handle to reset it.
- Hold `Shift` while dragging a handle to maintain horizontal symmetry.
- Hold `Alt`/`Option` while dragging a handle to maintain vertical symmetry.

Warp Selection

Stretch and compress your selection according to its rectangular perimeter by dragging the handles at each of the four corners.

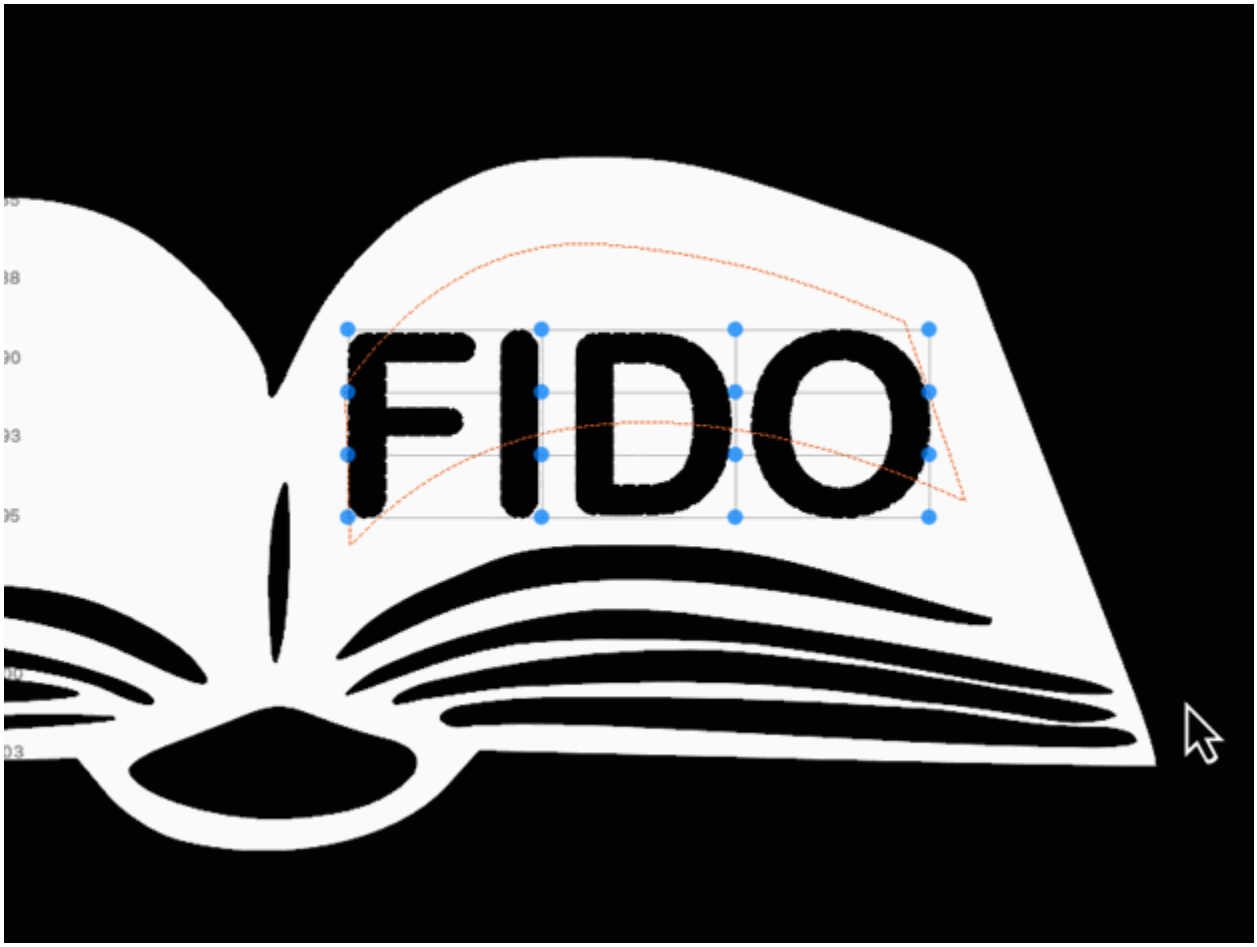


Use **Warp** to:

- Adjust the percieved perspective of an object (apply a perspective warp)
- Stretch or compress the corners of objects
- Create dramatic text effects

Deform Selection

Distort your selection in an organic manner by moving the 16 handles at the intersections of the overlaid grid.



Use **Deform** to:

- Fit text into the shape of another object
- Make an object fit the countours of another
- Create interesting organic shapes

Related Topics

- Transform Controls
- Shear
- Bend Text
- Apply Path to Text

Two-Point Rotate / Scale

Two-Point Rotate / Scale allows you to turn and resize objects around custom pivot points in your Workspace.

Using Two-Point Rotate/Scale

Select the objects you wish to edit and go to Arrange → **Two-Point Rotate / Scale**, or press `Ctrl`/`Cmd`+`2`.

The Status Bar will prompt you to **Select the first point - the center of rotation**. Any object can be Rotated and Scaled using this tool, including images, Text Objects, Primary Shapes, and custom paths.

Click anywhere in the Workspace to choose this point — you can choose a point on any object in your project (such as nodes, midpoints, intersections, corners), or even a point in empty space.

The Status Bar will then prompt you to **Click and drag the 2nd point, where you will rotate from**. As you click and drag, the object Rotates.

Hold `Shift` while dragging to also Scale the shape while rotating it.

As you drag, your cursor will Snap to existing objects in your Workspace, including lines, corner points, or other nodes.

Release the mouse to apply the previewed edits, or press `Esc` to discard the edits and return the object to its original position.

Matching the Sides of Two Objects

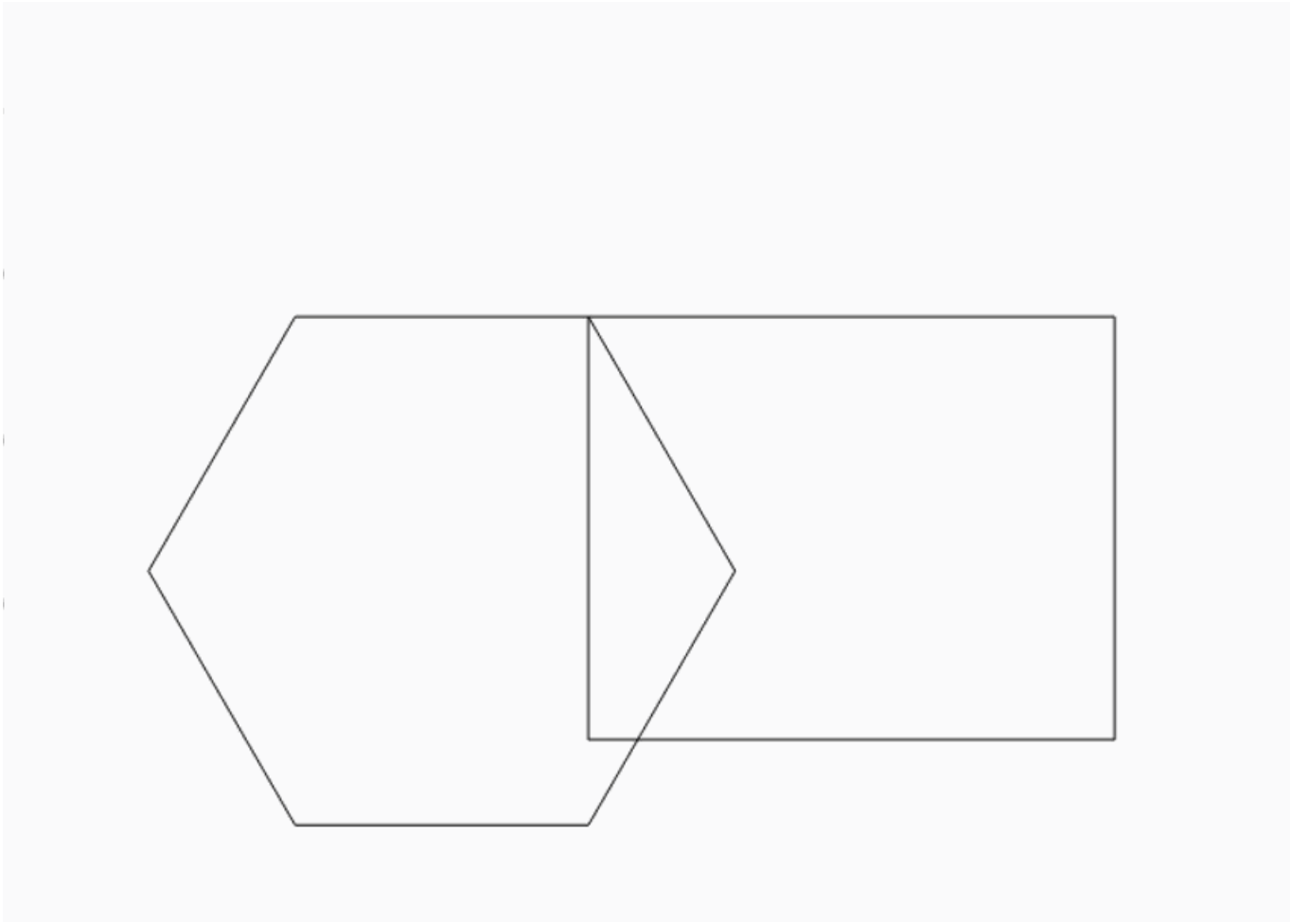
To align the sides of two objects and make their sides an equal length, start by aligning a node from each side of each object using the Select tool and Snapping. As the cursor hovers over a node, it will change to a crosshair, to assist in precise placement.

Next, select the object you wish to edit, open the **Two-Point Rotate / Scale** tool, and select the aligned node as the center of rotation.

Click and drag the node at the other end of the side on the edited shape to the point on the other end of the side on the stationary shape while holding `Shift`.

Release your mouse once it has snapped into place, to finalize the rotation.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=axO2M5asBbQ



Calibrating an Image to Known Dimensions

You can use **Two-Point Rotate / Scale** to straighten and calibrate an image to its real-life dimensions. This is especially useful when designing to fit existing, tangible items.

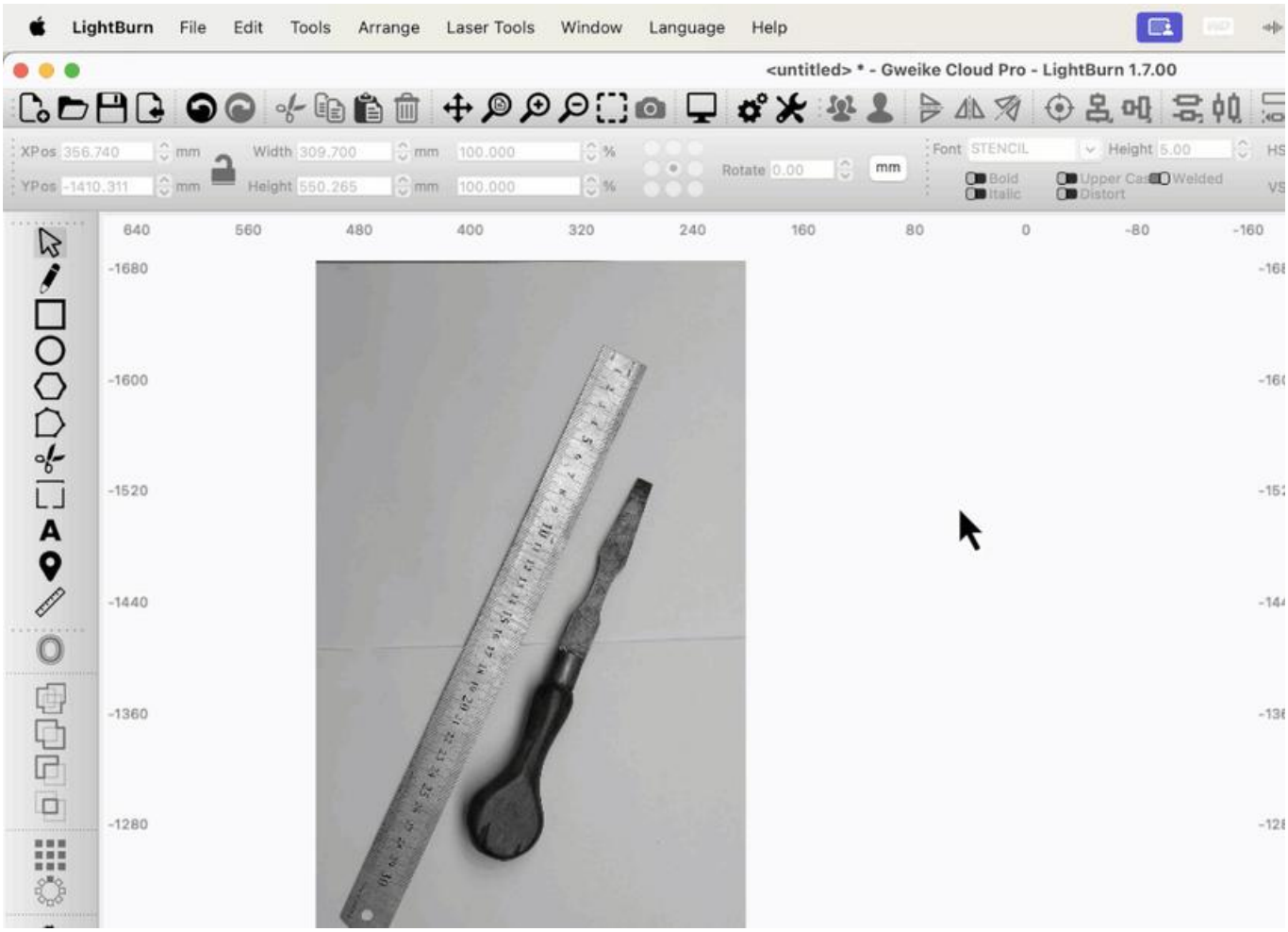
Take a photo or scan of your object with a ruler (or other object of known-size) next to it, and import that image into LightBurn.

Create a line the length of your ruler.

Align one end of the line to one end of the ruler.

Run the Two-Point Rotate / Scale tool on the image, with the first point being the aligned end, and the second being clicked from the other end of the ruler in the image, and dragged to the other end of the line.

After that, you can use the calibrated image in your project — for example, by Tracing it — and your project will be made to the correct, real-life scale.



Related Topics

- Align Tools
- Docking
- Snapping
- Transform Controls — Rotate

Create Rubber band Outline

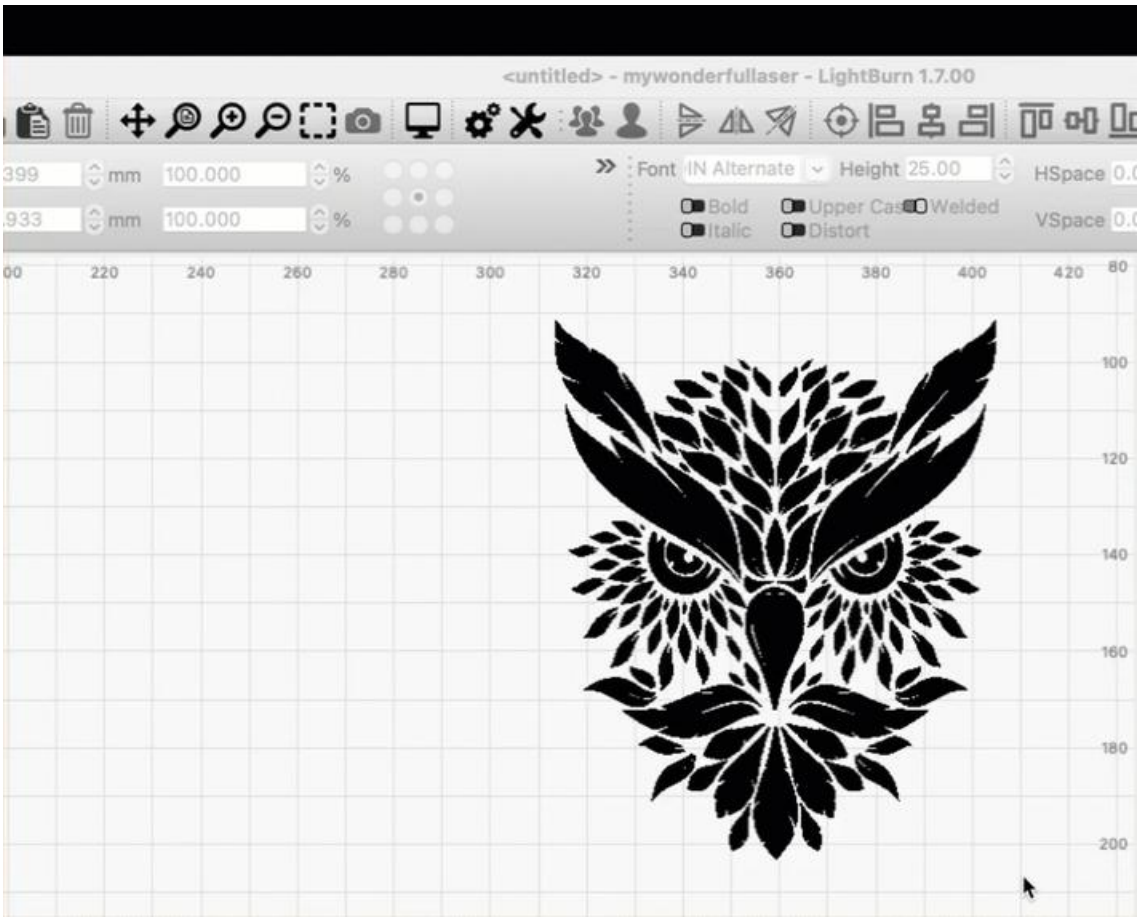
Create rubber-band outline automatically makes a new shape outlining the graphics in your current selection. The new shape is defined as the shortest possible path that can contain all graphics in your selection, as if a rubber band were stretched around them.

Use this tool to quickly create cut lines around images or complex graphics, or to negate Filled areas in vector engravings.

After making a selection, go to Arrange → **Create rubber-band outline from selection**.

The outline that's created will automatically be set to the last layer you selected from the Color Palette.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=R4nMnsy0v64



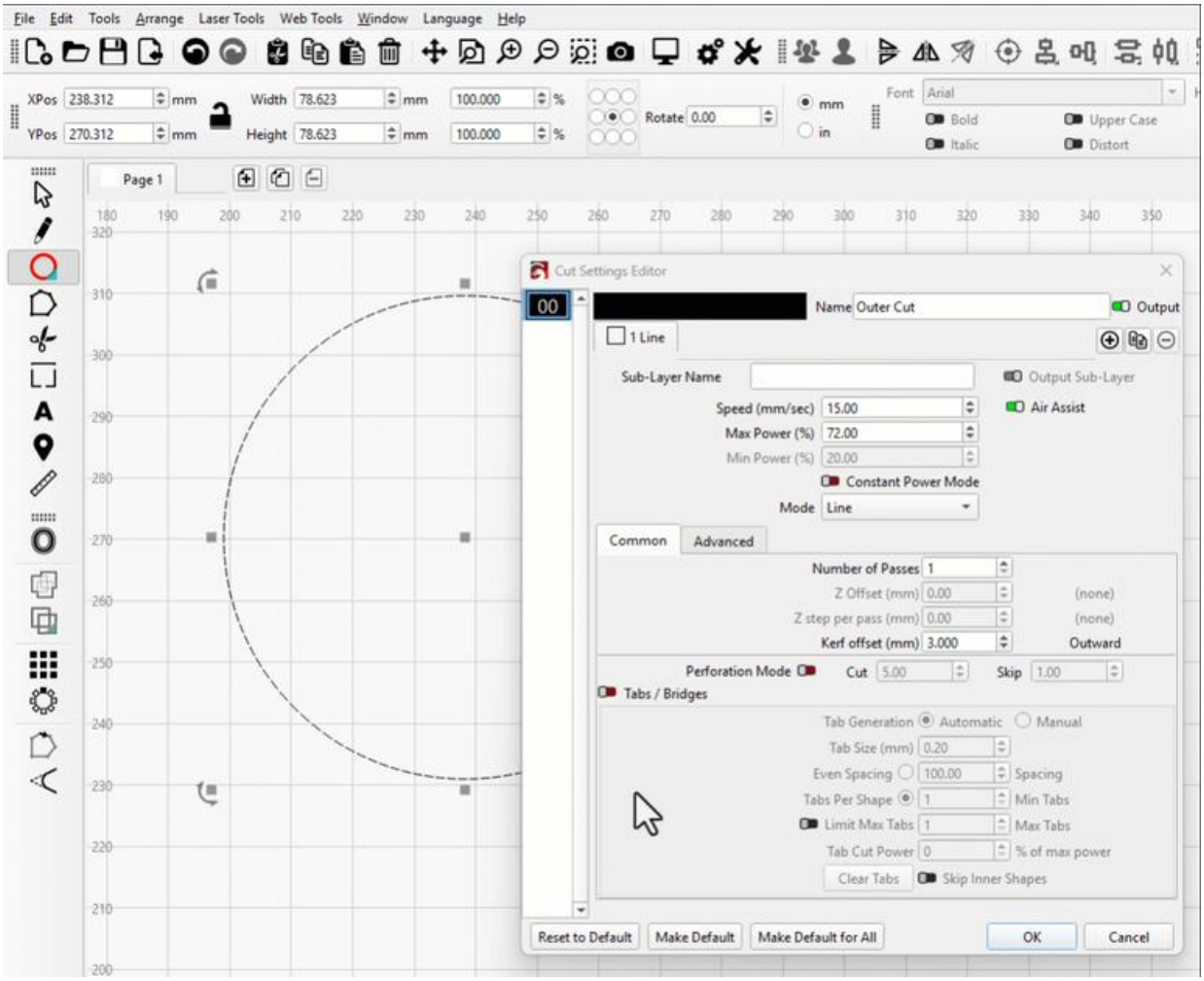
Related Topics

- Offset
- Selection Tools
- Draw Lines
- Edit Nodes

- Trace Image

Generate Path From Cut

Generate Path From Cut creates new paths based on your currently selected objects. The new paths include properties you've added through the Cut Settings Editor, such as Tabs, Kerf Offsets, and Perforations. The new paths generated are the cutting moves that are sent to the preview and the laser.



In this GIF, the circle has Perforation Mode and a large Kerf Offset (for visibility) applied, and a new path is created from it using Generate Path From Cut.

How to Use Generate Path From Cut

1. Prepare your shapes as you see fit.
2. Select the objects you want to generate paths from.
3. Go to Edit → **Generate Path From Cut**.
4. Your new paths are created without deleting the original objects.

Additional Information

Generate Path From Cut can be used to export designs for use with other programs while keeping the properties you set up in LightBurn, or anywhere you want to directly use or edit those properties. For instance:

- If you have a vinyl cutter or other similar CNC cutting tool, you could create an SVG that already includes tabs or perforated lines to import into that machine's control software.
- If you want to make alterations to a cut property, such as perforating part of a line, or altering part of a fill, you could use **Generate Path From Cut** and then use node editor or other design tools to edit the resulting shape.

Generate Path From Cut works on Fills and Images, but be cautious with particularly large images — the large number of paths produced may cause some lag for older computers with weaker hardware.

Convert to Cut

If you're looking for the **Convert to Cut** tool that was in LightBurn 2.1 or older, you're in the right place — it's been renamed to **Generate Path From Cut** in LightBurn 2.2.

Related Topics

- Generate Support Data
- Enable Debug Log
- Debug Drawing

Modifying and Combining

Flip and Mirror Shapes

Flip Vertical and **Flip Horizontal** mirror selected objects across their vertical or horizontal centers.

Mirror Across Line creates a mirrored duplicate of selected objects across a custom-selected line, at any angle.

Use these tools to correct orientation of graphics, or to speed up the design process when making symmetrical projects.

Flip Vertical and Horizontal

Flip Vertical and Horizontal are both available in the Arrange Menu, Arrange Toolbar, and by hotkey (see below).

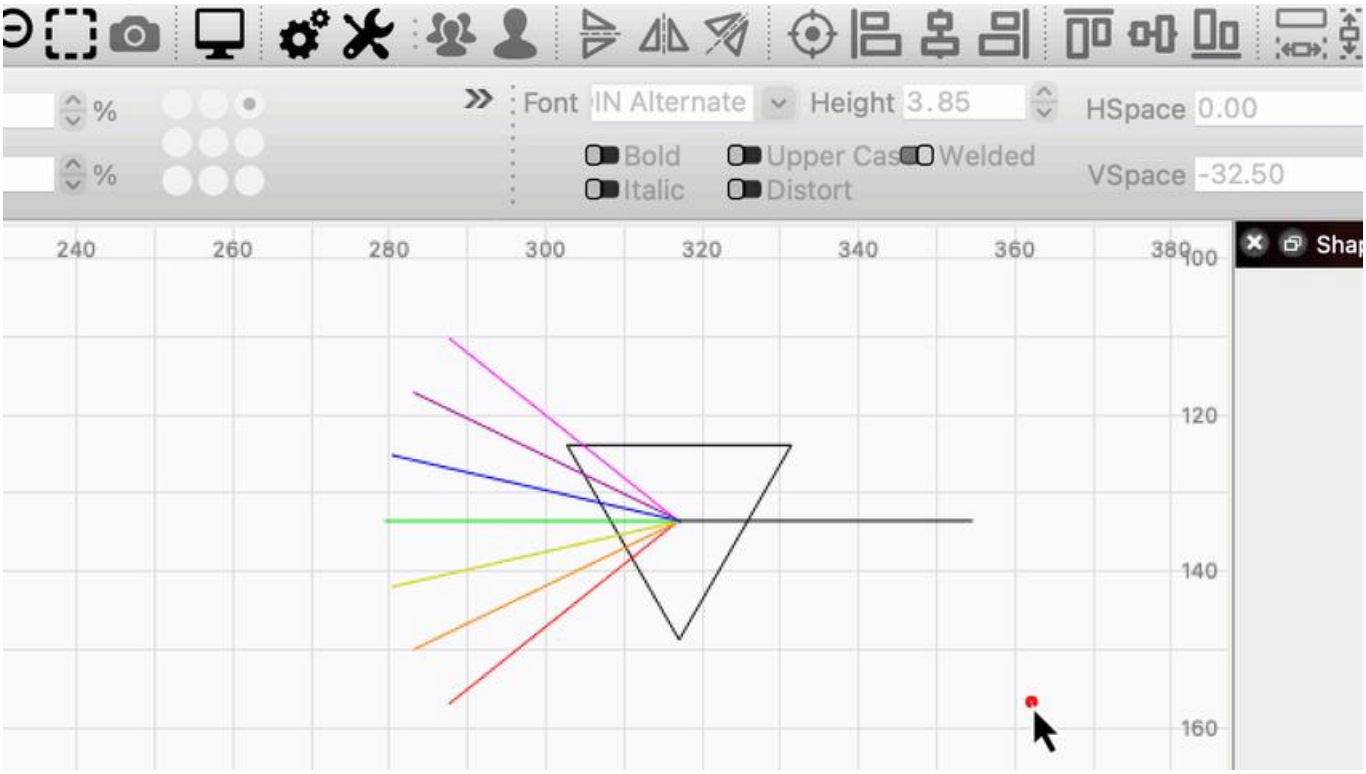


Flip Vertical mirrors all selected objects across the vertical center of the selection area. The vertical center is an imaginary line across the halfway point of a selection's full height.

Flip Horizontal mirrors all selected objects across the horizontal center of the selection area. The horizontal center is an imaginary line across the halfway point of a selection's full width.

Imagine the selection area as the smallest possible rectangle that can contain every object in your selection. The vertical and horizontal centers of a selection are the same as the vertical and horizontal centers of that imaginary rectangle.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=DDcRHTCxwwg



Mirror Across Line

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=0c1Q0vbtiTA

*Mirror Across Line *is available in the Arrange Menu, Arrange Toolbar, and by hotkey (see below).

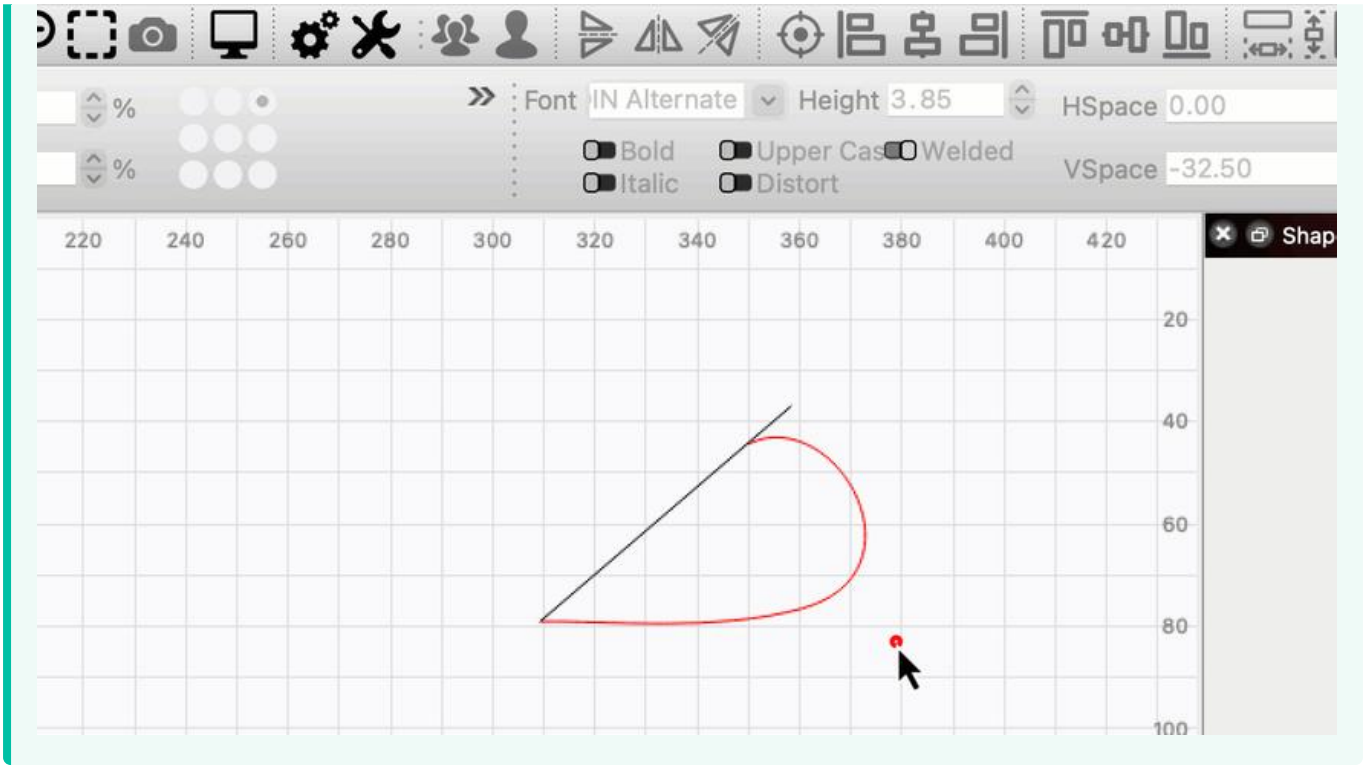


Mirror Across Line creates a mirrored duplicate of your selection across a custom-selected line.

- The line must be the last object added to your selection — in other words, you must first select the objects you'd like to Mirror, then select the line you'd like to Mirror them across.
- The line must be a straight line with only two points.

Use Mirror Across Line to create a symmetrical shape

Draw half the original shape, then mirror it to create the other half, like this:



Hotkeys

The keyboard shortcuts listed under **Edit Window in Focus** will only work when the Workspace — the area where you create and edit graphics — is in active focus.

What does it mean for a window to be in focus?

There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.

If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus. Selecting objects in the Workspace will also bring it into focus.

Action	Icon	Hotkey	Edit Window in Focus
Flip Vertical		`Ctrl` / `Cmd` + `V`	`V`
Flip Horizontal		`Ctrl` / `Cmd` + `H`	`H`
Mirror Across Line		`Ctrl` / `Cmd` + `Shift` + `M`	n/a

Related Topics

- Transform Controls
- Align
- Distribute
- Grid Array
- Circular Array
- Copy Along Path

Offset Shapes

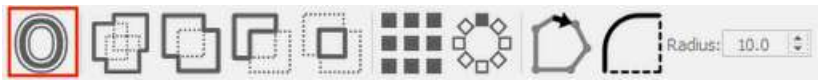
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=rEJ7ntlmEyU

The **Offset Shapes** tool creates a new shape by outlining the objects in your selection at a distance you specify, either inward, outward, or in both directions at once.

It's great for making borders around text, cut lines around engravings, inverting engravings, and creating a variety of patterns.

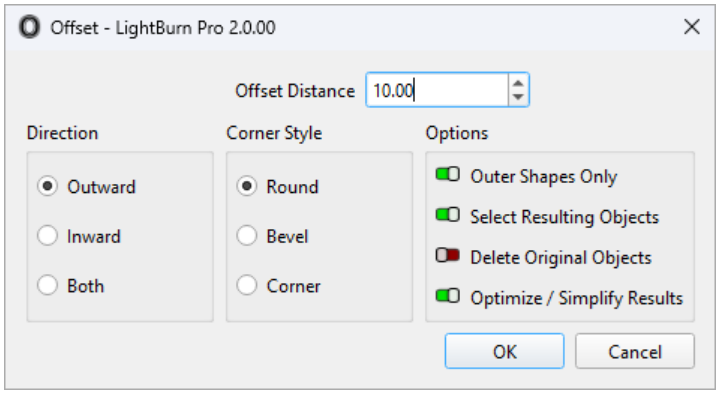
Offset Shapes works with any vector graphic, including paths, Primary Shapes, and Text Objects. You can offset multiple shapes, as well as Grouped shapes, all at the same time.

Using the Offset Shapes Tool



To open the **Offset Shapes** tool, first select all shapes you'd like to offset, then go to Tools → **Offset Shapes**, click the Offset button in the Creation Toolbar, or press `Alt`/`Option`+`O` to open the **Offset** dialog window, where you establish the settings that will determine the size, shape, and direction of your offset.

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Offset Distance

Offset Distance controls the amount the new shape will be offset away from the original. Adjust the distance by typing a value into the adjacent field, clicking the arrow buttons, hovering over it and scrolling with your mouse wheel, or dragging with two fingers, if you're using a trackpad.

Direction

Direction determines on which side of the original shapes the new offset shape will be created.

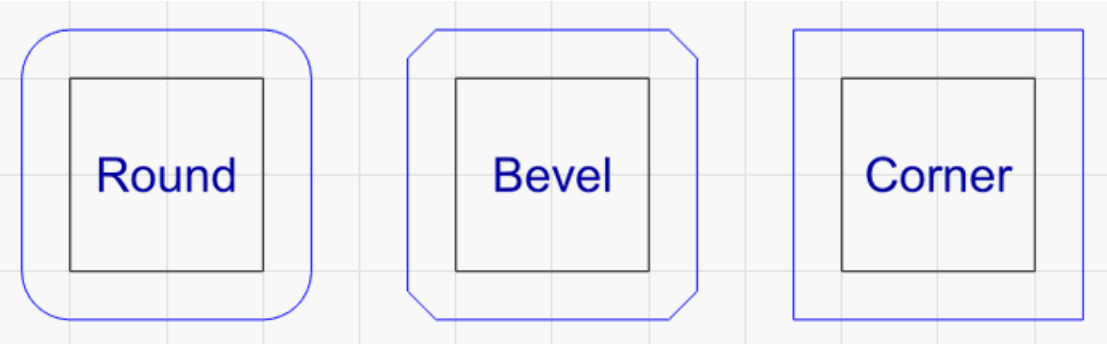
- **Outward** offsets away from the original shapes.
- **Inward** offsets in toward the center of the original shapes.

- **Both** creates two new offset shapes, one Outward and one Inward from the originals.

> **Note**
>
> You can only create Inward offsets for shapes that are closed, meaning they are complete, continuous loops whose start and end nodes are the same.

Corner Style

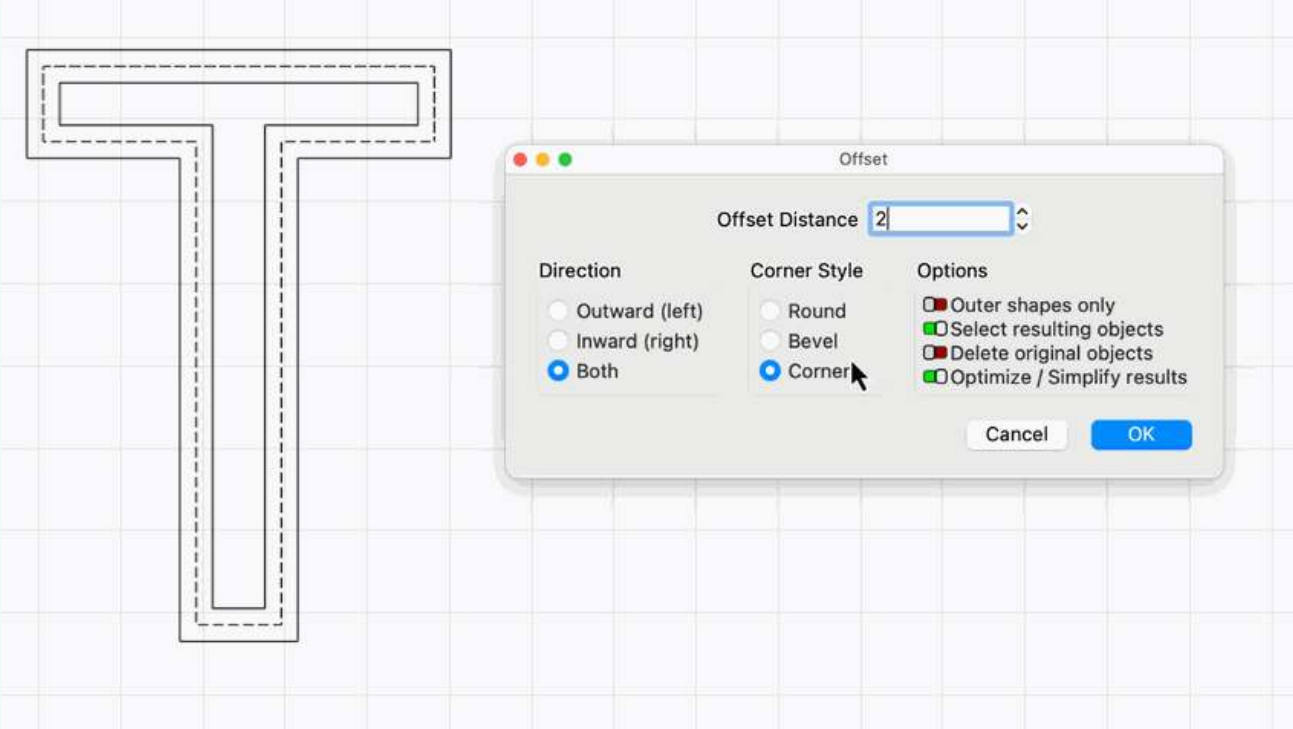
Corner Style controls the appearance of the corners in the offset shapes. You can choose **Round**, **Bevel**, or **Corner**.



Corner Style and Inward/Outward Corners

Corner Style affects outward- and inward-pointing corners differently — offset corners are only beveled or rounded when they're created in the direction a corner points. In the opposite direction, offset corners are always corners as well.

Notice the difference in the inward and outward offsets for the letter T below. The corners of the outer offset are beveled and rounded where the corners of the T point out, while the corners of the inner offset are beveled and rounded where the T's corners point in.



Options

Each switch beneath **Options** can be enabled or disabled. When enabled:

- **Outer shapes only**: will ignore any shapes contained within others when creating the offset, leaving just the outermost border.

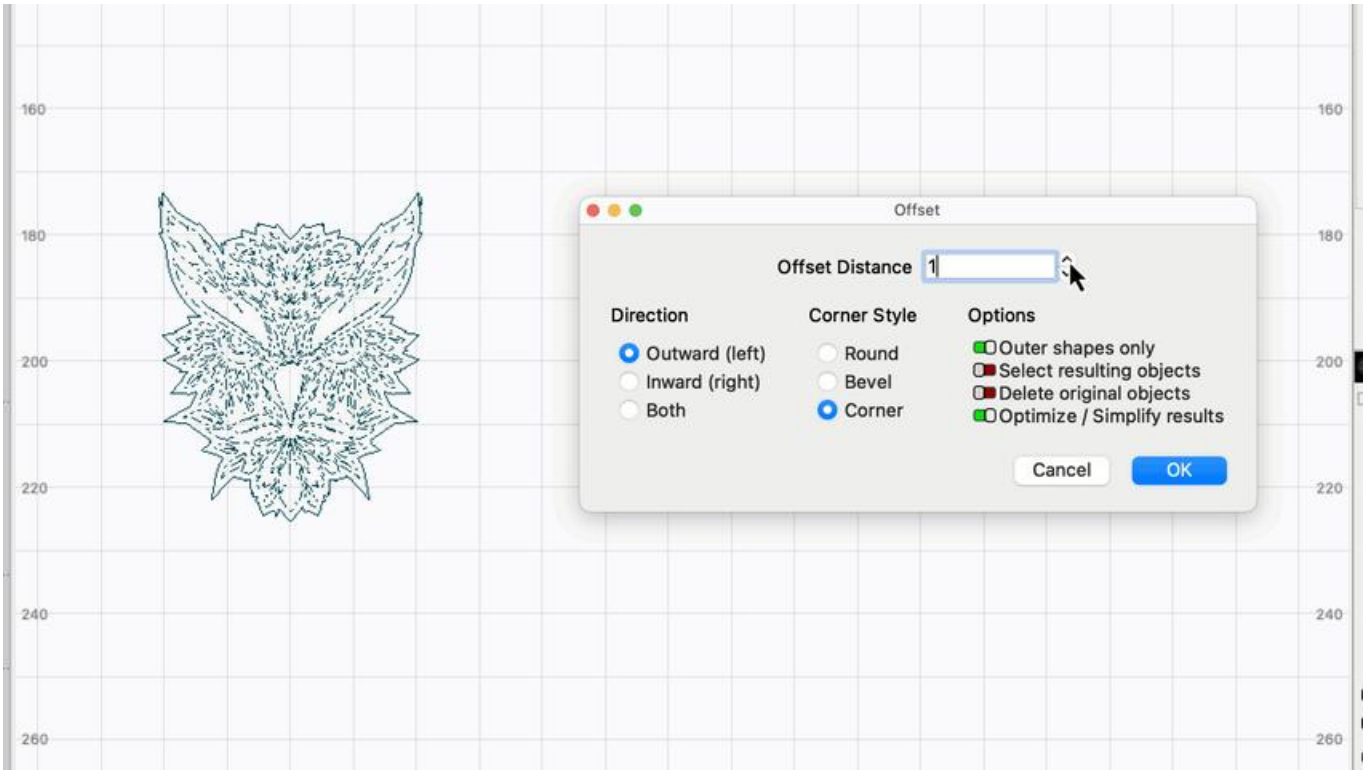
> **Use Offset Shapes to create a cut around a complex engraving**
>
> ▶ Watch the video on YouTube: https://www.youtube.com/watch?v=2xBJ8XGkp5g

- **Select resulting objects**: will select the new path you created upon completion of the offset.
- **Delete original objects**: will delete the objects that you had selected before opening this dialog, replacing them with the new offset.
- **Optimize / Simplify results**: will reduce the number of nodes in the result, great for reducing file size of complex objects such as image traces.

Previewing and Completing the Offset

When you open the **Offset** dialog window, the offset that will be created as the result of your settings will appear in your Workspace, and will live-adjust to reflect settings changes you make.

Click **OK** to create the offset, or **Cancel** to discard the settings and exit the window.



Tip

To repeat an offset with the last settings you used, select the shapes you wish to offset, then hold `Ctrl`/`Cmd` while clicking the **Offset** button. It will perform the offset operation using the previous settings, without bringing up the dialog.

Tips

- **Offset Shapes** can sometimes fix odd or broken vectors by running the offset with a **Distance** of 0.
- You can use Offset to invert/negate an engraving:

Use Offset Shapes to invert an engraving

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=EAlw371Ey0M

Related Topics

- Create Rubber-Band Outline
- Draw Lines
- Edit Nodes

Boolean Tools

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=rTYixcObMuU

Use the **Boolean** tools to combine existing shapes (or Grouped shapes) into new or more complex shapes. There are three Boolean options, along with one very similar option, **Weld**.

Using the Boolean Tools

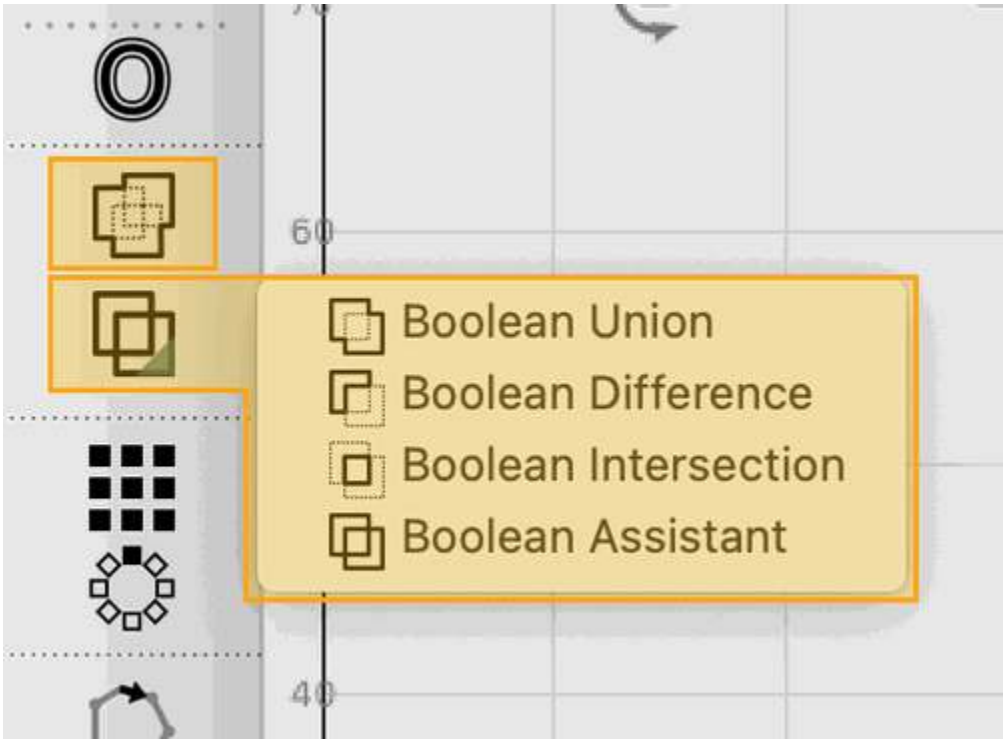
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=51cXqmNHgeI

All of the **Boolean** tools and **Weld** are available in the Modifiers Toolbar, Tools Menu, or by keyboard shortcut.

The **Weld** tool sits above the **Boolean** tools in the Modifiers Toolbar.

All Boolean tools are stacked into a single button.

- The Boolean Assistant is the default option, and clicking once will open it.
- You can click the arrow in the lower right corner of the icon, or click and hold anywhere on the icon, to open a submenu with the individual tools.
- Selecting an option will perform that operation, and will also make it the button's active operation, meaning you can subsequently click the button once to perform that operation again.



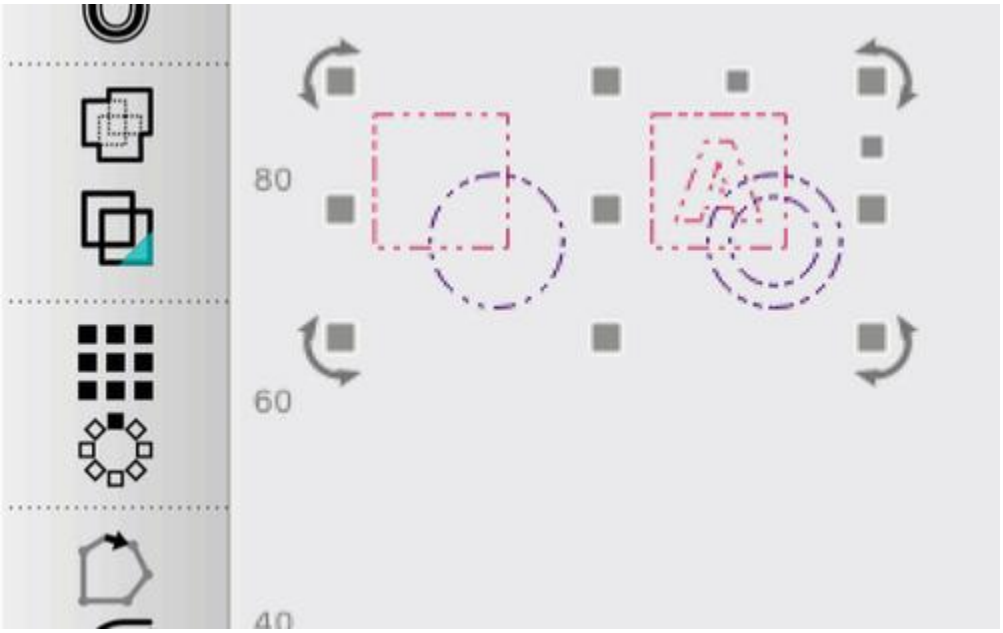
There are a few important things to keep in mind when using the Boolean tools and Weld:

- All selected shapes must be closed shapes, meaning they are complete, continuous loops whose start and end points are the same. If there are open shapes in your selection, the tools will be grayed out and unavailable.
- The Boolean tools work with vector objects only. If there are any images in your selection, the tools will be grayed out and unavailable.
- To use any of the Boolean tools, you must have exactly two shapes, two Groups of shapes, or one shape and one Group selected. If you have more than two shapes or Groups selected, the Boolean tools will be grayed out and unavailable.
- You can use Weld to combine more than two shapes at once.
- Grouping multiple shapes together tells LightBurn to treat them as a single shape. For example, Group an outer circle with an inner circle to treat it as a donut shape rather than two separate circles. Grouping shapes in this way can sometimes yield different results than you'll see with the same objects when they're not Grouped.

Boolean Union

 `Alt`/`Option`+`+`

Boolean Union merges two shapes into one.

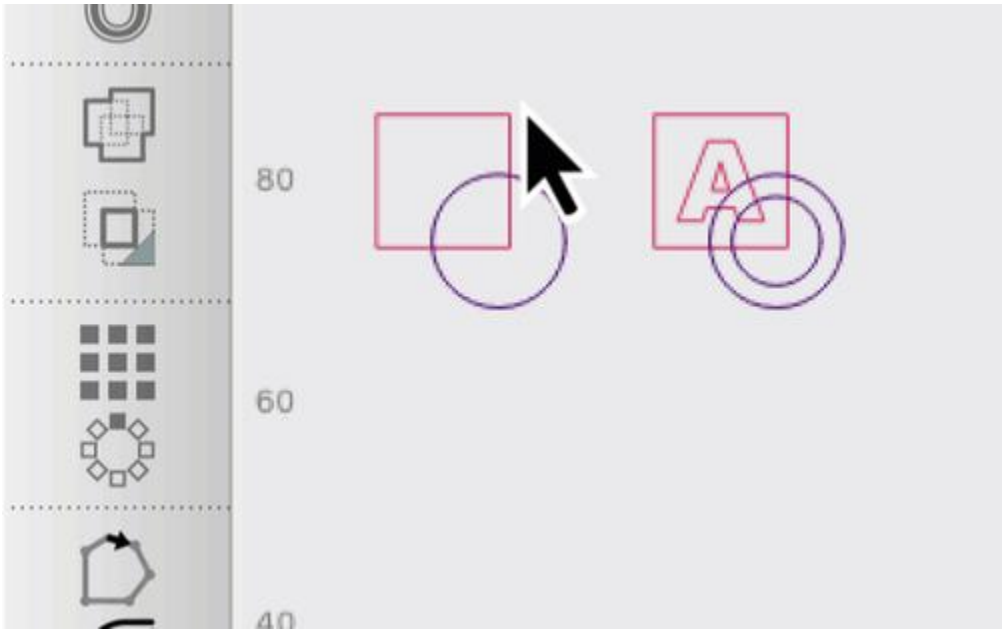


Boolean Subtract (Boolean Difference)

 `Alt`/`Option`+`-`

Boolean Subtract removes one shape from another.

The second shape you add to your selection is subtracted from the first, and deleted.



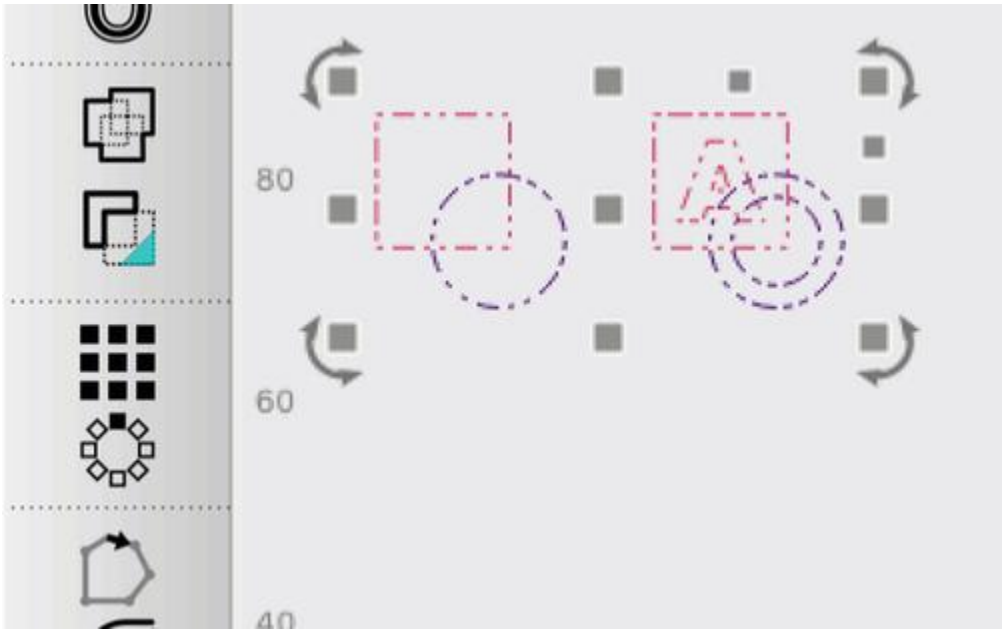
If you select the shapes in the incorrect order, and subtract the wrong shape from the other, Undo the action, then use Boolean Subtract again — the alternate shape will be removed instead.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=8LS1Vn3J_qQ

Boolean Intersection

 `Alt`/`Option`+`*`

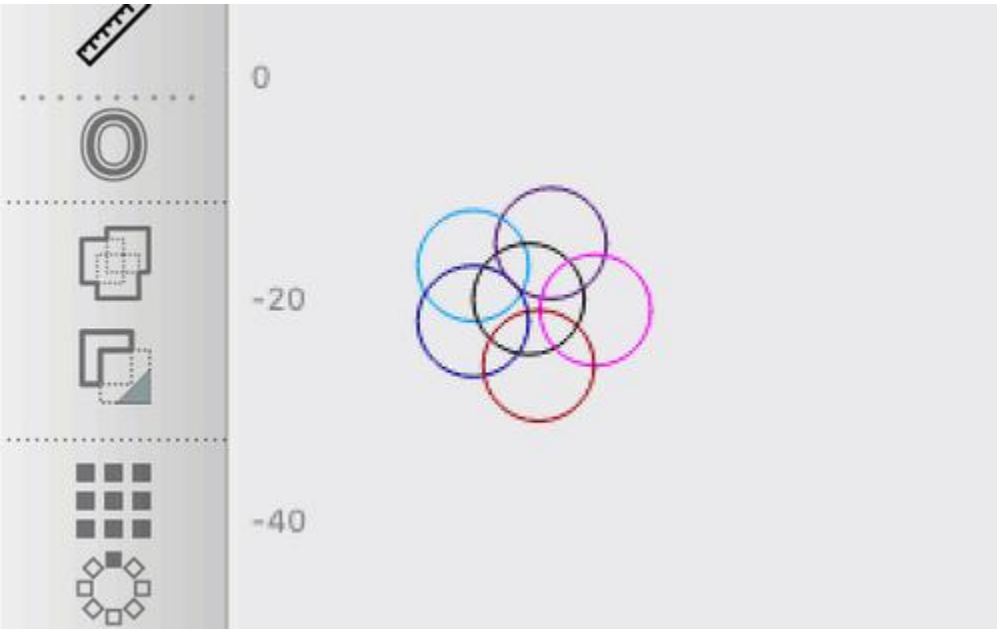
Boolean Intersection creates a new shape out of the overlapping area of two shapes.



Weld

Ctrl / Cmd + W

Weld works similarly to Boolean Union, but instead of being limited to combining only two shapes or two Groups of shapes, you can use Weld with any number of objects.



Boolean Assistant

Ctrl / Cmd + B

> **Note**
>
> If you have Beginner Mode enabled, all three Boolean tools will be replaced in the Modifiers Toolbar with a single icon that opens the Boolean Assistant.

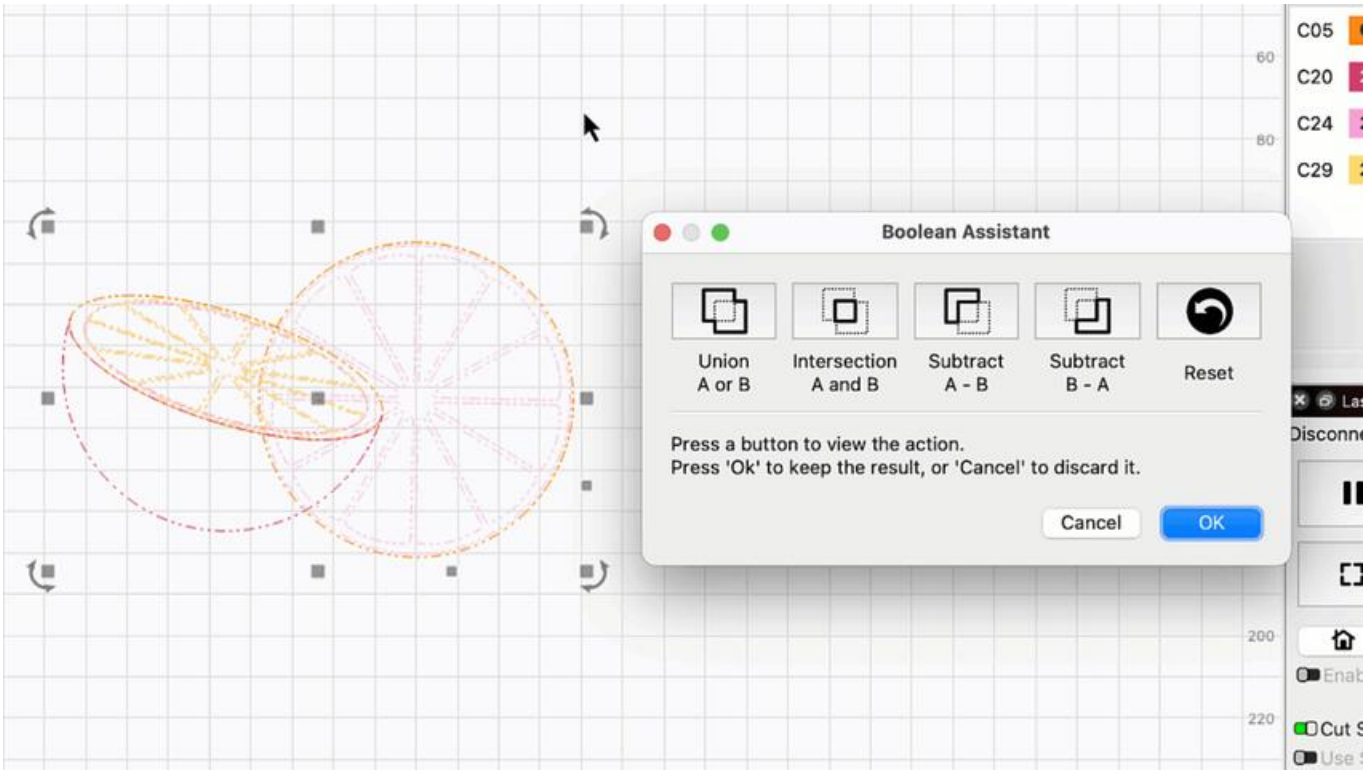
The **Boolean Assistant** is a dialog window that shows a preview of what each of the **Boolean** operations will do to your selected shapes, and gives you the option to click **OK** to confirm the change, or **Cancel** to discard it and close the Boolean Assistant.

To open the Boolean Assistant, go to Tools → **Boolean Assistant**.

Click an operation to see a preview of the result in your Workspace. If the shapes are not too complex, hovering over an operation without clicking will also show what the result will be.

Click or hover over **Reset** to return the shapes to their original appearance, while keeping the Boolean Assistant open.

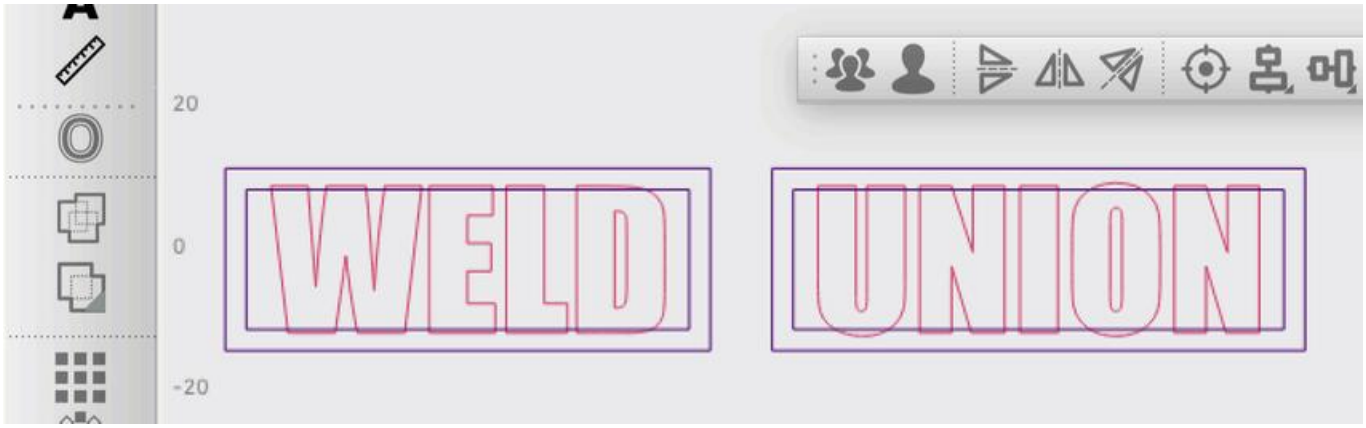
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=3O-g5O5QB_k



Weld vs. Boolean Union

In cases where some shapes are fully contained within other outer shapes, using **Weld** won't yield the result you're looking for, but Grouping related shapes together and using **Boolean Union** will.

If you Weld something and some or all of the middle content disappears — for example if you are trying to Weld text inside a ring, and the text disappears — use Boolean Union after making two Groups: one of the border pieces, and one of the middle content.



Hotkeys

Action	Icon	Hotkey
Boolean Assistant		Ctrl / Cmd + B
Boolean Union		Alt / Option + +
Boolean Subtract		Alt / Option + -
Boolean Intersection		Alt / Option + *
Weld		Ctrl / Cmd + W

Related Topics

- Cut Shapes
- Trim Shapes
- Grouping
- Masking

Cut Shapes

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=aHzZA_JKeXM

The **Cut Shapes** tool divides selected vector objects into separate shapes, using a closed shape as the cutting tool. The last shape you add to your selection is used as the cutting tool, and all other selected shapes are cut at the locations they intersect with the cutting tool.

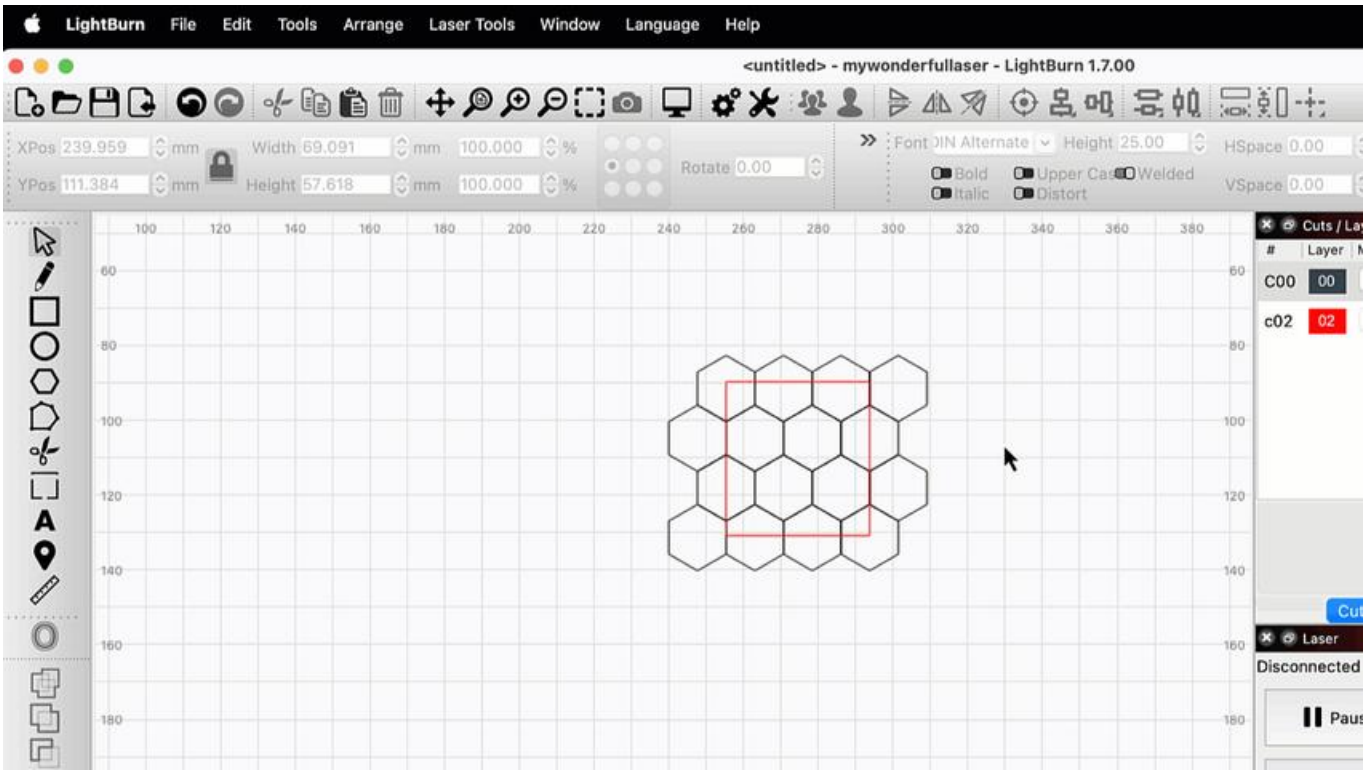
Using Cut Shapes

To use **Cut Shapes**, first select the shapes you wish to cut, then add the cutting tool shape to your selection.

Go to Tools → **Cut Shapes** or press `Alt`/`Option`+`Shift`+`C` to activate the tool.

Use Cut Shapes to cut a project into smaller sections that can be run separately, allowing you to create projects that are larger than your laser's work.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=dQmyIQ9OD4g



A few things to always keep in mind when selecting and using a cutting shape:

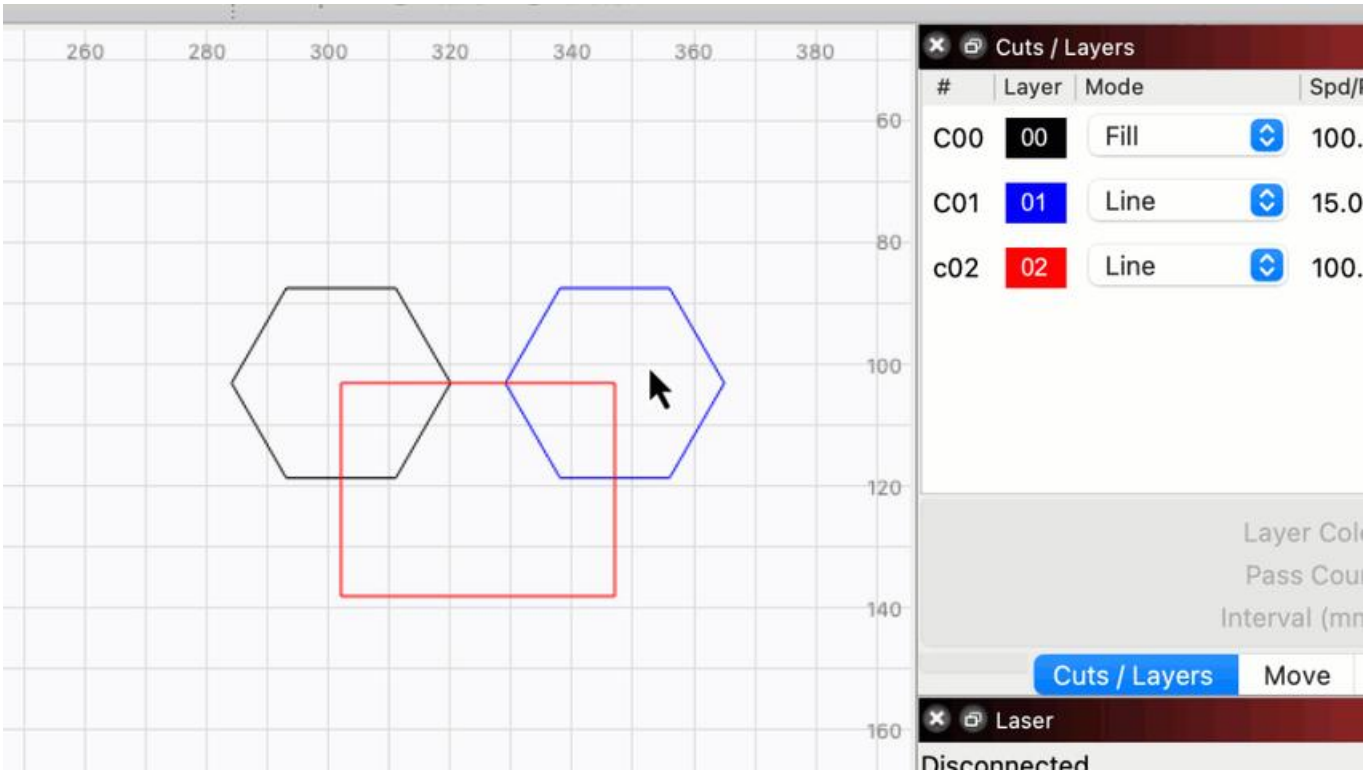
- The last shape you add to your selection is used as the cutting tool.
- The cutting tool must always be a closed shape, meaning it is a complete, continuous loop whose start and end points are the same.

- The cutting shape must not be Grouped with any other shapes. Ungroup any shape you'd like to use as a cutting shape.
- The cutting tool is deleted in the process — if you'd like the shape you're using as a cutting tool to remain in your design after using it with Cut Shapes, first Duplicate it, and then select just one copy to use with Cut Shapes.

And note the following about the resulting shapes:

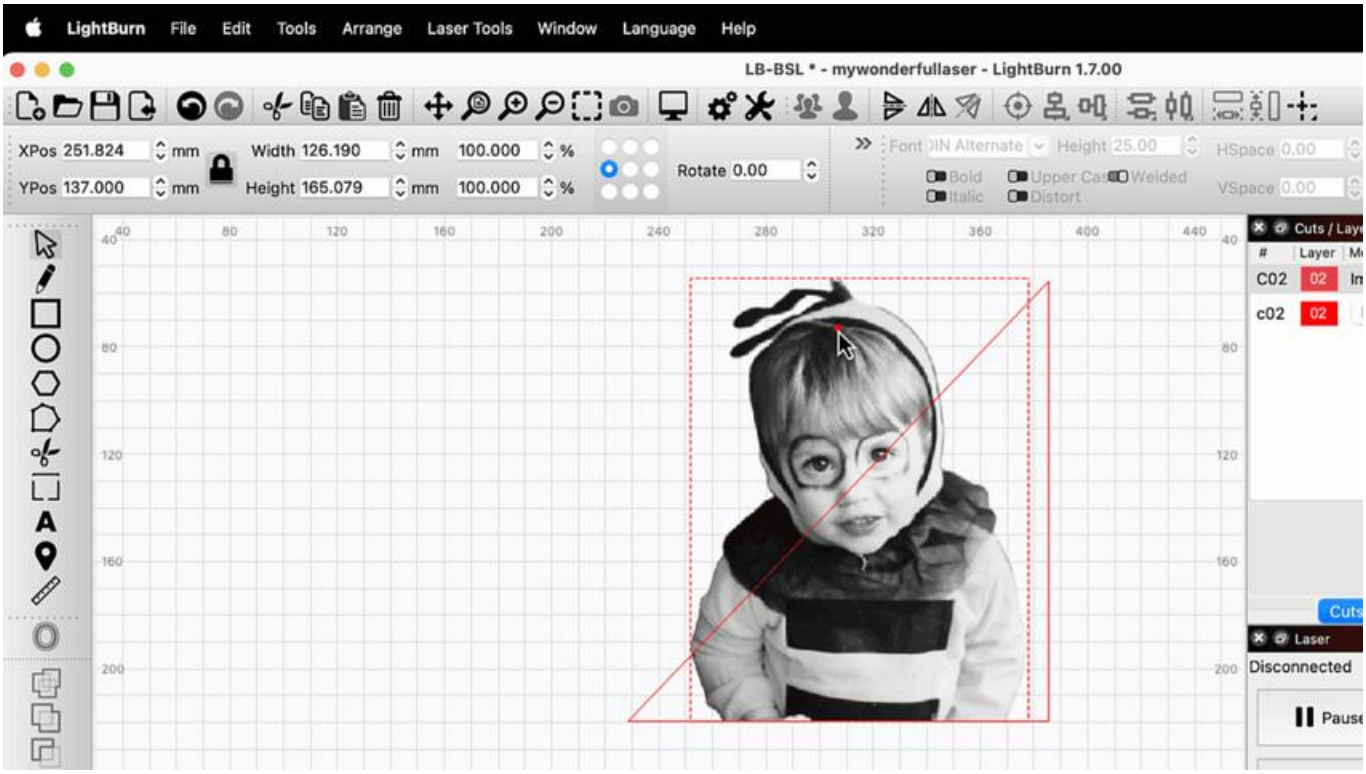
- The resulting shapes are automatically sorted into two Groups — one Group for shapes that were inside the cutting tool, and another for shapes that were outside it. You'll need to Ungroup the shapes to manipulate them individually.
- The resulting shapes are automatically closed if they were set to a Fill or Offset Fill layer, or left open if they were set to a Line or Tool layer.

  In other words, for shapes set to a Fill or Offset Fill layer, LightBurn will automatically add a new line on all new shapes, connecting the gap where the cutting shape severed the originals. If they are set to a Line or Tool layer, the gap will remain.



Cut Shapes and Images

If you use **Cut Shapes** with an image, LightBurn will automatically create two masked copies of the images — one copy will be masked to show the part of the image that was inside the cutting shape, and the other copy will be masked to show the part that was outside. The cutting shape will also be duplicated, and each copy will be reassigned to a Tool layer.



Troubleshooting

If **Cut Shapes** is grayed out in the Tools Menu, it means the object you selected last is not a valid shape to use as a cutting tool.

- The cutting tool must be a closed shape, meaning it is a complete, continuous loop whose start and end points are the same.
- The cutting tool must not be Grouped with any other objects.

Related Topics

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=rTYixcObMuU

- Trim Shapes
- Boolean Tools
- Apply Mask to Image
- Print and Cut — Cutting a Project Larger Than Your Laser's Workspace
- Open vs. Closed Shapes

Grid Array

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=WhjJvWE7T1k

The **Grid Array** tool creates copies of an object (or objects) in regularly spaced rows and columns, and includes options to adjust spacing, mirror the shapes, randomize orientation, and more.

Use the Grid Array tool to:

- Duplicate designs for batch production
- Maximize material utilization
- Create designs that require precise spacing
- Make interesting patterns

Use Grid Array to create seamless tiling patterns

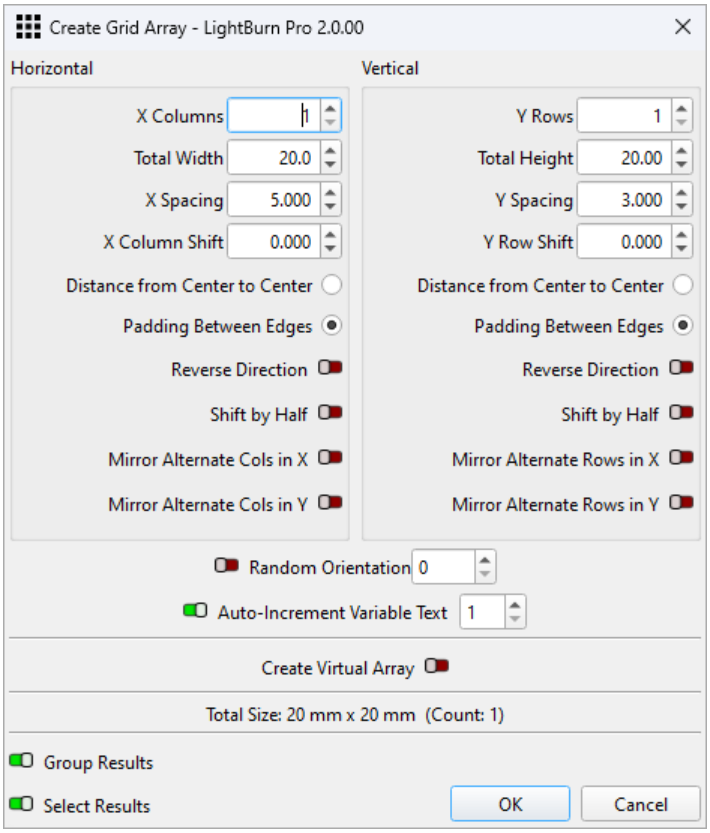
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=-mduelcFtQ8

Grid Array Settings

To open the **Create Grid Array** window, select all objects you'd like to array, then go to Arrange → **Grid / Array**, or click the Grid Array button in the Modifiers Toolbar.



Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



X Columns and Y Rows

Specify the number of copies to make in the horizontal (X) and vertical (Y) dimensions.

Total Width and Total Height

Alternative options for specifying the number of copies to make in each dimension. Set the values in these fields to match the size you'd like the grid to occupy, and LightBurn will set X Columns and Y Rows to whatever is necessary to fill the specified size without going over, taking into account the other settings below.

If you do not adjust these fields directly, they will automatically change to reflect the width and height of your grid, as determined by your other settings.

X Spacing and Y Spacing

Determine the distance between each copy, in the horizontal and vertical dimensions. For each dimension, you can select to space based on **Distance from center to center** of objects or **Padding between edges** of objects (see below).

Distance From Center to Center

When selected, **X** and **Y Spacing** will set the distance between the centers of each copy in the grid.

Padding Between Edges

When selected, **X** and **Y Spacing** will set the distance between sides of each copy in the grid.

X Column Shift and Y Row Shift

Offset every other column or row by the specified amount, shifting alternate columns or rows left, right, up, or down.

Reverse Direction

Changes the direction in which copies are produced. Can be enabled individually for each dimension.

Shift by half

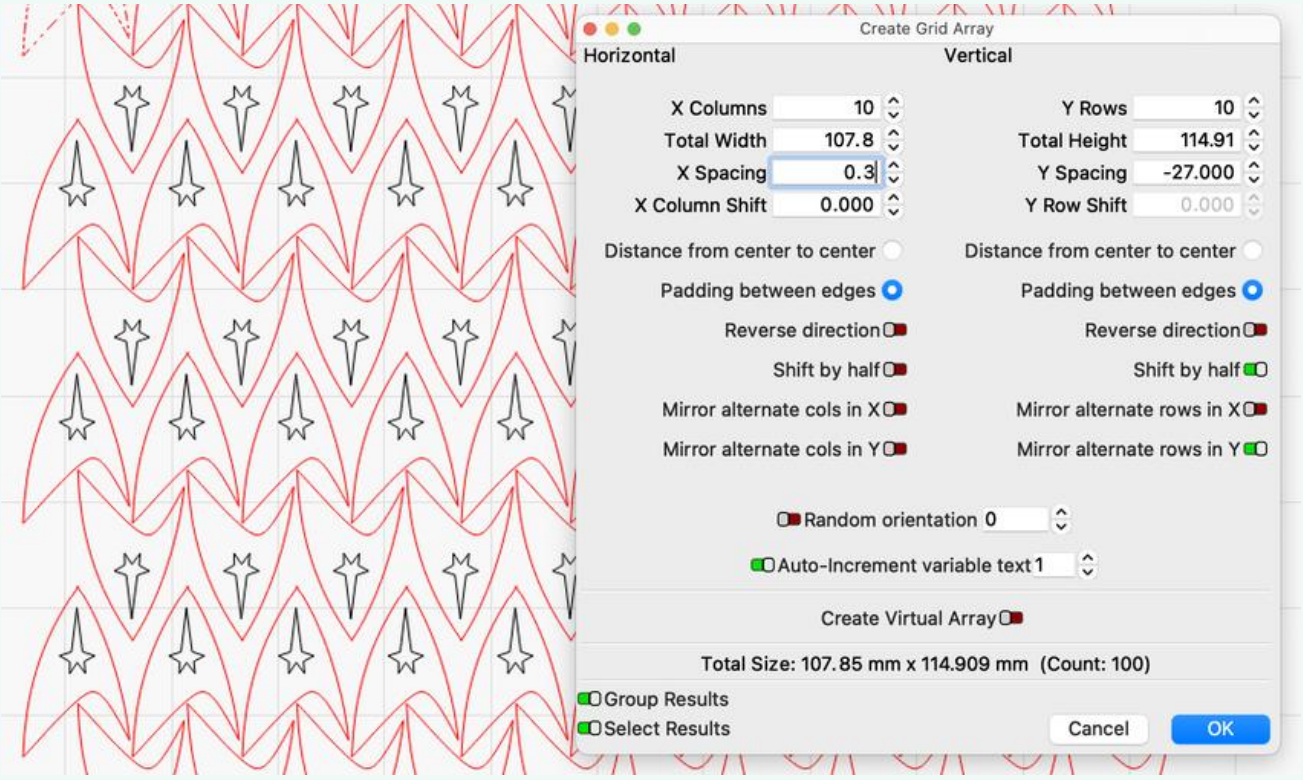
Offsets alternate columns or rows by half the total width or height of the original object.

Mirror Alternate Columns in X or Y and Mirror Alternate Rows in X or Y

Flip copies of the original object(s) in the selected dimension, for alternate columns or rows.

Maximize material utilization with Grid Array

Use the X/Y Spacing, X/Y Shift, and Mirror options individually or in combination with one another to efficiently lay out objects for maximum material utilization.



Random Orientation

Randomly rotates each copy of the original object(s). Use the field next to the Random orientation switch to enter a seed number that is used as the starting point of the randomly generated orientation. Re-using the same seed number will duplicate the same randomly generated orientation later on.

Auto-Increment Variable Text

Applies an **Offset** to text in each copy created in the array, automatically advancing output if you are using **Merge/CSV** or **Serial Number** formatting. The field next to the switch determines the amount by which each duplicates' Offset will advance.

See our documentation on using Variable Text for more information.

Use Grid Array to automatically increment Variable Text

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=5KONOAPuXV8

Create Virtual Array

Makes an array of synced clones of the original object(s), instead of independent copies. See below for more information.

Total Size and Count

Display the resulting array's size in and X and Y, and total number of copies, including the original.

Group Results

When enabled, automatically Groups all objects in the array after it's created.

Select Results

When enabled, all objects in the array are automatically added to your selection when it's created.

OK and Cancel

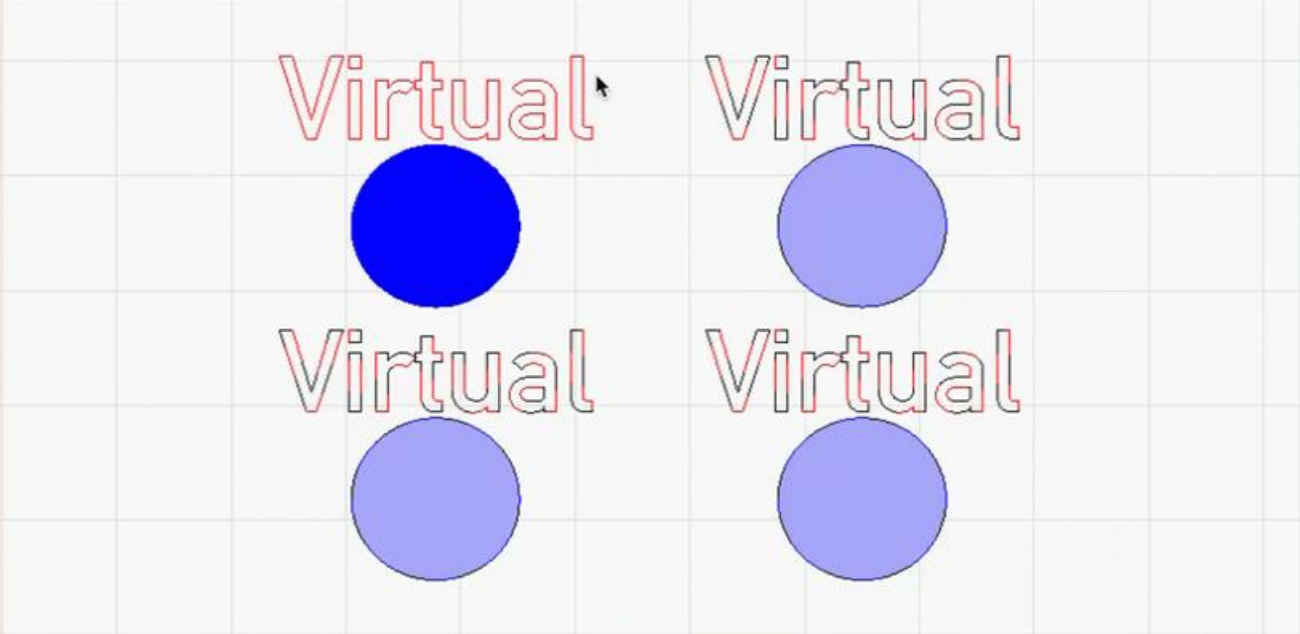
Press **OK** to confirm your settings and create the array, or **Cancel** to discard settings and not create the array.

Virtual Arrays

Enabling the **Create Virtual Array** option in the **Create Grid Array** window generates an array of synced clones, instead of making copies of the original objects.

The clones render with dashed outlines and muted fill colors, to indicate that they are virtual copies.

Any changes made to the original copy in the array will be applied to the clones, but the clones themselves are not selectable or adjustable.



Virtual Array Quick Tips

The Power of Virtual Arrays

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=2YeA8xxZ0Kk

Make Symmetrical Edits With Virtual Arrays

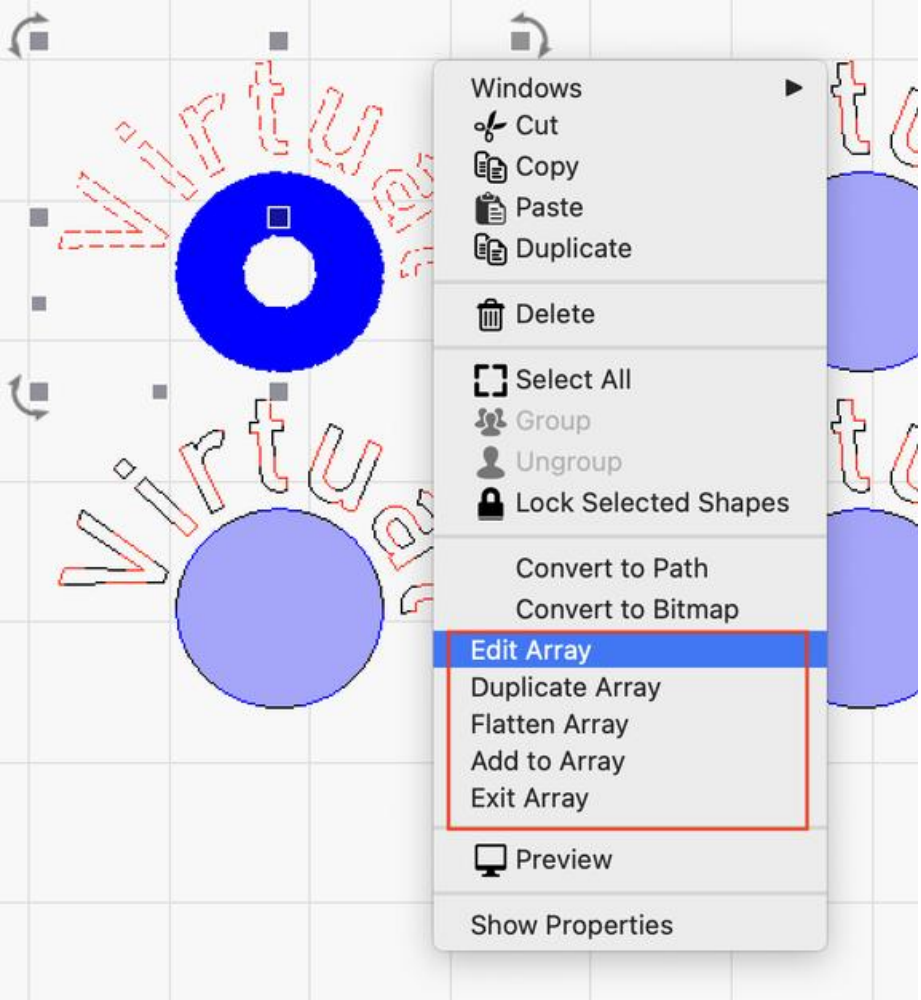
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=ZQdicJg8s4k

Modifying Virtual Arrays

Select and right-click any original object that is part of a **Virtual Array** to open a context menu where you'll see several additional actions you can perform after creating the array.

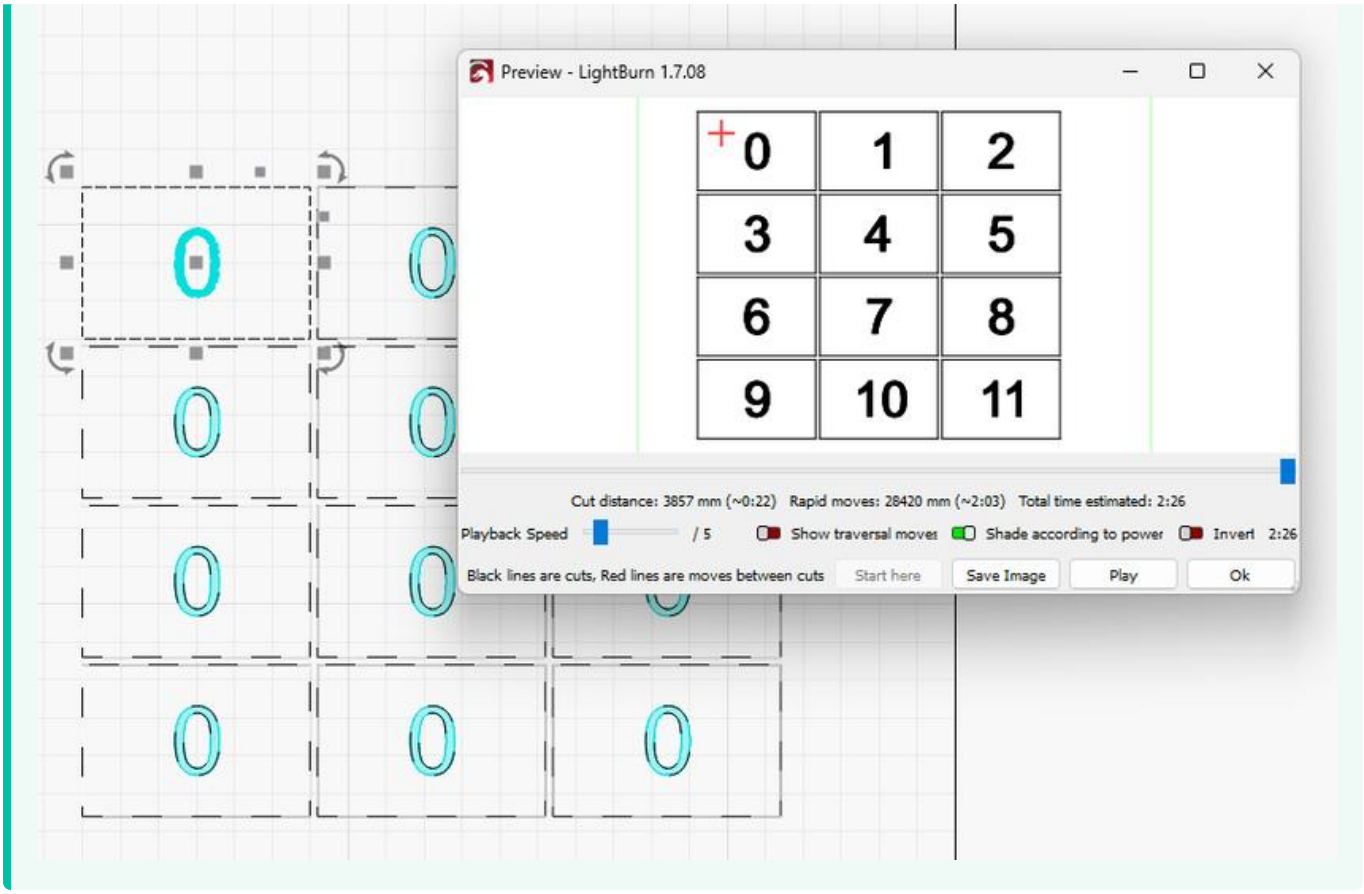
You do not have to select all original objects used in the array.

- **Edit Array** displays the **Create Grid Array** dialog again, allowing you to modify any of the parameters, and even convert the array into a normal array by unchecking **Create Virtual Array**.
- **Duplicate Array** duplicates the entire Virtual Array and all of the objects in it.
- **Flatten Array** converts the Virtual Array into a standard array where all objects are separate copies and no longer synced to the original objects.
- **Remove from Array** remove the currently selected object(s) from the virtual array.
- **Add to Array** adds any currently selected object not in the virtual array to the array. Note: to see this option, you must select at least one object that is currently in a virtual array and one object that is not.
- **Exit Array** removes the Virtual Array, without deleting the original copies.



Using Variable Text in Virtual Arrays

When using Variable Text in a **Virtual Array**, the Test button will not show the final output of the Variable Text in LightBurn's workspace. To see an accurate preview, you'll need to use the Preview window.



Related Topics

- Variable Text
- Circular Array
- Copy Along Path

Circular Array

Circular Array arranges multiple copies of an object — or multiple objects — in a radial pattern around either a point in space, or a central object.

Use the Circular Array Tool to:

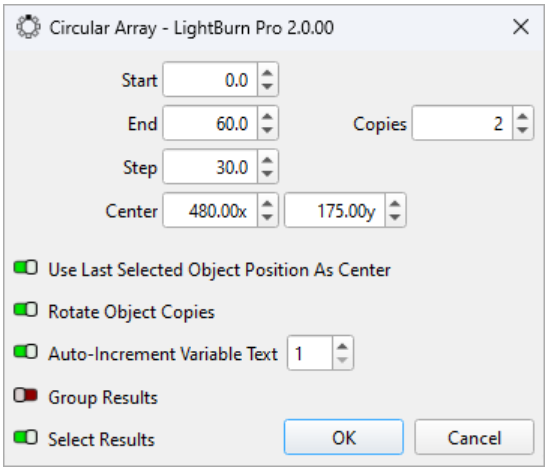
- Create organic radial patterns such as mandalas and flowers by duplicating simple shapes
- Create even spacing around a circle, such as numerals and markings on a clock face
- Make geometric radiating designs such as starburts or cogs
- Create decorative round frames or borders
- Distribute mounting holes in a circular formation

Circular Array Settings

To open the **Circular Array** window, select all objects you'd like to duplicate, then go to Arrange → **Circular Array**, or click the Circular Array button in the Modifiers Toolbar.



Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



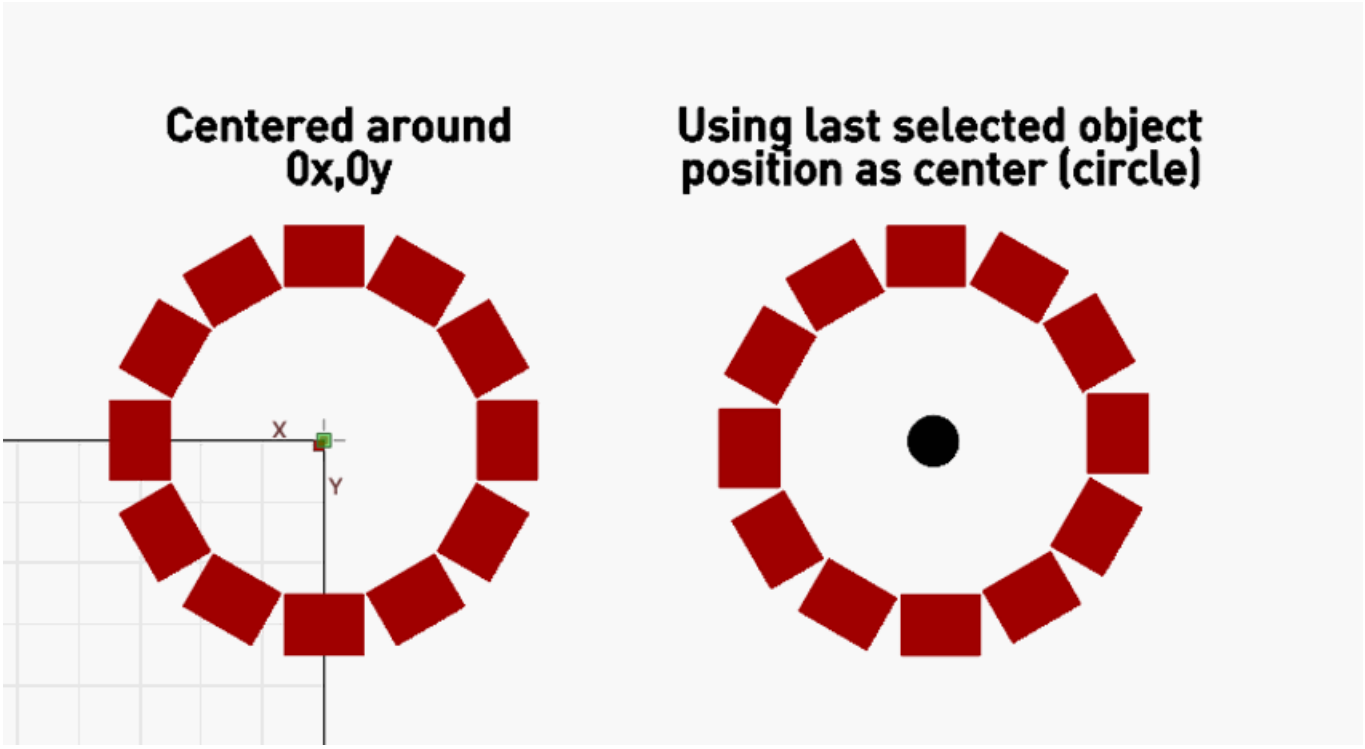
Copies

Enter the number of items you wish to appear in the final array into the **Copies** field.

Center of Rotation

There are two ways to set the center of rotation for your array:

- Enter the x and y coordinates of a spot in your Workspace into the **Center** fields.
- Turn on **Use last selected object position as center** to array the copies around the center of the last object you added to your selection.



Start and End

An array doesn't have to span the whole 360-degree rotation, you can limit the duplications to only part of the circle using the **Start** and **End** fields.

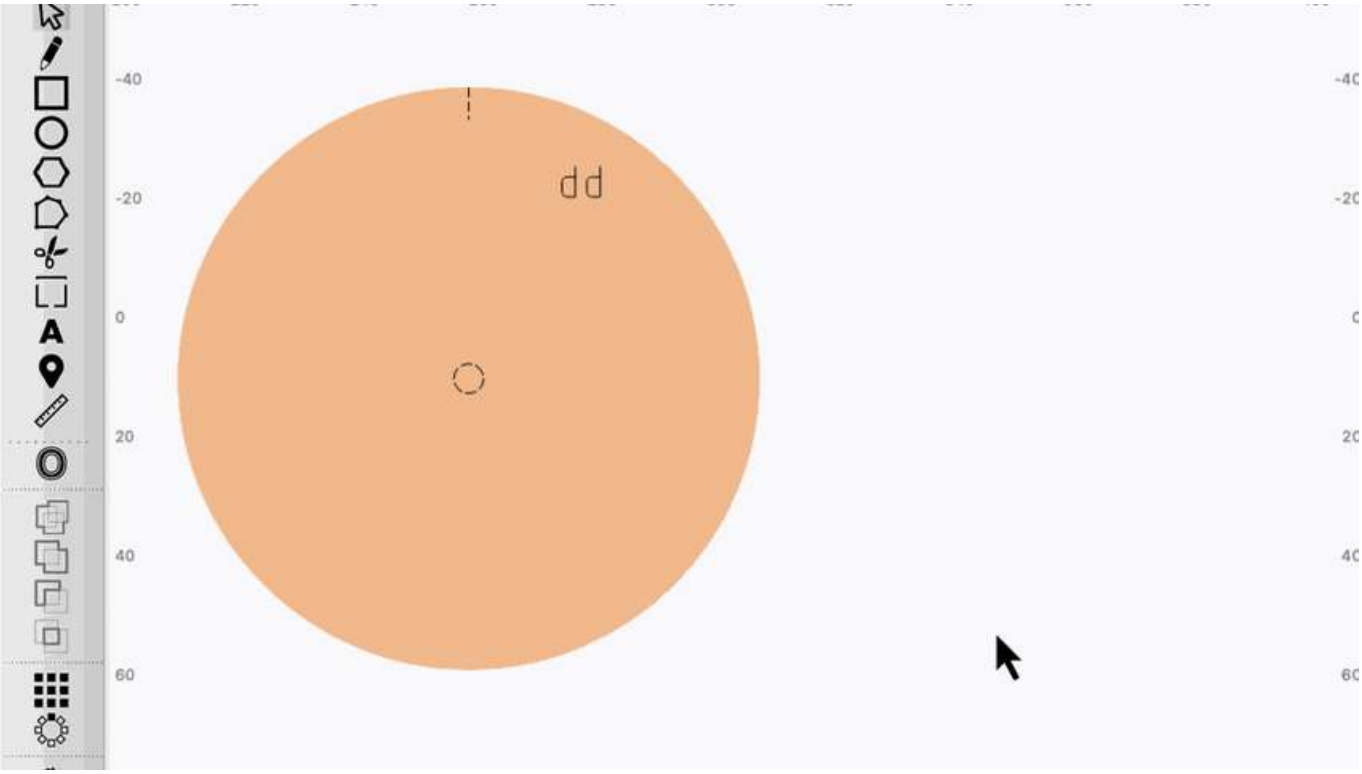


Step

You can specify the angle between the copies in the **Step** field. This value is mathematically tied to the **End** and **Copies** fields.

Rotate Object Copies

Enable **Rotate object copies** to turn copies so that the same part of each duplicate faces the center, like the tickmarks on a stopwatch. Disable it to keep the copies in their original rotation, like the numerals of a clock that remain upright for readability.



Auto-Increment Variable Text

Auto-Increment variable text works in conjunction with Variable Text, increasing the Offset value in the Text Options Toolbar value by the specified amount for each copy. Using Offsets allows you to use the same formatting expression to output different data.

Group Results

Enable **Group Results** to have the output automatically Grouped upon completion of the array.

Select Results

Enable **Select Results** to have output automatically selected upon completion of the array.

OK and Cancel

Press **OK** to confirm your settings and create the array, or **Cancel** to discard settings and not create the array.

Related Topics

- Grid Array
- Variable Text
- Copy Along Path

Copy Along Path

Use **Copy Along Path** to create repeated designs that follow a line, curve, or perimeter of a shape, for example:

- Decorative borders along, inside, or outside edges of a shape.
- Repetitive designs that flow along a line such as ropes, vines, and chains.
- Precisely-spaced patterns that follow a vector path, for example measurements on a curved ruler.

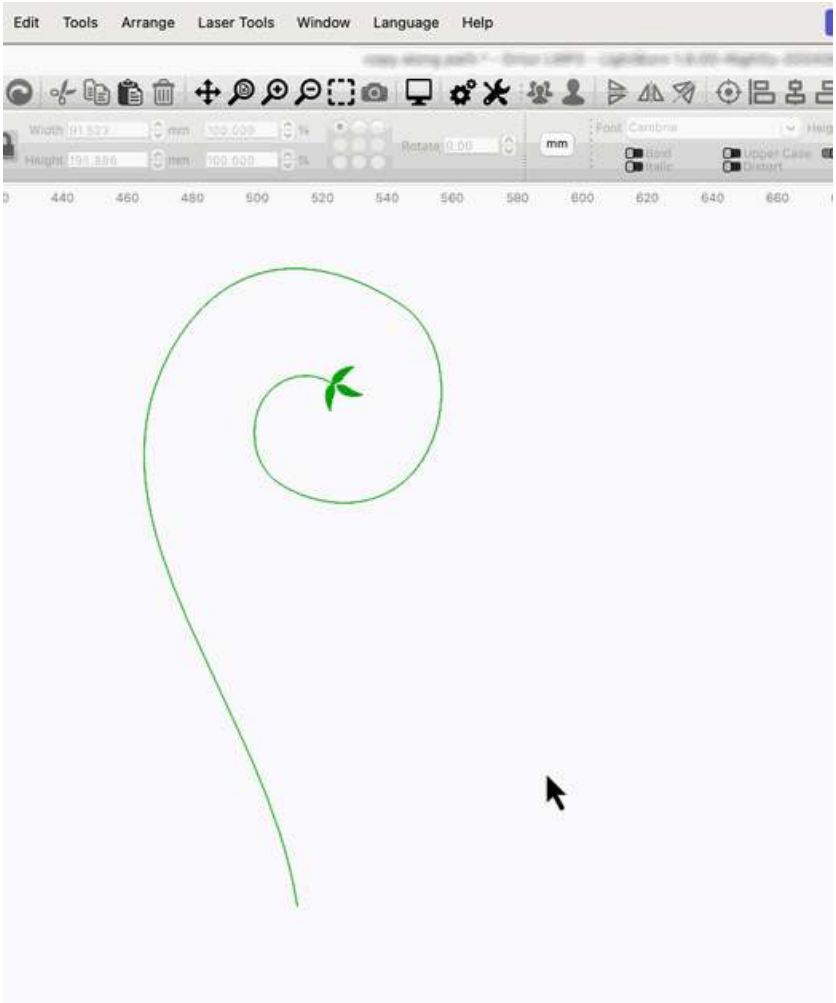
Using Copy Along Path

Copy Along Path works by duplicating copies of one object along the path of another.

Select the object to be copied first. Add the path to the selection (hold `Shift` while clicking the path), and go to Arrange → **Copy Along Path** to open the tool.

Adjust the options according to your needs — you'll see a preview of the result in the Workspace as you make changes.

Click **OK** to confirm the copies, or **Cancel** to discard them.



Requirements

The **object to be copied** must be:

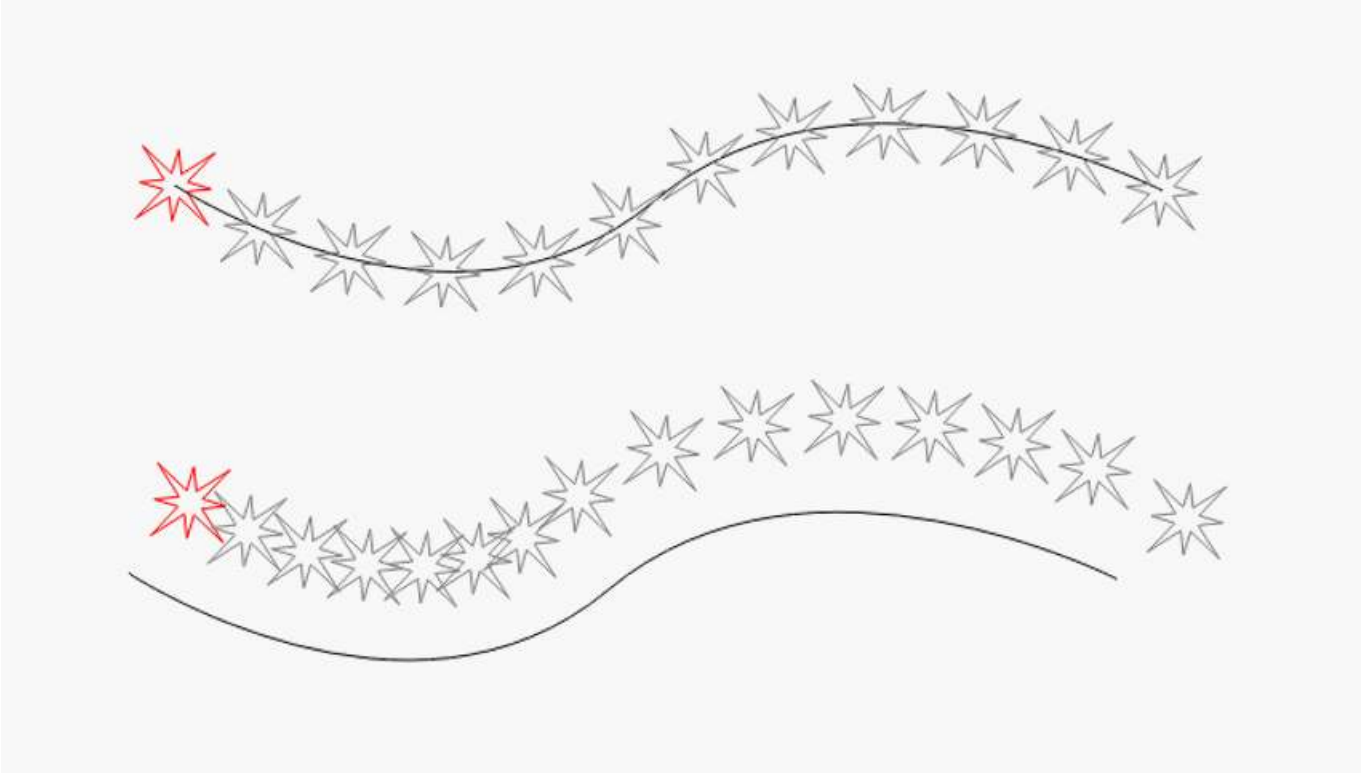
- A vector object — not an image
- A single object or a single Group of objects
- Selected first
-

The **path** must be:

- A vector — not an image
- A single, individual object — not a Group
- Selected second

Placement

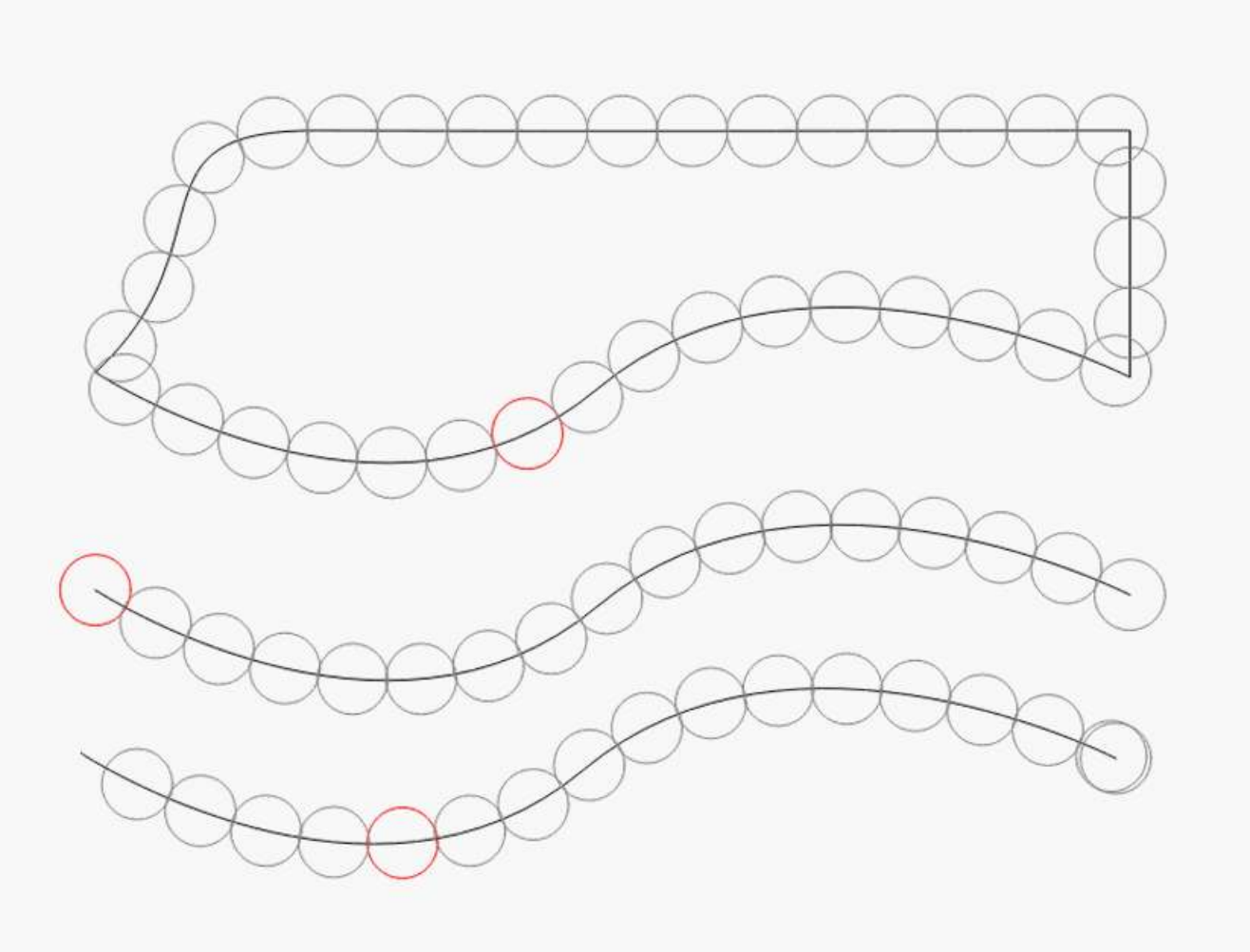
The placement of the original object to be copied relative to the start of the path determines the position of the copies.



Place the center of the object to be copied on the start of the path to superimpose each copy over the path.

Offset the object to be copied from the start of the path to maintain that distance from the path with each copy.

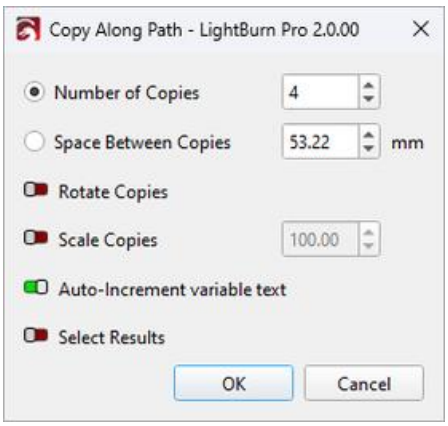
Open vs. Closed



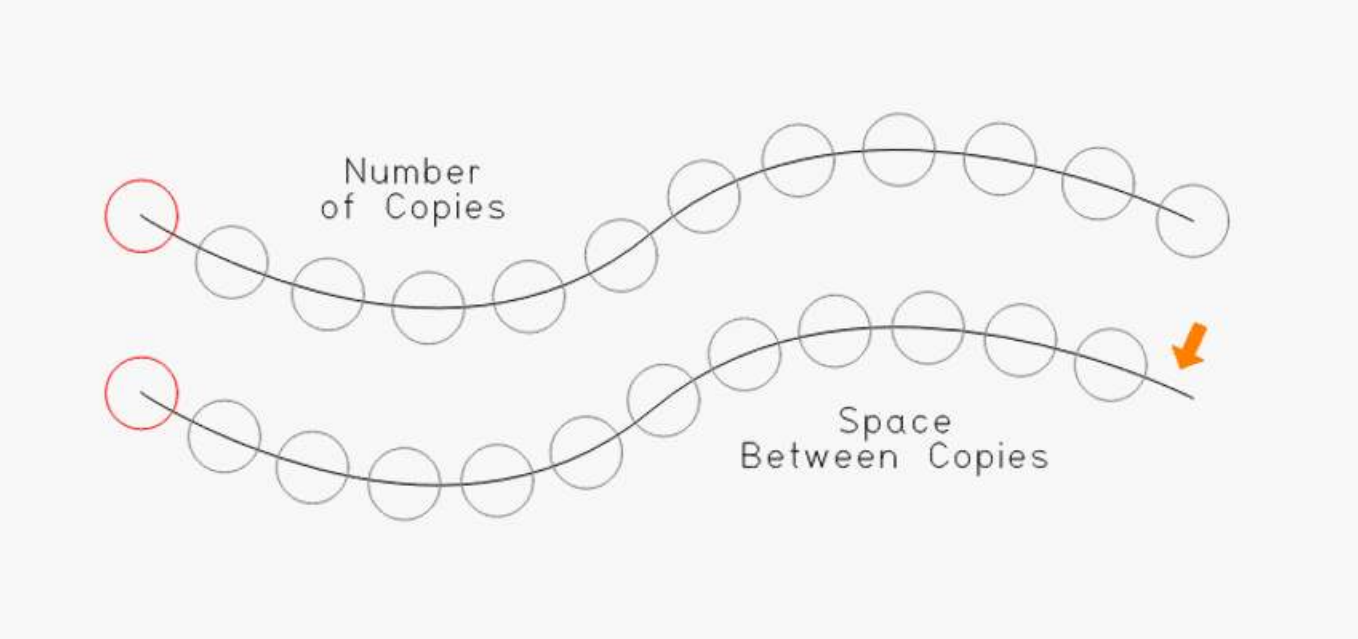
When using a closed path, the copied objects will start at whatever part of the path is closest, and then wrap around the circuit of the path to meet back up at the same point.

When using an open path, it's best to center the object to be copied over an end point of the path. This will create even spacing along the entire path.

Options



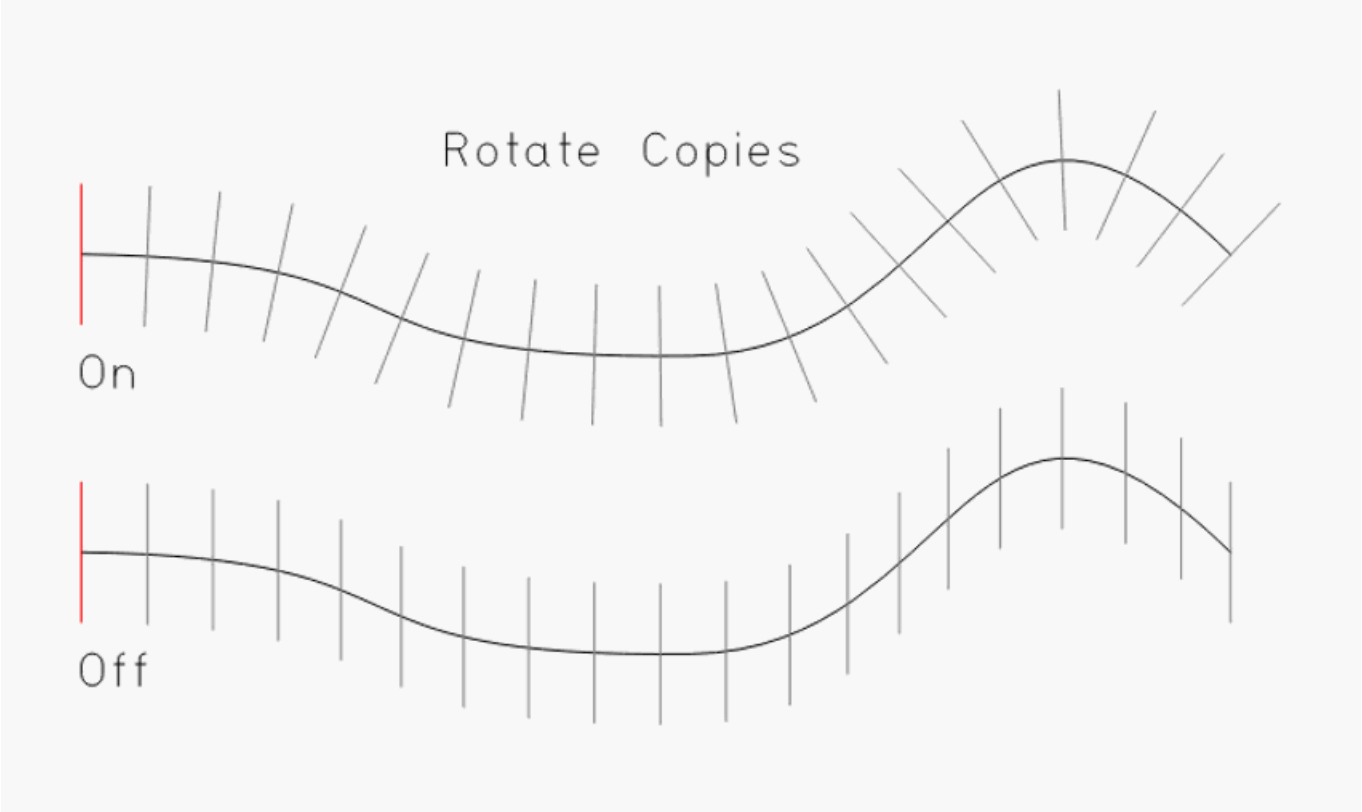
Spacing



The spacing of copied objects is measured from the centers of each copied object. There are two ways of spacing the copies objects along the path:

- **Number of Copies** spreads a certain quantity of them along the path, with even spacing between.
- **Space Between Copies** fills the path with as many copies as it can fit, given the spacing you provide. This usually leaves some excess line at the end of the path.

Rotate Copies



Enable **Rotate Copies** to create duplicates that are oriented acording to the changes in direction of the path. Disable this option to create objects that remain in the same orientation as the originally-copied object.

Scale Copies



Enable **Scale Copies** and enter a value in the adjacent field to incrementally change the size of each copy, with the final one reaching a size equal to this percentage of the original's size. A number larger than 100 will increase the sizes of the copies, a number lower than 100 will decrease them, and setting it to 100 (or disabling the switch) will keep the copies the same size as the original.

Auto-Increment Variable Text

Use **Auto-Increment variable text** in conjunction with Variable Text to add text-based codes that are substituted for other data when you send a project to your laser. Enabling Auto-Increment variable text will apply an Offset to each copied Text Object, so different data will output in the place of the same formatting expressions.

Select Results

Toggle this on to automatically select the resulting copies after clicking **OK**.

Troubleshooting

- **Copy Along Path** is not to be confused with Apply Path to Text, which attaches a single Text Object to a vector object, reshaping it to follow the contour of the chosen path.
- If your copies aren't appearing where you expected, check that their starting placement is set appropriately for your goals.
- If Copy Along Path is grayed out in the menu, it's likely that your objects are not suitable or they haven't been selected in the correct order.

Related Topics

- Apply Path to Text
- Circular Array
- Grid Array

Apply Path to Text

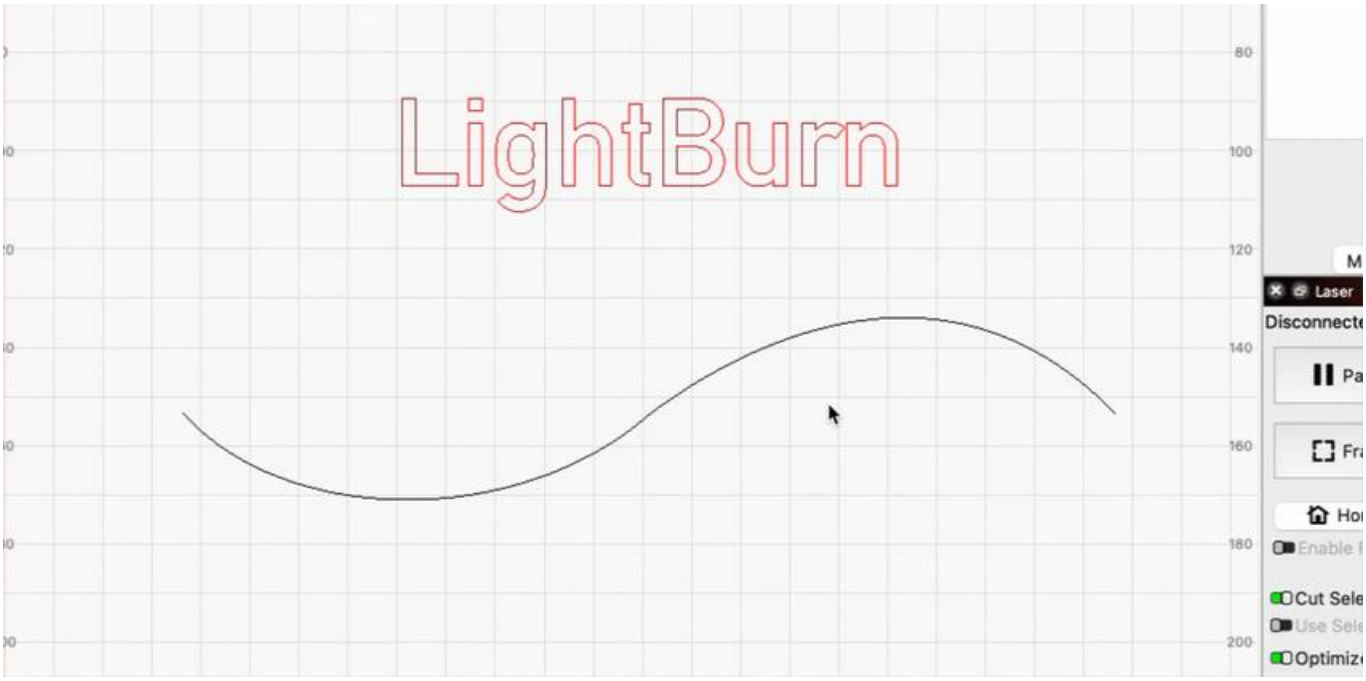
Apply Path to Text attaches text to a vector object, reshaping it to follow the contour of the chosen path. This tool makes it possible to use existing or custom-created paths to apply more complex curves to text than can be achieved using the text Bend tool.

Using Apply Path to Text

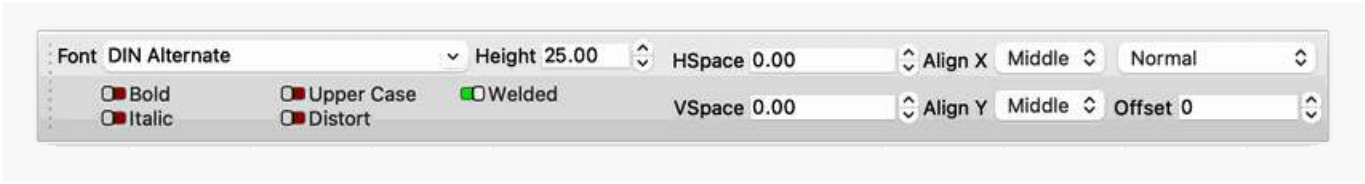
Select a single block of text and single shape at the same time, then go to Tools → **Apply Path to Text**, or right-click and select Apply Path to Text from the context menu.

The text is automatically moved to the location of the shape, and adjusted to follow its contour.

The text can no longer be independently moved after it is attached to the path, but all text attributes in the Text Options Toolbar can still be modified.



Making Adjustments in the Text Options Toolbar

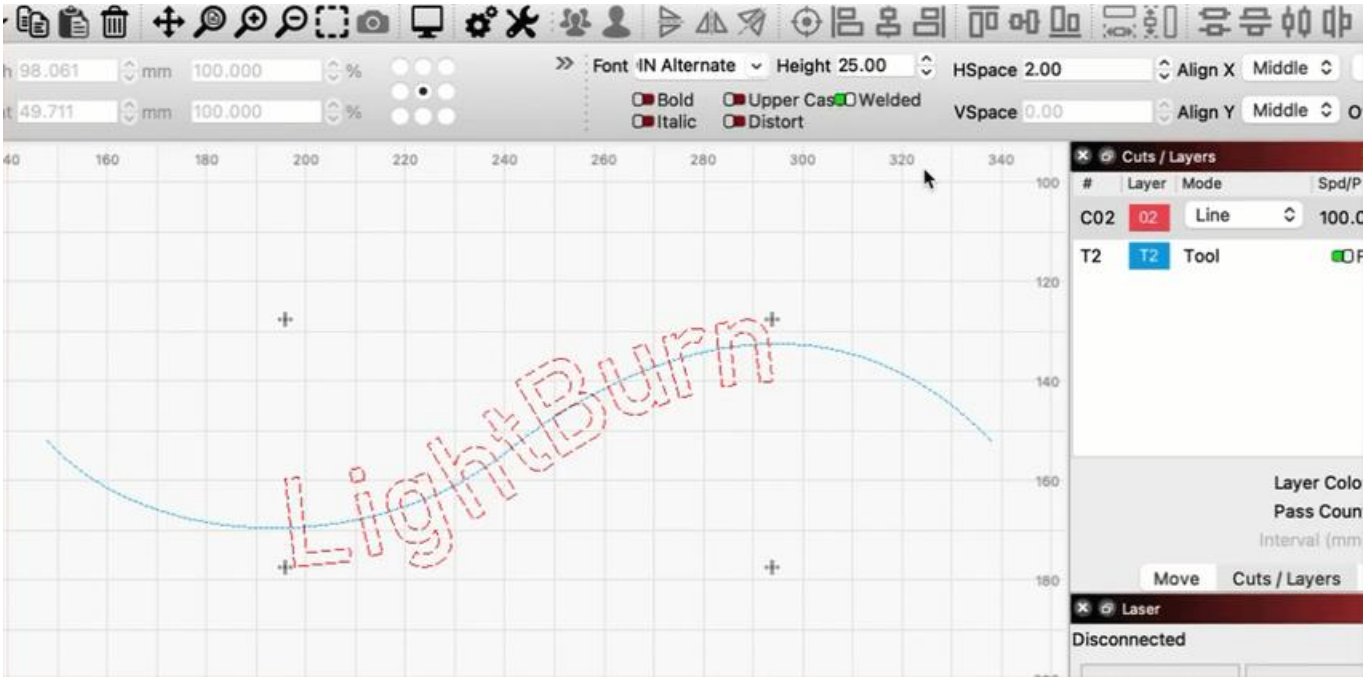


Note

In the case of text, horizontal always refers to the direction of the text as it is read, regardless of whether the Text Object is rotated. Vertical refers to the opposite direction, regardless of rotation.

The horizontal position of the text is oriented relative to the start and end points of the path using the **Align X** options. **Middle** aligns the text between the start and end points, **Left** aligns it so that the start of the text is next to the start point of the path, and **Right** aligns it so that the end of the text is next to the end of the path.

The vertical position of the text is positioned using the **Align Y** options. **Middle** places the text's vertical center on the path, **Bottom** places the bottom of the text directly on top of the path, and **Top** places the top of the text directly under the path.

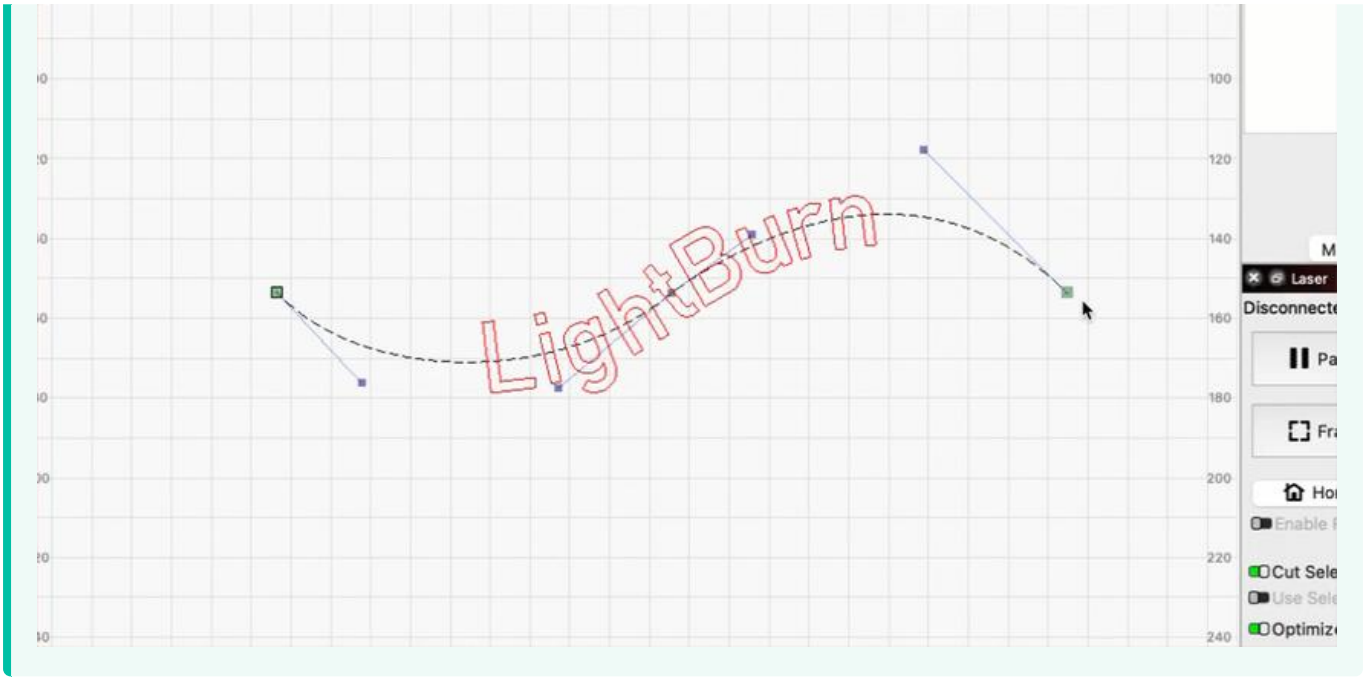


Manipulating and Deleting Paths

The position and shape of the path remain manipulable, and the text will automatically adjust to follow any changes made to the contour of the path.

Using Edit Nodes to manipulate paths

Use the Edit Nodes tool to manipulate nodes on a path that has already been applied to text, and adjust its appearance according to your visual preference.



To remove the text from the path, the path must be deleted. Deleting the path returns the text to its original orientation, but does not affect any settings in the Text Edit toolbar.

Tip

If you have applied text to a path that you do not want cut or engraved by your laser, but do not wish to delete either, assign the path to a Tool layer, as shown in the demonstrations above. Tool layers are special layers that are never sent to your laser.

Troubleshooting

- If text applied to a path is upside down or mirrored, you may need to Flip the text before applying the path to it, or Rotate or re-construct the path to correct the orientation.

  When manually creating a path with the Draw Lines tool, the first point you click is always the start point, and the last is always the end point.
- To use **Apply Path to Text**, you must have exactly one Text Object and one vector object selected. If the option is grayed out and unavailable, make sure the vector object you have selected is a single, connected path, and not multiple adjacent but disconnected paths. Use Edit Nodes or Auto-Join Selected Shapes to join disconnected paths.
- Grouped objects do not work with Apply Path to Text. **Ungroup** all text or vector objects you wish to use with this tool.

Related Topics

- Create and Edit Text
- Text Options Toolbar

- Copy Along Path
- Draw Lines
- Edit Nodes

Radius / Fillet

Use the **Radius Tool** to fillet (round over) sharp corners, take a round bite out of a corner (using a negative radius value), or turn a radius back into a sharp corner (remove a radius).

You can use Radiuses to:

- Round out corners for a softer appearance
- Create smooth transitions between sharp lines
- Form more complex shapes (varying sizes and types)
- Add rounded details to projects
- Round out corners of tabs, to enable easy assembly

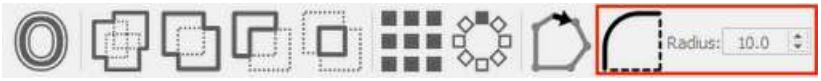
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=GSxphMRsOFk

Using the Radius Tool

You can apply a radius to any corner node of any Ungrouped path or Primary Shape, provided that the lines on either side of the node can accomodate the size of the radius. Radiuses can be applied to text if you first convert it to paths.

Add a Radius to a Path

To add a radius, first click the **Radius Tool** icon in the Modifiers Toolbar.

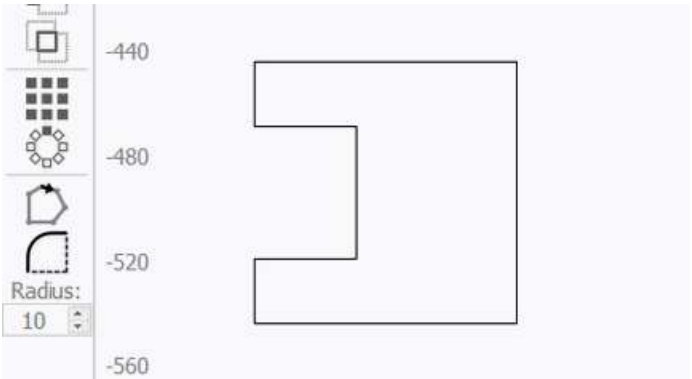


Set the radius value to the size of the radius you wish to create — there must be enough room between the corner and nearby points to accomodate this value.

If you change the radius to a negative number it takes a fancy little bite out of the corner.

Select the object you want to modify, and click on a corner to apply the radius.

To exit the Radius Tool, press Esc on your keyboard.



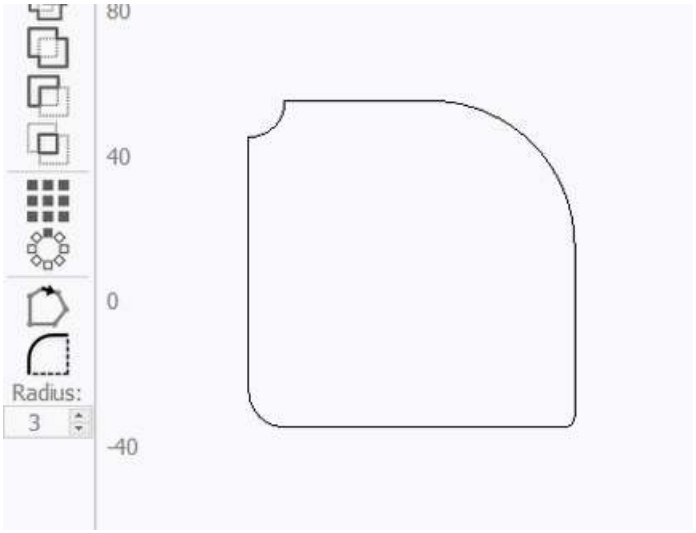
When the Radius Tool is active, the icon changes as you hover over a corner to indicate whether the Radius Tool can be used. If it can't be used, the cursor will change to indicate that there is not enough space.

Icon	Meaning
	Click to fillet corner
	Click to remove fillet
	Can't fillet — not enough space

Remove or Change a Radius

You can also use the Radius Tool to remove rounded corners (returning them to a sharp one) by clicking on them. This even works on imported designs that were made in different software, as long as the software exported actual arcs.

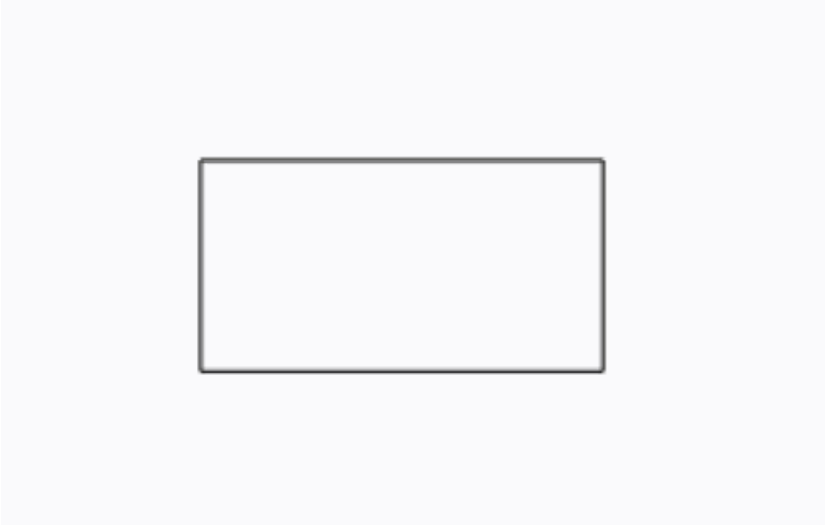
After removing a radius, you can then apply a differently-sized one to the corner, as normal.



Adding a Radius to a Rectangle

You can apply radiuses to Rectangles in the Shape Properties Window. This method has the advantage that resizing the rectangle won't change the size of the radius, allowing you to maintain consistency in a design.

Rectangles also have a handle that acts as a shortcut for the Radius value. With a Rectangle selected, hold `Ctrl` / `Cmd`, and a blue corner radius control will appear. Drag the blue handle away from the corner to increase the radius of all four corners, and drag back toward the corner to decrease it. Dragging vertically creates a reversed radius, with a bite taken out of the corner.



Troubleshooting

- The **Radius Tool** cannot be used with Grouped shapes. **Ungroup** any shapes you'd like to apply a radius to.
- If the cursor isn't appearing as you hover over a corner, make sure you've selected the Radius Tool in the Modifiers Toolbar. The Radius icon should be highlighted. Also check that you have selected the object you wish to edit by clicking on it.
- Radiuses can only be applied where there is room for the radius to be applied. If you see the following icon, it means there is not enough room:



- If a radius still can't be applied, it might be because the two lines that make up the corner aren't actually joined at that point. To check, move the corner using the Edit Nodes tool. If it splits off into two lines, bring the one you moved back toward the other so that they snap together and join, and then try the radius again.
- If you are having trouble converting a rounded corner back into a sharp one, it could be because there is not a true arc at the corner. This is most common with imported shapes, because some programs export arcs as a series of tiny lines. You can try recovering the arcs with Optimize Selected Shapes.

Related Topics

- Edit Nodes
- Primary Shapes: Rectangles, Ellipses, and Polygons
- Shape Properties
- Offset Shapes

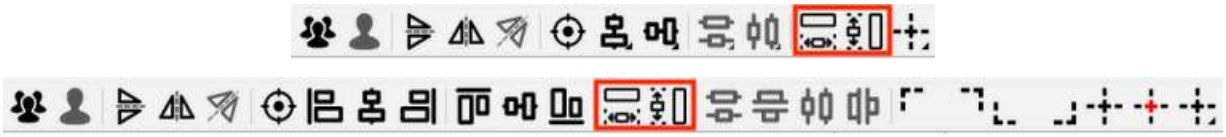
Make Same Width or Height

The **Make Same Width** and **Make Same Height** buttons resize all objects in your selection to the width or height of a single object in the selection.

By default, all selected objects will be resized to the width or height of the last object you added to the selection. For exceptions to that rule, see below.

Using the Make Same Width and Height Tools

Both buttons are available in the Arrange Toolbar and **Arrange (long) Toolbar**.



Option	Icon
Make Same Width	
Make Same Height	

Select all objects you wish to resize, followed by the object whose width and/or height you wish to match, then click either **Make Same Width** or **Make Same Height**.

Grouped objects are treated as single objects when using the Make Same tools — the overall width or height of an object or entire Group will change to match the width or height of the last-selected object or other Group.

Hold `Shift` while clicking to lock the aspect ratio of the objects you're resizing, adjusting both dimensions at once.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=fhgLBJxxae4

> **Note**
>
> Holding `Shift` does not change both dimensions of objects in your selection to match those of the last-selected object — it only matches the selected dimension to that of the last-selected object, and adjusts the alternate dimension to maintain the same proportional length to the changed dimension.
>
> For example, for a 40 mm (wide) x 20 mm (high) square, if you held `Shift` while clicking Make Same Width to match it to a 10 mm wide object, it would become a 10 mm x 5 mm square.

Resizing Priority

When using either of the **Make Same** options, all selected objects are matched to the width or height of a single object or Group in your selection.

The object that is matched to is determined according to the following priorities:

- Selection order: by default, all objects are resized to match the last object you added to your selection.
- Draw order: if you have nothing selected, then select multiple objects at once, they are matched to the first object in your selection's draw order. Use Push to front to move an object to the first position in the draw order.
- Combined order: if you add multiple objects at once to an existing selection, all selected objects are matched to the first object in the draw order of the objects you added to your selection last.
- If a single object in your selection is Locked, all objects are matched to the Locked object.
- If multiple objects in your selection are Locked, the Locked objects are not matched, and the rest of the objects are matched according to selection order or draw order.

Related Topics

- Numeric Edits Toolbar
- Align Tools
- Push in Draw Order
- Distribute

Resize Slots & Tabs

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=0Yep7sYi9Rs

Use **Resize Slots** to adjust the dimensions of slots or tabs in selected objects.

You can use Resize Slots to adjust a jointed design that was intended for a material of a different thickness, or to adjust slots or tabs back to their original size after you rescaled an entire design.

This tool works on two main forms of geometry:

- Slots, like those found in 3D puzzles or partitions in trays, somewhat similar to a traditional halved joint or dado.
- Tabs, like a tenon or the fingers in a box joint, usually found in boxes and drawers.

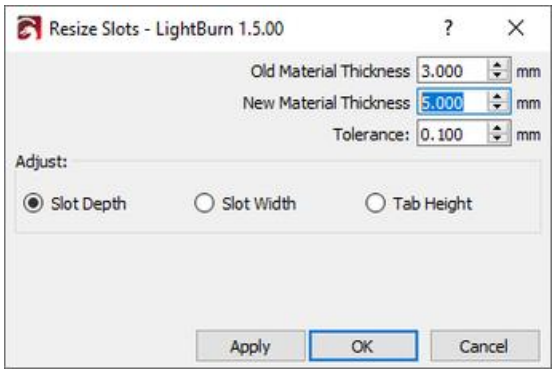
> **Note**
>
> This is a separate tool from Add Tabs, which creates small sections of cuts that the laser is commanded to skip, to prevent objects that you are cutting from immediately falling out of the base material.

Using Resize Slots

Before opening the **Resize Slots** tool, you'll need to know the existing length of the lines you want to adjust. Use the Measure tool to read the length of an individual slot or tab.

Select all objects with slots or tabs you'd like to resize, then go to Tools → **Resize Slots in Selection** to open the **Resize Slots** dialog window.

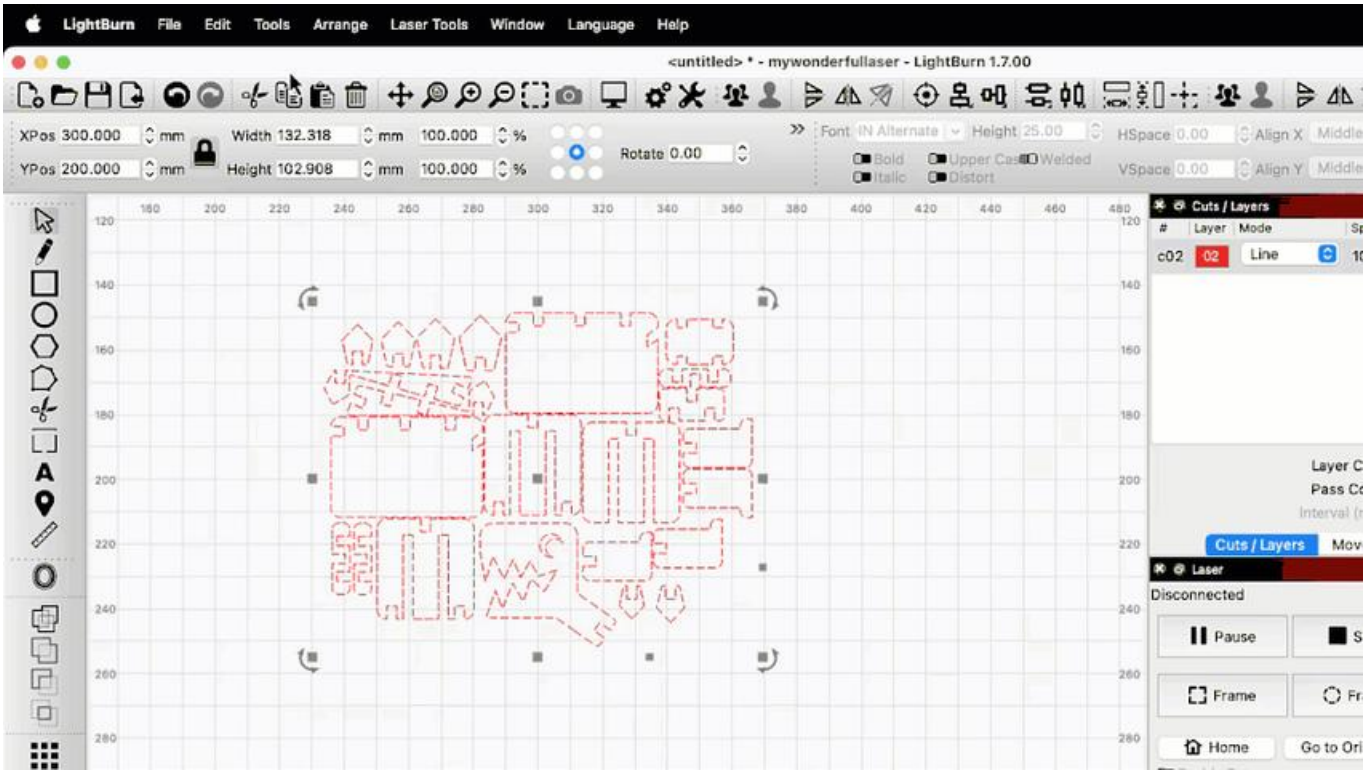
Settings



- **Old Material Thickness** defines the current length of the slots or tabs you want to identify and adjust.
- **New Material Thickness** defines the length you want to adjust the slots or tabs to.
- **Tolerance** specifies an added amount of padding to Old Material Thickness — slots or tabs that are within the range of Old Material Thickness plus or minus Tolerance will be identified by the resizer.

This helps if the slots or tabs vary slightly in length, or if there are kerf offsets built into the original design.

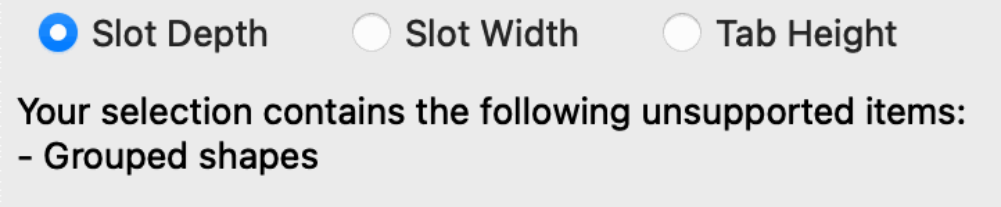
- Select **Adjust Slot Depth** to adjust the bottom/innermost line of slots. This allows you to keep the outer size of the assembled piece the same.
- Select **Adjust Slot Width** to adjust the sides of slots. Typically used when the slots are meant to intersect on the interior of a laser cut piece instead of on the corners, like boxes.py's tray insert.
- Select **Adjust Tab Height** to adjust the top/outermost line of tabs. This allows you to keep the interior size of the assembled piece the same, while making it bigger or smaller on the outside.
- Click **Apply** to implement a given set of adjustments and continue working in the Resize Slots dialog window.
- Click **OK** to implement of a set of changes and immediately exit.
- Click **Cancel** to immediately exit and discard changes, if you have not already pressed Apply to implement them. To revert changes you already implemented with Apply, exit the Resize Slots dialog window, then press `Ctrl`/`Cmd`+`Z` to Undo them.



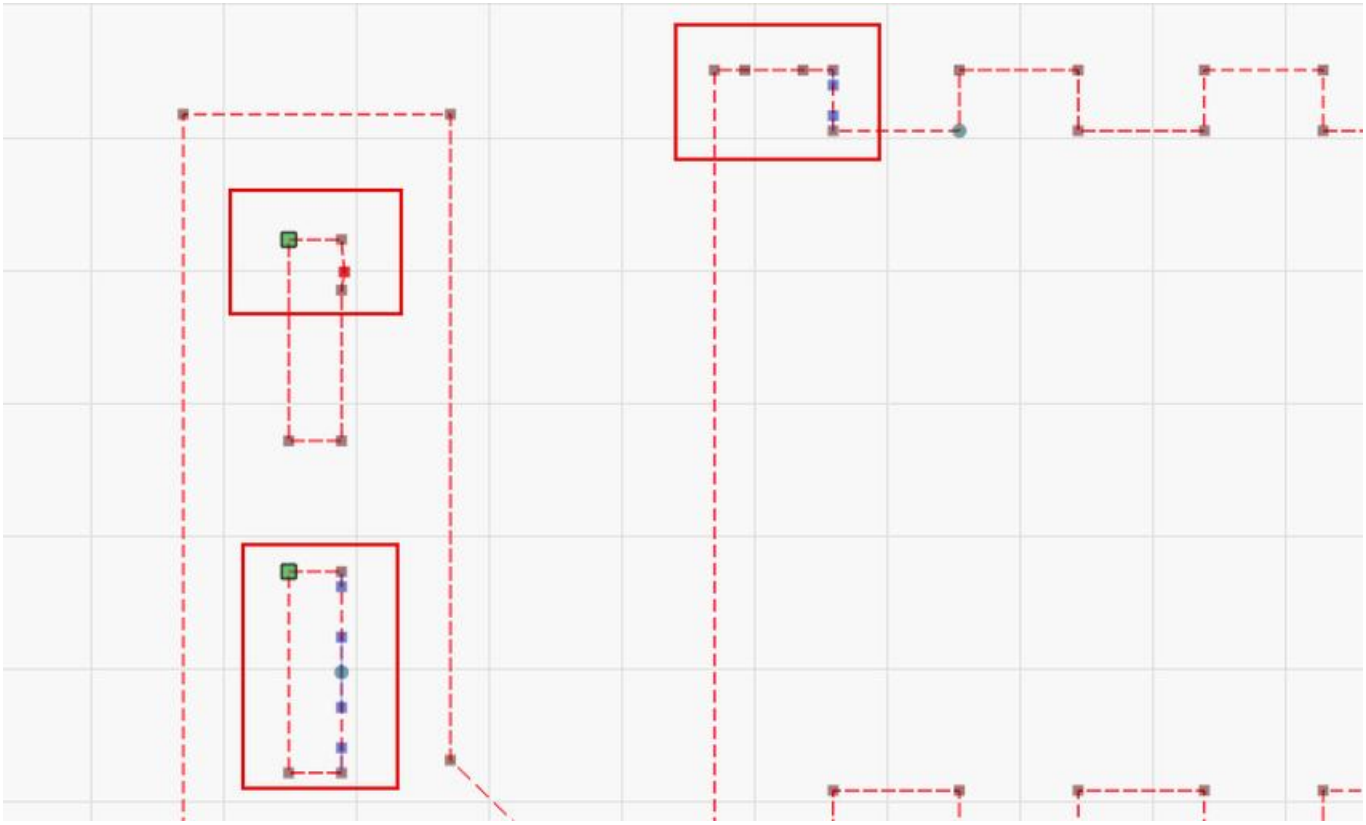
Troubleshooting

- The tool doesn't work on internal slots/cutouts/mortises, as it's tricky for the computer to determine which direction they should grow or shrink.

- The tool searches Ungrouped objects. Exit the Resize Slots dialog and Ungroup your objects if you see a warning message indicating that there are Grouped objects in your selection.



- The tool only works on sharp (not filleted/rounded) corners. Some tools (like boxes.py) default to adding small loops on internal corners and small fillets/rounded corners on external ones. Remove the external fillets using the Radius tool and the internal loops using the Offset tool, set to 0.
- The tool requires well-formed shapes as an input. If the shape has extra nodes, or if any of the lines are technically curves (even if they're straight), it will interfere with slot/tab detection. The highlighted sections in the image below show examples of problematic shapes. Use Optimize Selected Shapes to try to repair these shapes.



Related Topics

- Kerf Offset
- Measure
- Grouping
- Optimize Selected Shapes

Arrangement

Align Tools

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=jbSoR-FcHSY

The **Align** tools reposition selected objects to align them along their:

- Top, bottom, left, or right edges
- Vertical or horizontal centers
- Center points

By default, the **Align** tools move objects in your selection to align them to an edge or center of the last object you added to your selection. For exceptions to that rule, see below.

Align Options

The **Align** tools are available in the Arrange Menu and Arrange Toolbar.

In the standard Arrange Toolbar, they are presented in dropdown menus for the vertical and horizontal options:



In the **Arrange (long) Toolbar**, the Align tools are each presented individually:



Hotkeys

All seven **Align** options and their actions are listed in the table below.

The hotkeys listed under **Windows** and **macOS** will work at any time that you have objects selected.

The hotkeys listed under **Edit Window in Focus** will only work when the Workspace — the area where you create and edit graphics — is in active focus.

> **What does it mean for a window to be in focus?**
>
> There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.
>
> If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus. Selecting objects in the Workspace will also bring it into focus.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=g73qIhYD7LU

Align Option	Icon	Action	Windows	macOS	Edit Window in Focus (All Systems)
Align Centers		Aligns the center points of selected objects.	n/a	`Option` + `Fn` + `Left`	n/a
Align Vertical Centers		Aligns selected objects along their vertical centers.	`Alt` + `Pg Up`	`Option` + `Fn` + `Up`	`C`
Align Horizontal Centers		Aligns selected objects along their horizontal centers.	`Alt` + `Pg Dwn`	`Option` + `Fn` + `Down`	`E`
Align Left		Aligns selected objects along their left edges.	`Alt` + `Left`	`Option` + `Left`	`L`
Align Right		Aligns selected objects along their right edges.	`Alt` + `Right`	`Option` + `Right`	`R`
Align Bottom		Aligns selected objects along their bottom edges.	`Alt` + `Down`	`Option` + `Down`	`B`
Align Top		Aligns selected objects along their top edges.	`Alt` + `Up`	`Option` + `Up`	`T`

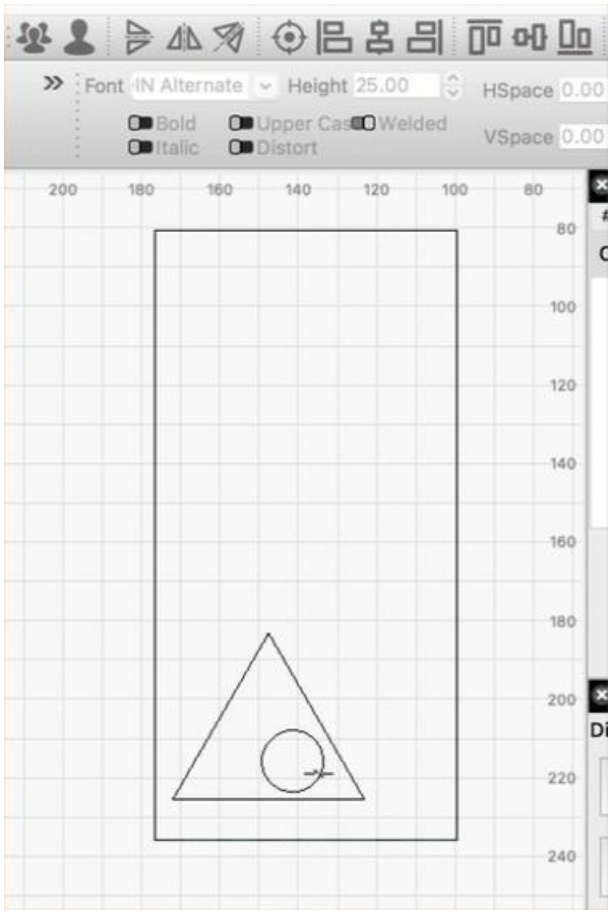
Using the Align Tools

Click a given **Align** icon or use its hotkey to align selected objects to the last object you added to your selection (exceptions below).

Edges of individual objects are defined according to their farthest points in each direction. For example, the top edge of a triangle is a horizontal line extending from its top point, its left edge is a vertical line extending from its bottom left corner, and so on.

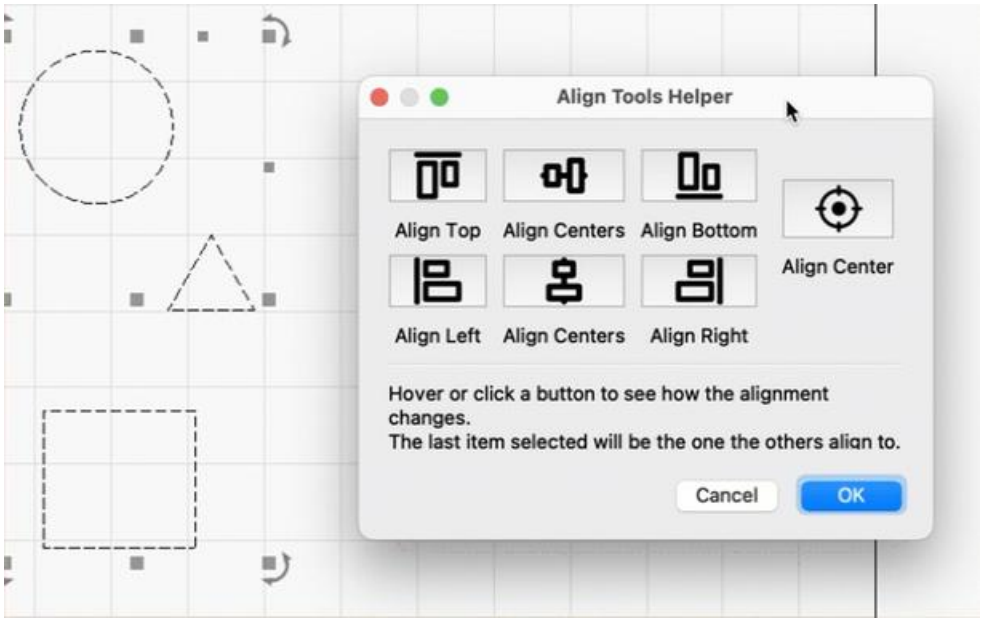
An object's vertical center is defined as a vertical line dividing it halfway along its total width, and its horizontal center is a horizontal line halfway along its total height. Its center point is the location where those lines intersect.

Grouped objects are treated as single objects when using the Align tools — they will always maintain their relative position to one another, and their edges and centers are defined according to the dimensions of the entire Group.



Align Tools Helper

If you're using Beginner Mode, clicking the **Align Left** icon in the Arrange Toolbar will open the **Align Tools Helper**.



Click or hover over any button in the Helper to see a preview of how alignment will change.

Click **OK** to confirm the changes, or **Cancel** to discard them.

Alignment Priority

> **Selecting an object to align to**
>
> After selecting multiple objects, hold down `Ctrl` and click twice on the object you want to align to. This will deselect and reselect it, making it the top priority for alignment.

When using any of the **Align** options, all selected objects move to align to an edge or center of a single object or Group in your selection.

The object that is aligned to is determined according to the following priorities:

- Selection order: by default, all objects are aligned to the last object you added to your selection.
- Draw order: if you have nothing selected, then select multiple objects at once, they are aligned to the first object in your selection's draw order. Use **Push to front** to move an object to the first position in the draw order.
- Combined order: if you add multiple objects at once to an existing selection, all selected objects are aligned to the first object in the draw order of the objects you added to your selection last.
- If a single object in your selection is Locked, all objects are aligned to the Locked object.
- If multiple objects in your selection are Locked, the Locked objects are not moved, and the rest of the objects are aligned according to selection order or draw order.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=SeqjyRnhgtM

Related Topics

- Automatic Guidelines
- Selection Tools
- Push in Draw Order
- Docking
- Distribute
- Move Together
- Move Selected Objects

Distribute & Move Together

The **Distribute** tools move selected objects to evenly space them either vertically or horizontally, with options to set the same distance between the centers, or edges of each object.

The **Move Together** tools move selected objects so that they are evenly distributed, and their edges are abutting along horizontal or vertical planes.

Distribute

The **Distribute** tools are available in the Arrange Menu and Arrange Toolbar.

In the standard Arrange Toolbar, they are presented in dropdown menus for the vertical and horizontal options:



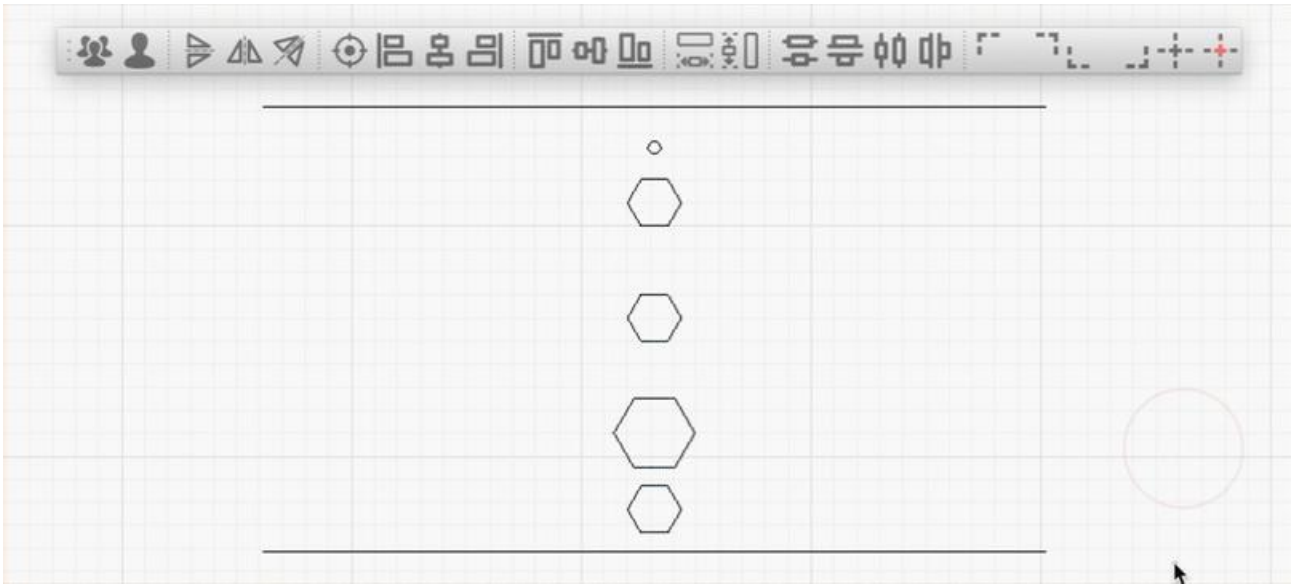
In the **Arrange (long) Toolbar**, the Distribute tools are each presented individually:



Distribute Vertically

Distribute V-Spaced creates even spacing between the edges of all selected objects in the vertical dimension.

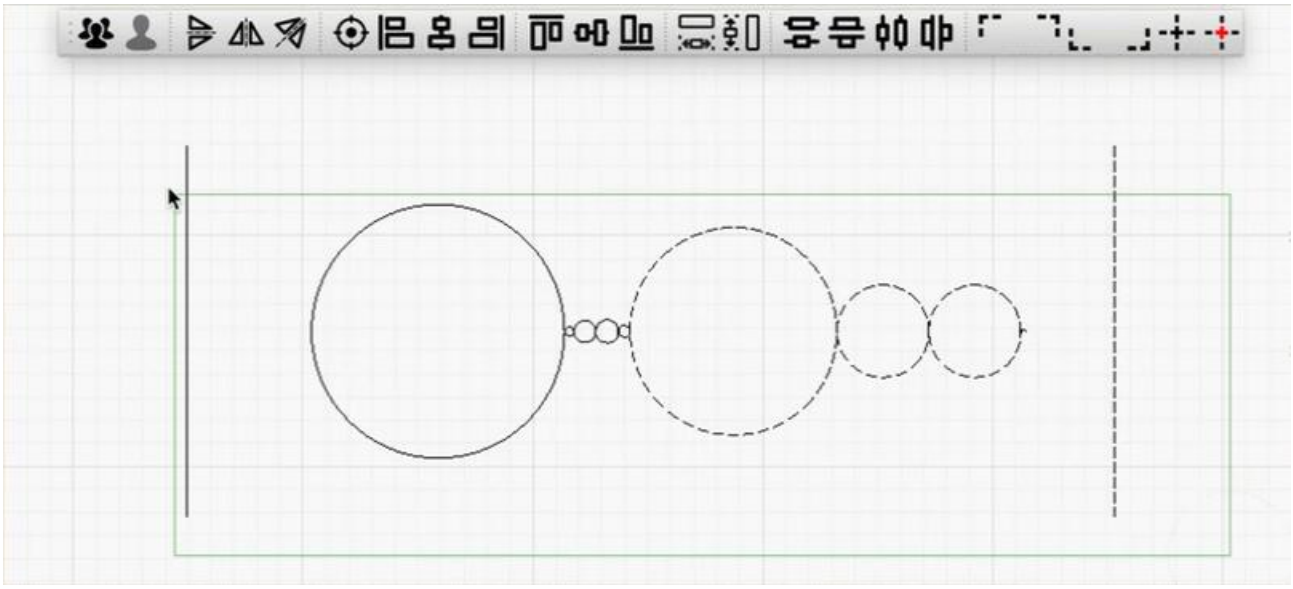
Distribute V-Centered creates even spacing between the centers of all selected objects in the vertical dimension.



Distribute Horizontally

Distribute H-Spaced creates even spacing between the edges of all selected objects in the horizontal dimension.

Distribute H-Centered creates even spacing between the centers of all selected objects in the horizontal dimension.

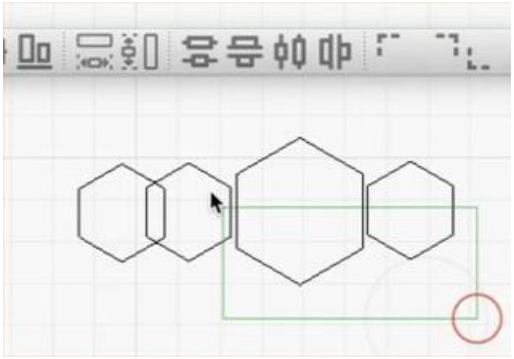


Notes on Spacing

The **Distribute** tools take an average of the current spacing between the edges or centers of your selected objects and move them to equalize that distance.

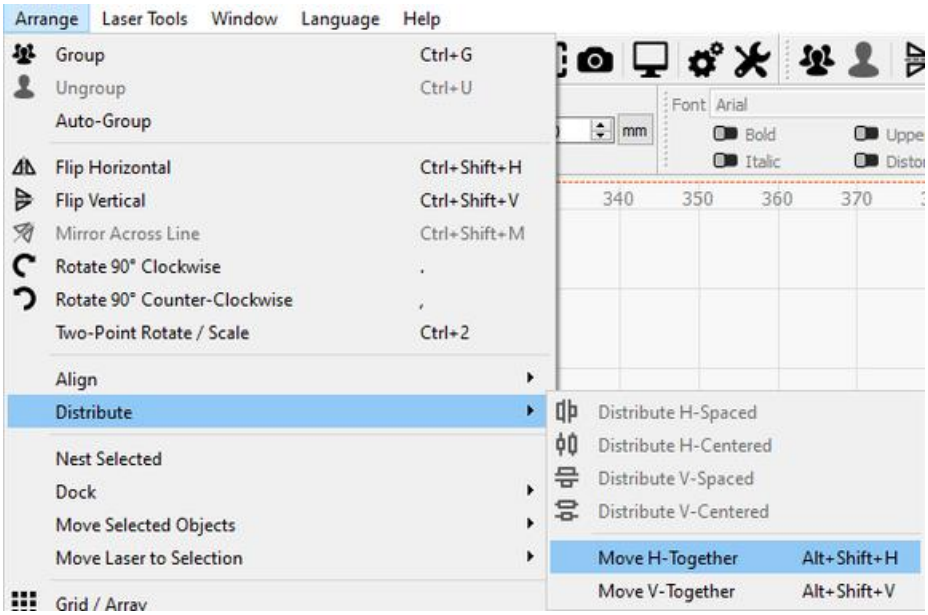
It is not always the case that the resulting average distributes objects such that there is positive spacing between all edges of all objects — only that the distance is even. If the amount of existing negative spacing (overlap) between objects in your selection exceeds positive spacing, the resulting distribution will also have overlap.

If large and small objects are distributed together with even spacing between their centers, they may overlap afterward, even if they were not overlapping to begin with.



Move Together

Both **Move Together** options are available in the **Distribute** submenu of the Arrange Menu, and by hotkey.

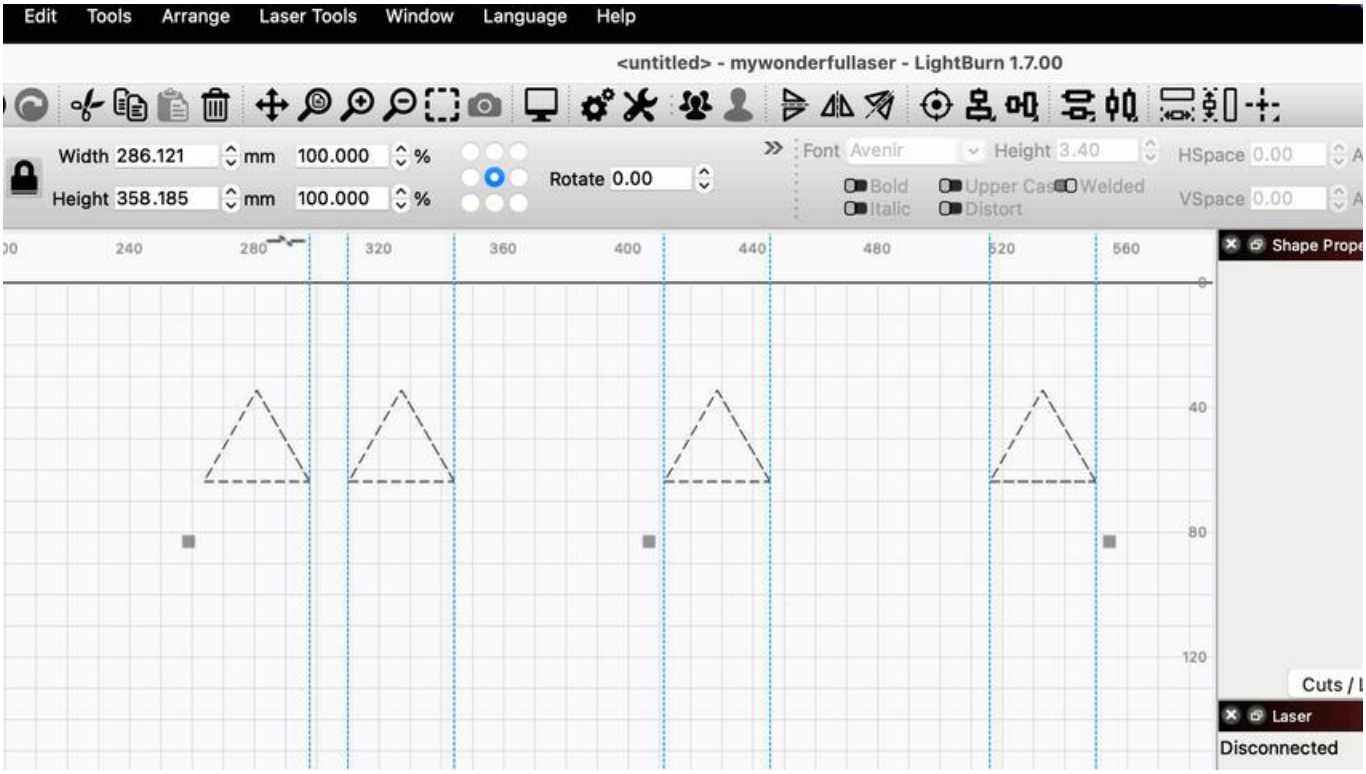


Move H Together

`Alt`/`Option`+`Shift`+`H`

Moves all selected objects so that they are evenly distributed, and their horizontal edges are abutting along vertical planes.

The last objected added to your selection is anchored in place, and all other objects will be moved relative to its location.

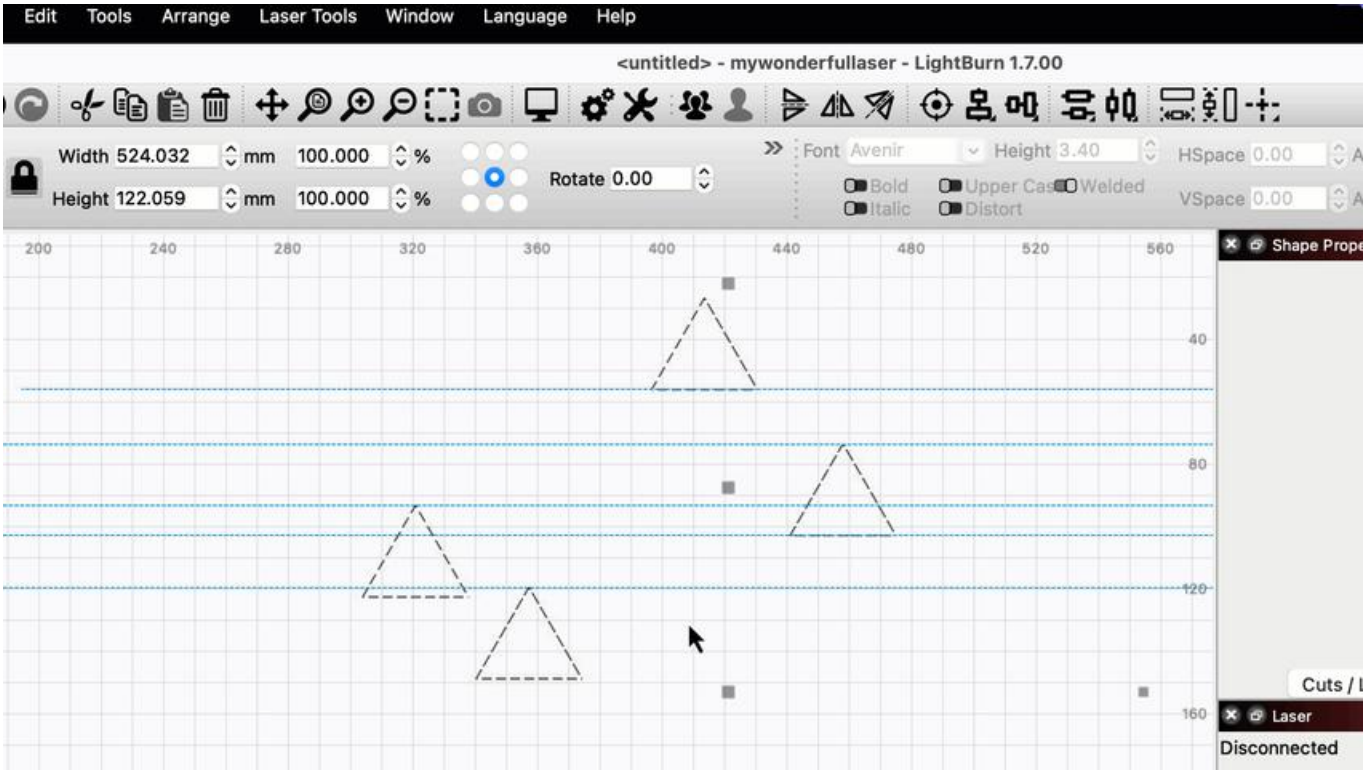


Move V Together

`Alt`/`Option`+`Shift`+`V`

Moves all selected objects so that they are evenly distributed, and their vertical edges are abutting along horizontal planes.

The last objected added to your selection is anchored in place, and all other objects will be moved relative to its location.



Related Topics

- Align Tools
- Docking
- Move Selected Objects
- Grid Array
- Selection Tools

Docking

The **Docking** tool moves selected objects left, right, up, or down, until they meet the edge of the Workspace or another object. Docking can bring objects together or move them in unison, making sure they maintain their relative position to one another.

Docking is helpful for laying out graphics to efficiently utilize space and material.

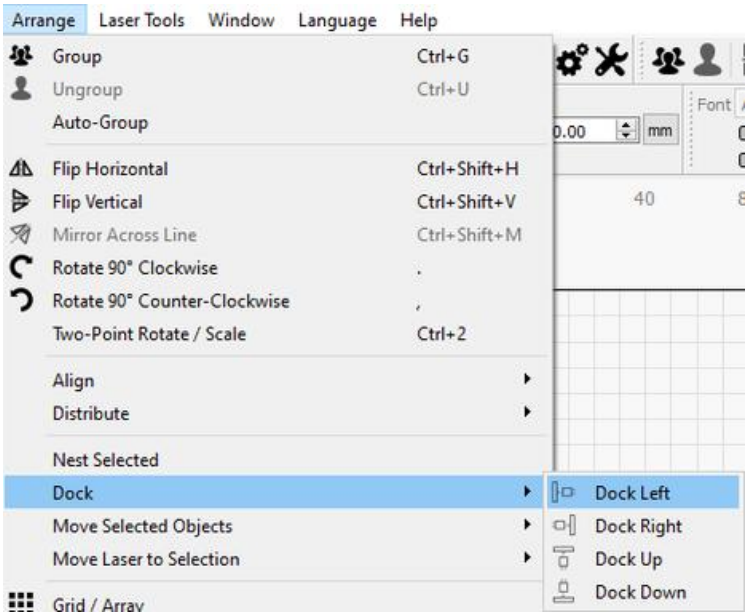
Accessing the Docking Tools

The **Docking** options are available in the **Dock** submenu of the Arrange Menu, or in the **Docking Toolbar**.

If you do not see the Docking Toolbar, go to Window → **Docking** to enable it. By default, it will appear on the far right of the top toolbar when enabled.



> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.



Docking Options

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=gmIcv56YXB4



Each of the **Docking** buttons moves selected objects in a specified direction, until they hit something — either the outer edge of another object, or an inner edge of your Workspace.

Objects contained within another object will not Dock to the inside edge of the object they are contained in, instead, they will continue past the inner edge until they hit the outer edge of another object, or an edge of your Workspace.

You can Dock objects:

- Left 
- Right 
- Up 
- Down 

In the **Docking Toolbar** there are three additional options to control Docking behavior.

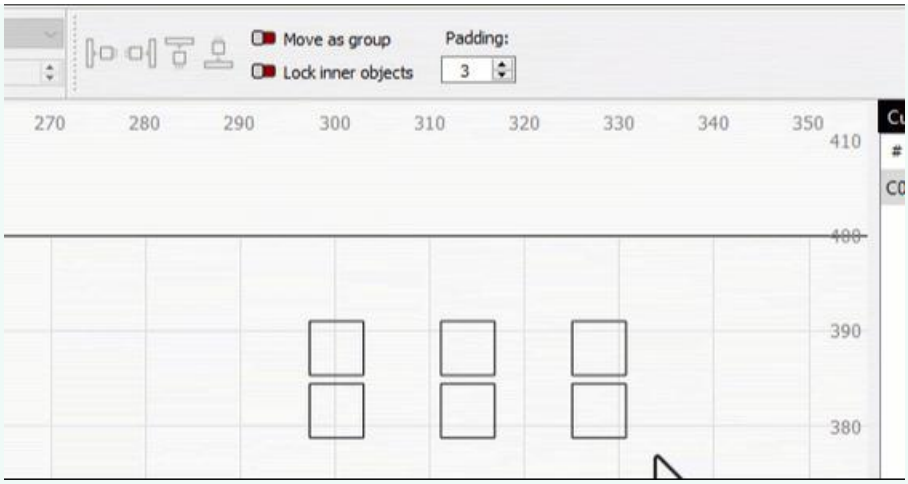
- **Move as group** tells LightBurn to treat all selected objects as a single group when Docking them — that means they will all stop when one object in the group hits the edge of another object or the Workspace, and all selected objects will maintain their relative position to one another when they're Docked.

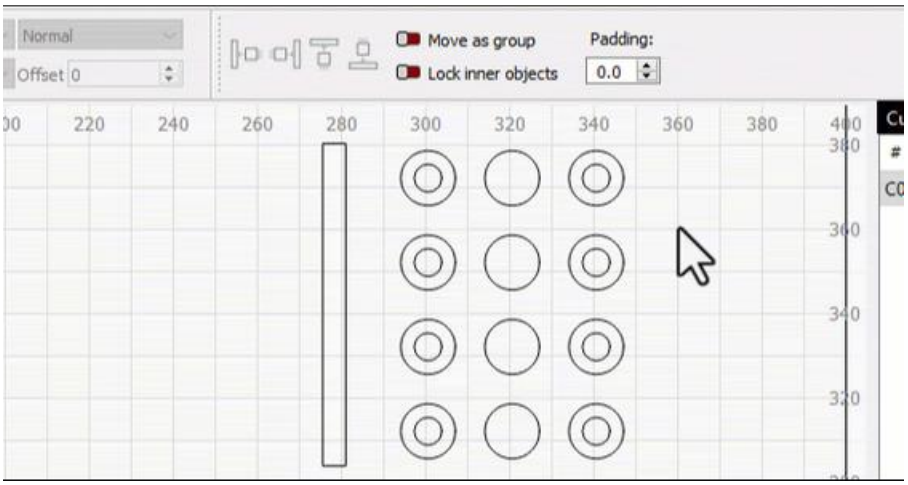
  > **Note**
  >
  > Grouped objects will always move together when Docked, regardless of whether Move as group is enabled.

- **Lock inner objects** maintains the relative position of objects contained within other objects, when both are Docked at the same time.
- **Padding** sets an amount of space to maintain between objects or the edge of the Workspace when docking.

  The units used for Padding (in / mm) are the same as the units you have selected in the Numeric Edits Toolbar or your Settings.

  > **Creating even Padding in both directions**
  >
  > Padding is only applied in the direction the objects are Docking, and if an object is already closer to another object or the edge of the Workspace than the Padding distance is set to, the object won't be moved. To apply consistent Padding in both directions, Dock objects away, then back again.





Related Topics

- Snapping
- Distribute
- Move Together
- Move Selected Objects
- Grid Array

Move Selected Objects

The **Move Selected Objects** tools move objects to a corner, midpoint, or center of your Workspace. If you have a Gantry style laser, it can also move objects to the last reported position of your laser head.

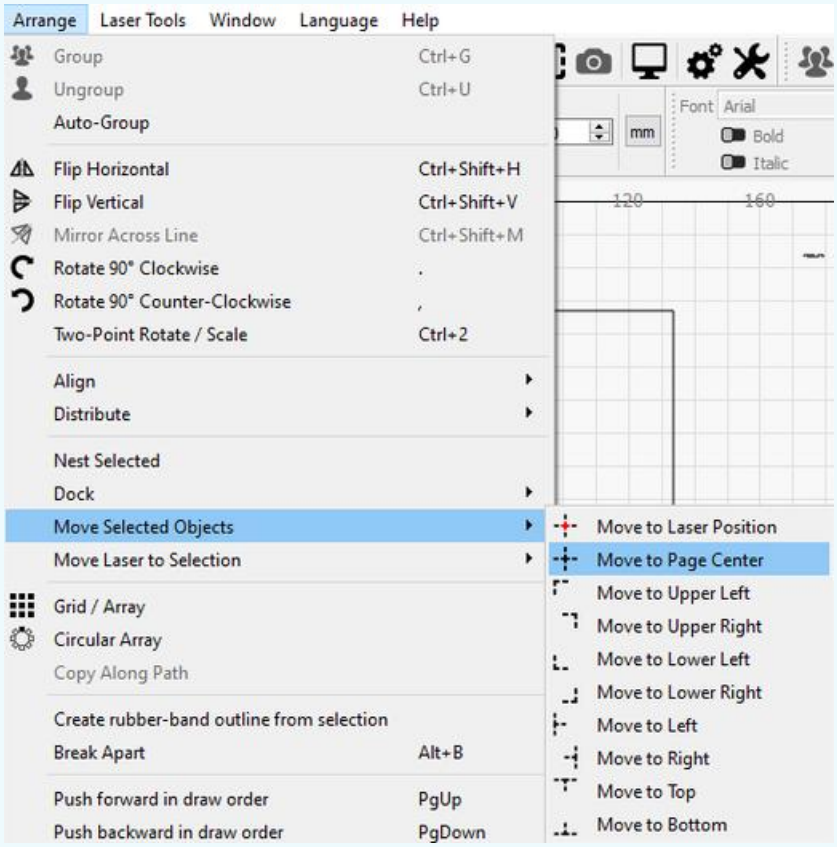
Use these tools to rearrange objects in your Workspace, or to align graphics in LightBurn to the location of material placed in your laser's bed, when using Absolute Coordinates positioning.

UI Locations

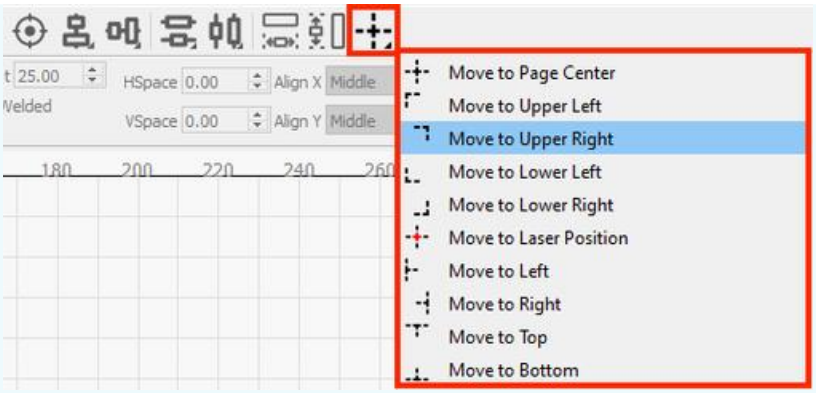
All **Move Selected Objects** tools are available in the Arrange Menu or as a drop down menu in the Arrange Toolbar.

In the **Arrange (long) Toolbar**, the options to move to the center of your Workspace, a corner of your Workspace, or the last reported position of the laser head are available in the full toolbar, while other options are available in a drop down menu.

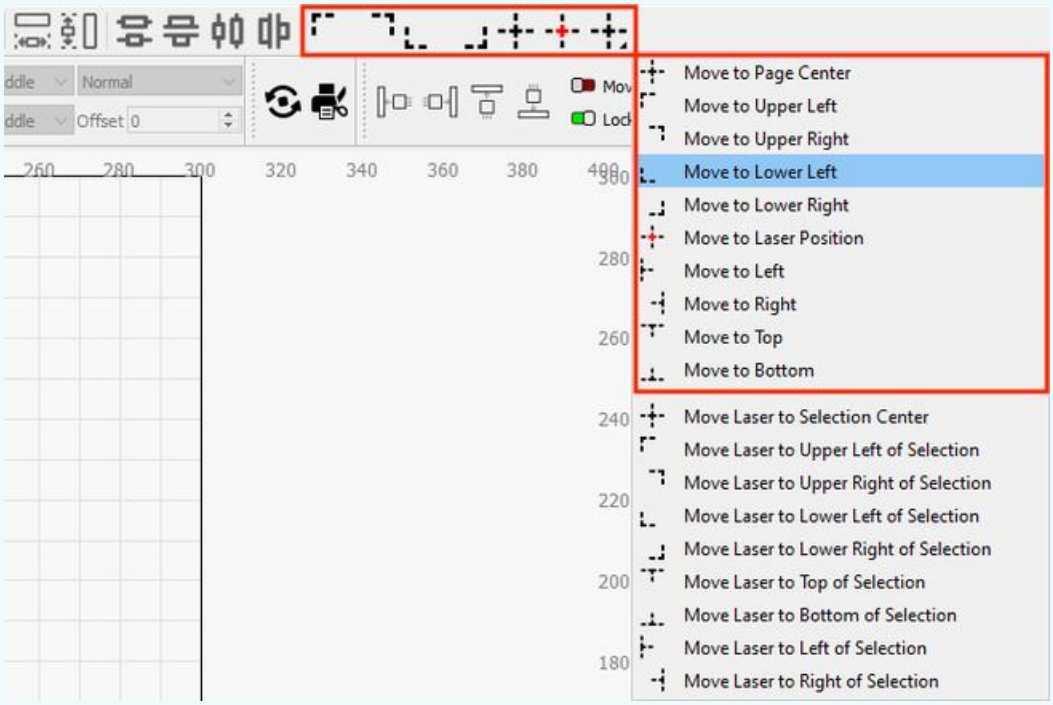
Move Selected Objects in the Arrange Menu



Move Selected Objects in the Arrange Toolbar



Move Selected Objects in the Arrange (long) Toolbar



Options and Icons

Info

Holding the `Ctrl` or `Cmd` key while pressing one of these buttons will move the laser to the indicated location on the selection, in machine coordinates, instead of moving the selected objects.

Warning

In order to use **Move to Laser Position**, LightBurn must be connected to your laser, and it must be capable of reliably homing and reporting its position accurately. If the position reported by the laser is inaccurate, using this feature may cause your laser to attempt to travel outside of its physical bounds.

Option	Icon	Action
Move to Laser Position		Moves selected objects in your Workspace to the location that corresponds to last reported position of the laser in machine coordinates. This option is not available for Galvo lasers.
Move to Page Center		Moves selected objects to the center of your Workspace.
Move to Upper Left		Moves selected objects to the top left corner of your Workspace.
Move to Upper Right		Moves selected objects to the top right corner of your Workspace.
Move to Lower Left		Moves selected objects to the bottom left corner of your Workspace.
Move to Lower Right		Moves selected objects to the bottom right corner of your Workspace.
Move to Left		Moves selected objects to the far left of your Workspace, centered between the top and bottom.
Move to Right		Moves selected objects to the far right of your Workspace, centered between the top and bottom.
Move to Top		Moves selected objects to the top of your Workspace, centered between the left and right.
Move to Bottom		Moves selected objects to the bottom of your Workspace, centered between the left and right.

Move to Page Center Hotkey

Press P to move selected objects to the center of your Workspace.

This hotkey only works if the Workspace / Edit Window — the area where you create and edit graphics — is in active focus.

What does it mean for a window to be in focus?

There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.

If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus.

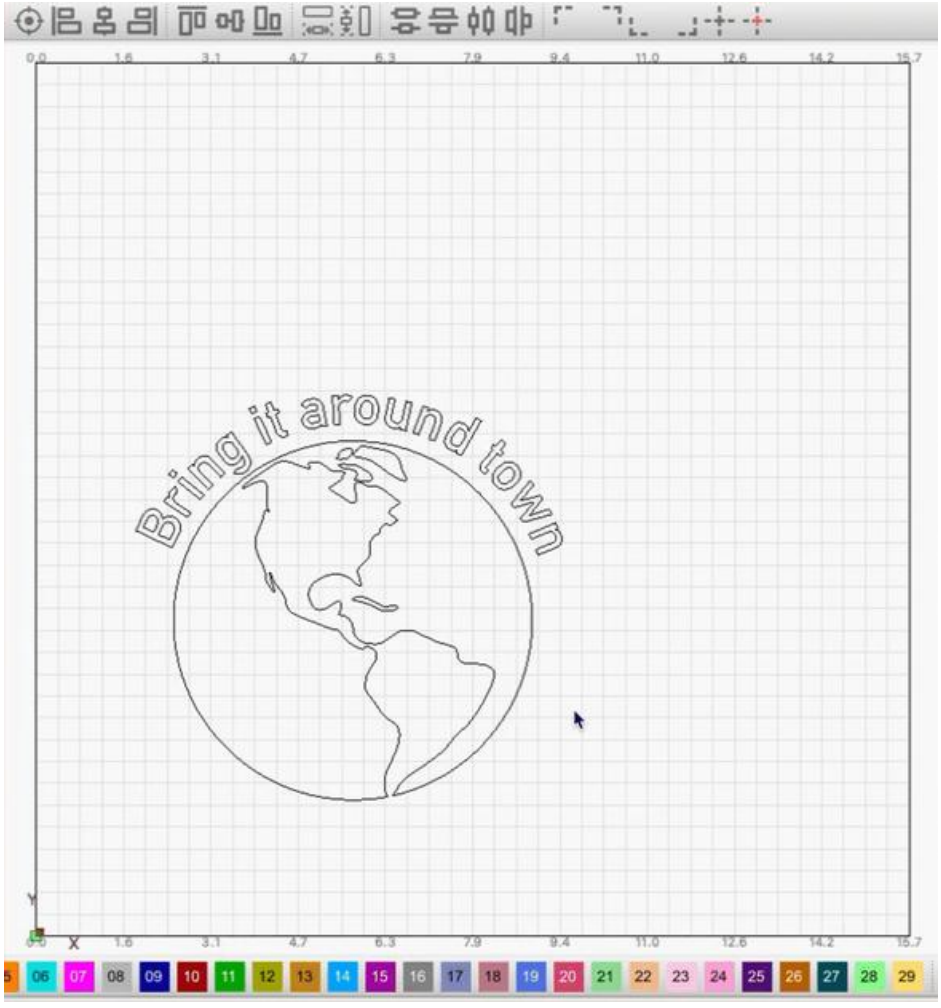
Movement of Selections

When using the **Move Selected Objects** tools, all objects in your selection move as a single unit, maintaining their relative position to one another.

Edges of selections are defined according to the farthest points of any object in the selection area, in each direction — imagine the selection area as the smallest possible rectangle that can contain every object in your selection. The edges of a selection are the same as the edges of that imaginary rectangle.

Selections are moved so that their edges abut the location you move them to — for example, the left edge of a selection will move to touch the left side of your Workspace when you use **Move to Left**, and the top and right edges of a selection will touch the top and right sides of your Workspace when you use **Move to Upper Right**.

When using **Move to Page Center**, the center point of the entire selection is moved to the center of your Workspace.



Related Topics

- Move Laser to Selection

- Align Tools
- Distribute
- Move Together
- Selection Tools

Nesting

True Nest

Nesting enables you to arrange objects to use less material. LightBurn 2.2 offers you a choice between **True Nest** (LightBurn Pro only), **Quick Nest**, and **SVGNest (External Tool)**.

	True Nest	Quick Nest	SVGNest (External Tool)
LightBurn release	LightBurn 2.2 Pro	LightBurn 2.1	LightBurn 1.1
Nests based on true shape of objects	□	×	□
Fully integrated in LightBurn	□	□	×
Works locally, internet not required	□	□	×
Read more	True Nest	Quick Nest	SVGNest (External)

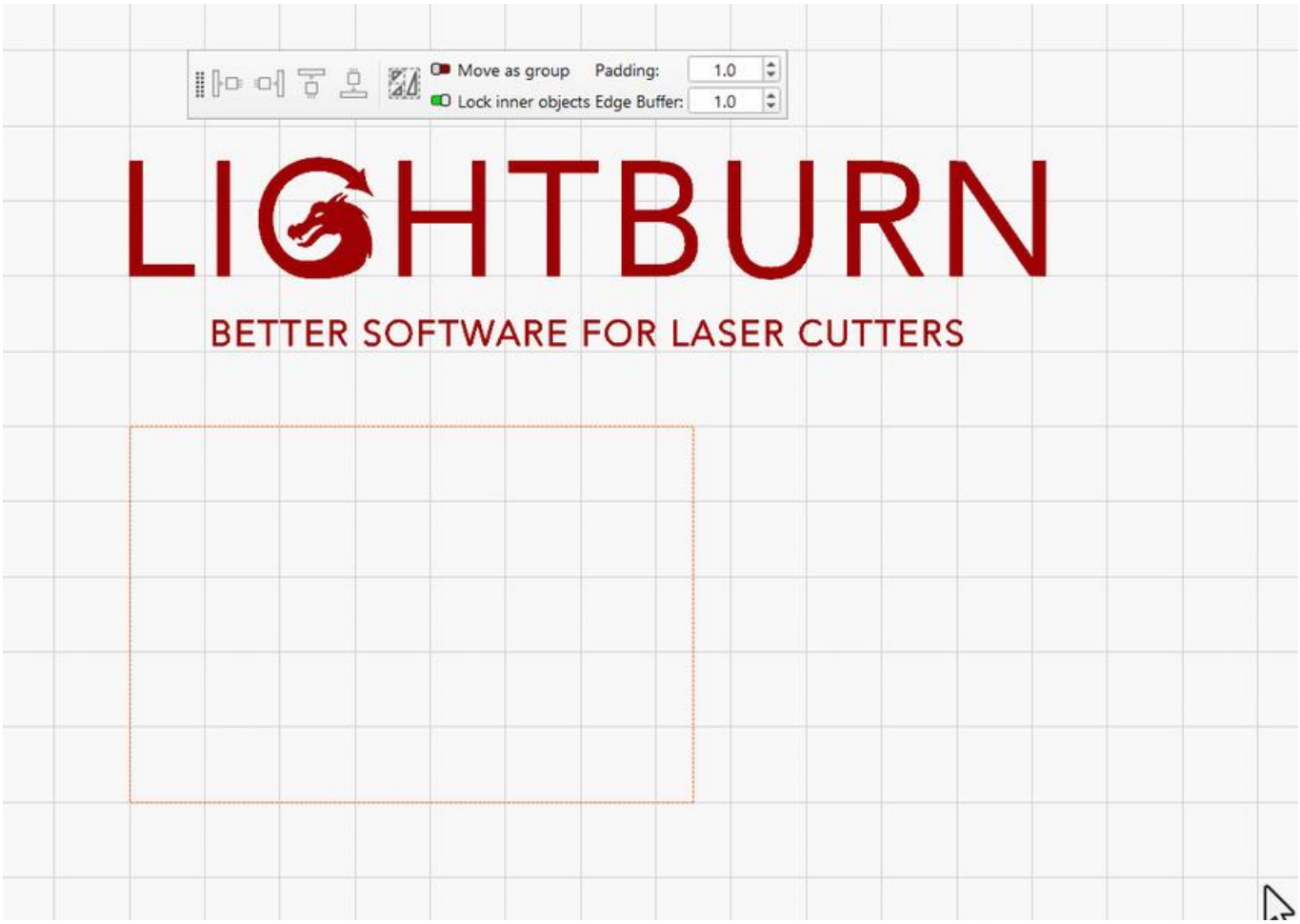
Using True Nest

You can find both True Nest and Quick Nest in the **Docking Toolbar** or by going to Arrange → **Nesting**.

True Nest requires LightBurn Pro. If you do not have LightBurn Pro, you can use Quick Nest or SVGNest (External), or you can try Pro and consider upgrading.

> **Accessing the Docking Toolbar**
>
> By default the Docking Toolbar is in the top right corner of the screen. Depending on your screen size, you may see a short version like in the screenshots below, with an arrow to the right to expand the toolbar. For the rest of this page, we'll use the full version of the toolbar in screenshots.
>
> To see all the options, you can expand the toolbar with the arrows to the right, drag it to the left to give it more space, drag it to another toolbar with more space, or drag it out of the toolbar area entirely. All four are shown below.

More About the Docking Toolbar



Getting Started

1. Create a rectangle to use as a container for the shapes you want to nest.
2. Select the container and all of the objects you want to nest.
3. Open the **Nesting Dialog** by clicking the **Nesting** button or going to **Arrange** → **Nesting**.
4. Select **True Nest** and adjust your Nesting options.
5. Click **OK** to move onto the next dialog.
6. Choose your Allowed Orientations and your optimization time (how long LightBurn will run the nesting process looking for a better layout) and click **Start Nesting** to begin the process.
7. While waiting for the nesting process you have three options:
   - Click **Stop Nesting** at any time to stop and accept the current layout.
   - Click **Next Page** (if Multipage is enabled) to start nesting the currently unnested objects on the next page.
   - Click **Cancel** to stop the process without keeping the changes.
8. After the time you've selected, LightBurn will exit the dialog and keep the best layout it's found.

Nesting Dialog

Mode

Choose between **Quick Nest** and **True Nest**.

Options

These options are shared between **Quick Nest** and **True Nest**.

Padding

Sets the minimum distance between the parts to nest. The units are either millmeters or inches, depending on your Design Units. Increase this number to leave more space between parts.

Edge Buffer

Sets the minimum distance between the parts to nest and the edge of the container you're nesting in. The units are either millmeters or inches, depending on your Design Units.

Lock Inner Objects

If **Lock Inner Objects** is enabled, inner objects move with the objects surrounding them, whether or not they're grouped. In the demonstration at the top of the page, **Lock Inner Objects** keeps the inner shapes of the letters and the dragon's eye in place while nesting without having to group them.

Lock Objects for Rotation

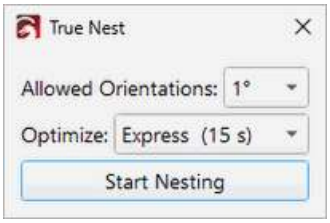
Enable this to lock the rotation of objects being nested. Disable this to allow LightBurn to rotate the objects as part of the nesting process. **Quick Nest** rotates in 90 degrees increments if this is enabled. **True Nest** offers more rotation options in a secondary dialog.

Allowing rotation can improve material usage, but may not work well for all designs, particularly when cutting parts from material with a visible grain.

Multipage

With **Multipage** enabled, if you try to nest more objects than fit in your target container, LightBurn will create additional pages with a copy of the original container to continue the nesting process. With **Multipage** disabled, LightBurn will nest as many objects as it can and leave the rest unnested.

True Nest-Specific Options



Allowed Orientations

The **Allowed Orientations** option controls how True Nest is allowed to rotate objects. Setting it to 1° allows True Nest to rotate in 1 degree increments to help squeeze objects into tight places. Setting it to 90° limits rotation to 90 degree increments, similar to how Quick Nest works. There are also a range of values in between the two.

If Lock Objects for Rotation is enabled, the value chosen here doesn't matter, because True Nest won't rotate objects.

Optimize

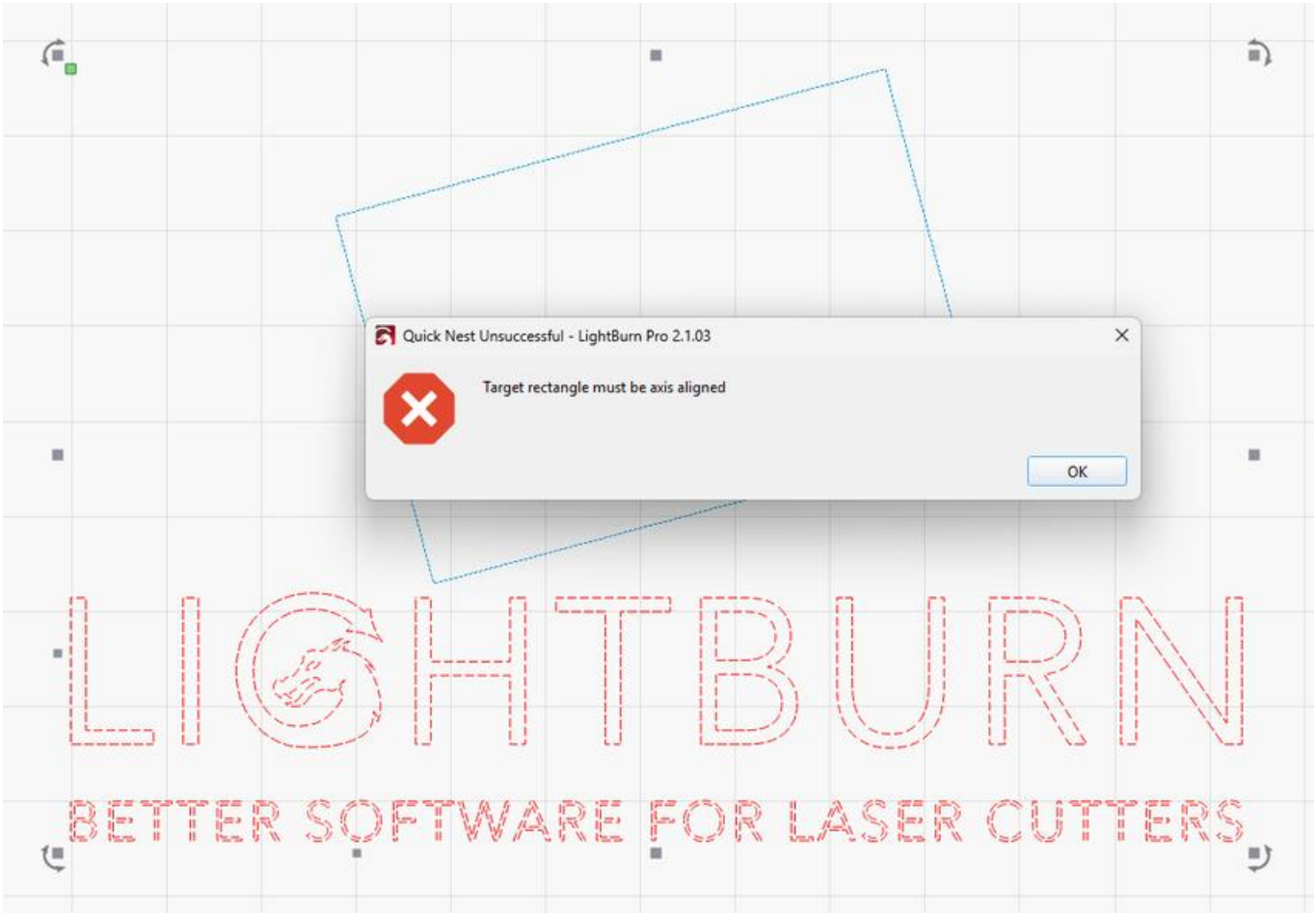
The **Optimize** setting lets you choose how long the nesting process will run for. Longer times can get you denser packing, but it's up to you whether the difference is worth the wait. If you do select a longer wait time, you can use the **Stop Nesting** button at any time to accept the current layout and move on.

Nesting Errors and Warnings

Target Rectangle Must be Axis Aligned

To use **Quick Nest**, the rectangle you're nesting into must have sides that are straight up and down and sideways.

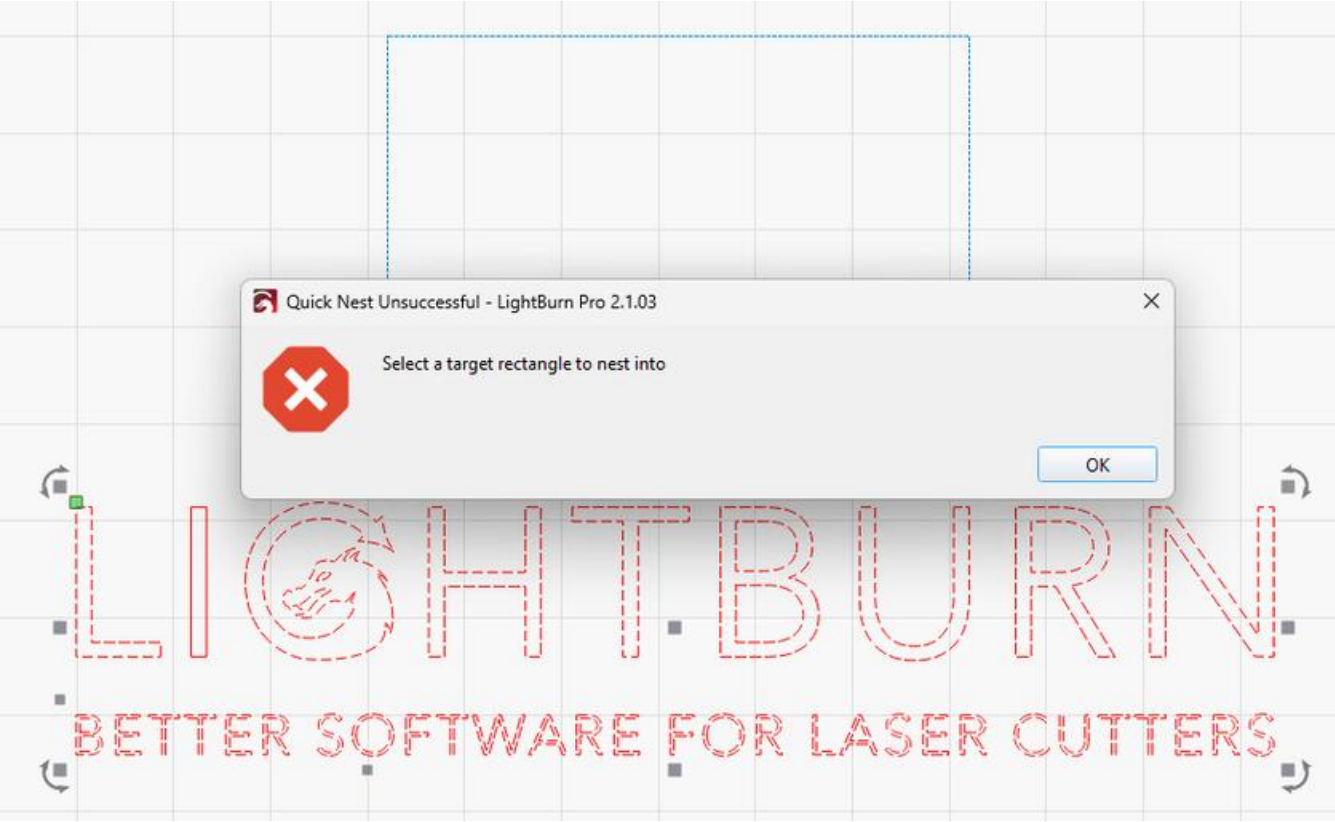
If you get this error, make sure your bounding box is a rectangle and that it's not at an angle.



Select a Target Rectangle to Nest Into

If you get this error message, LightBurn doesn't see a selected rectangle to use as a container.

Fix the error by selecting a rectangle to arrange your parts inside of in addition to the parts to be nested. In the example below, the blue rectangle wasn't selected, so LightBurn has nothing to nest the parts into.



Why is the I in LightBurn not selected?

In the screenshot above, the letter I in LightBurn isn't selected, because selecting it and not the larger blue box results in a different error. The reason for this is a bit technical, but if you're interested, keep reading.

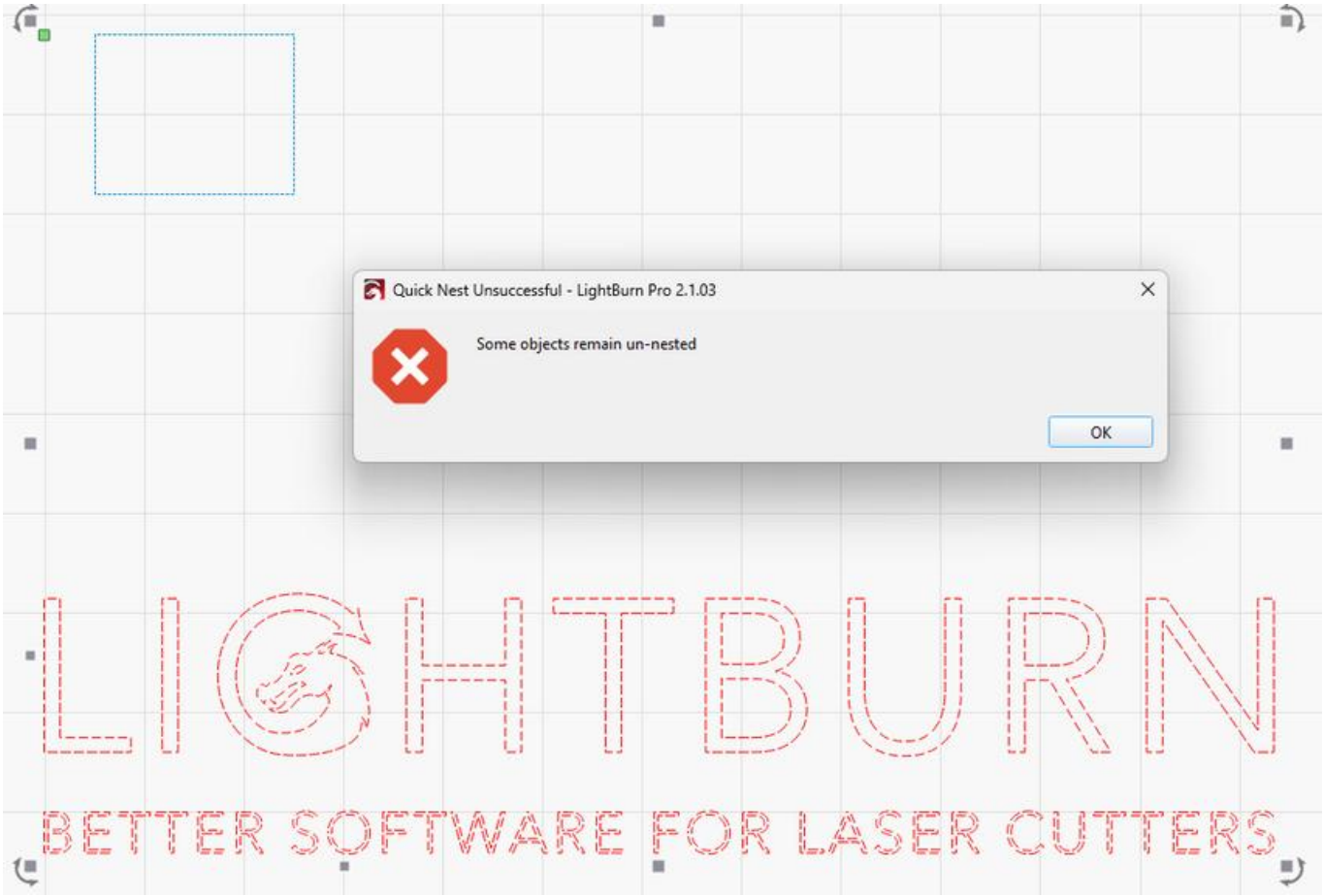
The letters used in the demo file here are actually paths, not text objects, and the letter I would be a valid rectangle to nest into if it were big enough. This isn't a problem when the larger blue rectangle is selected, but when I don't select that rectangle, Quick Nest tries to nest into the I. This actually could have resulted in one of two different error messages:

1. Optimization Yielded Zero Bounding Box: If none of the shapes selected are small enough to nest inside the I, Quick Nest will be unable to create a suitable arrangement and you will receive this error.
2. Some Objects Remain Un-Nested: If there are shapes small enough to be nested into the I, Quick Nest will nest the objects it can, and you will receive this error.

Some Objects Remain Un-Nested

If you get this error message, one or more of the parts you're trying to nest are to big to nest in your selected rectangle with the current settings.

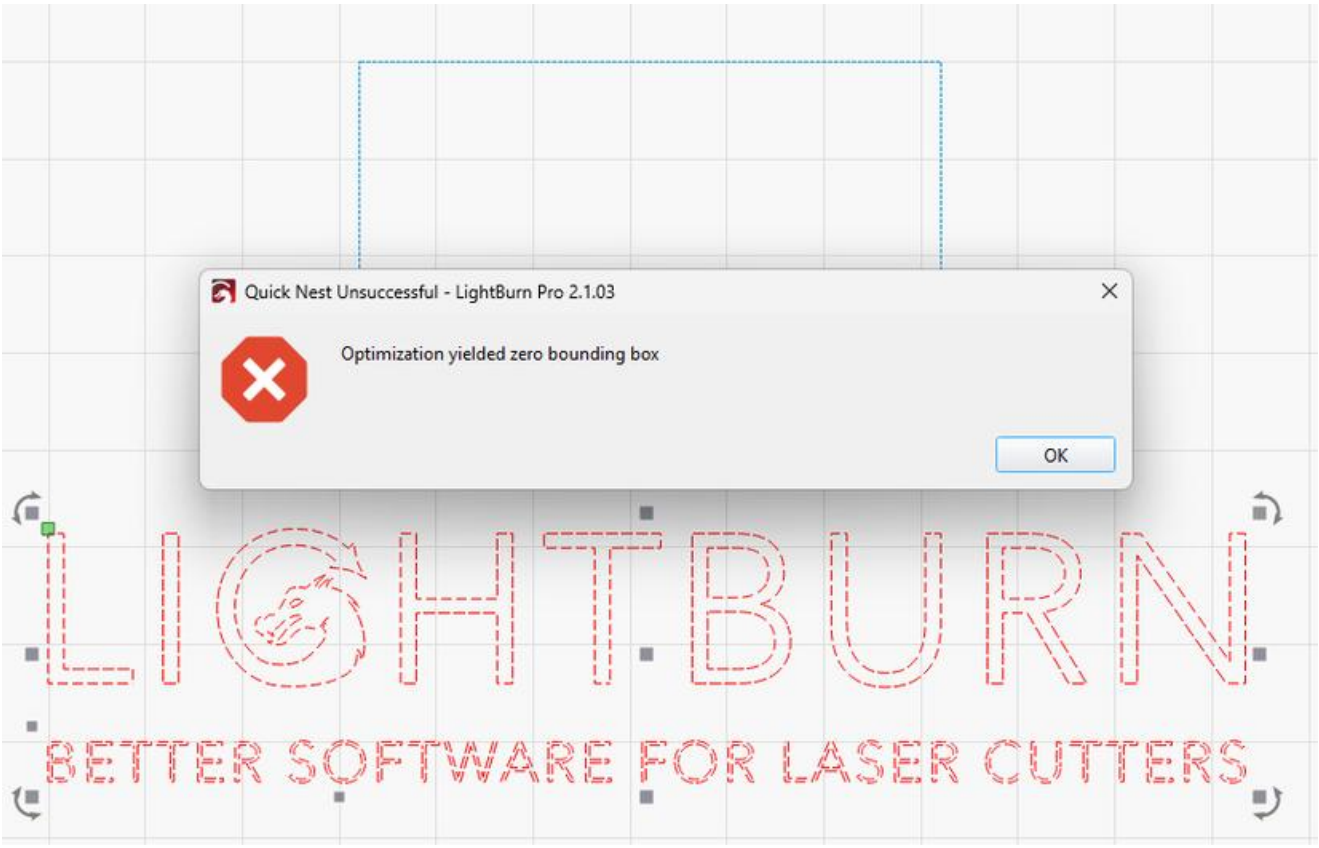
When you press **OK**, LightBurn will still nest as many of the parts as possible, leaving out the ones that can't be nested. If your target rectangle is big enough to nest your largest part, you will not see this error message.



Optimization Yielded Zero Bounding Box

If you get this error message, it means **True Nest** is unable to create a suitable arrangement at all.

In certain circumstances, if you don't select a rectangle to nest into you can get this error message instead of Select a Target Rectangle to Nest Into. In the screenshot below, the blue rectangle isn't selected, but the red I, which is actually a rectangle, is selected. Quick Nest tries to nest into the I and fails, resulting in the Zero Bounding Box error. More about how this happens.

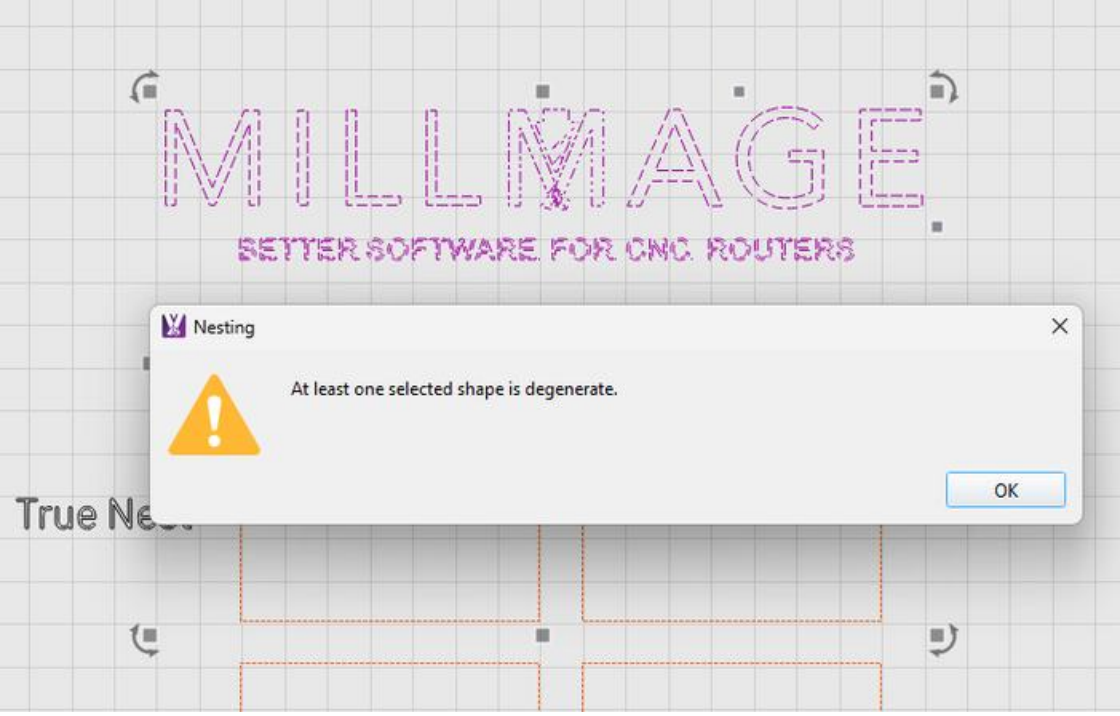


At Least One Selected Shape is Degenerate

If you get this warning message, it means at least one of the shapes you're trying to nest is degenerate, or invalid.

> **Degenerate Shapes?**
>
> Without getting into a full scale math lesson, a degenerate shape is a special case of a shape. Examples might include a line with a length of 0, or a triangle where all 3 points are on the same line.

When you press **OK**, LightBurn will proceed to nesting all valid shapes.



Related Topics

- SVGNest (External)
- Quick Nest
- Docking
- Align
- Distribute
- Move Together
- Move Selected Objects

Quick Nest

Nesting enables you to arrange objects to use less material. LightBurn 2.2 offers you a choice between **True Nest** (LightBurn Pro only), **Quick Nest**, and **SVGNest (External Tool)**.

	True Nest	Quick Nest	SVGNest (External Tool)
LightBurn release	LightBurn 2.2 Pro	LightBurn 2.1	LightBurn 1.1
Nests based on true shape of objects	□	×	□
Fully integrated in LightBurn	□	□	×
Works locally, internet not required	□	□	×
Read more	True Nest	Quick Nest	SVGNest (External)

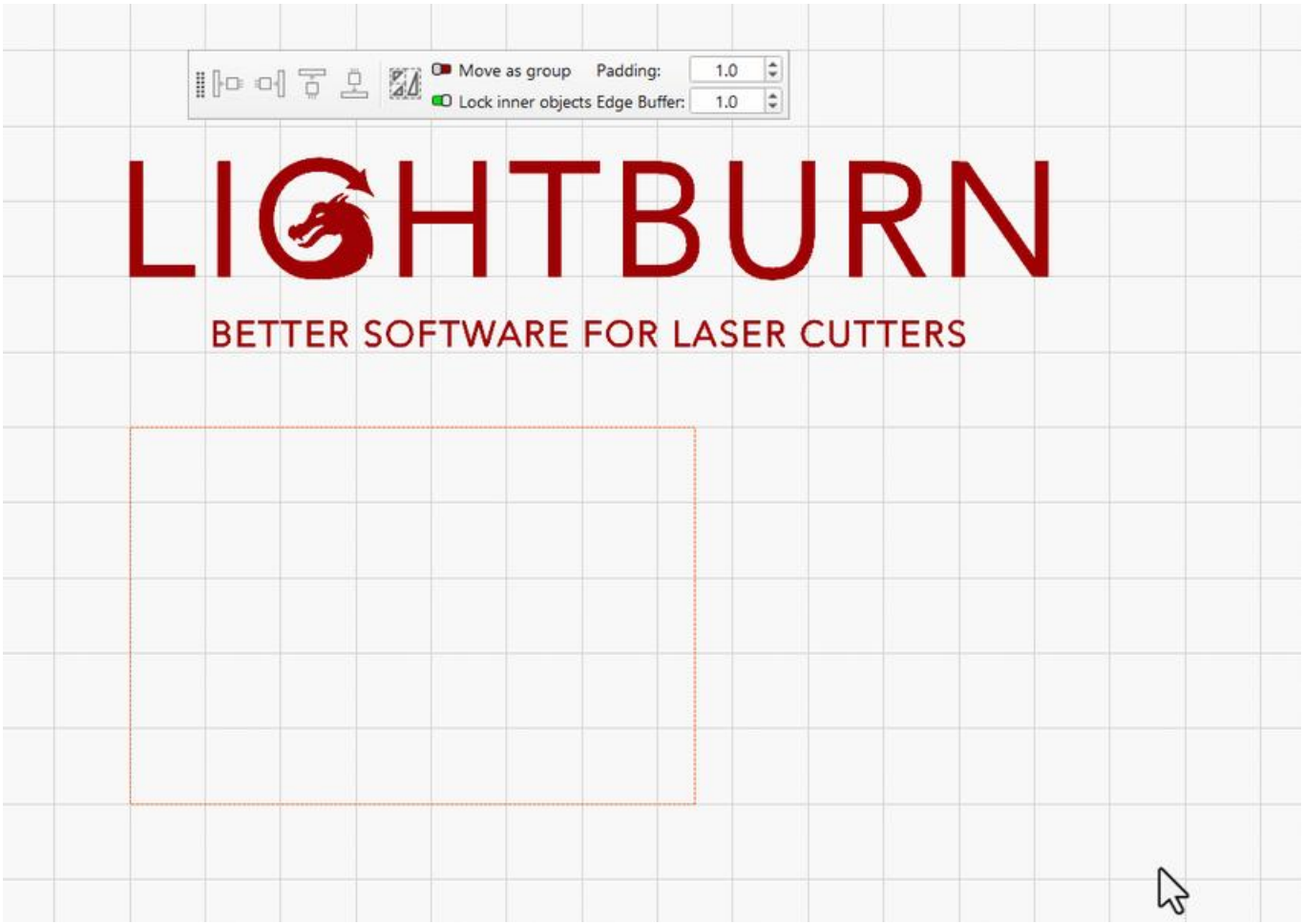
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=vjcYOBs3j-c

Using Quick Nest

You can find both Quick Nest and True Nest in the **Docking Toolbar** or by going to Arrange → **Nesting**.

> **Accessing the Docking Toolbar**
>
> By default the Docking Toolbar is in the top right corner of the screen. Depending on your screen size, you may see a short version like in the screenshots below, with an arrow to the right to expand the toolbar. For the rest of this page, we'll use the full version of the toolbar in screenshots.
>
> To see all the options, you can expand the toolbar with the arrows to the right, drag it to the left to give it more space, drag it to another toolbar with more space, or drag it out of the toolbar area entirely. All four are shown below.

More About the Docking Toolbar



Getting Started

1. Create a rectangle to use as a container for the shapes you want to nest.
2. Select the container and all of the objects you want to nest.
3. Open the **Nesting Dialog** by clicking the **Nesting** button or going to **Arrange** → **Nesting**.
4. Select **Quick Nest** and adjust your Nesting options.
5. Click **OK** to nest your shapes.

Nesting Dialog

Mode

Choose between **Quick Nest** and **True Nest**.

Options

These options are shared between **Quick Nest** and **True Nest**.

Padding

Sets the minimum distance between the parts to nest. The units are either millmeters or inches, depending on your Design Units. Increase this number to leave more space between parts.

Edge Buffer

Sets the minimum distance between the parts to nest and the edge of the container you're nesting in. The units are either millmeters or inches, depending on your Design Units.

Lock Inner Objects

If **Lock Inner Objects** is enabled, inner objects move with the objects surrounding them, whether or not they're grouped. In the demonstration at the top of the page, **Lock Inner Objects** keeps the inner shapes of the letters and the dragon's eye in place while nesting without having to group them.

Lock Objects for Rotation

Enable this to lock the rotation of objects being nested. Disable this to allow LightBurn to rotate the objects as part of the nesting process. **Quick Nest** rotates in 90 degrees increments if this is enabled. **True Nest** offers more rotation options in a secondary dialog.

Allowing rotation can improve material usage, but may not work well for all designs, particularly when cutting parts from material with a visible grain.

Multipage

With **Multipage** enabled, if you try to nest more objects than fit in your target container, LightBurn will create additional pages with a copy of the original container to continue the nesting process. With **Multipage** disabled, LightBurn will nest as many objects as it can and leave the rest unnested.

Quick Nest Limitations

Quick Nest uses bounding box nesting. It identifies the bounding box of each object — the smallest possible rectangle that can fully contain it — and then packs those bounding boxes as tightly as possible into the container you choose.

The great thing about Quick Nest is that it's fast, convenient, and powerful enough for most use cases. However, because it nests according to the bounding box and not the actual shape, complex shapes can't nest into each other as densely as they can with True Nest.

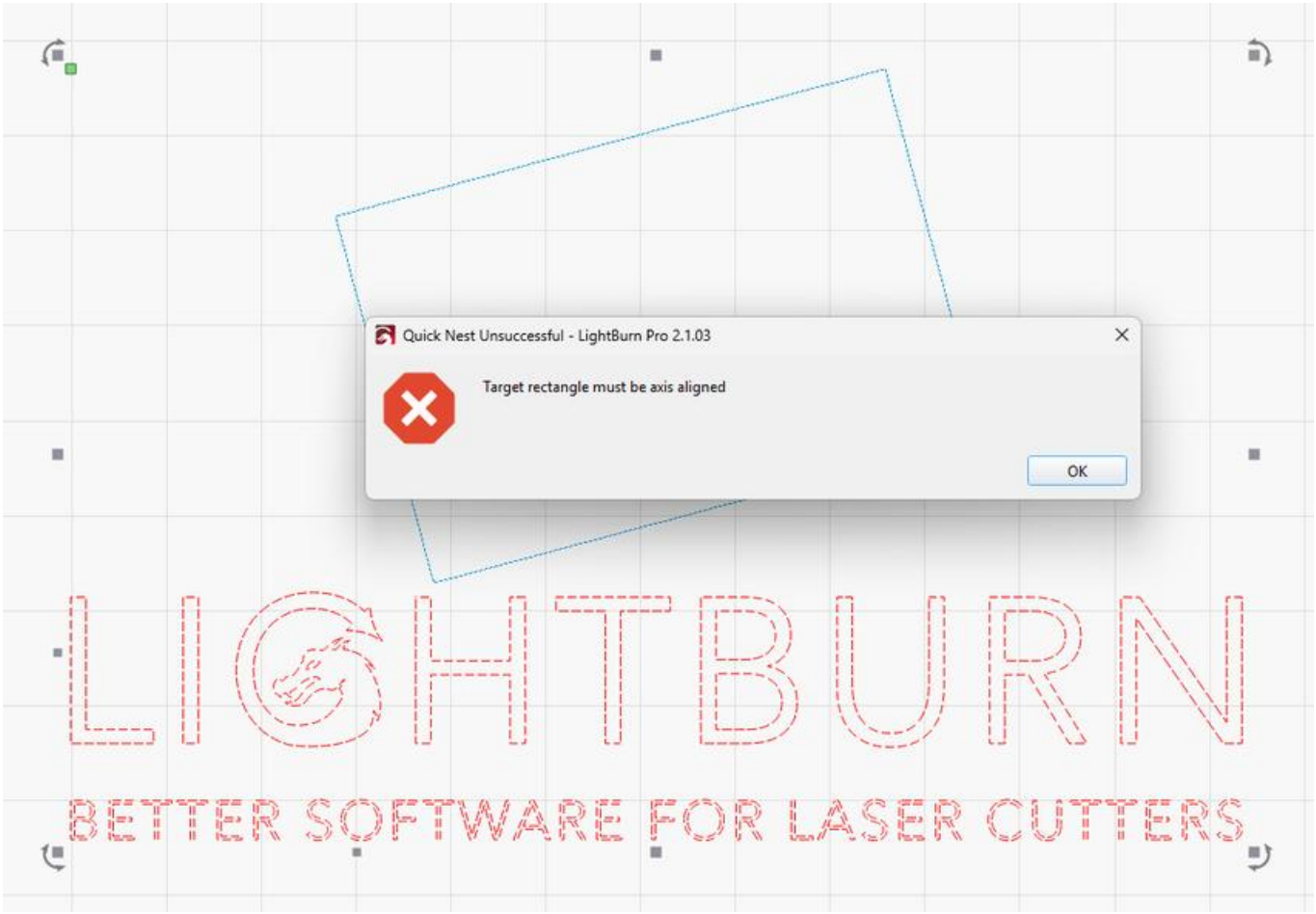
If you want denser packing, please take a look at True Nest, which is available in LightBurn Pro 2.2 and newer. If you're on an older version of LightBurn or are using LightBurn Core, you can also check out SVGNest (External), which partially automates the process of exporting your shapes to SVGnest.com and re-importing those shapes back into LightBurn.

Nesting Errors and Warnings

Target Rectangle Must be Axis Aligned

To use **Quick Nest**, the rectangle you're nesting into must have sides that are straight up and down and sideways.

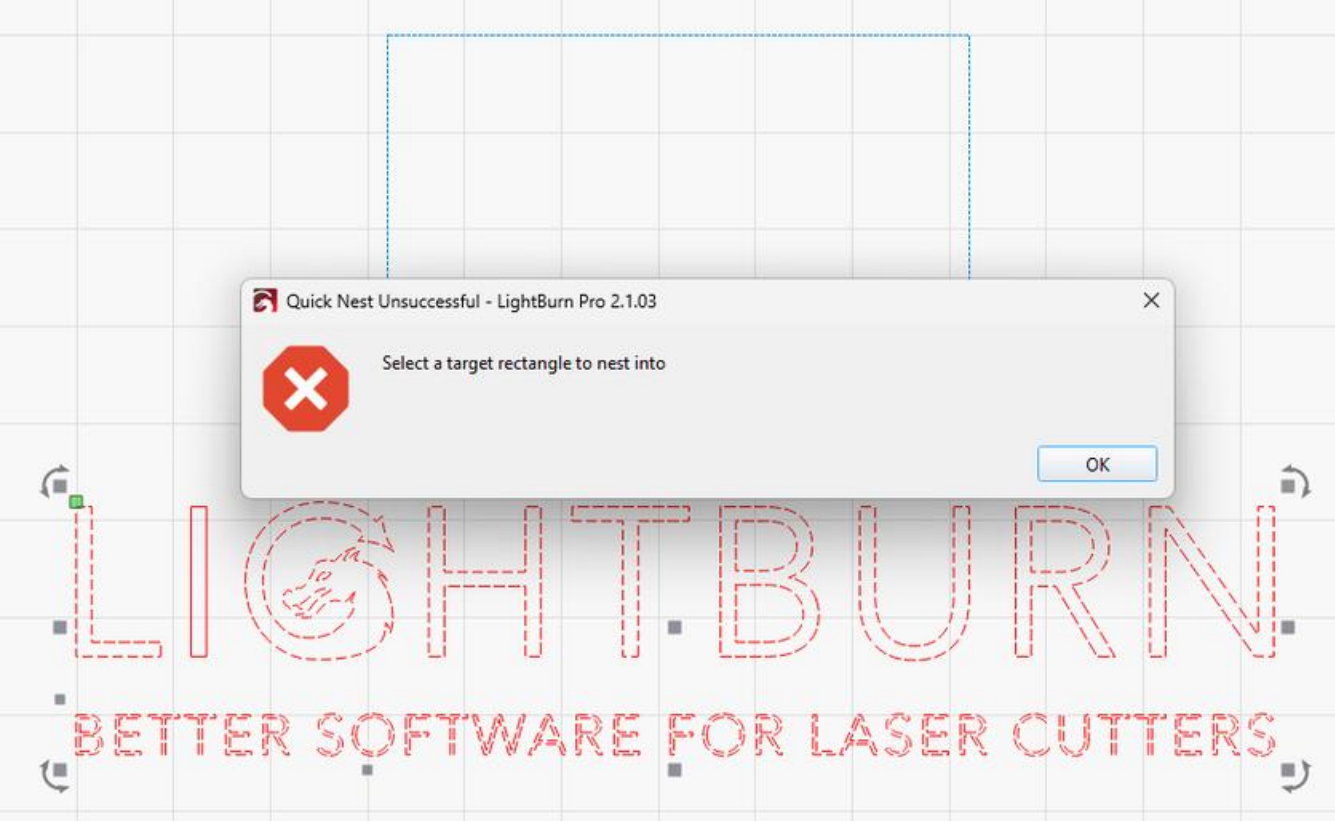
If you get this error, make sure your bounding box is a rectangle and that it's not at an angle.



Select a Target Rectangle to Nest Into

If you get this error message, LightBurn doesn't see a selected rectangle to use as a container.

Fix the error by selecting a rectangle to arrange your parts inside of in addition to the parts to be nested. In the example below, the blue rectangle wasn't selected, so LightBurn has nothing to nest the parts into.



Why is the I in LightBurn not selected?

In the screenshot above, the letter I in LightBurn isn't selected, because selecting it and not the larger blue box results in a different error. The reason for this is a bit technical, but if you're interested, keep reading.

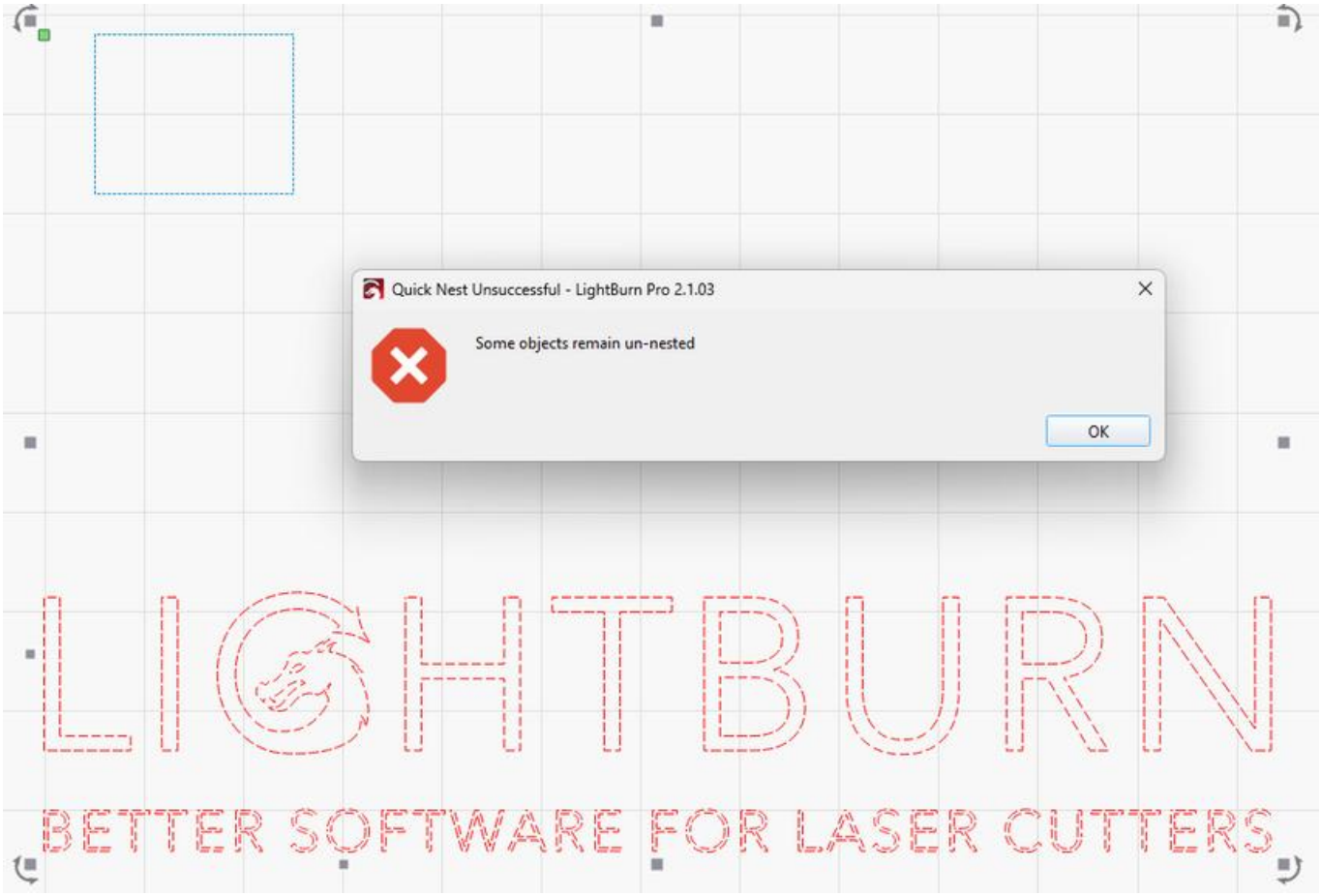
The letters used in the demo file here are actually paths, not text objects, and the letter I would be a valid rectangle to nest into if it were big enough. This isn't a problem when the larger blue rectangle is selected, but when I don't select that rectangle, Quick Nest tries to nest into the I. This actually could have resulted in one of two different error messages:

1. Optimization Yielded Zero Bounding Box: If none of the shapes selected are small enough to nest inside the I, Quick Nest will be unable to create a suitable arrangement and you will receive this error.
2. Some Objects Remain Un-Nested: If there are shapes small enough to be nested into the I, Quick Nest will nest the objects it can, and you will receive this error.

Some Objects Remain Un-Nested

If you get this error message, one or more of the parts you're trying to nest are to big to nest in your selected rectangle with the current settings.

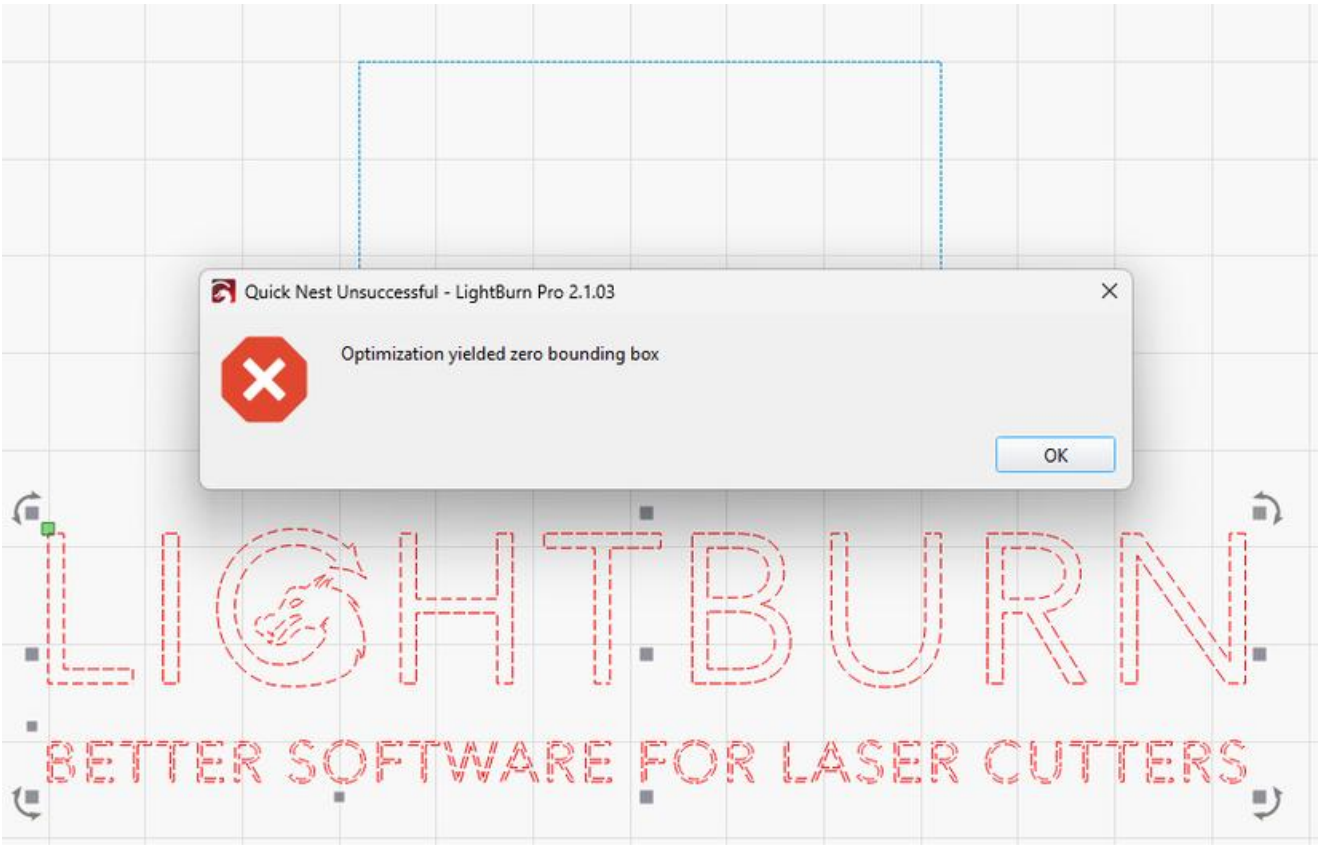
When you press **OK**, LightBurn will still nest as many of the parts as possible, leaving out the ones that can't be nested. If your target rectangle is big enough to nest your largest part, you will not see this error message.



Optimization Yielded Zero Bounding Box

If you get this error message, it means **Quick Nest** is unable to create a suitable arrangement at all.

In certain circumstances, if you don't select a rectangle to nest into you can get this error message instead of Select a Target Rectangle to Nest Into. In the screenshot below, the blue rectangle isn't selected, but the red I, which is actually a rectangle, is selected. Quick Nest tries to nest into the I and fails, resulting in the Zero Bounding Box error. More about how this happens.

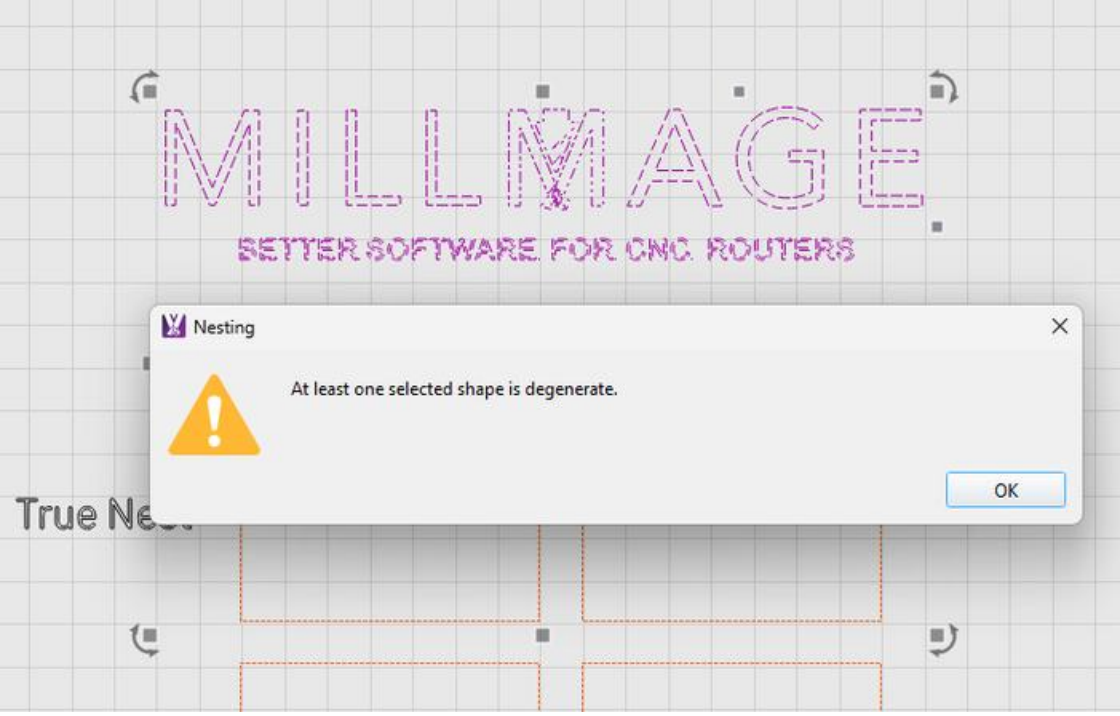


At Least One Selected Shape is Degenerate

If you get this warning message, it means at least one of the shapes you're trying to nest is degenerate, or invalid.

> **Degenerate Shapes?**
>
> Without getting into a full scale math lesson, a degenerate shape is a special case of a shape. Examples might include a line with a length of 0, or a triangle where all 3 points are on the same line.

When you press **OK**, LightBurn will proceed to nesting all valid shapes.



Related Topics

- SVGNest (External)
- Docking
- Align
- Distribute
- Move Together
- Move Selected Objects

SVGNest (External)

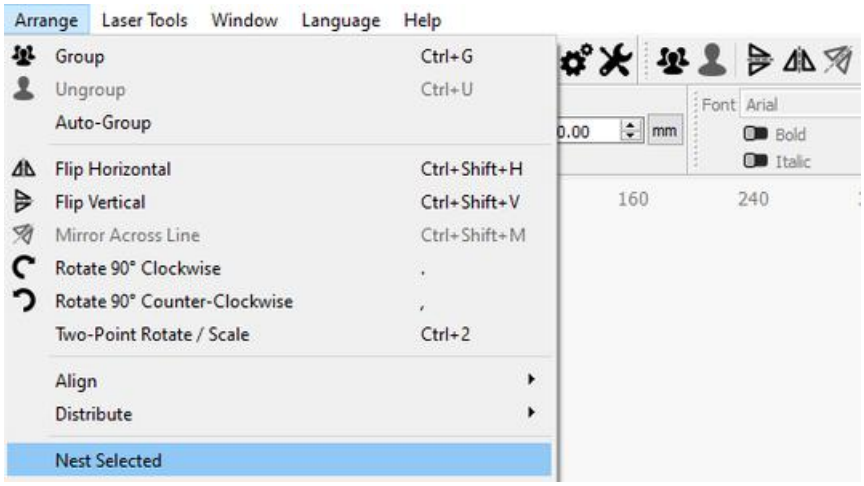
Nesting helps you arrange objects to use material more efficiently. LightBurn offers three options for nesting. This page is about LightBurn's integration with an external tool called **SVGNest**. LightBurn also has two built-in nesting tools: True Nest and Quick Nest. The **SVGNest** tool will automatically create and export an SVG of selected graphics, upload it to SVGnest.com, and re-import it after it nests the graphics contained in the file.

	True Nest	Quick Nest	SVGNest (External Tool)
LightBurn release	LightBurn 2.2 Pro	LightBurn 2.1	LightBurn 1.1
Nests based on true shape of objects	☐	×	☐
Fully integrated in LightBurn	☐	☐	×
Works locally, internet not required	☐	☐	×
Read more	True Nest	Quick Nest	SVGNest (External)

Using SVGNest

1. Select any graphics you'd like to nest along with a container shape — usually a shape that has the same dimensions as the material you are going to cut the nested graphics out of — then go to the Arrange Menu and click **SVGNest (External Tool)**.

> **Important Note on Inner Shapes**
>
> LightBurn does not export graphics that are contained inside another graphic — this means all of the graphics you'd like to nest must be outside the container shape you choose.
>
> Once your graphics have been nested within the container shape and re-imported, LightBurn will automatically move any shapes that were contained within them so they maintain their relative position.



2. Your default web browser will automatically open to SVGnest.com, and a dialog window will open in LightBurn, informing you that an SVG has been created and its file name (LightBurnNest.svg, along with a file path), has been copied to your clipboard.



3. On SVGnest.com, click **Upload SVG**. Paste the automatically copied name into the file browser, or navigate to the location the SVG saved to and select it.
4. Select the container shape, then click **Start Nest**.
5. When you are satisfied with the resulting nest, or the process is complete, click **Stop Nest**, then **Download SVG**. Save the nested SVG.
6. Back in LightBurn, click **OK** in the dialog box. Your file browser will open — navigate to the location of the SVG you downloaded from SVGnest, and import it into LightBurn.

> **Note**
>
> LightBurn will automatically delete the LightBurnNest.svg file it created and exported after you import the new file from SVGnest.

Related Topics

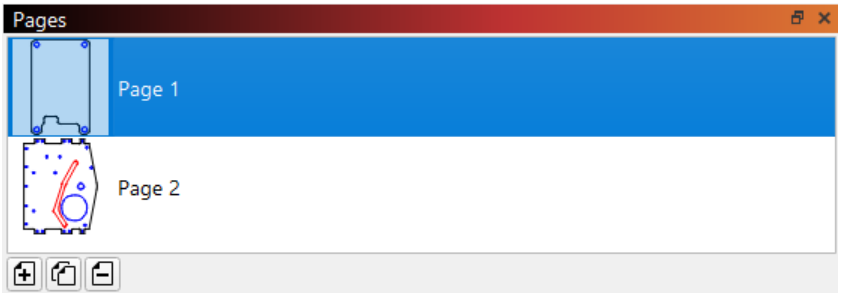
- Quick Nest
- Docking
- Grid Array
- Align
- Distribute
- Move Together
- Move Selected Objects

Pages Window

LightBurn's **Pages** feature allows you to create multiple pages within the same file. This can help with creating layouts that go across multiple physical sheets of material, or working with multiple related designs that use the same settings.

LightBurn's nesting features can also be set to nest across multiple pages.

Pages Overview



The **Pages** window contains a preview of all the pages in your current project and three buttons: add, duplicate, and delete.

The shapes contained on each page are separate, but cut settings and other file settings are shared across pages.

Accessing the Pages Window

The **Pages** window appears in the top right corner of the window by default, but if it's closed, you can reopen it by going to **Window** → **Pages**.

To restore the Pages window and all other windows to their default positions, got to **Window** → Reset to Default Layout

> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Pages Options and Settings

Page Preview

The preview area of the **Pages** window shows a thumbnail of each page along with the name of that page.

Right-click in this area to open a context menu allowing you to add, duplicate, delete, or rename pages.

Add Page

Click this button to add a new blank page to your project.

Duplicate Page

Click this button to duplicate the selected page - this creates a new page containing copies of all the objects you have on the selected page.

Delete Page

Click this button to delete the selected page.

Rename Page

To rename a page, either right-click in the Pages window and choose **Rename Page**, or double-click on the page in the Pages window to open the Rename Page dialog. Either way, enter the new name and press **OK**.

Nesting Onto Multiple Pages

When using LightBurn's Nesting tools, use the **Multisheet** option to have LightBurn automatically create additional pages when there are overflow objects beyond the first nesting container.

Change Draw Order

The draw order is an internal list of objects in your Workspace. By default, all objects are ordered according to when they were created or added to your project — the last object added to your project is the first in the draw order, the second to last added is the second in the draw order, and so on.

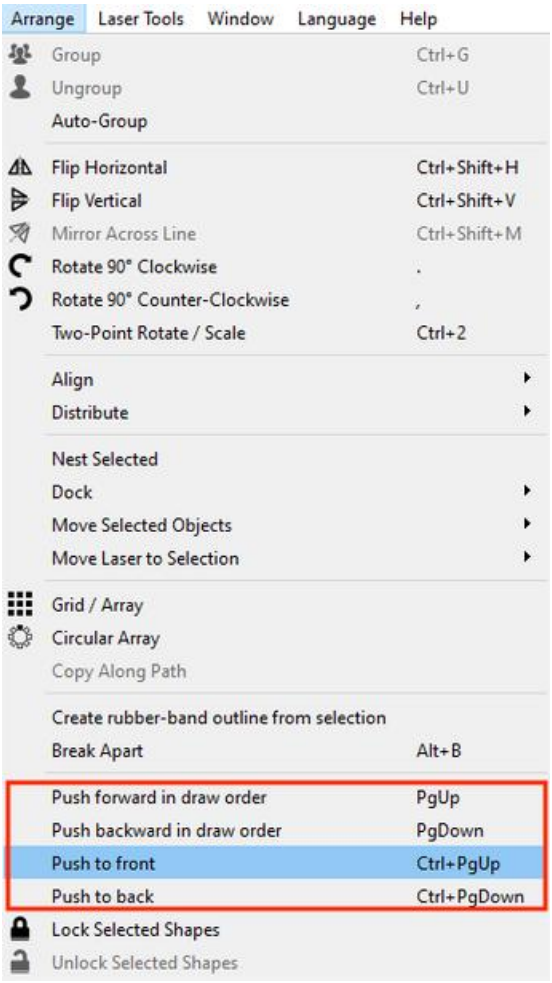
The **Push** options change the order of objects in the draw order. This is primarily useful when working with images and vector graphics. Vector graphics may end up hidden behind Images, making them difficult to manipulate — Push them forward to get them back in view.

> **Note**
>
> Changing the draw order only affects the display order when in Wireframe rendering mode, and not Filled. In Filled rendering mode, the display order will always match the layer order.

Push Tools and Hotkeys

All **Push** options are available in the Arrange Menu or by hotkey

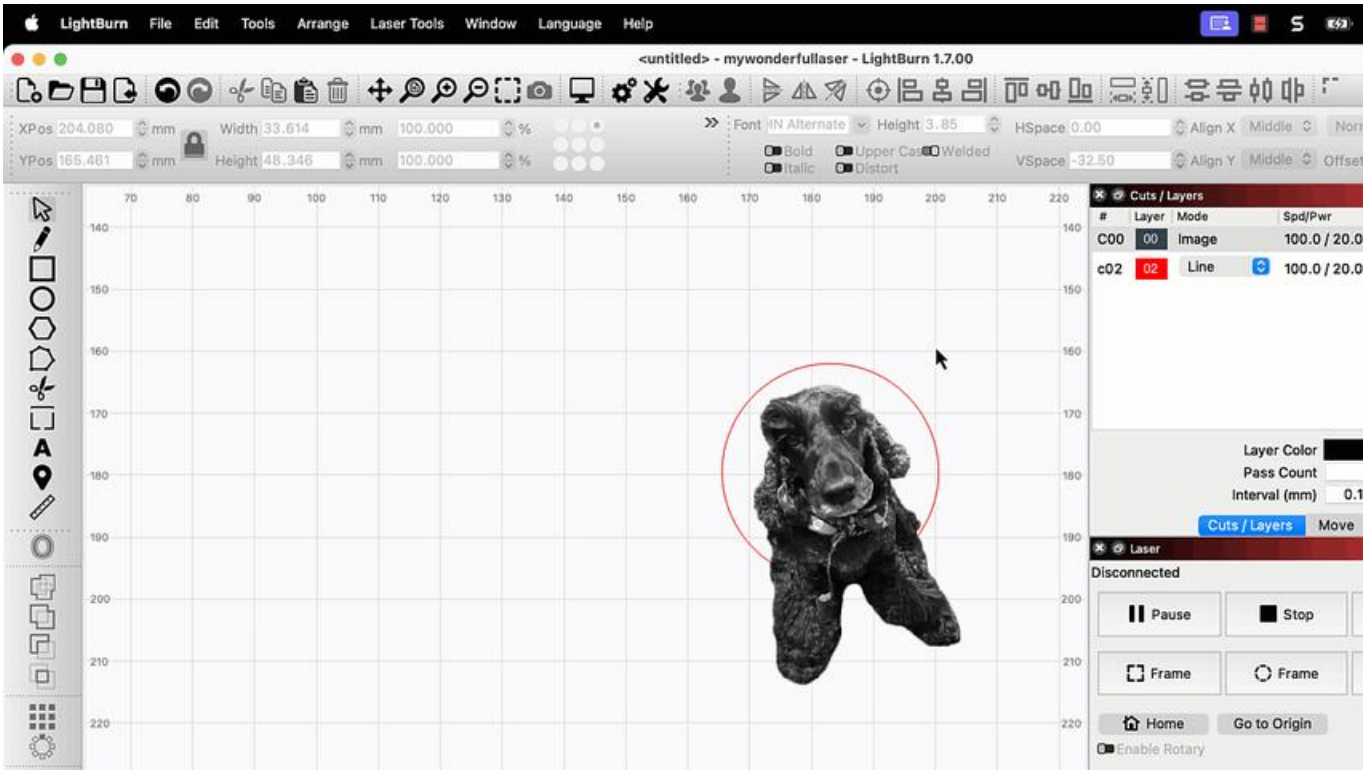
Push Option	Action	Hotkey
Push forward in draw order	Moves selected objects up one position in the draw order	`PgDwn`
Push backward in draw order	Moves selected objects up one position in the draw order	`PgUp`
Push to front	Moves selected objects to the first position in the draw order	`Ctrl`/`Cmd` + `PgUp`
Push to back	Moves selected objects to the last position in the draw order	`Ctrl`/`Cmd` + `PgDwn`



Using the Push Tools

Select an object and one of the **Push** options to move the selected object in the draw order.

If you have more than one object selected, they'll each move in the draw order, but maintain their position in the order relative to one another — in other words, if you Push the third and fourth objects in your draw order to the front, they'll become the first and second objects.



Related Topics

- View Style
- Cuts / Layers Window
- Align Tools Priority

Lock Shapes

The **Lock Shapes** tool locks selected objects in your design, preventing any form of editing (including moving, resizing, deleting, and more).

Use Lock Shapes to prevent unintentional design edits, secure graphics used for alignment, preserve templates, and restrict editing in collaborative projects.

Locking and Unlocking Objects

Lock selected objects by toggling the **Locked** state on in the Shape Properties Window, by going to Arrange → **Lock Shapes**, or by right-clicking the objects and choosing **Lock Selected Shapes**.

To Unlock and enable editing of a Locked object, toggle the Locked state off in the Shape Properties Window, go to **Arrange** → **Unlock Shapes**, or right-click the objects and choose **Unlock Selected Shapes**.

> **Tip**
>
> Use the Transform Control Toggles to restrict specific types of editing for the whole project.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=KUDsm1_v5Rk

Related Topics

- Transform Controls
- Transform Control Toggles
- Shape Properties

Snapping

LightBurn's automatic **Snapping** tools help align objects to other shapes in your Workspace, or ..to the Workspace grid itself — when you move objects close to a valid location, they will snap to that point.

As you hover your cursor over an object, the icon will change to indicate when it is over a node, line, midpoint, center, or intersection. Use these Snapping points when selecting, moving, or creating objects to ensure exact alignment.

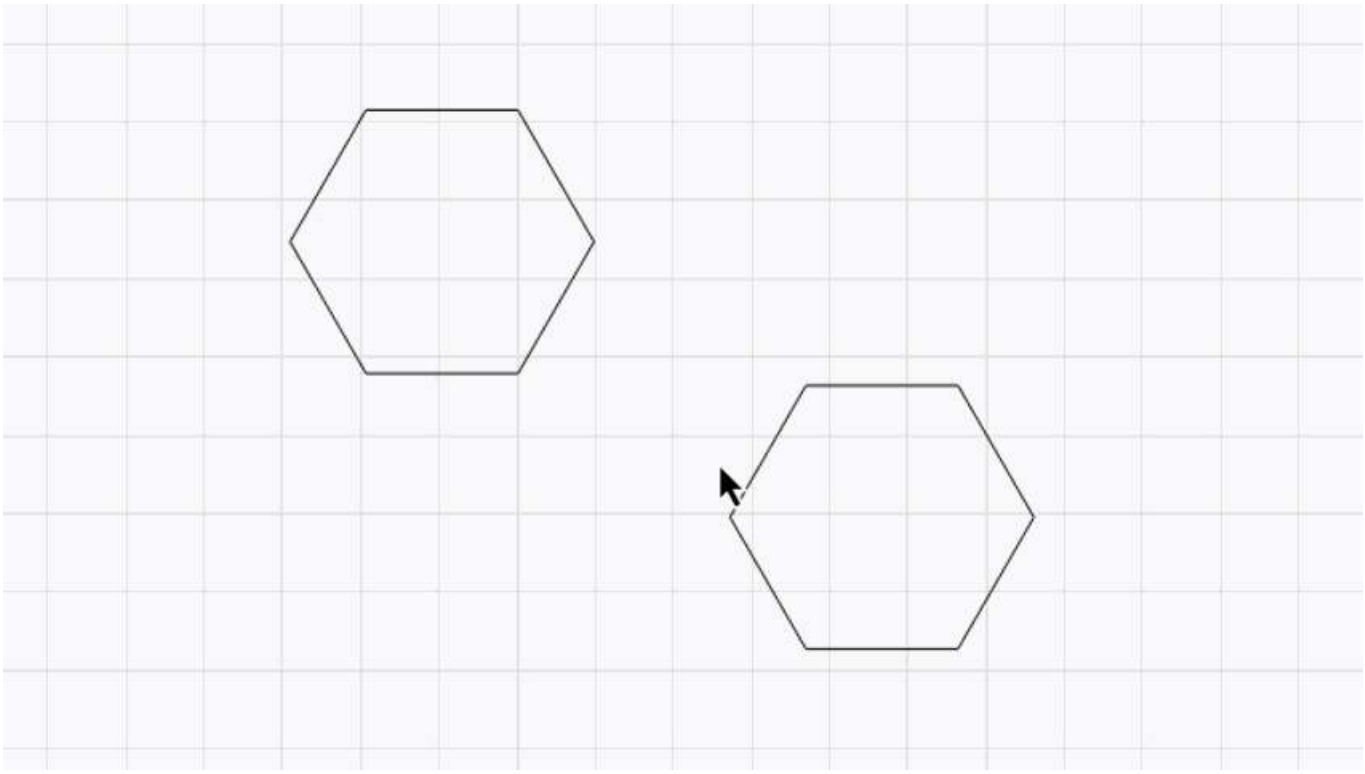
Object Snapping

Object Snapping Points

Snapping Point	Icon	Description
Node		The cursor is over any node in a vector graphic, or the corner of an image.
Midpoint		The cursor is over the exact halfway point between two nodes.
Center		The cursor is over the center point of an object.
Intersection		The cursor is over the intersection of two lines.
Line		The cursor is over a line, but not a node, midpoint, or intersection on that line.

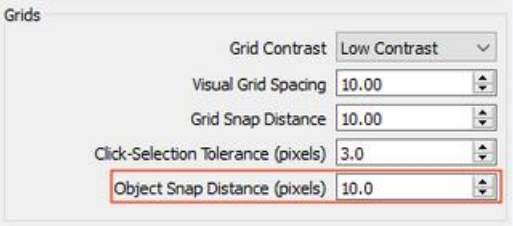
Moving to Snapping Points

When you move objects near enough to a **Snapping** point, they'll automatically align to that point, and the cursor will change to indicate the type of Snapping point they've aligned to.



Note

You can adjust how close your cursor must be to a point before Snapping behavior is activated in the **Units and Grids** tab of the Settings window. This value is set in pixels — when your cursor is within that distance from a valid point, it will automatically snap to it, and change appearance to indicate the type of point it is over.

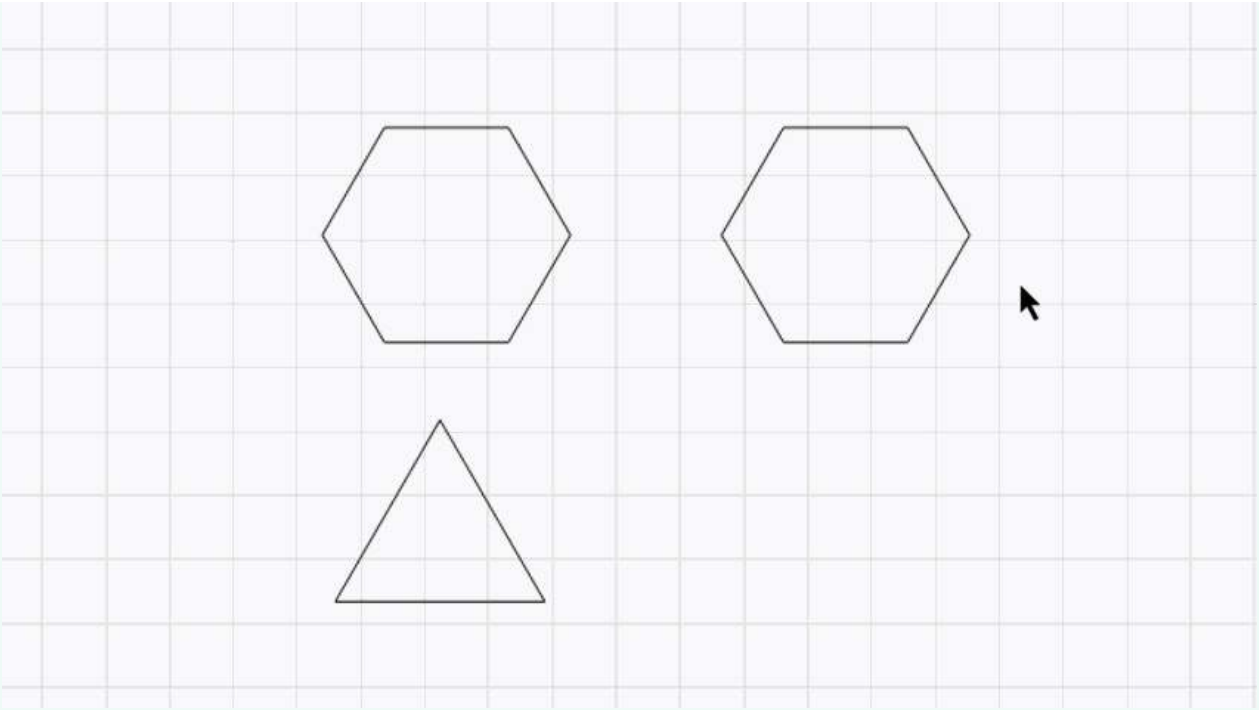


Selecting from Snapping Points

You can select and move objects directly from **Snapping** points by clicking and dragging from a location where the cursor has changed to one of the Snapping icons.

- To move multiple objects at once from a Snapping point on just one object, you'll need to select them all at once, or Group them first.

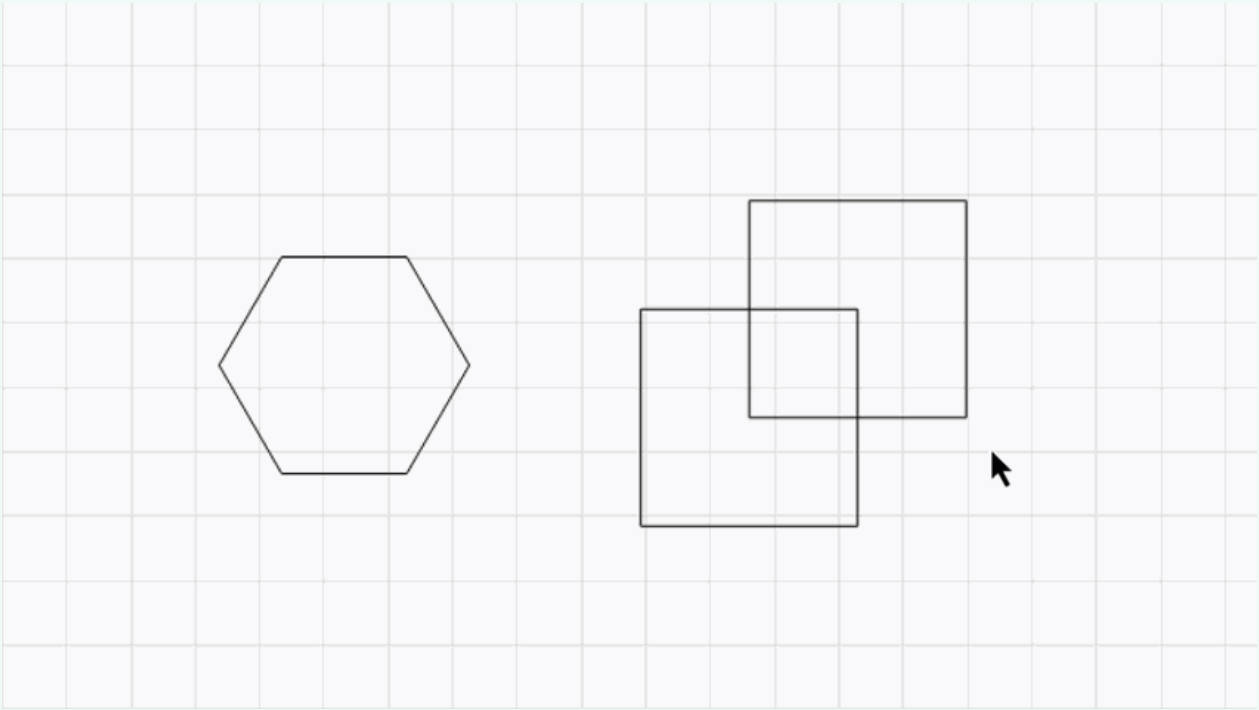
  Snapping multiple objects at once



- To move objects from a center point when using a Wireframe View Style, you'll need to select them before hovering your cursor over the center of the selection.
- The center of objects that are already selected is indicated by the four-arrow icon



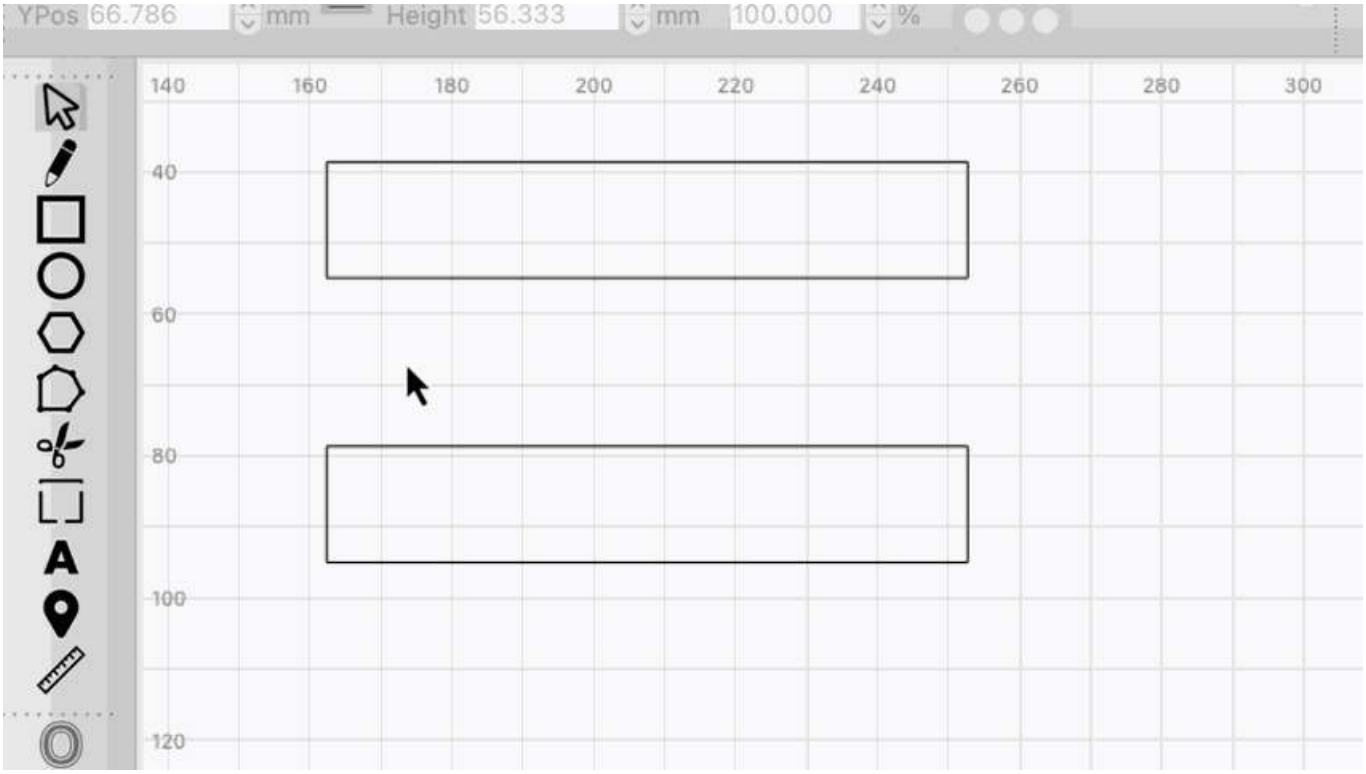
Selecting from a center point



Creating From Snapping Points

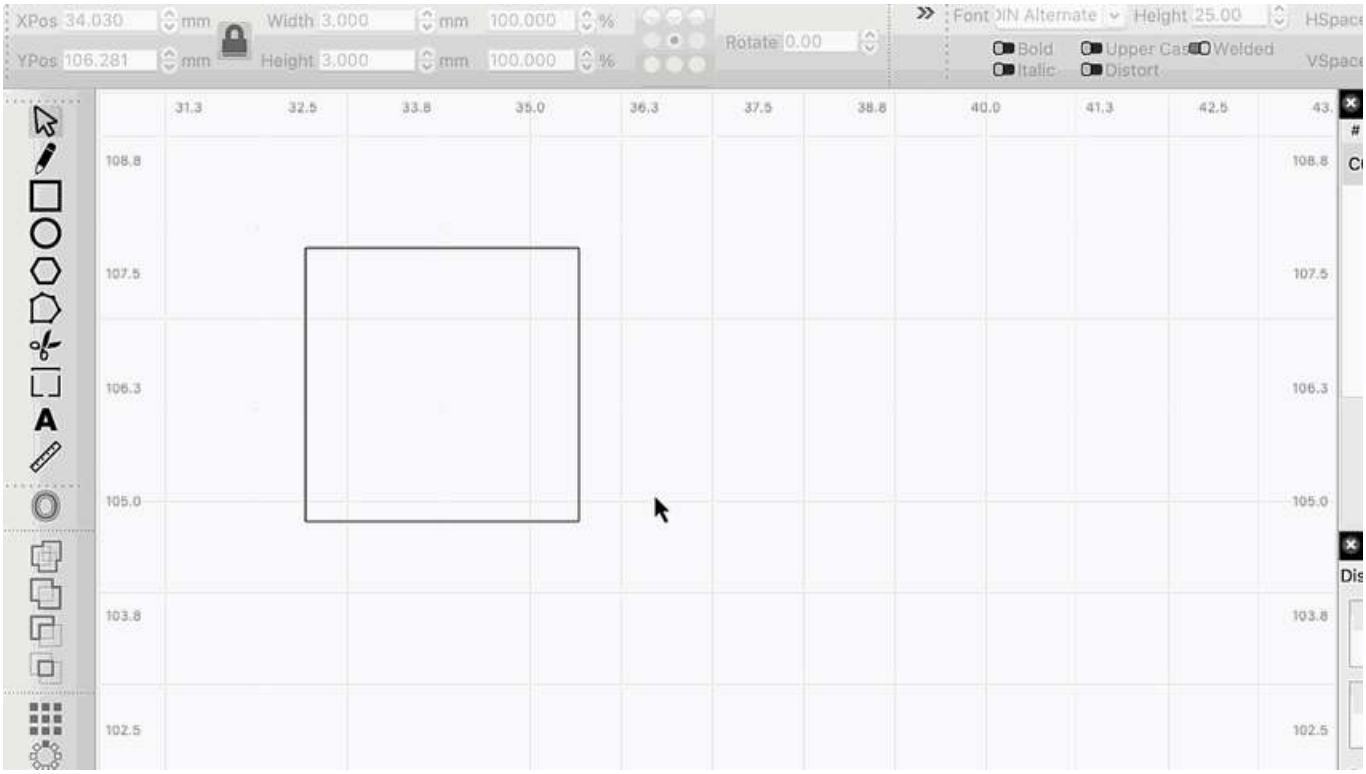
When creating new objects using the Draw Lines tool or any of the Primary Shapes tools, your cursor will change to indicate when it is over a **Snapping** point.

- Click and drag from any of those points to start a new line or shape at that point.
- When you drag near a Snapping point, the cursor will automatically align to the point, and change to indicate the type of point it is aligned over.
  - If you're using the Draw Lines tool, click again to place a node at that point.
  - If you're using a Primary Shape creation tool, release to finish creating the shape.



Snap to Grid

Grid Snapping automatically aligns your cursor to nearby grid corners when you're dragging objects or creating new ones.



Note

Grid Snapping points are determined by the Grid Snap Distance setting in the Settings window, and not the Visual Grid Spacing distance.

- Grid Snap Distance determines the size of the grid squares whose corners your cursor will snap to — for example, if your units are set to millimeters, and your Grid Snap Distance is set to 1, your cursor will snap to the corners of a grid of 1 mm x 1 mm boxes.
- Visual Grid Spacing determines the visual size of grid squares at the standard zoom level. As you Zoom In, you'll see a secondary 10 x 10 grid of squares within each larger square. Each smaller square is 1/10th the size of the Visual Grid Spacing value.



Keyboard Modifiers

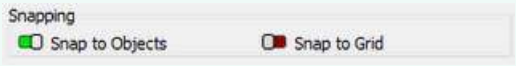
- Hold `Alt`/`Option` while dragging to make **Object Snapping** "stickier", meaning objects will automatically align to Snapping points from further away.

Tip

> If you're moving an object or creating a shape, this will also enable Alignment Guides — temporary visual indicators that show when an object's edge or center is aligned to an edge or center of another object in your Workspace.

- Hold `Ctrl`/`Cmd` while dragging to toggle either type of Snapping on or off for as long as the key is held down.

> **Tip**
>
> By default, this disables both types of Snapping. However, if you've disabled Snap to Objects or **Snap to Grid** in the Settings window, it will instead enable that type of Snapping for as long as the key is held down.



Related Topics

- Automatic Guidelines
- Align Tools
- Docking
- Edit Nodes

Automatic Guidelines

LightBurn features two types of automatic guidelines to help you align objects relative to one another, or along vertical or horizontal planes.

Alignment Guides are temporary visual guides that indicate when an object's edge or center is aligned to an edge or center of another object in your Workspace.

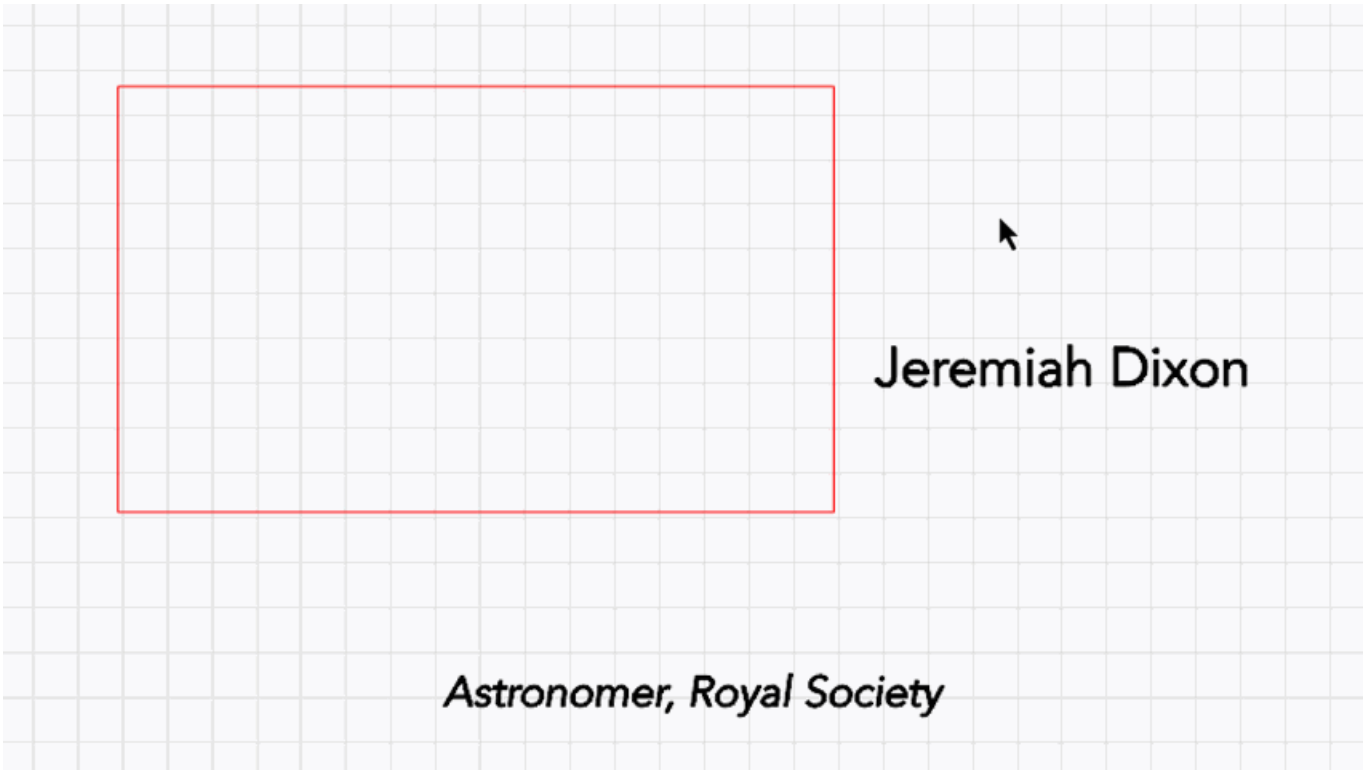
Draggable Guidelines are persistent lines set to the T1 Tool layer — they are perfectly vertical or horizontal by default, and you can snap other objects to them.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=TOF4luuVVTE

Alignment Guides

To enable **Alignment Guides**, hold `Alt`/`Option` while dragging a selected object or objects in your Workspace.

As you move the selection around, temporary visual guides will appear, indicating when an edge or midpoint of your selection has Snapped into alignment with an edge or midpoint of another object.

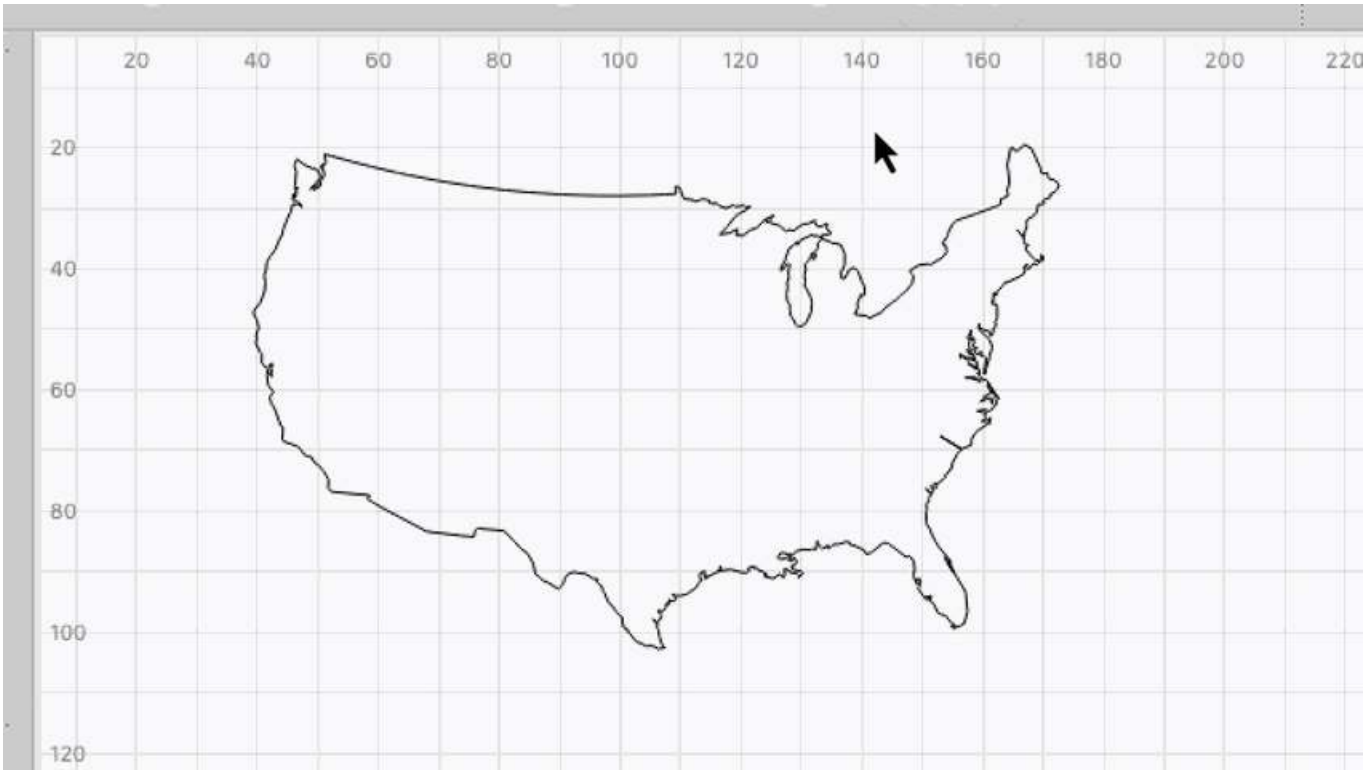


Draggable Guidelines

To create **Draggable Guidelines**, hover over the ruler to the top or left of the Workspace until the cursor changes to the line icon, then click and drag. Release when you've placed the Guideline where you want it — you'll also be able to adjust its position later.

Dragging from the top creates a perfectly horizontal guideline, and dragging from the left creates a perfectly vertical guideline.

These guidelines are set to the T1 Tool layer, and are treated like any other object in LightBurn. You can move them around, snap other objects to them, or Lock them to prevent accidental movement.



Warning

Since they're set to a Tool layer, Draggable Guidelines will never be output to your laser, but if you're using **Current Position** or **User Origin** as your Start From mode, you'll need delete them first, or toggle off the **Frame** option in the Cuts / Layers Window to exclude them when calculating **Origin**.

Related Topics

- Snapping
- Align Tools
- Docking
- Distribute and Move Together

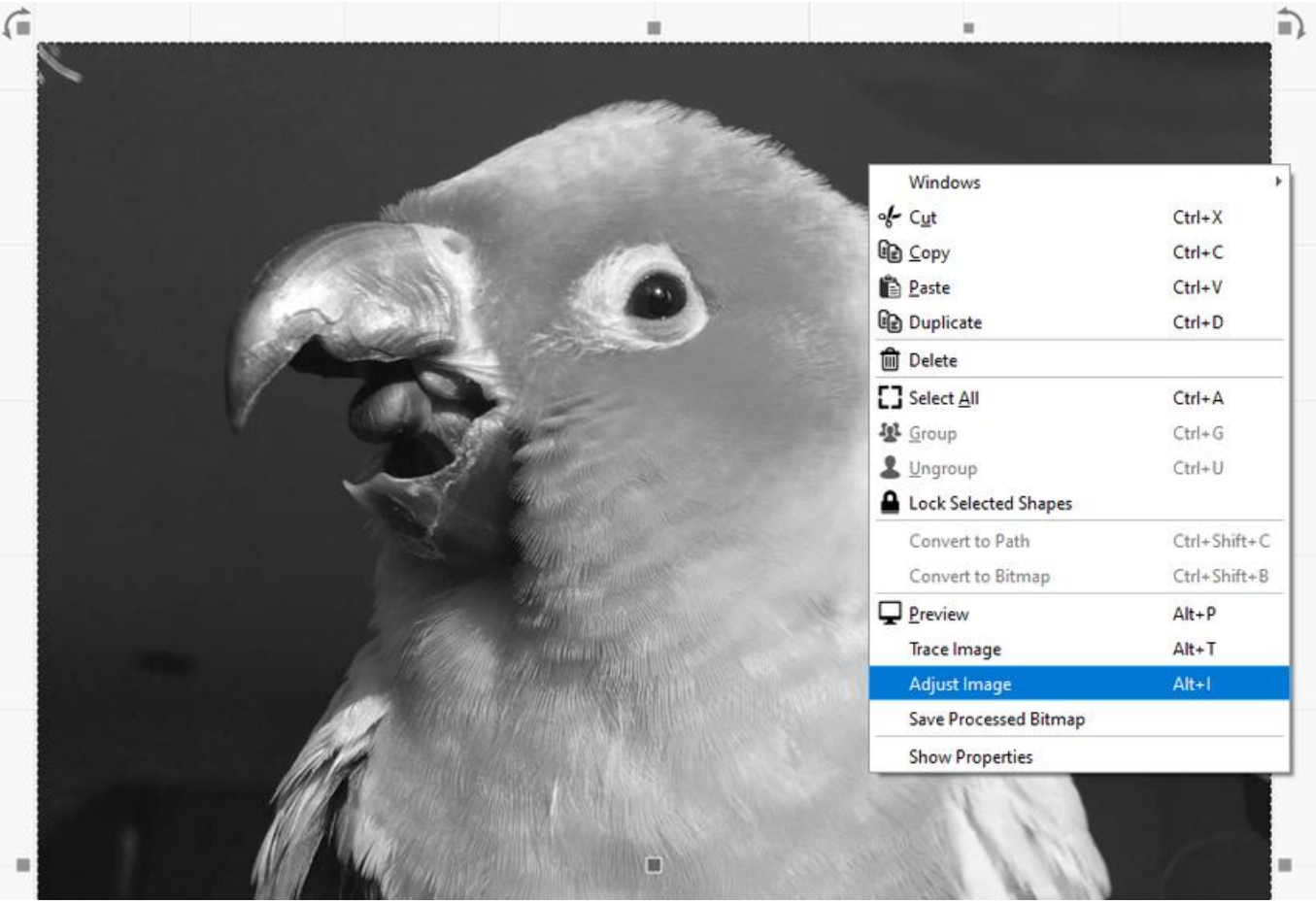
Image Tools

Adjust Image

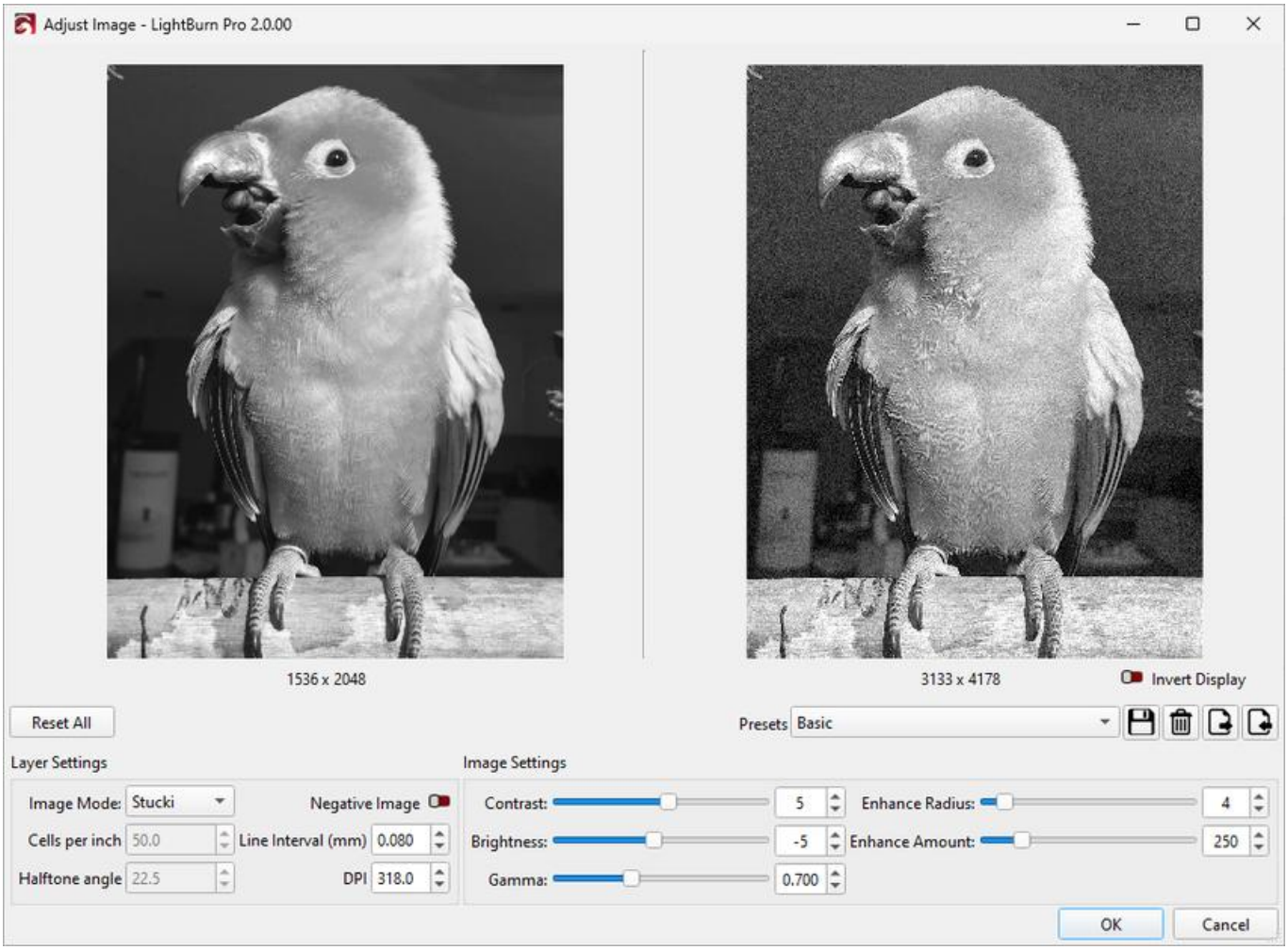
The **Adjust Image** tool streamlines the process of tweaking Image Settings such as **Contrast**, **Brightness**, and **Gamma** while also changing Layer Settings, allowing you to adjust all of these properties from a single window, and see an instant preview of resulting output.

To open the Adjust Image tool, do any of the following:

- Double click on the image.
- Select any image, right-click, and choose Adjust Image from the context menu
- Press `Alt`/`Option`+`I`.



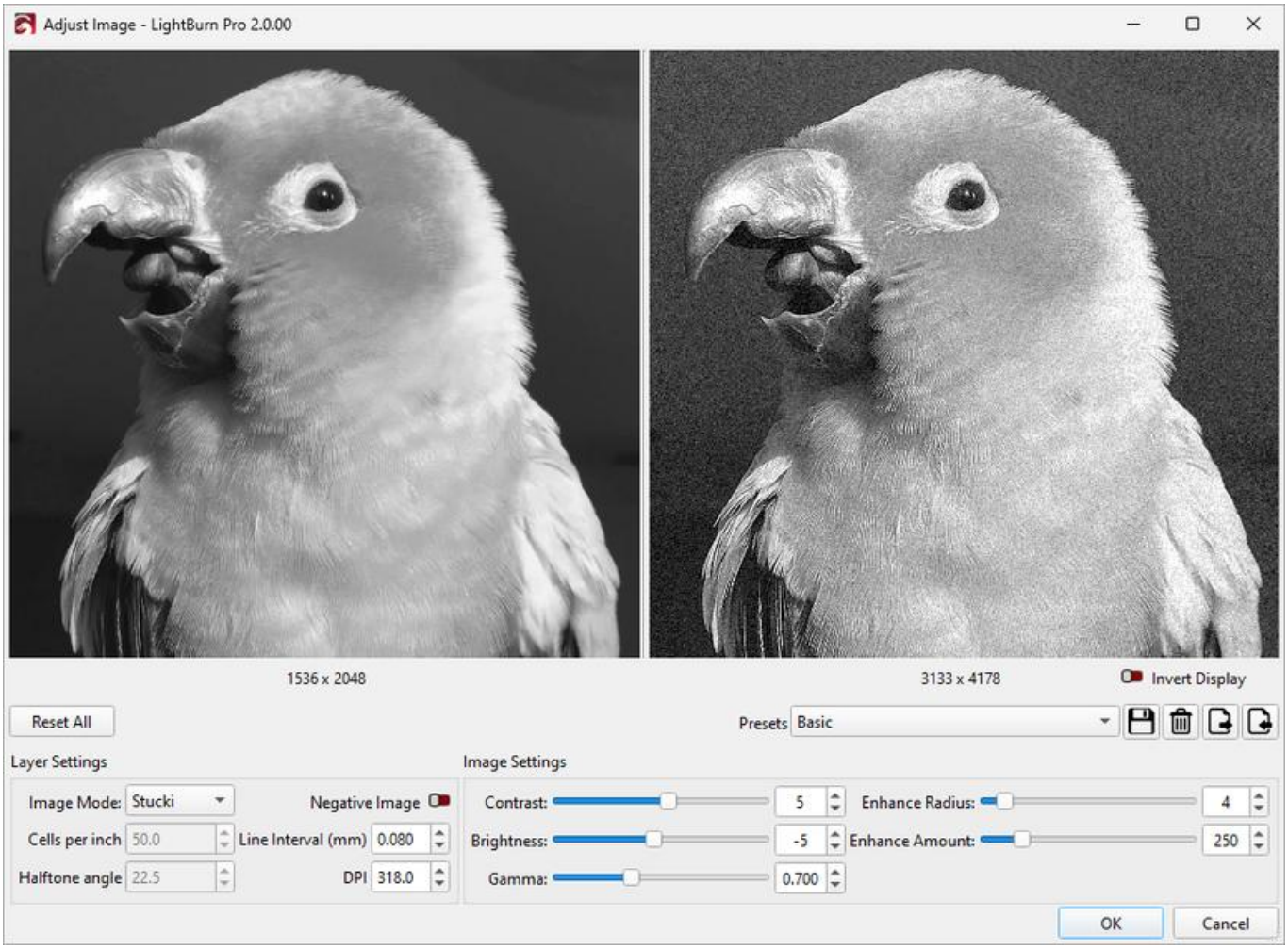
Adjust Image Window



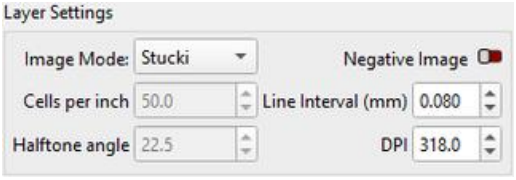
The original source image is displayed in the top left of the **Adjust Image** window. The result of your chosen settings is displayed as a processed image in the top right.

Making any adjustment allows you to see an immediate preview of the processed output, side-by-side against the source image.

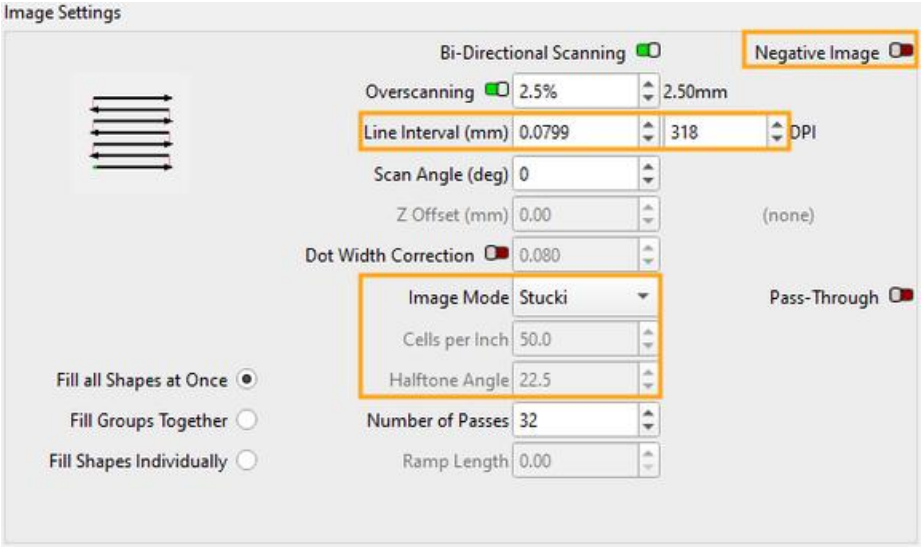
Click and drag to pan around the image, and use the mouse scroll wheel (or two-finger dragging if you're using a trackpad) to Zoom In and Out. The source image and processed image are kept in sync while zooming and moving around.



Layer Settings



The **Layer Settings** group on the left contains settings that correspond to those found in the Cut Settings Editor, as highlighted below:



For more information on these settings, see Cut Settings — Image Mode.

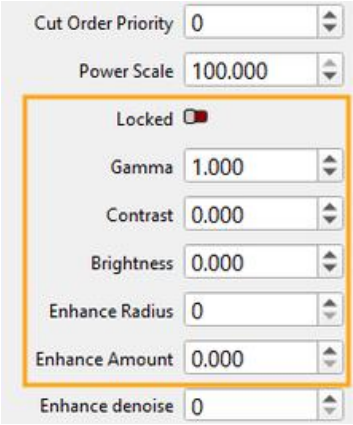
After adjusting settings, click **OK** to confirm and automatically apply changes to the Layer Settings for the image you are adjusting, or **Cancel** to discard the changes and leave the settings unchanged.

> **Note**
>
> These settings will affect any image on the same layer as the one you have selected.

Image Settings



The **Image Settings** group on the right contains settings that correspond to those found in the Shape Properties Window. Adjust these settings by dragging the sliders or directly editing the numeric value of each property.



For more information on these settings, see Shape Properties — Image.

As with **Layer Settings**, click **OK** to confirm changes, or **Cancel** to discard them.

Invert Display

Turn on the **Invert Display** option to show the processed image with black for the background and white for lasered lines.

Inverting provides a preview of what the resulting output will look like when engraved on slate, black painted tile, black anodized aluminum, or any other material where the marked area will be lighter than the surface of the unmarked material.

> **Note**
>
> This option affects the preview image only — it has no effect on actual output.

Presets



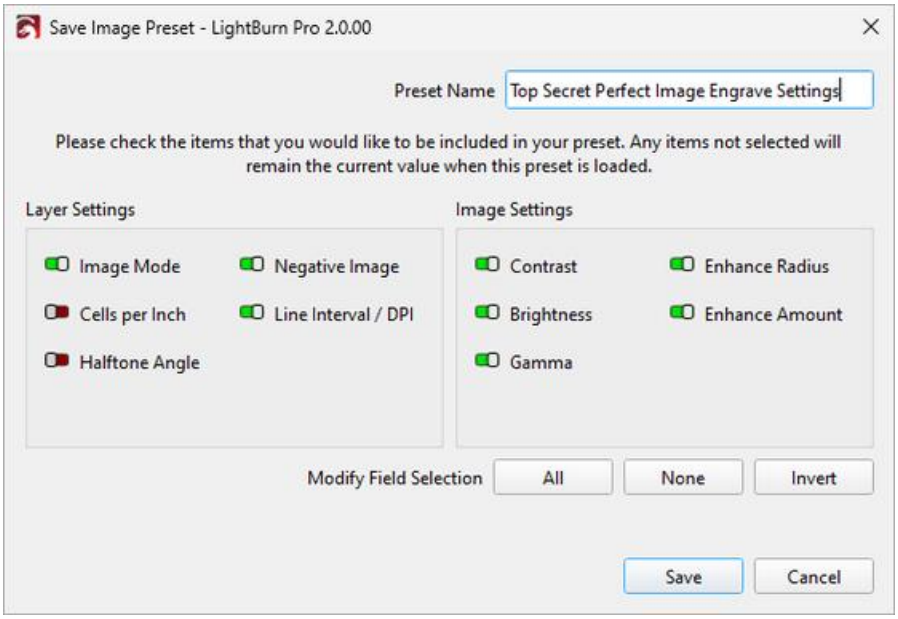
You can select **Presets** from the dropdown above **Image Settings** on the right side of the **Adjust Image** window.

Selecting a Preset automatically applies all the settings saved to it, overwriting your existing settings.

LightBurn provides two **Built-In Presets**, as well as the option to save your own frequently used image settings as **User Presets**.

- **Basic** is a good starting point for your own engravings or for creating User Presets.
- **Black Paint on White** is similar, but inverted for engraving light marks onto dark surfaces.

Save Image Preset



The first button next to the **Presets'** dropdown allows you to save your current settings as an image Preset.

1. Enter a name for your Preset in the **Preset Name** field near the top right of the window.
2. You can choose which **Layer** and **Image Settings** are included in the Preset by toggling the switches next to each option.
3. Click **All** to include all settings, **None** to exclude all settings, or **Invert** to include settings that are currently excluded, and exclude settings that are currently included.

   Excluded settings well be ignored when loading the Preset later.
4. Click **Save** to store the selected settings as a Preset, or **Cancel** to exit without saving.

Delete Image Preset

The second button allows you to **Delete** image **Presets**. Make sure the Preset you want to delete is selected in the dropdown first.

Import/Export Image Preset

The third and fourth buttons allow you to **Import** and **Export** image **Presets**, which is useful for backing up your Presets and for moving them between computers.

> **Tip**
>
> You can also include these Presets as part of a User Bundle.

Related Topics

- Cut Settings — Image Mode

- Shape Properties
- Apply Mask to Image
- Save Processed Bitmap

Apply Mask to Image

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=1q4WyVMfhXg

Use **Apply Mask to Image** to hide portions of images that you do not want to engrave, or to divide them into more complex engraving patterns.

By default, image masking doesn't delete the data of the removed part of the image, it just hides it from view and doesn't output those parts to the machine.

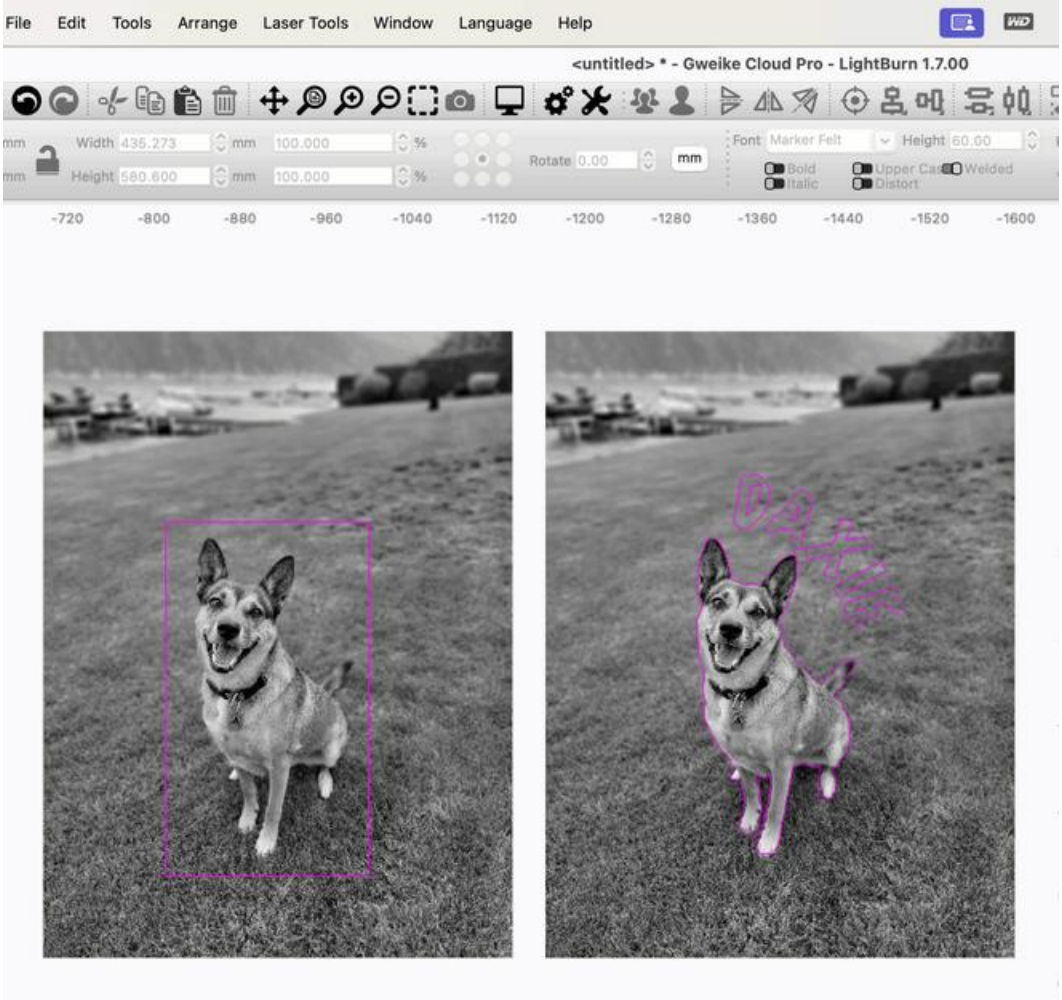
> **Tip**
>
> You can use masking to remove the background of an image, for example to isolate a pet or person from an photograph — a great way to get more clarity in an image that doesn't have a well-contrasted background.

Masking an Image

In addition to an image, masking requires a vector object, in any desired shape. The vector object must be:

- Placed on either a Tool or Line layer
- A closed shape, with a single, shared start and end point

You can use use rectangles to perform standard crops, but the shape can be a more complex path as well, or even a Group of vectors (including text) provided that the Group is all on the same layer. When shapes are placed on a shared layer, Grouped, and used for masking, the area between the shapes that would normally be solidly engraved with Fill Mode will be "filled" with the image instead.



To Mask an Image, select the image and vector, then go to Tools → **Apply Mask to Image**, or right-click the selection and choose Apply Mask to Image from the context menu.

To immediately Flatten the mask in the process, you can instead to go Tools → **Crop Image**. If you choose to Crop an image, you will be unable to remove or edit the mask.

> **Hidden Vector**
>
> If the vector is hidden by the image, use Push to Front to move it forward in the draw order if you're also using a Wireframe View Style.
>
> If you're using a Filled View Style, adjust the order of the layers in the Cuts / Layers Window.

Image Cropping vs. Masking

Using **Apply Mask to Image** and **Crop Image** are tools you can use to hide parts of an image you want LightBurn to ignore, but they work a little differently.

- **Apply Mask to Image** doesn't delete anything, and you can move and resize the mask relative to the image.

- **Crop Image** deletes the parts of the image that are removed, leaving only an image in the shape of the masking shape.

See the video here for more information.

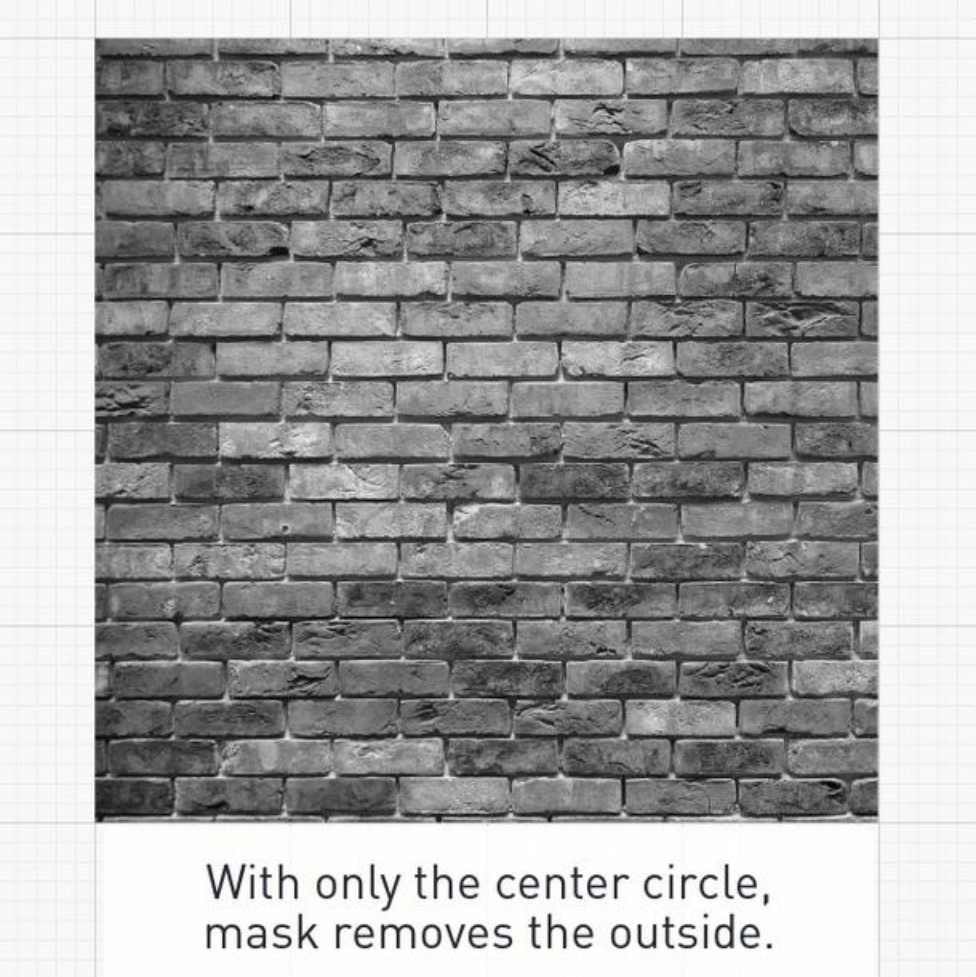
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=sf4KNLX71-k

Removing the Center of an Image

Sometimes, you might want to cut out the inside of an image while leaving the outer portion intact. You can do this by Grouping together nested shapes before masking.

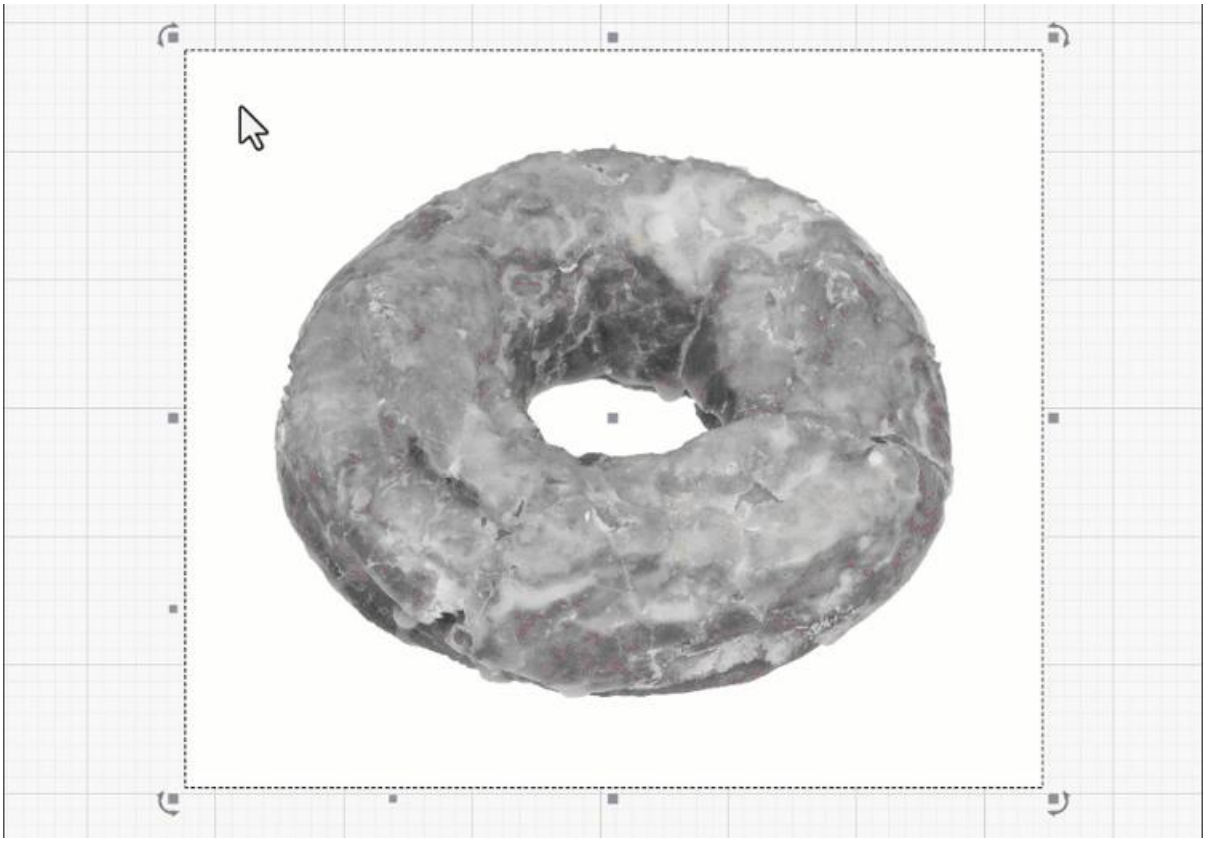
See below for detailed directions and an animation showing the difference in results between masking an image with just a circle or a circle inside of a rectangle.

1. Draw the vector shape(s) for what you want to remove.
2. Surround the shape(s) you drew with an outer shape. If the outer shape is completely outside of the image you're masking, the outer edges of the image will be unchanged. If the outer shape cuts into the image itself, the part of the image outside of your shape will be removed.
3. Group your inner and outer shapes together.
4. Select the grouped vectors and the image, and apply a mask by either going to **Tools → Apply Mask to Image** or right-clicking and choosing **Apply Mask to Image** from the context menu.



Combining Tracing and Image Masks

Using LightBurn's Trace tool on images with clearly defined edges can be a quick way to produce a mask to remove an unwanted background. In the example below, the donut is traced with Delete Image After Trace disabled, and then the traced shape is used as a mask to hide the background.



Removing a Mask (Restoring an Image)

You can fully restore the original image in two ways:

- Select the vector object you used for the mask and press `Backspace`/`delete` on your keyboard to delete it.
- Select the image, right-click it, and choose **Remove Mask from Image**.

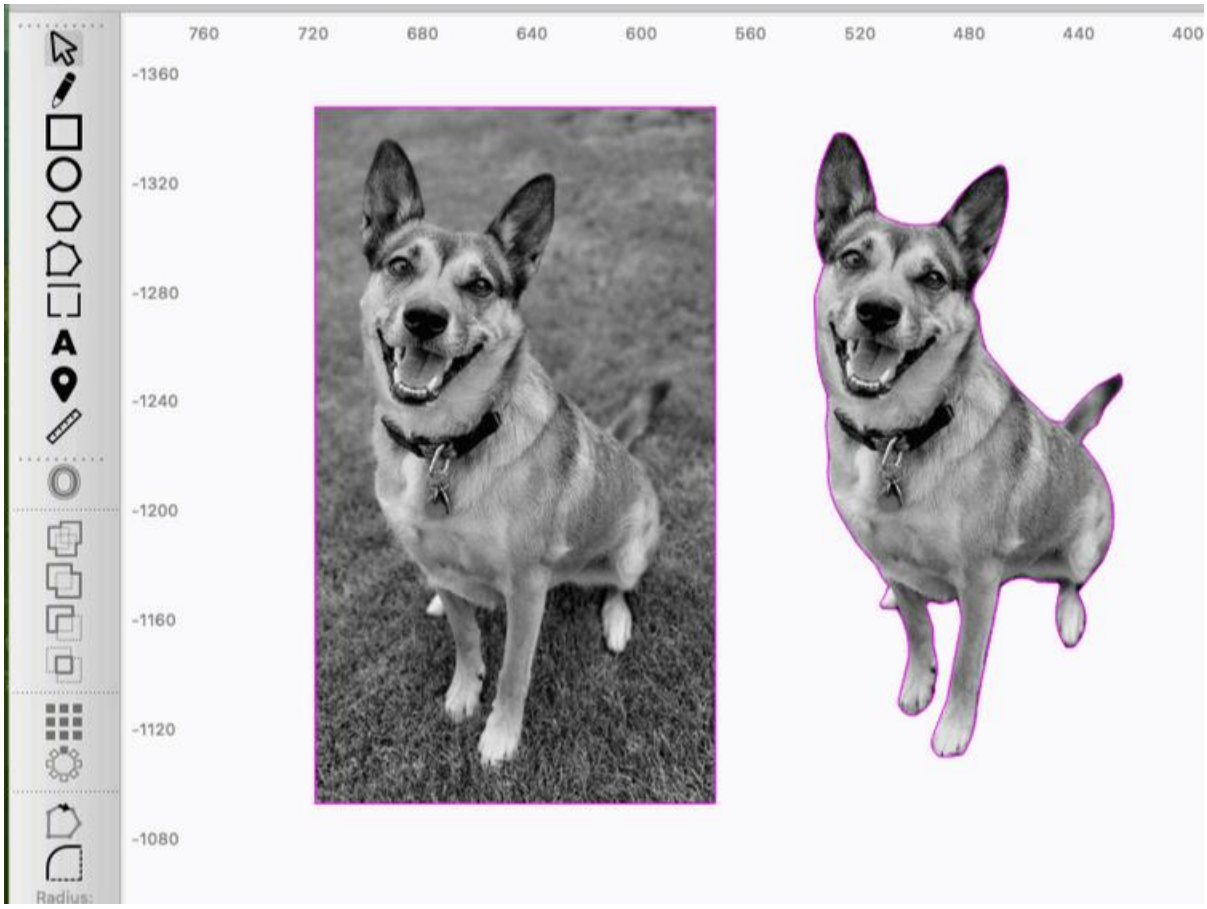


Editing a Mask

To change which parts of the image are revealed and hidden, you can edit the existing mask as you would any other vector object, for example with the Select, Edit Nodes, and Boolean tools.

Grouping

Once you have the mask as you want it, it's a good idea to Group the vector and image to avoid accidentally moving one without the other.



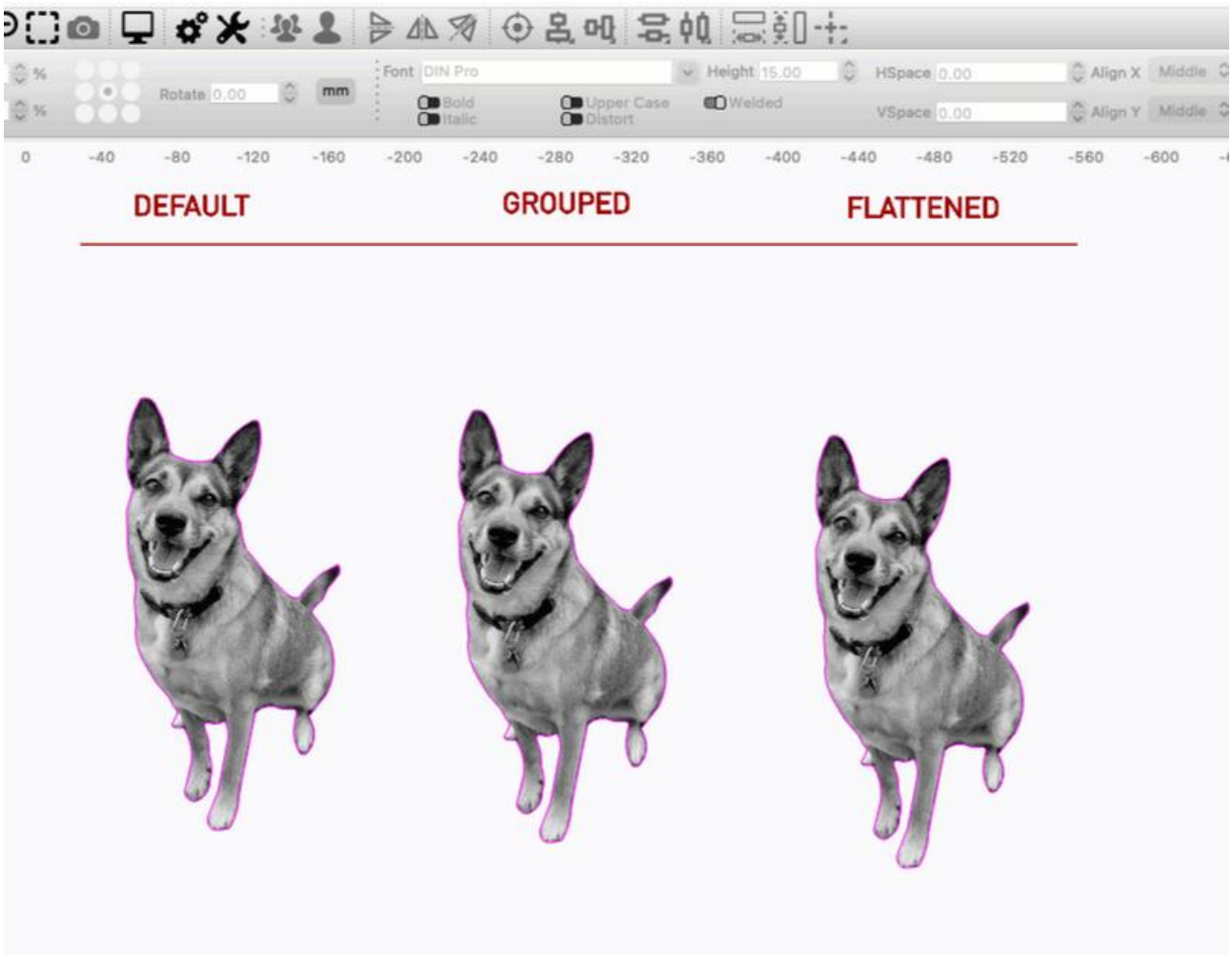
Flattening (Baking-In) a Mask

The downside of preserving the hidden parts of the image is that it can make masked images harder to align to other objects, but it is possible to permanently remove the hidden areas of a masked image (also called "baking in" the mask).

To do so, right-click the masked image and select **Flatten Image Mask**.

This also removes the masking shape, leaving you with a only an image in the shape it had been masked.

To accomplish this in one step, you can use **Crop Image** from the **Tools** menu.



Related Topics

- Cut Settings — Image Mode
- Adjust Image

Convert to Bitmap

Convert to Bitmap turns selected vector graphics into bitmap images.

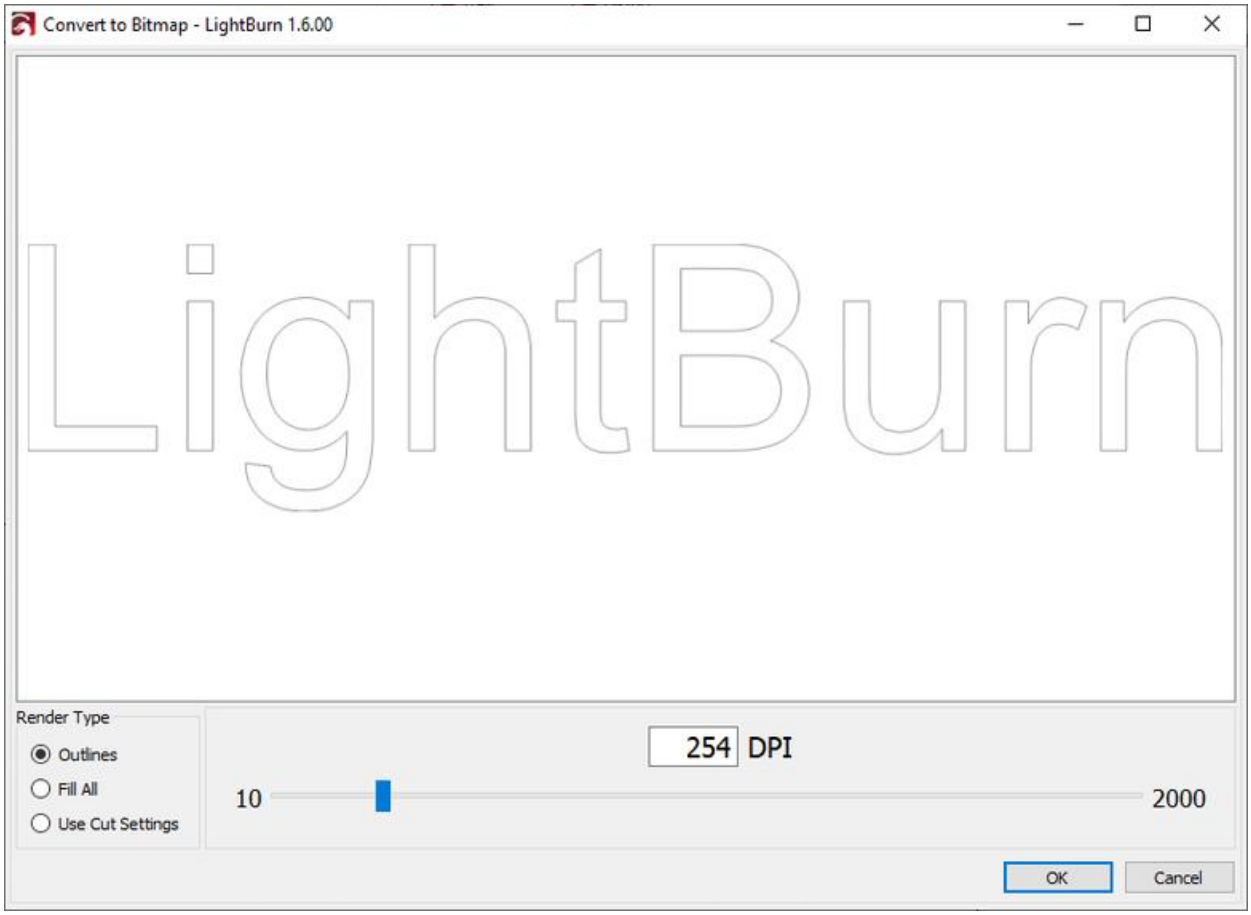
Vector graphics are made up of lines, curves, and nodes that define the contours of shapes, while bitmap images are made of a grid of shaded pixels.

See Images vs. Vectors for more information on the differences between images and vector graphics.

Once a vector graphic has been converted to a bitmap image, it is set to Image Mode and can be engraved using a dithered pattern.

Using Convert to Bitmap

Select all vector graphics you wish to convert and go to Edit → **Convert to Bitmap** (`Ctrl`/`Cmd` + `Shift` + `B`), or right-click and select Convert to Bitmap.



- At the top of the Convert to Bitmap window you'll see a preview of the bitmap image that will be created. As you adjust the options the preview will live-update.
- The **Render Type** option determines whether the resulting bitmap will show just the **Outlines** of the vector graphics, will **Fill All** areas between the outlines, or **Use Cut Settings** to determine whether to render outlines or solid fills depending on whether the graphics are assigned to a layer set to Line Mode.

> **Note**
>
> In order to Render a vector graphic as a solid, filled bitmap it must be a closed shape. See Open vs. Closed Shapes for more information.

- Set the **DPI** for the resulting bitmap by manually entering a value into the DPI field, or by adjusting the slider below it. DPI controls the density of the pixels in the bitmap — higher pixel densities lead to sharper images.
- The brightness value of each pixel is automatically set to 50% gray. After converting vector graphics to bitmap images, you can adjust their brightness using the Adjust Image tool.
- Press **OK** to create the bitmap image, or **Cancel** to discard the changes and leave the vector graphics as they are.
- The image produced will be in the last layer selected. If the image immediately disappears, check to see if it's on a layer that's not set to be visible.

> **Warning**
>
> Converting a vector graphic into a bitmap image deletes the vector graphic. To preserve the original graphic while creating a bitmap from it, Copy or Duplicate it, then convert the second copy.

Related Topics

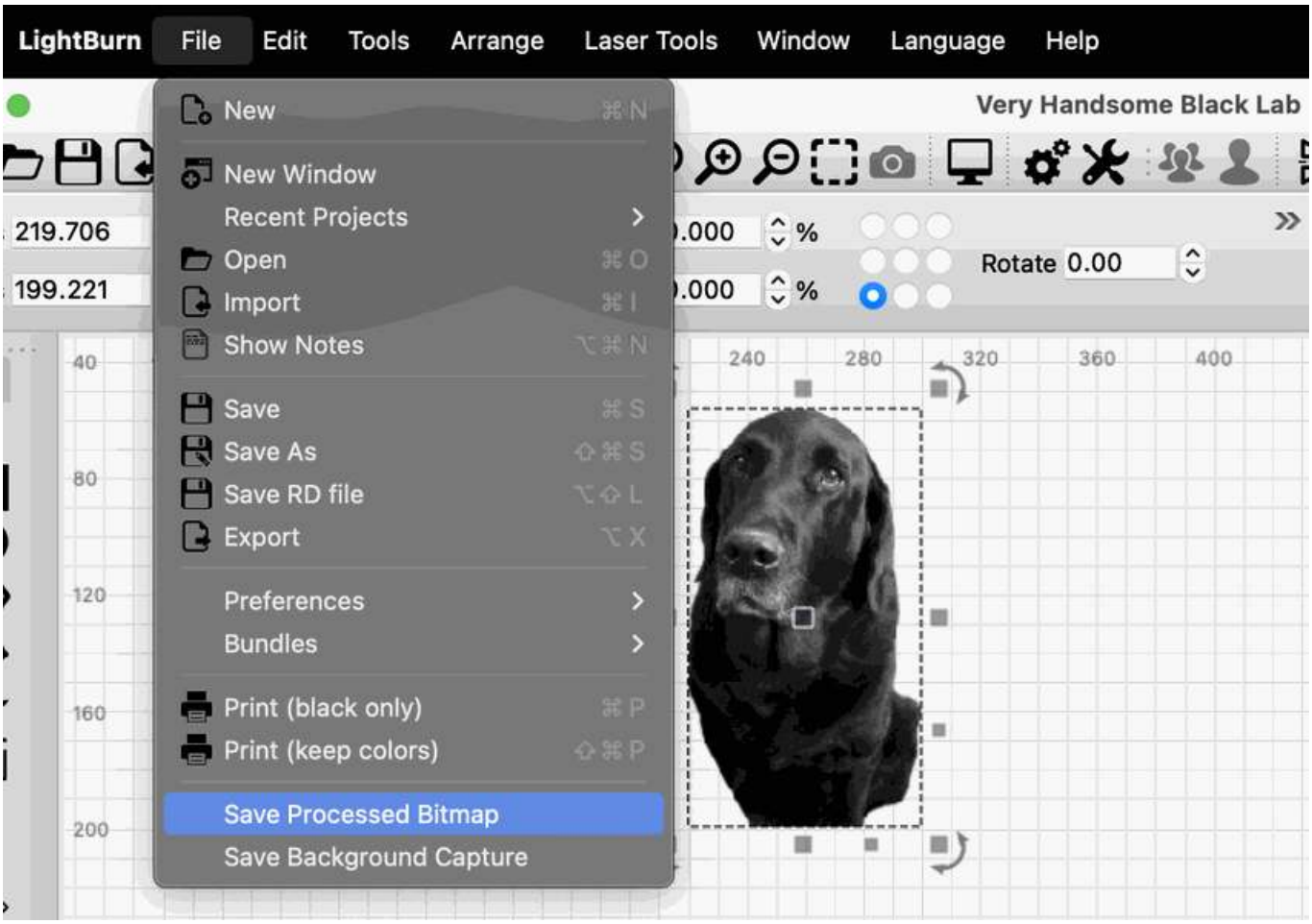
- Image Mode
- Adjust Image

Save Processed Bitmap

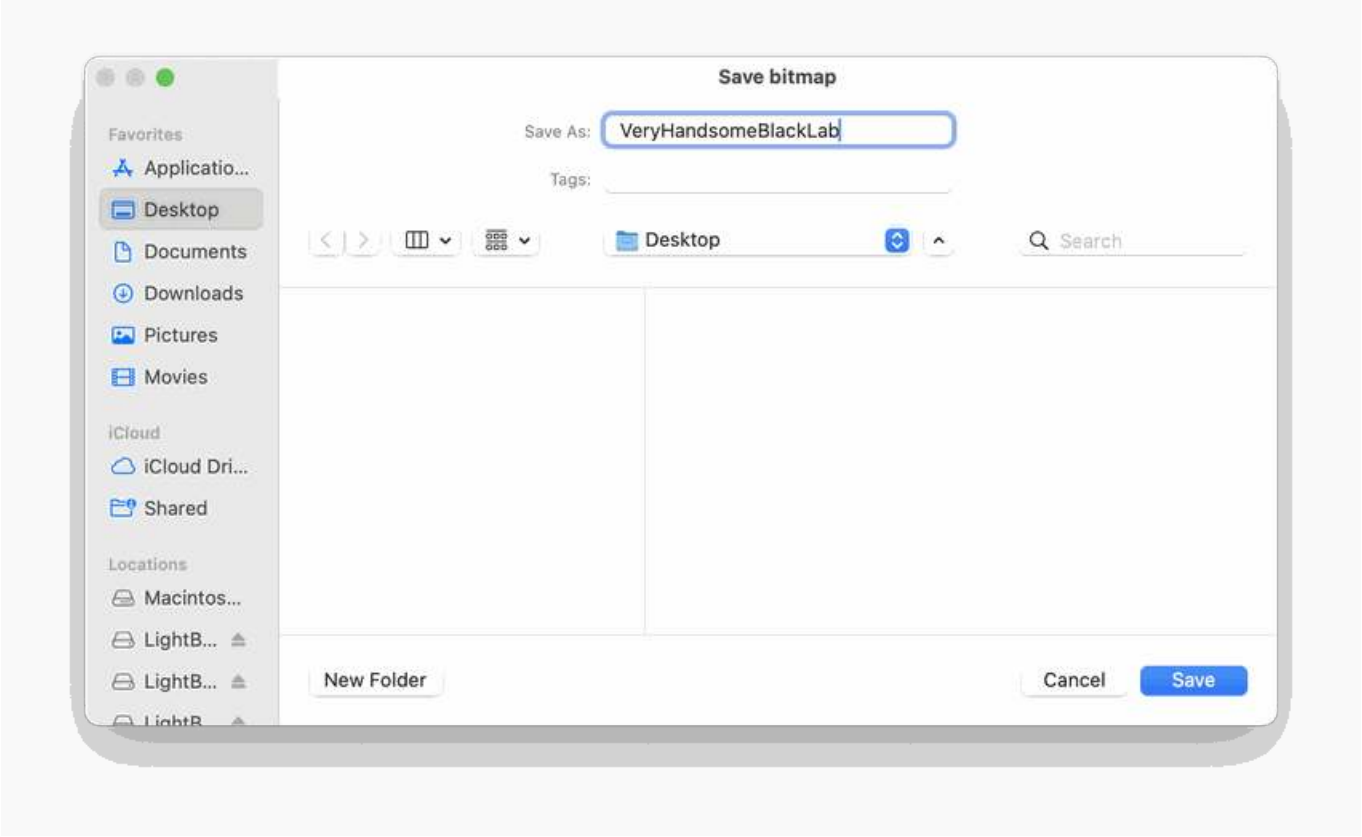
Use **Save Processed Bitmap** to save an image you have edited in LightBurn, including any changes made using Adjust Image, or any dithering mode applied to the image.

Saving Processed Bitmaps

To save a processed image, first select an image, then go to File → **Save Processed Bitmap** or right-click in your Workspace and select Save Processed Bitmap from the context menu.



Your system's file management window will open, where you can name and select a location to save the processed image. After, you'll be able to open the processed image using any software that can open images.





Related Topics

- Adjust Image
- Image Mode

Trace Image

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=ZDZ3QqTfdIk

Use the Trace Image tool to convert imported images into vector graphics.

This is useful if you want to edit an image that has high contrast or well-defined edges, including logos, silhouettes, cartoons, sketches, and handwriting. You can trace photographs as well, but results will vary based on the composition of the photograph.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=OXNKFEl6pjA

Trace Image Window

After selecting an image in your Workspace, select Tools → **Trace Image**, right-click the image and select **Trace Image** from the context menu, or press `Alt`/`Option`+`T` to open the Trace Image window.



In the image above, the purple lines are the vectors that LightBurn has produced from the image being traced. As you adjust settings in the **Trace Image** window, the purple lines will change to reflect the new vectors that will result from the updated settings.

Navigation and View Options

You can Zoom and Pan the preview using the same controls as in the Workspace, and double-click the middle mouse button to reset the view.

Fade Image

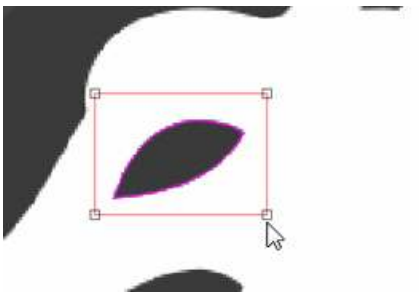
Click **Fade Image** to dim the image, increasing the visibility of the vectors (purple lines) in the preview.

Show Points

Click **Show Points** to toggle visibility of the nodes in the preview of the resulting vectors.

Clear Boundary

Click and drag in the preview to specify a portion of the image to trace. Click **Clear Boundary** to reset it.



Trace Options

Cutoff and Threshold

Cutoff and **Threshold** control the range of brightness values that LightBurn will include in what's traced. The **Cutoff** slider controls the lower (darker) end of the range of brightness values that LightBurn will include in the trace. The **Threshold** slider sets the upper (lighter) end.

Increase the **Threshold** value to include lighter/brighter shades of gray in what's traced. Increase the **Cutoff** value to remove darker shades of gray.

The values range from 0 to 255, and by default LightBurn will include values from 0 (black) to 128 (medium gray).

Trace Transparency

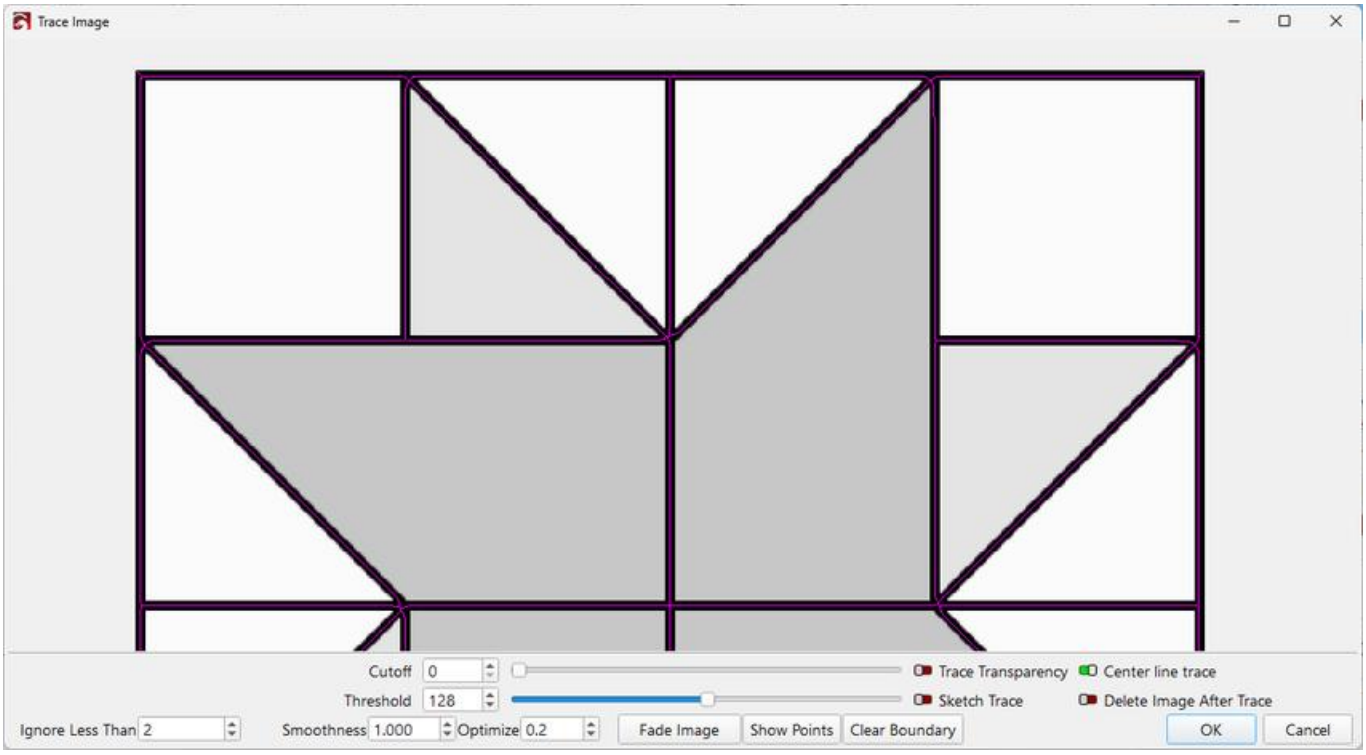
If the image you're tracing has a transparent background, you can trace the outline of the image quickly and easily by turning on **Trace Transparency**. When this is enabled, LightBurn will trace the border between transparent and non-transparent. Colors and internal details are ignored.

You can see an example of using Trace Transparency in the video here.

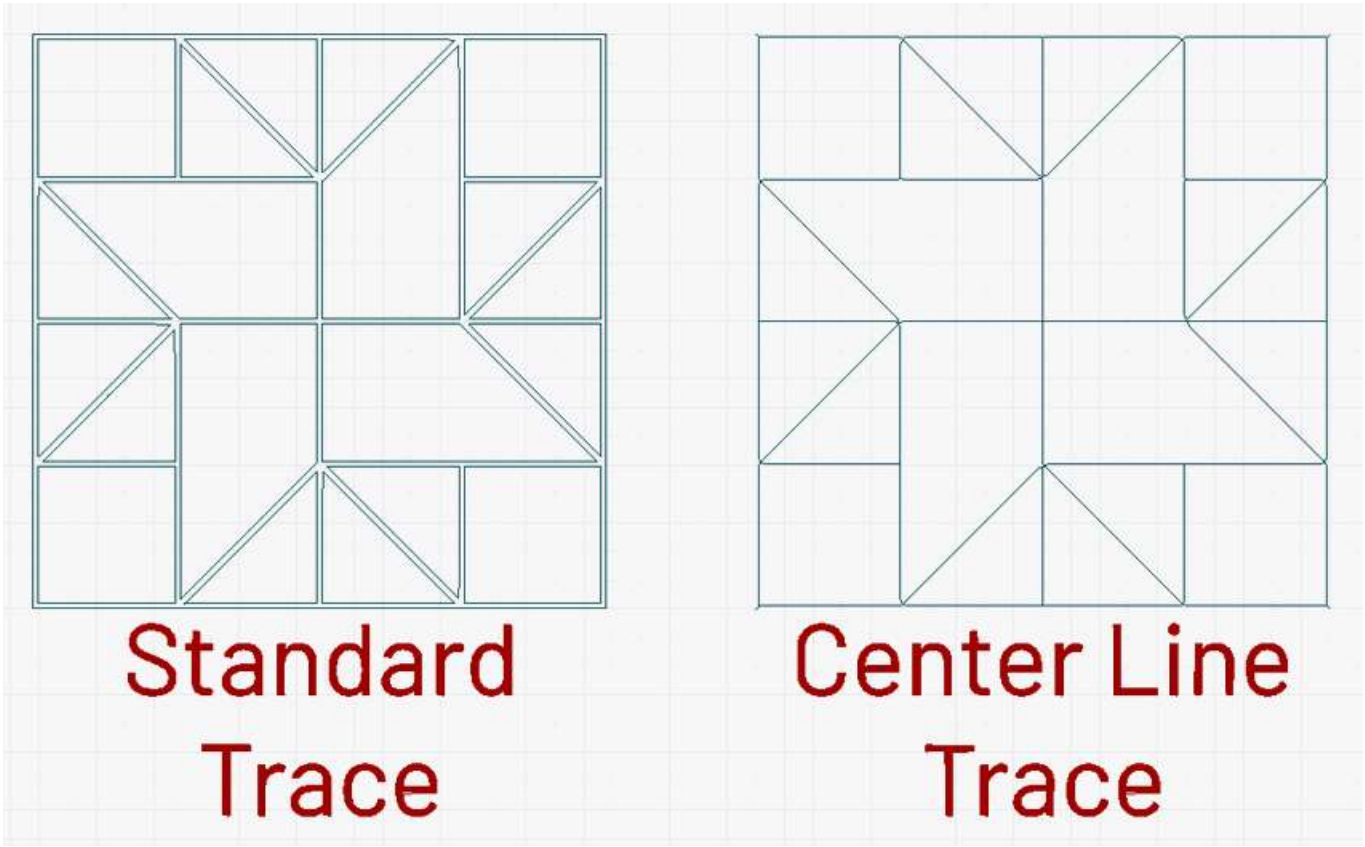
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=5O6Hun4_uS0

Center Line Trace

Use **Center Line Trace** to trace along the center of a line rather than around the edges. You can use Center Line Trace to turn line art like the quilt block below into vector paths to cut out.



The comparison image below shows a standard trace result next to Center Line Trace for the same image.



Sketch Trace

Turn on **Sketch Trace** for photographs of pages with uneven lighting — for example handwriting, sketches, and documents. LightBurn will look for edges in the image with this setting enabled, based on local differences in contrast, rather than relying on the brightness or darkness of the whole image.

Delete Image After trace

Turn on **Delete Image After trace** to automatically remove the image from your project upon hitting **OK**.

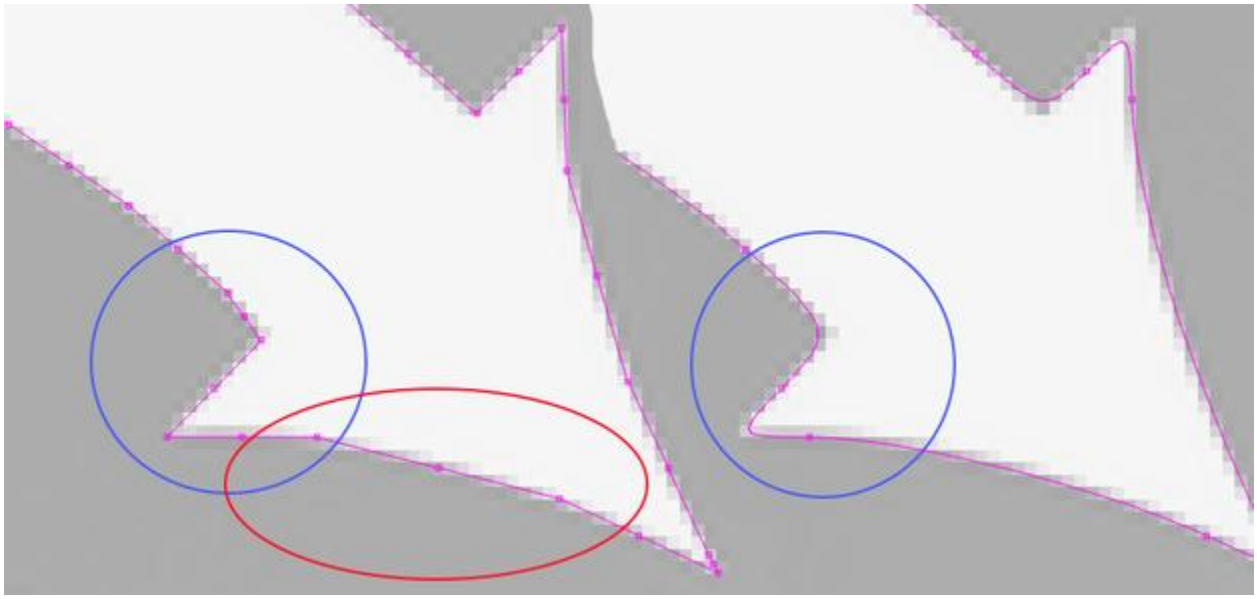
Ignore Less Than

Instructs LightBurn not to trace areas where the number of pixels in the valid brightness range is lower than this number. Increase the number of pixels in the **Ignore Less Than** field to clean up noisy images by removing small areas from the result.

This setting is useful for ignoring small details that may not be useful in the tracing.

Smoothness

Increase **Smoothness** to turn more line segments of the resulting vector into curves. Set it to **0.0** to create only lines (left), and **1.33** to create only curves (right).



Optimize

Increase the value in the **Optimize** field to reduce the number of nodes in the resulting vectors, similar to Optimize Selected Shapes.

A value of **0** leads to no optimization, while excessively high values can reduce accuracy. The default of **0.2** usually provides a good balance between accuracy of the result and node count.

Editing the Trace

After hitting **OK**, you will be presented with a Group containing the resulting vectors. To edit the shapes individually, you must first **Ungroup** them.

After they are Ungrouped, you can edit them as you would any other shape.

> **Hidden Vector**
>
> If you can't see the vector, it may be hidden by the original image, click and drag the image out of the way to reveal it, or Push the image to the back of the draw order.

Troubleshooting

Tracing works by automatically creating an approximation of the pixel-based image in vector format, and while LightBurn's trace function is extremely robust, it remains limited by both the computationally-difficult nature of this task, and the quality of the image it is presented with. In some instances fine details — for example sharp corners and small text — may not be converted well during this process. If changing the Trace options doesn't overcome this issue, exit the **Trace Image** window and try using Threshold Image Mode to engrave the original image instead of tracing it.

Related Topics

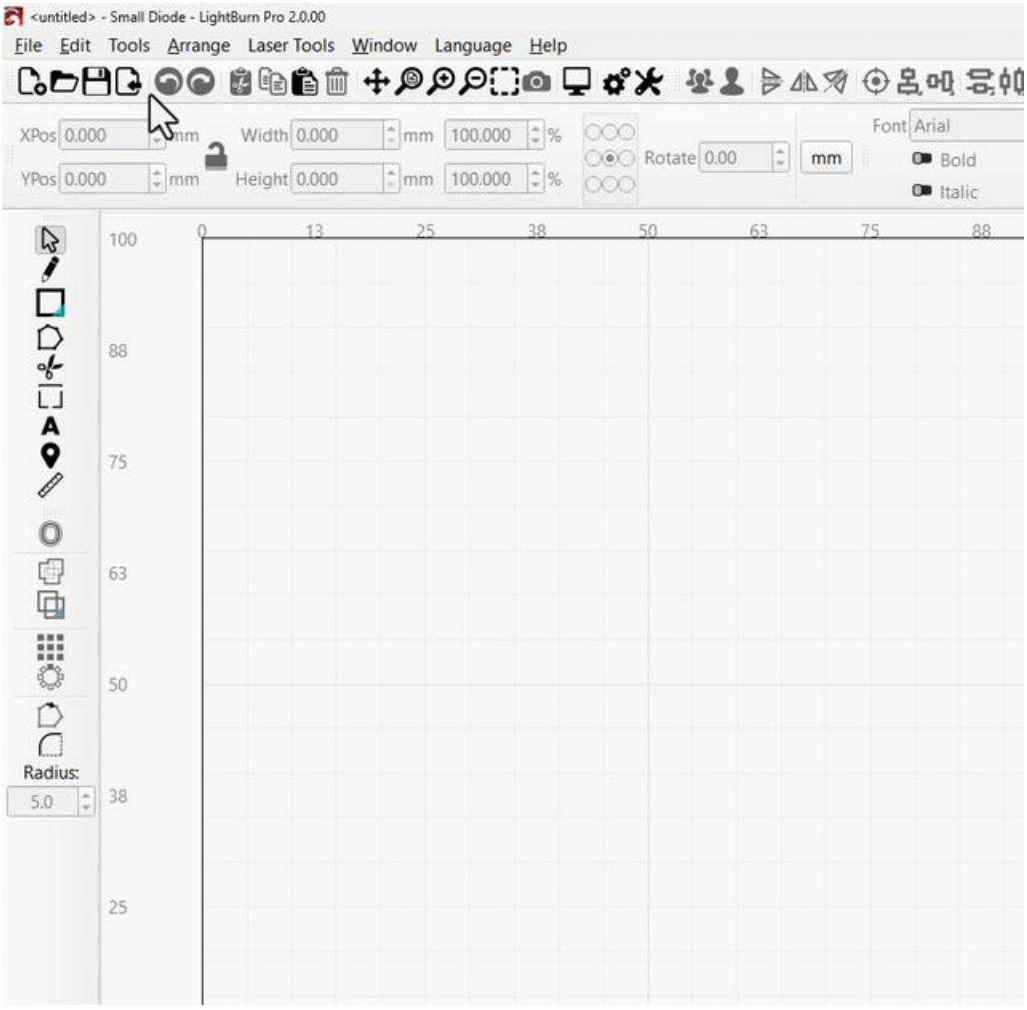
- Adjust Image
- Apply Mask to Image
- Convert to Bitmap
- Edit Nodes

Multi-File Trace Image

> **Labs Feature**
>
> Labs features are new tools in development that we couldn't wait to get into the hands of our users. Please experiment with caution, and report any unexpected results on our forum.

The **Multi-File Trace Image** tool automates image tracing for multiple files using LightBurn's Trace Image tool. Images are traced using LightBurn's default settings, and saved as SVG files in the same location as the source images.

Using Multi-File Trace Image



1. Go to Tools → **Multi-File Trace Image (Labs)** to open the file import dialog box.
2. In the dialog box, navigate to the folder containing images you want to trace.
3. Select all the images to be traced. Use `Shift` to select a range of files or `Ctrl`/`Cmd` to pick multiple files individually.
4. Click **Open** to select and trace files.

5. Go to the folder containing the source images to find your traced SVG files.

Image Options

The three commands under Edit → **Image Options** — **Refresh Image**, **Replace Image**, and **Replace Image to Fit** — provide ways to refresh or replace images in your project. This is most often useful if you've edited the image in a different editing software and want to reload or replace the image currently open in LightBurn.

Refresh Image

After importing an image into a LightBurn project, changes to the original image file aren't automatically reflected in LightBurn. Use **Refresh Image** to reload the image file in its most recent version.

Replace Image

Replace Image opens a dialog box to select a new image. The new image replaces the originally selected image in the LightBurn project.

Replace Image to Fit

Replace Image to Fit replaces the image as described above, automatically scaling it to fit in the space taken up by the originally selected image.

Troubleshooting

If these commands are unavailable, check the following:

- Make sure you have a pixel-based image selected in LightBurn.
- Make sure the image is not Grouped with other objects.

Related Topics

- Trace Image
- Adjust Image

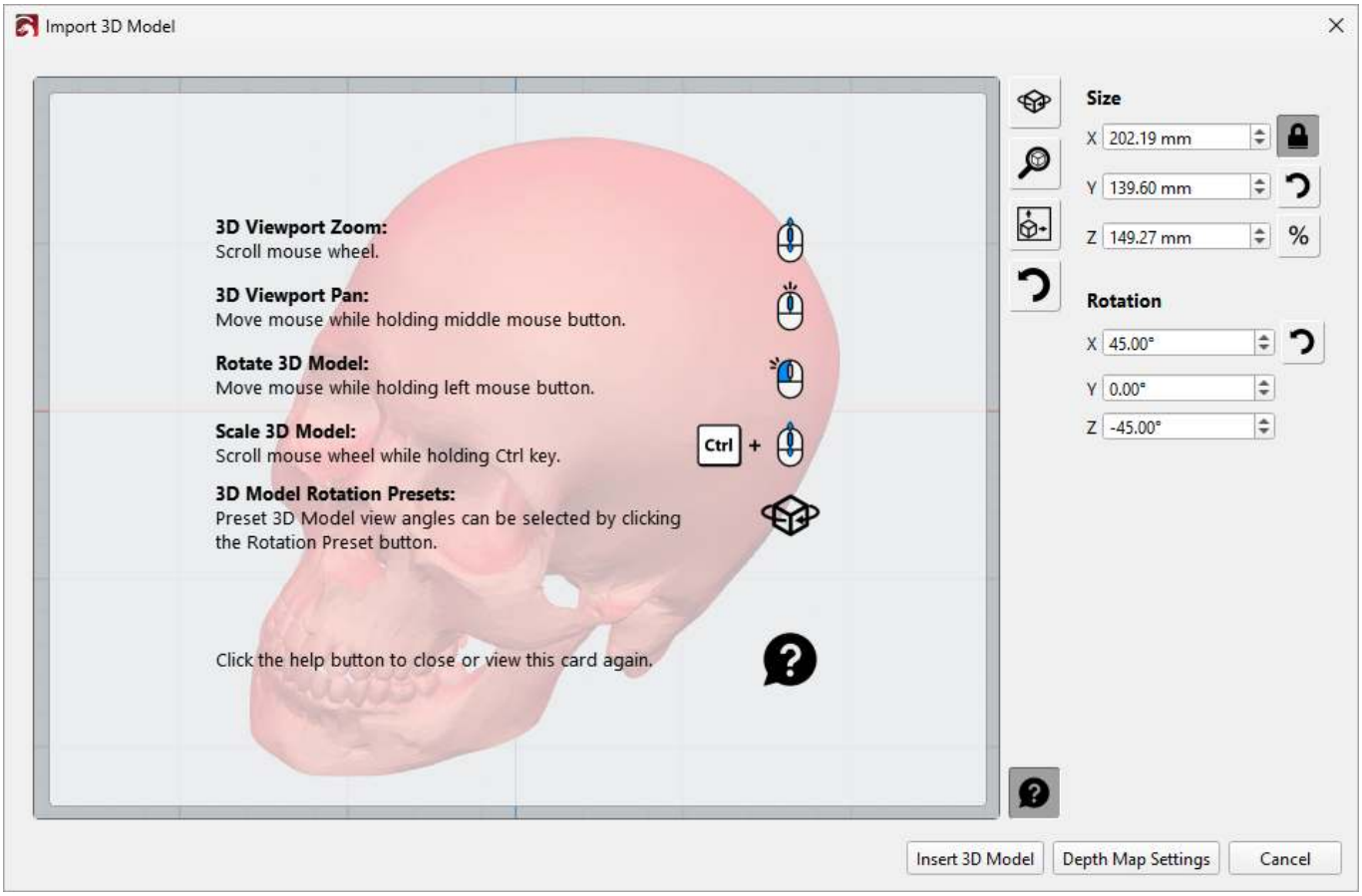
Import 3D Model

Starting with LightBurn 2.2, you can import 3D models directly into LightBurn to create depth maps for 3D engravings. LightBurn can import models in `.stl`, `.3mf`, `.amf`, `.obj`, `.off`, and `.ply` formats. The first three are commonly shared file formats, particularly for 3D printing.

To start importing a 3D model, either:

- Drag the 3D model you want to import from your computer's file explorer into LightBurn.
- Go to **File** → **Import** and select your 3D model.

Import 3D Model Window



Select a Preset Model Orientation

Click to select from a variety of preset model orientations. This can be useful for getting close to the orientation you want.

Zoom to Fit

Click to zoom to fit the model fully into the Import window.



Resize Model to Fit Workspace

Click to resize the model so that the depth map will fit fully into your laser's Workspace.



Reset Model Size and Orientation

Click to reset the size and orientation (position) to the model's default. Useful when you need to just reset your changes and start over.

Size

Use the numeric entry boxes and the buttons alongside them to adjust the size of the imported model.



Uniform Scale / Size for All Axes

Turn on to maintain the model's proportions when resizing. Turn off to allow stretching the model when resizing it.



Reset Scale / Size to Default

Click to reset the model's size to the default size it imported with.



Display Model Size as Percentage

Toggle this setting to control whether the size is displayed in actual measurements or a percentage of the default size.

Rotation

Use the numeric entry boxes and the reset button to adjust the rotation / orientation of the imported model.



Reset Model Rotation

Click to reset the mmodel's orientation to its default.



Help

Click the Help button to show an overview of mouse controls for the Import 3D Model window.

Depth Map Settings

Setting	Description
Bit Depth	Choose from 8-bit or 16-bit. 16-bit generates a wider range of shades (65,536), but can be slow with large models on older, weaker computers. 8-bit only generates 256 shades of gray, but requires fewer computer resources. The difference between the two is minimal if you're not doing at least 256 passes.
Depth Map Resolution	Select the Image Resolution for your depth map. Higher resolutions will produce more detail, but at the expense of creating a larger file requiring more processing power.

Setting	Description
Size	The total size of the image based on the size of the model and the **Depth Map Resolution** setting.
Transparent Background	Turn on to turn the background of the depth map transparent. By default, the background is black, which engraves to full depth.
Invert Depth Map	Invert the depth map, making darker (deeper) areas light and making lighter (shallower) areas dark.
Equalize Depth Map	Applies an equalization filter to the image. This can help highlight faint or fine details.
Depth Cutoff	Set the cutoff plane depth. Anything below the depth you select won't show up in the depth map. You can think of this like a floor the depth map sits on. Adjusting this setting turns off **Automatic Depth Cutoff**.
Automatic Depth Cutoff	If enabled, automatically sets the model's **Depth Cutoff**. Setting the Depth Cutoff manually will turn this off.
Generate Outline	Automatically includes an outline tracing the model.

Insert 3D Model

Generate and insert a **Depth Map** from the model based on your settings.

Cancel

Abandon the 3D Model Import process.

Working with a 3D Model in LightBurn

To make it easy to tell the difference between an ordinary image and a depth map created from an imported 3D model, the depth map is displayed in shades of red instead of gray.

Editing 3D Models

You can right click on the depth map and choose **Edit 3D Model** to re-open the Import 3D Model Window and change settings. You can also select **Generate Outline** to create a traced outline of the model.

Related Topics

- 3D Sliced Image Guide
- 3D Sliced Image Mode Settings

Laser Control

Cut Settings

Cuts / Layers Window

The **Cuts / Layers Window** shows the list of layers in your project. From the Cuts / Layers Window, you can assign different operations to specific layers, control the order the laser processes the layers, adjust an abbreviated set of settings for each layer, and more.

To adjust all layer settings, double-click any layer in the Cuts / Layers Window to open the Cuts Settings Editor, which contains all available settings.

Use the Color Palette to change the layer selected objects are assigned to.

> **About Layers in LightBurn**
>
> In LightBurn, different colors indicate each layer. These colors don't represent the final product's colors but instead differentiate each operation needed to complete the job. Each layer defines a process and a collection of settings that are assigned to objects within the project.
>
> In other words, layers let you tell the machine what it needs to do to produce the final work.
>
> Generally each color refers to a different layer, with one exception — if you add an image to a vector layer (or vice versa), then the layers will split to show two entries in the Cuts / Layers Window.

Accessing the Cuts / Layers Window

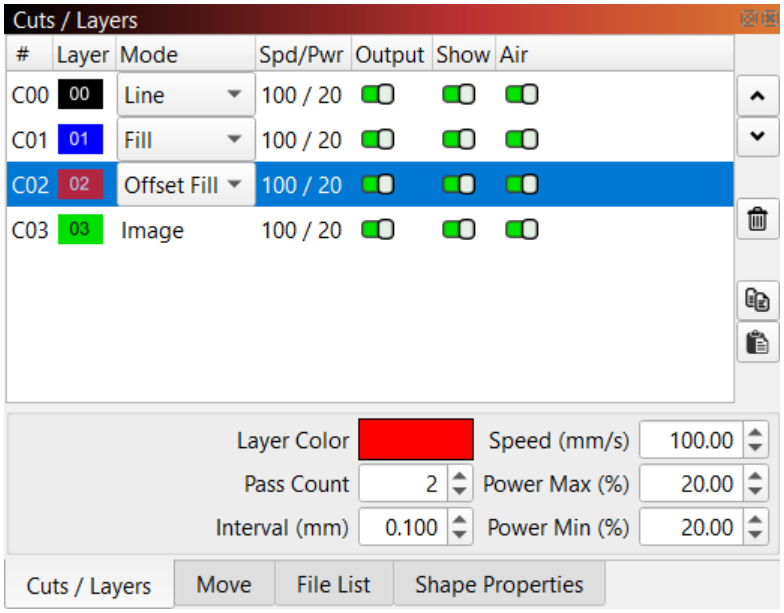
The **Cuts / Layers Window** is located in the upper right of LightBurn by default. If you have closed the Cuts / Layers Window, go to Window → **Cuts / Layers** to re-enable it. To restore it and all other windows to their default positions, go to **Window** → Reset to Default Layout.

> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Cuts / Layers Window Options and Settings

The type of laser you have active will affect which settings are presented in the **Cuts / Layers Window**. The image below shows the Cuts / Layers Window for a DSP laser, but not all options shown here are available for all lasers, and some lasers have additional options that are not shown, but are listed below.

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Columns

(Layer Name)

Shows the layer name. You can adjust a layer's name in the Cuts Settings Editor.

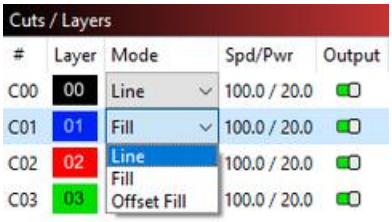
Layer

Shows the color and title of the layer. Colors and titles have matching entries in the Color Palette.

Mode

View and change the Layer Mode of layers in your project.

- You can adjust the mode of vector graphics between Line, Fill, and Offset Fill.
- Layers containing images are automatically set to Image and cannot be adjusted.
- Layers with multiple Sub-Layers are set to **Multi**. Open the Cut Settings Editor to adjust Sub-Layer modes and other settings.



Spd/Pwr

Shows Speed and Power settings.

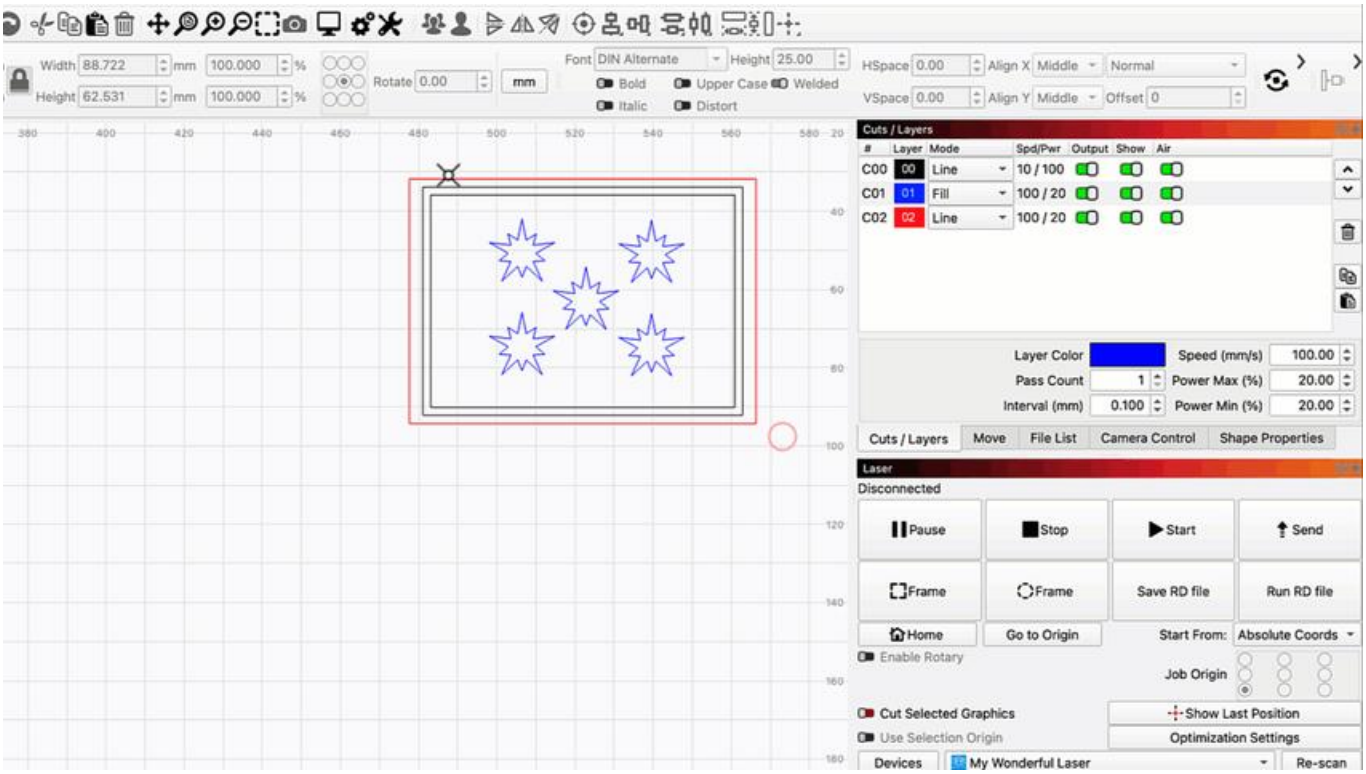
You can adjust these settings in Speed and Power Max fields beneath the list of layers, or by opening the Cut Settings Editor.

Output

> **Tip**
>
> Use Preview to verify that only intended layers are set to output.

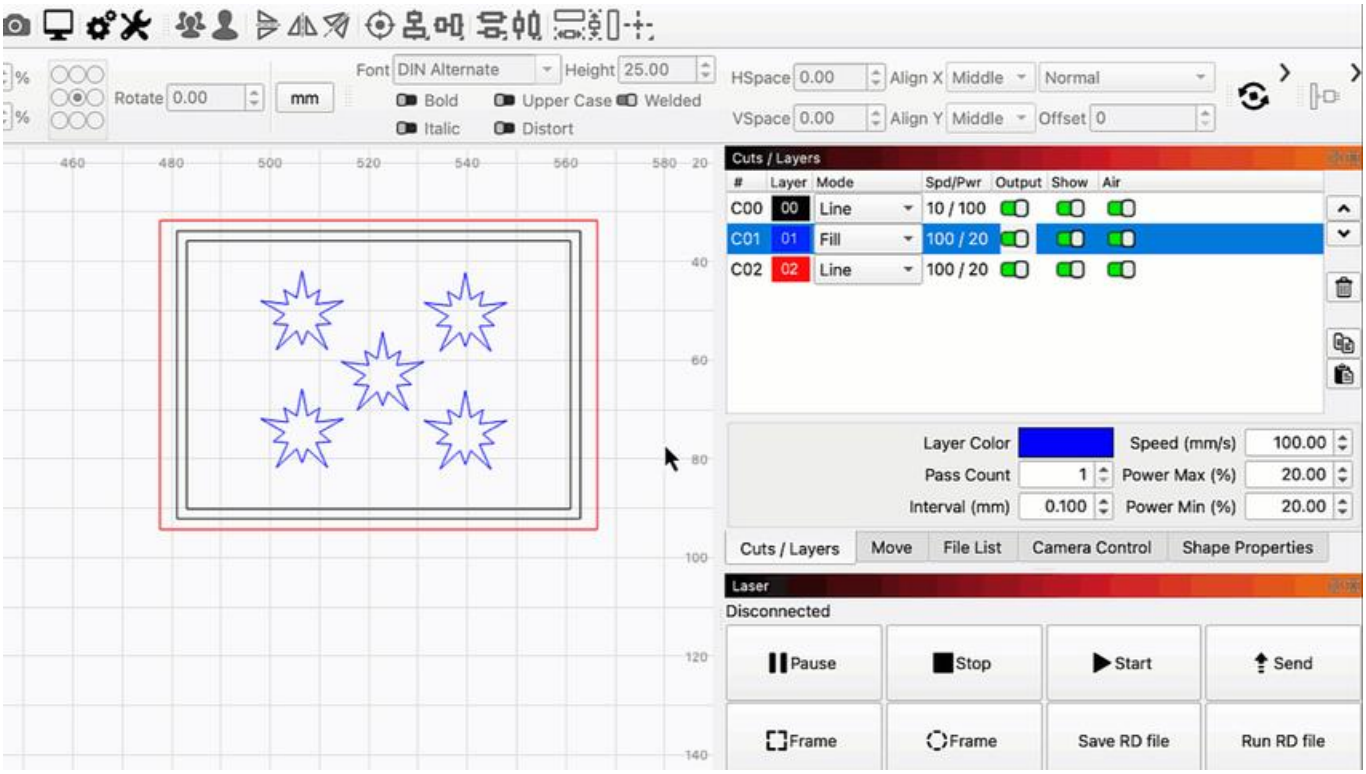
Toggle to enable or disable which layers are output when you Preview, Start, Send, or save a project in machine-specific format.

The color of layers not set to output will be faded in your Workspace.



Show

Toggle which layers are visible in your Workspace.



Air

> **Note**
>
> Available for Gantry lasers only.

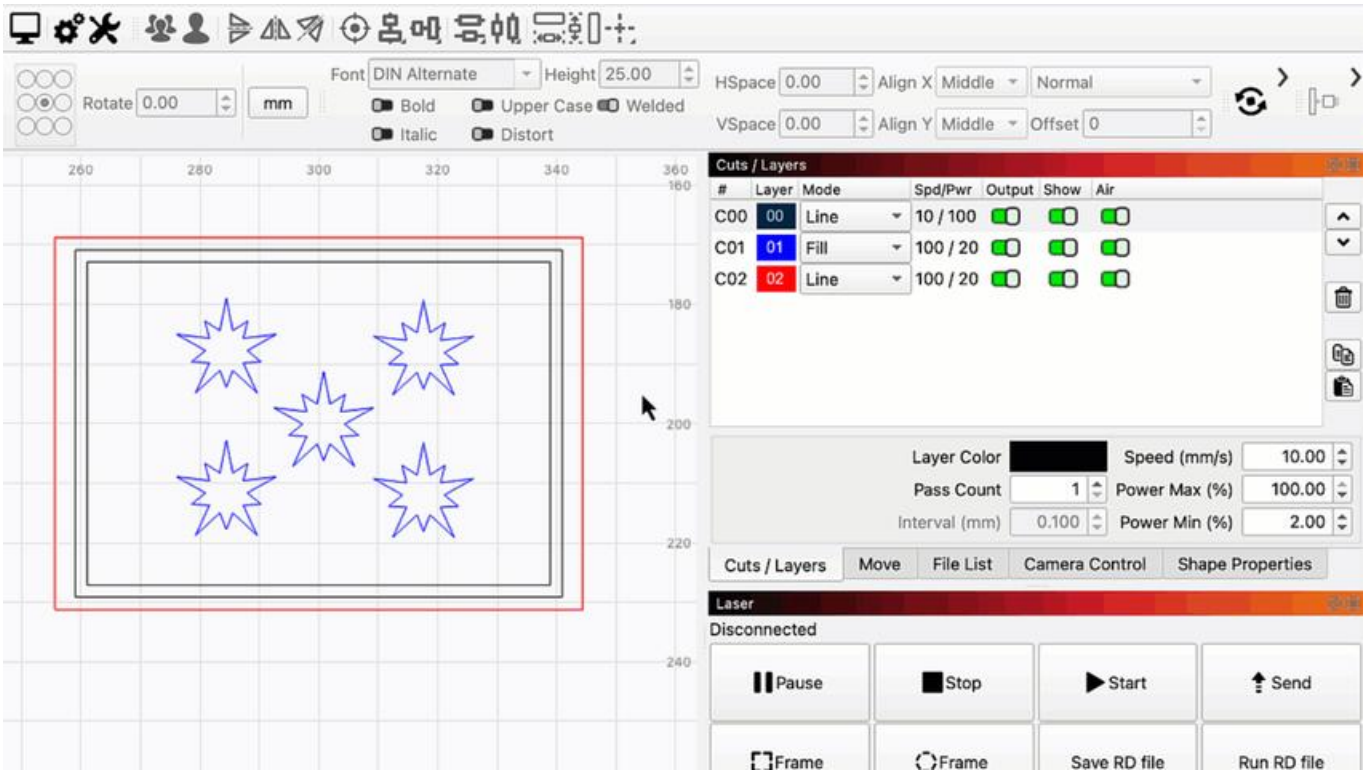
Toggle to enable or disable Air Assist for individual layers in your project.

Air assists blow air onto the surface of your material at the spot where the laser's beam strikes, which can help prevent soot accumulation and charring, and aid in cutting.

> **Note**
>
> Many lasers do not come with air assists, and others have manual air assists that you can't trigger through software. In those cases, toggling the Air switch will have no effect either way.
>
> Consult your laser's manufacturer if you're not sure if your laser has a built-in air assist.

Column Header Context Menu

Right-click anywhere over the column headers to open a context menu with options for enabling, showing, and sorting layers.



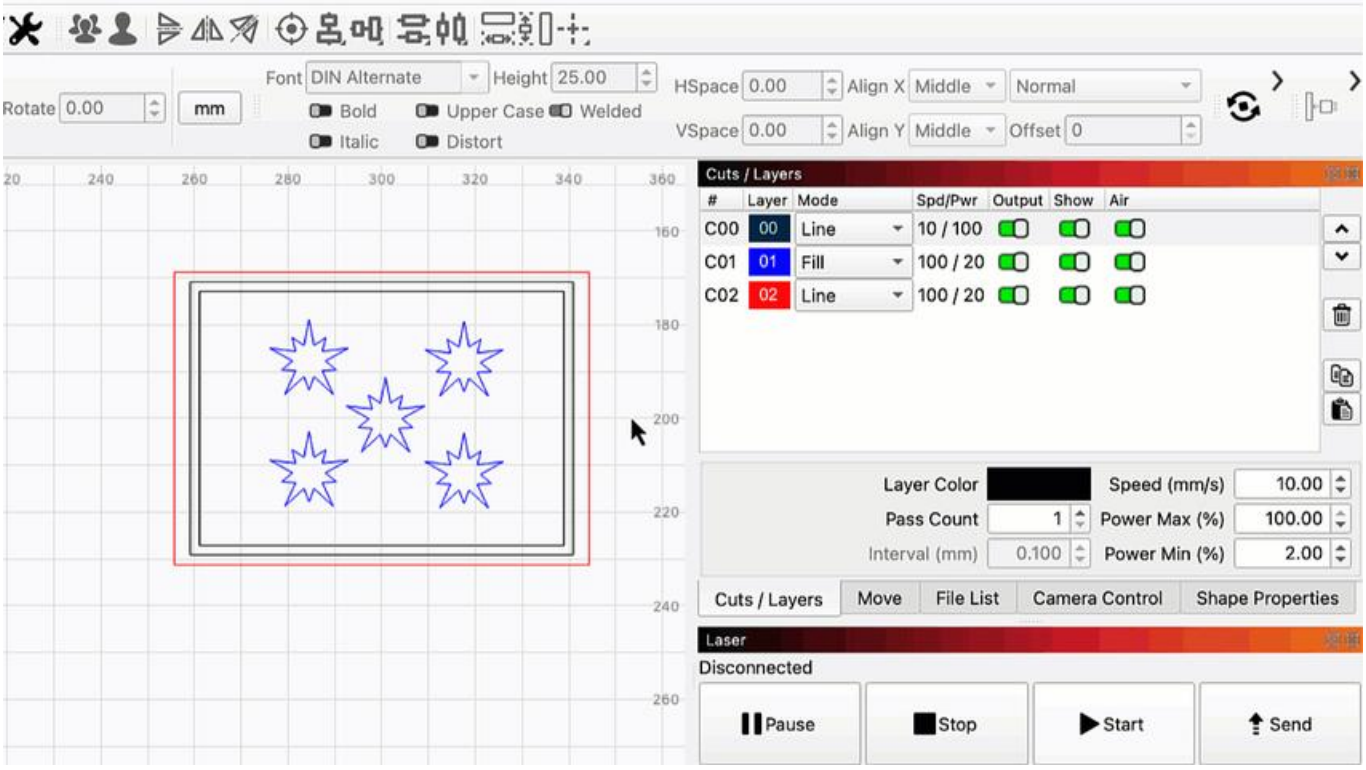
Option	Action
Enable all layers	Turn on the Output switch for all layers
Disable all layers	Turn off the Output switch for all layers
Invert enabled layer	Turn off the Output switch for all currently enabled layers and turn off for all currently enabled layers
Show all layers	Turn on the Show switch for all layers
Hide all layers	Turn off the Show switch for all layers
Invert visible layers	Turn on the Show switch for all hidden layers, and turn off for all visible layers
Enable all air assist	Turn on the Air switch for all layers
Disable all air assist	Turn off the Air switch for all layers
Invert all air assist	Turn off the Air switch for all layers it is currently enabled for, and turn off for all layers it is currently disabled for
Sort Cuts Last	Move cut layers to the bottom the list. See below for more information.

Layer Context Menu

Right-click over any layer in the list to open a context menu with options to disable, hide, flash, or select layers.

Flash to Highlight Layer

You can also hold down `Shift` and **right-click** a layer to flash the layer's contents.



Option	Action
Disable this layer	Turn off the Output switch for this layer
Disable all layers but this one	Turn off the Output switch for all layers except the layer you right-clicked
Hide this layer	Turn off the Show switch for this layer
Hide all layers but this one	Turn off the Show switch for all layers except the layer you right-clicked
Select all shapes in current layer	Select all shapes on the layer you right-clicked
Flash the content on this layer	Flash the objects assigned to the layer you right-clicked

Settings

Pass Count

Sets how many times the laser will repeat the shapes on this layer.

Speed

Adjust the Speed to run the laser at for this layer.

Power Max

Adjust the maximum power output level of the laser for this layer.

Power Min

Adjust the minimum power output level of the laser for this layer.

> **Note**
>
> This setting is only available when you're using a DSP laser or are engraving an image using Grayscale Mode, with any type of laser other than UV.

Interval

Adjust the space between each scan line in a Fill or Offset Fill operation.

Frequency

> **Note**
>
> Available Galvo lasers only.

Adjust the Frequency setting for this layer.

Q-Pulse

> **Note**
>
> Available MOPA and UV Galvo lasers only.

Adjust the Q-Pulse Width setting for this layer.

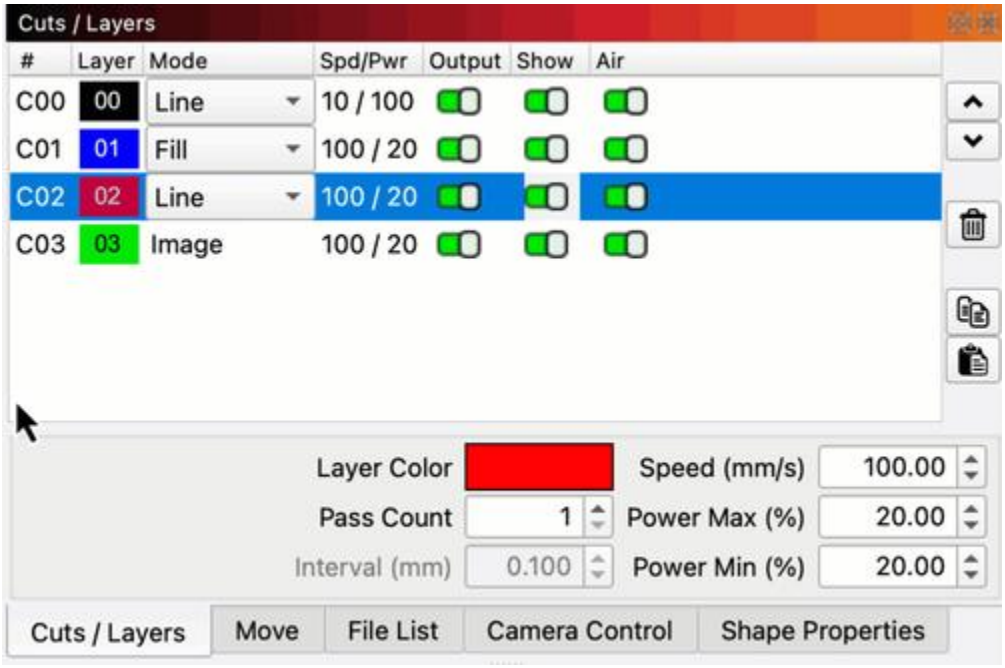
Rearrange Layers

By default, layer operations are performed by your laser in the order from top to bottom in which they are listed in the **Cuts / Layers Window**.

> **Note**
>
> See Optimization Settings for information on alternate options for ordering operations.

The order that layers are completed is often important, as pieces shift after being cut out. It's therefore best to engrave images and marking/scoring lines before any cuts through the material.

To change the order of the selected layer in the list you can click and drag to change its position or select a layer and click the up and down buttons in the window.



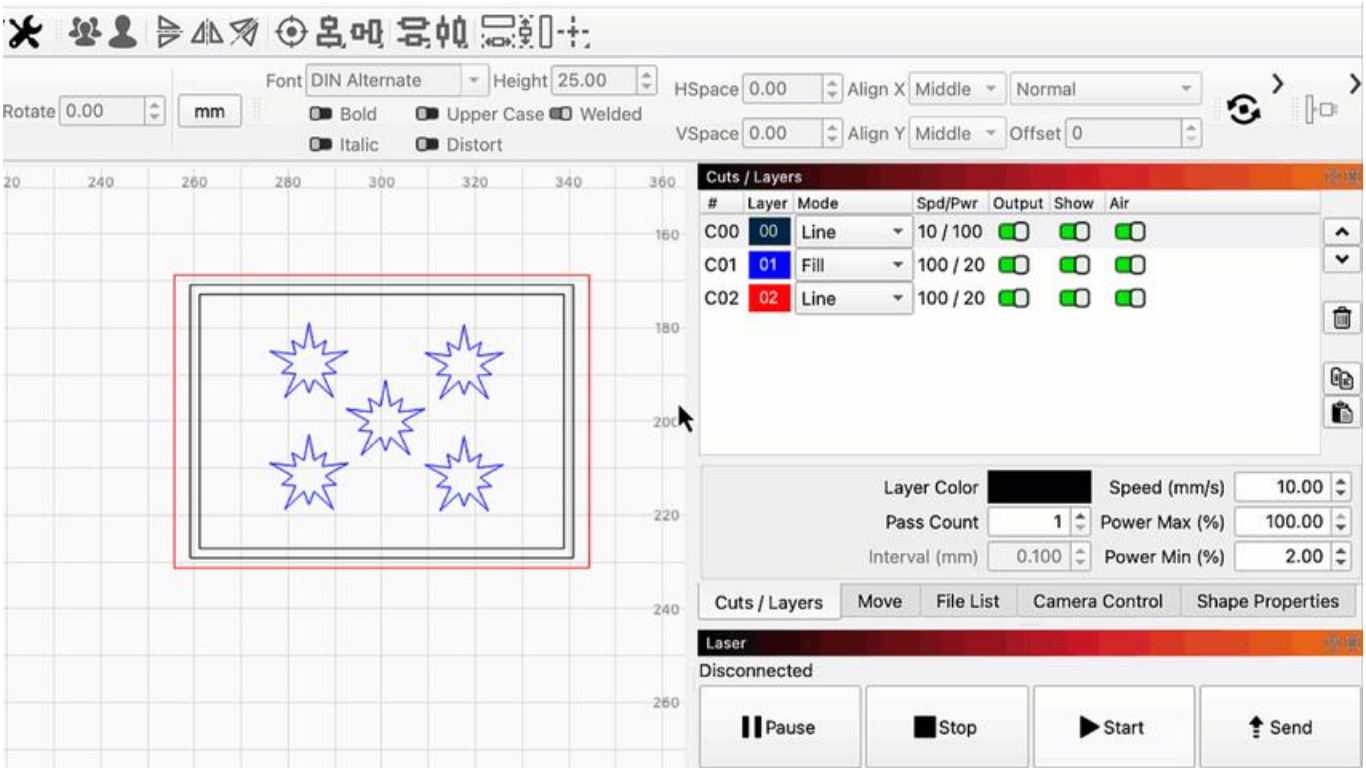
Sort Cuts Last

You can also automatically sort all layers, based on how likely they are to cut through the material — with the least likely being performed first.

To do so, right click any heading on the table (e.g. **Mode**) and choose **Sort Cuts Last**.

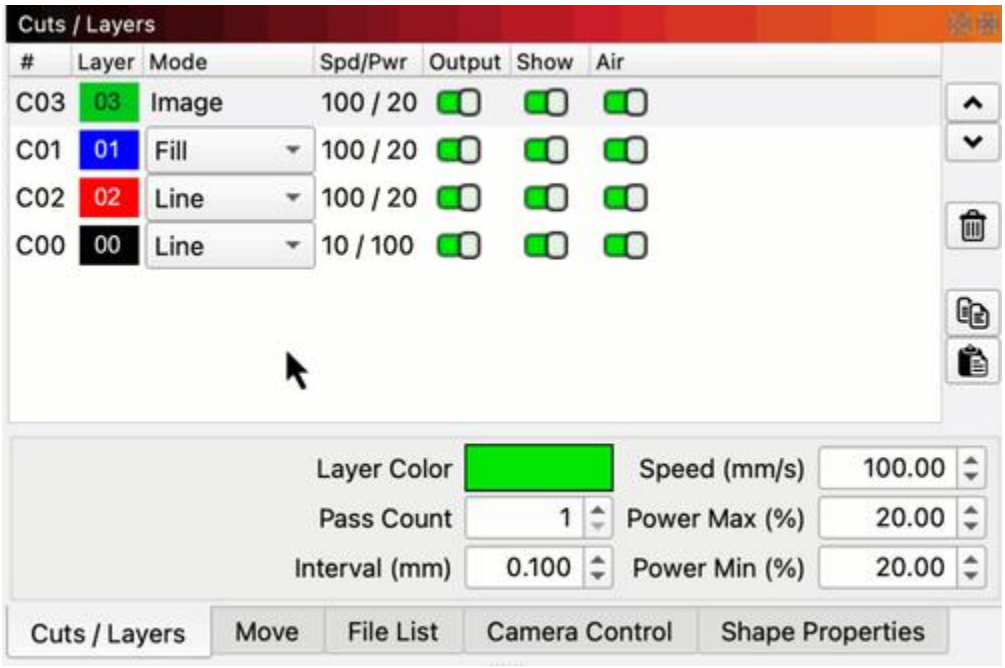
This process orders layers first by their strength — stronger layers have higher **Power**, lower **Speed**, and more **Passes**. It then pushes all Line Mode layers to the end, so all engraving operations are performed before cutting operations.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=avAbkTTa_cE



Copy Layer Settings

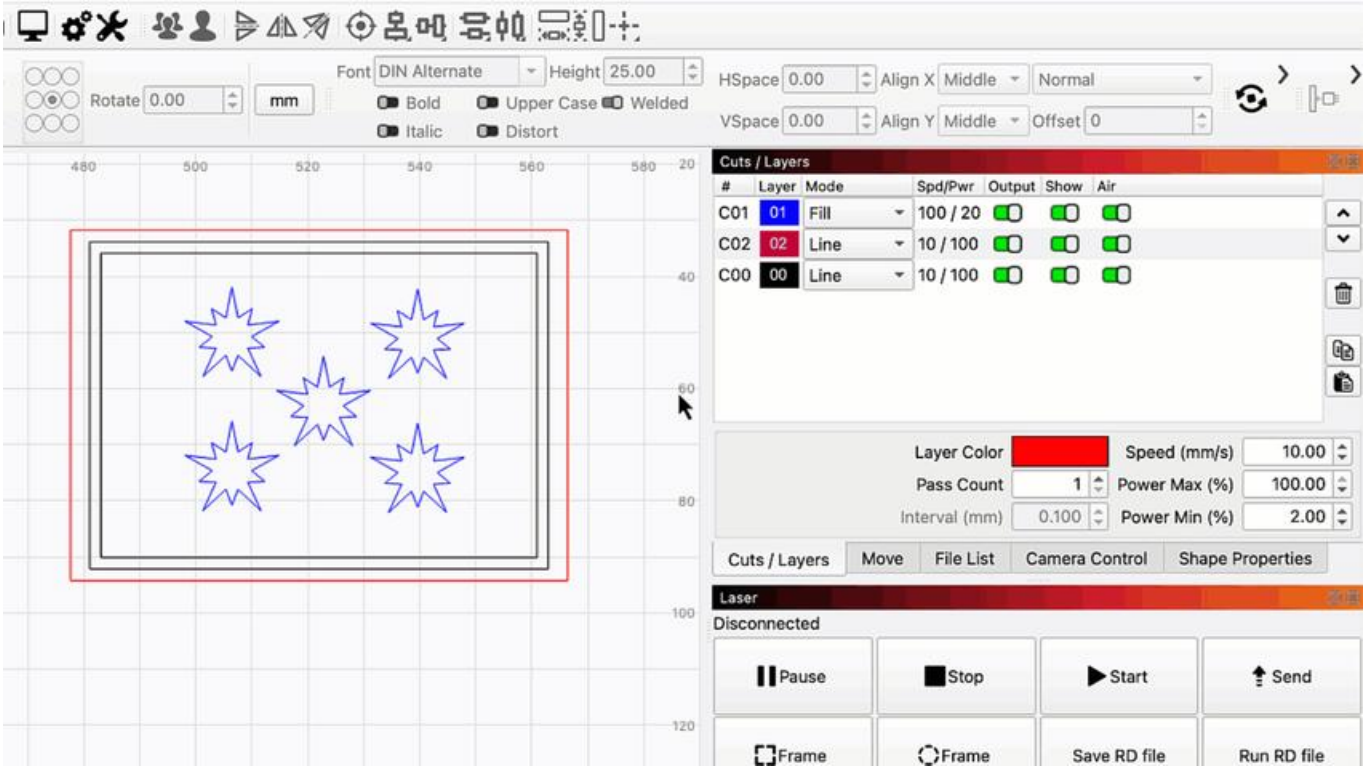
To copy settings between layers, first select a layer in the list and click **Copy** button to copy its settings. Next, select the layer you want to copy to, and click the **Paste** button.



Delete Layers

Click the **Delete** button to delete a layer and all objects assigned to it in your Workspace.

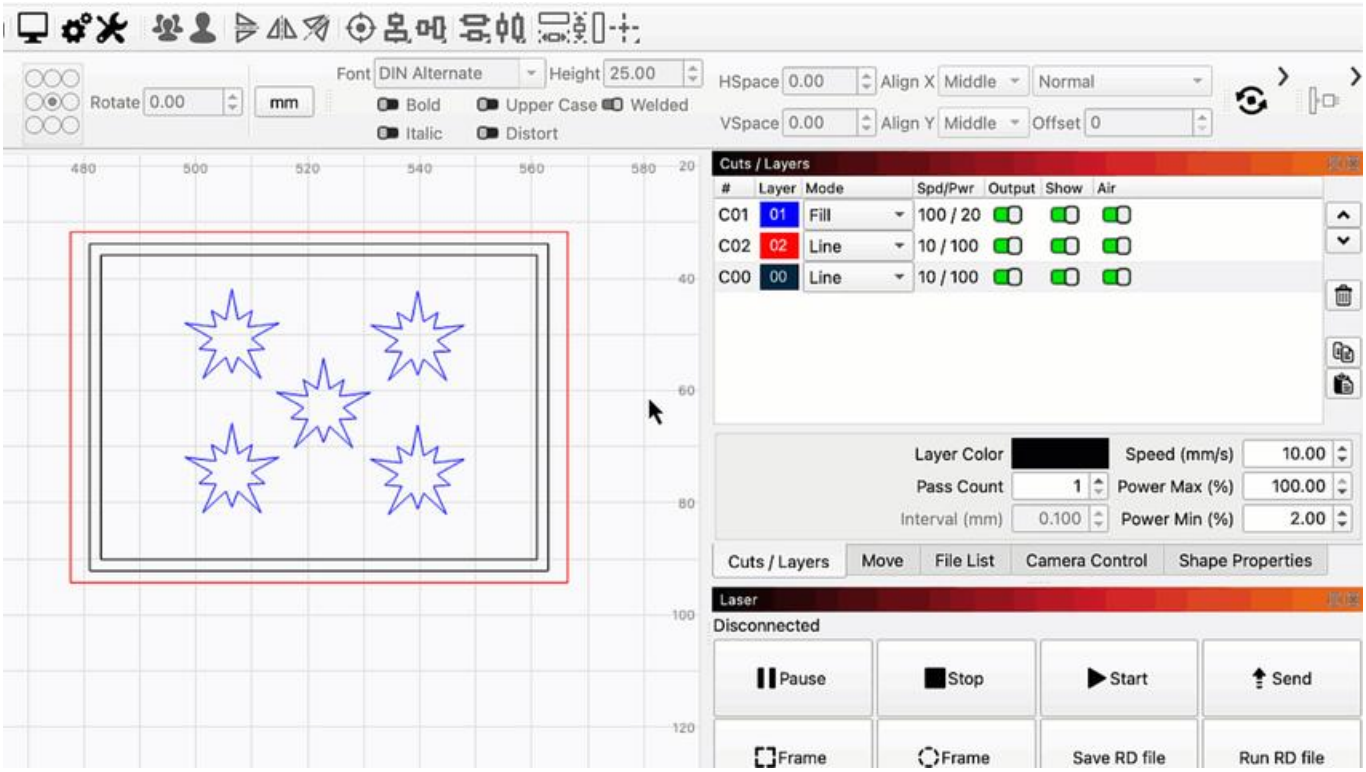
By default, you'll see a warning message asking if you're sure — hold `Shift` while clicking the Delete button to bypass the warning message.



Select All Objects on a Layer

In addition to right-clicking a layer and choosing **Select all shapes in current layer** from the context menu, you can hold `Shift` and left-click any layer in the list to select all objects on that layer.

You can also select any object in your Workspace, then go to Edit → Select all shapes in current layer.



Related Topics

- Cut Settings Editor
- Color Palette

Cut Settings Editor

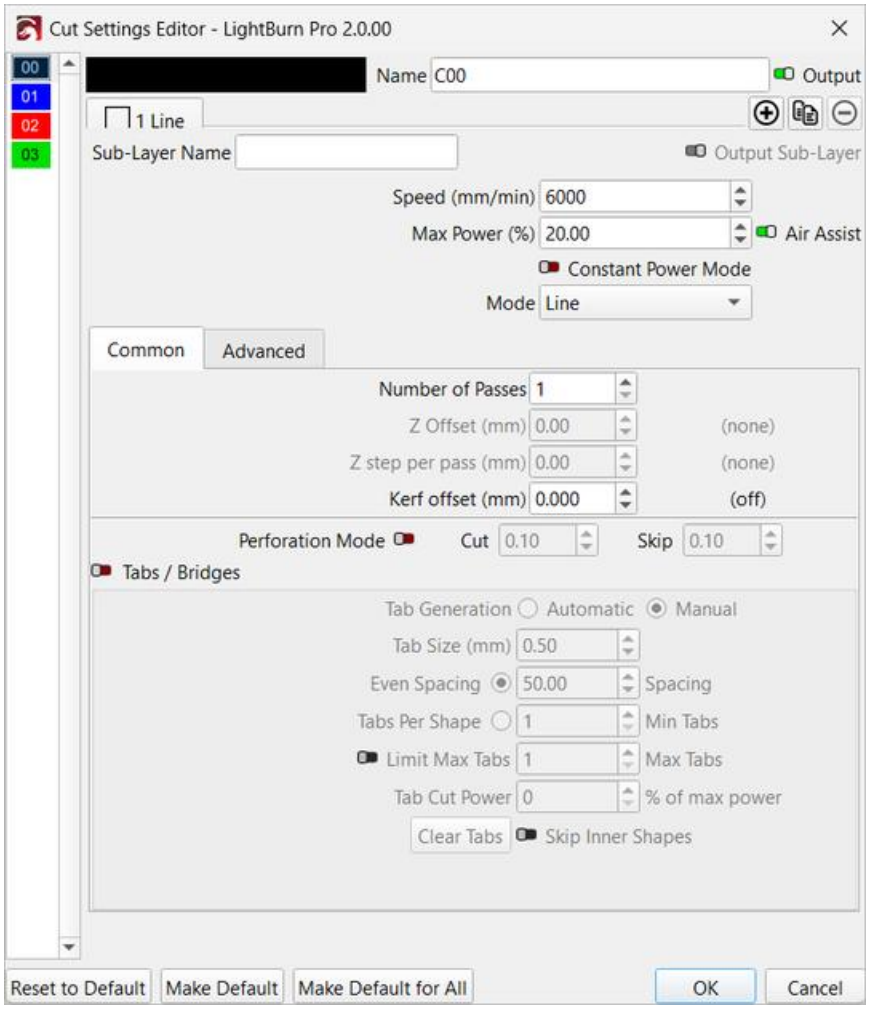
The **Cut Settings Editor** is where you adjust all settings for the layers in your project.

The settings presented in the Cut Settings Editor vary depending on the type of laser you're working with, and the Layer Mode the selected layer is set to.

In all variations of the Cut Settings Editor, the most essential settings — including **Speed**, **Power**, and **Mode** — are displayed at the top of the window. More specialized options follow below, split into two tabs — **Common** and **Advanced**.

Access the Cut Settings Editor by double-clicking any entry in the Cuts / Layers Window or Material Library.

The window will open to settings for the entry you double-clicked — click any other layer in the column along the left side to access the settings for that layer.



The Cut Settings Editor for a layer set to Line Mode, using a GCode-based laser.

> **Note**

> You can adjust a narrower set of critical settings directly in the Cuts / Layers Window, instead of opening the Cut Settings Editor.

The settings presented in the Cut Settings Editor depend on the selected Layer Mode and the type of laser you're working with. Select an option below to learn more about different groups of settings.

Main / Shared Settings

Settings shared by almost all types of lasers and Layer Modes, including critical settings like **Speed** and **Power**.

Read More

Line Mode

Tells your laser to trace a path along the outlines of geometry in your project.

Read More

Fill Mode

Tells your laser to etch or engrave parallel lines inside the bounds of closed shapes in your project.

Read More

Offset Fill Mode

Tells your laser to fill closed shapes with lines that follow the contour of the outer shape.

Read More

Image Mode

Works similarly to **Fill Mode**, but applies additional special settings to adjust output based on the pixels in your image.

Read More

Galvo-Specific Settings

Special settings that are available for users with Galvo style lasers only.

Read More

Sub-Layers

A tool for setting up and applying multiple cut/engraving settings to the same geometry or layers in your project.

Read More

Default Layer Settings

Built-in or custom-created settings stored on a per-layer basis, that you can reset to at any time.

Read More

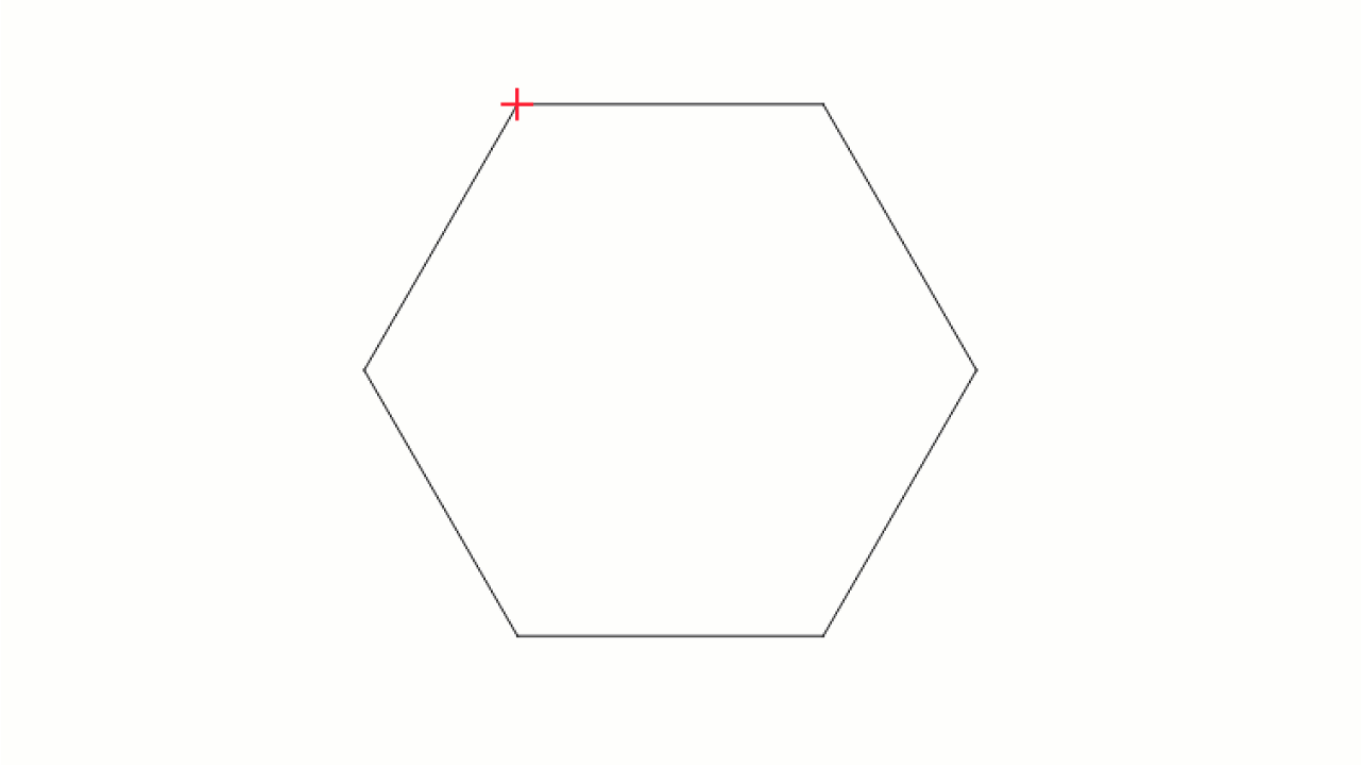
Note

Some options in the Cut Settings Editor are hidden if you have Beginner Mode enabled.

Line Mode

Line Mode tells your laser to trace a path around the contours of vector graphics.

If you command the laser to move quickly, or with low Power, it will likely just etch the surface (sometimes called scoring). If you move slowly and with high power, especially with a CO_2 laser, the laser will cut deeper, or even completely through the material. The only difference between surface marking and cutting with Line Mode is the Power and Speed you set.



Some settings, such as Wobble, are only available for Galvo lasers. See Galvo-Specific Cut Settings for more information about these settings.

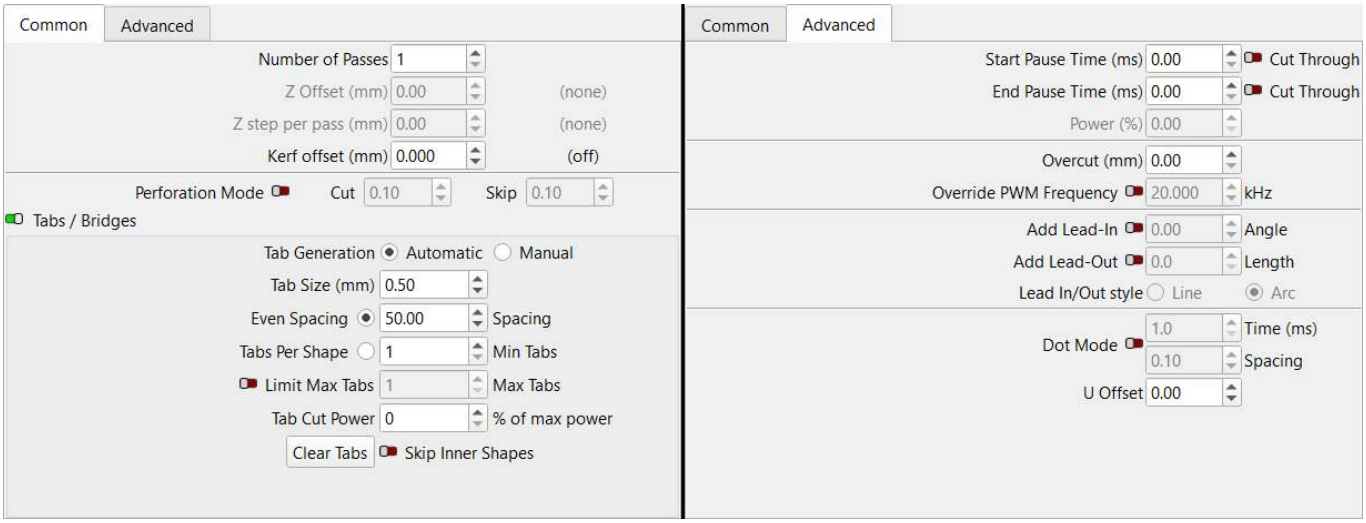
Line Mode Settings

There are several specialized options available in the Cut Settings Editor for layers set to **Line Mode**, split into two tabs, **Common** and **Advanced**.

> **Note**
>
> See Main / Shared Settings for information on applying essential settings like Speed and Power.
>
> See Additional Options for information on specialized settings not covered on this page.

Click any option in the images below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.

Many of the settings presented in the images below are available for users with DSP lasers only, but most are shared between all types of lasers.



Left: Common Line Mode settings tab

Right: Advanced Line Mode settings tab

Common Settings

Number of Passes

How many times the laser will repeat the shapes on this layer. Sometimes, when cutting thick material, trying to engrave very deep, or using a lower power laser, more than one pass may be necessary.

Z Offset

If you have Z movement enabled, and your controller supports it, you can use the **Z offset** setting to move the laser head closer to the material (inward) or farther away from it (outward). Positive values move the Z Axis inward, and negative values move it outward.

Focusing deeper into the material can sometimes help to cut thicker material, and lifting the laser away from the material can produce a thicker line.

> **Warning for Z Moves**
>
> Z moves have the potential to physically damage your laser by crashing the head into the material.
>
> If your system uses auto-focus, you may not be able to push the focus point lower, because most systems treat the auto-focus height as the lowest possible height (Z limit) to avoid crashing the laser head.

Z Step per Pass

When doing more than a single pass over a shape, the Z step per pass setting allows you to tell LightBurn to raise or lower the laser with each pass by the same amount.

By allowing you to shift the focus point deeper with each pass, this is often useful for cutting thick materials or achieving deep engravings.

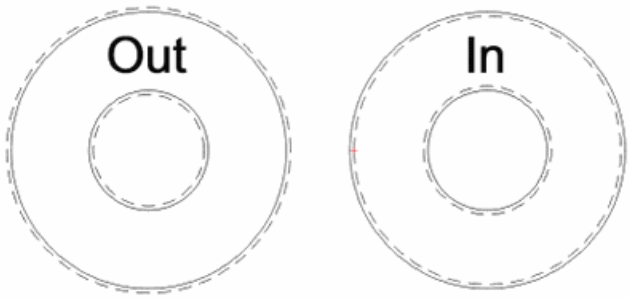
Kerf Offset

> **Closed Shapes Only**
>
> This feature only works with shapes that are closed. A shape is closed when it is a complete, continuous loop whose start and end points are the same. When a shape's start and end points are different, it is open.
>
> See Open vs. Closed Shapes for more information.

Kerf refers to the thickness or width of a cut made by a cutting tool. The kerf of a laser is much narrower than that of a saw blade, but the extra material removed can interfere with close-fitting parts such as tabbed boxes or inlays.

Kerf Offset compensates for the extra material removed by offsetting the path of the laser to the outside of closed shapes (for positive values) or the inside (for negative values). This doesn't modify the underlying shape, so you can use your design on lasers with different kerfs by adjusting the Kerf Offset value.

The image below shows how Kerf Offset (dashed lines) relates to the original shape (solid lines).



Perforation Mode

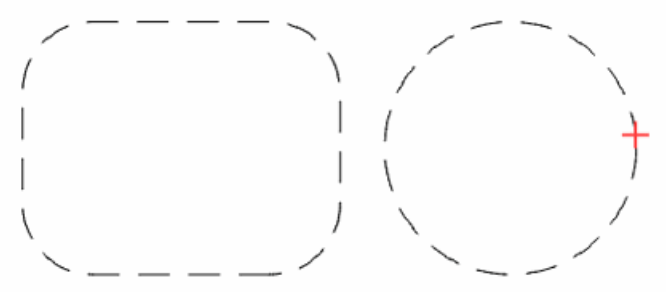
Enable **Perforation Mode** to specify a distance to **Cut**, followed by a distance to **Skip**.

As the laser traces the contours of shapes assigned to this layer, it will turn on over the specified Cut distance, and off over the specified Skip distance.

Perforation Mode is useful if you're trying to cut fold lines in card stock, stitch holes in leather, or make dashed lines.

> **Use Perforation Mode to reduce effective power output**

This can also help when using delicate materials with a powerful laser. With very low cut and skip values, the beam is rapidly switched on and off, reducing the effective power output. Try starting with cut and skip values of 0.1 mm each, and adjust the ratio between cut and skip to tune it further.



▶ Watch the video on YouTube: https://www.youtube.com/watch?v=vCZNB5B01_c

Tabs / Bridges

Tabs are small sections of cuts that the laser is commanded to skip, used to prevent objects that you are cutting from immediately falling out of the base material. When sized appropriately, the tabs can be broken with minimal force and the objects removed from the material after the job is finished cutting.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=xn_CJT6IoOk

See Add Tabs for information on creating and adjusting **Tabs / Bridges** settings.

Advanced Settings

Under the **Advanced** tab you'll find several less frequently-used options. Many of the options presented here are available only for particular types of lasers.

Note

The Advanced tab is not available if you're using a Galvo laser. See Galvo-Specific Cut Settings for information on advanced settings for Galvo lasers.

Start / End Pause Time

Use these options to add a delay, in milliseconds, at the beginning of a cut, the end, or both.

If you also enable the **Cut Through** option, the beam will be turned on during this pause, which can be used to prime the cut and help pierce through thick material. The **Power %** value sets the power of the laser during this pause.

Without the **Cut Through** toggle enabled, it can add a delay to let the gantry settle down after a rapid move, or allow time for an air assist solenoid to engage.

> **Note**
>
> This setting is not available for lasers with TopWisdom controllers. Dwelling without Cut Through enabled is not available for lasers with Trocen controllers.

Overcut

Enable **Overcut** to tell your laser to extend a cut by a specified amount past the end of closed shapes, on the final Pass.

Override PWM Frequency

If you have an RF Excited tube, this setting allows you to control the PWM frequency of the pulses sent to the laser, and can change the edge finish on certain materials.

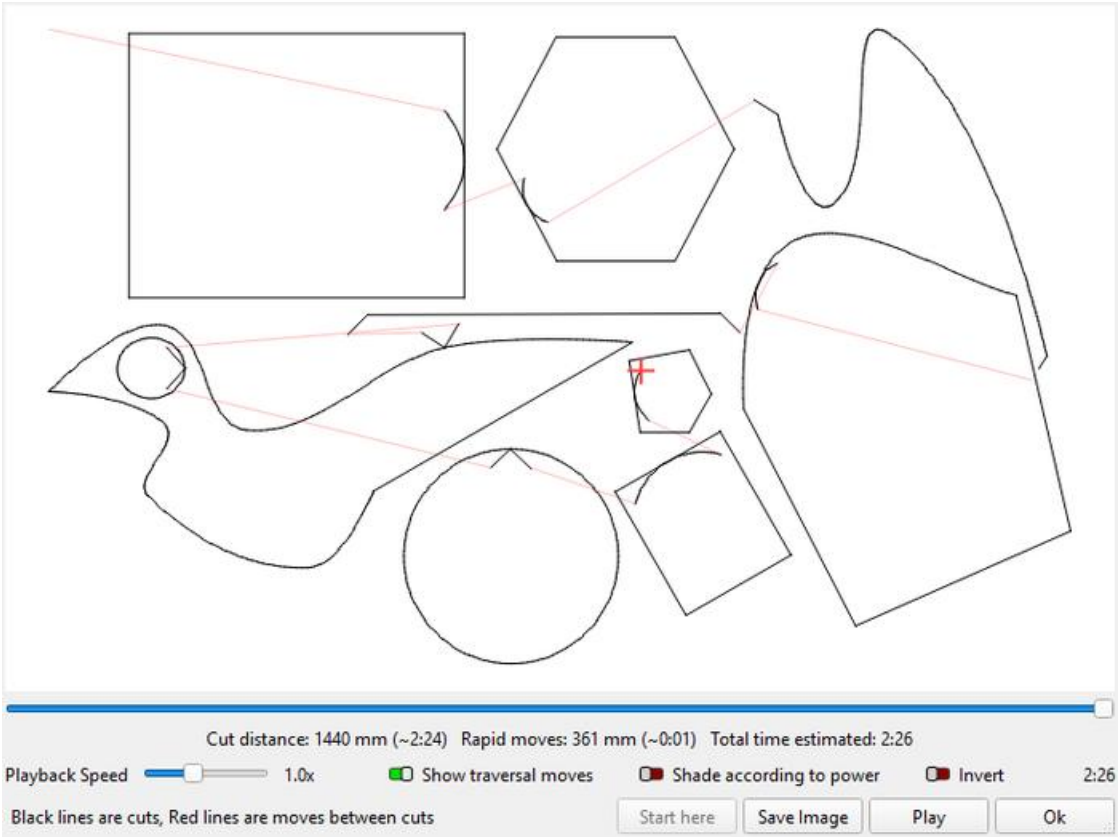
> **Note**
>
> This setting is only available for lasers with Ruida controllers.

Enable PPI

The **PPI** setting (Pulses Per Inch) lets you tell the laser to send an exact number of pulses per inch of travel, instead of varying the beam power directly. This is useful for delicate materials like paper, and is similar to using Perforation Mode, but is handled by the hardware itself.

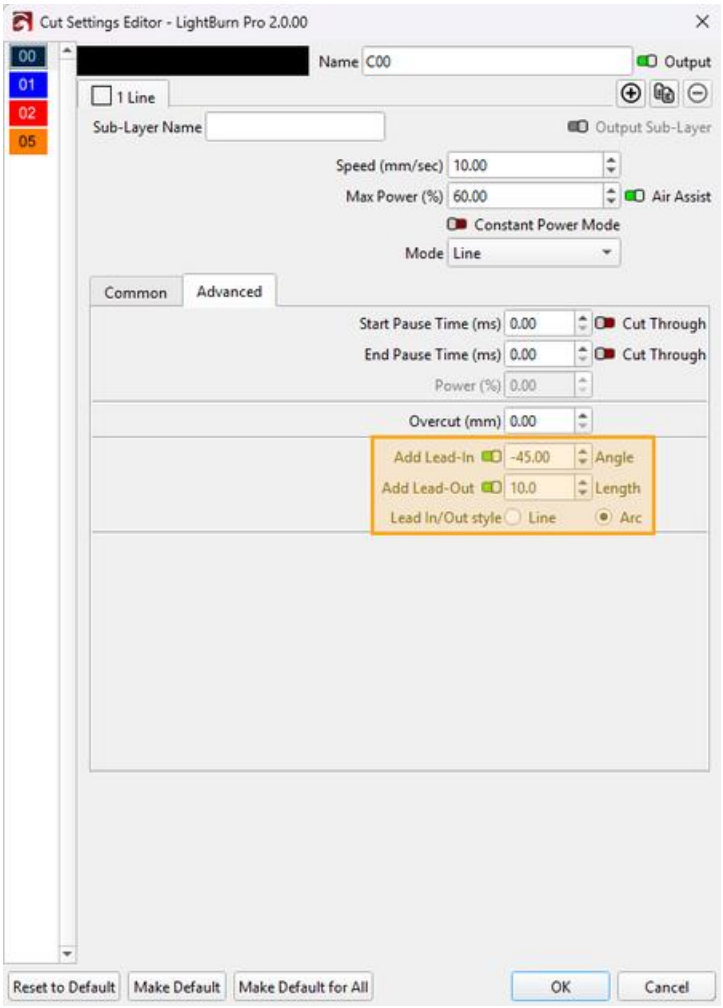
> **Note**
>
> This setting is only available for lasers with Trocen controllers.

Lead-In / Lead-Out

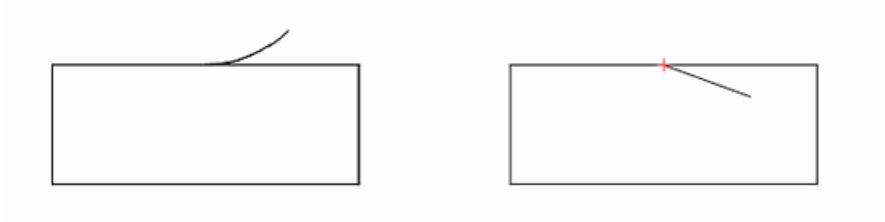


Lead-In and **Lead-Out** are optional lines or curves added to the start or end of a cut. You can toggle each on or off individually, and specify their **Angle**, **Length**, and **Style**.

When cutting thick material, particularly if using **Cut Through** delays, or cutting metal, there is often a mark left at the start of the cut that is slightly thicker than the rest, and can leave an indent on an otherwise smooth edge. With this setting, you can add a small line to the start or end of a cut, so that mark happens off the cut line itself.



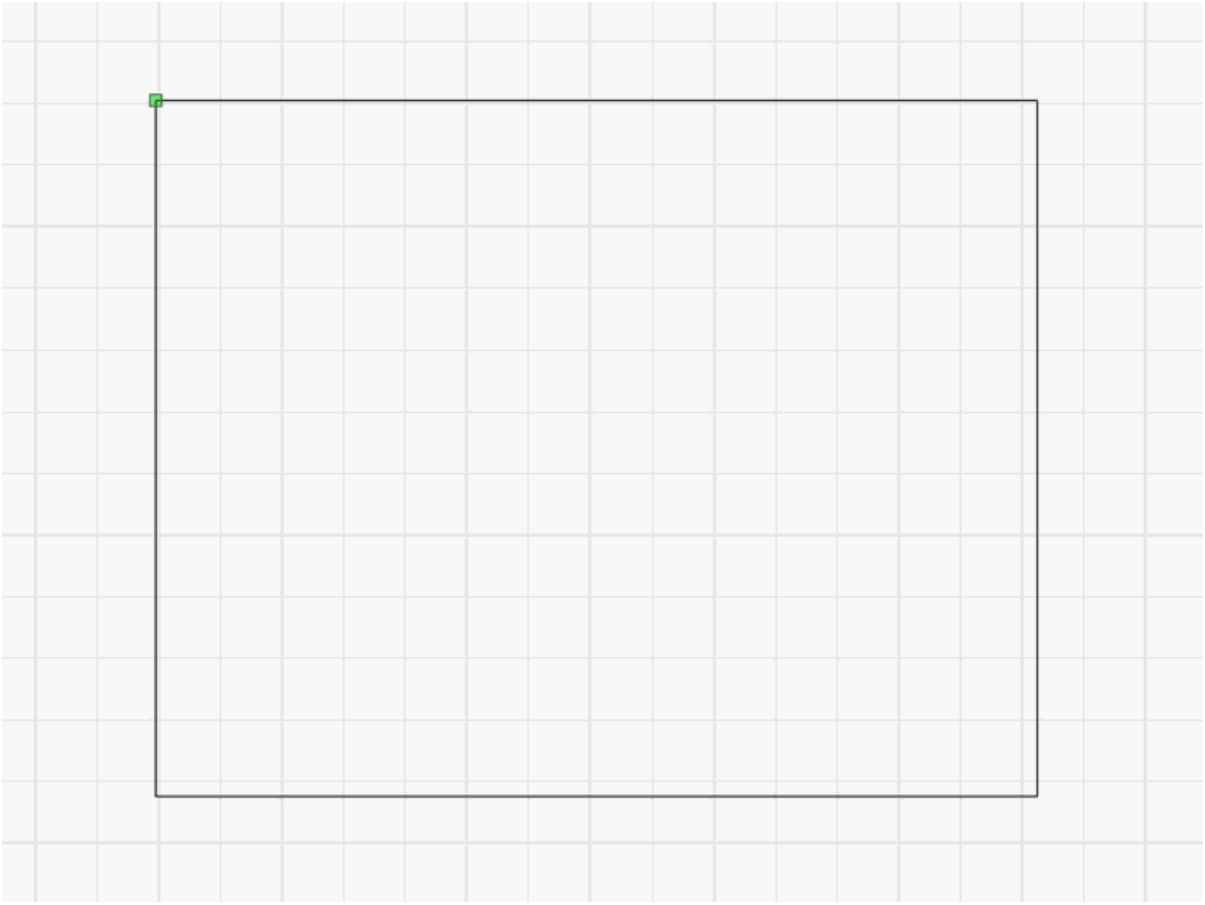
The **Angle** value controls the placement of the lead line — a positive number tells LightBurn to put the lead line outside the shape, and a negative number means to put it inside. You can choose to make the lead a straight **Line**, or a short **Arc**.



The shape above on the left is using an Arc set to +45 degrees, so it starts outside the shape. The one on the right is a Line set to -20 degrees, so it appears inside the shape.

Set Lead-In/Out Start Points

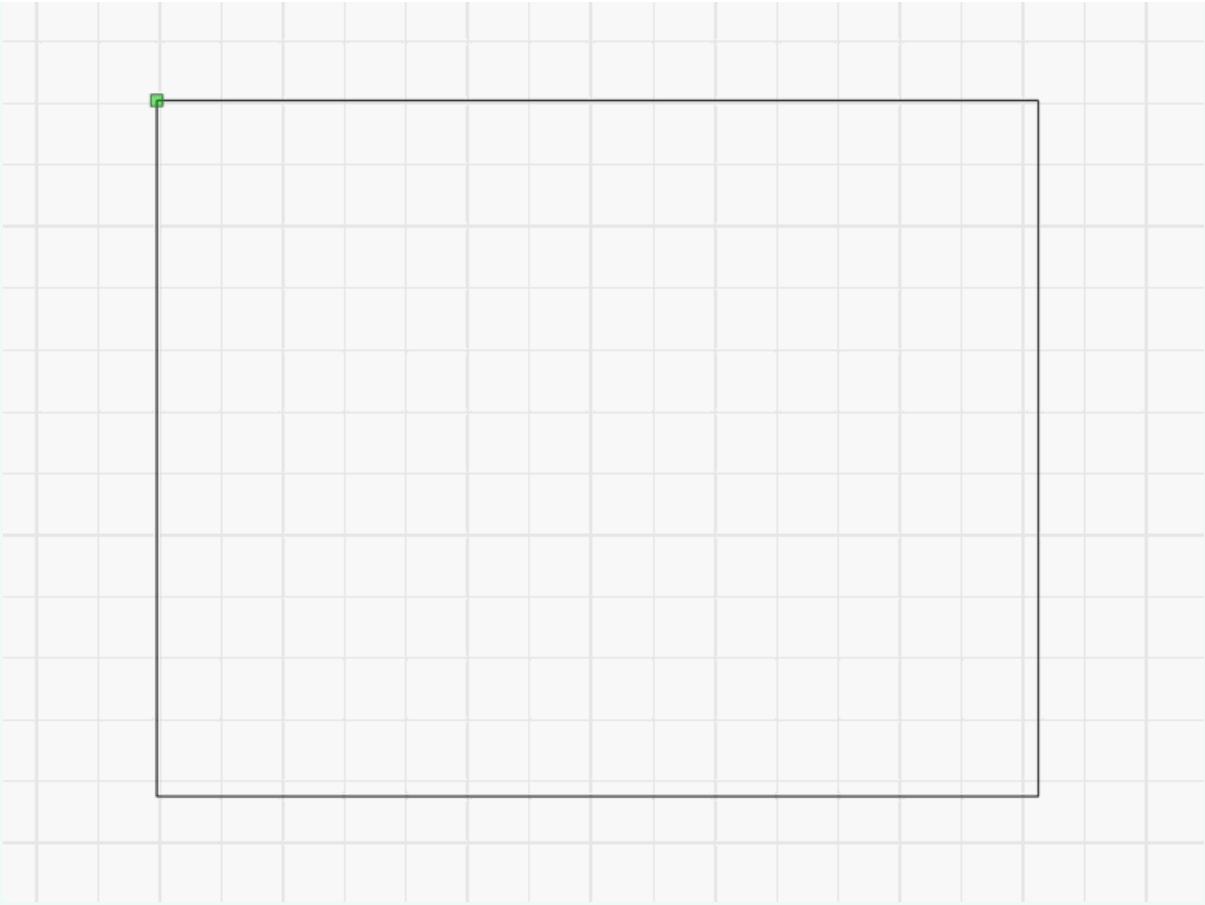
1. Click on the design element. Elements created with the Shape tool may not display any nodes or points along the selected elements, while path-based objects will display their nodes.



Rectangle Created With Create Shapes Tool | No Visible Nodes

2. With the desired shape selected, click on the Edit Nodes tool in the Creation toolbar and either:

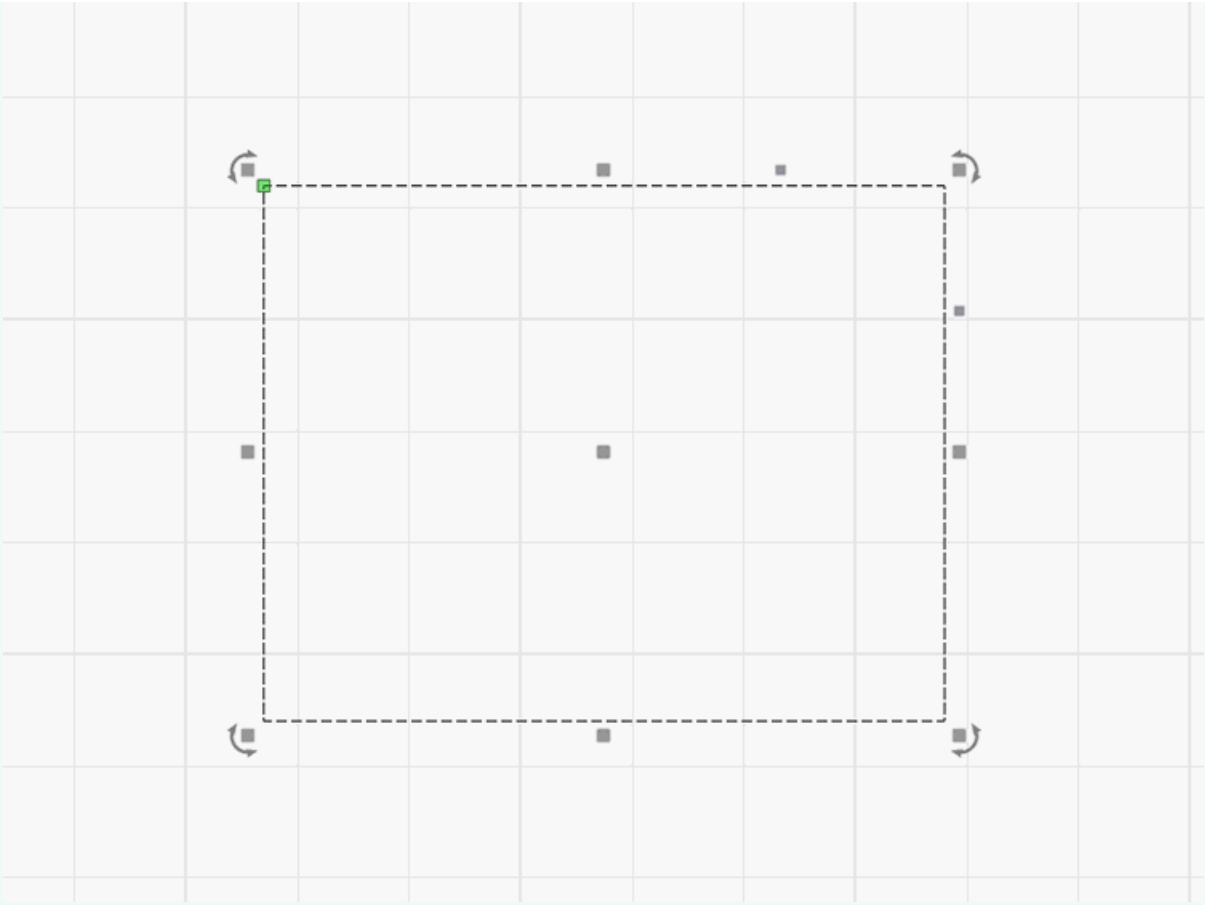
Objects Created With Draw Shapes Tool



No Visible Nodes

Convert to Path

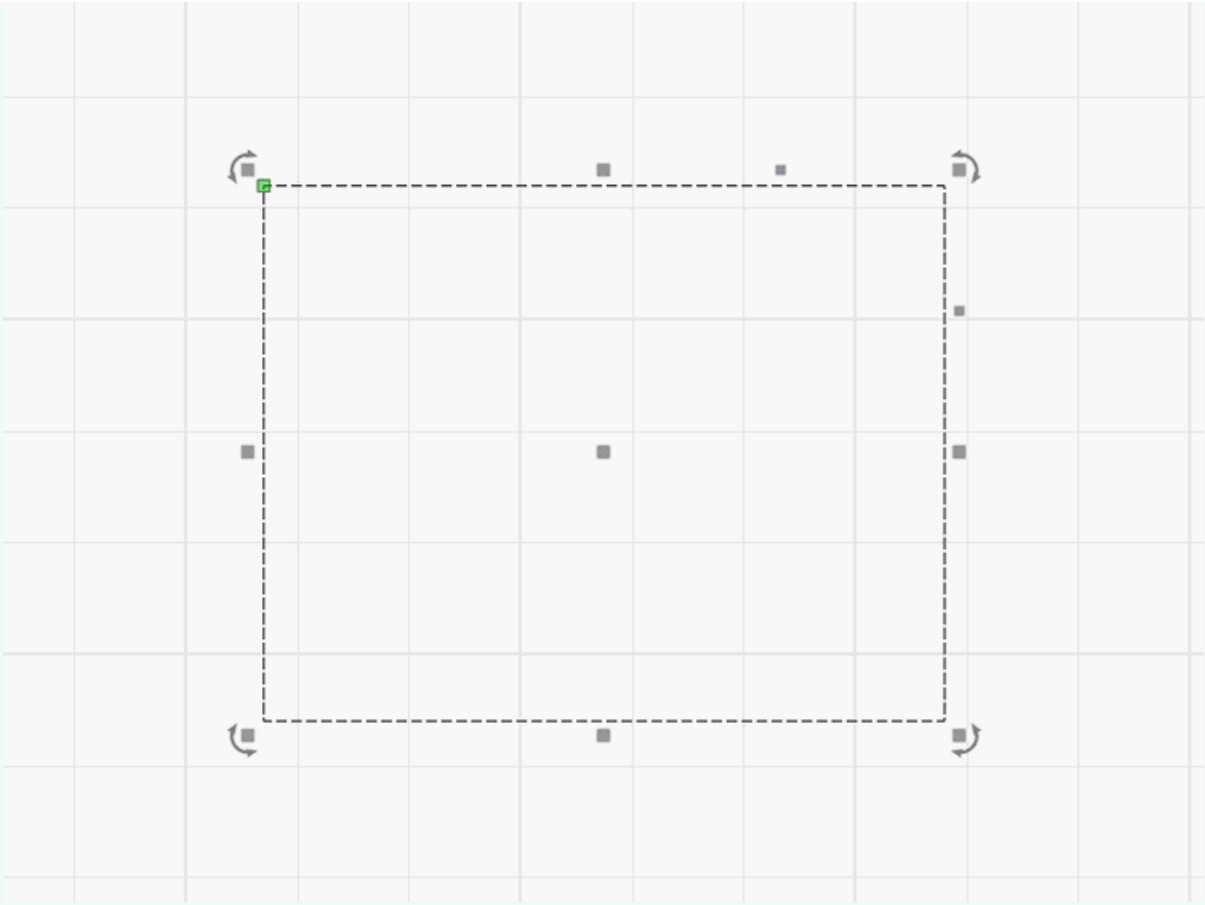
Objects created with the Draw Shapes tool may need to be converted to a Path object in order to access the object nodes.



Visible and Editable Nodes

a. With the object selected, click on the Edit menu and select Convert to Path, or on your keyboard press `Ctrl`/`Cmd` + `Shift` + `C`.

Objects Created With Draw Lines Tool or Path-based Objects



Visible and Editable Nodes

b. Continue to the next step.

3. Add an new node along the edge/face of the desired Lead-In/Out location. Curved shapes may be able to use existing nodes. Nodes that anchor acute angles (less than 90°) may not be compatible with internal Lead-In/Out locations.



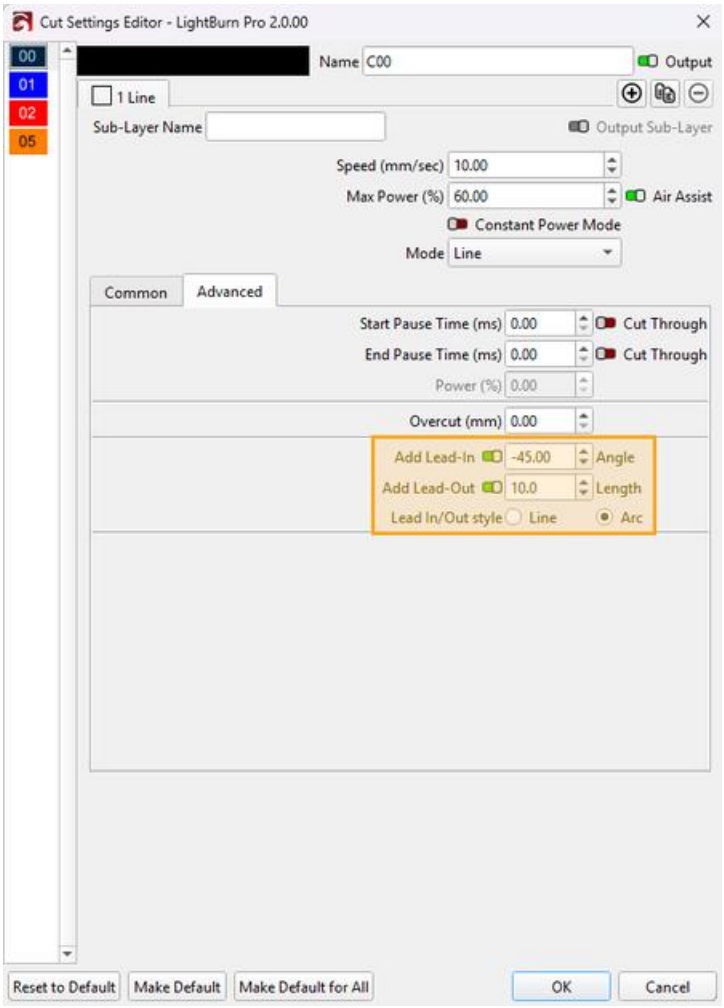
4. Select the Set Start Point tool and click on the new node to set that node as the shape's Lead-In/Out location.



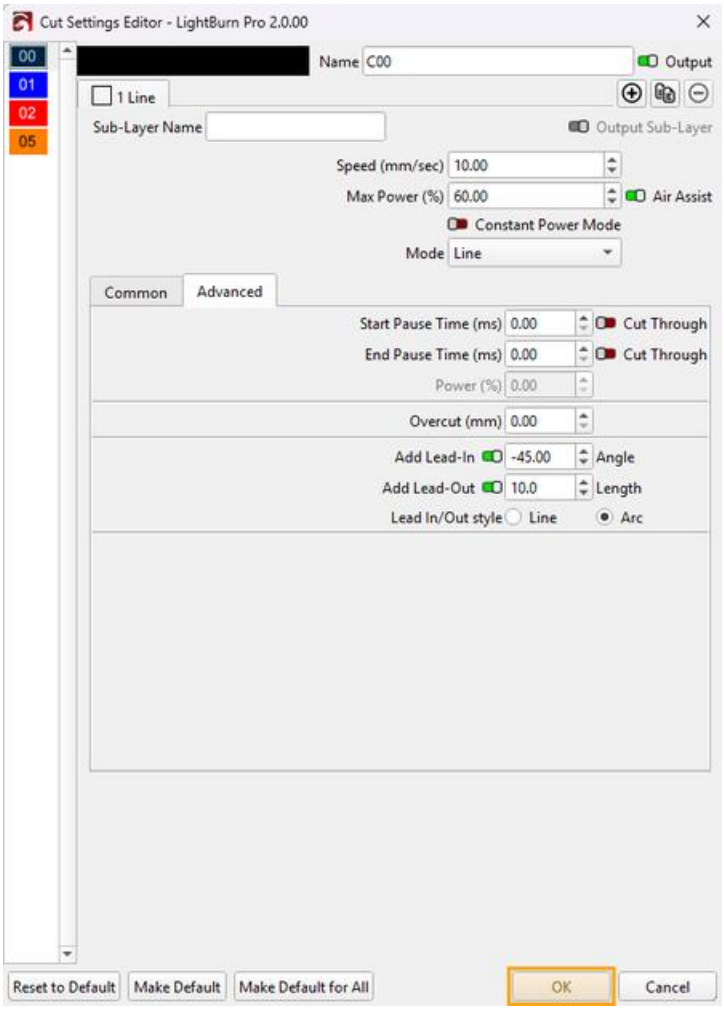
Activate Lead-In/Out at Start Points

Shapes with active Lead-In and Lead-Out points will share the same angle, length, and style settings on that layer. Assign shapes to different layers to use different Lead-In/Out settings in the same project.

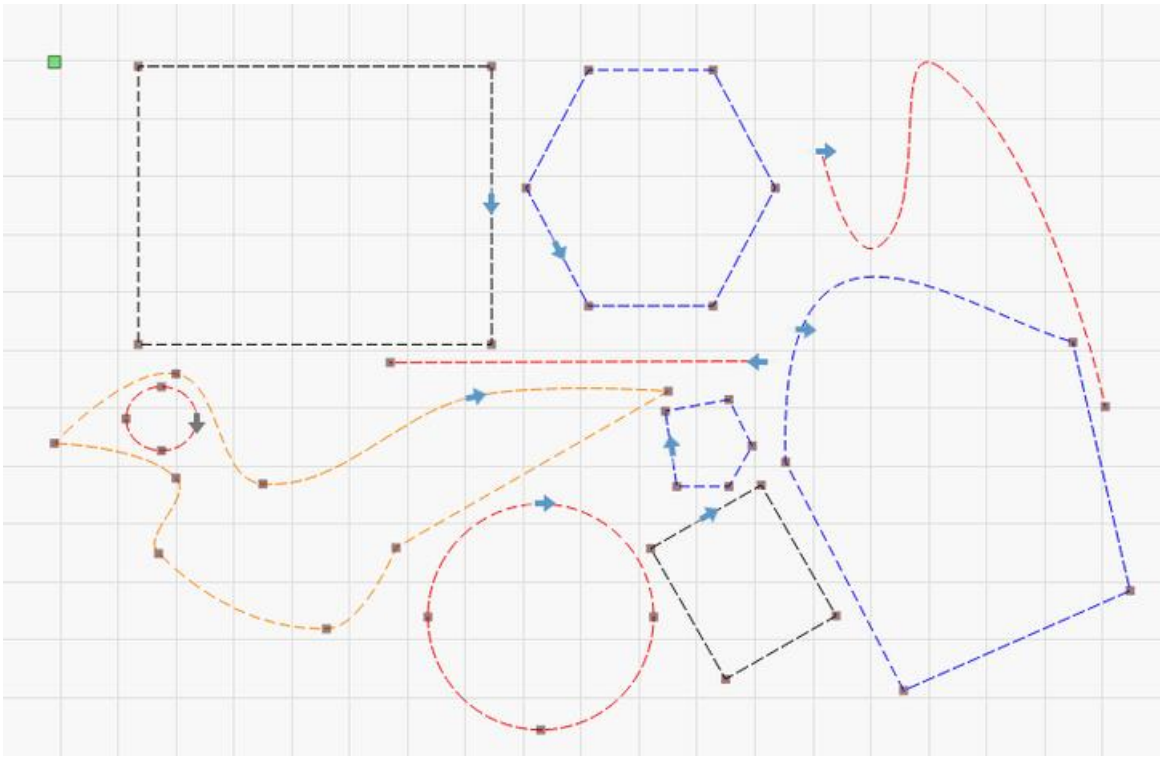
1. Open the Cut Settings Editor to activate the Lead-In/Out for that shape by double-clicking the assigned layer.



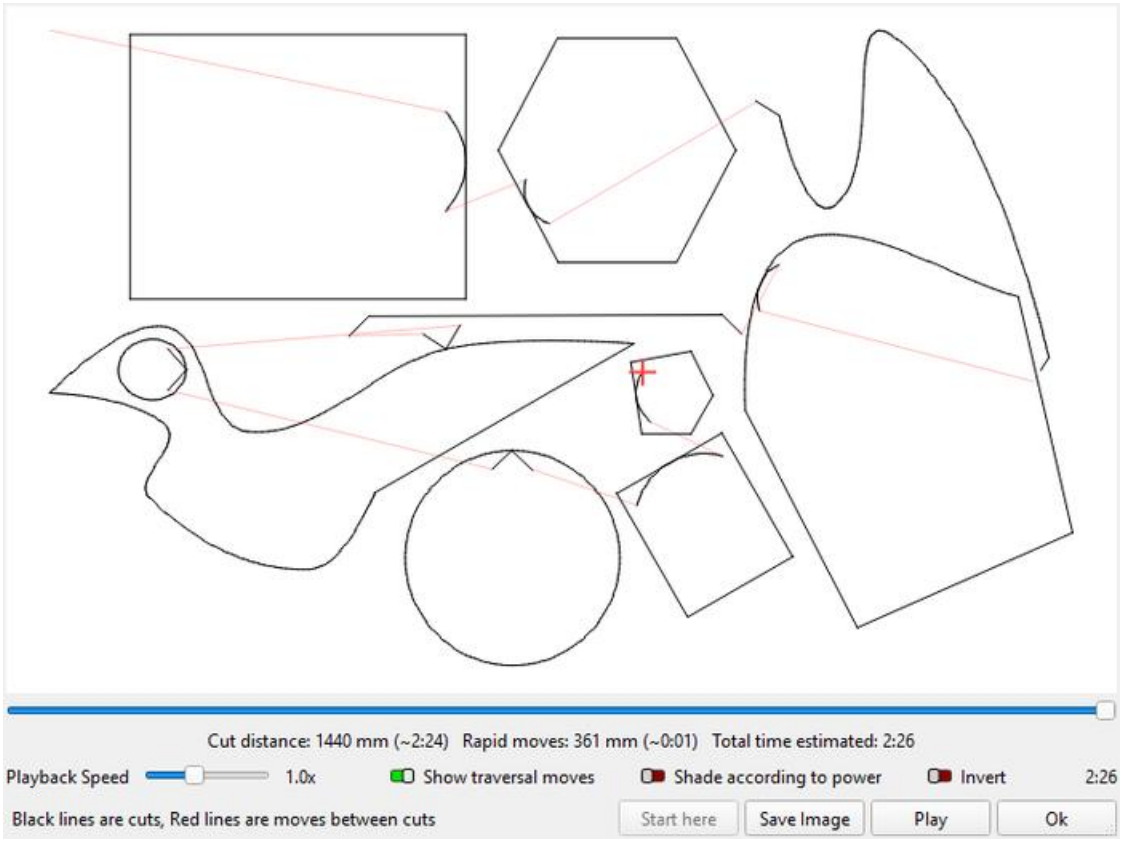
2. In the Advanced Tab, activate the Lead-In and/or Lead-Out toggle and set your
    1. Desired angle, with positive angles for Lead-In/Outs external to that object, or negative angles for internal Lead-In/Outs. Lead-In/Outs assigned to lines will use positive values for Lead-In/Outs above the line and negative values for Lead-In/Outs below the line.
    2. Desired Lead-In/Out length
    3. Choose between Line or Arc-shaped Lead-In/Outs.
3. Click on **OK** to save the Lead-In/Out settings for all shapes assigned to that layer. Use different layers for different Lead-In/Outs settings within the same project.



4. Assign additional Lead-In/Out points to any remaining shapes as desired and repeat the process for each additional layer as needed.



5. Click on the Preview Icon in the main toolbar to view the updated Lead-In/Outs for each assigned shape and start the job when ready through the Laser window.



Reducing Travel Moves

Set Cut Order Priority

Reduce the travel moves your laser cutter makes by manually setting the Cut Order Priority of each shape to align the Set Start Point assigned to one shape to the Start Point location on another shape.

1. Click on the Optimization Settings button in the Laser window to open the Cut Optimization Settings window.
2. Click on **Order by Priority** to use the **Cut Order Priority** assigned to each shape. Click on the **Remove** button as needed to remove existing entries or reorder by clicking and dragging on each **Order by** item.
3. Click **OK** to save the new job priority settings.
4. Open the Shape Properties window to assign the Cut Order Priority to each shape by selecting the **Shape Properties** tab found in the right-hand side of the main window by default and assign a priority to each shape starting from 0 (zero), onwards..

   > **Tip**
   >
   > The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

5. Open the Preview window by selecting the Preview window button in the main toolbar to view the updated cutting path and start the job once ready.

Dot Mode

When enabled, the laser will pause and pulse at regular intervals along the path, instead of cutting continuously. The **Time** value specifies the pause delay in milliseconds, and the **Spacing** parameter sets how far apart the pulses are.

This can be an effective way to do stitching holes, or cut very thin or delicate materials, but the constant pausing can shake the machine — using Perforation Mode is often preferable for this reason.

> **Note**
>
> This setting is only available for lasers with Ruida controllers.

U Offset

Enter a distance here to set a per-layer U Axis offset. To use this setting, your laser must support U Axis control, and you must enable the U Axis in Device Settings.

> **Note**
>
> This setting is only available for lasers with Ruida controllers.

Additional Options

The settings presented in the Cut Settings Editor depend on the selected Layer Mode and the type of laser you're working with. Select an option below to learn more about different groups of settings.

Main / Shared Settings

Settings shared by almost all types of lasers and Layer Modes, including critical settings like **Speed** and **Power**.

Read More

Line Mode

Tells your laser to trace a path along the outlines of geometry in your project.

Read More

Fill Mode

Tells your laser to etch or engrave parallel lines inside the bounds of closed shapes in your project.

Read More

Offset Fill Mode

Tells your laser to fill closed shapes with lines that follow the contour of the outer shape.

Read More

Image Mode

Works similarly to **Fill Mode**, but applies additional special settings to adjust output based on the pixels in your image.

Read More

Galvo-Specific Settings

Special settings that are available for users with Galvo style lasers only.

Read More

Sub-Layers

A tool for setting up and applying multiple cut/engraving settings to the same geometry or layers in your project.

Read More

Default Layer Settings

Built-in or custom-created settings stored on a per-layer basis, that you can reset to at any time.

Read More

Note

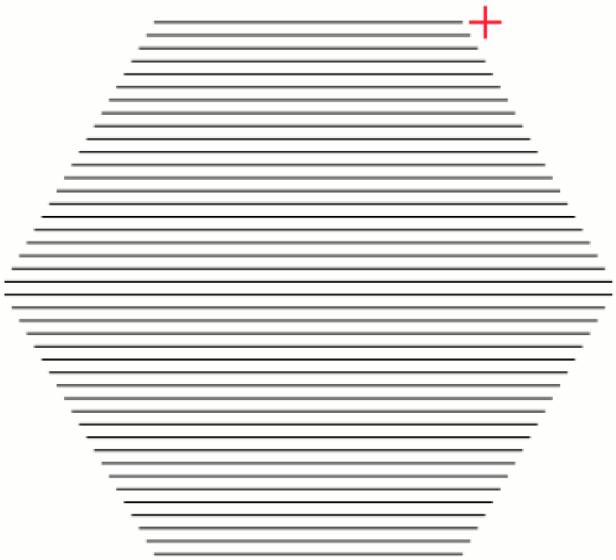
Some options in the Cut Settings Editor are hidden if you have Beginner Mode enabled.

Fill Mode

> **Closed Shapes Only**
>
> This feature only works with shapes that are closed. A shape is closed when it is a complete, continuous loop whose start and end points are the same. When a shape's start and end points are different, it is open.
>
> See Open vs. Closed Shapes for more information.

Fill Mode tells your laser to etch parallel lines within the boundaries of vector graphics.

The distance between each line is determined by the Line Interval setting, and the Speed and Power settings control the darkness or depth of the etch. Use low Intervals, low Speeds, and high Powers for deep engravings or dark marks, or tell your laser to move quickly at low Power to lightly etch or mark the surface.

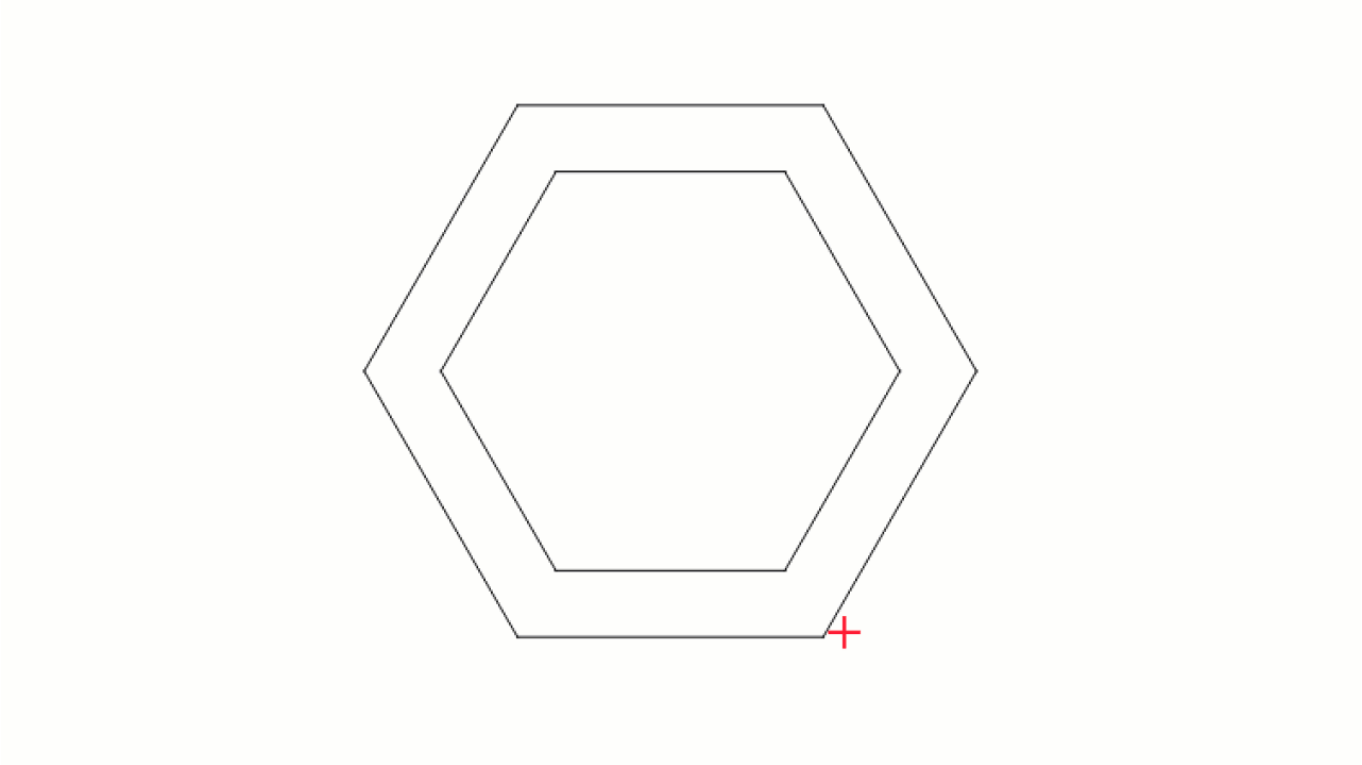


Some settings, such as Angle Increment and Auto-Rotate are only available for Galvo lasers. See Galvo-Specific Cut Settings for more information about these settings.

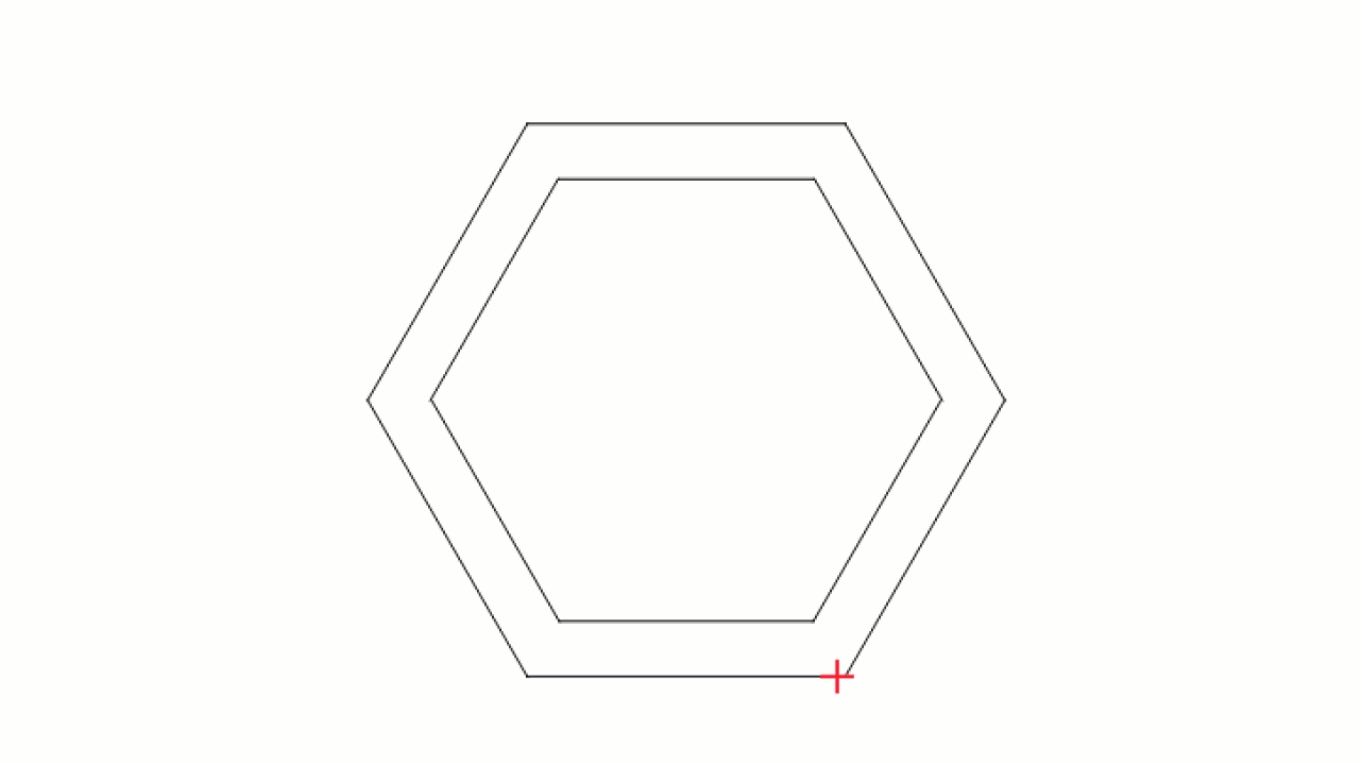
How Graphics Set to Fill Mode Interact

Graphics set to **Fill Mode** interact with one another differently depending on whether they are set to the same or different layers.

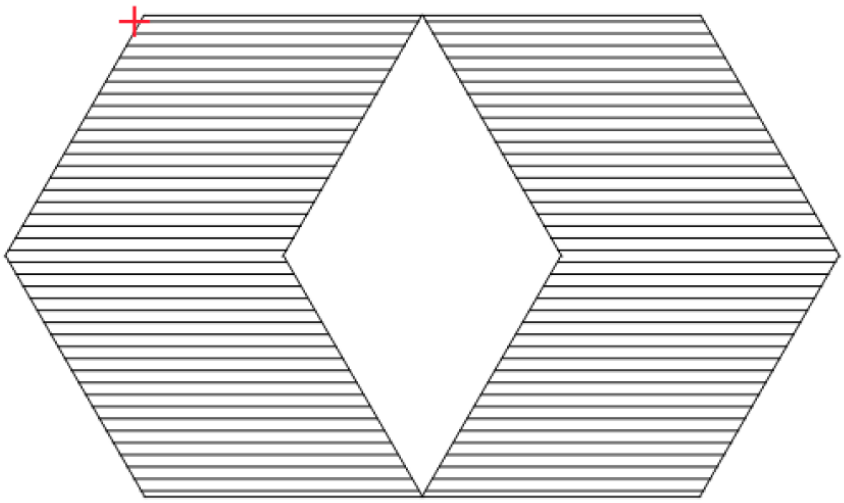
If two shapes are set to the same layer, and one is inside of the other, the area between the two outlines will be filled.



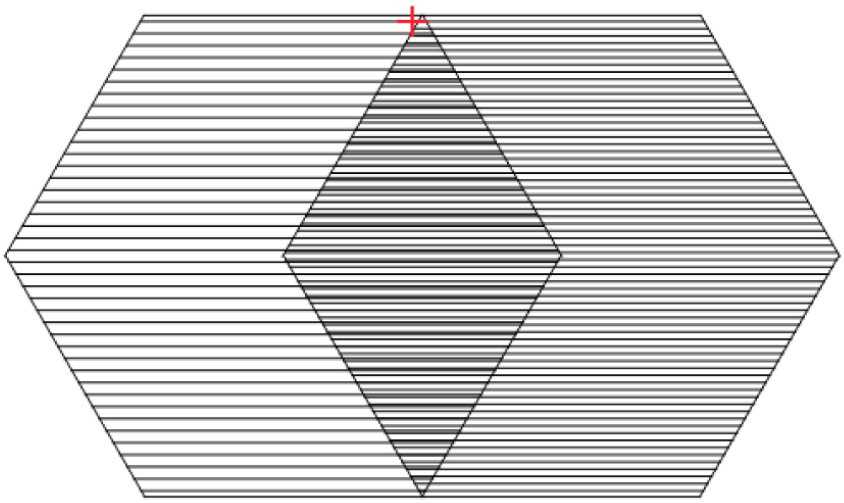
If those same two shapes are set to different layers, the entirety of the outer shape and the entirety of the inner shape will both be engraved, meaning the area within the inner shape will be engraved twice.



The same logic holds true when shapes only partially overlap — if the shapes are assigned to the same Fill Mode layer, the area where they overlap will not be engraved. If they are set to different Fill Mode layers, the area where they overlap will be engraved twice.



Partially overlapping shapes set to the same layer



Partially overlapping shapes set to different layers

Fill Mode Settings

There are several specialized options available in the Cut Settings Editor for layers set to **Fill Mode**, split into two tabs, **Common** and **Advanced**.

> **Note**
>
> See Main / Shared Settings for information on applying essential settings like Speed and Power.
>
> See Additional Options for information on specialized settings not covered on this page.

Click any option in the images below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.

Many of the settings presented in the images below are available for users with specific kinds of lasers, but most are shared between all types of lasers.



Left: Common Fill Mode settings tab

Right: Advanced Fill Mode settings tab

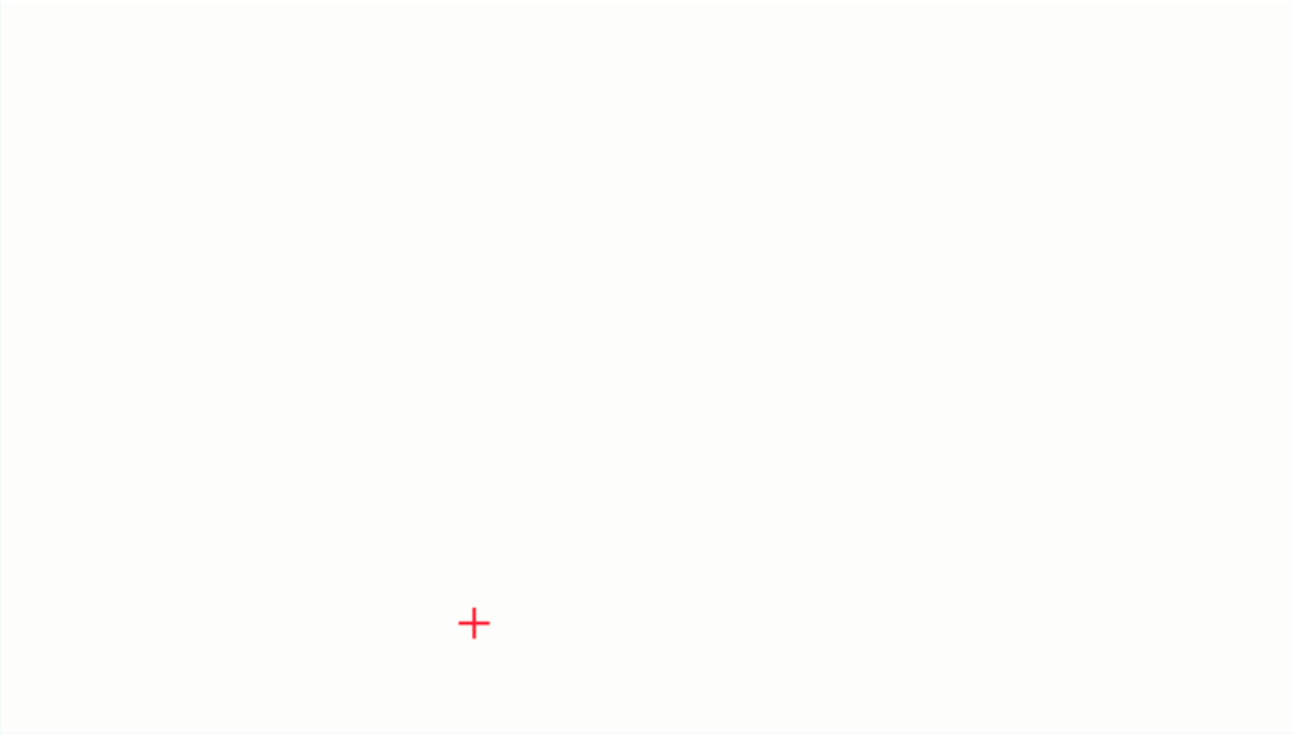
Common Settings

Bi-directional Fill

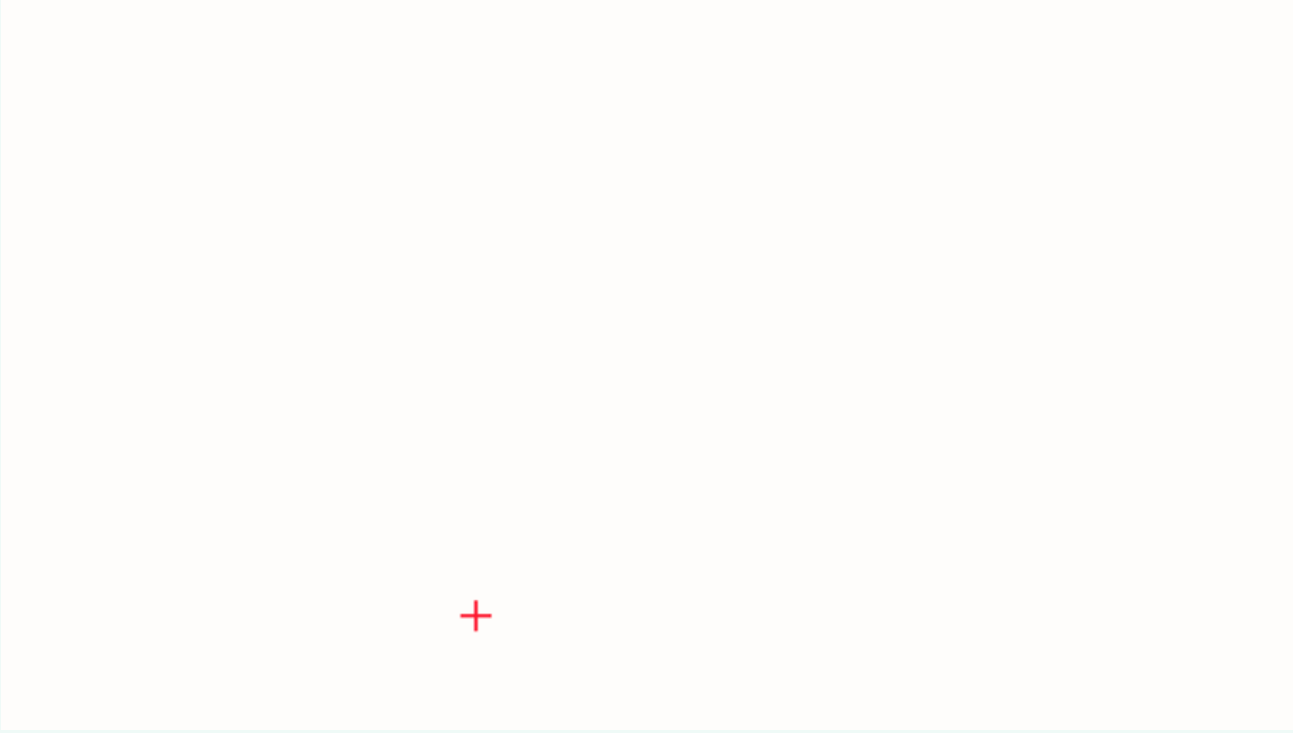
When enabled, the laser will engrave in a side-to-side sweeping motion with the laser on and engraving in both directions.

When disabled, the laser will engrave while traveling in one direction, then return to the start of the next line without engraving the return pass. Depending on the type of laser, enabling this can save significant time on long engraving jobs.

Bi-directional vs. Uni-directional Fill Comparison



Bi-directional fill

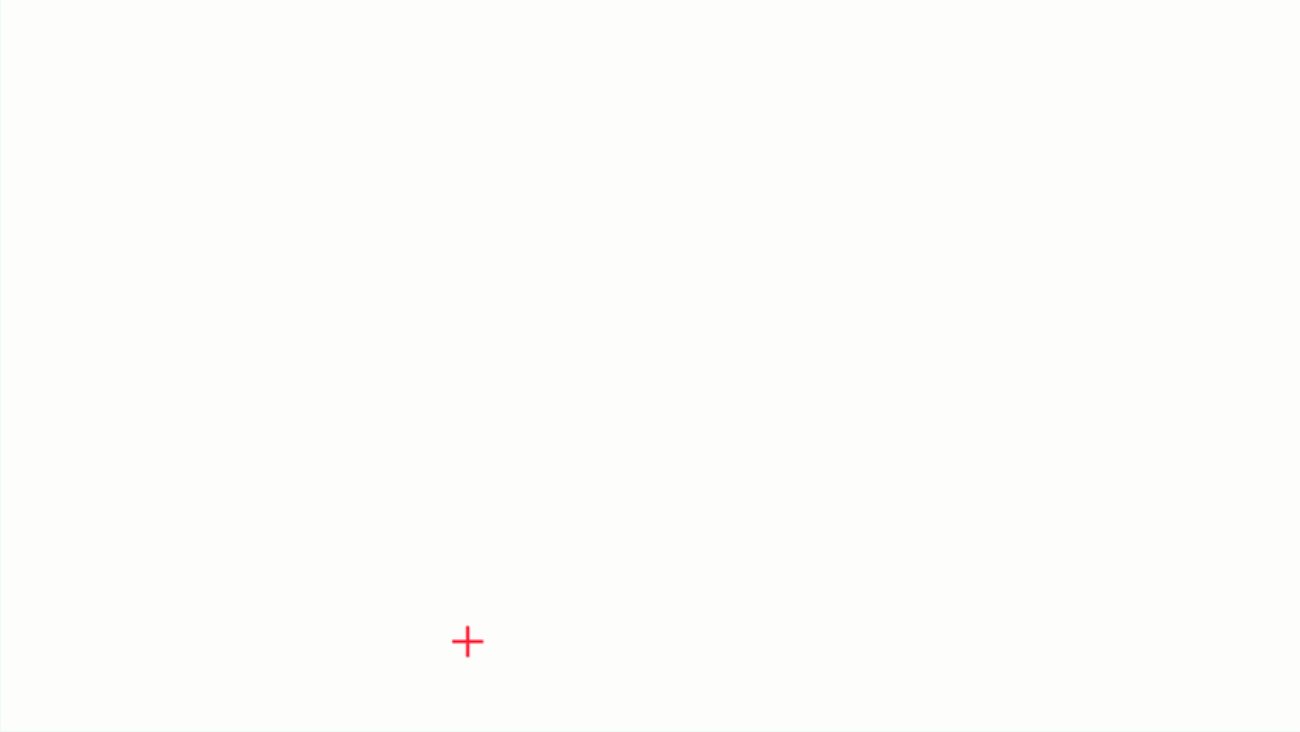


Uni-directional fill

Cross-Hatch

When enabled, the laser will run a second engraving pass 90 degrees rotated from the first, to ensure a consistent and thorough fill.

Cross-Hatch Demonstration



Overscanning

Note

This switch is not available for DSP controllers or Galvo lasers. DSP controllers handle Overscanning automatically, while Galvo lasers compensate for extra burning at edges with Delay settings.

When enabled, adds extra moves to the beginning and end of each line to give the laser time to speed up before firing, and slow down afterward. Overscanning distance is calculated as a percentage of Speed.

When **Overscanning** is not used or not set correctly, engraved jobs appear more burned at the edges and lighter in the middle, due to the laser slowing down to change scanning direction, while firing.

LightBurn can compensate for this by running the laser head outside the boundaries of the engraving, but with the laser off. This maintains an even speed on the engraving itself to provide a consistent engrave. You need a little more space on either side of the engraving to accommodate this Overscanning.

Excessive Overscanning: GCode

If Overscanning is enabled on a GCode laser and there isn't enough room at the edge of the job to accommodate the extra travel distance, you may see LightBurn's **Cut may be out of bounds due to overscan settings** error, or your laser's controller may return an ALARM:2 error.

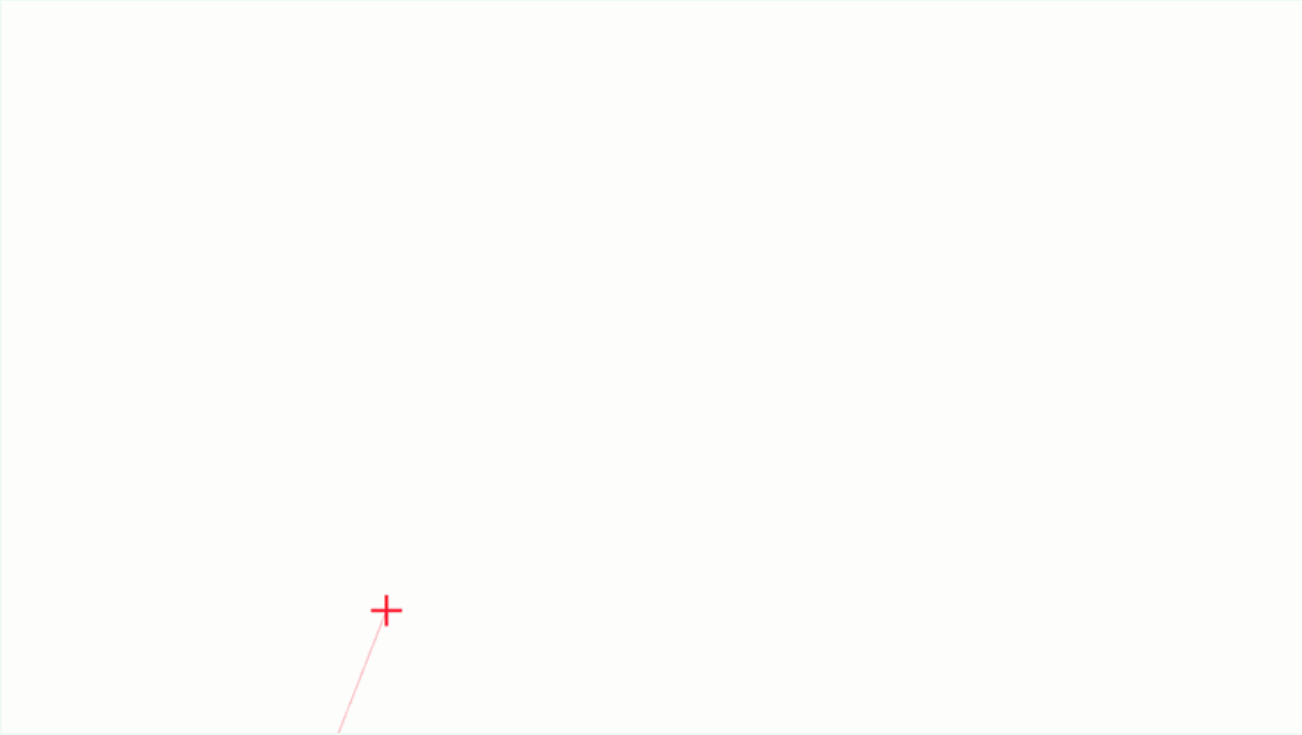
To correct the error, reduce Speed or Overscanning distance, or move your graphics in toward the center of your Workspace, if you are using Absolute Coordinates positioning. If you are using Current Position or User Origin to set a custom Origin, set it closer to the center of your laser's work area.

Excessive Overscanning: DSP

DSP controllers handle Overscanning in hardware, but the Speed value you enter affects the amount of distance required. If a job on a DSP laser requires Overscanning and there isn't enough room at the edge of the job to accommodate it, your DSP controller may return the **Not Enough Extend Space** error.

To correct the error, reduce Speed or move your graphics in toward the center of your Workspace, if you are using Absolute Coordinates positioning. If you are using Current Position or User Origin to set a custom Origin, set it closer to the center of your laser's work area.

Overscanning Demonstration



The extra Overscanning movements are indicated in red. Toggle on Show traversal moves in the Preview window to see these movements.

Line Interval

Controls the spacing between scanned lines, and indirectly controls **Lines per Inch**.

> **Optimizing Line Interval**
>
> Lower Line Interval values lead to a greater density of lines. The ideal Line Interval will produce lines that just touch, without overlapping.
>
> Increasing Line Interval to the maximum amount that doesn't leave gaps between lines will lead to reduced runtimes for your projects, with no loss in quality, but you may also need to increase Power to achieve dark engravings.
>
> Use the Interval Test or Material Test to find the optimal Line Interval for your laser and material.

Lines Per Inch

A different, often more intuitive way of controlling the Line Interval, or spacing between rows.

Scan Angle

Scan Angle defaults to 0, meaning the laser will scan back and forth horizontally across the image, progressing from the bottom of the image to the top. If you set this to 180, the laser will scan the image from top to bottom. Setting this to 90 will scan the laser vertically over the image, progressing from left to right.

> **DSP Scan Angle Warning**
>
> If you have a DSP controller, we advise only using scan angles that are a multiple of 90 degrees. Horizontal and Vertical scanning is supported natively by the hardware, and it will automatically handle Overscanning beyond the sides of the image to get the head to full speed before engraving starts. Scanning at non-90 degree angles is "emulated" using normal cutting moves. It works, but it's mostly useful as a style option, and not recommended for general use.

Number of Passes

How many times the laser will repeat the shapes on this layer. Sometimes, when cutting thick material, trying to engrave very deep, or using a lower power laser, more than one pass may be necessary.

Z Offset

If you have Z movement enabled, and your controller supports it, you can use the **Z offset** setting to move the laser head closer to the material (inward) or farther away from it (outward). Positive values move the Z Axis inward, and negative values move it outward.

Focusing deeper into the material can sometimes help to cut thicker material, and lifting the laser away from the material can produce a thicker line.

> **Warning for Z Moves**

Z moves have the potential to physically damage your laser by crashing the head into the material.

If your system uses auto-focus, you may not be able to push the focus point lower, because most systems treat the auto-focus height as the lowest possible height (Z limit) to avoid crashing the laser head.

Z step per Pass

When doing more than a single pass over a shape, the Z step per pass setting allows you to tell LightBurn to raise or lower the laser with each pass by the same amount.

By allowing you to shift the focus point deeper with each pass, this is often useful for cutting thick materials or achieving deep engravings.

Fill Grouping

The Fill Mode order-by options control which shapes are engraved at the same time by the laser. If you run your laser fast, or your laser accelerates slowly, it is often most efficient to scan things all at once, so the laser spends most of its time moving at the speed you've chosen, and less time changing direction.

If you are engraving slowly, your laser accelerates fast, or the design contains a lot of blank space, it can be more efficient to fill clusters of close shapes, or to fill the shapes one by one.

Fill All Shapes at Once: the default, this setting means that everything on this layer will be filled at the same time, sweeping back and forth across the whole job. If you are running the laser fast (300 mm/sec or more) this is usually the most efficient option, with some exceptions.

Fill Groups Together: this setting will fill all shapes in a Group at the same time.

Fill Shapes Individually: this setting fills all shapes one by one.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=ZRGSIm5y8aU

Tip

If you aren't sure which choice is ideal, try different options and use Preview to estimate the time.

Advanced settings

Under the **Advanced** tab you'll find several less frequently-used options. Many of the options presented here are available only for particular types of lasers.

Note

The Advanced tab is not available if you're using a Galvo laser. See Galvo-Specific Cut Settings for information on advanced settings for Galvo lasers.

Ramp Length

Use this setting to slope the sides of an engraving by varying the laser power. The ramp length controls how far from the design the slope will extend.

This setting is useful if you create rubber stamps, particularly if the designs have thin regions that need extra support.

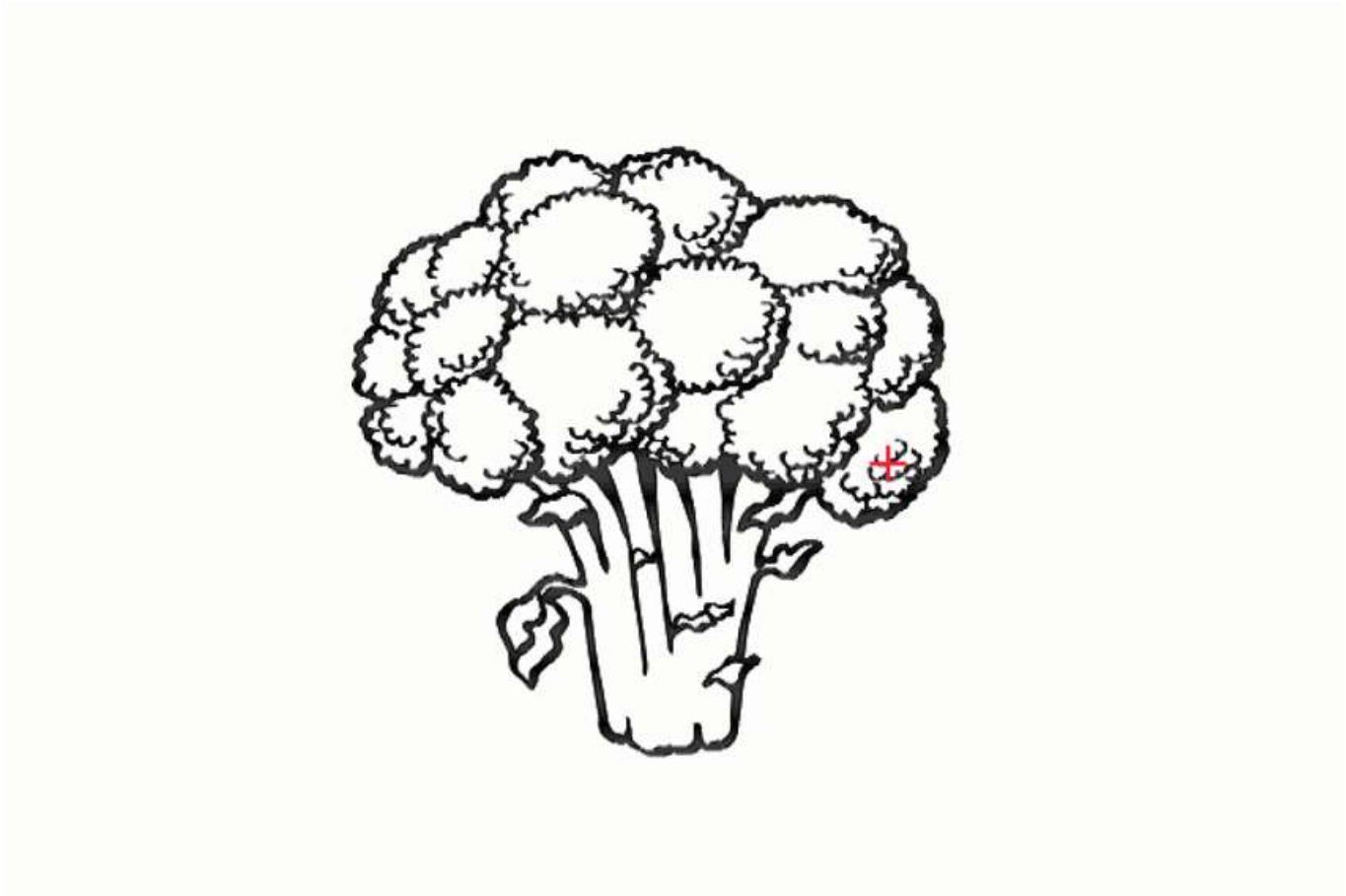
Enable **Ramp Outer Edge** to slope the outer sides of an engraving, and not just the inner sides (it's usually best to leave this setting disabled for stamp-making).

To entirely disable ramping on this layer, make sure that **Ramp Length** is set to 0.

> **Note**
>
> On a DSP controller, the Ramp varies the laser power from Min Power to Max Power, so you must set these differently to get a ramped side. Typically, you should set Min Power to be just above the firing threshold of your laser.

Flood Fill

Flood Fill calculates an engraving path that attempts to reduce or eliminate travel moves across blank space. It's useful for engraving designs with lots of empty space within graphics, where traveling across the empty portions would otherwise consume a great deal of time.



Flood Fill sometimes separates individual shapes into segments, engraving a portion of one, then a portion of another, before returning to the first shape (it may also jump within individual shapes to engrave disconnected portions). While Flood Fill will always scan back and forth at the angle you apply, it may change directions, going first from top to bottom, then bottom to top (or right to left, then left to right, and so on for any given angle).

Flood Fill is an especially useful option for machines that accelerate quickly, and for Galvo lasers in particular.

> **Note**
>
> For Galvo lasers, Flood Fill is presented in the **Common Settings** tab.

> **Warning**
>
> Flood Fill is very sensitive to machine tuning and backlash, and can cause gaps in your engraving if the design is too complex and/or your machine is not set up optimally. Use it sparingly, and use Preview to see how the option will engrave your design, so you aren't surprised by the path it takes.
>
> When using flood fill, the preview may be slow to load, especially for complex designs.

Override PWM Frequency

If you have an RF Excited tube, this setting allows you to control the PWM frequency of the pulses sent to the laser, and can change the edge finish on certain materials.

> **Note**
>
> This setting is only available for lasers with Ruida controllers.

Enable PPI

The **PPI** setting (Pulses Per Inch) lets you tell the laser to send an exact number of pulses per inch of travel, instead of varying the beam power directly. This is useful for delicate materials like paper, and is similar to using Perforation Mode, but is handled by the hardware itself.

> **Note**
>
> This setting is only available for lasers with Trocen controllers.

U Offset

Enter a distance here to set a per-layer U Axis offset. To use this setting, your laser must support U Axis control, and you must enable the U Axis in Device Settings.

> **Note**
>
> This setting is only available for lasers with Ruida controllers.

Additional Options

The settings presented in the Cut Settings Editor depend on the selected Layer Mode and the type of laser you're working with. Select an option below to learn more about different groups of settings.

Main / Shared Settings

Settings shared by almost all types of lasers and Layer Modes, including critical settings like **Speed** and **Power**.

Read More

Line Mode

Tells your laser to trace a path along the outlines of geometry in your project.

Read More

Fill Mode

Tells your laser to etch or engrave parallel lines inside the bounds of closed shapes in your project.

Read More

Offset Fill Mode

Tells your laser to fill closed shapes with lines that follow the contour of the outer shape.

Read More

Image Mode

Works similarly to **Fill Mode**, but applies additional special settings to adjust output based on the pixels in your image.

Read More

Galvo-Specific Settings

Special settings that are available for users with Galvo style lasers only.

Read More

Sub-Layers

A tool for setting up and applying multiple cut/engraving settings to the same geometry or layers in your project.

Read More

Default Layer Settings

Built-in or custom-created settings stored on a per-layer basis, that you can reset to at any time.

Read More

Note

Some options in the Cut Settings Editor are hidden if you have Beginner Mode enabled.

Image Mode

Several additional settings that affect the quality of the engraving are applied on a per-image basis, using the Adjust Image tool or Shape Properties Window.



Some settings, such as 3D Sliced Image Mode, are only available for Galvo lasers. See Galvo-Specific Cut Settings for more information about these settings.

Image Mode Settings

There are several specialized options available in the Cut Settings Editor for layers set to **Image Mode**. All settings are located in a single **Image Settings** tab.

> **Note**
>
> See Main / Shared Settings for information on applying essential settings like Speed and Power.
>
> See Additional Options for information on specialized settings not covered on this page.

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.

Some of the settings presented in the images below are available for users with specific kinds of lasers, but most are shared between all types of lasers.

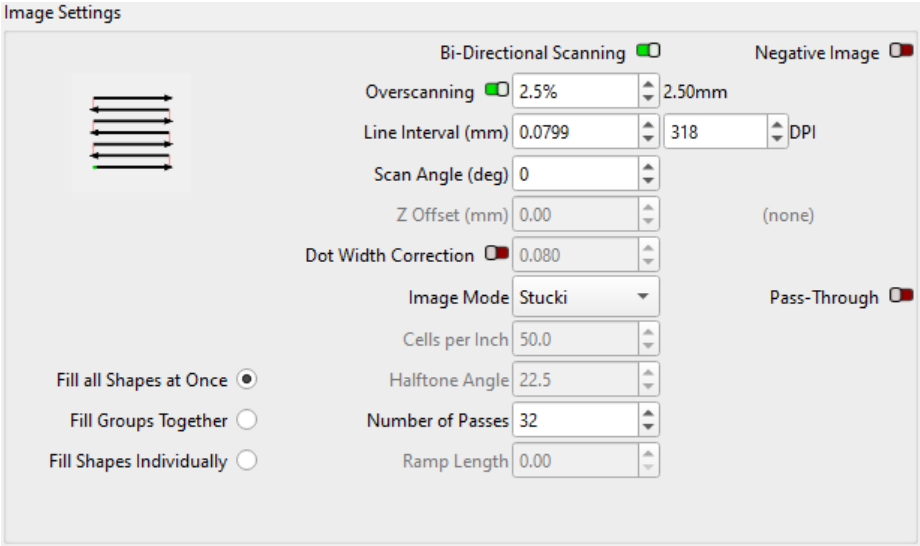


Image Mode settings for a GCode-based laser

Bi-directional Scanning

When enabled, the laser will engrave in a side-to-side sweeping motion with the laser on and engraving in both directions.

When disabled, the laser will engrave while traveling in one direction, then return to the start of the next line without engraving the return pass. Depending on the type of laser, enabling this can save significant time on long engraving jobs.

Bi-directional vs. Uni-directional Scanning Comparison

Bi-directional scanning



Uni-directional scanning

Negative Image

This will invert your image during engraving. Light becomes dark, and dark becomes light. This is useful for engraving slate or glass, where burned areas become lighter.

> **Tip**
>
> Use this in conjunction with the Invert switch in the Preview window to see how your engraving will look on dark surfaces.

Overscanning

> **Note**
>
> This switch is not available for DSP controllers or Galvo lasers. DSP controllers handle Overscanning automatically, while Galvo lasers compensate for extra burning at edges with Delay settings.

When enabled, adds extra moves to the beginning and end of each line to give the laser time to speed up before firing, and slow down afterward. Overscanning distance is calculated as a percentage of Speed.

When **Overscanning** is not used or not set correctly, engraved jobs appear more burned at the edges and lighter in the middle, due to the laser slowing down to change scanning direction, while firing.

LightBurn can compensate for this by running the laser head outside the boundaries of the engraving, but with the laser off. This maintains an even speed on the engraving itself to provide a consistent engrave. You need a little more space on either side of the engraving to accommodate this Overscanning.

Excessive Overscanning: GCode

If Overscanning is enabled on a GCode laser and there isn't enough room at the edge of the job to accommodate the extra travel distance, you may see LightBurn's **Cut may be out of bounds due to overscan settings** error, or your laser's controller may return an ALARM:2 error.

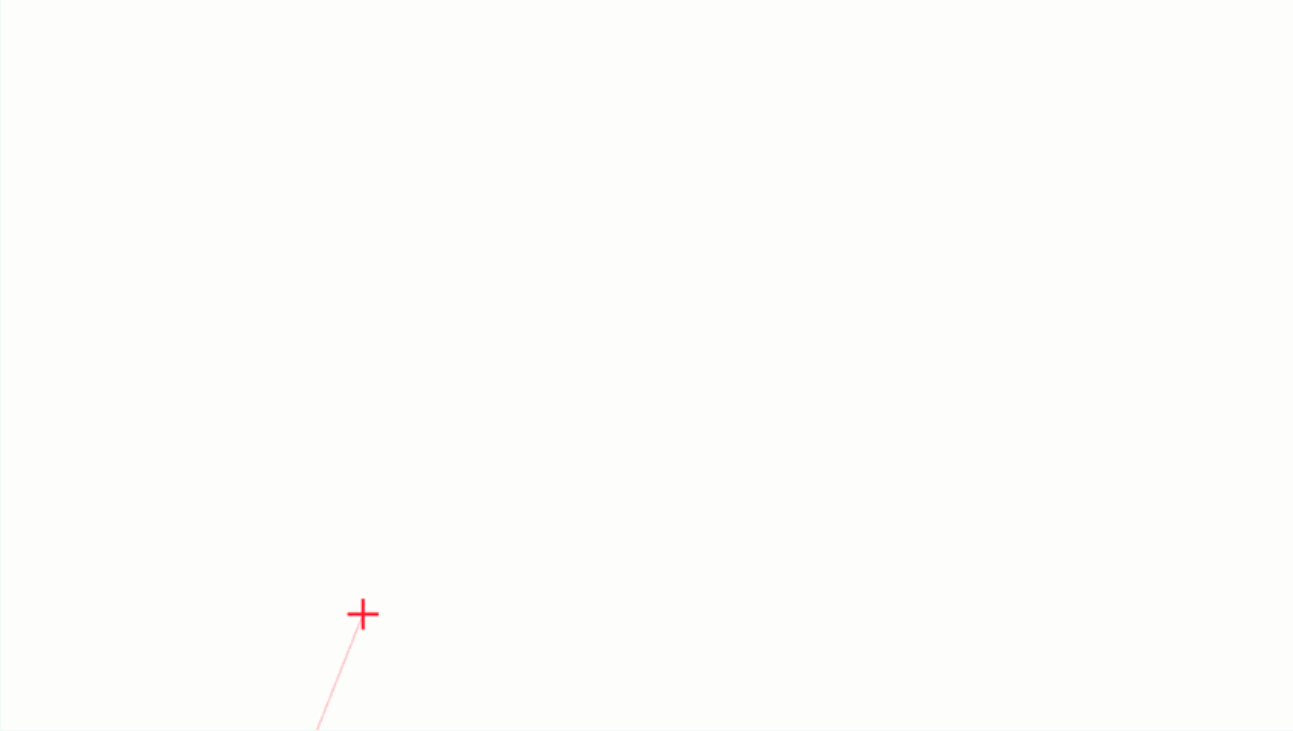
To correct the error, reduce Speed or Overscanning distance, or move your graphics in toward the center of your Workspace, if you are using Absolute Coordinates positioning. If you are using Current Position or User Origin to set a custom Origin, set it closer to the center of your laser's work area.

Excessive Overscanning: DSP

DSP controllers handle Overscanning in hardware, but the Speed value you enter affects the amount of distance required. If a job on a DSP laser requires Overscanning and there isn't enough room at the edge of the job to accommodate it, your DSP controller may return the **Not Enough Extend Space** error.

To correct the error, reduce Speed or move your graphics in toward the center of your Workspace, if you are using Absolute Coordinates positioning. If you are using Current Position or User Origin to set a custom Origin, set it closer to the center of your laser's work area.

Overscanning Demonstration



The extra Overscanning movements are indicated in red. Toggle on Show traversal moves in the Preview window to see these movements.

Line Interval

Controls the spacing between scanned rows, and indirectly controls **DPI**.

Optimizing Line Interval

Lower Line Interval values lead to a greater density of lines. The ideal Line Interval will produce lines that just touch, without overlapping.

Increasing Line Interval to the maximum amount that doesn't leave gaps between lines will lead to reduced runtimes for your projects, with no loss in quality, but you may also need to increase Power to achieve dark engravings.

Use the Interval Test or Material Test to find the optimal Line Interval for your laser and material.

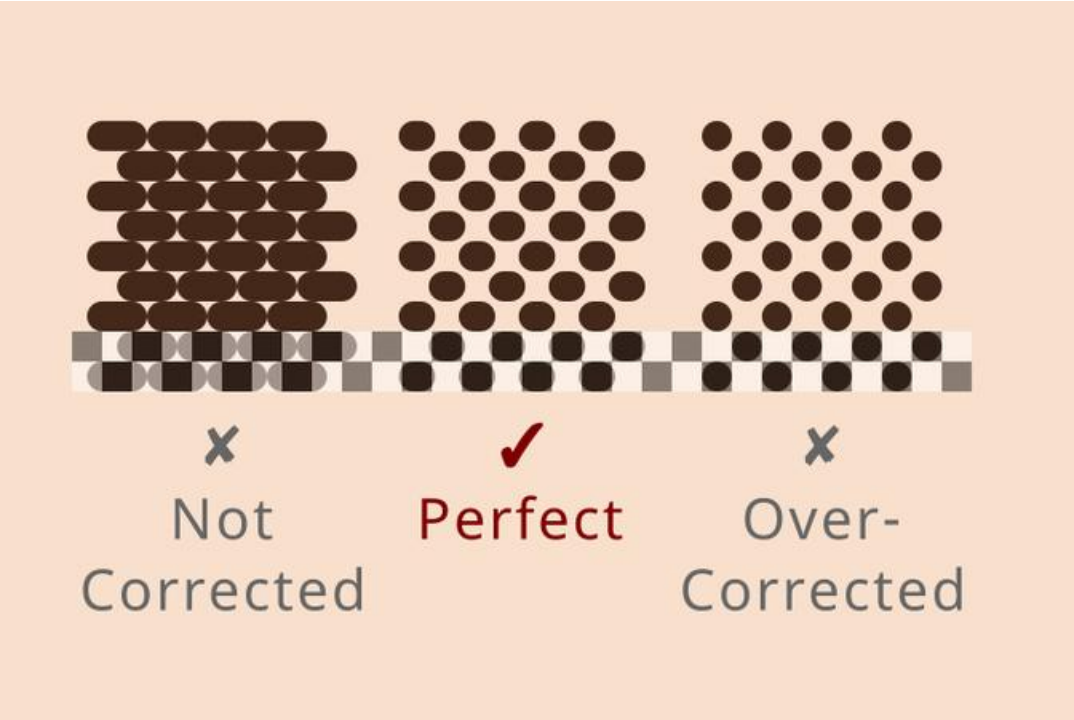
DPI (Dots Per Inch)

Controls the pixel density of the output — this is another way of representing **Line Interval**. DPI is 25.4 / Line Interval.

Dot Width Correction

Dot Width Correction compensates for the thickness of the laser's beam by shortening the length of engraved scan lines. The ideal Dot Width Correction value is about half the total thickness of the laser's beam. Due to varying affects of power output on different materials, this ideal value can vary depending on the material you're working with.

The number should always be smaller than whatever you're using as your interval setting. The valid range is from 0 to your Line Interval value.



Scan Angle

Scan Angle defaults to 0, meaning the laser will scan back and forth horizontally across the image, progressing from the bottom of the image to the top. If you set this to 180, the laser will scan the image from top to bottom. Setting this to 90 will scan the laser vertically over the image, progressing from left to right.

> **DSP Scan Angle Warning**
>
> If you have a DSP controller, we advise only using scan angles that are a multiple of 90 degrees. Horizontal and Vertical scanning is supported natively by the hardware, and it will automatically handle Overscanning beyond the sides of the image to get the head to full speed before engraving starts. Scanning at non-90 degree angles is "emulated" using normal cutting moves. It works, but it's mostly useful as a style option, and not recommended for general use.

Angle Increment

> **Process Change: Angle Increment**
>
> This setting is new in LightBurn 2.0. Previous LightBurn versions used crosshatching by default for 3D slicing. Now that users can set angle increment themselves, crosshatching is no longer automatic.
>
> To reproduce the crosshatching effect from previous LightBurn releases, set the **Angle Increment** to 90°.

The **Angle Increment** setting will determine how much the Scan Angle rotates between passes.

With the default setting of 0, all passes will be engraved in the same direction. Setting a non-zero number will tell LightBurn to rotate between passes. As an example, if the Scan Angle is set to 0 and the Angle Increment is set to 30°, subsequent passes will be at 0°, 30°, 60°, 90°, and so on.

Z Offset

If you have Z movement enabled, and your controller supports it, you can use the **Z offset** setting to move the laser head closer to the material (inward) or farther away from it (outward). Positive values move the Z Axis inward, and negative values move it outward.

Focusing deeper into the material can sometimes help to cut thicker material, and lifting the laser away from the material can produce a thicker line.

> **Warning for Z Moves**
>
> Z moves have the potential to physically damage your laser by crashing the head into the material.
>
> If your system uses auto-focus, you may not be able to push the focus point lower, because most systems treat the auto-focus height as the lowest possible height (Z limit) to avoid crashing the laser head.

Cells per Inch

Number of Halftone shading dots to compute per inch. Only enabled when the Halftone Image Mode is selected.

Halftone Angle

Angle of the Halftone shading pattern. Only enabled when the Halftone Image Mode is selected.

Number of Passes

How many times the laser will repeat the shapes on this layer. Sometimes, when cutting thick material, trying to engrave very deep, or using a lower power laser, more than one pass may be necessary.

Ramp Length

Use this setting to slope the sides of an engraving by varying the laser power. The ramp length controls how far from the design the slope will extend.

This setting is useful if you create rubber stamps, particularly if the designs have thin regions that need extra support.

This setting is only available if you have Threshold Image Mode selected.

> **Note**

> On a DSP controller, the Ramp varies the laser power from Min Power to Max Power, so you must set these differently to get a ramped side. Typically, you should set Min Power to be just above the firing threshold of your laser.

Pass-Through

When working with images that have been pre-processed for laser engraving, enable this setting to engrave the image as-is, rather than resampling using the Image Modes described below. **Line Interval/DPI** will be directly tied to the size of the image.

Image Mode

The Image Mode setting controls how LightBurn processes your image. Below you can see a description of each Image Mode in LightBurn, along with an example of how that mode changes the look of the example image. Click on the image previews to see a larger version.



Threshold



A simple on / off switch if the image is dark / bright at a given location. This should only be used for images that are two-color black/white to begin with, such as an image you dithered outside of LightBurn. Don't use this mode for grayscale or color images.

Ordered



Also called ordered dithering, this is a step above threshold for grayscale images, and uses densely packed on/off dots to approximate shading, using an ordered/regular grid pattern. This is ok for general use, but works best for images with large areas of solid fill, where diffusion dithering can cause unwanted artifacts (see below).

Atkinson



A good "in between" mode for solid color or smooth shaded images. It resembles Jarvis but preserves detail better, though struggles to show detail in very light and dark areas.

Dither



Also called error diffusion dithering, this is the best choice for smoothly shaded images, like photos. This also approximates shading with simple dots, but does so without evident patterning, and tends to produce more subtle shading.

Stucki



A good choice for smooth-shaded or photo images, with slightly faster processing than Jarvis. Stucki works somewhat similarly to Dither mode to produce smooth shading, and likewise struggles in midtones.

Jarvis



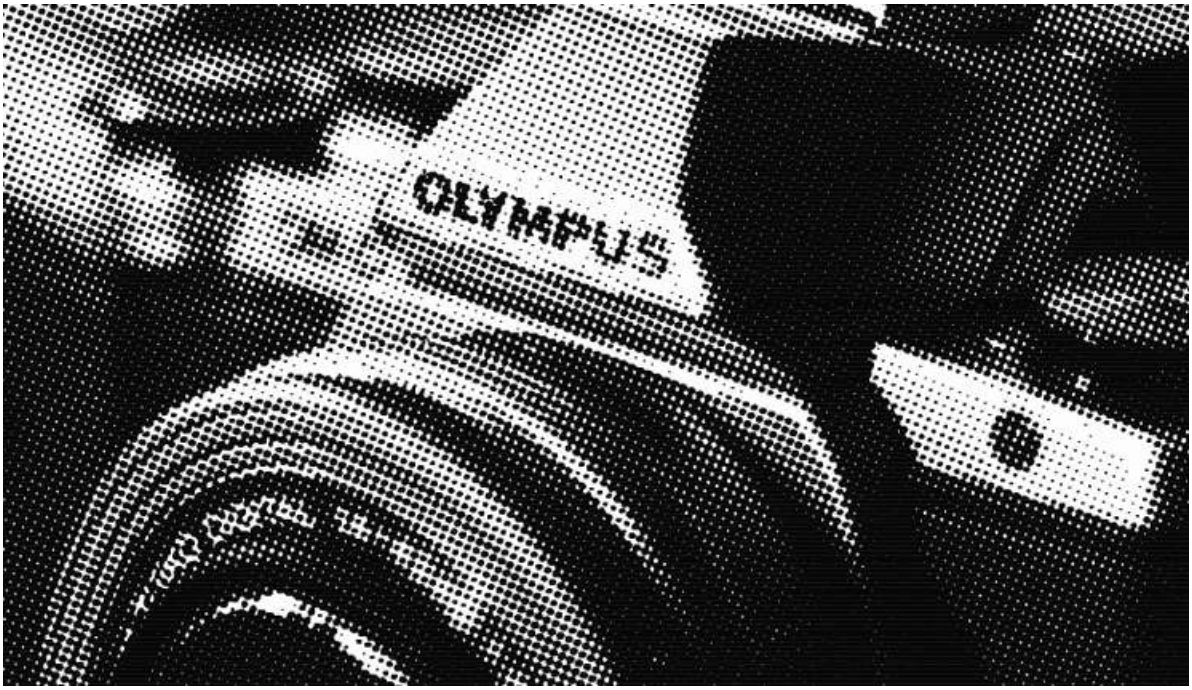
Generally the best choice for smooth-shaded or photo images, with slightly slower but better processing than Stucki. Jarvis works somewhat similarly to Dither mode to produce smooth shading, and doesn't struggle with midtones.

Newsprint



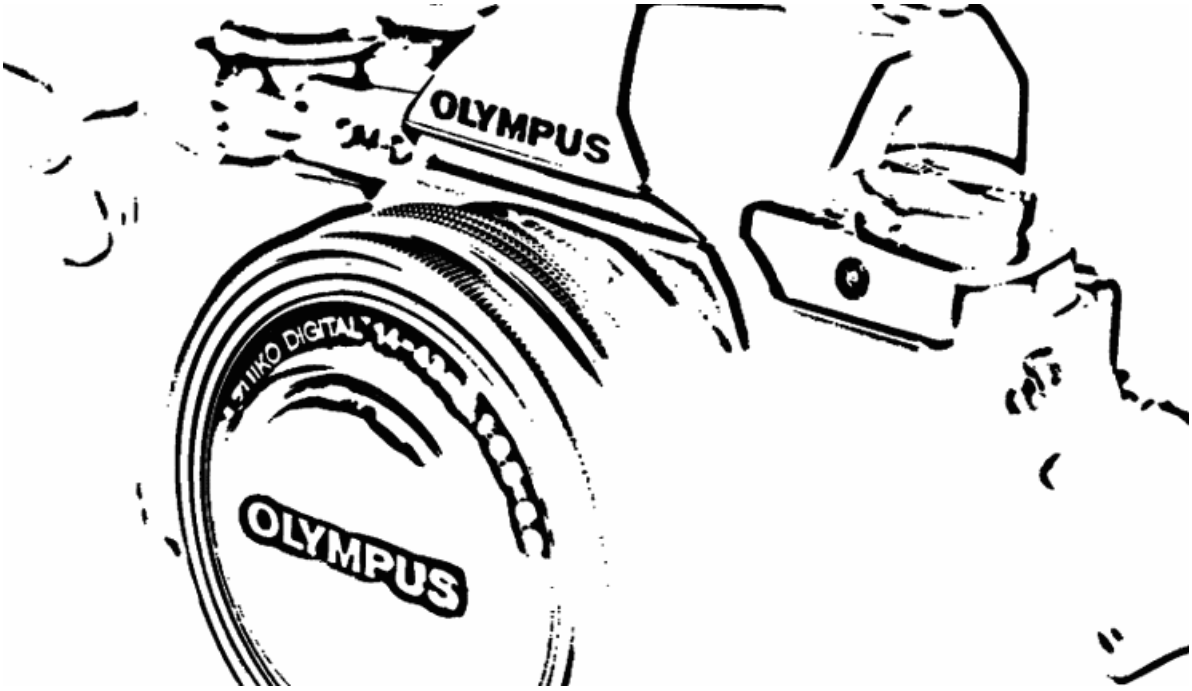
A decorative pattern that imitates newsprint halftoning by generating a grid of spots that are larger in darker areas. It has good tonal range, but is visibly patterned. Smoothest shading is achieved with higher DPI settings, or Smoothieware controllers.

Halftone



Good quality shading for high DPI images. It is similar to Newsprint but can have variable cell size and pattern angle.

Sketch



Useful for line drawings or handwriting, as it tries to detect hard edges. Sketch engraves only at the areas where light and dark values meet at hard edges (highpass filtering).

Grayscale



Varies power output as a percentage between Min and Max Power, using Min Power for the lightest shades and Max Power for the darkest. With a diode laser this can provide great shading but is harder to get right than plain dithering. With a CO2 laser, grayscale mode can achieve variable depth (3D) engraving rather than shading. Images typically need to be specifically created for this use.

> **Grayscale Image Mode is not compatible with UV lasers**
>
> UV lasers do not have direct control over power output - they vary output using frequency instead. When using Grayscale Image Mode with a UV laser, the power output will be locked to 20%, and the laser will output a solid gray rectangle. If you're using a UV laser, please select a different image mode.

> **Note**
>
> When Previewing a Grayscale engraving, be sure to enable Shade according to power, otherwise you will see a completely black Preview since, unlike other modes, Grayscale scans every portion of the image, just at varying levels of laser power.

Fill Grouping

The Fill Mode order-by options control which shapes are engraved at the same time by the laser. If you run your laser fast, or your laser accelerates slowly, it is often most efficient to scan things all at once, so the laser spends most of its time moving at the speed you've chosen, and less time changing direction.

If you are engraving slowly, your laser accelerates fast, or the design contains a lot of blank space, it can be more efficient to fill clusters of close shapes, or to fill the shapes one by one.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=ZRGSIm5y8aU

Tip

If you aren't sure which choice is ideal, try different options and use Preview to estimate the time.

Override PWM Frequency

If you have an RF Excited tube, this setting allows you to control the PWM frequency of the pulses sent to the laser, and can change the edge finish on certain materials.

Note

This setting is only available for lasers with Ruida controllers.

U Offset

Enter a distance here to set a per-layer U Axis offset. To use this setting, your laser must support U Axis control, and you must enable the U Axis in Device Settings.

Note

This setting is only available for lasers with Ruida controllers.

Additional Options

The settings presented in the Cut Settings Editor depend on the selected Layer Mode and the type of laser you're working with. Select an option below to learn more about different groups of settings.

Main / Shared Settings

Settings shared by almost all types of lasers and Layer Modes, including critical settings like **Speed** and **Power**.

Read More

Line Mode

Tells your laser to trace a path along the outlines of geometry in your project.

Read More

Fill Mode

Tells your laser to etch or engrave parallel lines inside the bounds of closed shapes in your project.

Read More

Offset Fill Mode

Tells your laser to fill closed shapes with lines that follow the contour of the outer shape.

Read More

Image Mode

Works similarly to **Fill Mode**, but applies additional special settings to adjust output based on the pixels in your image.

Read More

Galvo-Specific Settings

Special settings that are available for users with Galvo style lasers only.

Read More

Sub-Layers

A tool for setting up and applying multiple cut/engraving settings to the same geometry or layers in your project.

Read More

Default Layer Settings

Built-in or custom-created settings stored on a per-layer basis, that you can reset to at any time.

Read More

Note

Some options in the Cut Settings Editor are hidden if you have Beginner Mode enabled.

Offset Fill Mode

Closed Shapes Only

This feature only works with shapes that are closed. A shape is closed when it is a complete, continuous loop whose start and end points are the same. When a shape's start and end points are different, it is open.

See Open vs. Closed Shapes for more information.

Offset Fill mode works similarly to Fill Mode, but instead of telling your laser to scan parallel lines within the boundaries of a vector graphic, it tells your laser to etch concentric lines that follow the outline of the shape.

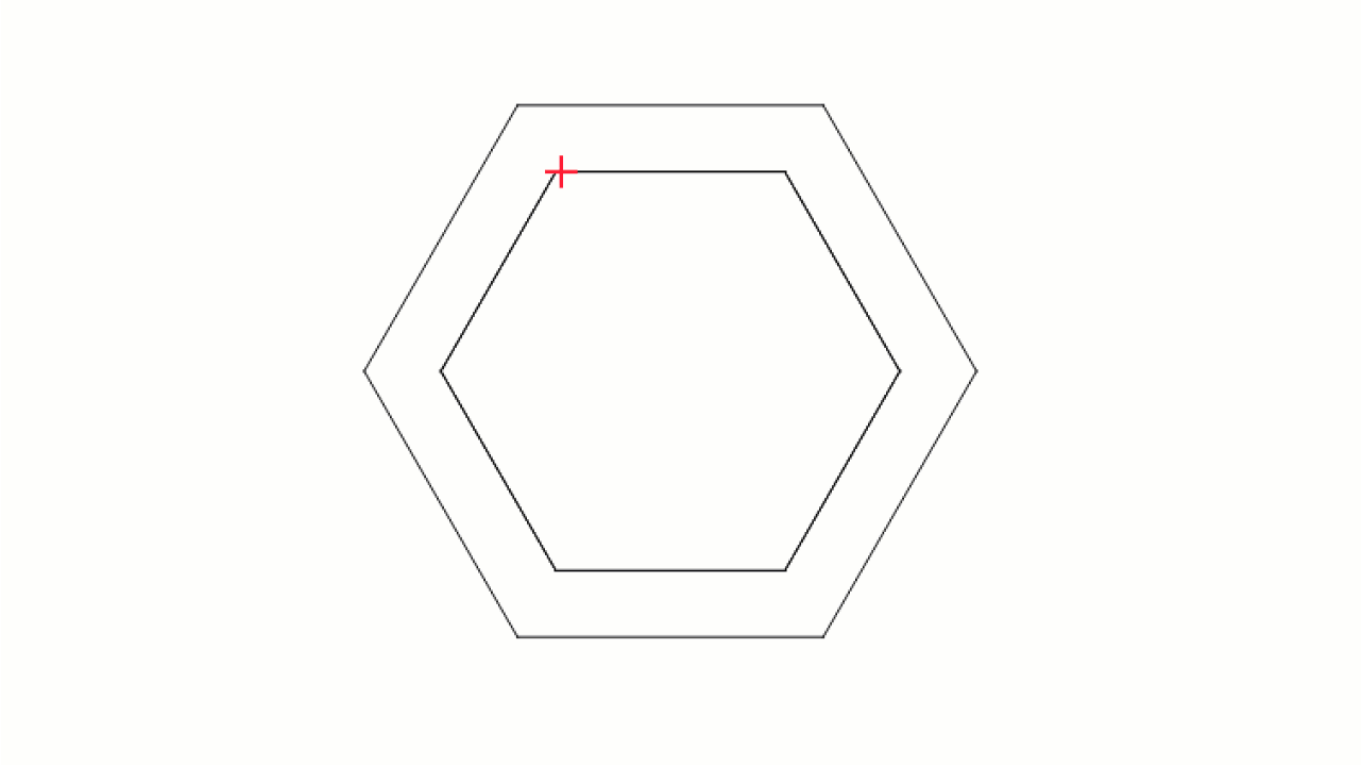
As with both Line and Fill Mode, Speed and Power control the depth and darkness of the etch, and, as with Fill Mode, the Line Interval setting controls the spacing of the lines.

Offset Fill is intended for designs with a lot of empty space within graphics, to cut down on the amount of travel time necessary to fill large, hollow shapes. If that description fits your design, Offset Fill will most likely be much faster than Fill Mode. If that doesn't apply to your graphics, it's usually best to use Fill instead of Offset Fill. Offset Fill is generally not a good choice for complex shapes.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=haLj85zVjBs

Offset Fill Computation Time

Offset Fill is very computationally expensive, and the more complex the design, or the smaller the Line Interval set, the more potential there is for the computation to cause LightBurn to hang. Sometimes graphics with stray nodes or unusual paths can confuse the calculation interminably.



Some settings, such as Angle Increment and Auto-Rotate are only available for Galvo lasers. See Galvo-Specific Cut Settings for more information about these settings.

Offset Fill Mode Settings

There are several specialized options available in the Cut Settings Editor for layers set to **Offset Fill Mode**, split into two tabs, **Common** and **Advanced**.

> **Note**
>
> See Main / Shared Settings for information on applying essential settings like Speed and Power.
>
> See Additional Options for information on specialized settings not covered on this page.

Click any option in the images below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.

Many of the settings presented in the images below are available for users with specific kinds of lasers, but most are shared between all types of lasers.



Left: Common Offset Fill Mode settings tab

Right: Advanced Offset Fill Mode settings tab

Common Settings

Bi-directional fill, Cross-Hatch, Overscanning, and Scan Angle are presented in this tab, but are grayed out and unavailable for use with layers set to **Offset Fill Mode**. To use these settings, set the layer to Fill Mode.

Line Interval

Controls the spacing between lines, and indirectly controls **Lines per Inch**.

Lower Line Interval values lead to a greater density of lines. The ideal Line Interval will produce lines that just touch, without overlapping.

Increasing Line Interval to the maximum amount that doesn't leave gaps between lines will lead to reduced runtimes for your projects, with no loss in quality, but you may also need to increase Power to achieve dark engravings.

Use the Interval Test or Material Test to find the optimal Line Interval for your laser and material.

Lines Per Inch

A different, often more intuitive way of controlling the Line Interval, or spacing between rows.

Number of Passes

How many times the laser will repeat the shapes on this layer. Sometimes, when cutting thick material, trying to engrave very deep, or using a lower power laser, more than one pass may be necessary.

Z Offset

If you have Z movement enabled, and your controller supports it, you can use the **Z offset** setting to move the laser head closer to the material (inward) or farther away from it (outward). Positive values move the Z Axis inward, and negative values move it outward.

Focusing deeper into the material can sometimes help to cut thicker material, and lifting the laser away from the material can produce a thicker line.

> **Warning for Z Moves**
>
> Z moves have the potential to physically damage your laser by crashing the head into the material.
>
> If your system uses auto-focus, you may not be able to push the focus point lower, because most systems treat the auto-focus height as the lowest possible height (Z limit) to avoid crashing the laser head.

Z step per Pass

When doing more than a single pass over a shape, the Z step per pass setting allows you to tell LightBurn to raise or lower the laser with each pass by the same amount.

By allowing you to shift the focus point deeper with each pass, this is often useful for cutting thick materials or achieving deep engravings.

Fill Grouping

The Fill Mode order-by options control which shapes are engraved at the same time by the laser. If you run your laser fast, or your laser accelerates slowly, it is often most efficient to scan things all at once, so the laser spends most of its time moving at the speed you've chosen, and less time changing direction.

If you are engraving slowly, your laser accelerates fast, or the design contains a lot of blank space, it can be more efficient to fill clusters of close shapes, or to fill the shapes one by one.

Advanced Settings

Under the **Advanced** tab you'll find a few less frequently-used options. The options presented here are available only for particular types of lasers.

Ramp Length and Flood Fill are presented in this tab, but are grayed out and unavailable for use with layers set to **Offset Fill Mode**. To use these settings, set the layer to Fill Mode.

> **Note**
>
> The Advanced tab is not available if you're using a Galvo laser. See Galvo-Specific Cut Settings for information on advanced settings for Galvo lasers.

Override PWM Frequency

If you have an RF Excited tube, this setting allows you to control the PWM frequency of the pulses sent to the laser, and can change the edge finish on certain materials.

> **Note**

This setting is only available for lasers with Ruida controllers.

Enable PPI

The **PPI** setting (Pulses Per Inch) lets you tell the laser to send an exact number of pulses per inch of travel, instead of varying the beam power directly. This is useful for delicate materials like paper, and is similar to using Perforation Mode, but is handled by the hardware itself.

Note

This setting is only available for lasers with Trocen controllers.

U Offset

Enter a distance here to set a per-layer U Axis offset. To use this setting, your laser must support U Axis control, and you must enable the U Axis in Device Settings.

Note

This setting is only available for lasers with Ruida controllers.

Additional Options

The settings presented in the Cut Settings Editor depend on the selected Layer Mode and the type of laser you're working with. Select an option below to learn more about different groups of settings.

Main / Shared Settings

Settings shared by almost all types of lasers and Layer Modes, including critical settings like **Speed** and **Power**.

Read More

Line Mode

Tells your laser to trace a path along the outlines of geometry in your project.

Read More

Fill Mode

Tells your laser to etch or engrave parallel lines inside the bounds of closed shapes in your project.

Read More

Offset Fill Mode

Tells your laser to fill closed shapes with lines that follow the contour of the outer shape.

Read More

Image Mode

Works similarly to **Fill Mode**, but applies additional special settings to adjust output based on the pixels in your image.

Read More

Galvo-Specific Settings

Special settings that are available for users with Galvo style lasers only.

Read More

Sub-Layers

A tool for setting up and applying multiple cut/engraving settings to the same geometry or layers in your project.

Read More

Default Layer Settings

Built-in or custom-created settings stored on a per-layer basis, that you can reset to at any time.

Read More

Note

Some options in the Cut Settings Editor are hidden if you have Beginner Mode enabled.

Main / Shared Settings

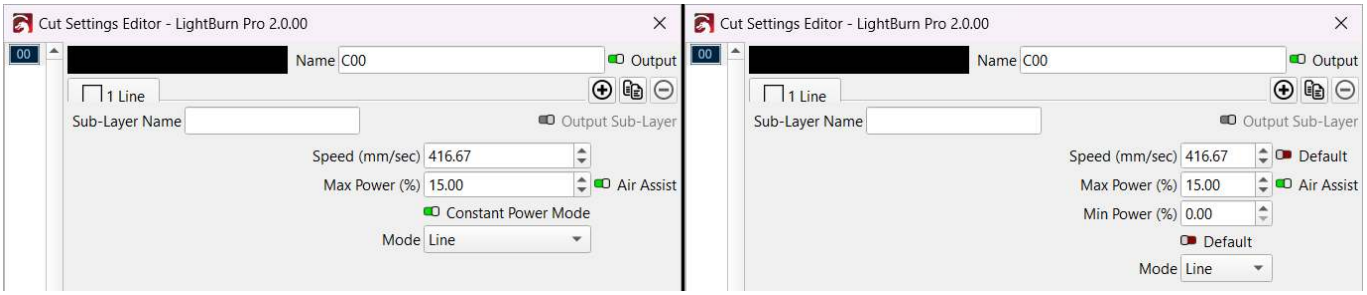
Several settings in the Cut Settings Editor are shared among nearly all laser types and Layer Modes.

> **Note**
>
> See Additional Options for information on specialized settings not covered on this page. See Galvo-Specific Cut Settings for cut settings only available with Galvo lasers.

Essential Settings

The settings near the top of Cut Settings Editor are fundamental to all laser operations. You can edit many of these settings directly in the Cuts / Layers Window as well.

Click any option in the images below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Left: Essential settings for a GCode-based laser

Right: Essential settings for a DSP laser

Name

You can rename layers to help keep track of their settings and purpose in your project.

Output Switch

Toggle the **Output** switch on or off to determine whether a given layer is included in the output when you Preview, Start, Send, or save a project in machine-specific format.

The color of layers not set to output will be faded in your Workspace.

Speed

The maximum speed the laser will reach when cutting or engraving your design.

Higher speed values can decrease the amount of time your laser spends completing a job, but also tend to produce lighter marks or shallower engravings than slower speed values. Cutting through many materials requires relatively slow speeds, though the exact values vary significantly depending on your laser and material.

> **Note**

Depending on your laser, its firmware settings, and the design you tell it to cut or engrave, the laser may not actually reach the speed you set. Consult your laser's manufacturer for guidance on the maximum speed your laser is capable of.

Always pay attention to units of distance and time when entering Speed settings

When entering **Speed** values recommended by your laser's manufacturer or other LightBurn users, make sure to use the same units of distance and time as those from the recommendation, or to convert the values to your preferred units.

A given number of **millimeters per second** is much faster than that same number in **millimeters per minute**. Mixing up units can lead to reduction in power output due to unexpectedly high speeds, or excessive power output — and even fire — due to unexpectedly slow speeds.

Change your Control Units in the **Dimensions / Units** tab of the **Device Settings** window. LightBurn automatically converts any existing values when you switch between units.

Default Speed Switch (Ruida Only)

Enable this switch to use the default **Speed** setting from the laser's controller. You can adjust this default through the controller's display.

Max Power

Tip

Use the Material Test to find optimal settings for your laser, material, and desired outcome.

The maximum power level the laser will reach when cutting or engraving your design.

High power levels produce darker marks and deeper engravings, and relatively high powers are required to completely cut through many materials.

GRBL Power Ramping

GRBL-based lasers ramp up to the maximum power level as they increase in speed, only reaching Max Power when they also reach the Speed you've set. If you set a Speed higher than your laser can actually reach, it will never reach Max Power either.

See Troubleshooting: Low or No Power Output for more information.

Min Power

Note

This setting is only shown when you're using a DSP laser or are engraving an image using Grayscale Mode, with any type of laser.

Min Power determines the power level the laser will use when it travels at slow speeds, such as when it slows down to reach the ends of lines, changes directions, or turns corners.

Some DSP controllers refer to this as the **Corner Power** for this reason.

The laser's controller will automatically ramp power between Min and Max Power as it accelerates to the maximum Speed you've set.

> **Min Power Troubleshooting**
>
> Reduce Min Power if your laser is scorching or charring your material when it turns corners, or increase it if it's having trouble cutting through material on corners.
>
> See Troubleshooting: Corners Are Too Dark, Too Light, or Missing for more information.

Default Power Switch (Ruida Only)

Enable this switch to use the default **Max** and **Min Power** settings from the laser's controller. You can adjust these defaults through the controller's display.

Constant Power Mode (GRBL Only)

By default, LightBurn uses **Variable Power** mode (M4), which ramps the power level of the laser up and down with the speed, producing a more consistent burn.

However, if you ask the laser to move faster than it's able, it will never reach the requested speed or the requested power level. It's generally best to stick with the default of Variable Power mode and adjust the speed to be within the range your machine can achieve, but if you want the same behavior as LaserGRBL, which defaults to **Constant Power** (M3), you can enable this option.

See Troubleshooting: Low or No Power Output for more information.

Air Assist

Enable this switch to turn on **Air Assist**, if your laser supports it. Air assist can help in cutting through material.

Users with GCode-based lasers can select whether their air assist is activated by an M7 or M8 command in Device Settings.

> **Note**
>
> This setting is available for DSP and GCode-based lasers only.
>
> If your laser has no air assist, or it must be activated manually, enabling and disabling this switch will have no effect.

Mode Dropdown Menu

Select Line, Fill, or Offset Fill from the dropdown menu to change how the laser will cut or engrave graphics assigned to a given layer.

Images are automatically set to Image Mode and this dropdown menu is not available for layers they are assigned to.

Additional Options

The settings presented in the Cut Settings Editor depend on the selected Layer Mode and the type of laser you're working with. Select an option below to learn more about different groups of settings.

Main / Shared Settings

Settings shared by almost all types of lasers and Layer Modes, including critical settings like **Speed** and **Power**.

Read More

Line Mode

Tells your laser to trace a path along the outlines of geometry in your project.

Read More

Fill Mode

Tells your laser to etch or engrave parallel lines inside the bounds of closed shapes in your project.

Read More

Offset Fill Mode

Tells your laser to fill closed shapes with lines that follow the contour of the outer shape.

Read More

Image Mode

Works similarly to **Fill Mode**, but applies additional special settings to adjust output based on the pixels in your image.

Read More

Galvo-Specific Settings

Special settings that are available for users with Galvo style lasers only.

Read More

Sub-Layers

A tool for setting up and applying multiple cut/engraving settings to the same geometry or layers in your project.

Read More

Default Layer Settings

Built-in or custom-created settings stored on a per-layer basis, that you can reset to at any time.

Read More

Note

Some options in the Cut Settings Editor are hidden if you have Beginner Mode enabled.

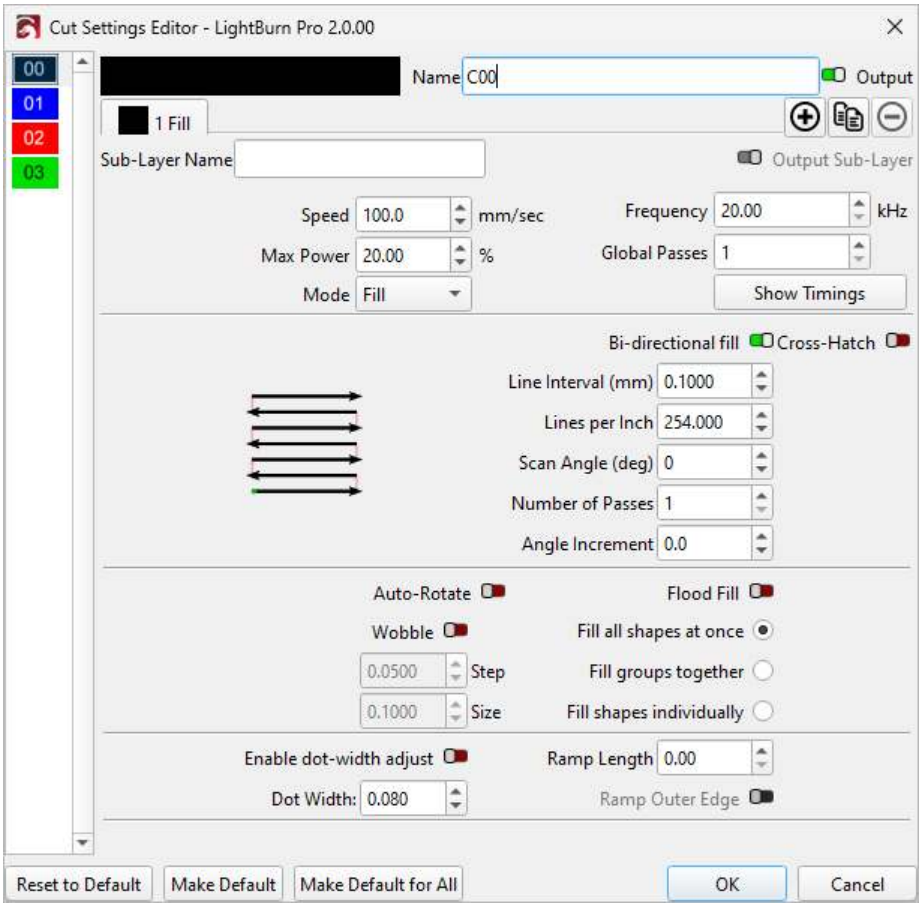
Galvo-Specific Cut Settings

Several settings are only available in the Cut Settings Editor when working with Galvo lasers with EZCAD2, EZCAD3, or BSL control boards.

Among these settings, some options are only available for specific types of Galvo lasers, or specific Layer Modes.

> **Note**
>
> See Main / Shared Settings for information on applying essential settings like Speed and Power.
>
> See Additional Options for information on specialized settings not covered on this page.

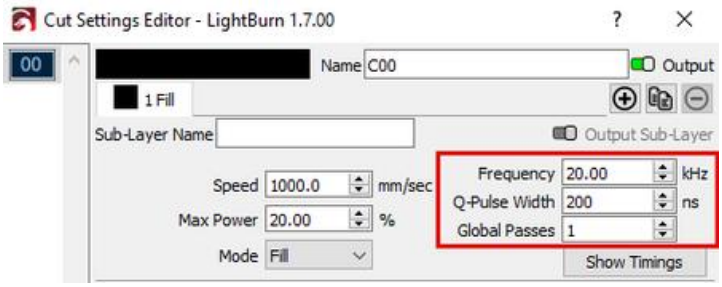
Click any option in the images below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Galvo-specific cut settings

Main Settings

For Galvo lasers, the following settings are available in the top portion of the Cut Settings Editor, along with other essential settings shared by all types of layers.



Frequency

The **Frequency**, in kHz, at which the laser will pulse during operation. Relatively low Frequencies yield fewer pulses at greater power output with each pulse, while relatively high Frequencies yield more pulses at lower power output with each pulse.

Frequencies in the mid-range of your laser's capability will often produce the most even markings, but the exact effect of given Frequency values varies greatly depending on your laser, material, and other settings, such as Power, Speed, and Line Interval.

> **Tip**
>
> Use the Material Test to try out different combinations of Frequency, Q-Pulse Width, and other settings on a given material.

Q-Pulse Width

> **Note**
>
> This setting is available for MOPA Fiber and UV Galvo lasers only.

Q-Pulse Width determines the duration of each pulse of your laser, in nanoseconds. As with Frequency, the exact effect of given Q Pulse Widths varies greatly depending on your laser, material, and other settings, such as Power, Speed, and Line Interval.

> **Note for UV Lasers**
>
> For UV lasers, the Power setting is grayed out because power output is controlled solely by adjusting Frequency and Q-Pulse Width.
>
> Max power is achieved by applying the correct combination of Frequency and Q-Pulse — your laser may have come with a chart that shows expected power outputs at different combinations. In general, you will see power increase as you increase Frequency and Q-Pulse values, a leveling off of power output in the middle-to-high values, and then a tailing off with very high values.

Global Passes

Enable this setting to repeat all Sub-Layers in the layer, in sequence, the specified number of times.

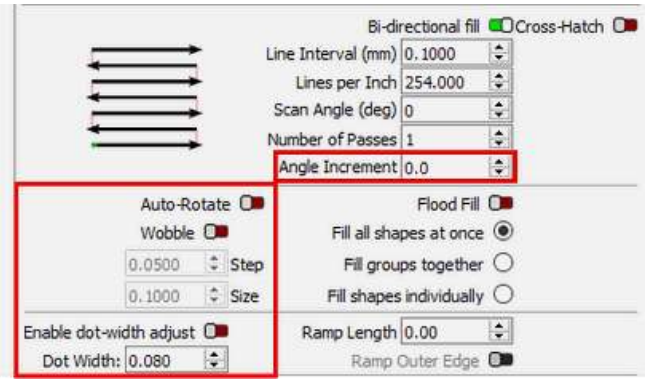
Use Laser 2

If you've enabled a second laser source in your Device Settings, enable this toggle to use the second laser source for this layer.

Specialized Settings

For Galvo lasers, the following settings are available in the bottom half of the Cut Settings Editor.

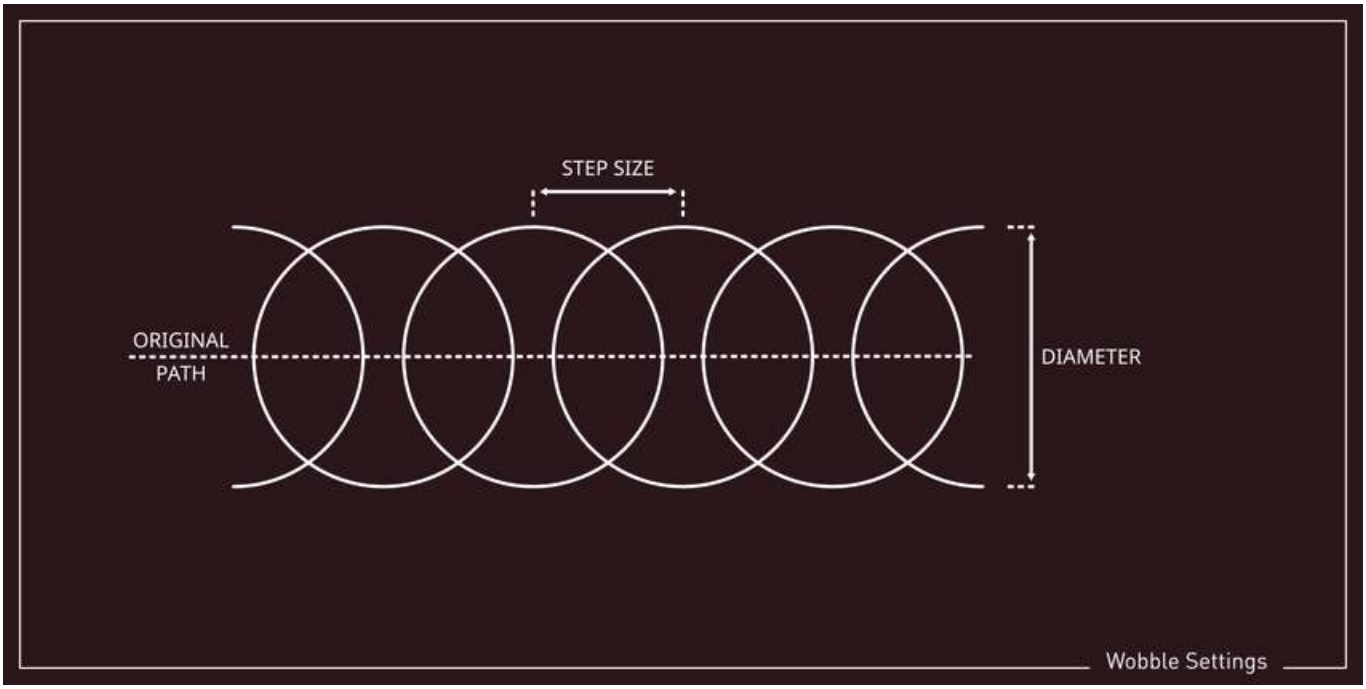
- Auto-Rotate and Angle Increment are only available for layers set to Fill Mode.
- Dot Width Adjust is available for layers set to Fill or Image Mode.
- 3D Sliced Image Mode and its related settings are only available for layers set to Image Mode.



Wobble

Because Galvo lasers are not optimized for cutting, it can be hard to apply enough power to a given area while using Line Mode to cut through material.

Enable Wobble to tell the laser to trace small circles along the path of the line, creating a thicker line and forcing the beam to spend longer in each section of the line. The **Step** value refers to how far apart the circles are, and the **Size** value refers to the diameter of the circles (in millimeters).



Angle Increment

For layers set to multiple Passes or Global Passes, adjusts the Scan Angle by this increment with each successive pass.

The Angle Increment setting applies to passes within a single job. Each time the job is run, the Scan Angle returns to its starting value. To change the Scan Angle each time the job is run, see Auto-Rotate.

Auto-Rotate

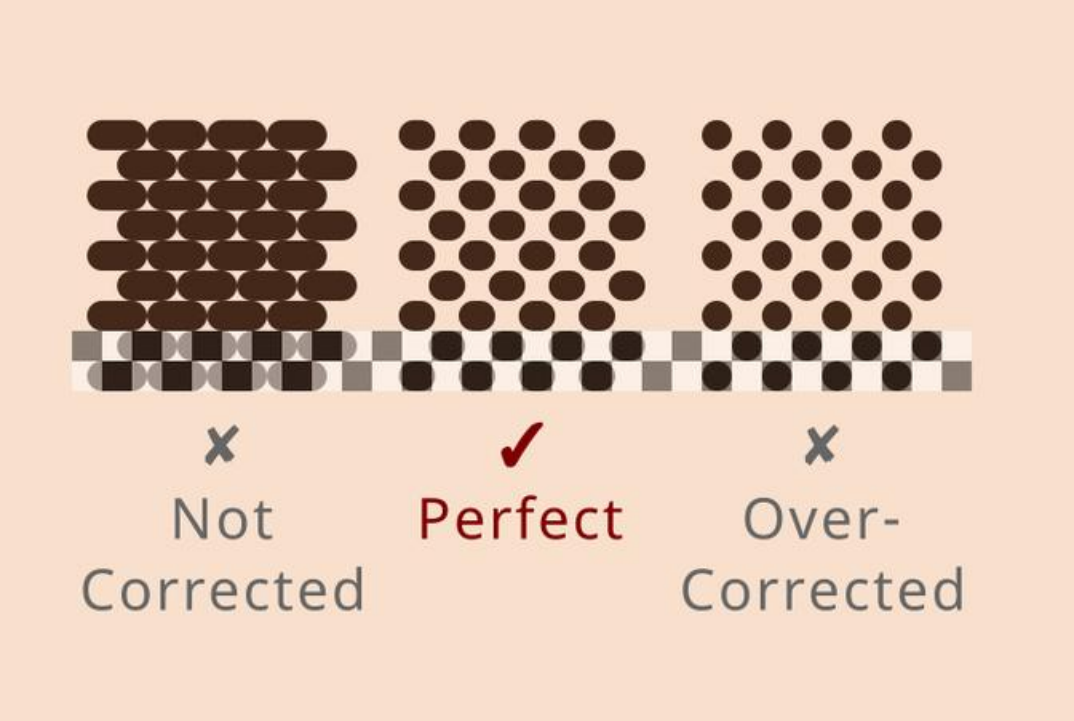
Enable to automatically adjust the starting Scan Angle with each run of a job. The Scan Angle will be increased by the Angle Increment each time the job is started, for instance by enabling Run Continuously, specifing a Repeat count in the Galvo Framing window, or manually Starting a job multiple times.

Click Reset Count to reset to the original Scan Angle.

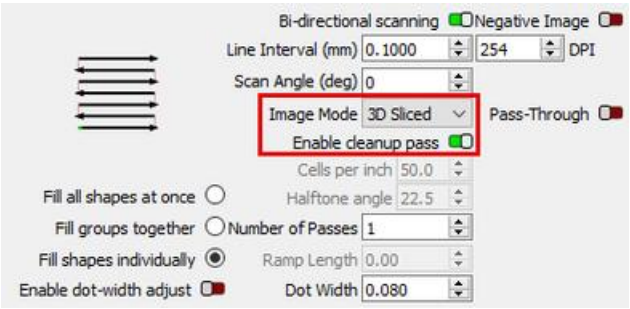
Dot Width Adjust

Dot Width Correction compensates for the thickness of the laser's beam by shortening the length of engraved scan lines. The ideal Dot Width Correction value is about half the total thickness of the laser's beam. Due to varying affects of power output on different materials, this ideal value can vary depending on the material you're working with.

The number should always be smaller than whatever you’re using as your interval setting. The valid range is from 0 to your Line Interval value.



3D Sliced Image Mode



3D Sliced is a special Image Mode for Galvo lasers only, which uses multiple passes to engrave a design one layer at a time. This mode is intended for use with depth maps – sometimes called height maps — which are specially designed grayscale images in which a pixel's brightness represents depth, rather than shading.

As of LightBurn 2.1, LightBurn support 16 bit depth maps. If you plan to run more than 256 passes, a 16 bit image is better, as it can contain more than 256 shades of gray.

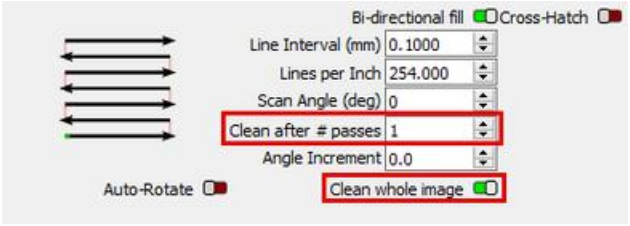
Lighter pixels are engraved with fewer passes than darker pixels, resulting in a deeper engrave in darker areas of the image. Enabling **Negative Image** inverts brightness, leading to lighter areas engraving deeper than darker areas.

See 3D Sliced Engravings for more detailed instructions on creating and working with grayscale depth maps.

Enable cleanup pass

Toggle on **Enable cleanup pass** to add a **Cleanup** Sub-Layer, where you can specify how often automatic cleanup passes should run, and adjust the settings for those passes.

Cleanup passes are typically run with a tighter Line Interval, higher Frequency, and lower Max Power than the main engraving passes, and are used to clear debris and produce a better finish.



Clean after # of passes

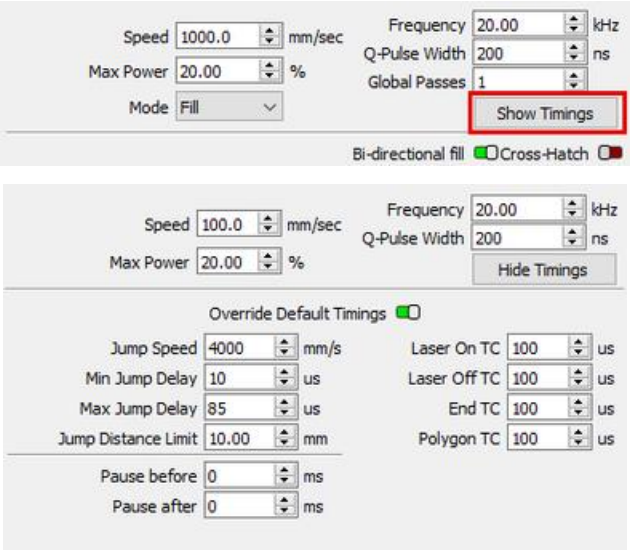
Specify the number of passes to run in between Cleanup passes.

Clean whole image

Enable **Clean whole image** to run Cleanup passes over the entire image, rather than recent passes only.

Timings

Click **Show Timings** to adjust per-layer timing settings. Click **Hide Timings** to return to the main tab of the Cut Settings Editor.



Override Default Timings

Enable **Override Default Timings** to set custom timings on a per-layer basis. If you leave it disabled, your laser will instead use the timings established in your Device Settings.

- Jump Settings control how the laser moves between cuts. Each jump causes the mirror to shake, with higher speeds and longer distances increasing the shaking. If you have wobbly lines at the start of a marking move, you may need to increase the delay and/or reduce the speed of jumps.
- Delay Settings control the delay between starting (or stopping) mirror movement and firing the laser. **Laser On/Off TC** and **End TC** control the delays for the start and end points of a path. **Polygon TC** controls the delays for other corners within a path.

Setting	Description
Jump Speed	The speed at which the galvo mirrors move between cuts. Higher values will make your job run faster at the cost of increased vibration for the mirrors.
Min Jump Delay	The minimum time the laser will wait to allow the galvo mirrors to stop shaking.
Max Jump Delay	The maximum time the laser will wait to allow the galvo mirrors to stop shaking.
Jump Distance Limit	Any jump above this limit will use the **Max Jump Delay** setting above. Jumps below this limit will use a delay value between the minimum and maximum settings.
Laser On TC	Microseconds the laser will wait to fire after the mirrors begin to move. Too high a value will leave gaps, while too low a value may cause excessive burning at the start of a cut.
Laser Off TC	Microseconds to delay turning off the laser at the end of a marking move. Too high a value may cause excessive burning at the end of a cut, while too low a value may cause gaps.
End TC	Microseconds to pause the laser's movement after completing a cut.
Polygon TC	Microseconds the galvo head will pause when turning corners. Too long a delay may cause excessive burning, while too short a delay can cause rounded or cut-off corners.

Pause Before

The number of milliseconds to pause before starting a Sub-Layer. Click the **Show Timings** button to show this setting or **Hide Timings** to hide it.

Pause After

The number of milliseconds to pause after finishing a Sub-Layer. Click the **Show Timings** button to show this setting or **Hide Timings** to hide it.

Additional Options

The settings presented in the Cut Settings Editor depend on the selected Layer Mode and the type of laser you're working with. Select an option below to learn more about different groups of settings.

Main / Shared Settings

Settings shared by almost all types of lasers and Layer Modes, including critical settings like **Speed** and **Power**.

Read More

Line Mode

Tells your laser to trace a path along the outlines of geometry in your project.

Read More

Fill Mode

Tells your laser to etch or engrave parallel lines inside the bounds of closed shapes in your project.

Read More

Offset Fill Mode

Tells your laser to fill closed shapes with lines that follow the contour of the outer shape.

Read More

Image Mode

Works similarly to **Fill Mode**, but applies additional special settings to adjust output based on the pixels in your image.

Read More

Galvo-Specific Settings

Special settings that are available for users with Galvo style lasers only.

Read More

Sub-Layers

A tool for setting up and applying multiple cut/engraving settings to the same geometry or layers in your project.

Read More

Default Layer Settings

Built-in or custom-created settings stored on a per-layer basis, that you can reset to at any time.

Read More

Note

Some options in the Cut Settings Editor are hidden if you have Beginner Mode enabled.

Sub-Layers

Sub-Layers allow you to assign multiple cut or engraving settings to the same layer and geometry. This can enable the use of single-setup etching and cleanup passes for Galvo lasers, as well as staggered engraving, cutting, and other operations for more complex jobs, regardless of machine type.

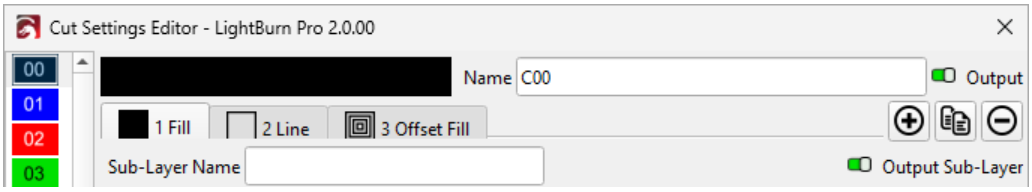
All settings of individual Sub-Layers are independently editable and presented as tabs in the Cut Settings Editor. Click a tab to access and adjust that Sub-Layer's settings.

You may have up to 11 Sub-Layers per layer.

> **Note**
>
> See Main / Shared Settings for information on applying essential settings like Speed and Power.
>
> See Additional Options for information on specialized settings not covered on this page.

Creating and Managing Sub-Layers

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Adding

Click the plus icon to create a new **Sub-Layer**, with the Default Settings of the parent layer.

Duplicating

Click the duplicate icon to create a new **Sub-Layer**, with identical settings to the currently-open Sub-Layer.

Deleting

Click the minus icon to delete the currently-open **Sub-Layer**. This option is not available if you have only one Sub-Layer.

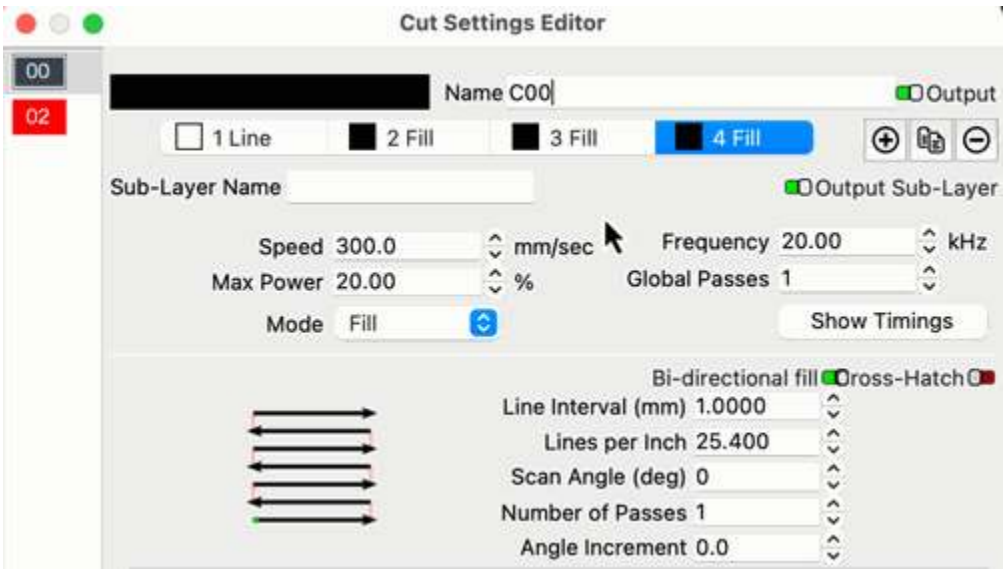
Enabling and Disabling

Toggle the **Output** switch on or off to determine whether a given **Sub-Layer** is included in the output when you Preview, Start, Send, or save a project in machine-specific format.

The universal Output switch at the top of the window determines whether an entire layer, including every Sub-Layer, is output.

Rearranging Sub-Layers

Click and drag a **Sub-Layer**'s tab to change the order in which it will be output. Sub-Layers are output in sequence, starting from left to right.



Naming Sub-Layers

To keep track of the purpose of individual Sub-Layers, you can give them unique names in the **Sub-Layer Name** field.

Sub-Layers in the Cuts / Layers Window

In the Cuts / Layers Window layers with **Sub-Layers** display their mode as **Multi**, and you may not edit the modes of individual Sub-Layers through that window. as seen below.

Click an individual Sub-Layer tab in the Cuts / Layers Window to edit a limited number of settings.

You can reorganize, as you can within the full Cut Settings Editor in the quick settings, by clicking and dragging. Sub-Layers are output in sequence, starting from left to right.

Legacy Behavior: Fill + Line

In older versions of LightBurn, Fill+Line mode allowed primitive Sub-Layer use to draw an outline around an engraved shape. This has since been superseded. The legacy behavior can be approximated with Sub-Layers, one for Fill and a second one for Line with the same base motion settings.

Danger

LightBurn will automatically convert pre-1.2 project files with Fill+Line settings to Sub-Layers in 1.2, including Fill+Line behavior. This produces two layers, one with your legacy Fill settings, and a subsequent layer with Line settings, to be run afterwards. Going backwards from current project files to pre-1.2 LightBurn is likely to result in breakages involving Sub-Layers due to how complex the changes are.

Additional Options

The settings presented in the Cut Settings Editor depend on the selected Layer Mode and the type of laser you're working with. Select an option below to learn more about different groups of settings.

Main / Shared Settings

Settings shared by almost all types of lasers and Layer Modes, including critical settings like **Speed** and **Power**.

Read More

Line Mode

Tells your laser to trace a path along the outlines of geometry in your project.

Read More

Fill Mode

Tells your laser to etch or engrave parallel lines inside the bounds of closed shapes in your project.

Read More

Offset Fill Mode

Tells your laser to fill closed shapes with lines that follow the contour of the outer shape.

Read More

Image Mode

Works similarly to **Fill Mode**, but applies additional special settings to adjust output based on the pixels in your image.

Read More

Galvo-Specific Settings

Special settings that are available for users with Galvo style lasers only.

Read More

Sub-Layers

A tool for setting up and applying multiple cut/engraving settings to the same geometry or layers in your project.

Read More

Default Layer Settings

Built-in or custom-created settings stored on a per-layer basis, that you can reset to at any time.

Read More

Note

Some options in the Cut Settings Editor are hidden if you have Beginner Mode enabled.

Default Layer Settings

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=86_obqHz7U4

Cut settings are normally pre-filled with the last settings used for a given layer, but LightBurn has an option to automatically reset to either universal settings or your own customized defaults each time you start a new project.

You can also reload custom or universal defaults at any time in the Cut Settings Editor.

Use the Material Library to save a wider range of settings

Material Libraries are also available for saving and reusing cut settings, and are recommended for users who need to store and reapply many distinct, frequently-used settings.

Note

See Main / Shared Settings for information on applying essential settings like Speed and Power.

See Additional Options for information on specialized settings not covered on this page.

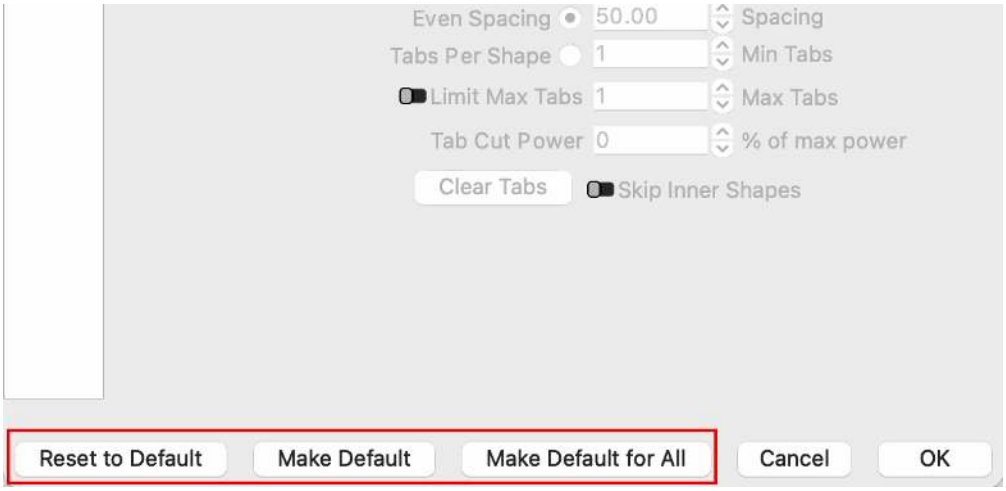
Setting Defaults

At the bottom of the Cut Settings Editor, there are three buttons to manage default layer settings.

Make Default: remember the settings for this palette color. Whatever your current settings are will become the default for this color. Default settings are saved separately for each device profile.

Reset to Default: reset the layer to the default settings. Hold `Shift` while clicking Reset to Default to reset to LightBurn's universal defaults.

Make Default for All: your current layer settings will become the default for all palette colors.



Always pay attention to units of distance and time when entering Speed settings

When entering **Speed** values recommended by your laser's manufacturer or other LightBurn users, make sure to use the same units of distance and time as those from the recommendation, or to convert the values to your preferred units.

A given number of **millimeters per second** is much faster than that same number in **millimeters per minute**. Mixing up units can lead to reduction in power output due to unexpectedly high speeds, or excessive power output — and even fire — due to unexpectedly slow speeds.

Change your Control Units in the **Dimensions / Units** tab of the **Device Settings** window. LightBurn automatically converts any existing values when you switch between units.

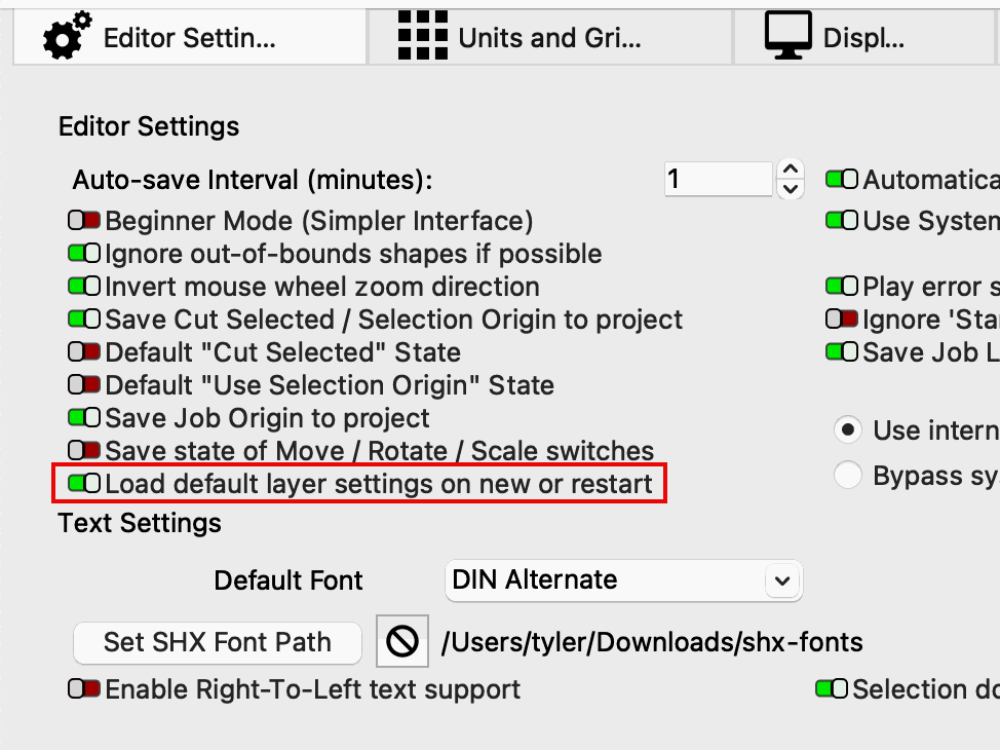
LightBurn's Universal Defaults

The universal default layer is set to Line Mode, at 6000 mm/min (100 mm/sec) speed and 20% power, with no advanced options enabled.

Layers switched to **Fill**, **Offset Fill**, or **Image** mode after having been restored to default will be set to the same speed and power as above, a **Line Interval** of .100, and to **Fill all shapes at once**. The default dithering mode of Image layers is **Stucki**.

Load Default Layer Settings on New or Restart

To automatically load defaults instead, enable **Load default layer settings on new or restart** in the **Editor Settings** tab of the Settings window.



Additional Options

The settings presented in the Cut Settings Editor depend on the selected Layer Mode and the type of laser you're working with. Select an option below to learn more about different groups of settings.

Main / Shared Settings

Settings shared by almost all types of lasers and Layer Modes, including critical settings like **Speed** and **Power**.

Read More

Line Mode

Tells your laser to trace a path along the outlines of geometry in your project.

Read More

Fill Mode

Tells your laser to etch or engrave parallel lines inside the bounds of closed shapes in your project.

Read More

Offset Fill Mode

Tells your laser to fill closed shapes with lines that follow the contour of the outer shape.

Read More

Image Mode

Works similarly to **Fill Mode**, but applies additional special settings to adjust output based on the pixels in your image.

Read More

Galvo-Specific Settings

Special settings that are available for users with Galvo style lasers only.

Read More

Sub-Layers

A tool for setting up and applying multiple cut/engraving settings to the same geometry or layers in your project.

Read More

Default Layer Settings

Built-in or custom-created settings stored on a per-layer basis, that you can reset to at any time.

Read More

Note

Some options in the Cut Settings Editor are hidden if you have Beginner Mode enabled.

Output and Positioning

Laser Window

The **Laser Window** is the hub for setting up and selecting a laser, **Framing** and **Starting** jobs, determining job positioning, and more.

Accessing the Laser Window

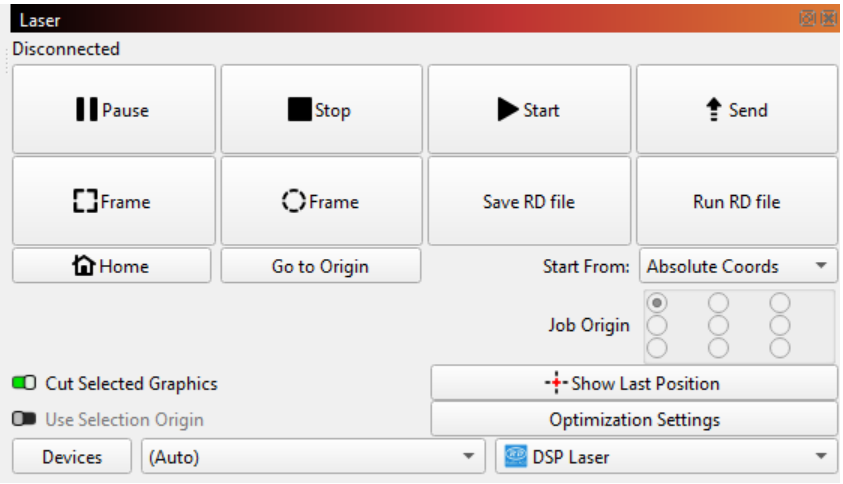
The **Laser Window** is located in the bottom right corner of LightBurn by default. If you have closed the Laser Window, go to Window → **Laser** to re-enable it. To restore it and all other windows to their default positions, go to **Window** → Reset to Default Layout.

> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Laser Window Options and Settings

The type of laser you have active, and certain options in the Settings window, including enabling Beginner Mode, will affect the way the **Laser Window** looks.

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



> **Note**
>
> The above image shows the default layout of the **Laser Window** as it appears for a DSP laser. Some settings that are not available for DSP lasers, or are not enabled by default, are listed in the page below. Check this page's table of contents if you do not see the setting you're looking for in the image.

Start

Commands your laser to immediately begin running your current project.

> **Note**
>
> If you have a Galvo laser, you'll see the Framing window first, unless you turn off **Require framing before start** in your Device Settings.

> **Pause and Stop from the keyboard**
>
> While your laser is running, you can use your keyboard's `Pause` key to Pause the laser and `Ctrl` +`Break` to Stop it. `Break` is typically on the same key as `Pause`, but more compact keyboards may not have this key.

Stop

Immediately aborts a currently running job.

Pause

Halts a running job. Click the **Resume** button to continue a paused job.

Send

The **Send** button, if your laser supports it, transfers the current job to the laser as a named file, so you can select and run it from the laser's controller.

When you use **Start**, the controller begins running the job as soon as sufficient data has reached the controller. Using Send instead of Start can sometimes help prevent file corruption during transfer.

Hold `Shift` while clicking Send to have the laser immediately start the job after the entire project has been sent.

> **Note**
>
> The Send option is only available for DSP lasers.

Framing

> **Note for Galvo users**
>
> This section covers Framing behavior for Gantry lasers. If you have Galvo laser, see Galvo Framing.

Use the two **Frame** buttons to preview the position of the job in your laser's work area, by commanding the laser to trace a path around the area where your project will output.

The speed at which the laser travels during Framing is determined by the value set in the **Speed** field in the Move Window.

Frame Continuously

Enable **Frame Continuously** in the **Basic Settings** tab of Device Settings to command your laser to run the Framing path repeatedly, until you tell it to stop. Framing Continuously is often helpful when adjusting material positioning in your laser's work area.

Laser on When Framing

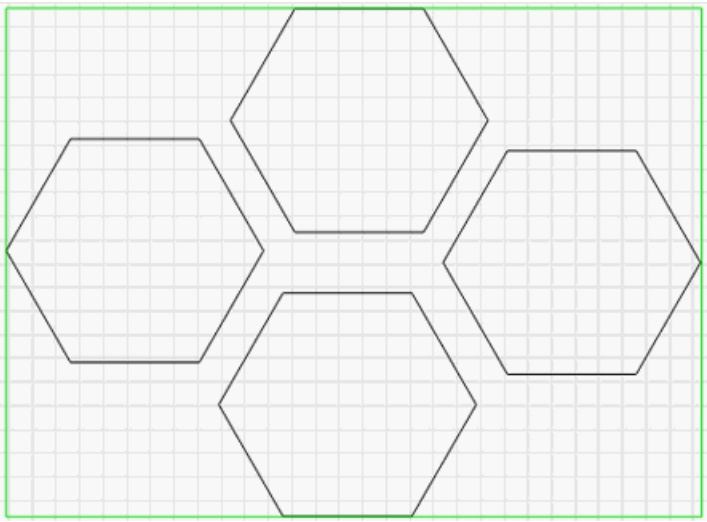
Enable **Laser on when framing** in the Device Settings window to **Fire** the laser during the Framing process, at the **Power** level set in the Move Window.

If Laser on when framing is not enabled, you can hold `Shift` while pressing Frame to enable the laser during the framing process.

> **Danger**
>
> This changes the default behavior of the laser and can cause the laser to fire when a user is not expecting it. Enable this setting with caution, with a very low power level, and only when using a diode laser.

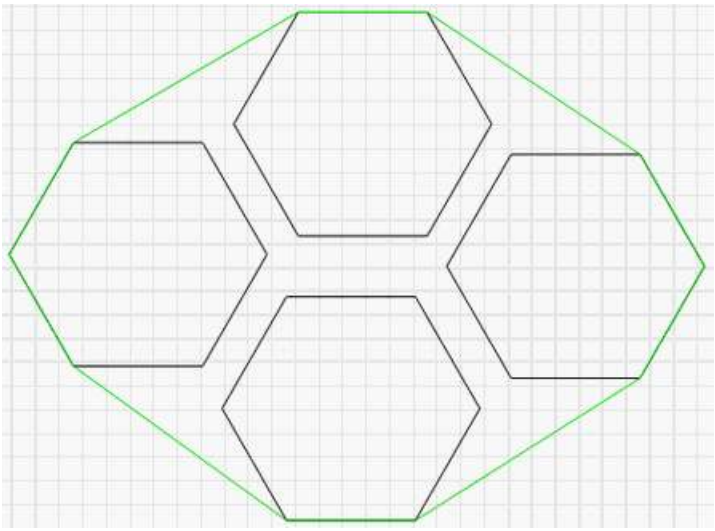
Bounding Box Frame 

Commands the laser to trace a path defined as the smallest possible rectangle that will fully contain all graphics you're sending to the laser.



Rubber Band Frame 

Traces the smallest possible path that fully contains all graphics in the design you're sending to the laser, as if a rubber band were stretched around them.



> **Tip**
>
> Rubber Band Framing is useful for lining up jobs with irregular shapes where a box outline doesn't fit well. For example, a long, thin diagonal shape, or a triangle.

Save Machine Files

This button will change depending on the type of laser you're using — it may say **Save GCode** (for GRBL, Smoothieware, or Marlin controllers), **Save RD File** (Ruida), **Save OUT File** (TopWisdom), or **Save UD5 File** (Trocen).

All options save your current project as a machine-ready file, in the appropriate format for your type of laser. You can transfer this file to your machine to run it without ever having to directly connect using LightBurn.

These options are also available in the File Menu.

Run Machine Files

Use this option to load and run a file previously saved in the appropriate format for your type of laser. To use this option, LightBurn must be directly connected to your laser.

This button will change depending on the type of laser you're using — it may say **Run GCode**, **Run RD File**, **Run OUT File**, or **Run UD5 File**.

Home

For GCode lasers: Commands your laser to execute a homing cycle, during which it moves toward the home position looking for the switches that activate when it hits the boundary.

For DSP lasers: Moves the head of the laser to the machine origin.

Homing is how your laser locates its 0,0 position, and all subsequent positioning information is determined relative to that point.

> **Warning**
>
> Do not use this button if your laser does not have homing switches. If you are unsure, contact your laser's manufacturer.

Go to Origin

Commands your laser to jog to the position currently set as the **User Origin**.

See Coordinates and Job Origin for more information.

Start From and Job Origin

These two controls determine where the job is placed on the bed of your laser.

See Coordinates and Job Origin for more information.

Cut Selected Graphics

This toggle switch tells LightBurn to only send the portion of your design that is currently selected when you click **Start**, **Send**, or **Save Machine File**.

See Cut Selected Graphics for more information.

Use Selection Origin

Only available in combination with **Cut Selected Graphics**, the **Use Selection Origin** button tells LightBurn to calculate origin relative to selected graphics only, and not the whole design.

See Cut Selected Graphics – Use Selection Origin for more information.

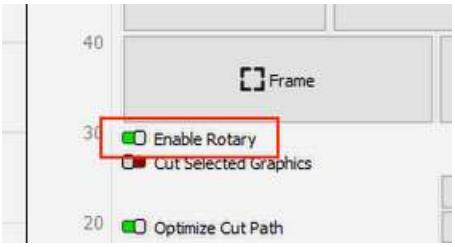
Show Last Position

Places a cross-hair cursor in the Workspace at the last reported location of the laser head. If you jog the laser within LightBurn using any of the positioning tools, the position will be updated.

The position does not update live — for example, it will not update while a job is running. Live updating might encourage some users to watch the screen rather than their laser — lasers should never be left unattended while running.

Enable Rotary

This switch controls whether **Rotary Mode** is enabled. When enabled, rotational commands are sent to the laser to control rotary motion.



In order to see this switch in the **Laser Window**, you must turn on Show rotary enable in main window in your Settings/Preferences.

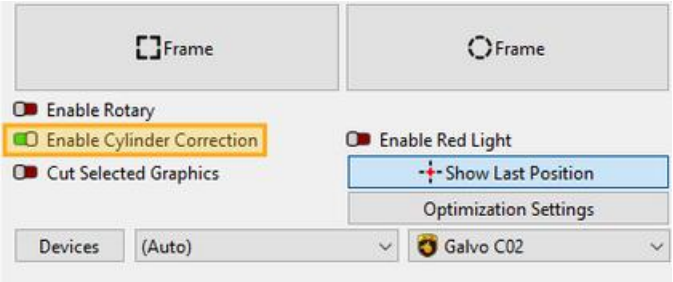
Rotaries are used to turn cylindrical objects so that their surfaces can be marked by a laser. Before using your laser with a rotary, you must first establish several important parameters in the **Rotary Setup** window that control the rotary's motion.

See Rotary Mode for more information.

Enable Cylinder Correction

> **Note**
>
> This setting is available for Galvo lasers only.

This switch controls whether **Cylinder Correction** is enabled. When enabled, LightBurn warps the output of your design to correct for distortion caused by engraving cylindrical objects, without using a rotary.



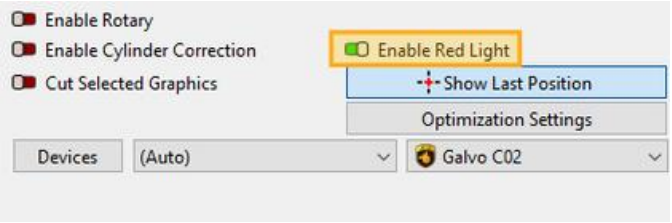
In order to see this switch in the **Laser Window**, you must turn on Show cylinder correction enable in main window in your Settings/Preferences.

Before using Cylinder Correction, you must first establish several important parameters in the **Cylinder Correction Setup** window that determine how LightBurn will warp your output.

See Cylinder Correction for more information.

Enable Red Light

> **Note**
>
> This setting is available for Galvo lasers only.



This switch controls whether your laser's Red Dot light is enabled. In order to see this switch in the **Laser Window**, your laser's Red Dot light must be controlled by an IO Port, and the appropriate port must be set in your Device Settings.

Optimization Settings

This button opens the **Optimization Settings** window, where you can adjust various options that control the order in which your graphics are cut or engraved by the laser.

See Optimization Settings for more information.

Devices

Opens the **Devices** window, where you can create, remove, or edit device profiles for lasers you want to use with LightBurn.

> **Right-click to reset the USB connection**
>
> Right-clicking the Devices button closes and re-opens the connection to the currently selected device to reset it. This currently does not work with BSL devices.

See Devices for more information.

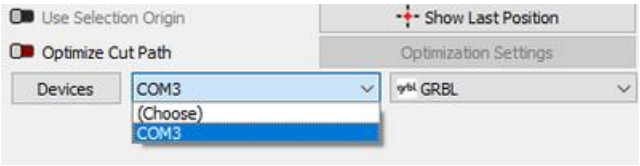
Device Selection Dropdown

If you've created multiple device profiles for multiple lasers, select the laser you are currently working with the dropdown menu.



Serial Port Dropdown

LightBurn will attempt to automatically select the correct serial port and connect to your laser, but you may need to manually select it from this dropdown. When connected, the **Disconnected** message in the top left corner of the Laser window will change to **Ready**. If you are using a GCode-based laser, you will see a startup message in the Console window.



You may need to right-click the **Devices** button in the Laser window to refresh the list of available ports.

If you're having trouble connecting to your laser, see Troubleshooting: Connection Problems.

Device Status

The readout in the top left of the **Laser Window** indicates the current status of the laser.

- **Disconnected** indicates that the laser is not connected to LightBurn.



- **Ready** indicates that the laser is connected to LightBurn and ready to receive commands.

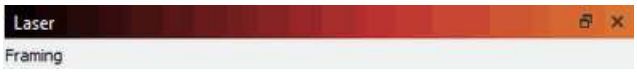


- **Busy** indicates that the laser is connected to LightBurn and actively completing a process.

  The progress bar indicates the percentage of commands that have been streamed to the laser, along with a time estimate indicating how long the laser has been streaming, and how much longer it will continue streaming. These estimates can be close to the total runtime of the project, but are not identical.



- **Framing** appears for Galvo lasers only, to indicate that the Framing process is active.



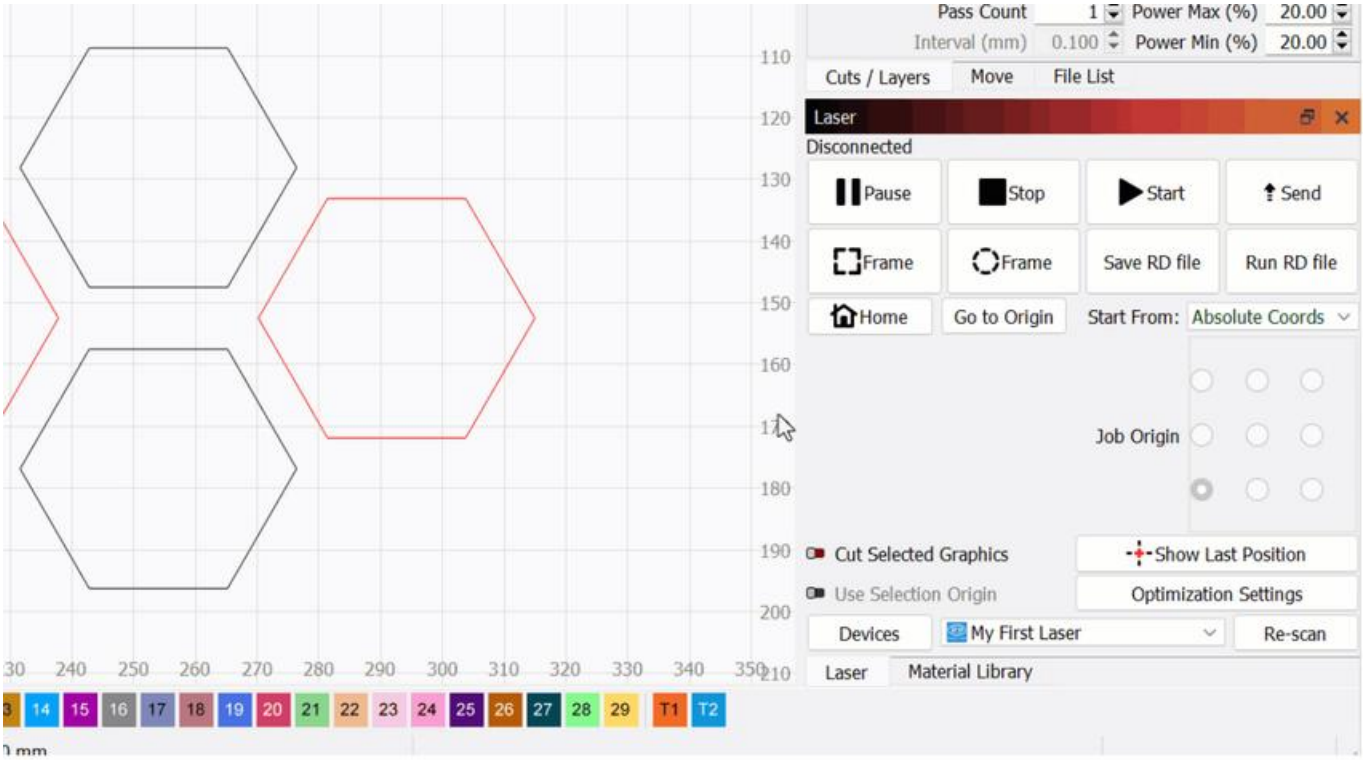
Coordinates and Job Origin

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=aphiLTUWcjE

There are three options for telling LightBurn where to output your project within the work area of your machine, each available in the **Start From** menu in the Laser Window.

- **Absolute Coordinates** tells LightBurn to output graphics in your laser's work area at a location that matches their position in your Workspace.
- **Current Position** tells LightBurn to output graphics relative to the position of the laser head at the moment you start a job.
- **User Origin** tells LightBurn to output graphics relative to a custom-defined origin that you set before starting a job.

The 9-dot **Job Origin** selector beneath the dropdown menu determines the orientation of output relative to the Current Position or User Origin. It will be grayed out if you have Absolute Coords selected, but clickable if you choose Current Position or User Origin.



Note for Galvo Users

If you have a Galvo laser, you won't see the Start From mode dropdown menu or 9-dot Job Origin selector in the Laser Window — that's because Galvo lasers always use Absolute Coordinates positioning.

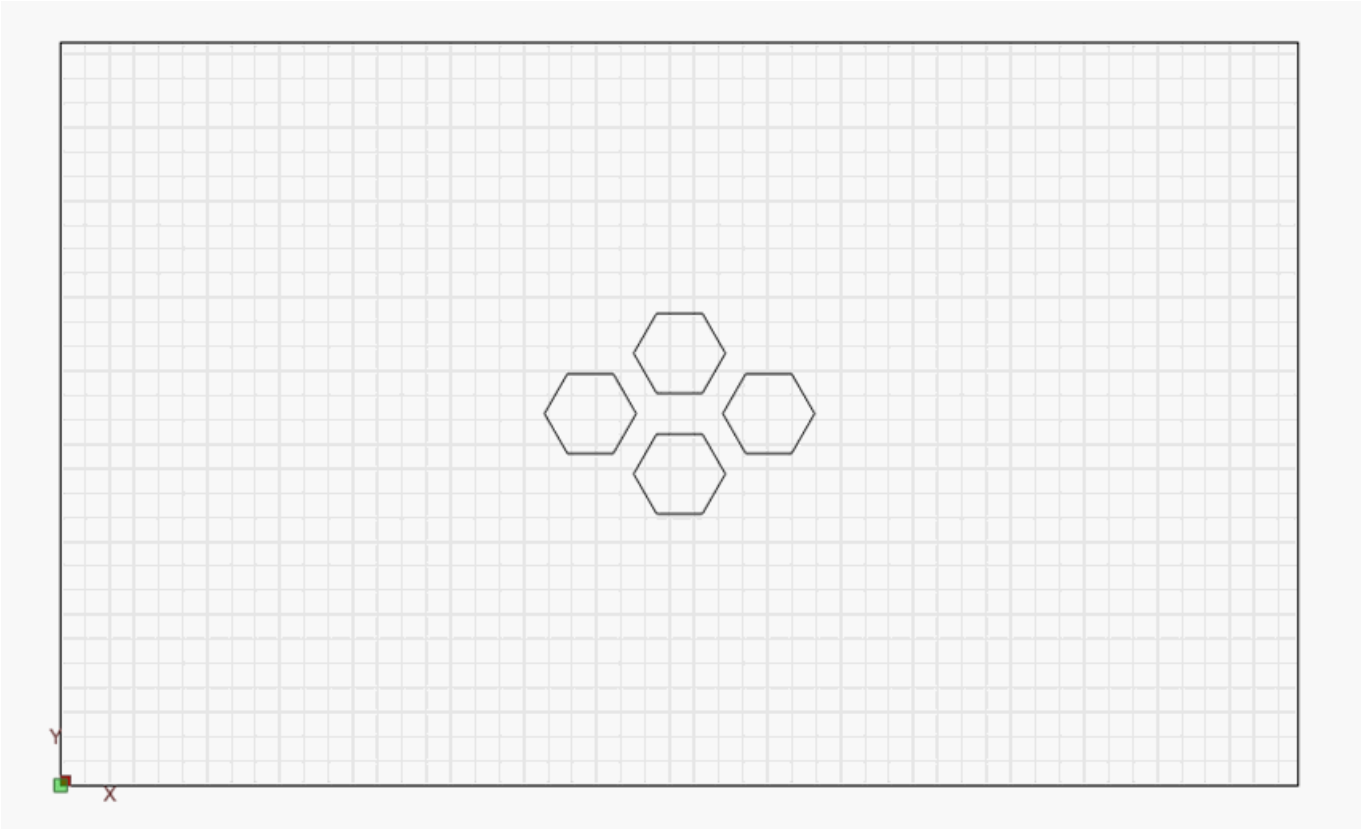
Absolute Coordinates

Absolute Coordinates is the most intuitive option. The grid in your LightBurn Workspace has a coordinate system that matches the physical work area of your laser. The 0,0 position is always the location of your laser's **Machine Origin**.

With **Absolute Coords** selected as your Start From mode, anything you place in the LightBurn grid will be output in the corresponding location in your machine's work area.

> **Note**
>
> Using absolute positioning requires a laser with a fixed origin. If you have a small diode laser that does not have homing switches, you will need to manually home the machine. See: Machines without homing sensors / limit switches.

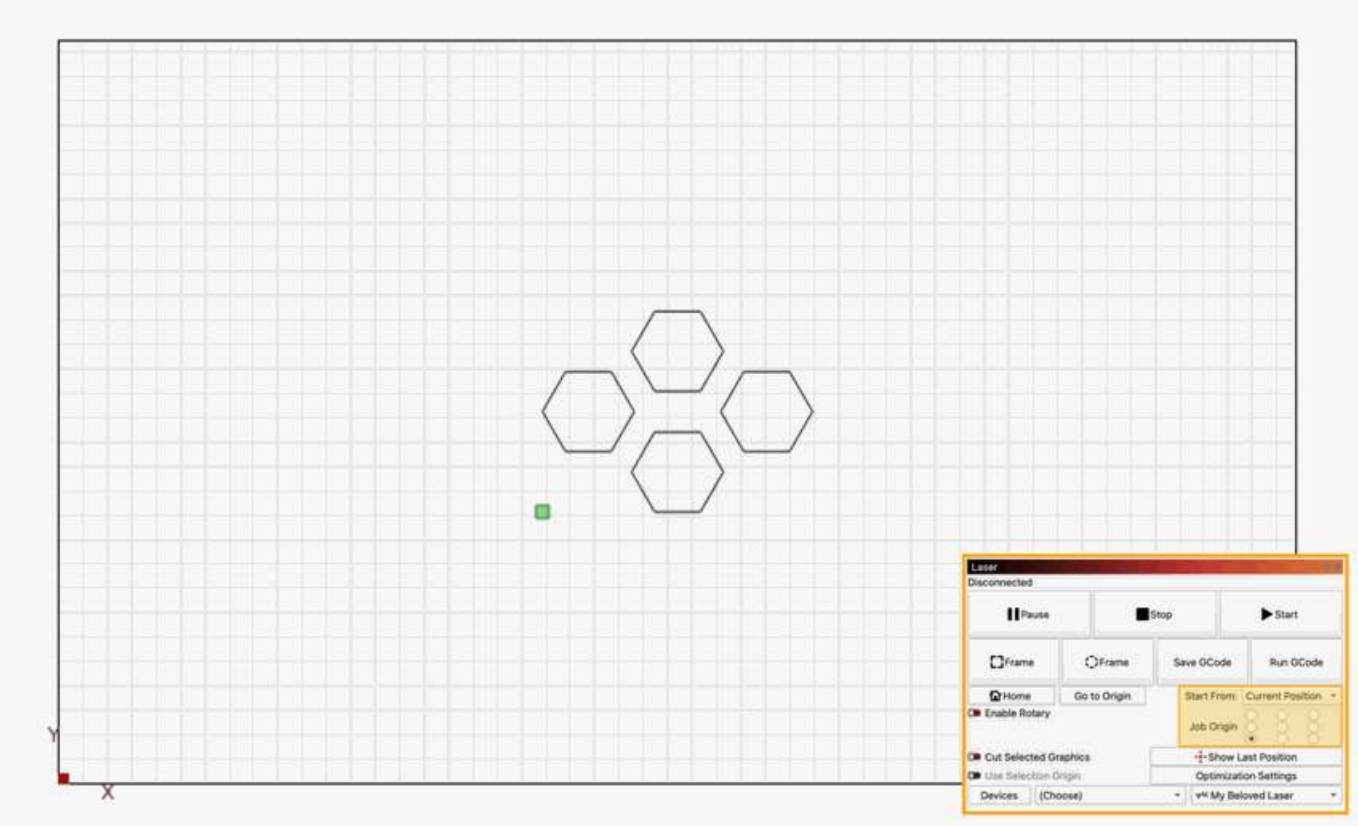
In the image below, the four hexagons placed in the middle of the LightBurn grid will be cut in the middle of the machine work area. The green square in the lower left of the image represents the **Job Origin**, and the red square in the same place shows the **Machine Origin**. When using Absolute Coords, these are always in identical locations.



Current Position

With **Current Position** selected, your job outputs relative to wherever the laser head is located when you press the **Start** button. The **Job Origin** control in the **Laser Window** tells LightBurn how to orient output relative to the laser head.

In this image, we're starting from the Current Position, with the Job Origin set to the lower left:

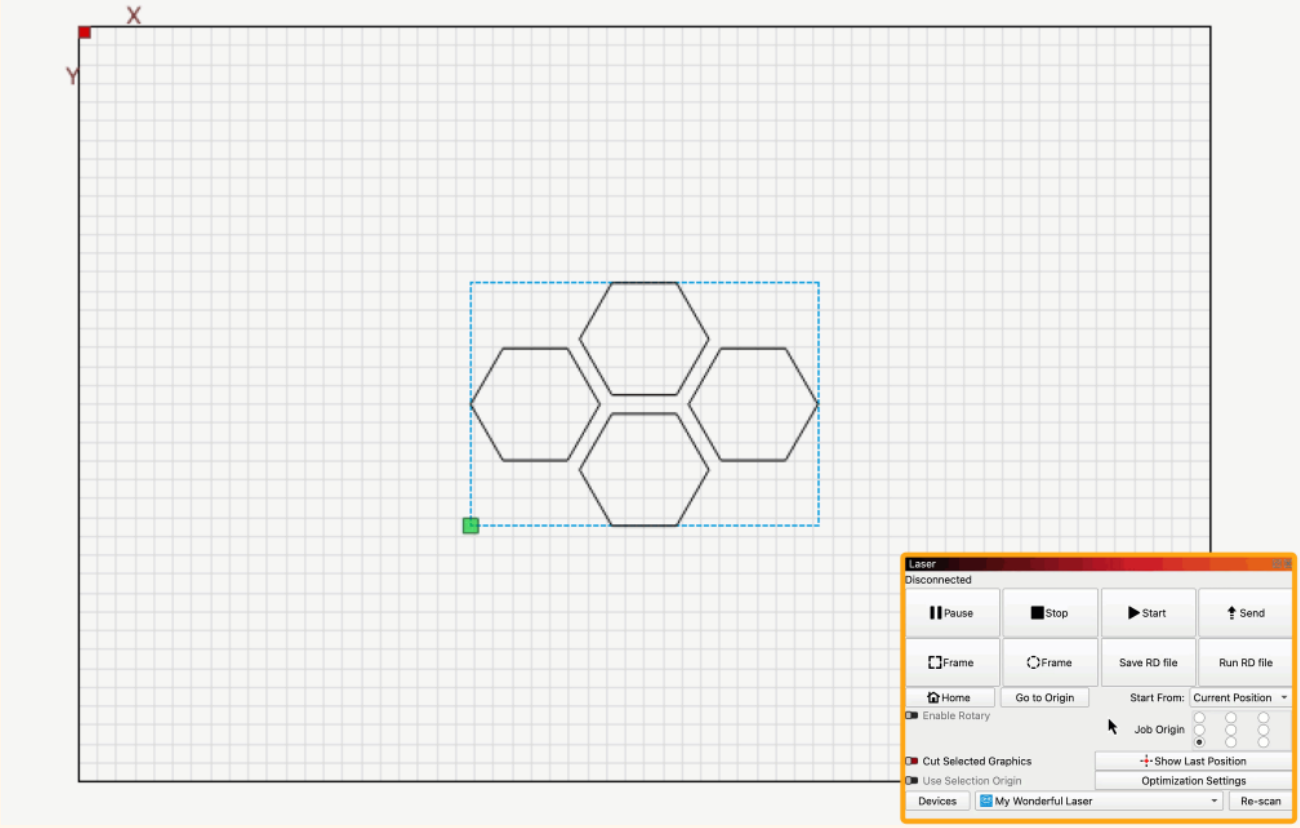


Notice that the green Job Origin indicator has moved. This represents the position of the laser when you start the job, so the laser is going to move slightly up and to the right from wherever it is, cut the four hexagons, and go back to where it started.

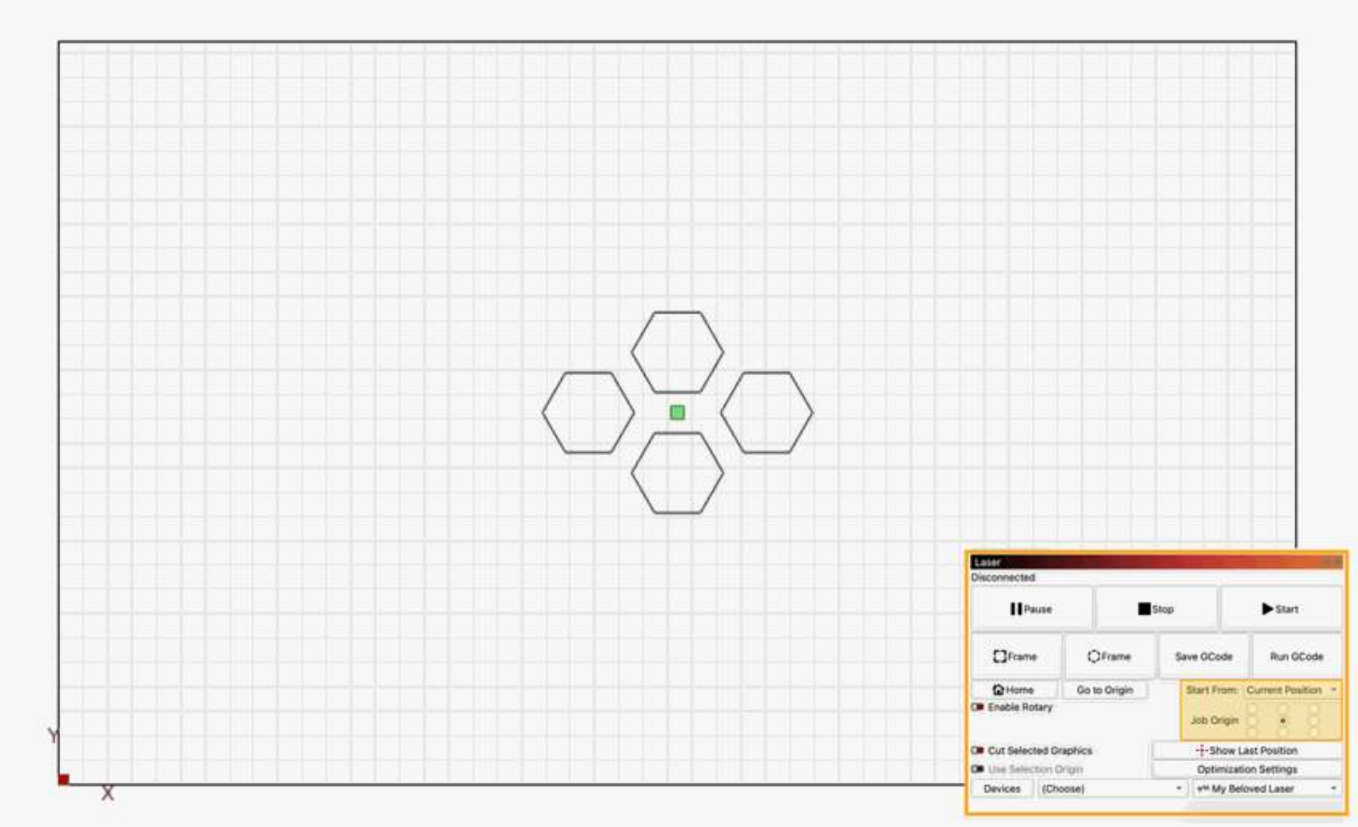
Why is the Job Origin offset from the graphics?

When using Current Position or User Origin, the green Job Origin indicator is always located on a corner, side, or at the center of an imaginary box — the smallest possible rectangle that can fully contain all graphics you're sending to the laser.

Below, we've added a Tool Layer to represent that imaginary box. Notice how the Job Origin moves around the box as we change the location in the 9-Dot selector.



Now imagine that you want to etch this four hexagon pattern onto a coaster or a phone case. Lining it up from the lower left is not easy. If you change the Job Origin setting to center, you get this instead:

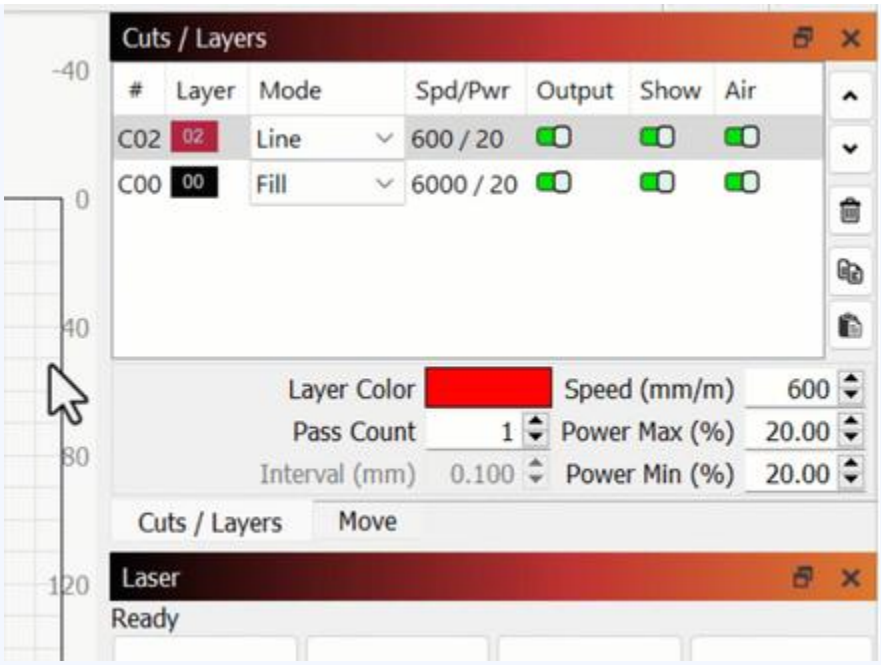


The job will be centered around the current position of the laser head. If you position the laser head directly over the center of the item you want to etch, the output will be centered on the item.

Jogging Your Laser

Both Current Position and User Origin require you to jog your laser into a desired position to define a custom origin. LightBurn has a few tools for adjusting the location of your laser's head within its work area — the most straightforward option for beginners is to use the arrow keys in the Move Window. By default, the Move Window is hidden behind the Cuts / Layers Window — click the Move tab to expose it.

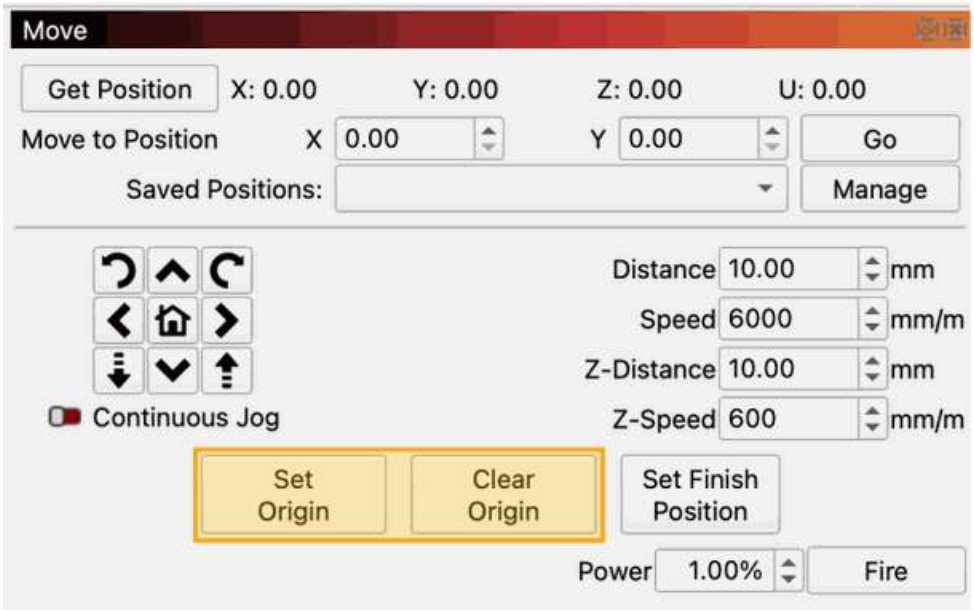
The arrow keys move your laser up, down, right, or left. You can set the amount the laser travels with each click in the **Distance** field, and the rate at which it travels in the **Speed** field.



User Origin

User Origin works almost exactly the same as **Current Position**, except that the starting location is programmable. To set a User Origin, you must first jog your laser to the location you want your job to start from, then set the origin.

Some lasers with digital displays, such as those with Ruida controllers, have an `Origin` button which you press to set the origin. GCode-based systems use the **Set Origin** button in the Move Window in LightBurn to do the same thing.



For supported systems, use **Clear Origin** to remove and reset a custom-set origin.

After setting the origin, you are free to jog your laser around its work area. With User Origin selected as the **Start From** mode, the laser will always move back to that programmed location and start the project from there.

You can also use the **Go to Origin** button in the Laser Window to command your laser to travel back to the origin you've set.

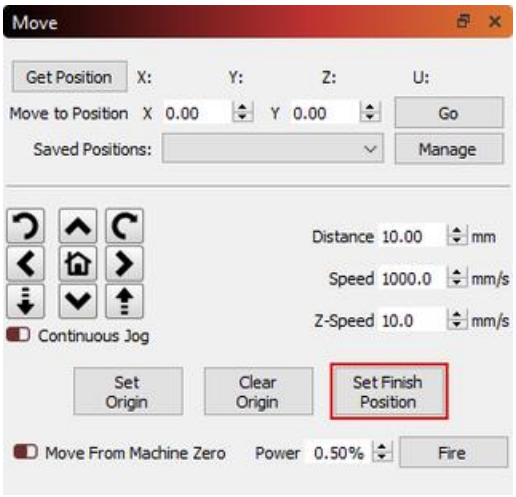
As with Current Position, the **Job Origin** setting determines how output will be oriented relative to the user-defined origin.

Finish Position

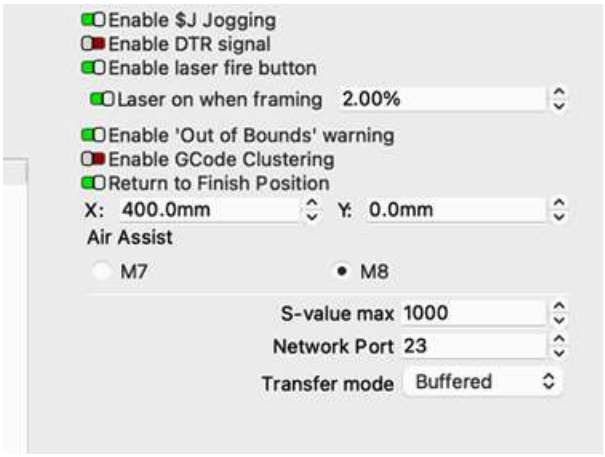
> **Note**
>
> The **Finish Position** option is only available for GCode-based controllers. DSP controllers manage this internally.

LightBurn allows you to control where you want the head of the laser to return after a job is finished. By default, it will return to 0,0, but this is not always optimal for all machines.

To change your laser's Finish Position, go to the Move Window and use the arrows to position your laser head wherever you would like it to return to after the job is finished, then click the **Set Finish Position** button. Your machine will now remember that spot for future jobs. Finish Position can be adjusted at any time by repeating the above steps for a new location.



Users with GCode-based lasers can also adjust or disable the Finish Position in Device Settings.


Enable $J Jogging
Enable DTR signal
Enable laser fire button
Laser on when framing 2.00%
Enable 'Out of Bounds' warning
Enable GCode Clustering
Return to Finish Position
X: 400.0mm
Y: 0.0mm
Air Assist
M7
M8
S-value max 1000
Network Port 23
Transfer mode Buffered

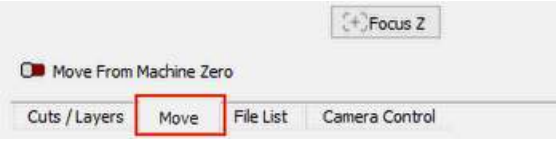
Move Window

The **Move Window** is used primarily for jogging, positioning, and homing the laser.

Accessing the Move Window

By default, the **Move Window** is docked behind the Cuts / Layers Window on the upper right side of your screen. Click the Move tab to bring the Move Window to the front.

If you have closed the Move Window go to Window → **Move** to re-enable it. To restore it and all other windows to their default positions, go to **Window** → Reset to Default Layout.



> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Move Window Options and Settings

The type of laser you have active and certain options in the Settings window — including enabling Beginner Mode — will affect the way the **Move Window** looks.

Click any option in the image below to jump to more information, or scroll down for a list of options and descriptions.



> **Info**
>
> Ruida Controller Devices: The **Set Origin**, **Clear Origin**, or **Set Finish Position** controls will not be displayed as these are handled by the controller itself. LightBurn will generally show only controls you can use.
>
> Galvo Devices: The Move window is not available for EZCad2/JCZFiber devices. EZCad3 and BSLFiber devices have an alternate Move window, described on the EZCad3 page

Get Position

The **Get Position** button queries the controller for its current location and displays the current X, Y, Z, and U Axis coordinates.

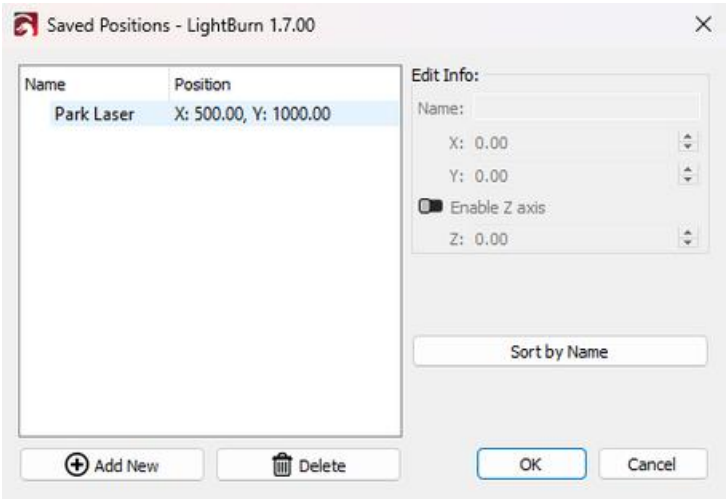
Move to Position

Selecting the **Go** button will move the laser to the X and Y Axis coordinates in the **Move to Position** entry boxes.

Saved Positions

The **Saved Positions** drop-down displays a list of previously stored positions. Use Saved Positions for jogging to known locations on the work area — for example a specific corner, the location of your rotary tool, and commonly used jigs. Manage the contents of this list by clicking the **Manage** button.

Managing Saved Positions



1. Add new positions to the Saved Positions list by pressing the **Add New** button and entering the desired coordinates.
2. Activate Z Axis movements by toggling the **Enable Z Axis** option and updating the Z Axis coordinate.
3. Optionally, press the **Sort by Name** to sort the position list alphabetically.
4. Press **OK** to save the list.

Manual Movement

Move the laser by a specific amount in a given direction through a process called Jogging.

Move Window Directional Arrows

Jog the laser throughout its work area by clicking any of the arrow buttons that surround the **Home** button. Each click will move the laser by amount entered in the **Distance** field, at the rate set in the **Speed** field.

> **Note**

The Speed settings also control the Framing speed and the movement speed of the Position Laser tool.

If your laser has a motorized Z Axis, the dashed up and down arrows move the Z Axis up or down.

The rotational arrows jog the A Axis, commonly used for rotaries.

Note

If you're using an A Axis rotary, the Distance field determines degrees of movement of the rotary axis with each click.

Continuous Jog

The **Continuous Jog** toggle allows supported lasers to move continuously, by pressing and holding any of the directional arrows. The laser will keep moving until the button is released.

Note

This switch is available for all DSP lasers. Users with GCode-based lasers running GRBL version 1.1f or later may need to toggle on Enable $J Jogging in the Device Settings window to reveal this switch.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=q9ypyQFFf8I

Keyboard Shortcuts

Each keyboard shortcut will move your laser by the amount entered in the **Distance** field, at the rate set in the **Speed** field.

- **Jog Laser Left**: `Alt`/`Option` + `Ctrl`/`Cmd` + `[`
- **Jog Laser Right**: `Alt`/`Option` + `Ctrl`/`Cmd` + `]`
- **Jog Laser Up**: `Shift` `Ctrl`/`Cmd` + `]`
- **Jog Laser Down**: `Shift` `Ctrl`/`Cmd` + `[`

Keypad Jogging

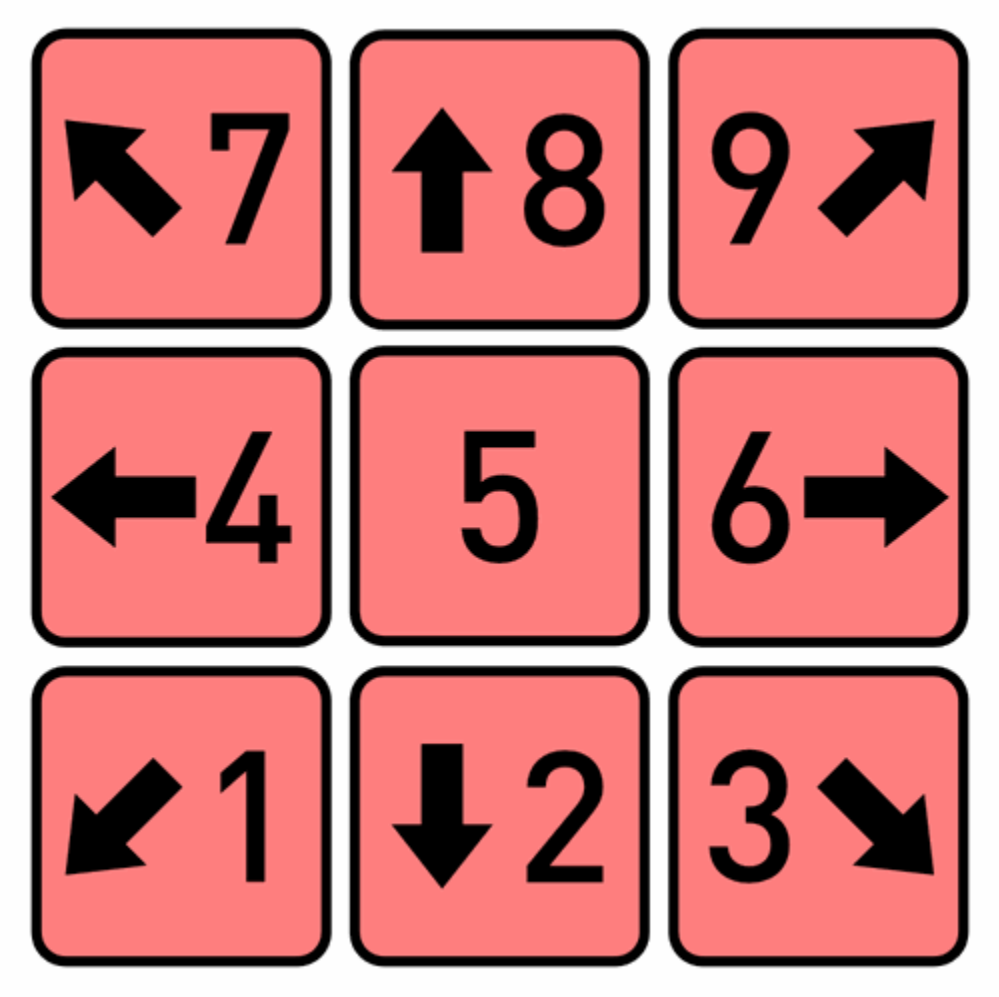
You can use a keypad to Jog your laser. In order to use keypad Jogging, you must enable **Numlock** on your keyboard, and the **Edit Window** (Workspace) — the area where you create and edit graphics — must be in focus.

What does it mean for a window to be in focus?

There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.

If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus. Selecting objects in the Workspace will also bring it into focus.

Each press will move your laser by amount entered in the **Distance** field, at the rate set in the **Speed** field. If Continuous Jog is enabled, the laser will Jog until you release the key.



Arrange Menu → Move Laser to Selection

The Jog options are also available in the Arrange Menu, within the Move Laser to Selection submenu.

Move From Machine Zero

The **Move From Machine Zero** toggle moves the laser from machine zero instead of from the current position of the laser.

Instead of determining the amount of travel, the **Distance** field controls the coordinate location the laser will travel to.

Device-specific Buttons

Focus Z

Focuses the Z Axis of the laser. To use this function, your laser must have a movable Z Axis and have an automatic focusing feature that is supported by LightBurn.

Set Origin

Creates a custom-set starting point, known as the User Origin.

Clear Origin

Press **Clear Origin** to remove and reset a custom-set User Origin.

Set Finish Position

Press **Set Finish Position** to establish the position the laser head will move to after a job is finished.

See Finish Position for more information.

Fire

Turns your laser on at the **Power** level specified in the adjacent field.

Use the **Fire** button to enable the laser at low power for focusing, Framing, and positioning.

> **Danger**
>
> The Fire button should only be used for diode lasers, which generally don't have a red dot pointer for Framing.
>
> Always wear proper eye protection when Firing your laser. Consult your laser's manufacturer for information on the proper eye protection required for your laser.
>
> This should never be used for a CO_2 laser, which has an invisble beam that could blind you or start a fire.

See Device Settings — Enable Laser Fire Button for more information.

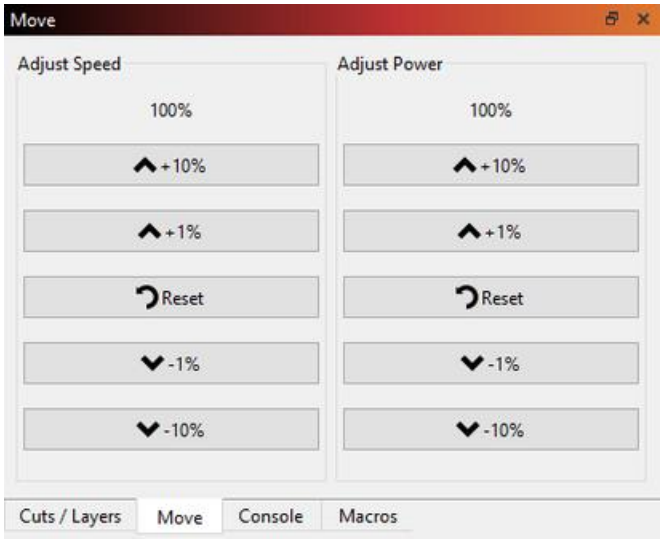
Power

Sets the **Power** level the laser will use when you press the **Fire** button, or when Framing, if you have enabled Laser on when framing in the **Device Settings** window.

GCode Only — the Move Window During an Active Job

For GCode lasers only, the **Move Window** changes in appearance while LightBurn is actively streaming a project to your laser.

The original interface is replaced by new buttons which allow you to adjust the Speed and Power settings of your active project.



The buttons beneath **Adjust Speed** and **Adjust Power** each adjust those settings as a percentage of the original values — not by absolute values.

As you increase or decrease Speed, Power will automatically adjust to match the change, but you can increase or decrease Power without affecting Speed.

You can click **Reset** to restore either setting to your project's original values.

Adjusting settings through the Move Window only affects the active job, and makes no change to your original project settings.

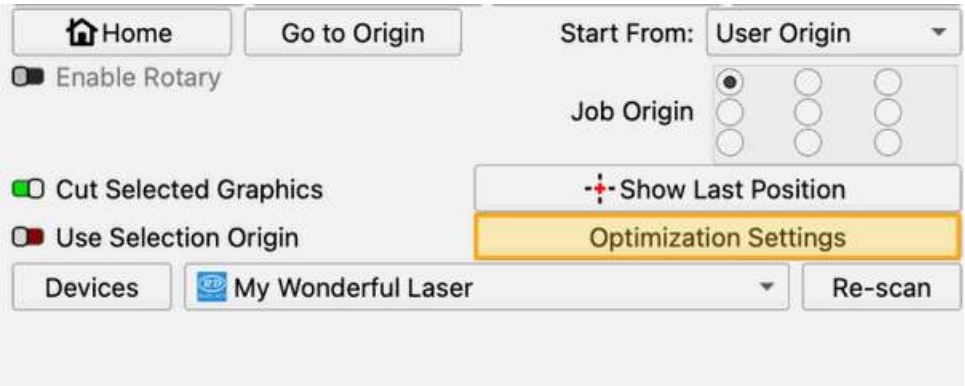
Optimization Settings

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=w1hG1wv8tS4

LightBurn's **Cut Planner** controls the order in which your laser cuts or engraves graphics in your project. The **Cut Optimization Settings** window is where you adjust the variables by which the Cut Planner decides on cut order and pathing.

To open the Cut Optimization Settings window, click the **Optimization Settings** button in the Laser Window.

Optimization Settings only take effect when the Enable Optimization and output re-ordering switch in the Optimization Settings window is toggled on. If Optimize Cut Path is toggled off, graphics are output in the order in which they were first created.

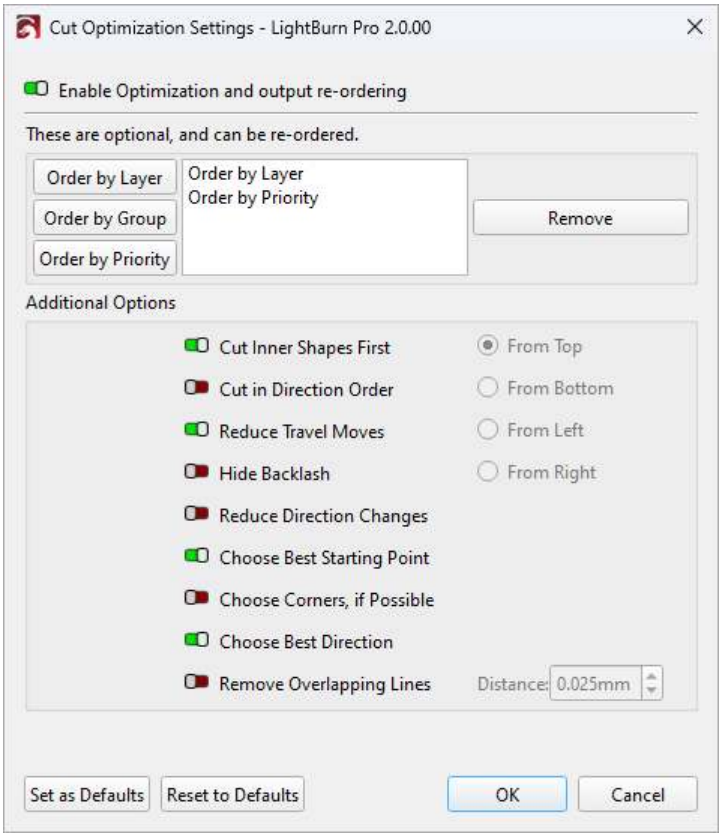


Tip

After adjusting **Optimization Settings**, you can see the resulting path the laser will take, along with a runtime estimate, in the Preview window.

Optimization Settings Window

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Enable Optimization and Output Re-ordering

Toggle this setting on to enable the **Optimization Settings** described below. To disable optimizations without losing your settings, turn the toggle off.

If this switch is disabled, the laser will cut or engrave graphics in the order in which they were drawn.

Order By

The **Order by** options determine the initial ordering parameters for your jobs, which is the primary way in which LightBurn decides the order of cuts. You can choose to Order by Layer, Group, Priority, or combinations of these parameters.

Using the Order by Options

When determining the order of cuts, the **Cut Planner** sorts all objects in your project into ordered lists based on the parameters you've selected. If you select multiple Order by options, their order within the list is important.

After splitting all the objects in your projects into lists based on the first Order by parameter, the Cut Planner then splits each of those lists based on the next Order by parameter in the list, and repeats that process for however many parameters you've selected.

> **Example lists**

With the default settings shown in the screenshot above — **Order by Layer** positioned first in the list, and **Order by Priority** positioned second — the Cut Planner will first split all objects in your projects into lists based on the Layer they are assigned to, and will then sort the objects in each layer based on the Priority you've set for each object.

With 2 layers and 3 objects on each layer, each set to a different Priority, the lists will look like this:

1. Objects on Layer 1
    1. Objects set to Priority 0
    2. Objects set to Priority 1
    3. Objects set to Priority 2
2. Objects on Layer 2
    1. Objects set to Priority 0
    2. Objects set to Priority 1
    3. Objects set to Priority 2

If the positions in the ordered list are reversed so that Order by Priority is first, and Order by Layer is second, the lists will look like this:

1. Objects set to Priority 0
    1. Objects on Layer 1
    2. Objects on Layer 2
2. Objects set to Priority 1
    1. Objects on Layer 1
    2. Objects on Layer 2
3. Objects set to Priority 2
    1. Objects on Layer 1
    2. Objects on Layer 2

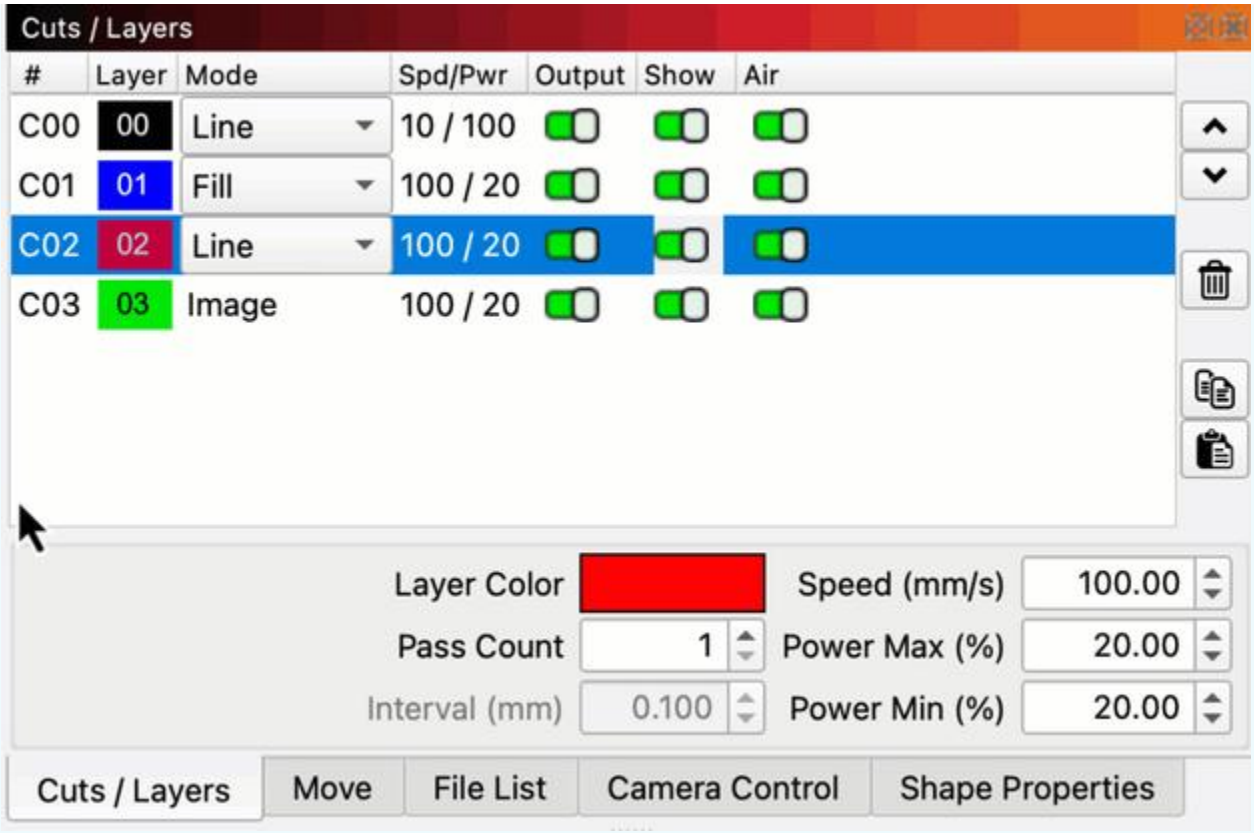
After sorting, each set of shapes created through this process is then optimized with the options in the lower section of the window.

Clicking any of the Order by options adds it to the list of ordering parameters to use. If an Order by option is already on the list, clicking again moves it to the top of the list, making it the first parameter the Cut Planner will sort by.

Order by Layer

Orders cuts according to the position of the layers in the Cuts / Layers Window.

Move layers up and down in the Cuts / Layers Window by selecting and dragging them, or using the buttons to the right of the list



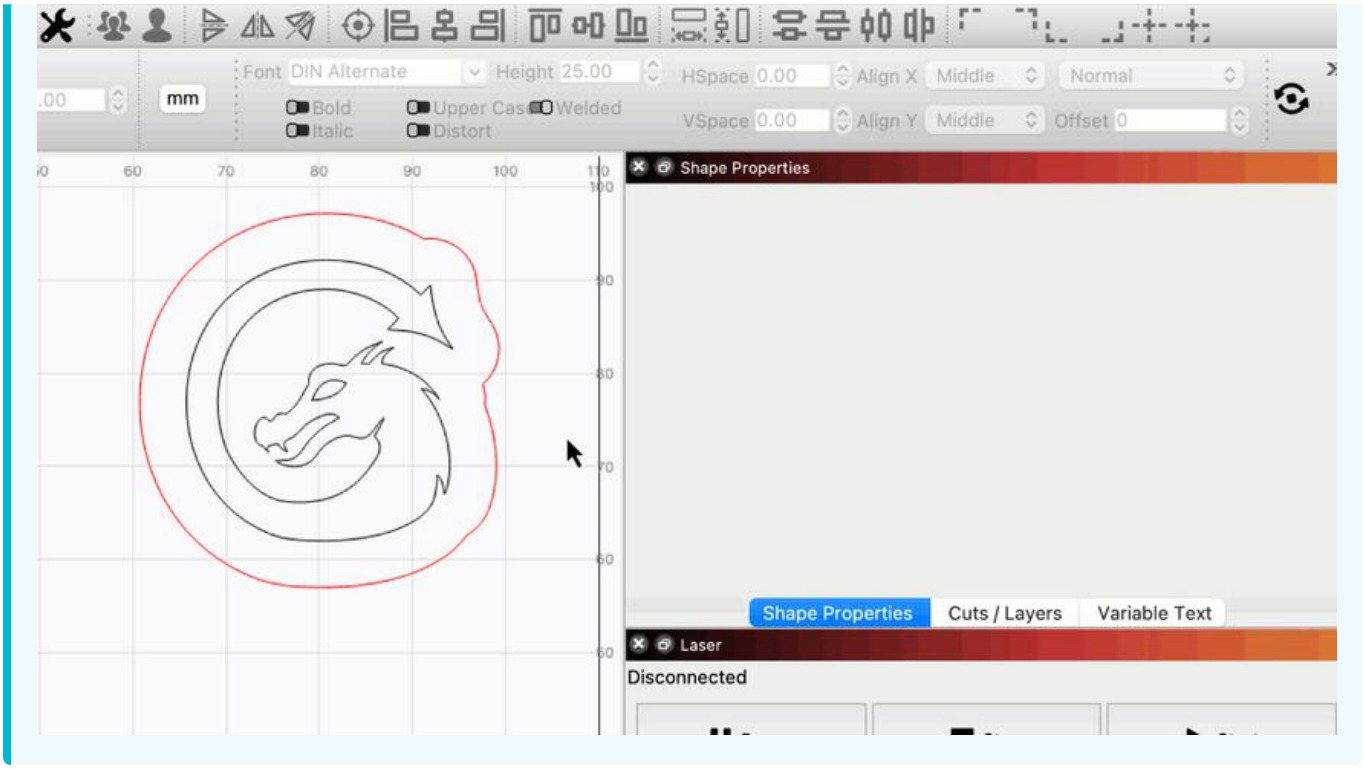
Order by Group

Orders cuts by Group — all objects in a given Group are cut together, then the next Group of objects, and so on. Ungrouped objects will all be treated as a single group for the purposes of this setting.

Order by Priority

Orders cuts according to individual **Cut Order Priority**. Priorities are ordered such that 0 is the first priority, 1 is the second, and so on. Assign Cut Order Priority in the Shape Properties Window.

Select objects and go to the Shape Properties Window to adjust their Cut Order Priority



Remove

Select an **Order by** option from the list and click **Remove** to remove it from the list of ordering parameters.

Optimizations

The following optimizations are applied to all objects sent to your laser, after they are sorted into lists according to the above ordering parameters.

Cut inner shapes first

If there is an object within another object, and both are being cut, tells your laser to cut the inner object before the outer one. The outer shape must be closed in order for this optimization to apply.

> **Tip**
>
> This is often useful to ensure engravings and inner cutouts are completed before cutting a shape out of the stock material, which can cause the shape to drop or shift, depending on how the material is held in place.

Cut in direction order

Attemps to order the shapes in your project so they're cut in specified direction — Top to Bottom, Left to Right, etc.

When the **Cut in direction order** switch is enabled, you can select one of the buttons on the right to determine order.

Reduce travel moves

Attempts to reduce non-cutting travel moves by ordering cuts based on proximity to other shapes.

Hide backlash

This option is similar to the **Backlash Repay Optimize** setting found in RDWorks — it produces a cutting order that reduces or eliminates the misalignment between the start and end points of a cut caused by loose or flexing belts, or other forms of play in the mechanical parts of the laser. Enabling this option will force some of the other options, to give it the most flexibility when planning the cutting path.

Reduce direction changes

Attempts to choose nearby cuts that allow the laser to keep moving in the same direction, which keeps it moving faster.

Choose best starting point

Allows the **Cut Planner** to start a cut at any point within a shape, not just the first point.

> **Tip**
>
> This options works best in combination with Reduce travel moves.

Choose corners, if possible

Attempts to start cuts at sharp corners, to minimize burning or staining on the surface of an item.

Choose best direction

Attempts to choose the direction to cut that will result in the most efficient path.

Remove Overlapping Lines

Tells the **Cut Planner** to remove lines that are overlapping between two graphics, and would cause the laser to cut in the same place twice.

The **Distance** field to the right of the switch controls how close two lines need to be in order for the Cut Planner to remove one of them when sending the job to your laser.

> **Workshop Video: Save Time with Remove Overlapping Lines**
>
> ▶ Watch the video on YouTube: https://www.youtube.com/watch?v=W1rmMIHDGJs

> **Quick Tip Video: Remove Overlapping Lines**
>
> ▶ Watch the video on YouTube: https://www.youtube.com/watch?v=tHTnb6MURdU

Defaults

Set as Defaults

Saves your current **Optimization Settings** as your custom defaults.

Each time you start a new project, your Optimization Settings revert to these defaults.

Reset to Defaults

Restores your current **Optimization Settings** to your default settings.

Hold `Shift` while clicking **Reset to Defaults** to restore to LightBurn's standard default settings (as shown in the screenshot above).

OK

Saves any adjustments you've made in the **Optimization Settings** window, overwriting previous settings.

Any Optimization Settings you save are also stored within `.lbrn` and `.lbrn2` files, and opening any previously saved file will automatically overwrite your current settings with those saved to the file.

Cancel

Discards any adjustments you've made in the **Optimization Settings** window, reverting to previous settings.

Related Topics

- Preview
- Cuts / Layers Window
- Shape Properties
- Grouping

Cut Selected Graphics / Use Selection Origin

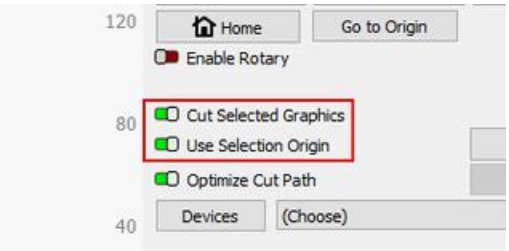
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=Om-6FRoy6Z0

By default, LightBurn outputs all graphics in your Workspace when you Preview, Frame, Start, Send, or save a project in a machine-specific file format. This works well if you have one design in each project, but is less ideal when storing multiple designs within the same file.

In order to output only specific design elements, you can enable **Cut Selected Graphics** instead.

Use Selection Origin allows you to choose whether to orient output relative to the entire project's Origin, or just the Origin of the graphics in your selection.

Both switches are available in the Laser Window.



Note

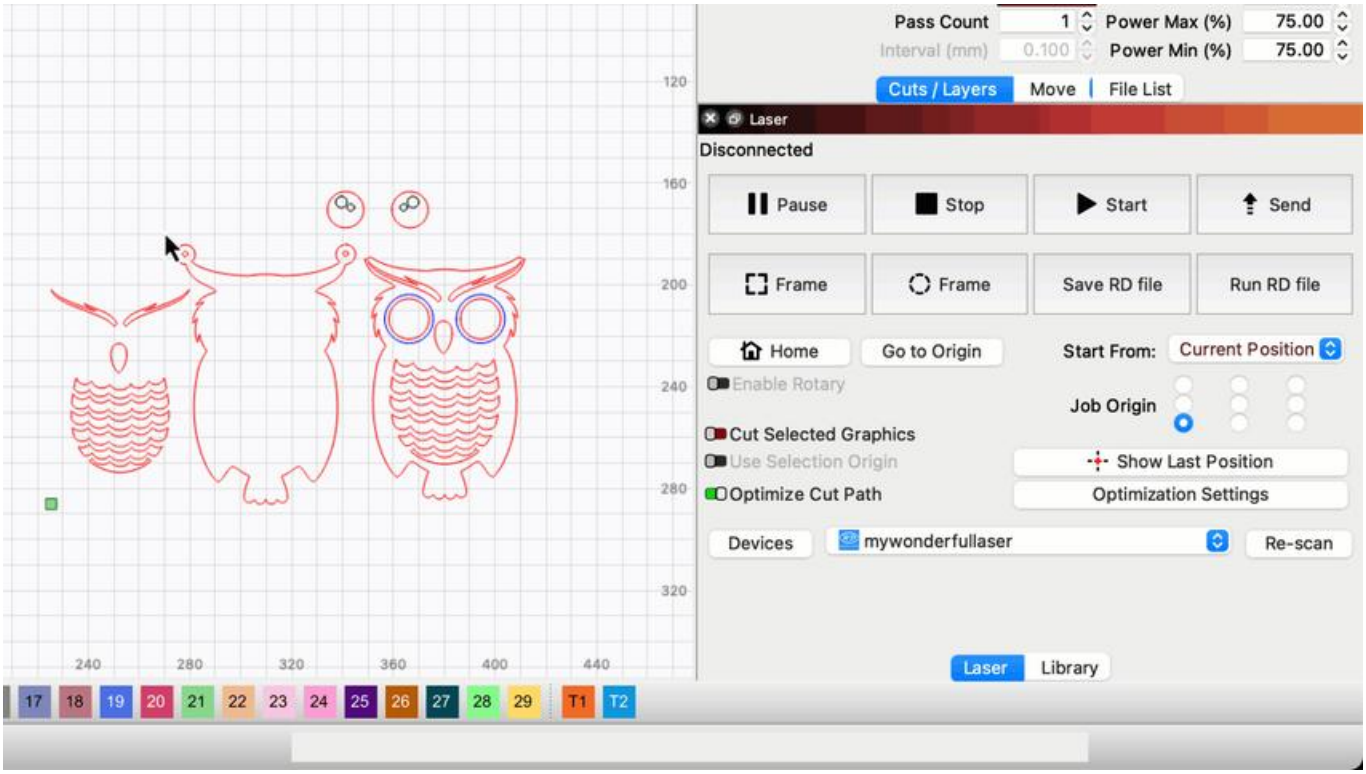
Use Selection Origin will be grayed out if you've selected **Absolute Coords** as your Start From mode, or do not have Cut Selected Graphics enabled.

Cut Selected Graphics

Enable **Cut Selected Graphics** to tell LightBurn to only output the portion of your design that is currently selected.

Tip

The Preview window only shows graphics that will be output to your laser — use it to check that you've selected just the graphics you want.

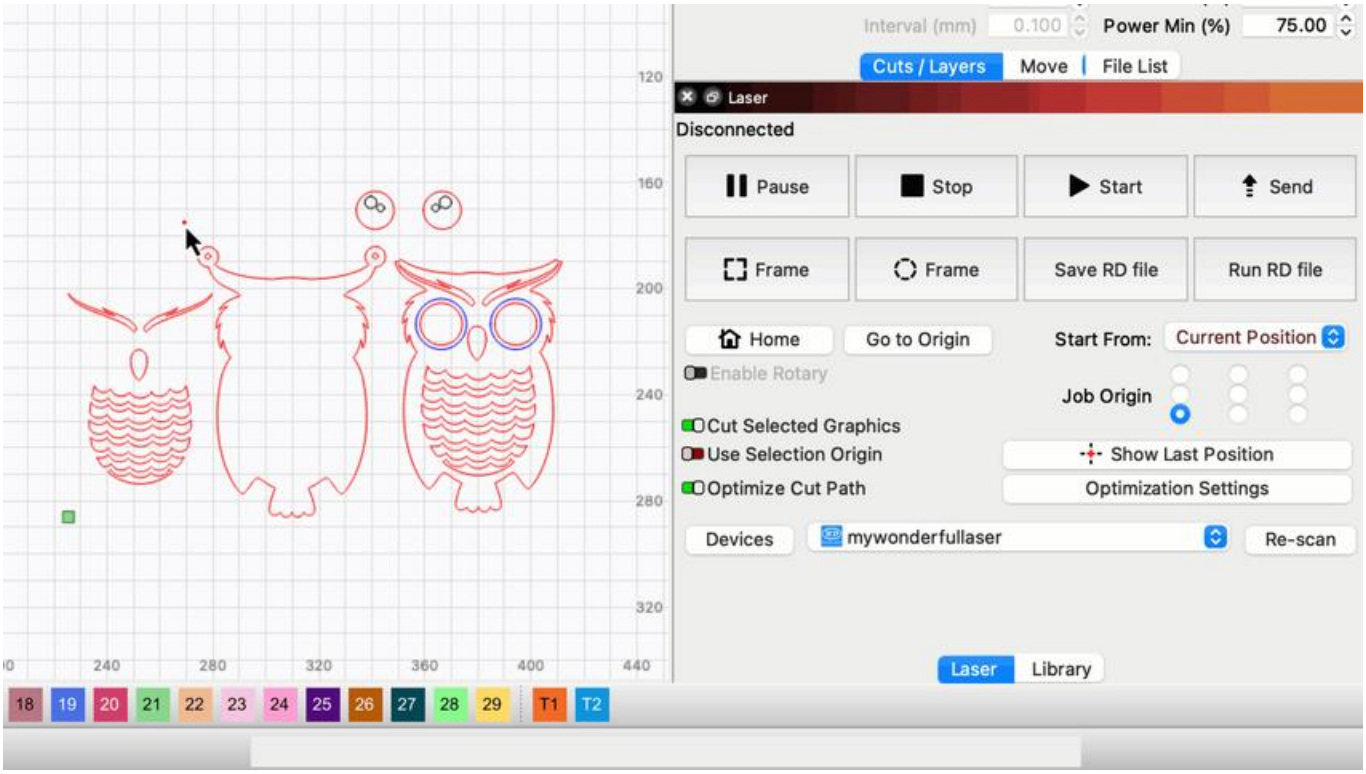


Use Selection Origin

When used in combination with **Cut Selected Graphics**, the **Use Selection Origin** button tells LightBurn to calculate the Job Origin from only the parts that are selected, not the whole design.

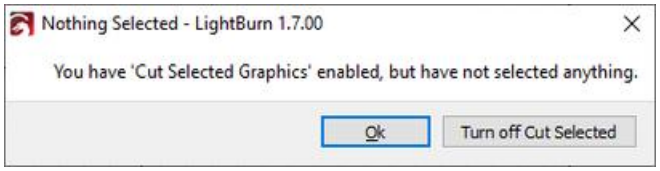
- If you have many different graphics in a project that are intended to be cut completely separately, enabling Use Selection Origin simplifies the process of positioning output to your laser, by excluding unselected graphics when calculating the Origin.
- If you're looking to rerun just one part of a larger design that didn't cut through, disabling Use Selection Origin will ensure that the cut lines up, by using the entire design to calculate the origin.

The Job Origin is indicated on your design as a green square. You'll see its position move as you toggle Cut Selected Graphics and Use Selection Origin on and off in different combinations.



Troubleshooting

If **Cut Selected Graphics** is enabled, but you have nothing in your selection, and attempt to **Preview**, **Start**, **Send**, or **Save** a project in machine-specific format, you'll see a warning message, and be given the option to click **Ok** and abort the process but leave Cut Selected Graphics enabled, or **Turn off Cut Selected** to abort the process while automatically disabling Cut Selected Graphics at the same time.



Related Topics

- Laser Window
- Coordinates and Job Origin
- Preview

Position Laser

The **Position Laser** tool is a quick way to jog your laser to a location in its physical work area, by clicking in your LightBurn Workspace.

In order to use this tool, LightBurn must be directly connected to your laser, and your laser must be properly homed and reporting its position accurately.

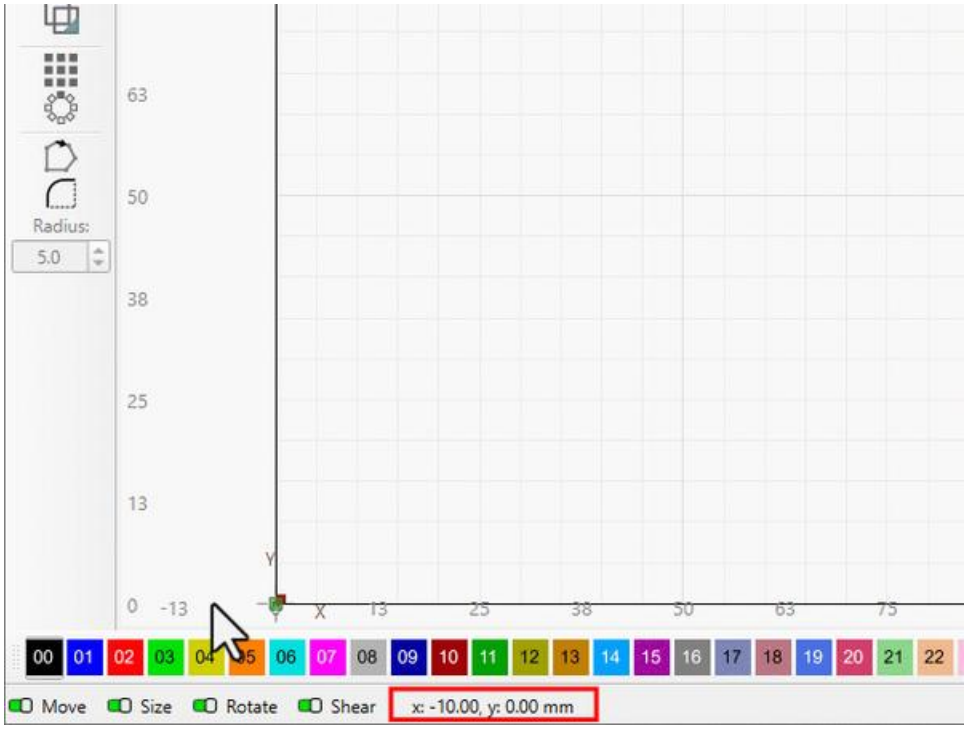
To enable Position Laser, go to Tools → **Position Laser**, click the Position Laser button in the left side toolbar, or press `Alt`/`Option`+`L`.

> **Use With Caution**
>
> If your laser is not properly homed and reporting its position accurately, using this tool may cause your laser to attempt to travel outside its physical bounds, in some cases even crashing into the side rails.
>
> Before using this tool, make sure nothing is in the path of your laser.



Using Position Laser

The grid in your LightBurn Workspace has a coordinate system that matches the physical work area of your laser. The 0,0 position is always the location of your laser's Origin. You can see the exact location of your cursor at any given position in the Status Bar.



After enabling Position Laser, click anywhere in your Workspace, and LightBurn will issue a command to send your laser to the corresponding location in its physical work area.

The speed at which the laser travels when using Position Laser is determined by the value set in the **Speed** field in the Move Window.

Position Laser automatically turns itself off after about 10 seconds, in case you accidentally leave it on.

Related Topics

- Move Laser to Selection
- Move Selected Objects

Set Start Point

Use the **Set Start Point** tool to control where your laser begins cutting a shape, and in which direction.

By default, the Start Point is the first point of the shape, and the direction is the same as the direction of the animated dash pattern that indicates when an object is selected.

Set Start Point is available in the Modifiers Toolbar, located on the left side of the LightBurn window, by default.

Using Set Start Point

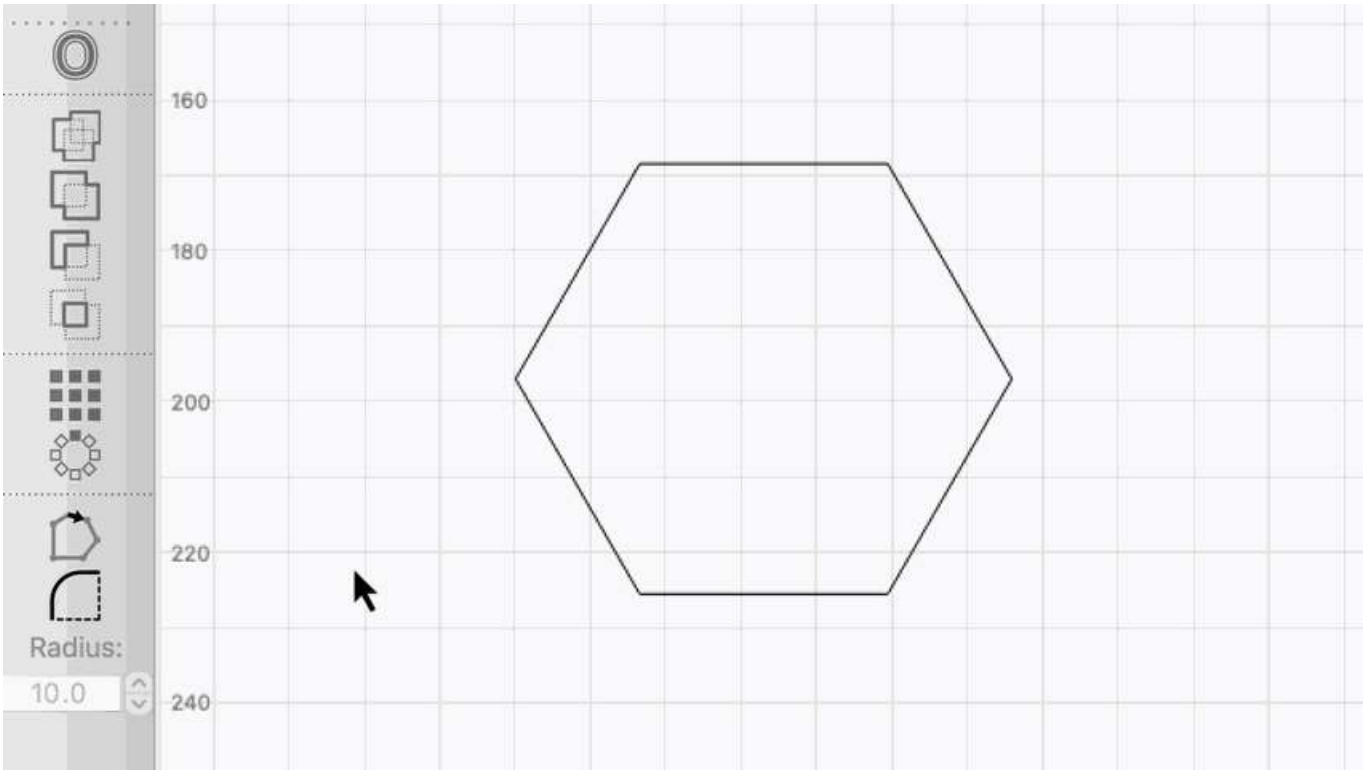
Click the **Set Start Point** icon in the Modifiers Toolbar to activate it.



If you had an object selected before you activated the Set Start Point tool, you'll see its nodes and the current Start Point. To select a new or different object, click any object in your Workspace.

The default Start Point is indicated by a gray arrow, while a user-defined Start Point is indicated by a blue arrow.

- Click any node to move the Start Point to that node.
- Hold `Shift` while clicking to reverse the direction. Click again without holding `Shift` to return it to the default direction.
- Hold `Ctrl`/`Cmd` while clicking any node to revert the Start Point and direction back to the default.
- You can also reverse the direction by going to Edit → **Reverse Shape Direction**.



Additional Notes

- Use the Edit Nodes tool to add a custom node to use as a Start Point, if one does not already exist at the point you'd like a cut to begin.
- Default and user-defined Start Points are overridden by certain Optimization Settings. If you select **Choose best starting point** or **Choose best direction**, LightBurn will select a different Start Point if doing so will reduce overall cutting time.
- The Start Point does not affect the direction of Fill or Image engravings. To change the direction of an engraving, invert the **Scan Angle** from 0° to 180°.

Related Topics

- Edit Nodes
- Line Mode

Galvo Framing

Framing is a function of Galvo lasers that outlines projects in your laser's work area using a red dot light, allowing you to see where your design will output on a physical object.

Click the **Frame** button in the Laser Window or press `F1` to open the **Live Framing** window.

> **Note**
>
> You may need to hold the `Fn` key while pressing `F1` to open the Live Framing window.

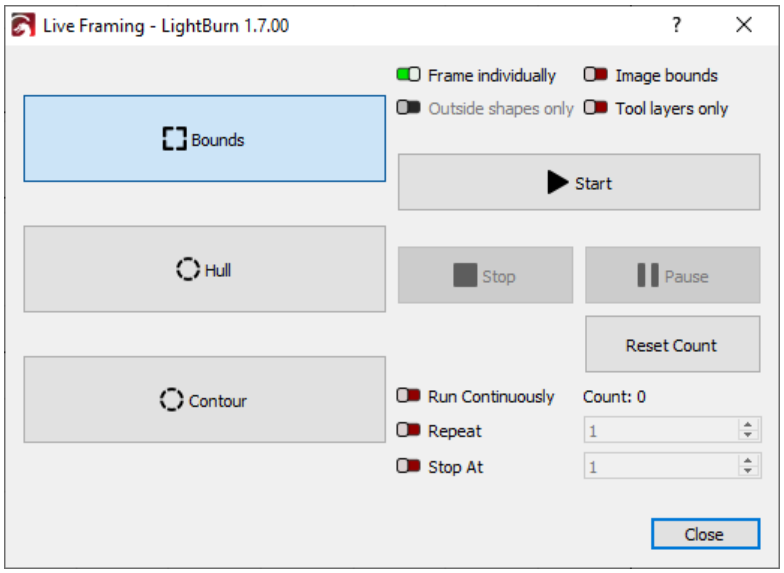
By default, the Live Framing window will also open when you press the **Start** button in the Laser Window, but you can disable **Require framing before start** in your Device Settings.

The Live Framing window contains options that determine the style of framing, which graphics are framed, and how many times to repeat a given project.

You can **Start**, **Stop**, and **Pause** your projects from the Live Framing window, which remains open while your job is running.

While framing, you can nudge, rotate, and rescale graphics, adjusting them in your Workspace and live-updating them in your laser's work area.

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Framing Styles

The different options determine how the red dot light represents your shapes, and which shapes are shown.

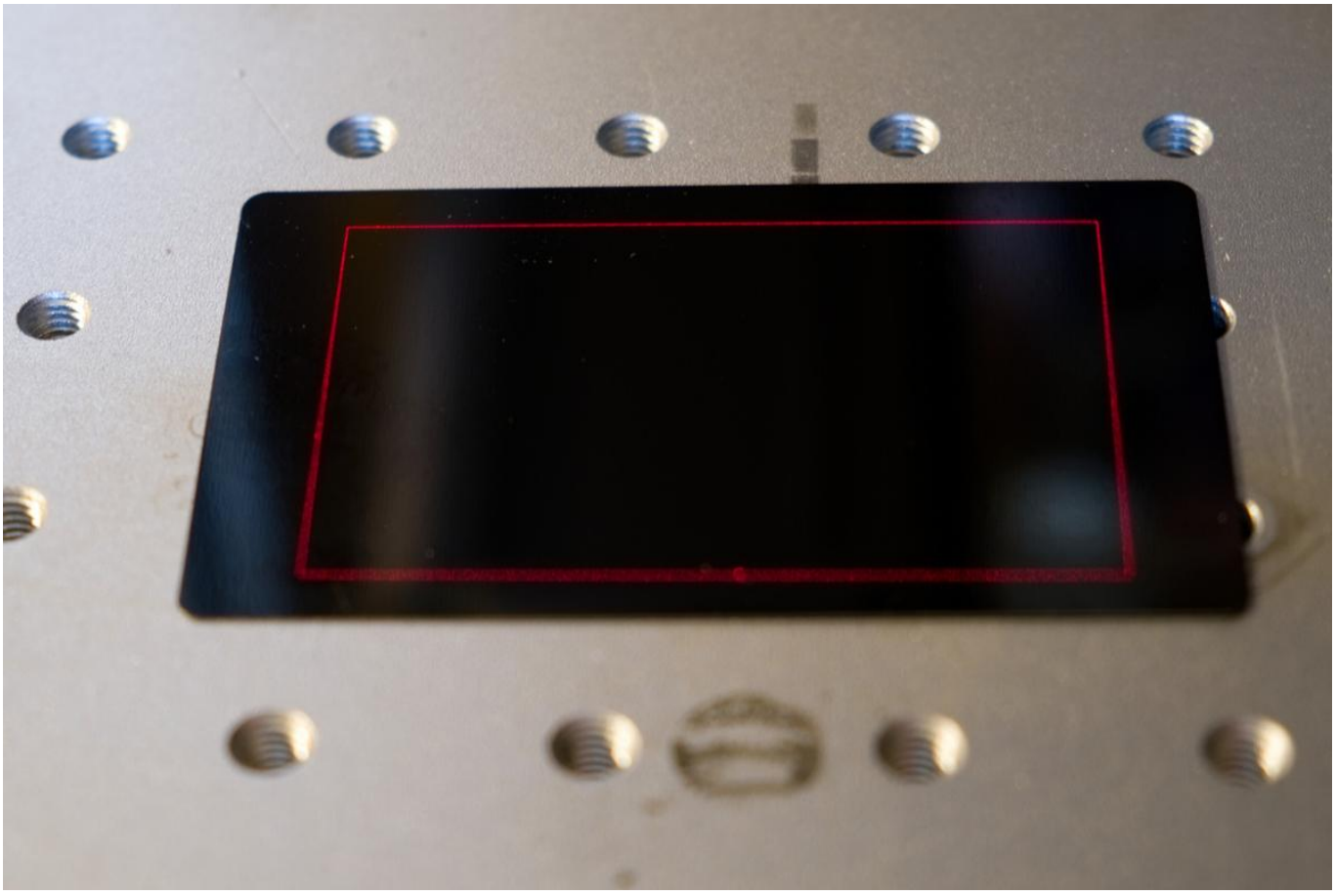
> **Tip**

> Press F1 while framing to cycle through the **Bounds**, **Hull**, and **Contour** framing modes.

Bounds

Commands the red dot light to trace a path defined as the smallest possible rectangle that will fully contain all graphics you're sending to the laser.

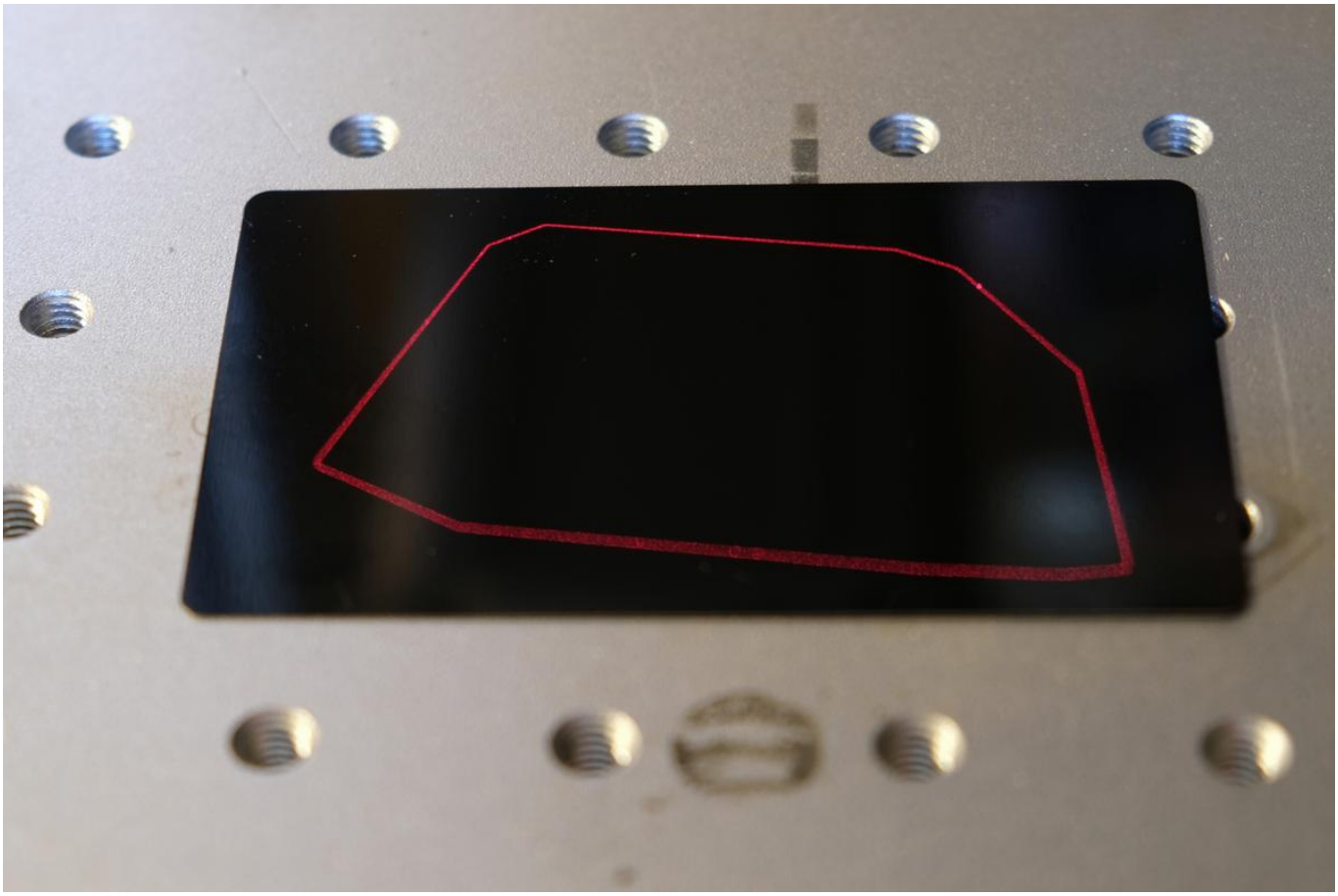
This mode is extremely fast, but doesn't represent complex shapes well.



Hull

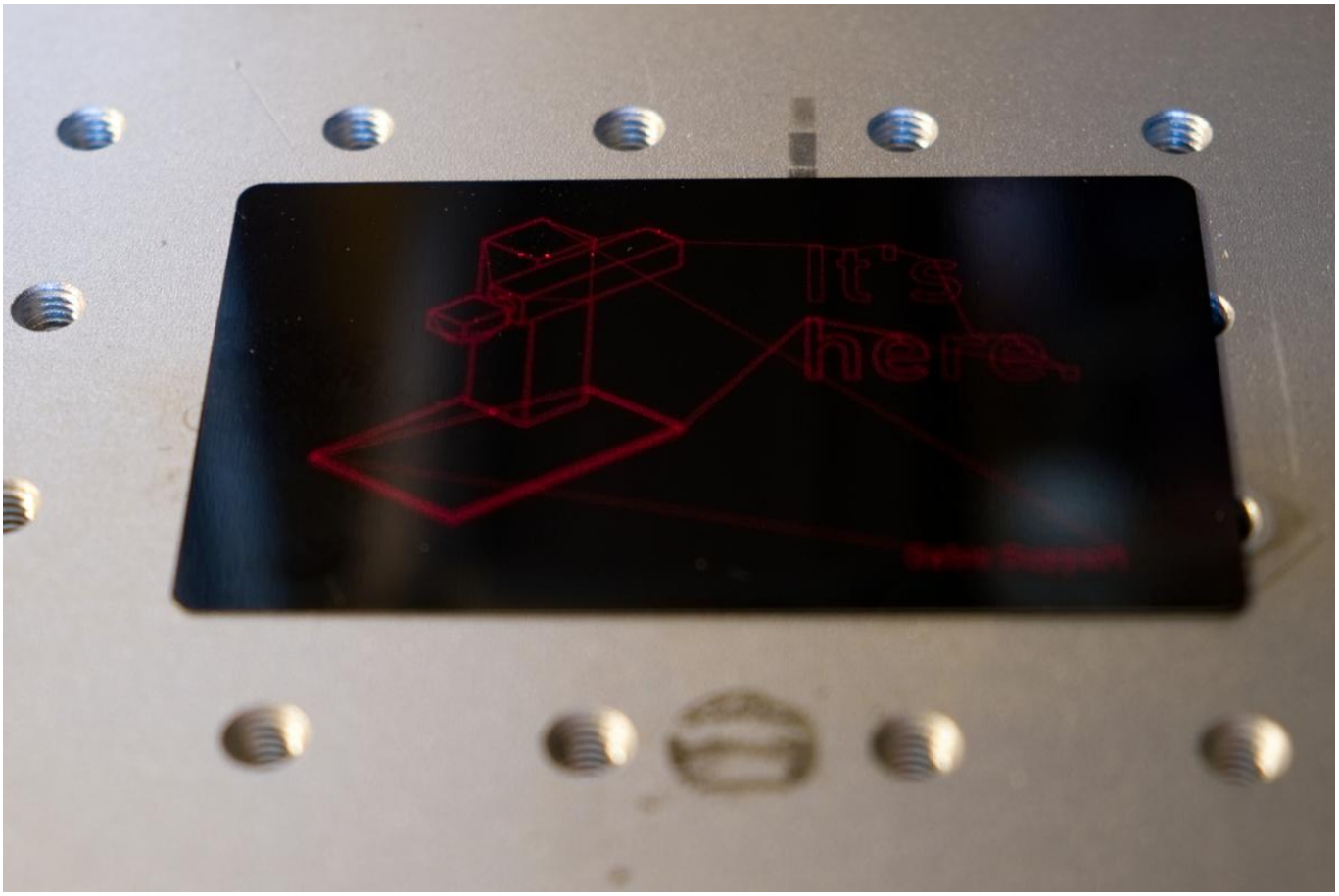
Commands the red dot light to trace the smallest possible path that fully contains all graphics in the design you're sending to the laser, as if a rubber band were stretched around them.

It is nearly as fast as **Bounds**, but gives a much closer fit for rounded or smooth shapes.



Contour

Commands the red dot light to trace a path that follows the exact contours of your shapes — it takes the longest to draw, but it's the most accurate.



Additional Options

Frame Individually

Determines whether to trace a single **Bounds** or **Hull** frame around everything in your project, or around each individual shape.

This option is grayed out when using **Contour** framing.

Image Bounds

Attempts to quickly trace the outlines of any images you have in your design, rather than only showing the rectangular boundary of the entire image.

LightBurn determines the outline based on transparency and whitespace in the image — accuracy may vary.

Outside Shapes Only

Tells the software not to trace inner shapes, like the insides of letters — typically this doesn't affect placement, but allows a faster update of the frame (this only affects **Contour** mode).

Tool Layers Only

Tells LightBurn to only frame shapes that are on one of the Tool layers.

Job Control

Start

Commands your laser to immediately begin running your current project.

Stop

Immediately aborts a currently running job.

Pause

Halts a running job. Click the **Resume** button to continue a paused job.

Job Counter and Multiple Runs

Count

This value is incremented at the end of each successful run of your file. You can use this to keep track of how many parts you have marked, or how many passes of a single job you've run.

Reset Count

Resets the **Count** value to 0.

Run Continuously

Tells LightBurn to re-start the job automatically after it finishes, and update the **Count** after each run.

- Turn the switch off while a job is running to stop the laser when the current run completes.
- Press Stop to immediately halt the laser.

This is useful when you are doing very deep engraving, and are unsure how many passes a job will need.

> **Warning**
>
> Ensure your laser is properly supervised, even during long jobs with multiple passes.

Repeat

Enable this switch and set a value in the adjacent field to tell LightBurn to run the entire job the specified number of times, and then stop.

If you turn off this switch while a job is running, the laser will stop when the current run completes.

> **Tip**
>
> To run multiple passes of specific layers, sub-layers, or combined sequences of sub-layers, see the **Number of Passes** and **Global Passes** settings in the Cut Settings Editor.

Stop At

Enable this switch and set a value in the adjacent field to disable the Start button if the Count is greater than or equal to the specified **Stop At** value.

This option is especially useful when the job is externally triggered (e.g. with a foot switch), as LightBurn will ignore the external trigger once the Count reaches the Stop At value.

Nudging, Rotating, and Rescaling

While framing, you can use the keyboard to nudge, rotate, and rescale your graphics.

Key(s)	Result
`Left` / `Right` / `Up` / `Down`	Move graphics by 5 mm
`Shift` + arrow keys	Move graphics by 20 mm
`Ctrl` + arrow keys	Move graphics by .5 mm
`Page Up`	Scale graphics up
`Page Down`	Scale graphics down
`.` (period)	Rotate graphics 15° clockwise
`,` (comma)	Rotate graphics 15° counter-clockwise
`Shift`+`.` (period)	Rotate graphics 1.5° clockwise
`Shift`+`,` (comma)	Rotate graphics 1.5° counter-clockwise
`Ctrl`+`.` (period)	Rotate graphics 45° clockwise
`Ctrl`+`,` (comma)	Rotate graphics 45° counter-clockwise

If your keyboard doesn't have `Page Up` or `Page Down`, try using `Fn`+`Up` or `Fn`+`Down`.

Troubleshooting

- If the framing location or scale is inaccurate, you may need to adjust your laser's focus, or fine-tune the values found in Device Settings → Galvo and Basic Settings → Red Dot.

  See our video guide for more information on adjusting red dot alignment.

  ▶ Watch the video on YouTube: https://www.youtube.com/watch?v=2Ec4-F49aV0

- If **Contour** framing doesn't work, try toggling off **Enable Blanking Delay**, in Device Settings, under **IO Port Settings**.

- For information on framing behavior when using a rotary, see Rotary Mode (Galvo).

Move Laser to Selection

The **Move Laser to Selection** tools move your laser in its physical work area to a location that corresponds to a position on your current selection in your LightBurn Workspace.

Use these tools to align your laser and physical objects to the positions of graphics in your LightBurn grid.

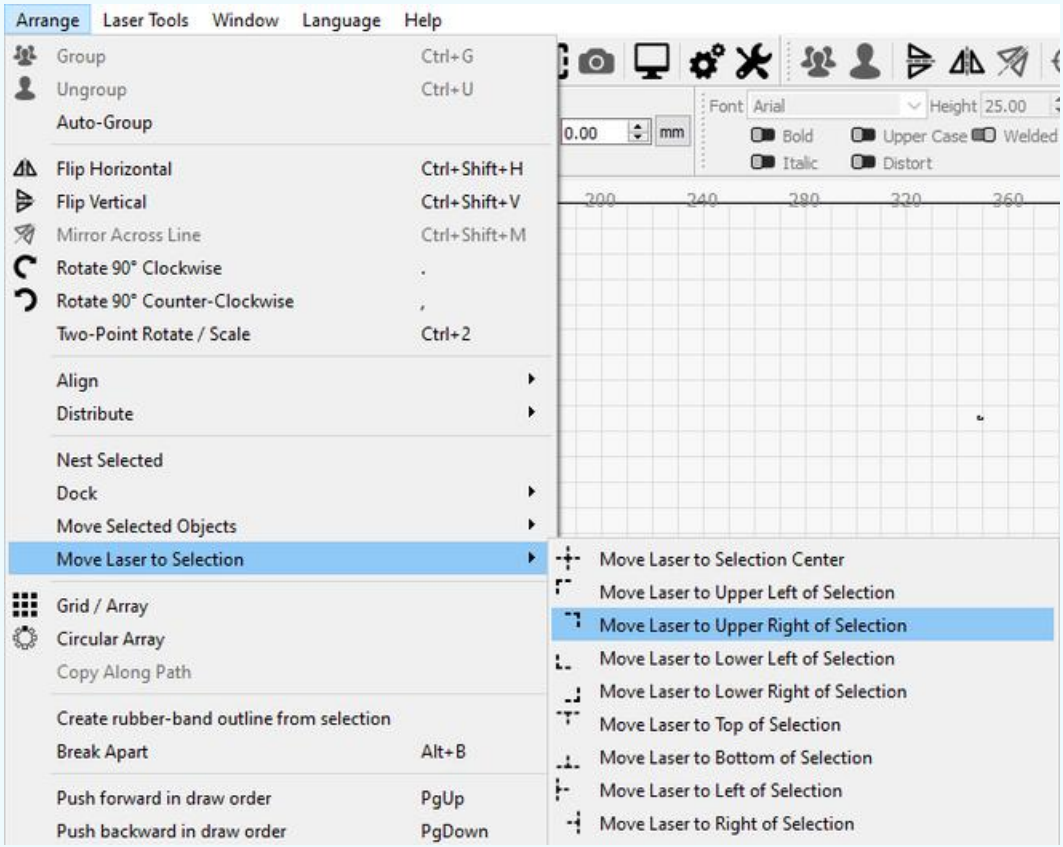
> **Warning**
>
> In order to use **Move Laser to Selection**, LightBurn must be connected to your laser, and it must be capable of reliably homing and reporting its position accurately. If the position reported by the laser is inaccurate, using this feature may cause your laser to attempt to travel outside of its physical bounds.

UI Locations

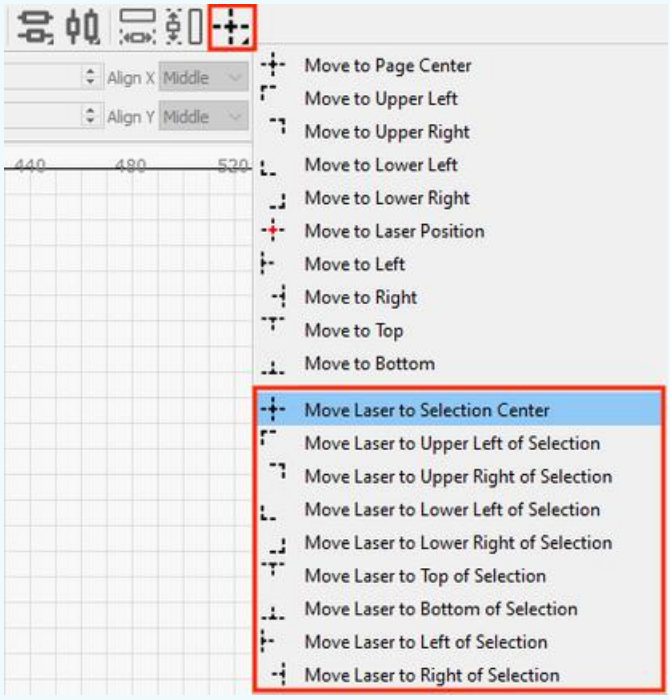
All **Move Laser to Selection** tools are available in the Arrange Menu or in drop down menus in the Arrange Toolbar and **Arrange (long) Toolbars**.

You can also hold the `Ctrl` or `Cmd` key while pressing one of the Move Selected Objects buttons to move the laser head to the location on the selection, rather than moving the objects.

Move Laser to Selection in the Arrange Menu

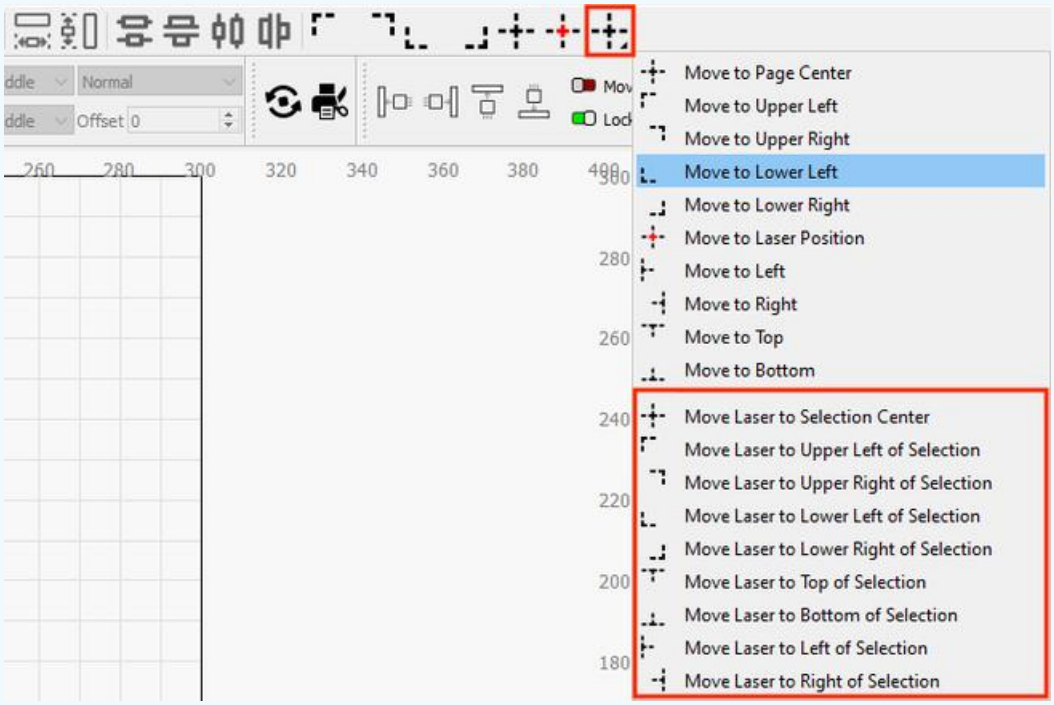


Move Laser to Selection in the Arrange Toolbar



Move Laser to Selection in the Arrange (long) Toolbar

Hold `Ctrl`/`Cmd` while clicking the Move Selected Objects buttons in the Arrange (long) Toolbar to move the laser to the chosen location on your selection instead, or open the dropdown menu to select any option.



Options and Icons

All below options move the laser to the location in its physical work area that corresponds with the location of the chosen point on your current selection in your LightBurn Workspace.

The grid in your LightBurn Workspace has a coordinate system that matches the physical work area of your laser. The 0,0 position is always the location of your laser's Origin.

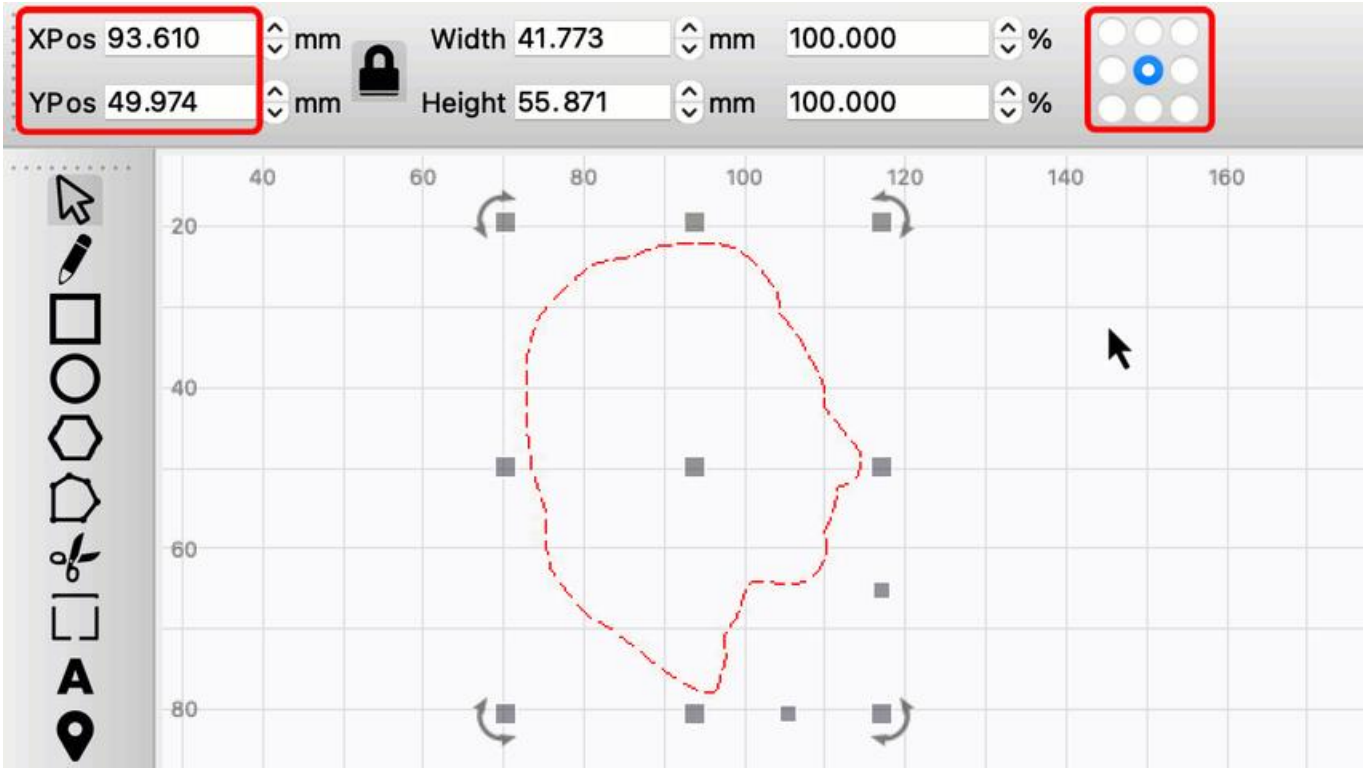
For example, if the bottom right corner of your selection is at 100 mm in the horizontal (**X**) dimension, and 200 mm in the vertical (**Y**) dimension in your LightBurn Workspace, selecting **Move Laser to Lower Right of Selection** will move the laser head to X: 100 and Y: 200 in its physical work area.

Option	Icon	Action
Move Laser to Selection Center		Moves laser to the center of selected objects.
Move Laser to Upper Left of Selection		Moves laser to the top left corner of selected objects.
Move Laser to Upper Right of Selection		Moves laser to the top right corner of selected objects.
Move Laser to Lower Left of Selection		Moves laser to the bottom left corner of selected objects.
Move Laser to Lower Right of Selection		Moves laser to the bottom right corner of selected objects.
Move Laser to Left of Selection		Moves laser to the left-center of selected objects.
Move Laser to Right of Selection		Moves laser to the right-center of selected objects.
Move Laser to Top of Selection		Moves laser to the top-center of selected objects
Move Laser to Bottom of Selection		Moves laser to the bottom-center of selected objects.

Selection Coordinates

Edges and centers of selections are defined according to the farthest points of any object in the selection area, in each direction — imagine the selection area as the smallest possible rectangle that can contain every object in your selection. The edges and center of a selection are the same as the edges and center of that imaginary rectangle.

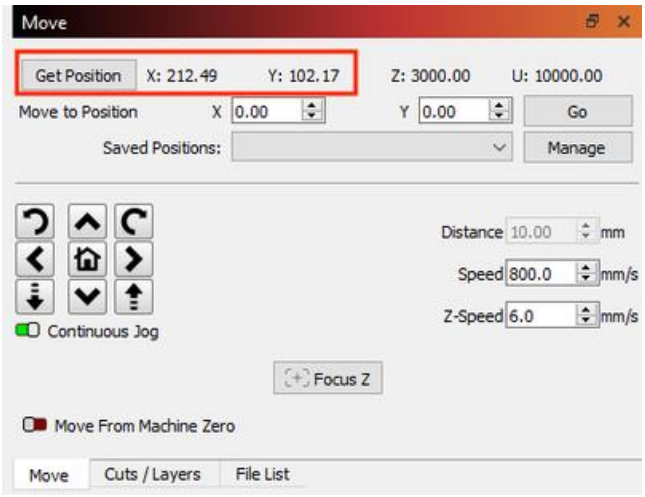
You can read the coordinates of any location on your selection in the Numeric Edits Toolbar. Click a point in the 9-dot control to see the horizontal position (**XPos**) and vertical position (**YPos**) of that location on your selection, in your Workspace.



Laser Coordinates

You can read your laser's current cooridinates in the Move Window, by clicking the **Get Position** button.

> **Note**
>
> The **Move Laser to Selection** tools can only move your laser in the **X** and **Y** dimensions.



Related Topics

- Move Selected Objects

- Position Laser
- Selection Tools
- Coordinates and Origin
- Move Window
- Numeric Edits Toolbar

Quality Optimization

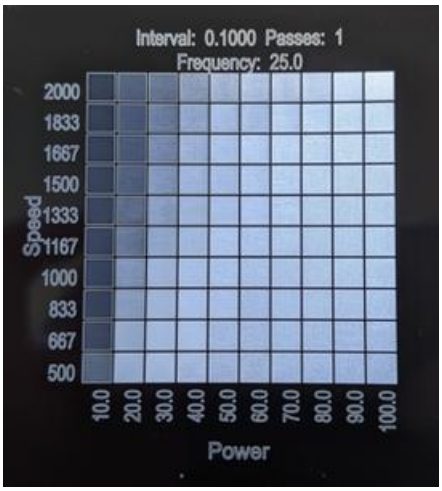
Material Test

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=pTx0RG_4Wbs

LightBurn has a built-in test pattern generator to help you find the best settings for your laser and any material you're working with.

To open the **Material Test Generator**, go to the Laser Tools Menu and select **Material Test**.

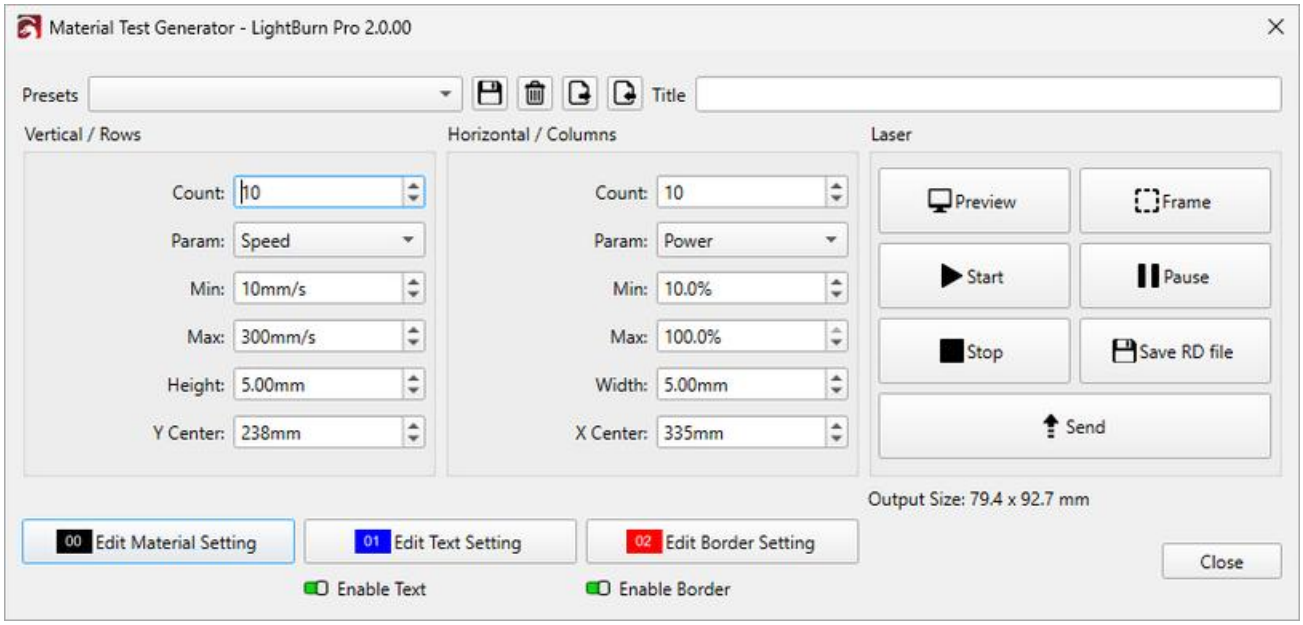
Material Test Example



A material test showing different speeds and powers on black anodized aluminum

On the left side, each row is labeled with the engraving speed. Along the bottom, columns are labeled with the power level used. At the top, the sample shows additional universal settings — in this case, **Interval**, **Passes**, and **Frequency** — that are used for each box in the grid.

Using The Material Test Generator



The Material Test dialog as it appears for a DSP laser

After opening the **Material Test Generator**, you'll be able to configure your test pattern. By default, the Material Test Generator will create a 10x10 grid of boxes with varying **Power** and **Speed**. Use the **Param** dropdown to select different parameters to test.

Positioning

The output location of the test grid is determined by the **Start From** mode you select before opening the Material Test Generator. Current Position and User Origin work exactly as they do for all other output, while Absolute Coordinates location is controlled by the **X/Y Center** options in the Material Test configuration.

Test Configuration

Use the settings below to configure your test grid. The parameters tested can be **Power**, **Speed**, **Interval**, or **Passes**. **Frequency** and Q-Pulse width can also be tested, if your laser supports those settings.

Test Grid Settings	Explanation
Count	How many rectangles are in each row/column
Param	The setting being tested along this axis
Min	The lowest value to be used for this setting
Max	The highest value to be used for this setting
Height/Width	The height/width of each rectangle in the test grid
Y Center/X Center	The position of the center of the test grid (if using Absolute Coordinates)

Cut Settings

The three buttons at the bottom of the dialog box open the Cut Settings Editor, where you can adjust the settings to use in your test. There are two switches to enable or disable the text labels and border.

Button	Explanation
Edit Material Setting	Base settings for the test — the two settings chosen in **Param** will be varied in the grid, but all other settings applied here will apply to every test box
Edit Text Setting	Settings to use for the text labels
Enable Text	This switch controls whether text labels are output
Edit Border Setting	Settings to use for the border around the test grid — available in LightBurn 1.5 and higher
Enable Border	This switch controls whether the border around the test grid is output

Always pay attention to units of distance and time when entering Speed settings

When entering **Speed** values recommended by your laser's manufacturer or other LightBurn users, make sure to use the same units of distance and time as those from the recommendation, or to convert the values to your preferred units.

A given number of **millimeters per second** is much faster than that same number in **millimeters per minute**. Mixing up units can lead to reduction in power output due to unexpectedly high speeds, or excessive power output — and even fire — due to unexpectedly slow speeds.

Change your Control Units in the **Dimensions / Units** tab of the **Device Settings** window. LightBurn automatically converts any existing values when you switch between units.

Running the Test

The buttons on the right side allow you to Preview the test grid, Frame to check output location, run the laser, and more.

Laser Controls	Explanation
Preview	Generate a preview of the test grid
Frame	Frame the test grid to see its position on the material
Start	Start running the material test
Pause	Pause the material test
Stop	Stop running the material test
Save RD File/Save GCode	Device-dependent — if available, saves the test grid in RD or GCode format
Send	Device-dependent — if available, sends the material test to the laser

Material Test Cutting and Engraving Order

The order in which test boxes are cut or engraved is determined according to an ascending list of potential risk of burning or charring material.

In other words, boxes with the highest **Speed**, lowest **Power**, lowest **Interval**, and fewest **Passes** will run first, followed by the next highest, lowest, fewest, and so on.

Saving and Exporting

There is also a dropdown allowing you to select a saved **Preset**. This is useful if you have custom settings you like to run frequently, such as a smaller test pattern you want to use on metal tags, or a range of settings you want to test on various woods.

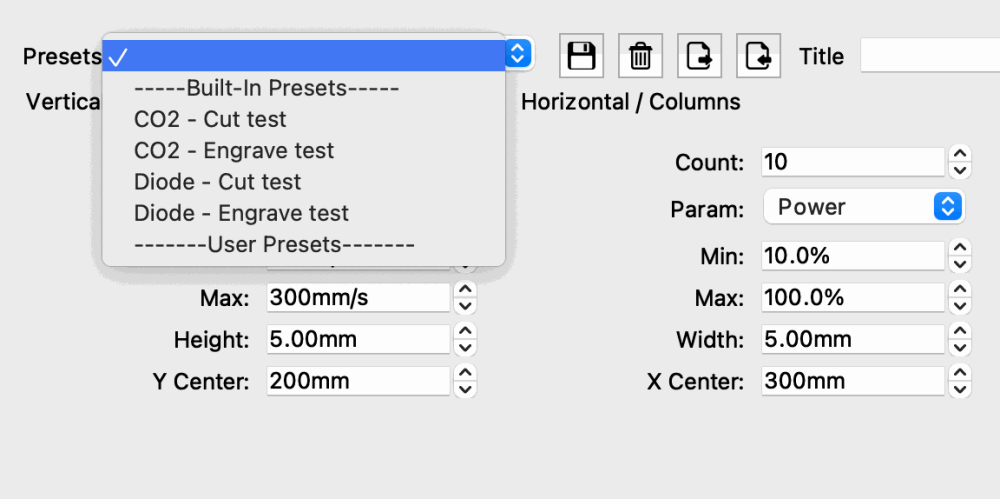
At the top are buttons allowing you to save, delete, export, and import presets.

To locate the presets saved on your computer, go to **File → Preferences → Open Prefs Folder**. You'll find the `material_test_presets.lbmt` file inside the `presets` folder.

Presets can also be exported in a User Bundle.

Built-in Presets

LightBurn provides four **Built-in Presets** for users with Diode or CO_2 lasers. All four are selectable from the dropdown menu in the **Material Test Generator** dialog window.



▶ Watch the video on YouTube: https://www.youtube.com/watch?v=TfJ5fI2J-LI

Related Topics

- Material Library
- Interval Test
- Focus Test

Focus Test

> **Tip**
>
> You can also use the Focus Test to try out different amounts of defocusing. Defocusing your laser produces thicker lines where it marks the material, which in turn allows you to increase the Line Interval setting on Fill layers, and speed up jobs.

The **Focus Test** generates and runs a test pattern to help you identify the optimal focal height for your laser.

Focal height affects the strength and size of the laser dot where it meets the material. A laser that is perfectly focused will have the strongest power output and smallest dot size possible.

The Focus Test pattern is made up of a user-specified number of lines, each engraved at a different Z Axis height. Text is engraved beneath each line to indicate the Z height the line was engraved at.

Go to Laser Tools → **Focus Test** to open the Focus Test setup window, where you set the parameters used to generate the test pattern.

Preparation

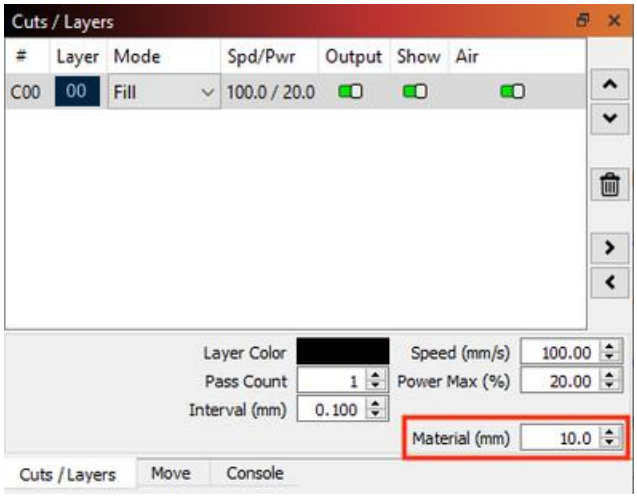
Z Axis Control

In order to use the **Focus Test**, your laser must have a motorized Z Axis that is controllable through LightBurn, and Z Axis control must be enabled in your Device Settings.

You'll receive a warning message if Z Axis control is not enabled in your Device Settings, and you attempt to open the Focus Test.



If you are not using **Relative Z moves**, you must set the **Material** height for the material you will be testing on in the Cuts / Layers Window, before opening the Focus Test. You'll need at least one object in your Workspace in order to adjust the Material height — any objects in your Workspace will be ignored when the Focus Test pattern is sent to your laser.



Material

You'll be engraving a test pattern onto a piece of material placed in your laser's bed. It's best to have the material loaded into your laser before opening the **Focus Test**, so you can set and check positioning.

Your laser's optimal focus doesn't depend on the type of material you're engraving on, but if you're purposely testing a range of defocused heights, you may need to run unique tests on each different material you work with to get the most accurate results.

Positioning

The **Focus Test** pattern will be output in your laser's work area according to the **Start From** mode and **Job Origin** you select before opening the Focus Test.

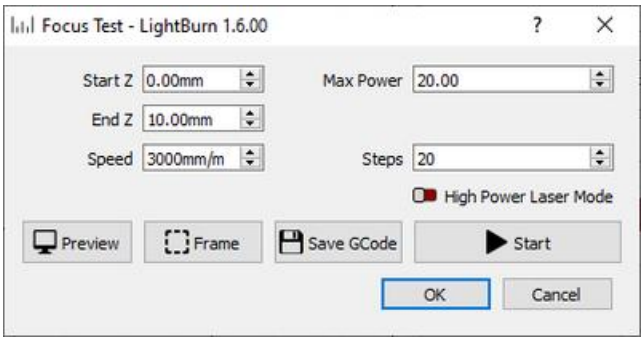
If you select **Absolute Coords**, the Focus Test pattern will be centered in your laser's work area.

Current Position and **User Origin** work exactly as they do for all other output.

See Coordinates and Job Origin for more information on these settings.

Focus Test Setup Window

GCode



- **Start Z** determines the Z height of the first line in the **Focus Test** pattern.
- **End Z** determines the height of the final line in the test pattern.

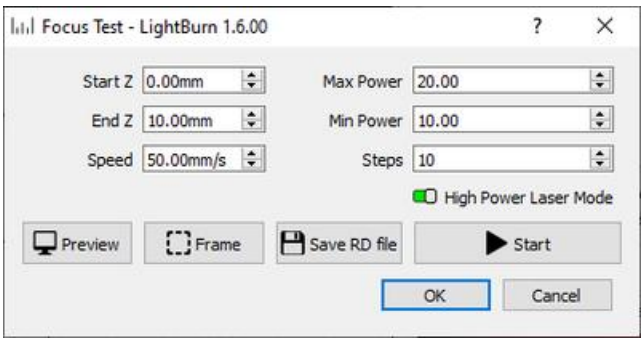
> **Note**
>
> If you are using **Relative Z moves only**, Start Z, End Z, and all Z heights tested in between are determined relative to the current Z position of your laser, at the time the test begins. In other words, the first line will be engraved at the laser's current Z position plus the value you enter in the Start Z field, and so on for all Z heights tested, until the End Z.
>
> If you are not using Relative Z moves only, Start Z, End Z, and all Z heights tested in between are determined relative to the **Material** height you enter in the Cuts / Layers window before opening the **Focus Test**.

- **Steps** determines the total number of lines to create in the test pattern. The total number of lines is equal to Steps + 1. The amount that Z height will advance between each step is equal to: (End Z - Start Z) / Steps.
- **Speed** sets the speed at which your laser will engrave the lines.
- **Max Power** sets the power at which your laser will engrave the lines.

> **Tip**
>
> Set Max Power just high enough to mark your material — if the power is set too high, the laser may char or cut through your material, making it challenging to determine which Z height was the optimal focal height.

- Turn on **High Power Laser Mode** to enable Perforation Mode for the labels beneath each line, if they are overburning. Perforation Mode switches the laser off and on during cuts, reducing overall power output.
- Preview shows a preview of the Focus Test pattern, as it will output to your laser.
- Frame commands your laser to travel around the area the Focus Test pattern will be output to in your laser's work area.
- **Save GCode** saves the test pattern in GCode format.
- **Start** commands your laser to begin engraving the Focus Test pattern.
- Press **OK** to save your Focus Test settings — saved settings will be present when you open the Focus Test later.
- Press **Cancel** to discard any changes you've made to the Focus Test settings — settings will revert to previously saved settings when you open the Focus Test later.

DSP



- **Start Z** determines the Z height of the first line in the **Focus Test** pattern.
- **End Z** determines the height of the final line in the test pattern.

> **Note**
>
> If you are using **Relative Z moves only**, Start Z, End Z, and all Z heights tested in between are determined relative to the current Z position of your laser, at the time the test begins. In other words, the first line will be engraved at the laser's current Z position plus the value you enter in the Start Z field, and so on for all Z heights tested, until the End Z.
>
> If you are not using Relative Z moves only, Start Z, End Z, and all Z heights tested in between are determined relative to the **Material** height you enter in the Cuts / Layers window before opening the **Focus Test**.

- **Steps** determines the total number of lines to create in the test pattern. The total number of lines is equal to Steps + 1. The amount that Z height will advance between each step is equal to: (End Z - Start Z) / Steps.
- **Speed** sets the speed at which your laser will engrave the lines.
- **Max Power** sets the power at which your laser will engrave the lines, once it reaches the Speed you've set.

> **Tip**
>
> Set Max Power just high enough to mark your material — if the power is set too high, the laser may char or cut through your material, making it challenging to determine which Z height was the optimal focal height.

- **Min Power** sets the power at which your laser will begin engraving the lines. As the laser accelerates, power will increase until the laser reaches maximum Speed, and Max Power.
- Turn on **High Power Laser Mode** to enable Perforation Mode for the labels beneath each line, if they are overburning. Perforation Mode switches the laser off and on during cuts, reducing overall power output.
- Preview shows a preview of the Focus Test pattern, as it will output to your laser.
- Frame commands your laser to travel around the area the Focus Test pattern will be output to in your laser's work area.
- **Save RD file** saves the test pattern in RD file format.

- **Start** commands your laser to begin engraving the Focus Test pattern.
- Press **OK** to save your Focus Test settings — saved settings will be present when you open the Focus Test later.
- Press **Cancel** to discard any changes you've made to the Focus Test settings — settings will revert to previously saved settings when you open the Focus Test later.

Interpreting Results

After engraving the test pattern, inspect the results.

The thinnest line in the engraving is the one that was run at the Z height closest to your laser's optimal focal height.

To dial in the optimal focal height further, you can re-run the **Focus Test** — select the two lines that look closest to ideal, and set **Start Z** and **End Z** to match their Z heights.

Related Topics

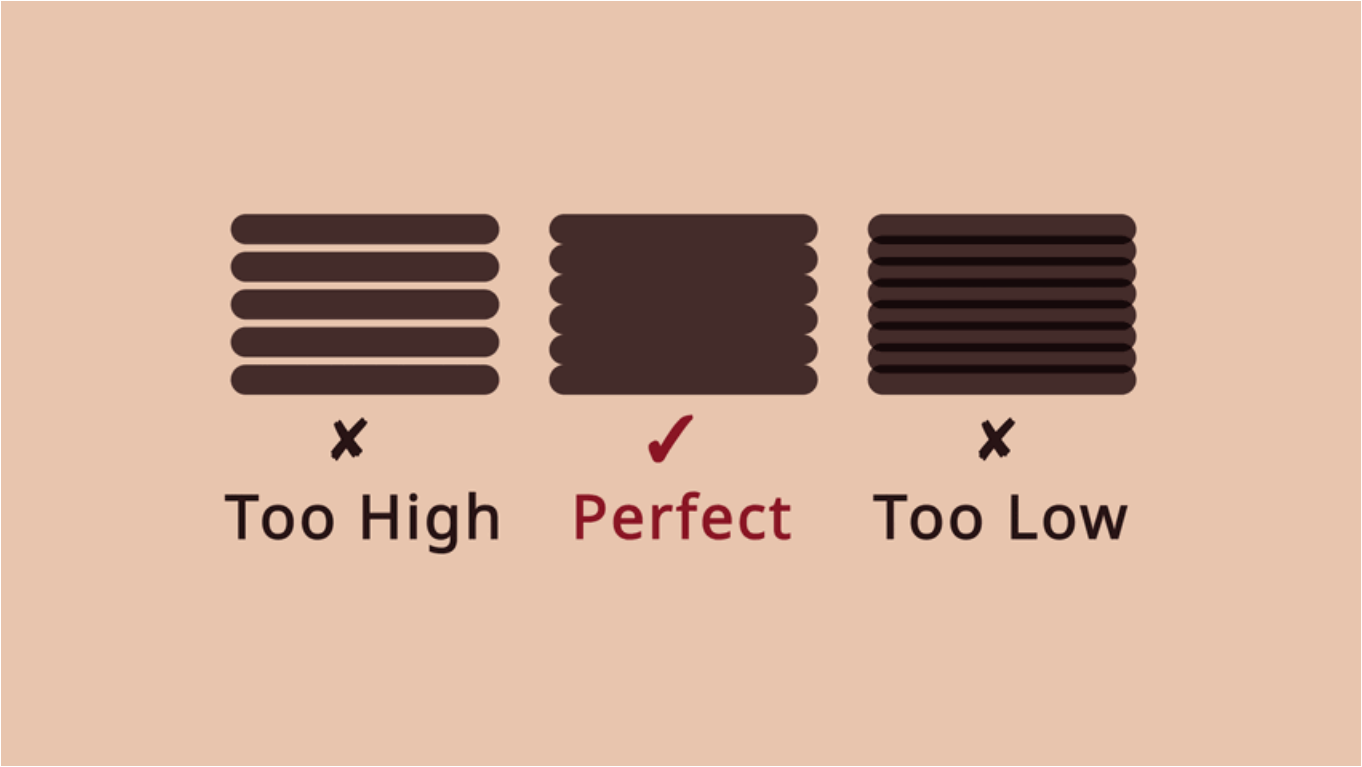
- Material Test
- Interval Test
- Z Axis control
- Fill Mode - Interval

Interval Test

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=kpMunzp2NM8

The **Interval Test** provides a quick way to determine the optimal **Line Interval** for a given combination of machine, material, and focal length.

The Interval Test pattern is made up of a user-specified number of sample squares, each engraved at a different Line Interval. Text is engraved next to each square to indicate the Line Interval it was engraved at.



Line Interval is the distance between scan lines when raster engraving, making it one of the key settings determining the quality of engravings on Image and Fill Layers. It is mathematically tied to the **Lines per Inch** value.

Both Line Interval and Lines per Inch are set in the Cut Settings Editor.

Unlike in traditional print media, a higher density of scan lines (lower interval) doesn't necessarily produce a higher quality image when laser-engraving — an optimal value must be found.

If Line Interval is set too low, the engraving time is vastly increased, and the images appear dark and muddy.

If the Line Interval is set too high, more power is required to achieve the same depth and darkness, and gaps may appear between the lines. Stepping may also appear on curved edges.

Using the Line Interval Test

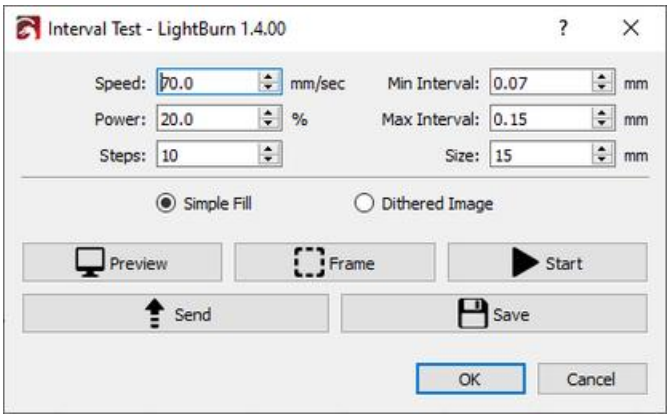
Preparation

Place a scrap piece of the material to be tested into the laser, and focus the laser as per your manufacturer's instructions.

The location of the test on your material is determined by the **Start From** mode and the **Job Origin** that you select before opening the Interval Test. If you select **Absolute Coords**, the test will be centered on your laser's bed. **Current Position** and **User Origin** work exactly as they do for all other output. See Coordinates and Job Origin for more information on these settings.

Interval Test Setup Window

Open the **Interval Test** by navigating to Laser Tools → **Interval Test**, and adjust the settings:



- **Speed** sets the maximum speed the laser head will travel. The Material Test can help you find the optimal speed.
- **Power** sets the maximum power output the laser source should deliver. The Material Test can help you find the optimal power.
- **Steps** determines the number of sample squares to make. Reducing this will reduce the material needed, but will also result in larger differences between each test square.
- **Min Interval** determines the minimum distance between lines to test (try 0.08 mm as a start).
- **Max Interval** determines the maximum distance between lines to test (try 0.16 mm as a start).
- **Size** sets the size of each sample square. Reducing this will reduce the material required, but may make the results harder to read.
- Select **Simple Fill** to created sample squares with a solid black fill.
- Select **Dithered Image** to create sample squares that are a gradient from black to white. These are often easier to read and provide more information to those seeking to engrave images.
- Preview shows a preview of the Interval Test pattern, as it will output to your laser.
- Frame commands your laser to frame the area the Interval Test pattern will be output to in your laser's work area.
- **Save** saves the test pattern in the appropriate file format for your type of laser.
- **Start** commands your laser to begin engraving the Interval Test pattern.

- Press **OK** to save your Interval Test settings — saved settings will be present when you open the Interval Test later.
- Press **Cancel** to discard any changes you've made to the Interval Test settings — settings will revert to previously saved settings when you open the Interval Test later.

Interpreting Results

Results are best observed under good lighting and with magnification, if available. Look for the sample square where the scan lines touch without overlapping.

Adjust your **Line Interval** setting in the Cut Settings Editor, and save your settings to the Material Library for future use.

Related Topics

- Focus Test
- Material Test

Material Library

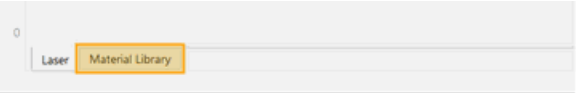
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=jt8M2cJlkH4

LightBurn's **Material Library** provides a place to store, organize, and quickly re-apply preset Cut Settings for different operations and materials.

Accessing the Material Libary

In the default layout, the **Library Window** is the tab after Laser, in the bottom right of the screen.

If you have closed the Library Window, go to Window → **Library** to re-enable it. To restore it and all other windows to their default positions, go to **Window** → Reset to Default Layout.



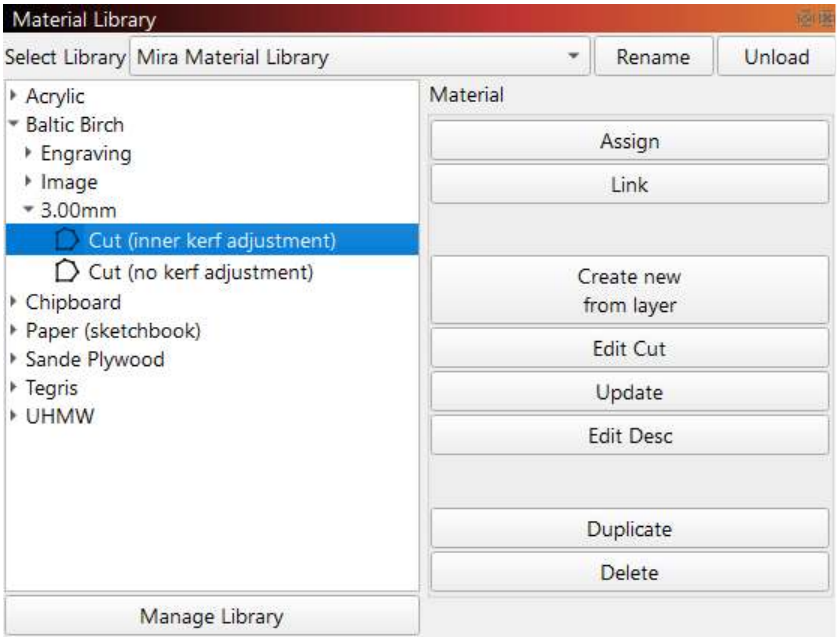


Tip

The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

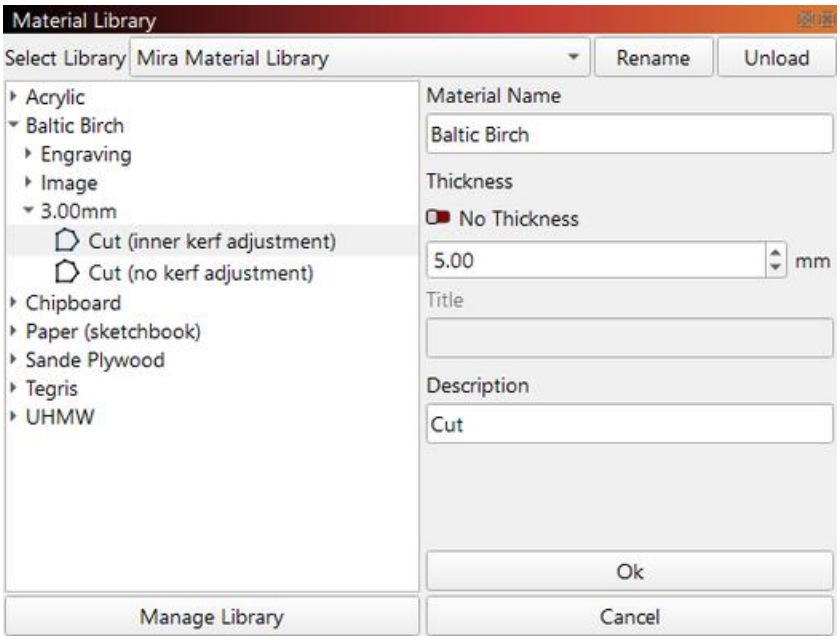
Options and Settings

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Creating New Material Library Presets

Click **Create new from layer** to save settings from the active layer (including Sub-Layers) to a new preset entry in the Library. Enter the below details and hit **OK** to save the material preset to the active Library.



- **Material Name**: The highest folder within the Library to include this material in.
- **Thickness**: The Z-height of the material to be cut through. Appears as the sub-folder name when in use.
- For surface operations like "Engrave" or "Score" that don't depend on the material thickness enable **No Thickness** and enter the operation into **Title**. This disables the Thickness option, and appears as the sub-folder name when used.

- **Description**: Provide greater detail about these settings, for example "Cut — Quick and Dirty". Appears as the name of the entry in the list.

All saved presets are organized in collapsible folders on the left side of the **Library Window**.

Using Material Library Presets

To apply presets to a layer in your project, first activate the layer by selecting a graphic set to that layer, or clicking the layer in the Cuts / Layers Window.

Next, select a preset from the list in the **Library Window**, and click Assign or Link.

Assign

Applies the preset settings from the selected entry to the active layer.

After applying the presets to the layer, you may adjust them in the Cut Settings Editor, and edits made in the Cut Settings Editor will not affect the Library entry, or vice-versa.

Link

Syncs the active layer to the settings stored in the selected preset entry. This disables the ability to edit the layer in the Cut Settings Editor, but any changes made to the Library entry will be automatically applied to the layer.

> **Tip**
>
> This is often useful for files that you'll be running again in the future, so that future improvements to your custom presets will be automatically applied in your project.

Modifying Presets

You can modify existing presets by selecting them in the list, then clicking one of the options in the right side of the window.

Edit Cut

Opens the Cut Settings Editor for this preset. After making changes, click **OK** to override the existing entry in the Library with your changes.

Update

Click **Update** to update your selected Material Library entry with the settings from your currently selected layer.

Edit Description

Click **Edit Description** to modify the **Name**, **Thickness** and **Description** for the highlighted entry.

Duplicate

Copies the highlighted preset to a new entry in the Library.

> **Tip**

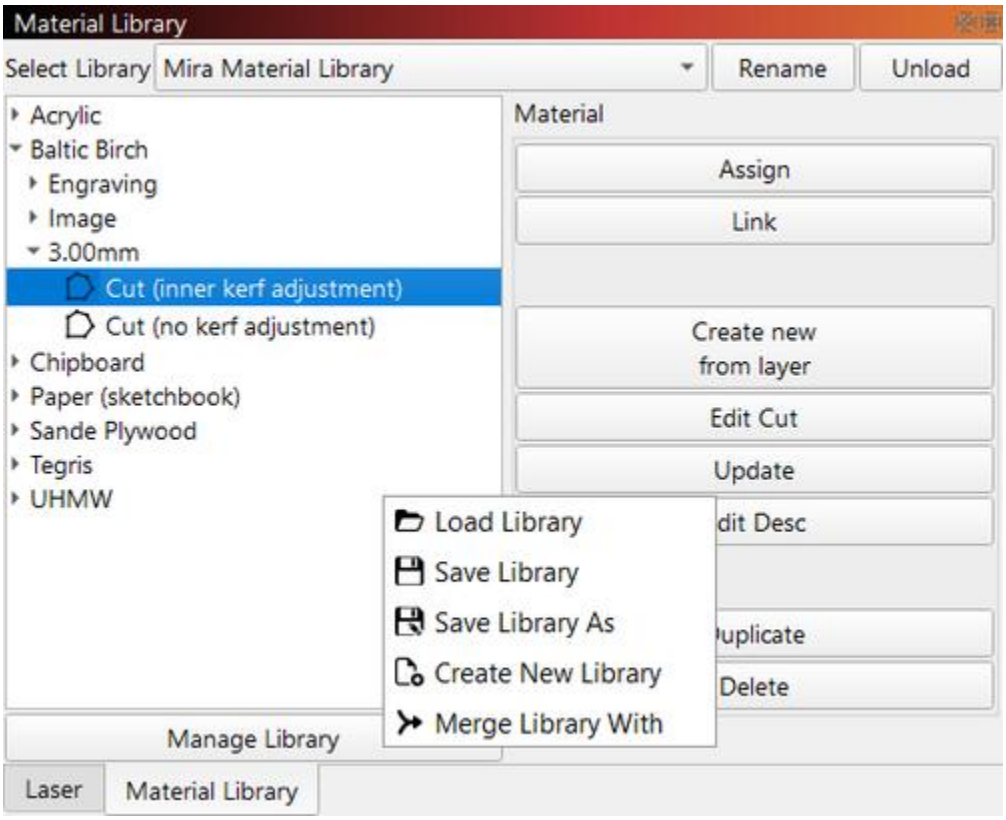
> Combined with **Edit Description** and **Edit Cut**, you can keep the same **Material Name** and **Thickness** to create nested entries (e.g. 3 mm cut, scan, and image). Keeping just the Name while changing the Thickness allows nesting of different thicknesses of the same material.

Delete

Permanently removes the selected preset from the Library.

Managing Material Libraries

Click the **Manage Library** button to open a menu of options for opening, saving, or combining libraries.



Load Library

Open your system's file browser, where you can navigate to the location of a previously saved `.clb` file and select it, to add it to the **Select Library** dropdown list.

Save Library

Stores or overwrites a previously a stored Library with an updated version containing any changes you've made since you last saved the Library. Libraries are saved with the `.clb` file extension.

Save Library As

Saves a copy of the active Library to your files with a different file name. Useful for archival purposes.

Create New Library

Creates a blank Library.

Merge Library With

Combines two Libraries together. Useful when importing settings from external parties.

Select Library

Select one of the Loaded Libraries from the dropdown list.

Rename

Changes the name of the Library.

Unload

Removes the Library from the **Select Library** list.

Advanced Usage

Multiple Lasers

If you have multiple lasers, it's typically best to create a **Material Library** for each laser.

When you save a Library, it is linked to the currently active device. In the future, when you select that device, the Library associated with it will load automatically.

When you change Libraries, LightBurn will check any Linked layers against the newly-loaded Library. If there's a Linked preset in each of the old and new Libraries that have matching details — **Material Name**, **Thickness/Title**, **Description** — LightBurn will automatically Link the layer to the preset in the new Library, and update the layer's Cut Settings accordingly.

Multiple Computers

Multiple computers can access a single **Material Library** file hosted on a network drive or from cloud service storage (e.g. Dropbox, Google Drive, iCloud, OneDrive, etc.). To open up a shared Material Library on a different computer, use the Load button in the **Library Window**.

> **Note**
>
> If you have multiple computers pointing to the same Material Library file, updates made from one computer won't show on the other computer(s) until LightBurn is restarted or the Library is reloaded with the **Load** button.

Related Topics

- Cut Settings Editor
- Color Palette

- Art Library

Modes and Advanced Control

Print and Cut

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=ddRW12VKIwY

Print and Cut aligns a current project to existing **Target Positions** — also called registration marks — in a previously output design.

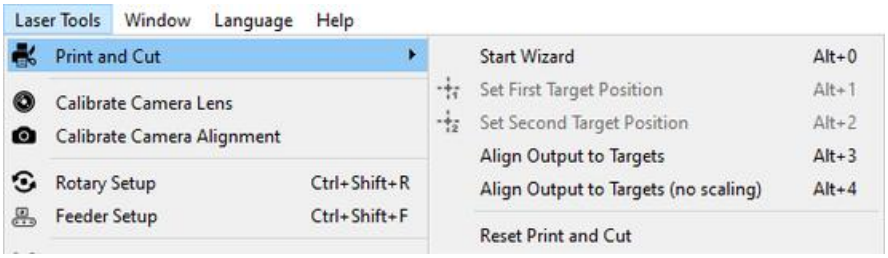
Print and Cut works by measuring the difference in location, orientation, and scale between two Target Positions in your laser's physical work area and the positions of two corresponding graphics in your LightBurn Workspace.

Use Print and Cut to cut around a pre-printed design, to cut a project larger than your laser's work area, to run a second pass over a previous cut, or to restart a halted job.

To open the Print and Cut Wizard, go to Laser Tools → **Print and Cut** → **Start Wizard**, press `Alt`/`Option`+`0`, or click the Print and Cut Icon in the Modes Toolbar.



All individual Print and Cut steps are also selectable from the Print and Cut submenu, without opening the wizard.



Print and Cut Requirements and Notes

- **Print and Cut** is only available for Gantry style lasers.
- In order to use Print and Cut, your laser must be connected to your computer, reporting its position accurately, and properly homed — for best results, you'll need a laser capable of automatic homing.
- You must use Absolute Coordinates positioning with Print and Cut. When you **Align Output to Targets** with Print and Cut, Absolute Coordinates will automatically be selected as your **Start From** mode — switching out of that mode will cause misalignment.
- When aligning your laser to the **Target Positions**, you must jog it using the commands in the Move Window, its controller's keypad (if applicable), or the button in the Print and Cut Wizard. You must not move your laser by hand at any point in the process.

- The accuracy of Print and Cut depends on the reliability of your laser's motion system and the precision of your laser's beam or red dot pointer's alignment to the Target Positions.
  - If your laser slips, snags, or loses position in any other way due to to mechanical problems, you will be unable to achieve accurate alignment with Print and Cut. See Troubleshooting: Mechanical Issues for advice on addressing those issues.
  - If the beam or red dot is not perfectly aligned to the center of the Target Positions, output will not be perfectly aligned either.
  - Alignments with sub-millimeter accuracy are achievable, but you should anticipate minor deviations in most cases.

Using Print and Cut

Notes on Target Positions

To use **Print and Cut**, you must align your laser to two **Target Positions** in a previously output design, and have corresponding, selectable Target Position graphics in your LightBurn Workspace. LightBurn uses the reported position of the laser over the Target Positions to align new output to the existing design.

The standard use case of Print and Cut is to align laser output to registration marks that were specially added to a file in order to use them as Target Positions later. In this case, the original file is intended to be printed by one device, and cut out (or around) by a laser later on.

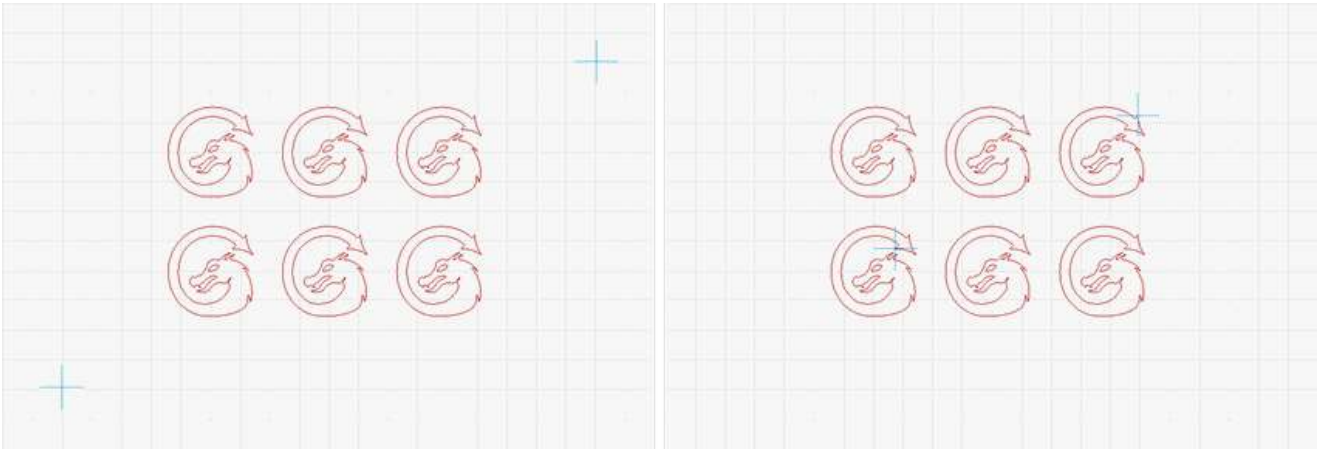
- In order to align to a previously printed design, the printed file must be in a format that can be Imported by LightBurn.
- The Target Position graphics must always maintain their relative position to the design after it is first output, in order to accurately align to them later.

  > **Tip**
  >
  > Lock all objects in your project to ensure they always maintain their relative position to one another.

- The Target Position graphics must be a single object or single Group of objects.
- Output is aligned relative to the center of the object or Group used as a registration mark.
- The farther apart the Target Positions are positioned from one another, the greater the accuracy of your alignment will be.
- Crosshair marks make ideal Target Positions. Include them in any printed design you intend to cut with your laser.
- In LightBurn, crosshairs must consist of one vertical and one horizontal line, Grouped together.

  > **Tip**
  >
  > Create a circle with lines between its top and bottom and left and right midpoint Snapping locations to make a perfect crosshair.

- The crosshairs cannot be images, or part of a larger image. They must be individually selectable vector graphics.
- Set the crosshairs to a Tool layer if you do not want them to be cut or engraved by your laser.
- To cut a project larger than your laser, you can divide it and engrave sets of crosshairs with each section, and use them to align the next section to the previous section.
- You can also center crosshair marks over sharp corners in an existing cut or engraving, and align to those corners. Useful for running a second pass over an incomplete cut, or recovering halted jobs.



Left: Crosshairs imported from a pre-existing design for use as Target Positions.
Right: Crosshairs placed over sharp corners in pre-existing design, to use as Target Positions.

See below for more information on alternate use cases:

- Rerunning a project or recovering a halted job
- Cutting a project larger than your laser's workspace

Aligning Your Project to Target Positions

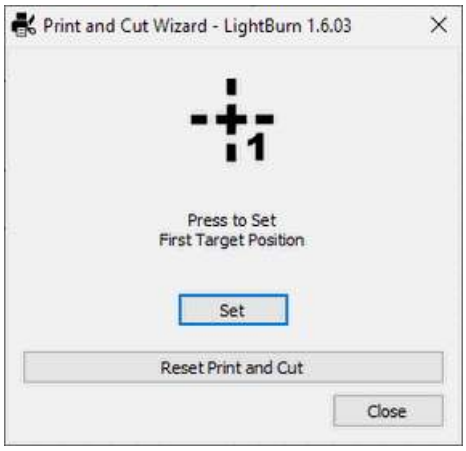
1. Start by Importing or creating a project that contains Target Position graphics — ideally crosshair marks — following the above guidance.
2. Go to Laser Tools → **Print and Cut** → **Start Wizard**, or press `Alt`/`Option`+`0` to open the Print and Cut Wizard.

   > **Using Print and Cut from outside the Wizard**
   >
   > Optionally, you can choose not to open the wizard and instead set Target Positions and align output from the Print and Cut submenu, or by using hotkeys.

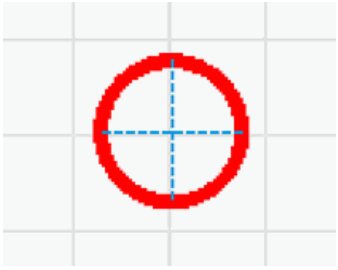
3. In your LightBurn Workspace, select the first crosshair mark.
4. Using the buttons in the Move Window or on your controller's keypad (if applicable), jog your laser over the first Target Position in the existing output in your laser's work area.
5. Use the laser Fire button in the Move Window or the Pulse button on your laser's controller (if applicable) to align the laser's beam to the center of the Target Position. If your laser has a red dot pointer, position it over the center of the Target Position instead of firing or pulsing the laser.

> **Warning**
>
> Make sure you are wearing properly rated safety glasses before firing or pulsing your laser. Consult your laser's manufacturer to make sure your safety glasses are properly rated for your laser.

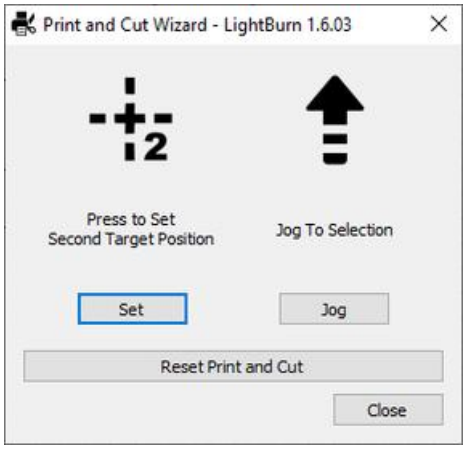
6. After aligning the laser to the Target Position, click **Set** to set the laser's current location as the first Target Position. If you are not using the Print and Cut Wizard, select **Set First Target Position** from the Print and Cut submenu, or press `Alt`/`Option`+`1`.



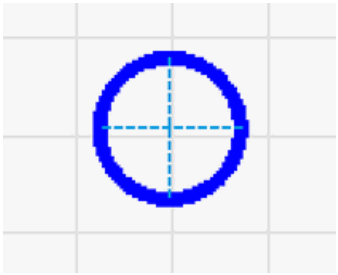
7. The selected crosshair mark in your Workspace will be highlighted in red.



8. Select the second crosshair mark in your LightBurn Workspace.
9. Jog your laser over the second Target Position in the existing output. The **Jog** button beneath **Jog To Selection** in the Print and Cut Wizard will automatically jog your laser to the predicted location of the second Target Position. Use the Move Window or controller's keypad to correct or fine-tune alignment if the predicted location is imprecise.
10. After verifying alignment, click Set to set the laser's current position as the second Target Position. If you are not using the Print and Cut Wizard, select **Set Second Target Position** from the Print and Cut submenu, or press `Alt`/`Option`+`2`.

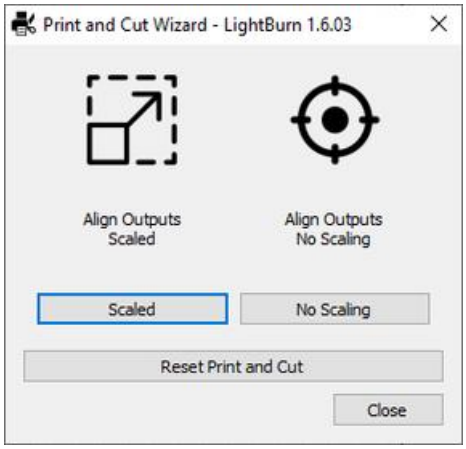


11. The second crosshair mark in your Workspace will be highlighted in blue.

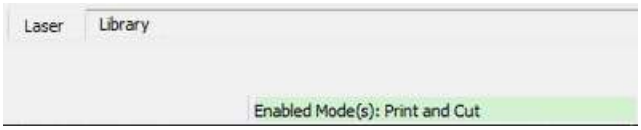


12. Select **Scaled** (`Alt`/`Option`+`3`) or **No Scaling** (`Alt`/`Option`+`4`) to align the current project in your LightBurn Workspace to the Target Positions in your laser's work area. In the Print and Cut submenu, these options appear as **Align Output to Targets** and **Align Output to Targets (no scaling)**.
    - The Scaled option resizes your project to account for any discrepancy between the size of your project in LightBurn and the size of the pre-existing output in your laser's work area, based on the distance between the Target Positions.
    - The No Scaling option aligns the project but does not resize it according to the distance beteween the Target Positions.
13. The wizard will close when you select one of the above options. You can select an alternative option by re-opening the wizard, selecting it from the Print and Cut submenu, or by using the appropriate hotkey.

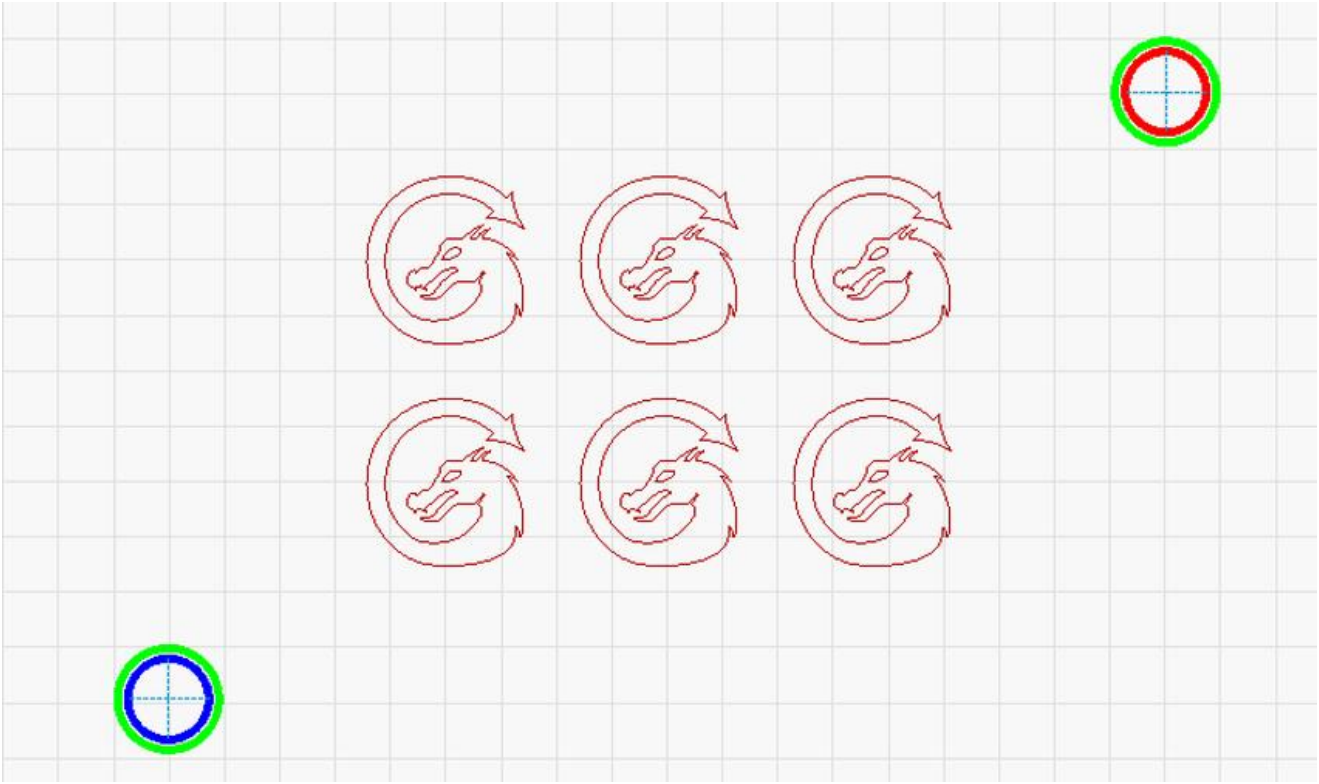
    Choose **Reset Print and Cut** to remove the current alignment and start the process over.



14. The **Enabled Mode(s): Print and Cut** indicator will appear at the bottom right corner of your screen:



And green outlines will appear around both crosshair marks:



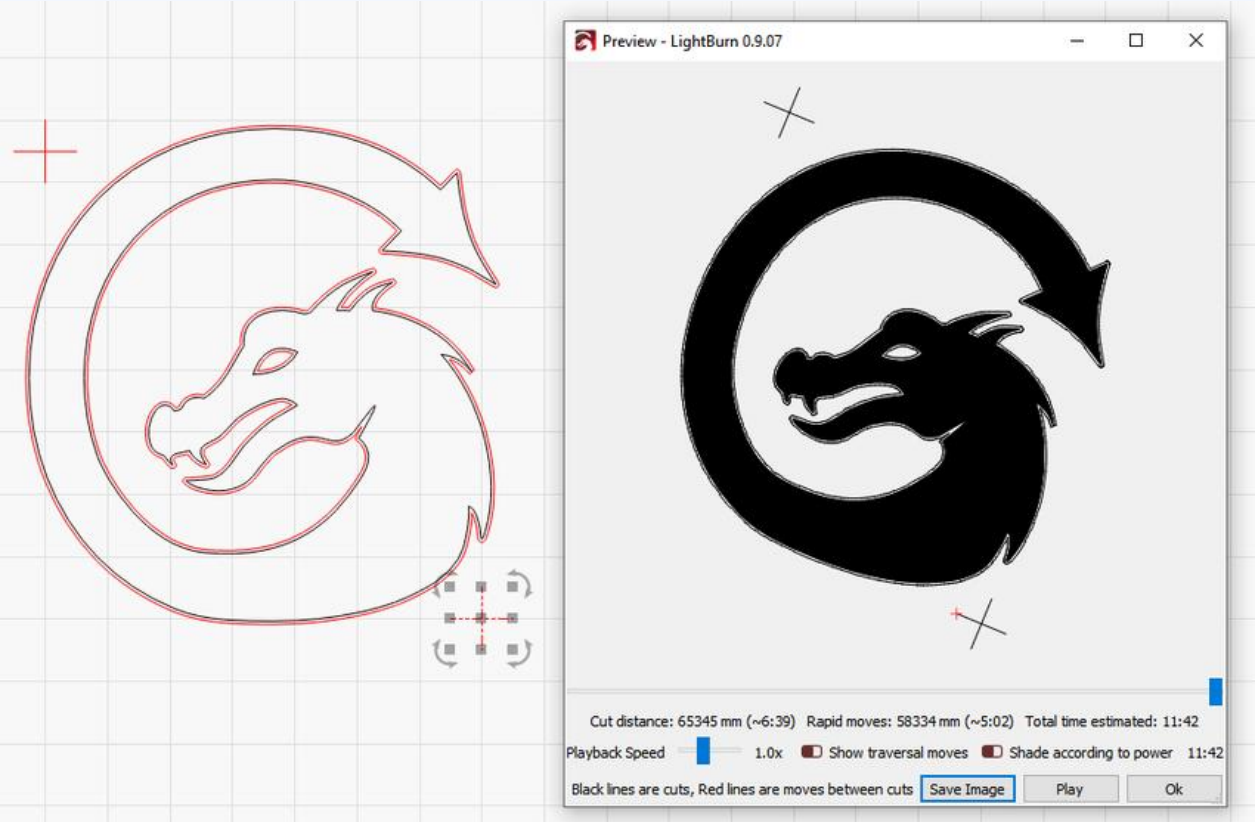
Warning

Absolute Coordinates will automatically be selected as your **Start From** when you enable Print and Cut. Make sure not to switch to a different Start From mode.

15. After verifying that your project will output correctly by Framing and viewing it in the Preview window, Start the job.

Expected appearance of Print and Cut Preview

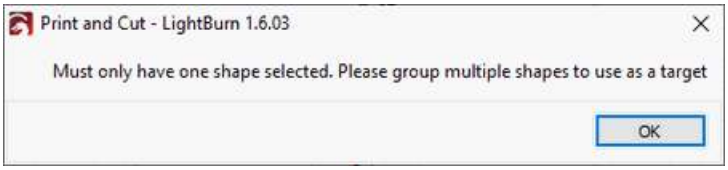
You should expect to see your project rotate in the Preview window, if the location of the Target Positions you've set indicates that a change in orientation is required.



Error Messages

There are a few error messages you may receive when setting the **Target Positions**.

Multiple Shapes Selected



You may not use multiple shapes as a registration mark, unless they are Grouped. If you are using a crosshair mark as your Target Position graphics, you must Group the lines before setting the positions.

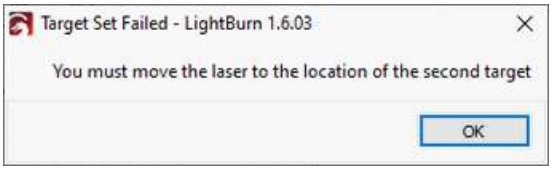
Need to Select a Different Target



After setting your first Target Position, you did not select a different Target Position graphic in your LightBurn Workspace, before clicking to set the second position.

Select the second graphic, and try again.

Must Move the Laser

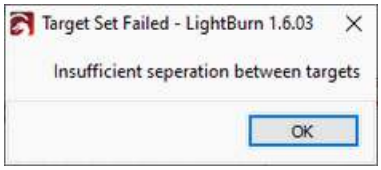


The laser has reported no movement from the position it was in when you set the first Target Position.

Make sure the laser is still connected and reporting its position. If the laser has disconnected, you must re-home and start the process over.

Make sure you are moving the laser using the jog buttons in the Move Window or on its controller's keypad (if applicable), and not by hand.

Insufficient Separation



The laser has reported a change in position, but the second Target Position is too close to the first. Make sure you have jogged your laser the whole way to the second position using the jog buttons in the Move Window or on its controller's keypad (if applicable), and not by hand.

If the Target Positions are physically too close together, you must amend your original design to increase their separation. That requires re-printing or re-outputting the original design, and making sure the design in your LightBurn Workspace is updated to match.

Alternate Use Cases

Rerunning a Project or Restarting a Halted Job

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=pOl75orZw3I

If a design has suitable sharp corners, you do not need to have special **Target Positions** in the original design to use **Print and Cut**.

Using sharp corners in existing output allows you to align a project that was not originally intended to be used with Print and Cut, either to run a second pass over an incomplete cut, or to align to and restart a halted job.

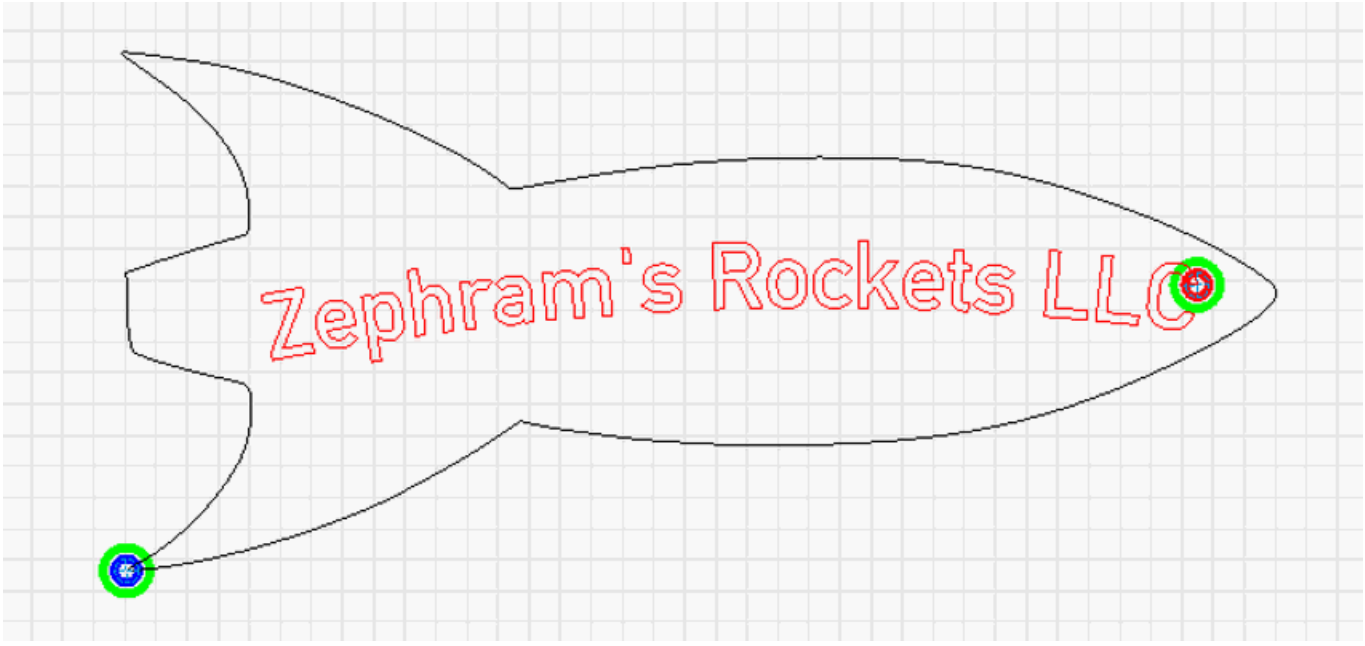
To align to corners of existing output, you must create two crosshair marks and position them so that they are perfectly centered over two sharp corners in your design in LightBurn.

> **Tip**
>
> Use Snapping to ensure that the centers of the crosshairs are directly over the points of the corners.

Move the laser to the corresponding corners in the existing output, and select the appropriate crosshair for each corner when setting the Target Positions.

These crosshairs do not ever need to be output to the laser, so they should be set to a Tool layer.

> **Alternate method for restarting a halted job**
>
> ▶ Watch the video on YouTube: https://www.youtube.com/watch?v=WmOGoHiYQu4



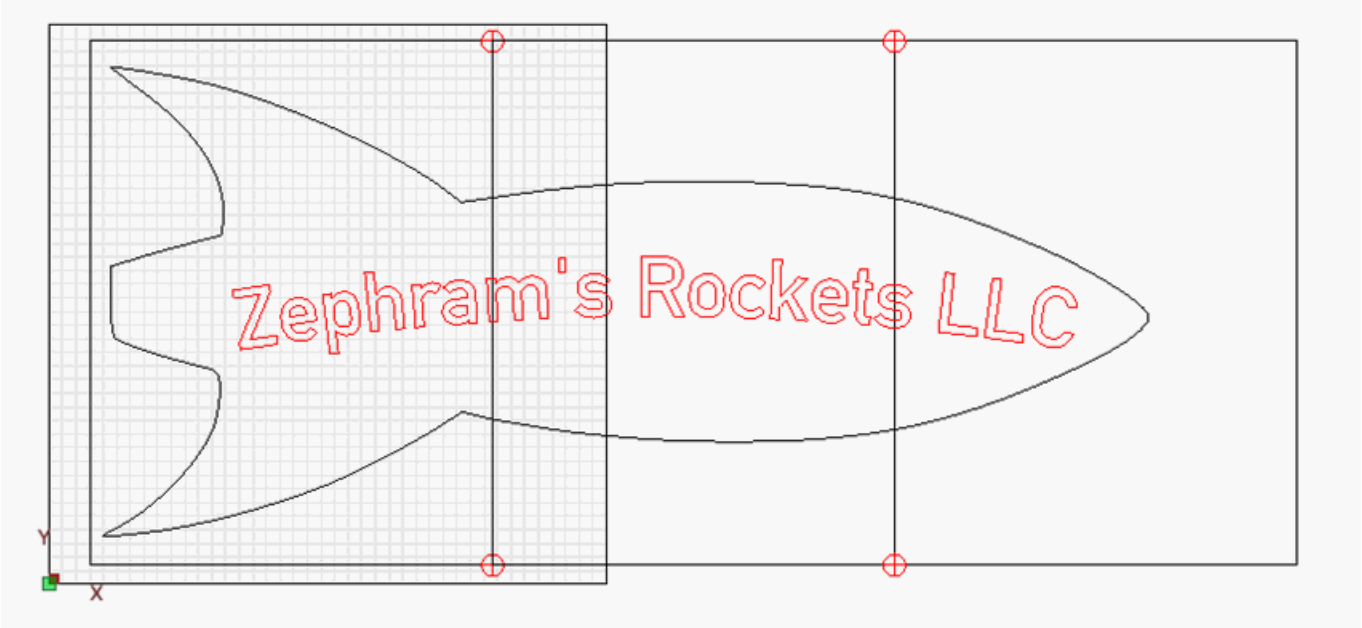
Cutting a Project Larger Than Your Laser's Workspace

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=n__saOKVupA

In order to cut a project larger than your laser's work area, divide it into sections using Cut Shapes, and add crosshair marks to each section, to use as **Target Positions** to align the subsequent section to.

Be sure to divide the sections and position the crosshair marks such that each section and the crosshairs from the previous section can fit in your laser's work area.

After a set of crosshairs of each section have been engraved, set them to a Tool layer, so they are not output to your laser again — be sure not to delete or move them, you'll still need them for alignment. Set the rest of the graphics in each completed section to a Tool layer as well, or delete them.

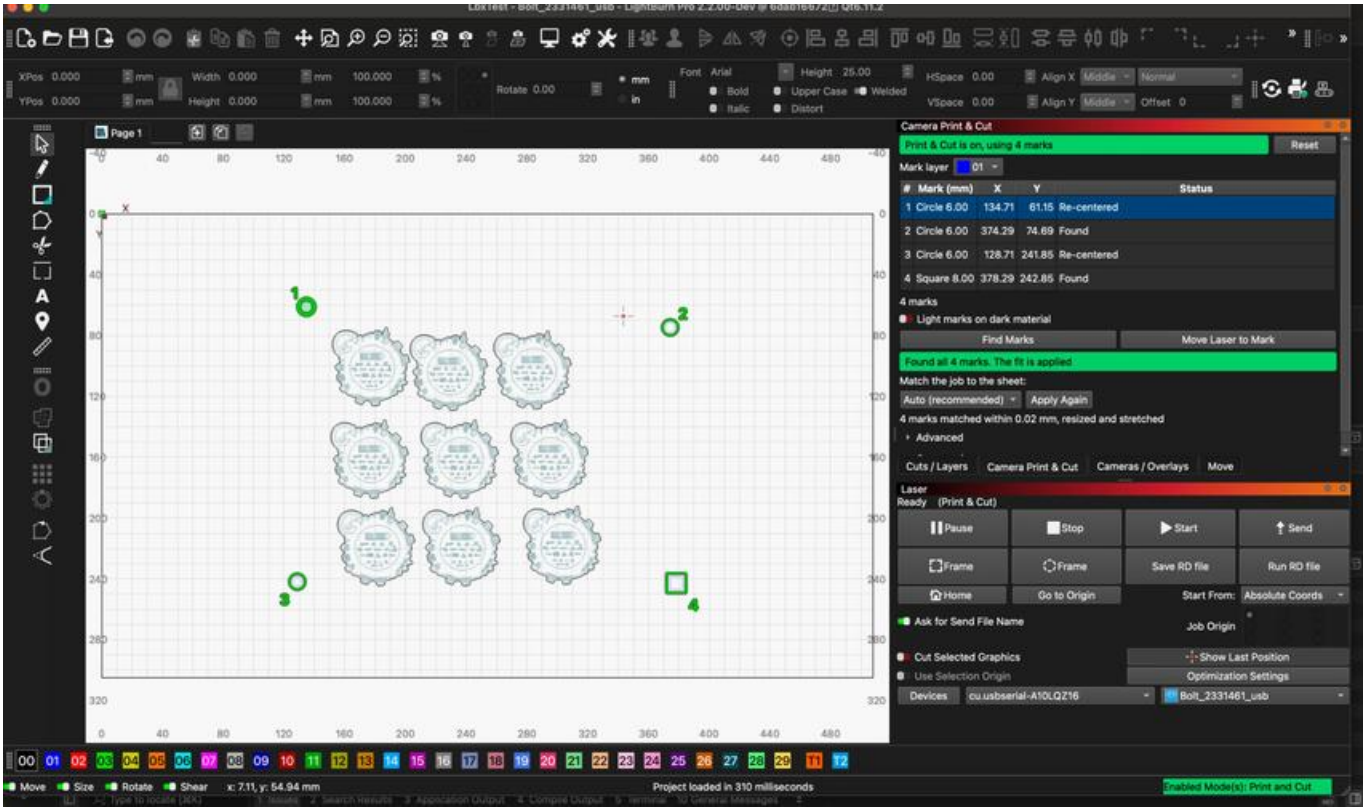


Related Topics

- Coordinates and Origin

Camera Print and Cut

Use the **Camera Print & Cut** window to line up a job with a printed sheet automatically using a Head Camera. LightBurn moves the laser head to each registration mark on the sheet, finds it, and moves the job to match. Open it from Window → **Camera Print & Cut**.



About Camera Print & Cut

Print & Cut lines up a job with something you've already printed, such as a sticker sheet, by matching registration marks in your design to the same marks on the material. The classic method uses two marks that you find by hand, jogging the laser onto each one.

Camera Print & Cut does the finding for you. It uses three or more marks, so it can correct for a sheet that is shifted, rotated, or printed slightly too large or small. It can also correct for a printer that stretches the sheet in one direction.

The marks can either be printed or engraved by the laser itself in an earlier job.

Before You Begin

You need:

- A gantry laser (one with a head that moves around), connected to LightBurn. Galvo lasers are not supported.

- A laser with a Head Camera (one mounted on the laser head) that is calibrated, aligned, and included in the Overlay. See Cameras / Overlays Window for more information.
- **Three or more registration marks** in your design, on a layer of their own, not all in a straight line. Four marks near the corners of the sheet work best.

Registration marks can be:

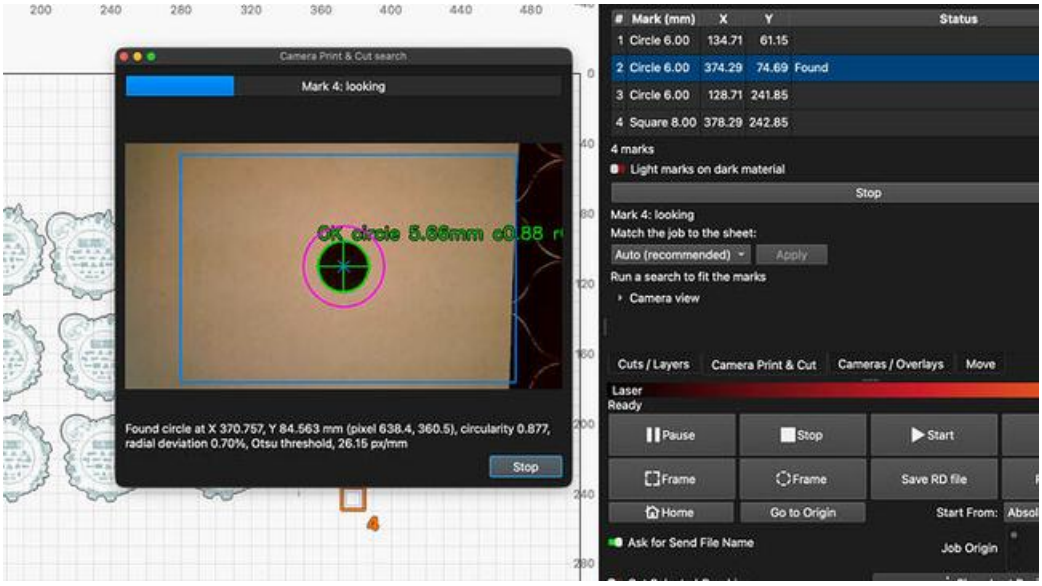
Shape	Notes
Filled circle	Recommended. 5 mm works well.
Filled square	Any rotation.
Donut (ring)	One shape with a hole, or a circle inside a circle.

Marks should be dark on a light material. For light marks on dark material, turn on **Light Marks on Dark Material**.

> **Tip**
>
> Put the marks on a layer you don't use for anything else. LightBurn turns that layer's output off for you, so the marks aren't cut.

Using Camera Print & Cut

1. Load your design, with its registration marks, and print it at 100% scale.
2. Go to **Window** → **Camera Print & Cut** to open the Camera Print & Cut Window.
3. LightBurn chooses the **Mark Layer** for you. Check to make sure it's correct, and if not, change the Mark Layer to the one you want to use. The marks and their locations appear in the list and are highlighted in the workspace.
4. If you're using light marks on dark material, turn on **Light Marks on Dark Material**.
5. Place the printed sheet in the laser.
6. Jog the head so the camera can see the mark named in the yellow line in the window, for example "Park the head over mark 1".
7. Click **Find Marks**. LightBurn finds that mark, then moves to each of the others in turn.
8. When every mark is found, the fit is applied automatically, and the window says **Found all 4 marks. The fit is applied**.
9. If you want to be really certain, check the fit: select a mark in the list and click **Move Laser to Mark**.
10. Run your job as usual. It is moved onto the printed sheet.



Camera Print & Cut stays applied until you click **Reset**, choose **Reset Print & Cut**, or start a new search. The two-point Print & Cut targets and Camera Print & Cut replace each other: setting either one clears the other.

Camera Print & Cut Window Reference

Mark Layer

Choose the layer that holds your registration marks. LightBurn lists every shape on that layer as a mark, except text, images, and barcodes, and turns the layer's output off, so the marks are not cut. If you turn the output back on, the window warns you.

When the chosen layer has no usable marks, for example after you open or import a file, LightBurn looks for the layer that holds your marks and chooses it for you. It looks for a layer with only marks on it: three or more small circles, squares, or donuts. They can be mixed, such as one square among circles to show which way up the sheet is. If it can't find one, it asks you to put the marks on a layer of their own and choose that layer. A layer you choose yourself is kept, even before it has marks on it.

Mark List

Each row is one mark:

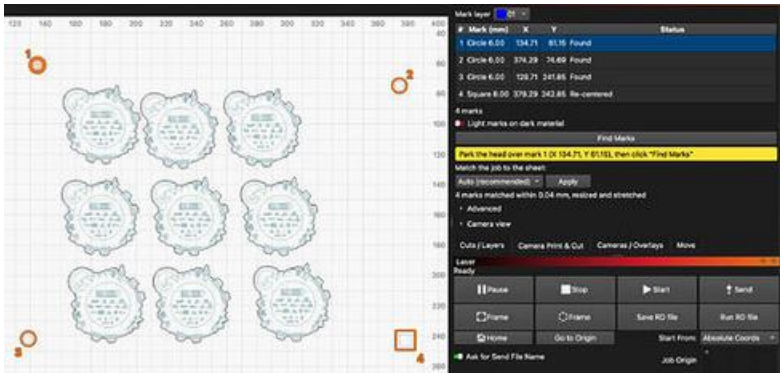
Column	Shows
#	The mark's number. The same number is shown next to the mark in the Workspace.
Mark	The mark's shape and size.
X, **Y**	Where the mark is in your design.
Status	What the search found, such as **Found**, **Not found**, **Skipped**, or **Out of reach**.

Hover a row to see where the camera found that mark. If a shape is not recognized, its row shows **Unknown**: double-click its shape to choose one.

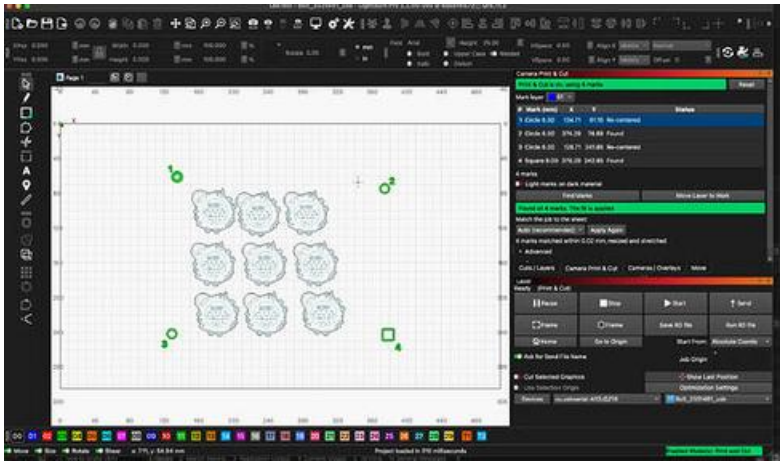
Select a row to highlight that mark in the Workspace.

Marks in the Workspace

While the Camera Print & Cut window is open, every mark is outlined in orange, with its number beside it, so you can find marks even on a hidden layer or a pale layer color. Circles and donuts get a circle, and squares a square turned to match the mark. The mark selected in the list has a thicker, filled outline. Once the fit is applied, the outlines of the marks it used turn green.



This screenshot shows the marks in orange, with one selected, before the fit is applied



This screenshot shows the rings in green after the fit is applied.

Find Marks, Search Here, and Skip

Click **Find Marks** to find every mark. Before you click it, jog the head so the camera can see the mark the window names. You can choose a different starting mark by selecting it in the list.

LightBurn searches around where it expects each mark. If it can't find one, it stops and shows **Mark # not found**. You can:

- Jog the head near the missing mark, then click **Search Here**.
- Click **Skip** to leave the mark out and carry on, as long as at least three marks remain.

Click **Stop** at any time to end the search.

Match the Job to the Sheet

Choose how the job may be adjusted to match the marks:

Option	What it does
Auto (recommended)	Chooses the best of the options below for the marks found.
Keep size	Moves and rotates the job only.
Resize evenly	Also resizes the job, by the same amount across and down.
Resize and stretch	Resizes across and down separately, for a printer that stretches the sheet. Needs four or more marks to be checked properly.

Below it, a line tells you how closely the marks matched, for example **4 marks matched within 0.08 mm, resized evenly**.

Apply and Reset

The fit is applied automatically when every mark is found and they all match. Otherwise, check the warning, then click **Apply** to use the fit anyway.

While a fit is applied, a green bar at the top of the window says **Print & Cut is on**. Click **Reset** beside it to turn Camera Print & Cut off.

If one mark doesn't match the others, the window names it in red, for example **Mark 3 doesn't match the others: it is 0.7 mm off**. This usually means it was found on the wrong mark, or the printed sheet is damaged near it. The fit is not applied automatically in this case.

Move Laser to Mark

Once a fit is applied, select a mark in the list and click **Move Laser to Mark**. The laser moves, without firing, to where the camera found the mark, so you can check it with your red dot pointer or a low-power pulse. If the laser is not centered on the mark, check your head camera's alignment.

Advanced

Click **Advanced** to see the fit in detail: how far each mark is from the fit, the overall error (RMS), and the scale and rotation applied to the job. Most jobs don't need this.

Camera View

Click **Camera view** to see the last image the camera captured, with the mark it found outlined.

To check one mark without moving the head, select it in the list, jog the head so the camera can see it, and click **Find Mark in View**. The line below the image says where the mark was found and how big it is.

Messages

Problems show in red, and things to check in orange:

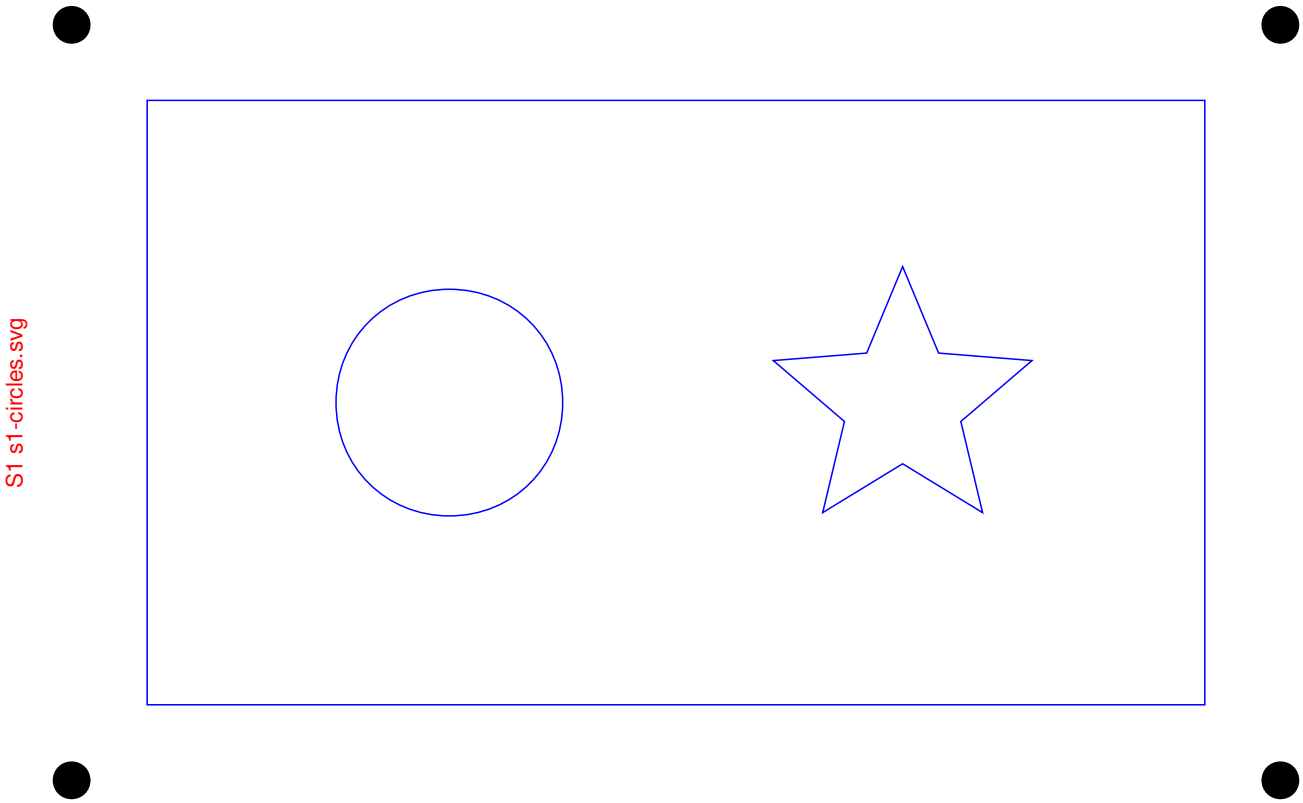
Message	What to do
Needs a ready head camera that is included in the overlay	Connect the head camera, and turn on **Include in Overlay** in its right-click menu in the Cameras window.
Align the head camera first	Run the head camera alignment in the Cameras window.
No marks on layer …	Choose the layer your marks are on, or put them on the chosen layer.
Needs at least 3 marks	Add marks. Four near the corners work best.
The marks are in a line	Move at least one mark off the line.
… not recognized	Double-click the mark's shape in the list and choose one.
Layer … is set to output, so the marks will be cut	Turn the layer's output off in **Cuts / Layers**.
Connect the laser first	Connect to your laser.

Tips

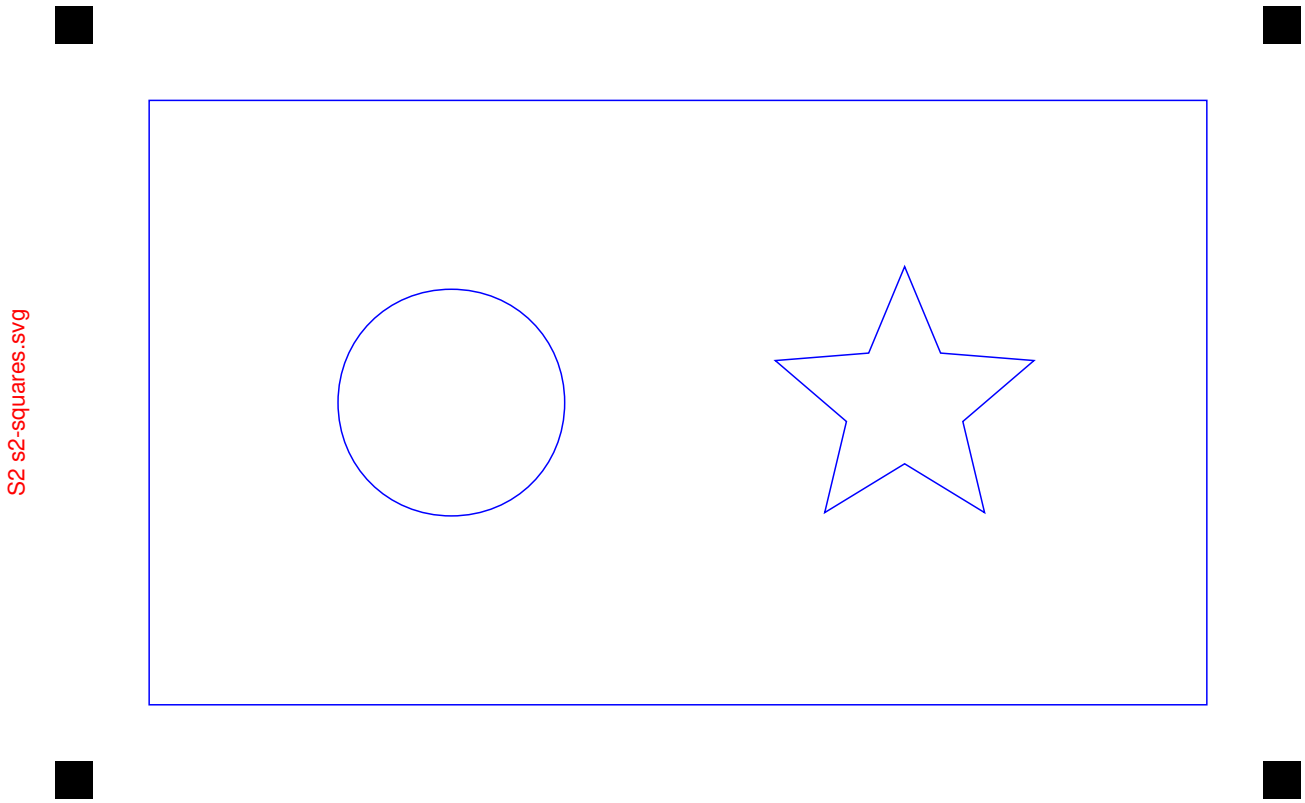
- Print at 100% scale. **Auto** corrects small printer scaling, but not "fit to page". A sheet printed more than about 10% too large or too small, or stretched more than about 15% in one direction, is not found: the camera view shows the marks labelled "wrong size", or not recognized at all.
- Use four marks, one near each corner of the sheet, for the best fit over the whole sheet.
- Laser-made marks work well: engrave filled circles in one job, then use them as marks for the next.
- A whole sheet of designs can share one set of marks. Every mark on the mark layer is used for one fit across the sheet.

Sample Files

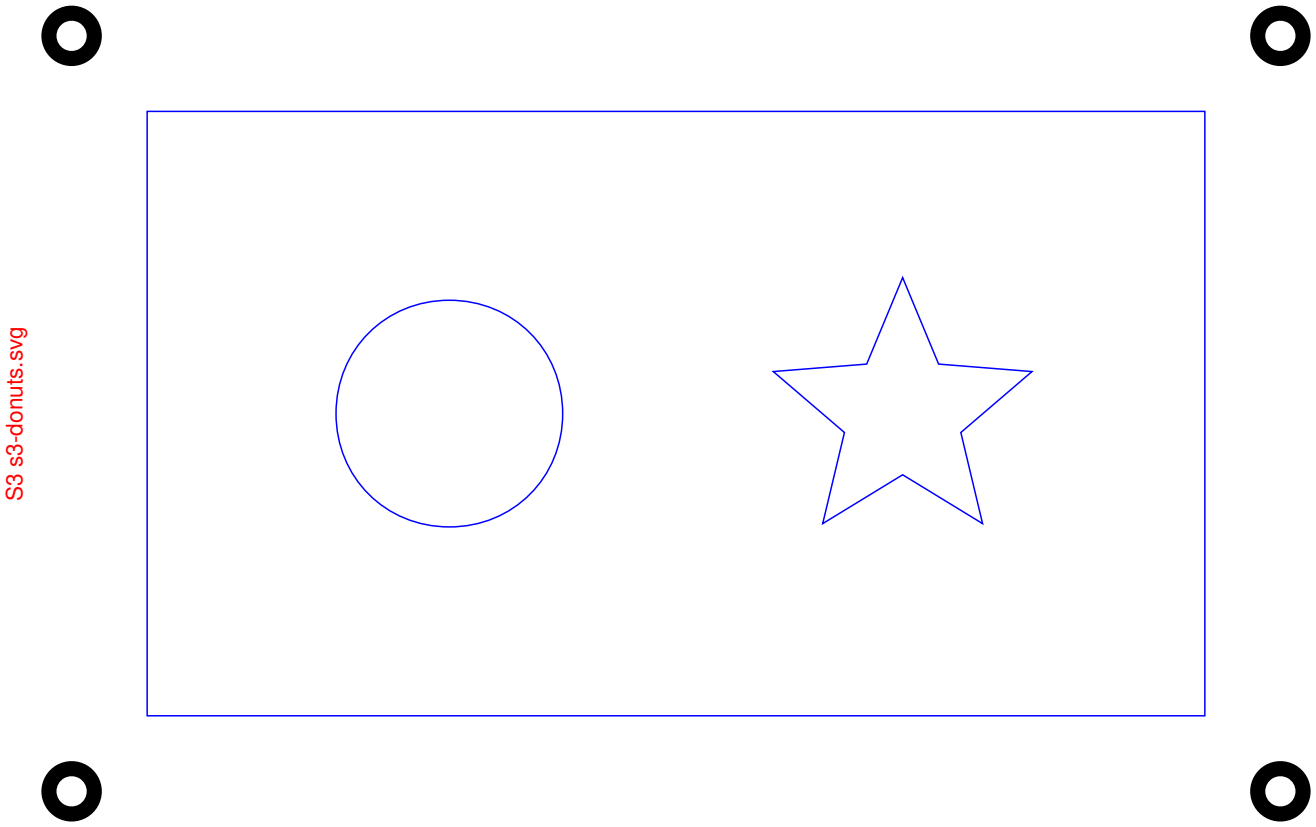
You can import or open these SVG files in LightBurn to test out **Camera Print & Cut**.



Circle and Star with Circular Marks



Circle and Star with Square Marks



Circle and Star with Donut Marks

Related Topics

- Print & Cut
- Cameras Window
- Window Menu

Rotary Mode

Rotaries are used to turn cylindrical objects so that their surfaces can be marked by a laser. In order to use your laser with a rotary, you must first establish several important parameters in the **Rotary Setup** window that control the rotary's motion.

> **Info**
>
> If you're working with a tapered object, use the largest end of the taper for the object diameter.

To open Rotary Setup, go to Laser Tools → **Rotary Setup** (`Ctrl`/`Cmd`+`Shift`+`R`), or click the rotary icon in the Modes Toolbar.

For more information on setting up and using a rotary with your type of laser, see:

Rotary Mode (DSP) **Rotary Mode (GCode)** **Rotary Mode (Galvo)**

You can enable or disable **Rotary Mode** by toggling the **Enable Rotary** switch in the **Rotary Setup** window, or, if Show rotary enable in main window is enabled in your Settings/Preferences, by toggling the switch in the Laser Window.

When Rotary Mode is enabled, you'll see a status indication in the bottom right corner of your screen:



Rotary Mode (DSP)

> **Note**
>
> The Rotary Setup window is not available for lasers with TopWisdom or Trocen controllers. See below for more information on setting up a rotary with these controllers.

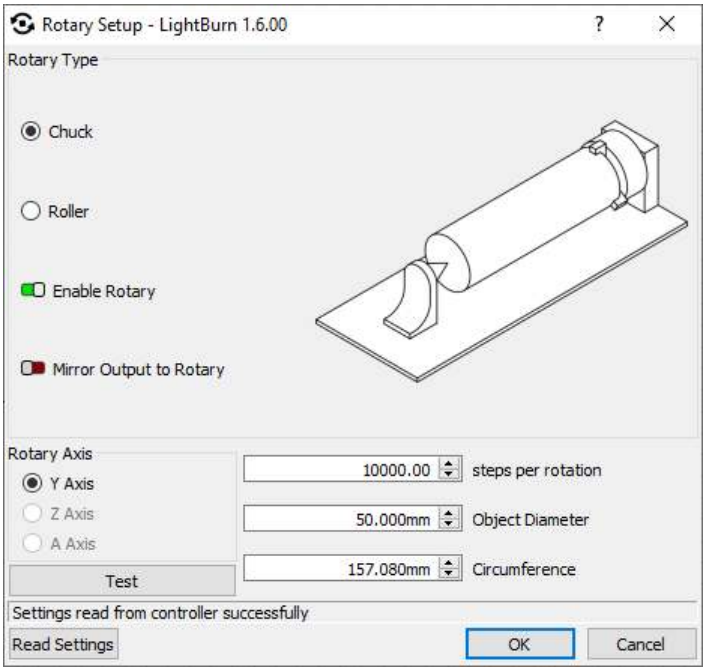
Rotaries are used to turn cylindrical objects so that their surfaces can be marked by a laser. In order to use your laser with a rotary, you must first establish several important parameters in the **Rotary Setup** window that control the rotary's motion.

> **Info**
>
> If you're working with a tapered object, use the largest end of the taper for the object diameter.

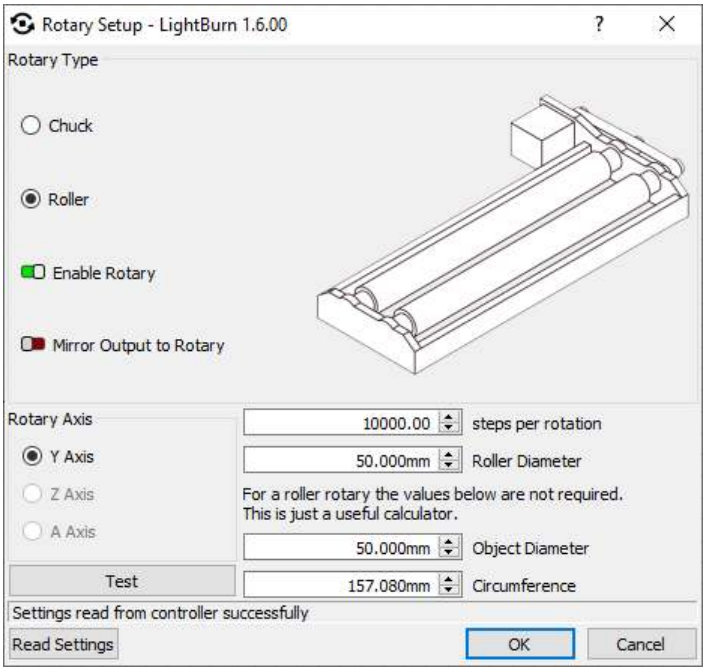
To open Rotary Setup, go to Laser Tools → **Rotary Setup** (`Ctrl`/`Cmd`+`Shift`+`R`), or click the rotary icon in the Modes Toolbar.

Rotary Setup (Ruida)

Click any option in the images below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Rotary Setup for Ruida Controllers (Chuck)



Rotary Setup for Ruida Controllers (Roller)

Rotary Type

Select whether your rotary is a **Chuck** or **Roller** style rotary. Chuck rotaries have adjustable jaws that grab and hold objects, while Roller rotaries have revolving rollers or wheels that objects are placed on top of.

Enable Rotary

> **Tip**
>
> If Show rotary enable in main window is enabled in your Settings/Preferences, you can also enable or disable Rotary Mode using the switch in the Laser Window.

This switch controls whether **Rotary Mode** is enabled. When enabled, rotational commands are sent to the laser to control rotary motion, and output is split into segments.

When Rotary Mode is enabled, you'll see a status indication in the bottom right corner of your screen:

Mirror Output to Rotary

Enable the **Mirror Output to Rotary** switch if output to the rotary is inverted in the dimension of rotation.

Rotary Axis

For most lasers with Ruida controllers, the **Rotary Axis** is automatically set to **Y** and cannot be changed.

If your rotary is connected to a U Axis port, the controller will handle transferring the commands the correct axis, provided the firmware version running on the controller supports U Axis rotary output. In some cases, you may need to manually select the U Axis through the controller.

Consult your laser's manufacturer for more information on U Axis rotary support.

Rotary Settings

Steps Per Rotation

> **Note**
>
> With a **Roller** style rotary, it is the roller or wheel driven by the motor that must make one complete rotation when you click the Test button, and not an object on the rotary.

Set the **steps per rotation** value to the number of motor steps required to spin the rotary one complete rotation.

If this value is incorrect, output will be shrunk or stretched.

In most cases, steps per rotation will be provided by the manufacturer of your rotary or machine — consult the manufacturer if the steps per rotation value was not provided in the instructions included with your device.

Test

When the **Test** button is pressed, the rotary should turn a full 360 degrees, pause, then rotate back to the starting point. If the rotary turns too far, reduce the steps per rotation value. If it does not turn far enough, increase it.

Roller Diameter

This option is only presented for **Roller** style rotaries.

In most cases, **Roller Diameter** is provided by your rotary's manufacturer. If not, carefully measure the diameter of the roller or wheel driven by the rotary motor.

Once this measurement is accurately identified and set, it never needs to be adjusted again. If it is inaccurate, output will be shrunk or stretched.

Object Diameter

> **Tip**
>
> Use high precision calipers to guarantee the accuracy of your diameter measurement.

For **Chuck** style rotaries, carefully measure and enter the **Object Diameter** of the item on your rotary. This value must be updated every time you place an object with a different diameter on your rotary. If it is inaccurate, output will be shrunk or stretched.

For **Roller** style rotaries, Object Diameter does not affect output, but can be entered here to calculate the **Circumference** of your object.

> **Tip**
>
> Object **Circumference** and **Object Diameter** are linked — if you manually adjust the value for one measurement, LightBurn will automatically calculate the other.

If your object is tapered, check out our Taper Warp tool. Use the largest diameter of your object as the Object Diameter value.

Circumference

You can choose to directly measure and enter your object's **Circumference** using a tape measure. As with Object Diameter, this value must be as accurate as possible when using a **Chuck** style rotary, but does not affect output when using a **Roller Rotary**.

> **Tip**
>
> You can use the Circumference value when adjusting the size of your graphics. For full wrap designs, the combined height of your graphics should be identical to the Circumference.

Read Settings

Instructs LightBurn to read rotary settings settings from your laser's controller.

Writing Changes

After changing rotary settings, you must power-cycle the controller, as the internal logic isn't adjusted for the new settings until you do.

Change the setting, press **OK** to close the **Rotary Setup** window and write the changes to the controller, then power off the control board and power it back on.

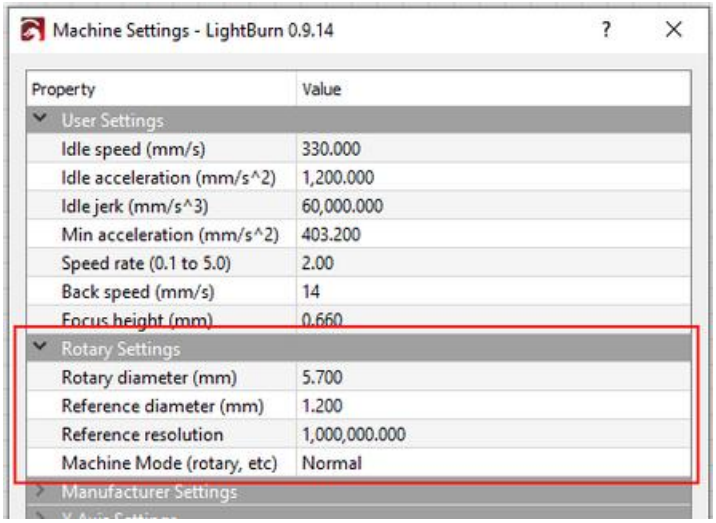
You may need to press the Esc (or stop) button on the controller to prevent it from trying to home itself, if the rotary is plugged into the Y Axis port and the laser cannot travel in Y.

Rotary Setup (TopWisdom and Trocen)

TopWisdom

Users with TopWisdom controllers must enable Rotary Mode and set parameters through the Machine Settings of the controller itself.

In LightBurn, go to Laser Tools → Machine Settings to enable Rotary Mode as well as view and adjust the rotary controls.



The options presented here are the same as those available in AutoLaser, the stock software for the TopWisdom. For more information on these settings, refer to this manual, or contact your machine's manufacturer.

Trocen

Users with Trocen controllers must enable Rotary Mode and adjust settings through the controller display. For more information on these settings, refer to this manual, or contact your machine's manufacturer.

Positioning

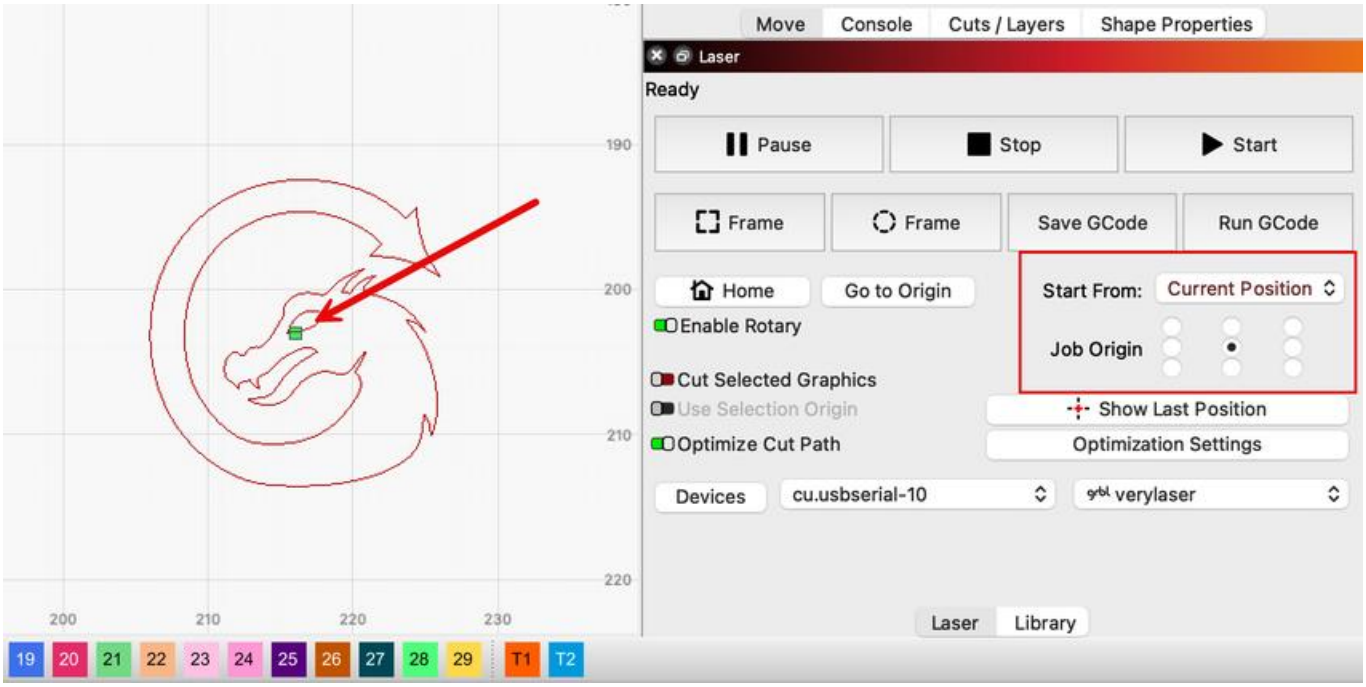
> **Tip**
>
> If your rotary is plugged into your laser's Y Axis, position the laser head toward the center of its work area before plugging in the rotary.

In most cases, Current Position is the best **Start From** mode to use when working with a rotary.

Align the object under your laser head in the X dimension. Output will be positioned on the object in the X dimension relative to the laser head and **Job Origin** selection. The Job Origin is indicated on your graphics by a green square. Parts of the graphic to left of the green square will output to the left of your laser head, and parts to the right will output to the right.

Turn the rotary or object on the rotary to determine where the engraving will output on the object in the dimension of rotation. Output in this dimension is also determined relative to your Job Origin selection — the rotary will turn to reach graphics positioned above or below the Origin.

See Coordinates and Job Origin for more information on the Start From mode and Job Origin settings.



In the above example, output will be centered around the starting position of the laser head.

> **Note**
>
> For some Trocen controllers, the Start From mode must always be selected through the controller display.

Troubleshooting

- If your output is shrunk or stretched in the direction of rotation, either **steps per rotation**, **Object Diameter** (if working with a **Chuck** style rotary), or **Roller Diameter** (if working with a **Roller** style rotary), is incorrect.
  - Use the **Test** button in the **Rotary Setup** window to check steps per rotation. The rotary should make one 360 degree rotation before returning to the start. If you are working with a Roller rotary, it is the roller or wheel driven by the motor that must make one complete rotation and not an object on the rotary. If it turns too far or not far enough, adjust steps per rotation.
  - Use high precision calipers to double-check the Object or Roller Diameter value.
- If the controller returns a "Y Slop Over" or similar error when you try to start a job, jog the rotary using the Y Axis arrows until it reports a position near to the center of its work area (half the max travel distance in Y.)
- If an object slips on a Roller rotary, try:
  - Wrapping the rollers with a non-slip material, or even rubber bands.
  - Placing weights inside the object, such as ball bearings or other small round objects, to help press the object against the rollers.

    - Going to Laser Tools → Machine Settings and reducing the Acceleration value of your rotary axis, in your machine's firmware settings.

- If you end up with a flat part at the bottom of the engraving, it means your rotary gear mechanism has backlash. Usually, tightening any belts in the rotary device will fix this, but, if not, you can place a small line a few mm below the bottom of your design, set to very low power (0.1%) and set it as the first layer to run. The rotary will spin to this position, then back up over that few millimeters of gap, and will take up the backlash with that movement.

Related Topics

- Taper Warp

Rotary Mode (GCode)

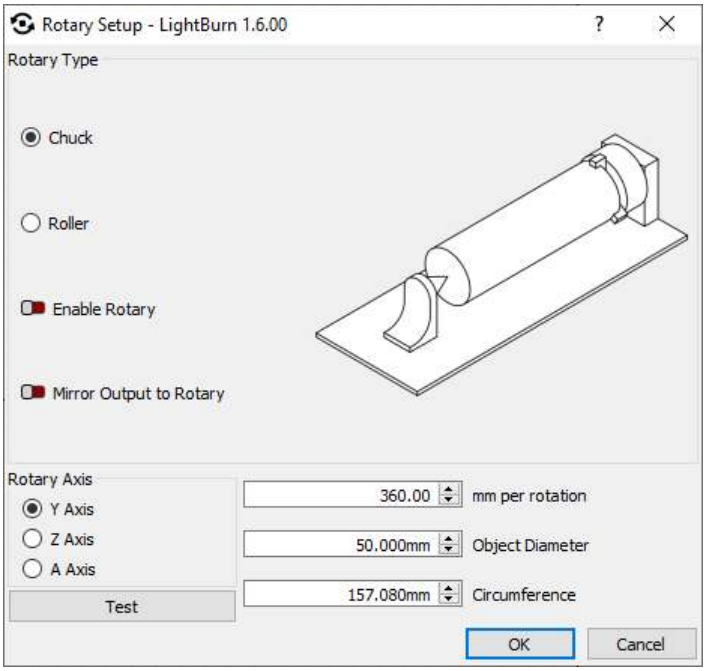
Rotaries are used to turn cylindrical objects so that their surfaces can be marked by a laser. In order to use your laser with a rotary, you must first establish several important parameters in the **Rotary Setup** window that control the rotary's motion.

> **Info**
>
> If you're working with a tapered object, use the largest end of the taper for the object diameter.

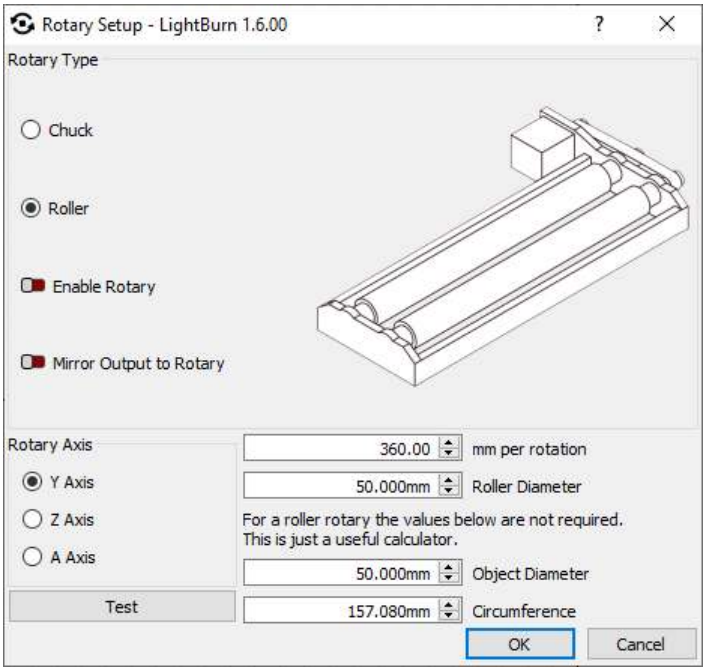
To open Rotary Setup, go to Laser Tools → **Rotary Setup** (`Ctrl`/`Cmd`+`Shift`+`R`), or click the rotary icon in the Modes Toolbar.

Rotary Setup

Click any option in the images below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Rotary Setup for GCode (Chuck)



Rotary Setup for GCode (Roller)

Rotary Type

Select whether your rotary is a **Chuck** or **Roller** style rotary. Chuck rotaries have adjustable jaws that grab and hold objects, while Roller rotaries have revolving rollers or wheels that objects are placed on top of.

Enable Rotary

> **Tip**
>
> If Show rotary enable in main window is enabled in your Settings/Preferences, you can also enable or disable Rotary Mode using the switch in the Laser Window.

This switch controls whether **Rotary Mode** is enabled. When enabled, rotational commands are sent to the laser to control rotary motion, and output is split into segments.

When Rotary Mode is enabled, you'll see a status indication in the bottom right corner of your screen:

Mirror Output to Rotary

Enable the **Mirror Output to Rotary** switch if output to the rotary is inverted in the dimension of rotation.

Rotary Axis

Select which axis LightBurn will output rotational commands to.

Select **Y Axis** if you are unplugging your Y Axis stepper motor wire and plugging it into the rotary instead. Some lasers use the Y Axis with a dedicated port and switch.

Select **Z Axis** or **A Axis** if you are plugging your rotary into a dedicated rotary axis. Consult your laser's manufacturer for help identifying the correct axis.

Rotary Settings

MM Per Rotation

> **Note**
>
> With a **Roller** style rotary, it is the roller or wheel driven by the motor that must make one complete rotation when you click the Test button, and not an object on the rotary.

Set the **mm per rotation** value to the distance of commanded movement in millimeters required to spin the rotary one complete rotation.

If this value is incorrect, output will be shrunk or stretched.

In most cases, mm per rotation will be provided by the manufacturer of your rotary or machine — consult the manufacturer if the mm per rotation value was not provided in the instructions included with your device.

Test

When the **Test** button is pressed, the rotary should turn a full 360 degrees, pause, then rotate back to the starting point. If the rotary turns too far, reduce the mm per rotation value. If it does not turn far enough, increase it.

Roller Diameter

This option is only presented for **Roller** style rotaries.

In most cases, **Roller Diameter** is provided by your rotary's manufacturer. If not, carefully measure the diameter of the roller or wheel driven by the rotary motor.

Once this measurement is accurately identified and set, it never needs to be adjusted again. If it is inaccurate, output will be shrunk or stretched.

Object Diameter

> **Tip**
>
> Use high precision calipers to guarantee the accuracy of your diameter measurement.

For **Chuck** style rotaries, carefully measure and enter the **Object Diameter** of the item on your rotary. This value must be updated every time you place an object with a different diameter on your rotary. If it is inaccurate, output will be shrunk or stretched.

For **Roller** style rotaries, Object Diameter does not affect output, but can be entered here to calculate the **Circumference** of your object.

> **Tip**
>
> Object **Circumference** and **Object Diameter** are linked — if you manually adjust the value for one measurement, LightBurn will automatically calculate the other.

If your object is tapered, check out our Taper Warp tool. Use the largest diameter of your object as the Object Diameter value.

Circumference

You can choose to directly measure and enter your object's **Circumference** using a tape measure. As with Object Diameter, this value must be as accurate as possible when using a **Chuck** style rotary, but does not affect output when using a **Roller Rotary**.

> **Tip**
>
> You can use the Circumference value when adjusting the size of your graphics. For full wrap designs, the combined height of your graphics should be identical to the Circumference.

> **Notes for Smoothieware Users**
>
> Before configuring the above, you may need to set up the rotary axis on your controller. LightBurn sends rotary moves as angle values, and the GCode controller translates those angles into actual movements. To do this, it needs to know the correct number of motor steps to take for one degree of movement on the rotary motor.
>
> If you have a Smoothieboard or other uncommon GCode-based controller, the number is usually 200 times your microstepping multiplier (usually 8 or 16) times any gear reduction, divided by 360.
>
> For a Cohesion3D or HolgaMods rotary, this is usually
>
> 3200 steps (200 x 8 micro steps x 2x reduction) / 360 = 8.88888888 steps per degree 6400 steps (400 x 8 micro steps x 2x reduction) / 360 = 17.7777778 steps per degree
>
> The steps per degree number, along with acceleration and maximum speed will need to be set in the appropriate location for the controller. With Smoothieware, those are the **delta** settings in the config.txt file on the controller.

Positioning

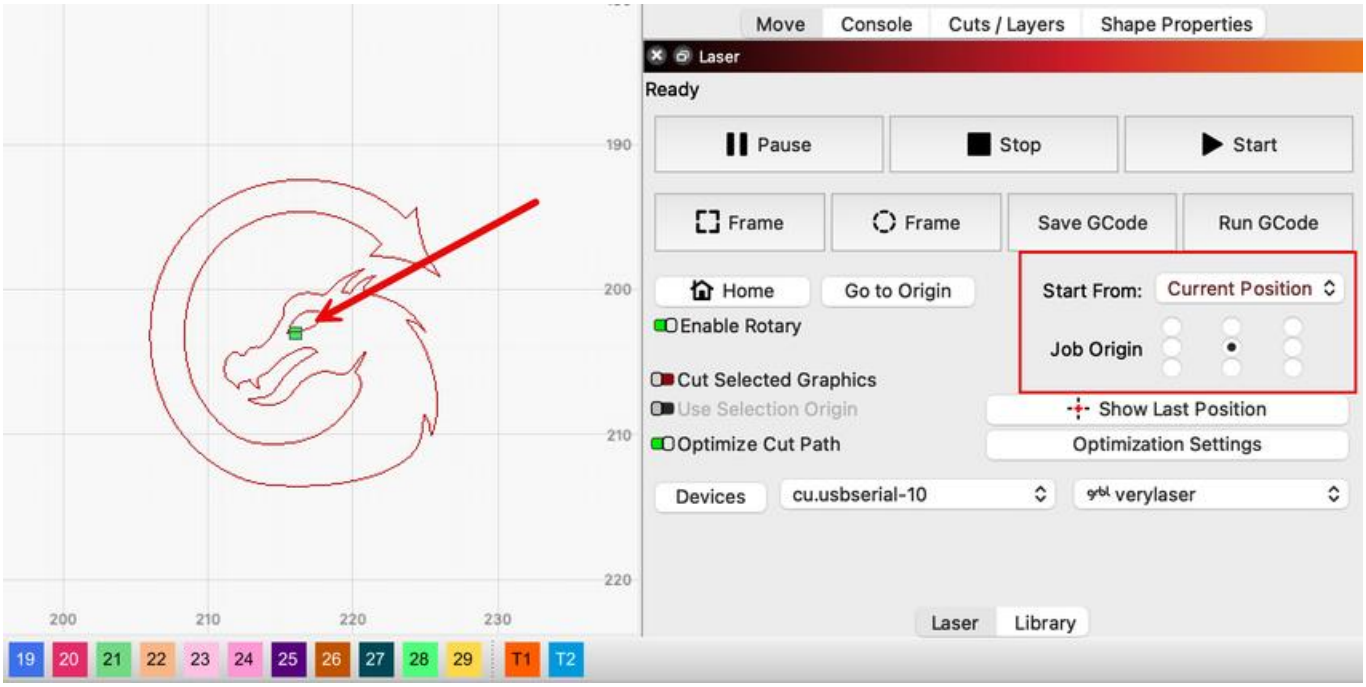
> **Tip**
>
> If your rotary is plugged into your laser's Y Axis, position the laser head toward the center of its work area before plugging in the rotary.

In most cases, Current Position is the best **Start From** mode to use when working with a rotary.

Align the object under your laser head in the X dimension. Output will be positioned on the object in the X dimension relative to the laser head and **Job Origin** selection. The Job Origin is indicated on your graphics by a green square. Parts of the graphic to left of the green square will output to the left of your laser head, and parts to the right will output to the right.

Turn the rotary or object on the rotary to determine where the engraving will output on the object in the dimension of rotation. Output in this dimension is also determined relative to your Job Origin selection — the rotary will turn to reach graphics positioned above or below the Origin.

See Coordinates and Job Origin for more information on the Start From mode and Job Origin settings.



In the above example, output will be centered around the starting position of the laser head.

Troubleshooting

- If your output is shrunk or stretched in the direction of rotation, either **mm per rotation**, **Object Diameter** (if working with a **Chuck** style rotary), or **Roller Diameter** (if working with a **Roller** style rotary), is incorrect.
  - Use the **Test** button in the **Rotary Setup** window to check mm per rotation. The rotary should make one 360 degree rotation before returning to the start. If you are working with a Roller rotary, it is the roller or wheel driven by the motor that must make one complete rotation and not an object on the rotary. If it turns too far or not far enough, adjust mm per rotation.
  - Use high precision calipers to double-check the Object or Roller Diameter value.
- If your controller returns an Alarm:2, Soft Limits error, enter `$20=0` in the Console Window to temporarily disable Soft Limits. When you are done working with the rotary, be sure to enter `20=1` to re-enable Soft Limits in your machine's firmware.

> **Additional Information**

> This error is due to the amount of commanded travel on the rotary axis exceeding the Max Travel limit for the Y Axis, as set in your laser's firmware. During flat engraving, this protects the machine from attempting to travel out of bounds and potentially damaging itself. Since the rotary can turn forever, it is ok to disable this setting when engraving with a rotary, but take care not to give the laser commands that could cause it to travel out of bounds in the X dimension.

- If the object slips on a Roller rotary, try:
  - Wrapping the rollers with a non-slip material, or even rubber bands.
  - Placing weights inside the object, such as ball bearings or other small round objects, to help press the object against the rollers.
  - Going to Laser Tools → Machine Settings and reducing the Acceleration value of your rotary axis, in your machine's firmware settings.
- If you end up with a flat part at the bottom of the engraving, it means your rotary gear mechanism has backlash. Usually, tightening any belts in the rotary device will fix this, but, if not, you can place a small line a few mm below the bottom of your design, set to very low power (0.1%) and set it as the first layer to run. The rotary will spin to this position, then back up over that few millimeters of gap, and will take up the backlash with that movement.

Related Topics

- Taper Warp

Rotary Mode (Galvo)

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=St8PWtp17K0

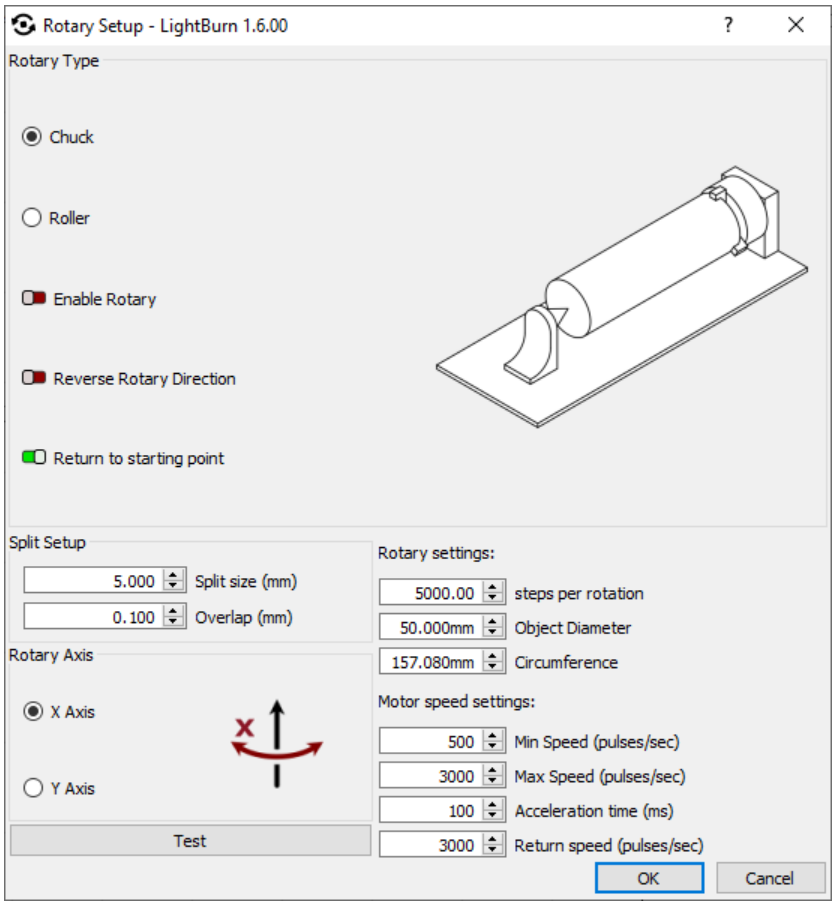
Rotaries are used to turn cylindrical objects so that their surfaces can be marked by a laser. In order to use your laser with a rotary, you must first establish several important parameters in the **Rotary Setup** window that control the rotary's motion.

> **Info**
>
> If you're working with a tapered object, use the largest end of the taper for the object diameter.

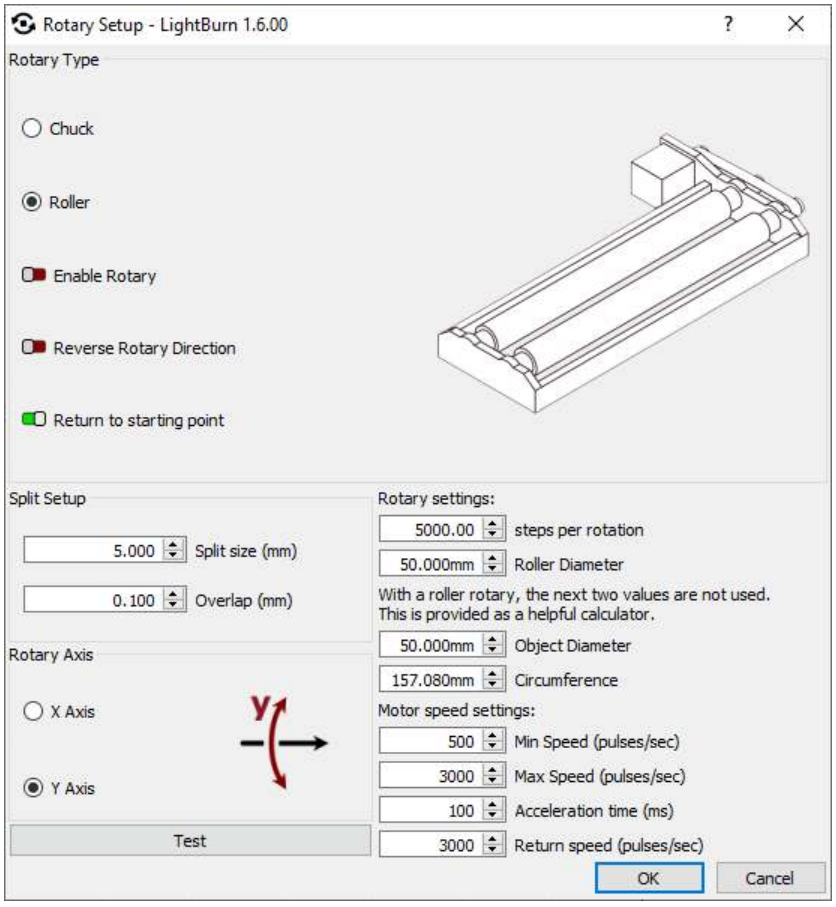
To open Rotary Setup, go to Laser Tools → **Rotary Setup** (`Ctrl`/`Cmd`+`Shift`+`R`), or click the rotary icon in the Modes Toolbar.

Rotary Setup

Click any option in the images below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Rotary Setup for Galvo (Chuck)



Rotary Setup for Galvo (Roller)

Rotary Type

Select whether your rotary is a **Chuck** or **Roller** style rotary. Chuck rotaries have adjustable jaws that grab and hold objects, while Roller rotaries have revolving rollers or wheels that objects are placed on top of.

Enable Rotary

> **Tip**
>
> If Show rotary enable in main window is enabled in your Settings/Preferences, you can also enable or disable Rotary Mode using the switch in the Laser Window.

This switch controls whether **Rotary Mode** is enabled. When enabled, rotational commands are sent to the laser to control rotary motion, and output is split into segments.

When Rotary Mode is enabled, you'll see a status indication in the bottom right corner of your screen:

Rotary Settings

Steps Per Rotation

> **Note**
>
> With a **Roller** style rotary, it is the roller or wheel driven by the motor that must make one complete rotation when you click the Test button, and not an object on the rotary.

Set the **steps per rotation** value to the number of motor steps required to spin the rotary one complete rotation.

If this value is incorrect, output will be warped or disconnected.

In almost all cases, steps per rotation will be provided by the manufacturer of your machine, either as a screenshot of the rotary params page in EZCAD or in a text document.

Common steps per rotation values for Galvo rotaries are 9,600, 12,800, and 25,600.

Test

When the **Test** button is pressed, it should rotate a full 360 degrees, pause, then rotate back to the starting point.

Roller Diameter

This option is only presented for **Roller** style rotaries.

In most cases, **Roller Diameter** is provided by your rotary's manufacturer. If not, carefully measure the diameter of the roller or wheel driven by the rotary motor.

Once this measurement is accurately identified and set, it never needs to be adjusted again. If it is inaccurate, output will be warped or disconnected.

Roller Diameter

This option is only presented for **Roller** style rotaries.

In most cases, **Roller Diameter** is provided by your rotary's manufacturer. If not, carefully measure the diameter of the roller or wheel driven by the rotary motor.

Once this measurement is accurately identified and set, it never needs to be adjusted again. If it is inaccurate, output will be warped or disconnected.

Object Diameter

> **Tip**
>
> Use high precision calipers to guarantee the accuracy of your diameter measurement.

For **Chuck** style rotaries, carefully measure and enter the **Object Diameter** of the item on your rotary. This value must be updated every time you place an object with a different diameter on your rotary. If it is inaccurate, output will be warped or disconnected.

For **Roller** style rotaries, Object Diameter does not affect output, but can be entered here to calculate the **Circumference** of your object.

> **Tip**
>
> Object **Circumference** and **Object Diameter** are linked — if you manually adjust the value for one measurement, LightBurn will automatically calculate the other.

If your object is tapered, check out our Taper Warp tool. Use the largest diameter of your object as the Object Diameter value.

Circumference

You can choose to directly measure and enter your object's **Circumference** using a tape measure. As with Object Diameter, this value must be as accurate as possible when using a **Chuck** style rotary, but does not affect output when using a **Roller Rotary**.

> **Tip**
>
> You can use the Circumference value when adjusting the size of your graphics. For full wrap designs, the combined height of your graphics should be identical to the Circumference, if the rotary is set to the **Y Axis**, or the width, if it is set to the **X Axis**. See below for information on Axis selection.

Split Setup

Split Size

When working with a rotary and Galvo laser, graphics are split into segments (slices). Each slice is marked on the object, then the rotary turns, the next slice is marked, and so on, until the entire graphic has been marked.

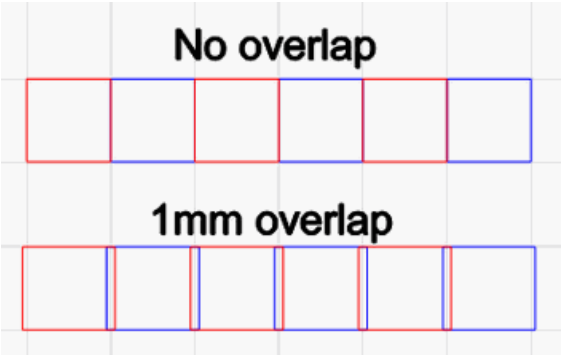
The size of each segment is determined by the **Split size** setting. If your object is tapered, irregularly shaped, or not perfectly aligned with the rotary axis, using a small Split size can help reduce gaps or misalignment of the splits.

A larger Split size will reduce the time spent running the job, but it can be harder to dial in the settings such that there are no overlaps or gaps.

Overlap

> **Tip**
>
> If you are removing paint, anodizing, or other surface coating, using a small overlap is recommended. If you are annealing or marking the material directly, adding overlaps may produce visible artifacts.

To help eliminate visible gaps between slices, you can tell LightBurn to overlap them. Enter an **Overlap** value to produce that much overlap on the ends of each slice, like this:



Rotary Axis

> **Tip**
>
> When rotating on the X Axis, you'll want to engrave along the Y Axis, with a Scan Angle of 90 or -90. When rotating on the Y Axis, you'll want to engrave along the X Axis, with a Scan Angle of 0 or 180.

Select the **X Axis** if your rotary is oriented across your laser's work area from top to bottom (or vice versa), and will be rotating in the direction of X Axis motion.

Select the **Y Axis** if your rotary is oriented across your laser's work area from right to left (or vice versa), and will be rotating in the direction of Y Axis motion.

Motor Speed Settings

The following settings control how fast the rotary moves and accelerates. In most cases, you do not need to adjust these settings from their default values.

Min Speed

The minimum speed at which in the rotary will rotate, in pulses/second.

> **Tip**
>
> If your rotary is losing steps, or an object on a Roller type rotary is slipping, reduce Max Speed and increase Acceleration time.

Max Speed

The maximum speed at which in the rotary will rotate, in pulses/second.

Acceleration Time

Time to accelerate the rotary to full speed, in milliseconds. Lower values mean the rotary will accelerate faster.

Return Speed

The speed at which the rotary will return to its starting position, in pulses/second.

Reverse Rotary Direction

Enable the **Reverse Rotary Direction** switch if output is backwards or sliced in the wrong order.

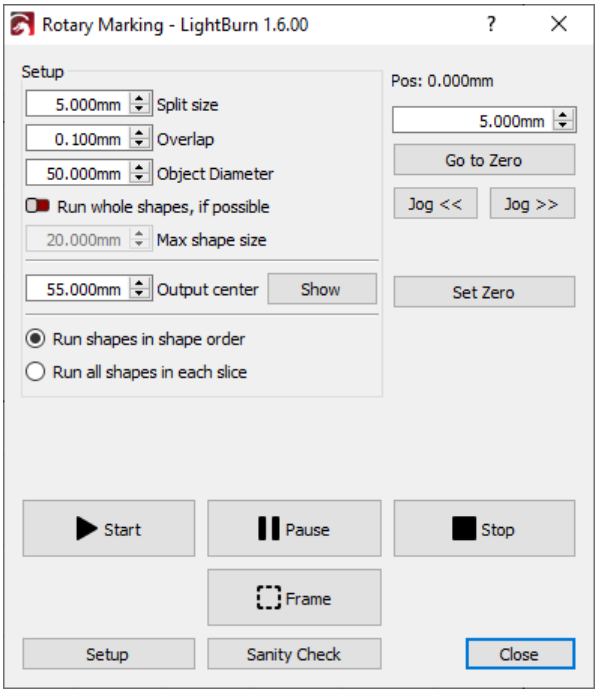
Return to Starting Point

Enable **Return to starting point** to have the rotary return to its starting point after the job is complete.

Rotary Marking Window

When Rotary Mode is enabled, pressing the **Start** button in the Laser Window will bring up the **Rotary Marking** window.

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Split Size and Overlap

Split size and **Overlap** in the Rotary Marking window are linked to the values set in **Rotary Setup** window. Changing either value in either window automatically changes it in both places.

You will see a warning message if you set Split size lower than the smallest possible Split size. Minimum split size is equal to the circumference of your object divided by **steps per rotation**. For a roller style rotary, it is the circumference of the roller divided by steps per rotation.

See above for more information on these settings.

Object Diameter (Rotary Marking Window)

Object Diameter is linked to the value set in the **Rotary Setup** window. Changing the value in either window will automatically update it in both places.

See above for more information.

This option is not presented in the Rotary Marking window when working with a Roller style rotary.

Run Whole Shapes, If Possible

Enabling **Run whole shapes if possible** tells LightBurn to create splits in such a way that shapes are not broken into segments. The rotary will jog to the center of each valid shape and mark it in its entirety. This can produce a higher quality finish without gaps or misalignment within shapes.

Max Shape Size

When **Run whole shapes if possible** is enabled, **Max Shape Size** determines the maximum size of objects that the laser will run in their entirety. Objects larger than the Max Shape Size will be split into segments.

> **Warning**
>
> If Max Shape Size is set too high, the laser's beam may go out of focus toward the edges of large shapes due to the curvature of the object on your rotary, and mark weakly, or not at all.

Output Center

Output will be centered on this line on the axis your rotary is set to. By default it is set to half of the maximum length of your Workspace, meaning output defaults to the center of the laser's field.

This setting allows you to shift the center position of your rotary output to compensate for the center of the rotary not lining up precisely with the center of the field. This value cannot be negative.

Show

The **Show** button commands your laser to **Frame** a line in the location **Output Center** is set to.

Run Shapes in Shape Order

Shapes will be run in their planned order, allowing the rotary to rewind. This option is best suited to Chuck style rotaries.

Run All Shapes in Each Slice

All shapes in each slice will be run before the rotary advances, preventing the rotary from rewinding. This option is best for Roller style rotaries, where objects may lose position due to rewinding.

> **Run shapes in shapes order vs. Run all shapes in each slice behavior comparison**
>
> Imagine your project has two layers, and you've selected **Order by Layer** in Optimization Settings.

If you select **Run shapes in shape order**, the rotary will run all splits containing the first layer, rewind back to the beginning, and then run all splits containing the second layer.

If you select **Run all shapes in each slice**, the rotary will run the first layer and then the second layer on each split, before advancing to the next split, until the job is complete.

Rotary Positioning

Position

The value displayed next to **Pos.** shows the current location of the rotary along the axis of rotation.

Go to Zero

Commands the rotary to return to the zero point on the axis of rotation.

Jog

The **Jog << / >>** buttons command the rotary to turn forward or backward in the increment set in the field above the **Go to Zero** button.

Set Zero

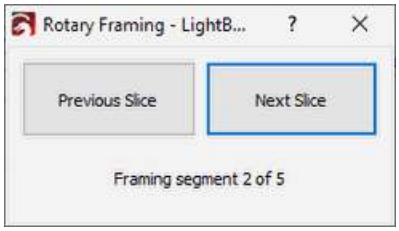
Sets the current position of the rotary as the zero point.

Start, Pause, Stop

These options work exactly as they do in the Laser Window during flat marking operations to begin (**Start**), temporarily pause (**Pause**) a job until your press **Resume**, or permanently halt (**Stop**) a job.

Frame

Frames individual slices that show where graphics will be engraved on the surface of your object.



Click **Previous Slice** and **Next Slice** to command the rotary to turn forward or backward to frame each slice in order.

The current segment being framed and total number of segments are displayed beneath the buttons.

Framing slice size

The size of slices when framing is distinct from the user-specified **Split size** used during engraving. Instead, the amount of slices is automatically determined based on the size of your object and the graphics you'll be engraving.

Larger objects have less curvature and therefore more area that the red dot can reach and adequately frame, often resulting in fewer framing slices for larger objects than smaller ones.

Setup

Opens the Rotary Setup window.

Sanity Check

Checks whether your settings make sense, including whether your **Scan Angle** is set to engrave along the dimension of rotation, or have enabled **Run whole shapes if possible** but selected **Fill all shapes at once** in the Cut Settings Editor.

Positioning Graphics

> **Tip**
>
> Enable Show work area center cross for a clear visualization of your Workspace's center lines.

When using a rotary with a Galvo device, the center line of your LightBurn Workspace represents the starting position of the rotary. If the rotary is set to the **Y Axis**, it is represented by the horizontal center line, and if it's set to the **X Axis**, it is represented by the vertical center line.

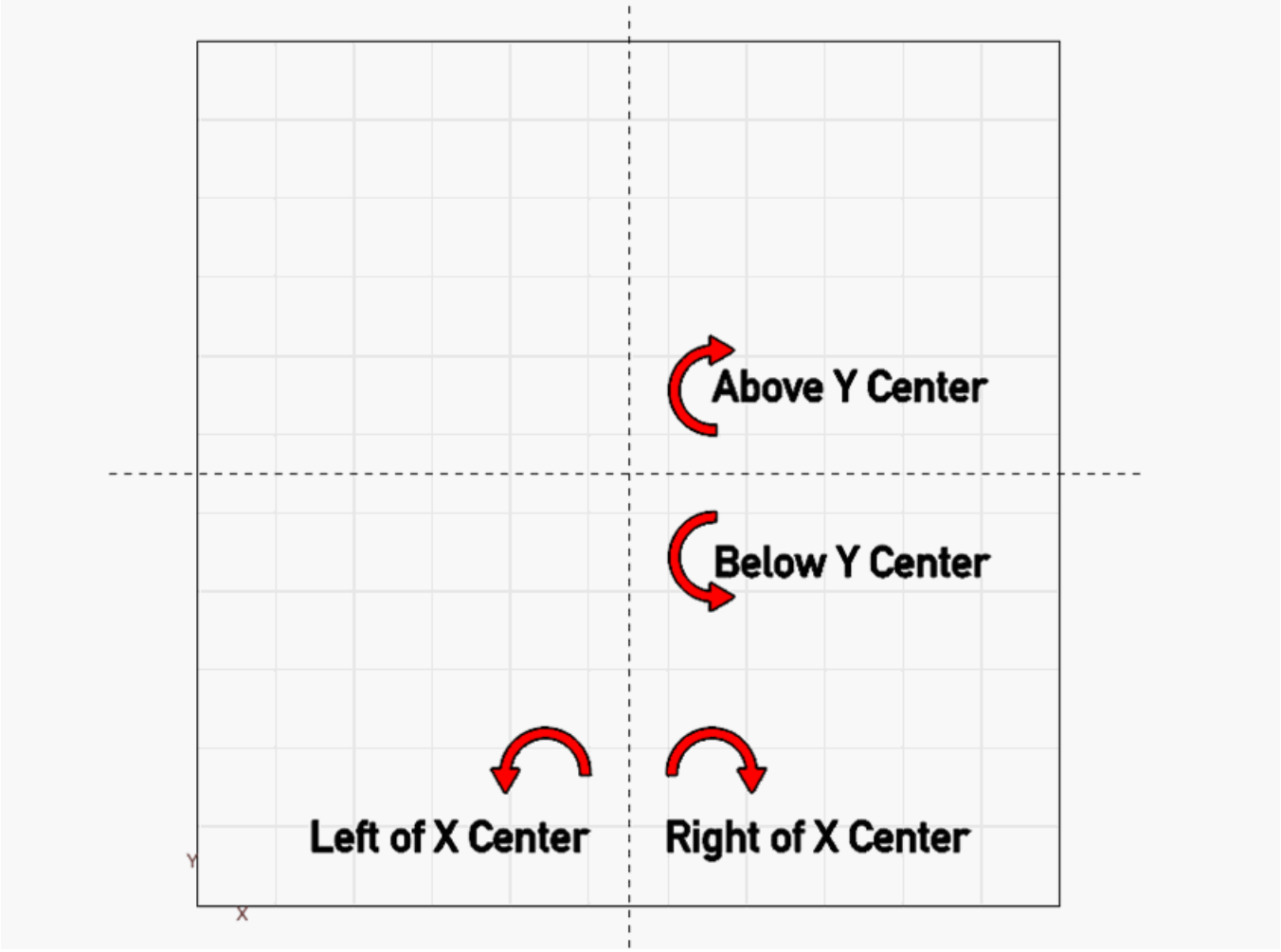
The farther away from the center line a graphic or part of a graphic is positioned in LightBurn, the more the rotary will turn to reach the location it will be marked on your object.

If graphics are straddling the center line, the rotary will first turn in one direction to reach the furthest extent of the graphics, then begin marking, eventually crossing the zero point to mark graphics positoned on the other side of the center line.

> **Tip**
>
> Use the **X** and **Y Position** fields in the Numeric Edits Toolbar to guarantee precise positioning.

To have the rotary begin marking without turning first, position your graphics so that their edge is just above or below (if the rotary is set to Y) or to the left or right (if the rotary is set to X) of the center line of your Workspace.

Graphics can be positioned as normal in the dimension the rotary is not set to rotate on, to adjust the location they will output to along the object you're marking.



Troubleshooting

- If output is stretched or there are gaps between slices, double check the **steps per rotation** and **Object Diameter** values. If you are using a Roller style rotary, check **Roller Diameter** instead of Object Diameter.
  - In the Rotary Setup window, make sure the rotary — and not an object on the rotary — makes a single 360 degree rotation before returning to the starting location when you click the **Test** button.
  - Use high precision calipers for best results when measuring the diameter of your object or roller.
- If output to your rotary is sent to the far left or bottom of your laser's work area, check to make sure the Output Center value is not set to 0. By default, it should be set to half of your Workspace's maximum width or height.
- If your graphics are only outputting as a line or narrow band on your object, switch to the alternate Rotary Axis setting, or change the physical orientation of your rotary to match the current setting.

- If output to your rotary is backwards or sliced in the wrong order, enable Reverse Rotary Direction.

Related Topics

- Cylinder Correction
- Taper Warp
- Repeat Marking

Repeat Marking

The **Repeat Marking** tool enables the use of a **Rotating** or **Linear** axis table with Galvo lasers, to move parts past a laser while running repetitions of identical jobs.

By specifying an **Increment** of movement as well as a desired number of repetitions, or **Count**, you can command your laser to mark a design, move an external axis by the specified increment, then run the project again, and repeat that process until it has completed the set number repetitions.

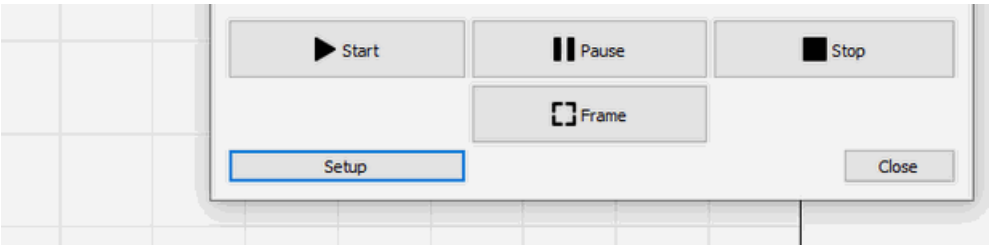
To open the Repeat Marking tool, go to Laser Tools → **Repeat Marking**.

> **Warning**
>
> The Repeat Marking tool is separate from, and incompatible with, Rotary Mode. When using Repeat Marking, make sure that Rotary Mode is disabled.

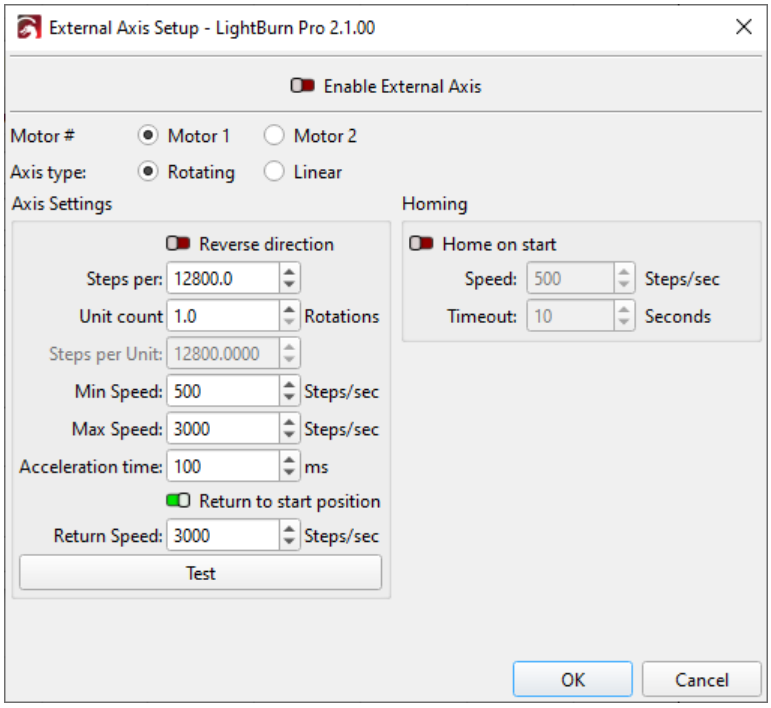
External Axis Setup

Before you use **Repeat Marking** for the first time, you must configure the settings for the **Rotating** or **Linear** axis table connected to your laser.

Click the **Setup** button in the Repeat Marking window to open the **External Axis Setup** window.



Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Motor

If your laser has two motor outputs, select which is connected to your **Rotating** or **Linear** table.

Axis Type

Select whether your device is a **Rotating** table that can make continuous, complete revolutions, or a **Linear** table that moves back and forth along an axis.

Axis Settings

Orientation

If your Axis Type is set to **Linear**, select whether it is orientated along your work area's **X** or **Y Axis**.

Reverse Direction

Enable **Reverse Direction** to tell your device to travel in the opposite direction it travels by default, without changing its orientation or rewiring it.

Steps Per

This value is the number of pulses required for the stepper motor to make one complete revolution.

Your table's manufacturer should provide you with the correct value for your device, or you can access its stepper driver and read or adjust the value there.

Unit Count

The amount of movement of the table that results from one complete rotation of the stepper motor.

- For **Rotating** tables the unit of movement is **Rotations**.
  - For direct-driven tables, this value should be set to 1.

    - For belt-driven tables, the value may vary based on gear ratio.

- For **Linear** tables the unit of movement is **mm**.
    - This value should be set to match the amount of linear movement of your table that results from one complete revolution of its stepper motor.

Your table's manufacturer should be able to provide you with the correct value for your device — if you're unsure consult the documentation for your device, or contact the manufacturer.

Steps Per Unit

This value is automatically calculated by dividing your Steps per value by your Unit Count.

The calculated value equals the number of steps required to move your table in a single increment of your **Unit**:

- 1 complete rotation for **Rotating** tables
- 1 mm for **Linear** tables

Min Speed

The minimum speed at which in the table will travel, in **Steps/second**.

Max Speed

The maximum speed at which in the table will travel, in **Steps/second**.

Acceleration Time

Time to accelerate the table to full speed, in milliseconds. Lower values mean the table will accelerate faster.

Return to Start Position

Enable **Return to starting point** to have the table return to its starting point after the job is complete.

Return Speed

The speed at which the table will return to its starting position, in **Steps/second**.

Test

For **Rotating** tables, when the **Test** button is pressed, and both Steps per and Unit Count are correct, the table will rotate a full 360 degrees, pause, then rotate back to the starting point.

For **Linear** tables, when the **Test** button is pressed, and both Steps per and Unit Count are correct, the table will move 50 mm, then back to its starting point.

Homing

Home on Start

Enable **Home on start** to have LightBurn send a command to home a **Linear** table at the start of a job.

Note that homing may require special wiring — do not enable this setting if you are not sure if your table is wired for homing.

Speed

The speed at which the table will home.

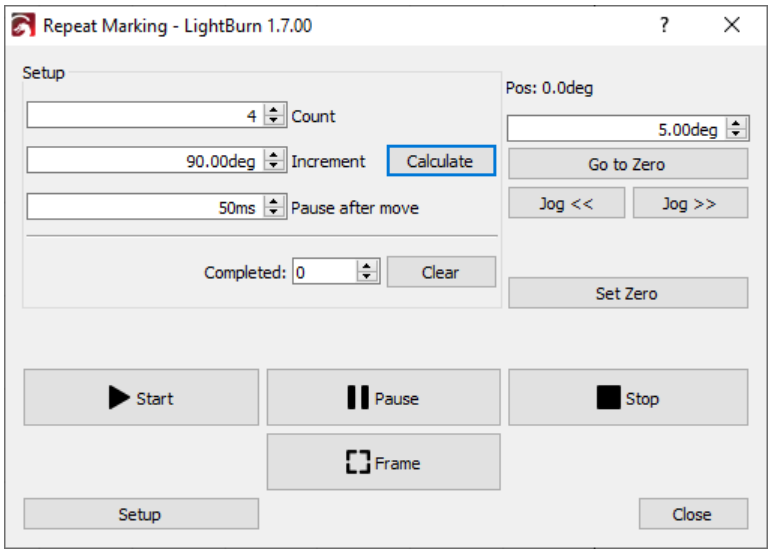
Timeout

Amount of time before the table will abort the homing process.

Repeat Marking Window

Use the **Repeat Marking** window to jog your table into its starting position, set the number of repetitions it will run, and determine the amount of movement between each repitition.

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Count

The number of times (repetitions) your laser will mark a given set of graphics.

Your laser will mark a single set of graphics once, then the table will rotate or move, and the laser will mark the same set of graphics again, and so on until it reaches the **Count** you've set.

Increment

For **Rotating** tables, the angle, in degrees, to rotate the axis between repetitions.

For **Linear** tables, the distance, in millimeters, to move the axis between repetitions.

Calculate

The **Calculate** button is only available if the axis type in **External Axis Setup** is set to **Rotating**, and can automatically determine the correct rotation **Increment** based on the number of positions in your Rotating table. To use it, set **Count** to the number of positions in your table, then click Calculate.

Afterward, you can change the Count to any other number of repetitions, but make sure not to click Calculate again, or the correct Increment value will be lost.

Pause After Move

If your **Rotating** table wobbles when it stops, adding a pause here gives it time to settle before marking starts again.

Completed

The total number of runs of a given project, from the time you first open the **Repeat Marking** window, until you close it. Use this number to track the total number of parts you have marked.

Clear

Resets the **Completed** count to zero.

Positioning

Position

The value displayed next to **Pos.** shows the current location of the **Rotating** or **Linear** table.

Go to Zero

Commands the table to return to the zero point on the axis.

Jog

The **Jog << / >>** buttons command the table to turn (or move) forward or backward in the increment set in the field above the **Go to Zero** button.

Set Zero

Sets the current position of the table as the zero point.

Start, Pause, Stop

These options work exactly as they do in the Laser Window during regular marking operations to begin (**Start**), temporarily pause (**Pause**) a job until your press **Resume**, or permanently halt (**Stop**) a job.

Frame

Framing works as it does with regular marking operations.

> **Tip**
>
> The **Rotating** or **Linear** table will not jog to Frame each repetition — to verify that your **Increment** and **Steps per rotation** or **Steps per mm** values are correct before running a job, try Jogging the table by the amount you've set for your Increment, and Framing multiple positions on the table.

Setup

Click the **Setup** button to open the External Axis Setup window and configure the settings for your table table.

Related Topics

- Rotary Mode (Galvo)
- Variable Text

Split Marking

Split Marking enables the ability to mark designs larger than a Galvo laser's work area using a **Rotating** or **Linear** axis table.

By specifying a Split Size, you can tell LightBurn to divide your design into segments, then mark individual segments between movements of the external axis, continuing until the entire design has been marked.

To configure your external axis' motion parameters and enable the Split Marking tool, click the conveyor icon in the Modes Toolbar.

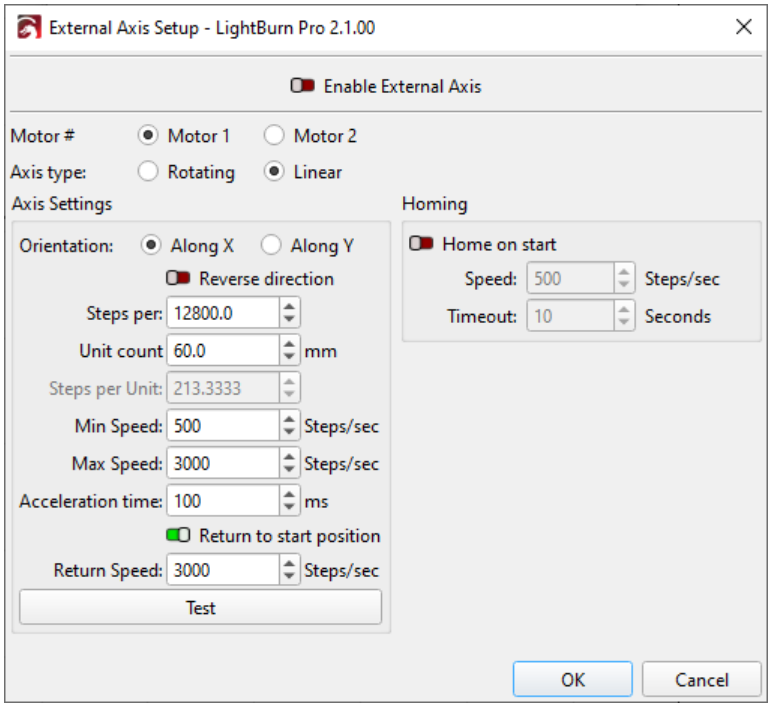
Split Marking Setup

Before you use **Split Marking** for the first time, you must configure the settings for the **Rotating** or **Linear** axis table connected to your laser.

Click the **Conveyor** icon in the Modes Toolbar to open the **Split Marking Setup** window.

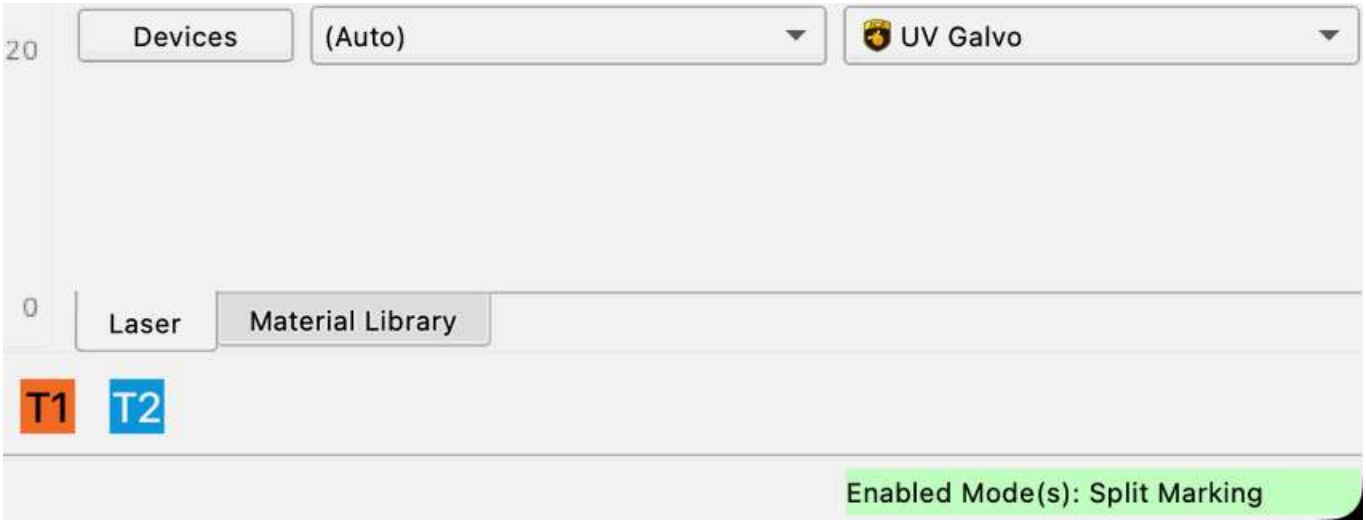


Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Enable Split Marking

This switch controls whether **Split Marking Mode** is enabled. When enabled, you'll see a status indication in the bottom right corner of your screen:



Motor

If your laser has two motor outputs, select which is connected to your **Rotating** or **Linear** table.

Axis Type

Select whether your device is a **Rotating** table that can make continuous, complete revolutions, or a **Linear** table that moves back and forth along an axis.

Axis Settings

Orientation

If your Axis Type is set to **Linear**, select whether it is orientated along your work area's **X** or **Y Axis**.

Reverse Direction

Enable **Reverse Direction** to tell your device to travel in the opposite direction it travels by default, without changing its orientation or rewiring it.

Steps Per

This value is the number of pulses required for the stepper motor to make one complete revolution.

Your table's manufacturer should provide you with the correct value for your device, or you can access its stepper driver and read or adjust the value there.

Unit Count

The amount of movement of the table that results from one complete rotation of the stepper motor.

- For **Rotating** tables the unit of movement is **Rotations**.
  - For direct-driven tables, this value should be set to 1.
  - For belt-driven tables, the value may vary based on gear ratio.

- For **Linear** tables the unit of movement is **mm**.
  - This value should be set to match the amount of linear movement of your table that results from one complete revolution of its stepper motor.

Your table's manufacturer should be able to provide you with the correct value for your device — if you're unsure consult the documentation for your device, or contact the manufacturer.

Steps Per Unit

This value is automatically calculated by dividing your Steps per value by your Unit Count.

The calculated value equals the number of steps required to move your table in a single increment of your **Unit**:

- 1 complete rotation for **Rotating** tables
- 1 mm for **Linear** tables

Min Speed

The minimum speed at which in the table will travel, in **Steps/second**.

Max Speed

The maximum speed at which in the table will travel, in **Steps/second**.

Acceleration Time

Time to accelerate the table to full speed, in milliseconds. Lower values mean the table will accelerate faster.

Return to Start Position

Enable **Return to starting point** to have the table return to its starting point after the job is complete.

Return Speed

The speed at which the table will return to its starting position, in **Steps/second**.

Test

For **Rotating** tables, when the **Test** button is pressed, and both Steps per and Unit Count are correct, the table will rotate a full 360 degrees, pause, then rotate back to the starting point.

For **Linear** tables, when the **Test** button is pressed, and both Steps per and Unit Count are correct, the table will move 50 mm, then back to its starting point.

Homing

Home on Start

Enable **Home on start** to have LightBurn send a command to home a **Linear** table at the start of a job.

Note that homing may require special wiring — do not enable this setting if you are not sure if your table is wired for homing.

Speed

The speed at which the table will home.

Timeout

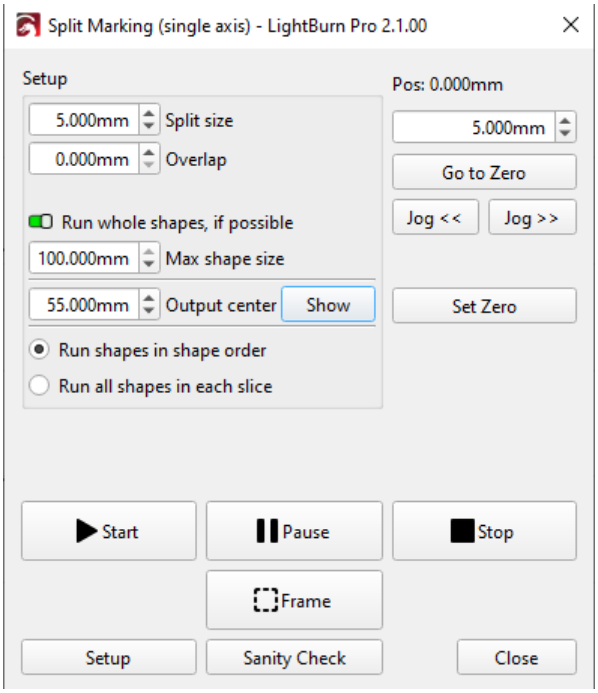
Amount of time before the table will abort the homing process.

Split Marking Window

When **Split Marking** is enabled, pressing the **Start** button in the Laser Window will bring up the Split Marking window.

Use the Split Marking window to jog your table into its starting position and establish the settings that determine how LightBurn will segment your graphics and move your table during marking.

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Split Setup

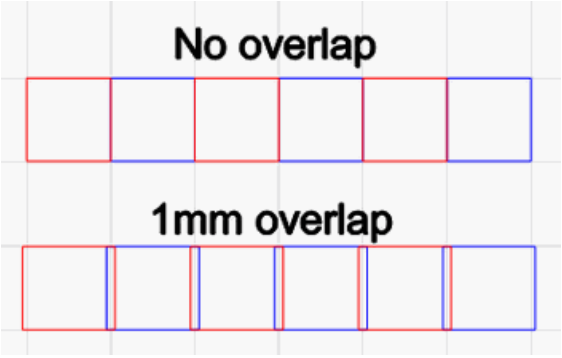
Split Size

In Split Marking mode, graphics are divided into slices (segments). Each slice is marked on the object, the table advances, the next slice is marked, and so on, until the entire graphic has been marked.

The size of each segment is determined by the **Split size** setting. A larger Split size will reduce the time spent running the job, but it can be harder to dial in the settings such that there are no overlaps or gaps.

Overlap

To help eliminate visible gaps between slices, you can tell LightBurn to overlap them. Enter an **Overlap** value to produce that much overlap on the ends of each slice, like this:



Run Whole Shapes, If Possible

Enabling **Run whole shapes if possible** tells LightBurn to create splits in such a way that shapes are not broken into segments. The table will jog to the center of each valid shape and mark it in its entirety. This can produce a higher quality finish without gaps or misalignment within shapes.

Max Shape Size

When **Run whole shapes if possible** is enabled, **Max Shape Size** determines the maximum size of objects that the laser will run in their entirety. Objects larger than the Max Shape Size will be split into segments.

Output Center

Output will be centered on this line on the axis your table is set to. By default it is set to half of the maximum length of your Workspace, meaning output defaults to the center of the laser's field.

This setting allows you to shift the center position of your rotary output to compensate for the center of the rotary not lining up precisely with the center of the field. This value cannot be negative.

Show

The **Show** button commands your laser to **Frame** a line in the location **Output Center** is set to.

Run Shapes in Shape Order

Shapes will be run in their planned order, allowing the table to both advance and reverse.

Run All Shapes in Each Slice

All shapes in each slice will be run before the table advances, preventing the table from reversing.

Run shapes in shapes order vs. Run all shapes in each slice behavior comparison

Imagine your project has two layers, and you've selected **Order by Layer** in Optimization Settings.

If you select **Run shapes in shape order**, the laser will mark all splits containing the first layer, then reverse the table back to the beginning, and then run all splits containing the second layer.

If you select **Run all shapes in each slice**, the laser will run the first layer and then the second layer on each split, before advancing to the next split, until the job is complete.

Table Positioning

Position

The value displayed next to **Pos.** shows the current location of the table along the external axis.

Go to Zero

Commands the table to return to the zero point on the linear or rotating axis.

Jog

The **Jog << / >>** buttons command the table to advance forward or backward in the increment set in the field above the **Go to Zero** button.

Set Zero

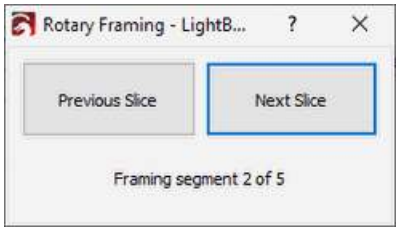
Sets the current position of the table as the zero point.

Start, Pause, Stop

These options work exactly as they do in the Laser Window during standard marking operations to begin (**Start**), temporarily pause (**Pause**) a job until your press **Resume**, or permanently halt (**Stop**) a job.

Frame

Frames individual slices that show where graphics will be engraved on the surface of your object.



Click **Previous Slice** and **Next Slice** to command the table to jog forward or backward to frame each slice in order.

The current segment being framed and total number of segments are displayed beneath the buttons.

> **Framing slice size**
>
> The size of slices when framing is distinct from the user-specified **Split size** used during engraving.

Setup

Opens the Split Marking Setup window.

Sanity Check

Checks whether your settings make sense, including whether you have enabled **Run whole shapes if possible** but selected **Fill all shapes at once** in the Cut Settings Editor.

Related Topics

- Rotary Mode (Galvo)
- Repeat Marking

Feeder Setup

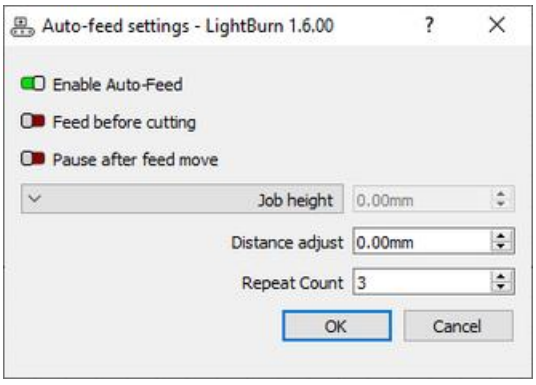
Auto-feeders are conveyers that feed materials such as fabric or leather into a laser in stepped increments. The laser runs repetitions of the same engraving or cutting job between each increment.

Go to Laser Tools → **Feeder Setup** (`Ctrl`/`Cmd`+`Shift`+`F`) to open the **Auto-feed settings** window, where you can set the parameters by which your auto-feeder will feed material into your laser, and the laser will run repeat jobs.

> **Note**
>
> Feeder Setup is only available for DSP lasers with Ruida or Trocen controllers.

Auto-Feed Settings

Ruida



- **Enable Auto-Feed** turns on Auto-Feed mode. This setting must be enabled in order to operate your auto-feeder.
- **Feed before cutting** commands the feeder to move material into the laser before the laser performs its first run of the job.
- **Pause after feed move** tells the laser to pause after a feed move, before cutting. This option is not available on all controllers.
- There are three options for feed distance in the drop-down menu. Feed distance controls the amount that the feeder will advance your material with each repetition.
  - **Job Height**: the feeder will advance the same distance as the height of the job you send to the laser.

    > **Tip**
    >
    > You can see your job's height in the Numeric Edits Toolbar after selecting all graphics you'll be outputting to the laser.

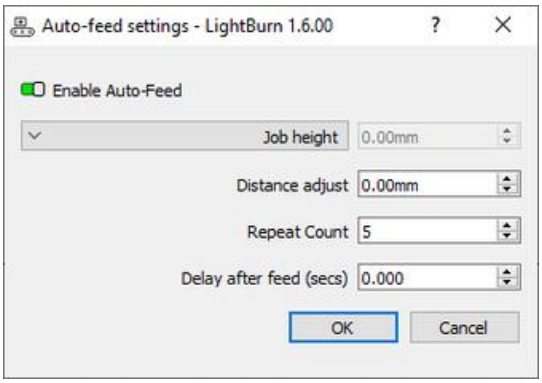
  - **Y Axis length**: the feeder will advance the same distance as the length of your laser's bed in the Y dimension.

- **Manual**: the feeder will advance by the amount you specify in the field next to the drop-down menu.

- **Distance Adjust** offsets the feed distance by the amount entered. For example, with Job Height selected, and a project height of 50 mm, entering 5 mm in the Distance Adjust field will command the feeder to advance 55 mm with each repetition. Offsets can also be negative.
- **Repeat Count** determines the number of times the laser will run the job and the feeder will advance.
- Press **OK** to save Auto-feed settings or **Cancel** to discard any settings changes made and restore to previous settings.

> **Warning**
>
> Be sure to click OK to save settings after enabling or disabling Auto-Feed mode, or adjusting any settings. Cancelling or "X-ing" out of the Auto-feed settings window will not apply the changes you've made.

Trocen



- **Enable Auto-Feed** turns on Auto-Feed mode. This setting must be enabled in order to operate your auto-feeder.
- There are three options for feed distance in the drop-down menu. Feed distance controls the amount that the feeder will advance your material with each repetition.
  - **Job Height**: the feeder will advance the same distance as the height of the job you send to the laser.

    > **Tip**
    >
    > You can see your job's height in the Numeric Edits Toolbar after selecting all graphics you'll be outputting to the laser.

  - **Y Axis length**: the feeder will advance the same distance as the length of your laser's bed in the Y dimension.
  - **Manual**: the feeder will advance by the amount you specify in the field next to the drop-down menu.

- **Distance Adjust** offsets the feed distance by the amount entered. For example, with Job Height selected, and a project height of 50 mm, entering 5 mm in the Distance Adjust field will command the feeder to advance 55 mm with each repetition. Offsets can also be negative.
- **Repeat Count** determines the number of times the laser will run the job and the feeder will advance.
- **Delay after feed (secs)** tells the laser to pause for the specified amount of time after each feed.
- Press **OK** to save Auto-feed settings or **Cancel** to discard any settings changes made and restore to previous settings.

> **Warning**
>
> Be sure to click OK to save settings after enabling or disabling Auto-Feed mode, or adjusting any settings. Cancelling or "X-ing" out of the Auto-feed settings window will not apply the changes you've made.

Related Topics

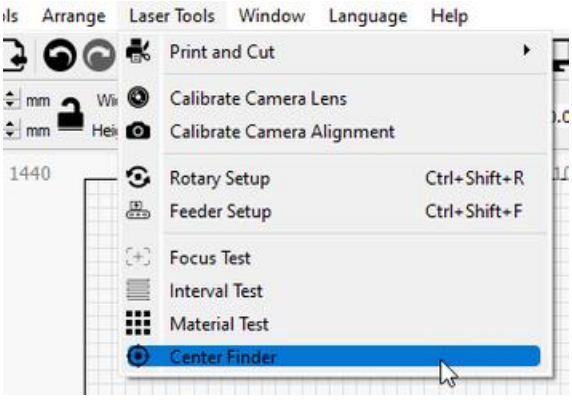
- Rotary Mode (DSP)

Center Finder

The **Center Finder** is a tool for finding the center of circular stock, as well as defining that stock in your LightBurn Workspace.

By jogging your laser to three positions on a circle (or other regular shape), and setting those positions in the **Center Finder Wizard**, LightBurn can calculate the circle that passes through those three points, jog your laser to its center, and create a circle in your LightBurn Workspace matching those dimensions, to represent your stock.

The Center Finder is available in the Laser Tools Menu.



Using the Center Finder

First Steps

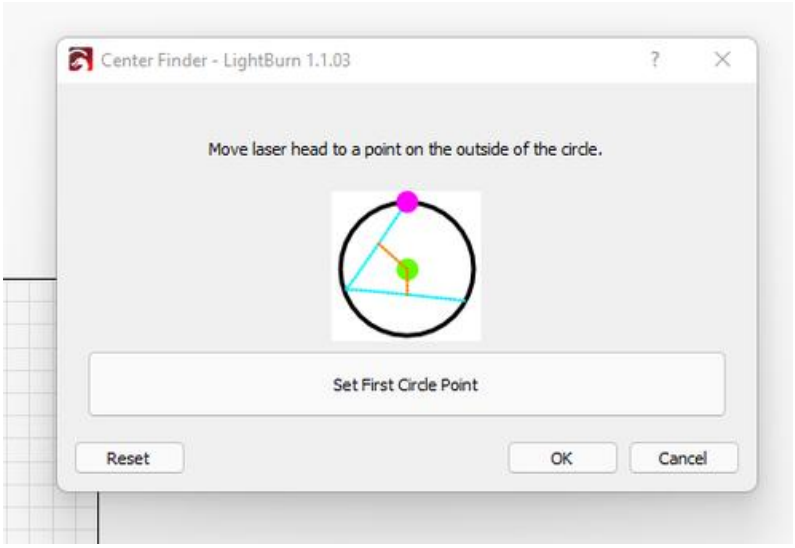
- Home your machine.
- If your machine does not have homing switches, make sure to start the machine with the laser head at the 0,0 point of its work area — that is typically the lower left corner for GCode-based devices.
- If the stock you are referencing is light or can be moved easily, make sure to secure it so it does not get bumped during the operations.

Set the Points

Point 1

Jog the laser head to the first position on the edge of your circle, as shown in the wizard, then click **Set First Circle Point**.

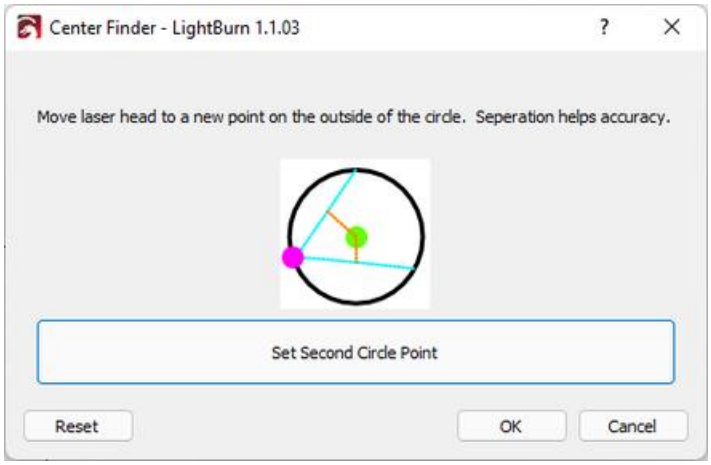
> **Warning**
>
> You must jog the laser head with the laser control panel or from the Move Window. **Center Finder** will not work if you move the laser head by hand.





Point 2

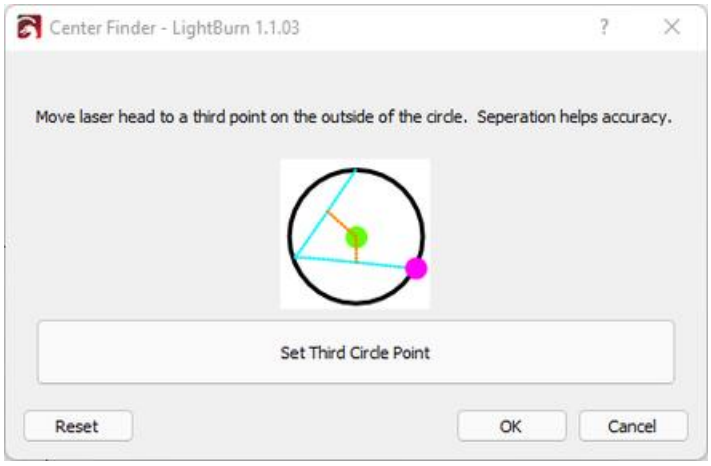
Jog the laser head to the next position on the edge of your circle, as shown in the wizard, then click **Set Second Circle Point**.

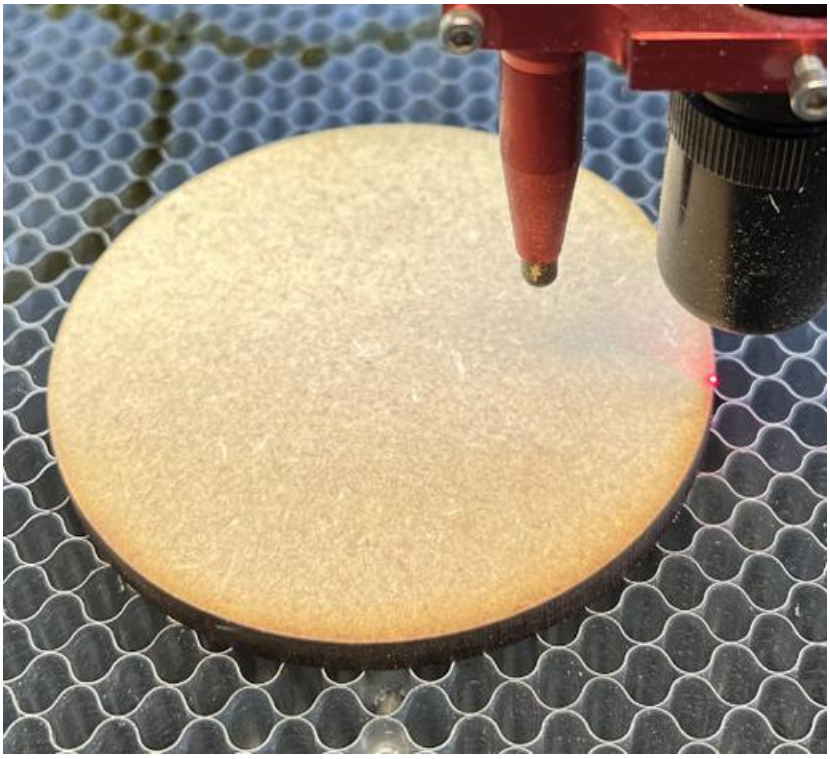




Point 3

Jog the laser head to the last position on the edge of your circle, as shown in the wizard, then click **Set Third Circle Point**.

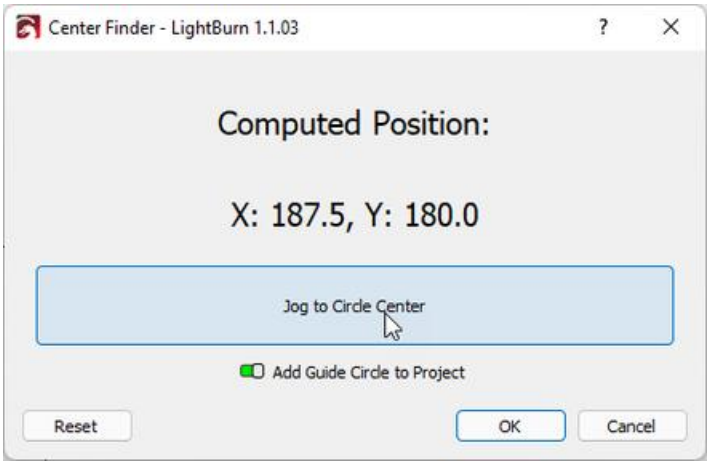


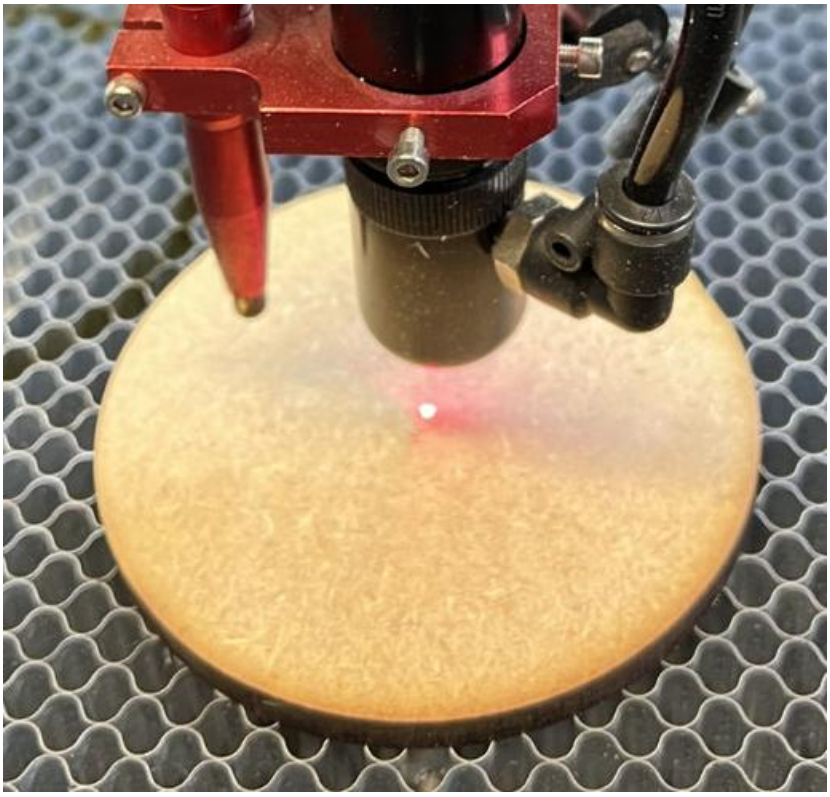


Align to Center

Move to Center

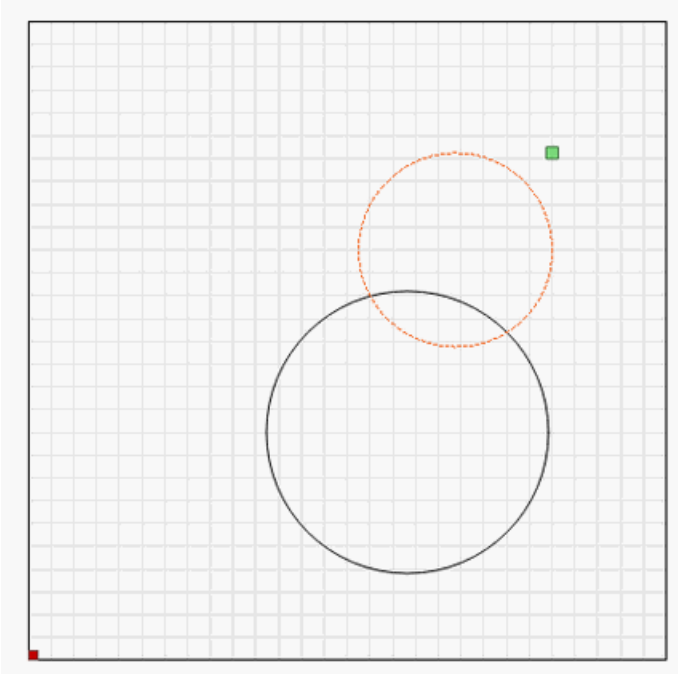
The last step will show you the calculated center of your circle and allow you to jog to that point.





Create Guide Circle

Toggle on **Add Guide Circle to Project** to create a circle on the T1 Tool layer, that will provide a representation of your stock in your LightBurn Workspace.



Set Origin

Absolute Coordinates

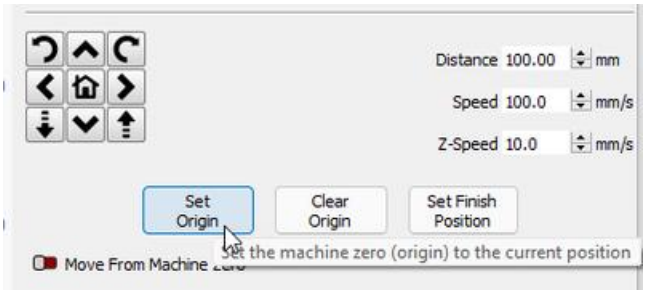
If you use Absolute Coordinates for your workflow, move your design to the guide circle, and make sure your **Start From** mode is set to **Absolute Coords**.

User Origin — Center

If you use User Origin make sure the center radio button is selected as **Job Origin**.

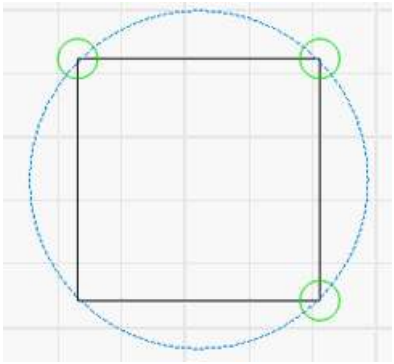


After you move the laser to the center of your stock, press the **Origin** button on your DSP control panel or the **Set Origin** button in the Move Window.

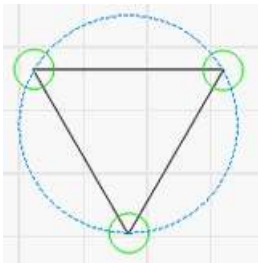


Notes

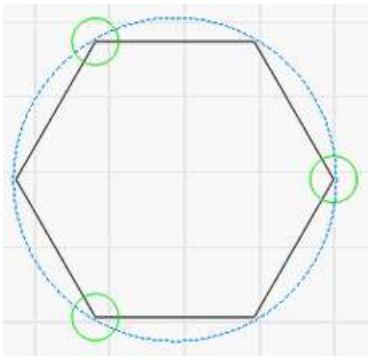
- You can set the points in any order, and they do not need to be positioned exactly as shown in the reference photos — they need only be far enough apart from one another along the perimeter of the shape that they are valid for calculating the center. The reference photos are meant as guidelines for acceptable distance.
- This tool can find the center of any shape that can be described with a circle diameter:
  - Squares (pick 3 corners)



- Equilateral Triangles

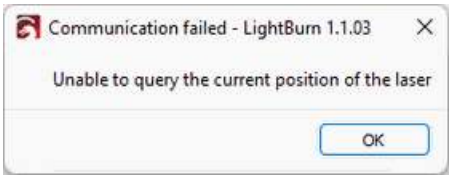


- Hexagons



Error Messages

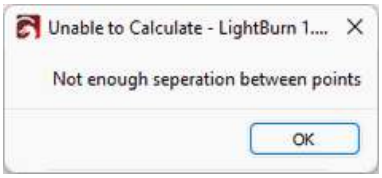
Unable to Query the Current Position of the Laser



This happens when LightBurn is unable to get the position of the laser from the controller.

- Often, the laser is still moving, or another command is still in process. Click **OK**, and try again after waiting a few seconds.
- If the warning persists, make sure the controller is still connected and is showing **Ready** at the top of the Laser Window. If you are unable to connect, see Troubleshooting: Connection Problems.

Not Enough Separation Between Points



You have not moved the laser away from the last point or have not moved it far enough to make an acceptable calculation.

- Try moving farther away and setting the point again.
- You must jog the laser using its control panel or the Move Window, and not by hand.

Unable to Calculate Center

LightBurn can't calculate center, or it would be outside your laser's work area.

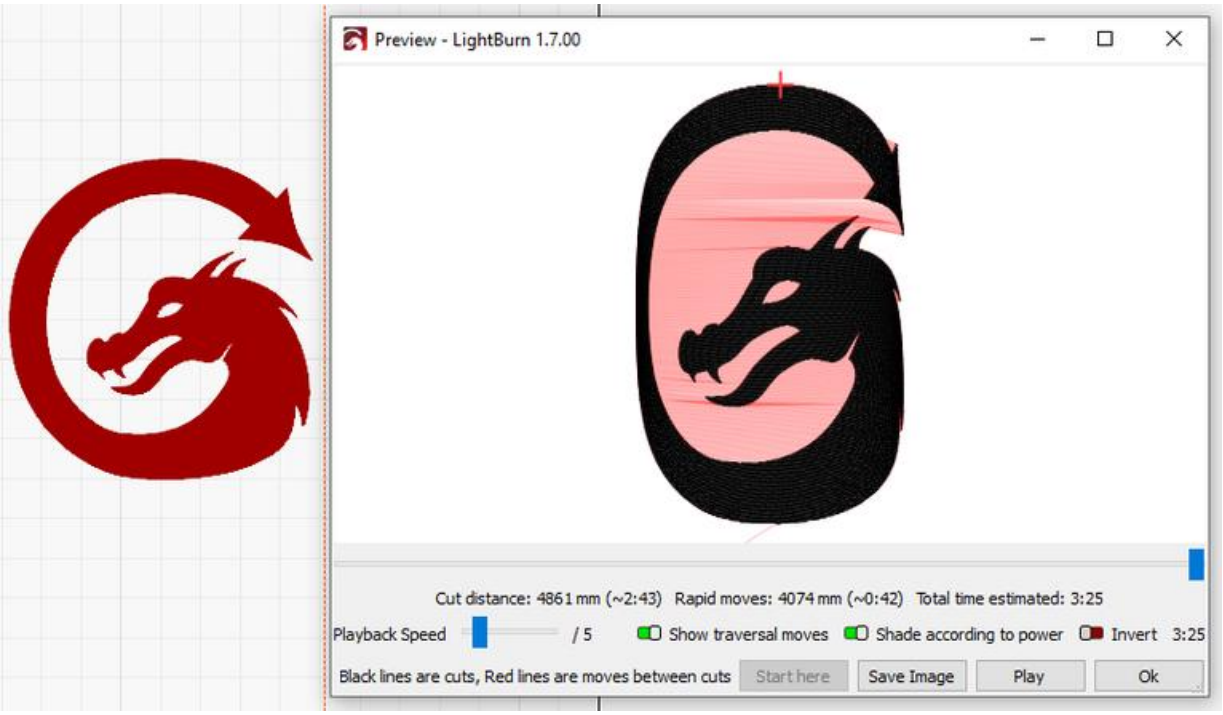
- This typically happens on machines without homing switches that were not manually homed properly.
- Restart your machine with the head at your machine origin and try to run the wizard again.

Troubleshooting

- The accuracy of the **Center Finder** depends on the accuracy of your point placement. If you have a red dot laser that is accurate, one trick is to try to split the dot on the edges of your stock.
- If you set a point incorrectly, hit the **Reset** button. This will clear all points saved. After, restart from the current point. You can set the points in any order.

Cylinder Correction

Cylinder Correction warps the output of your design to correct for distortion caused by engraving cylindrical objects, without using a rotary. This tool is currently only available for Galvo lasers. EZCAD users may be familiar with this tool by the name ProjectMark.

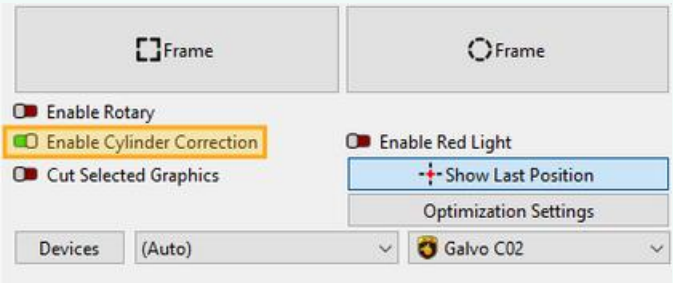


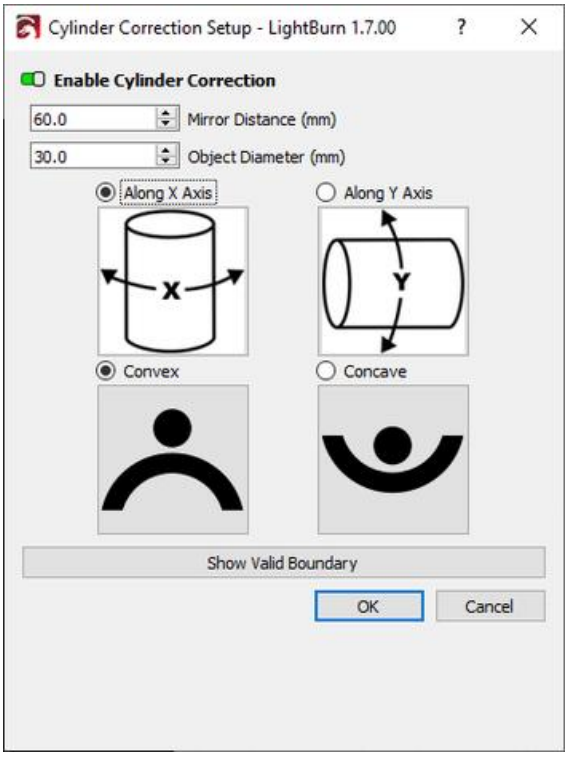
Using Cylinder Correction

Open the **Cylinder Correction Setup** window by going to Laser Tools → **Cylinder Correction Setup**, or clicking the Cylinder Correction button in the Modes Toolbar.

Make sure **Enable Cylinder Correction** is turned on.

> **Tip**
>
> If Show cylinder correction enable in main window is enabled in your Settings/Preferences, you can also enable or disable Cylinder Correction Mode using the switch in the Laser Window.





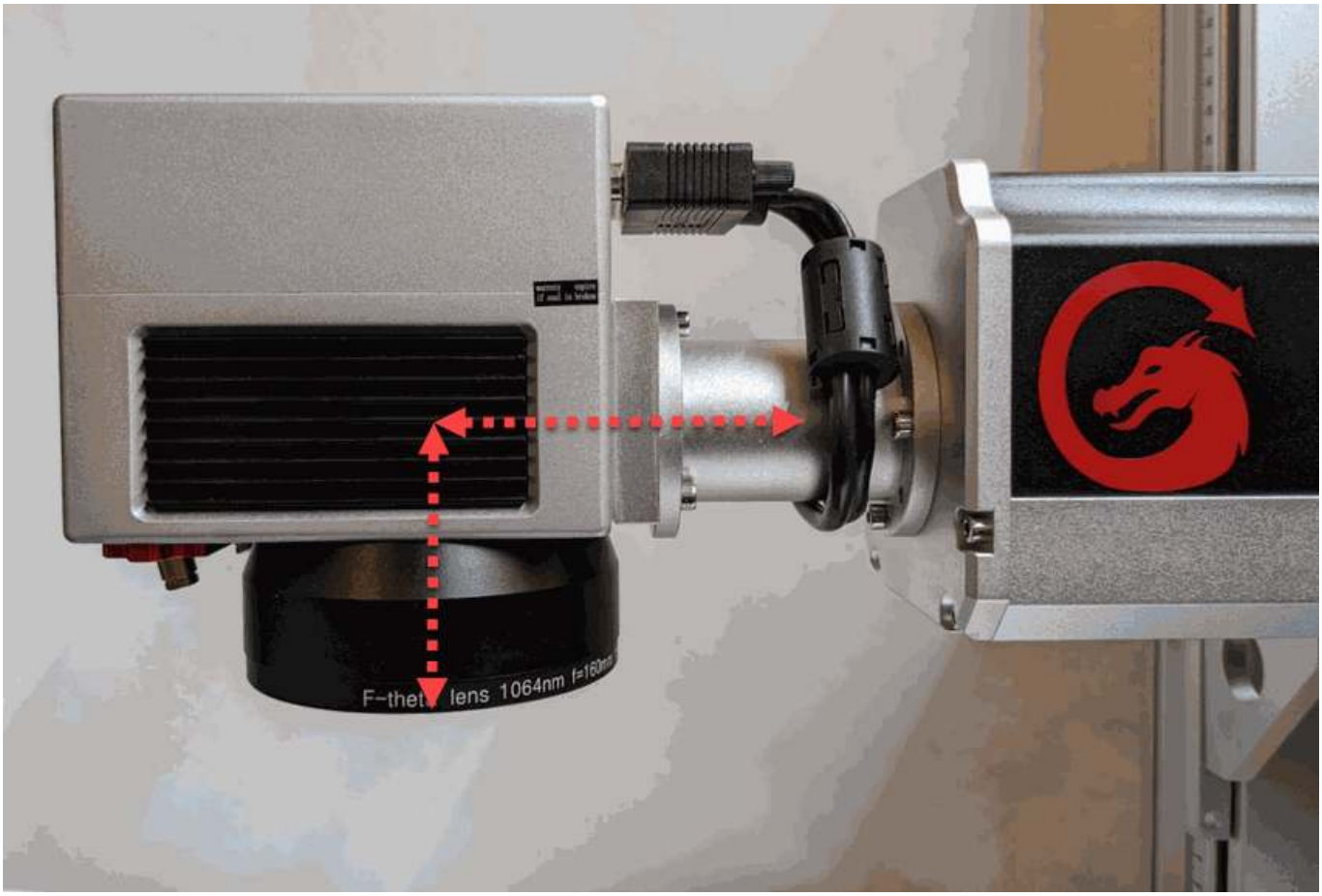
From the setup window, you'll adjust the **Mirror Distance**, **Object Diameter**, axis to curve around, and direction of the curve.

LightBurn will calculate the Cylinder Correction assuming the design to be output is centered on the object. If you are unable to place the item you're engraving at the center of the laser's working area, move your design to match the actual location of the object.

Mirror Distance

> **Warning**
>
> This is not the focus distance between the object and the lens, but the distance to the galvo mirrors themselves. Keep reading for help measuring this.

To use **Cylinder Correction**, you'll need to identify where your mirrors are. The mirrors are typically centered along a tube connecting the galvo head to the laser source. In the image below, a horizontal line marks the center of this tube.



Measure the vertical distance from this line to the bottom of the lens — shown with a vertical line in the image above — and add this to your focus distance to find the mirror distance. We recommend saving this measurement so you don't have to keep re-measuring.

Object Diameter

Measure the diameter of the object and enter it into this box.

X or Y Axis

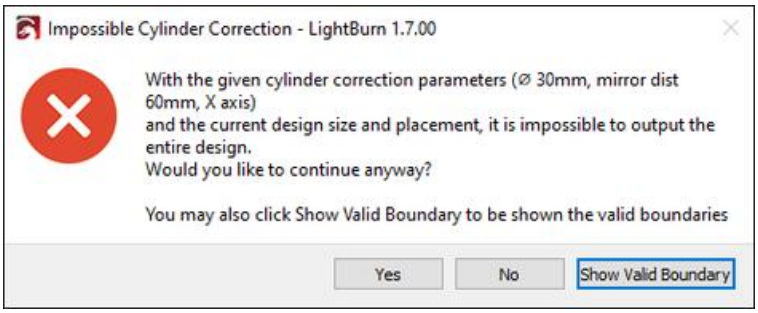
Select whether the object curves along the X (horizontal) axis or Y (vertical) axis. Only one of the two can be selected. The images below the options show a simplified representation of what each option would look like.

Convex or Concave

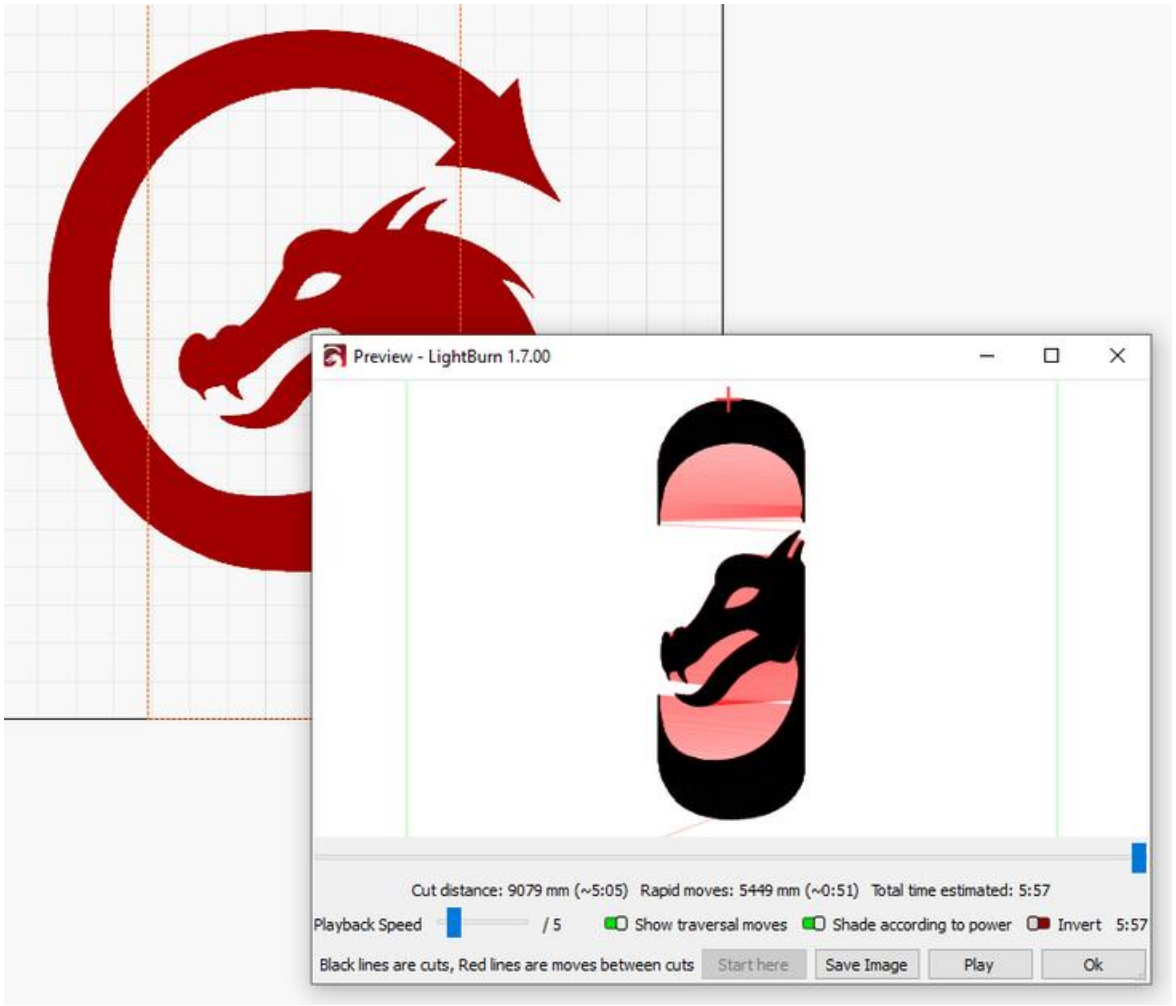
Select whether the object is **Convex** — curving outwards to make up the outer surface of a cylinder — or **Concave** — curving inwards to make up the inner surface of a cylinder. As with the X and Y Axis, the images show a simplified representation of Convex and Concave curves.

Show Valid Boundary

There are limits to where on a given object your laser can actually reach. The **Show Valid Boundary** button will insert a rectangle on the T1 Tool layer into your design to show the area that can be engraved.

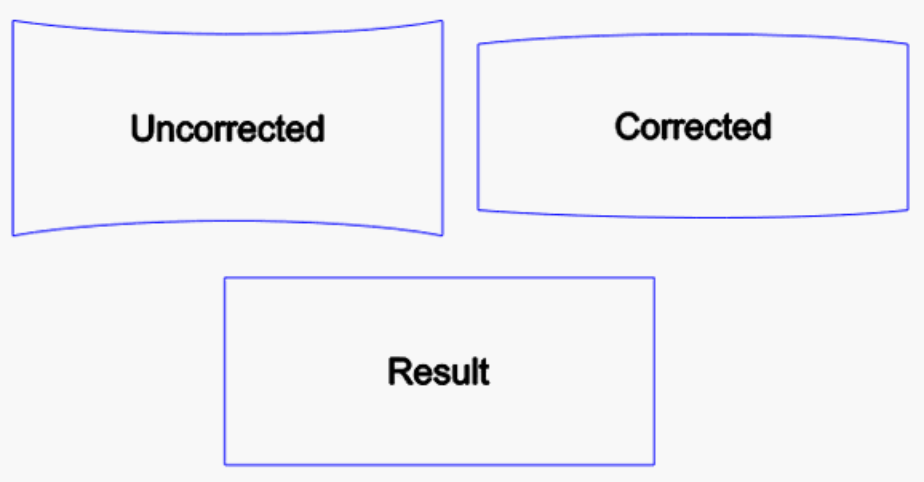


If you try to Preview or run a cylinder-corrected engraving while a portion of your design is outside of the valid area, LightBurn will pop up a warning asking you whether you'd like to continue anyway, show the valid boundary, or cancel. If you engrave anyway, LightBurn will automatically cut off the portions of the design outside the valid boundary area, as shown below.



How It Works

Galvo lasers project the laser from a single point above your object. When marking a cylindrical object, the increased travel distance for the laser as the surface of the object curves away causes the shape to widen and bend, similarly to the uncorrected rectangle in the diagram below.

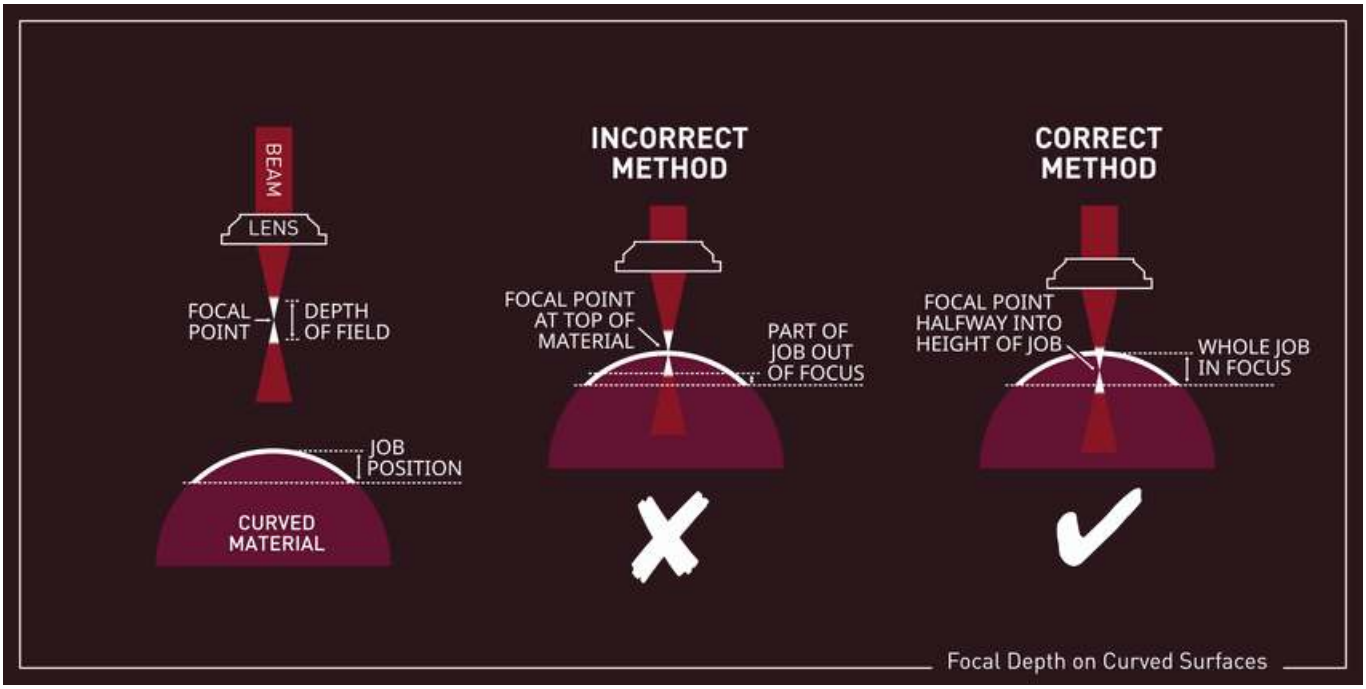


Cylinder Correction alters the output to warp your design in the opposite direction, negating the warping caused by the curve of the object. If your measurements are correct, this produces clean, undistorted output like the result shown in the diagram above.

Lens Selection

When using **Cylinder Correction**, you'll be marking across a larger range of focal distances than is typical when marking flat surfaces. For instance, if you have a 75 mm diameter tumbler and are engraving a 50 mm wide graphic, there will be about 10 mm difference in focal distance across your design.

For best results, use a lens with a larger field (working area) if possible. Wider field lenses have a lower power density but a longer focal range. To take advantage of the full focal range, focus your lens below the upper surface of the cylinder, as shown in the diagram below.



Troubleshooting

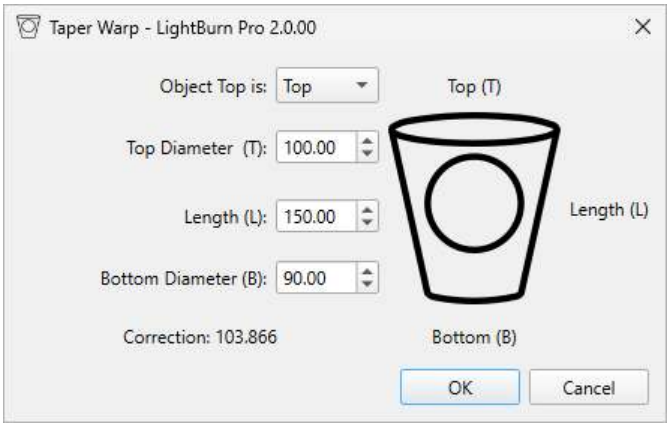
- If the outer parts of your design are missing, make sure that it's fully within the valid boundaries.
- If the placement of your engraving is wrong, check that the location of your design matches the actual location of the object you're marking. Confirm that the placement is right by framing before running the job.

Related Topics

- Rotary Mode
- Taper Warp

Taper Warp

The **Taper Warp** tool assists in rotary engravings on tapered objects by adjusting graphics to compensate for the difference in diameter between the top and the bottom of the object. Open the Taper Warp window by going to Laser Tools → **Taper Warp**.

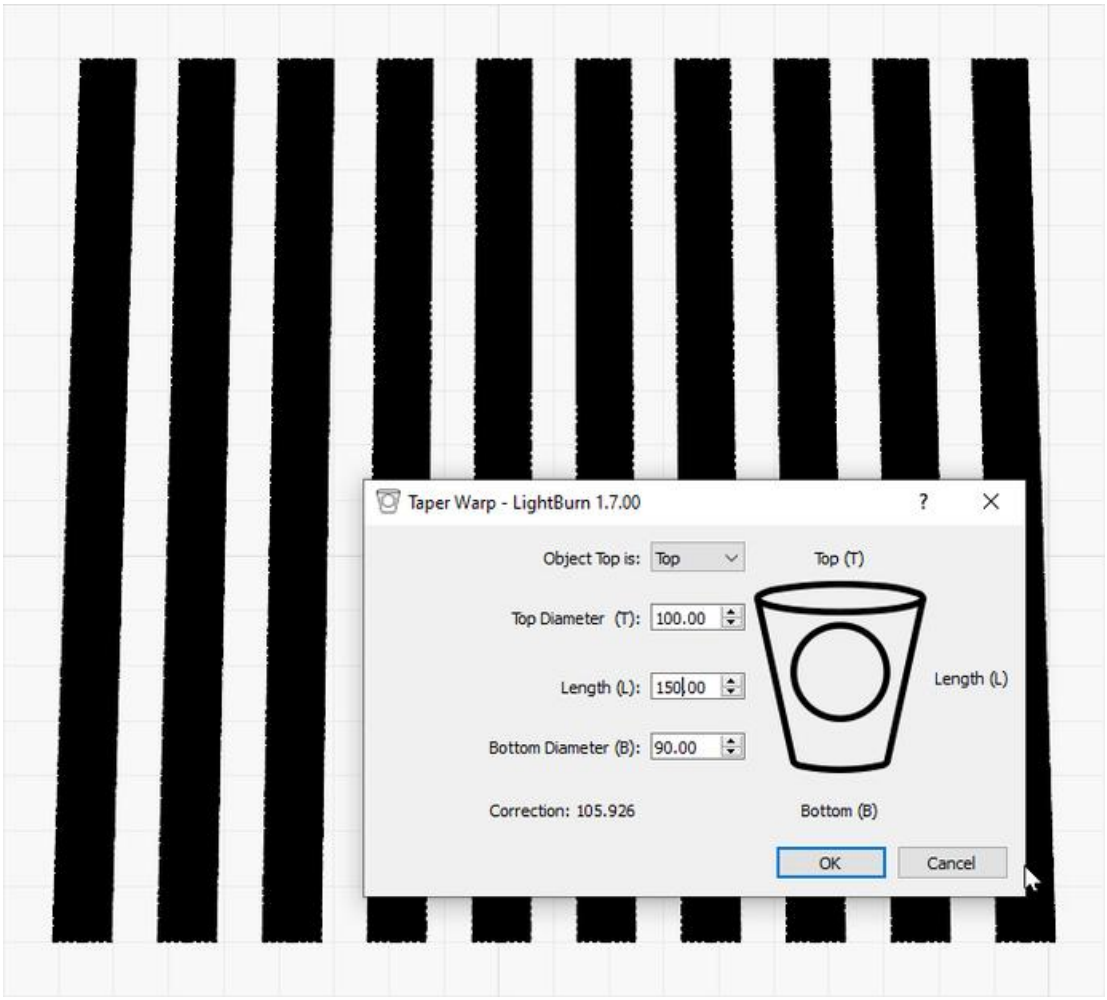


Using Taper Warp

> **Warning**
>
> Taper warp operates in millimeters. If your project is set to inches, please either switch to millimeters before using taper warp or convert your measurements to millimeters (multiply them by 25.4).

Select the artwork you want to apply **Taper Warp** to. Open the **Taper Warp** window and adjust the settings shown to match your object. The diagram on the right side of the window helps illustrate what the settings refer to.

As you adjust the settings, you'll see a preview of the effects of the Taper Warp in your Workspace. The image below shows the effect of Taper Warp on parallel rectangles.



Press **OK** to confirm settings and adjust the graphics, or **Cancel** to discard the changes.

When engraving your object, make sure that it's positioned so that the surface to engrave on is parallel to the bed of the laser.

Object Top

Set **Object Top** to either Top, Left, or Right so LightBurn knows where the top and bottom of the taper are — this is where the top of the physical object you're engraving is.

Top Diameter

Measure the diameter of the top of the object. Be as precise as possible – calipers can be very helpful, especialy dealing with round objects.

Length

Measure the length of the object and enter it here.

Bottom Diameter

Measure and enter the diameter of the bottom of the object.

Troubleshooting

- **Taper Warp** corrects for the difference in diameter between the top and bottom of the object. It does not account for different focal lengths or distances from the laser. Make sure the object you're engraving is held in the rotary so the surface to engrave is parallel to the laser's bed.
- The results of Taper Warp depend on the accuracy of the measurements you enter — if Taper Warp does not properly compensate for the taper of your object, double-check your measurements.

Related Topics

- Rotary Mode
- Cylinder Correction
- Warp and Deform

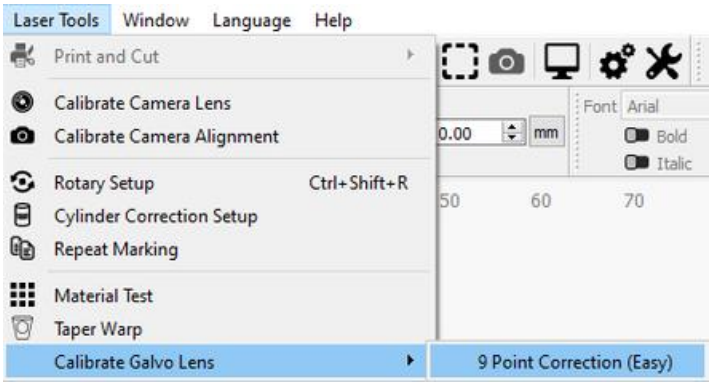
Galvo Lens Calibration

All Galvo lenses have distortion which must be corrected in order for output from your laser to have an appearance that's true to the design you see in your LightBurn Workspace.

When working with a new laser, you can usually import lens correction values provided by your machine's manufacturer when you first set up your laser with LightBurn.

If your manufacturer didn't provide the necessary file, or you purchase a new lens for a laser you already own, however, you may need to use LightBurn's **Galvo Lens Calibration** wizard to find the appropriate corrections.

To open the Galvo Lens Calibration wizard, go to Laser Tools → **Calibrate Galvo Lens** → **9 Point Correction (Easy)**.



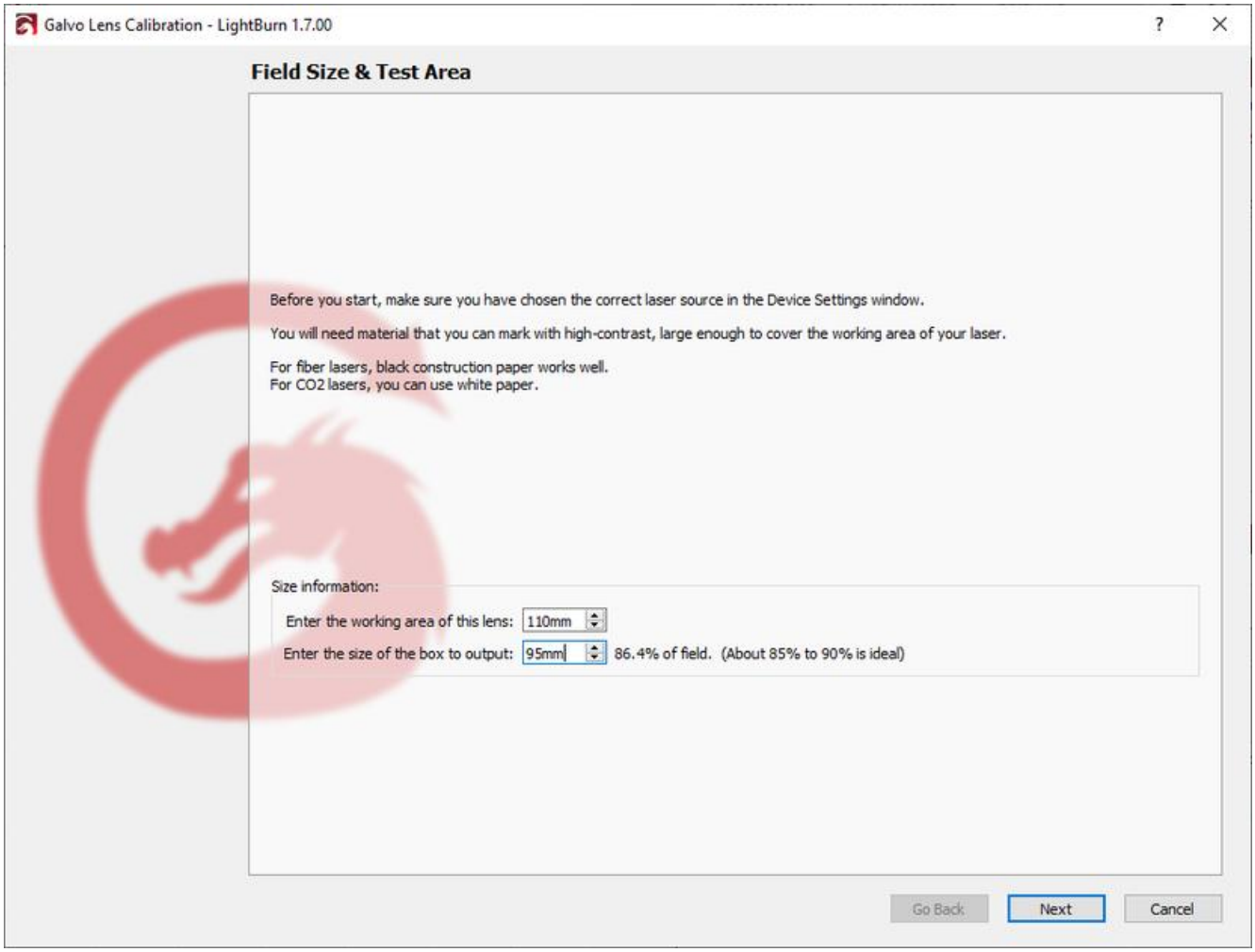
Requirements

To complete the **Galvo Lens Calibration** process, you will need:

- A properly set up device profile for your laser, with the correct **Laser Type** and (if applicable) **Fiber Type** selected in the Device Settings window.
- A material your laser is capable of marking with high contrast. Black construction paper works well for lasers with Fiber sources, and white paper works well for lasers with CO_2 sources.
- A ruler or calipers, to measure the distance between markings made on the material you selected.

Galvo Lens Calibration Wizard

Field Size and Test Area

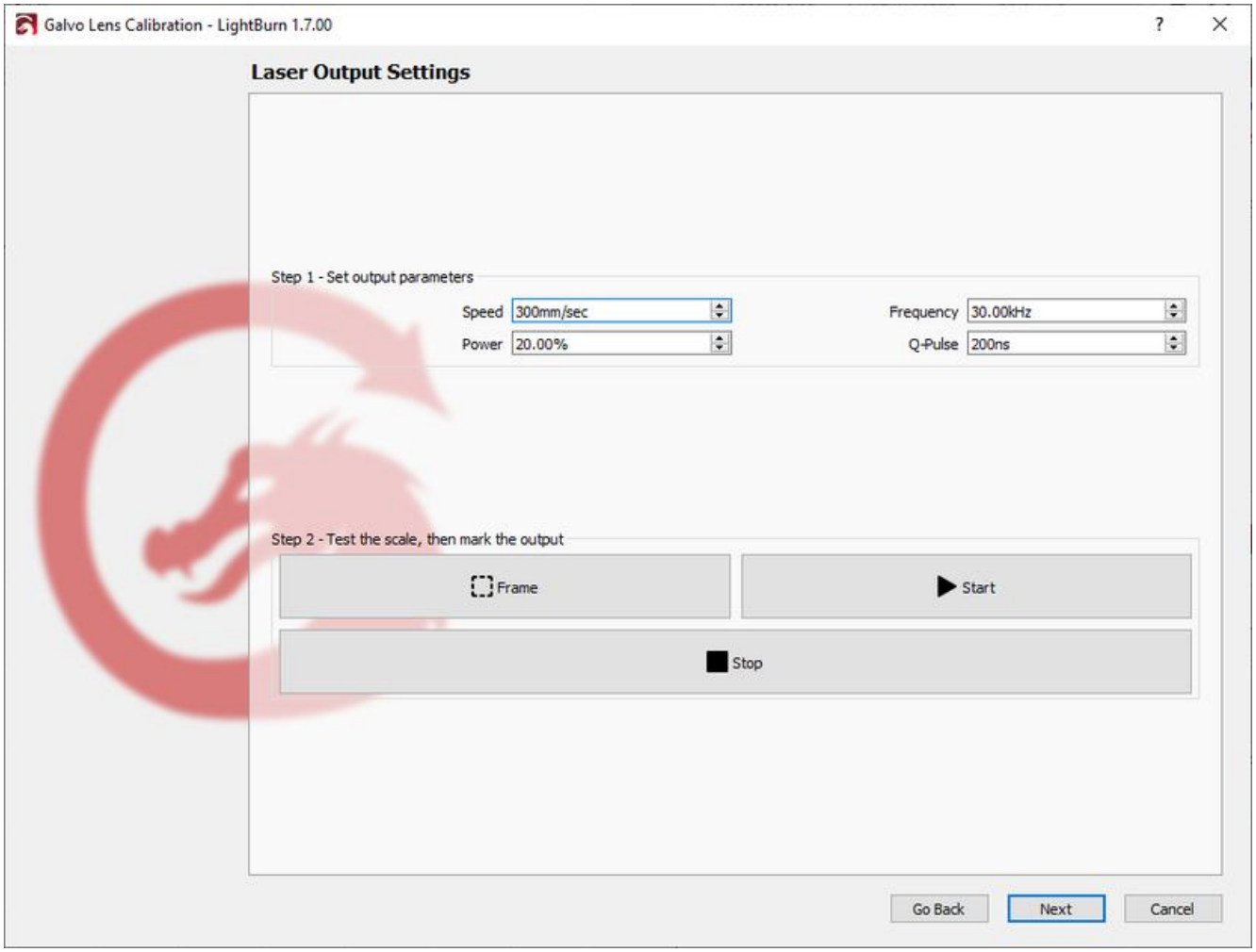


Each of the fields in this step will be automatically populated based on the **Field** setting in the Device Settings window, but can be adjusted manually if those settings are inaccurate.

Enter the **working area** of your lens (this value is usually provided by the manufacturer) as well as the **size of the box to output** — a value between 85% and 90% of the working area is ideal.

Click **Next** to continue to the next step, or **Cancel** to exit the process (progress will not be saved).

Laser Output Settings



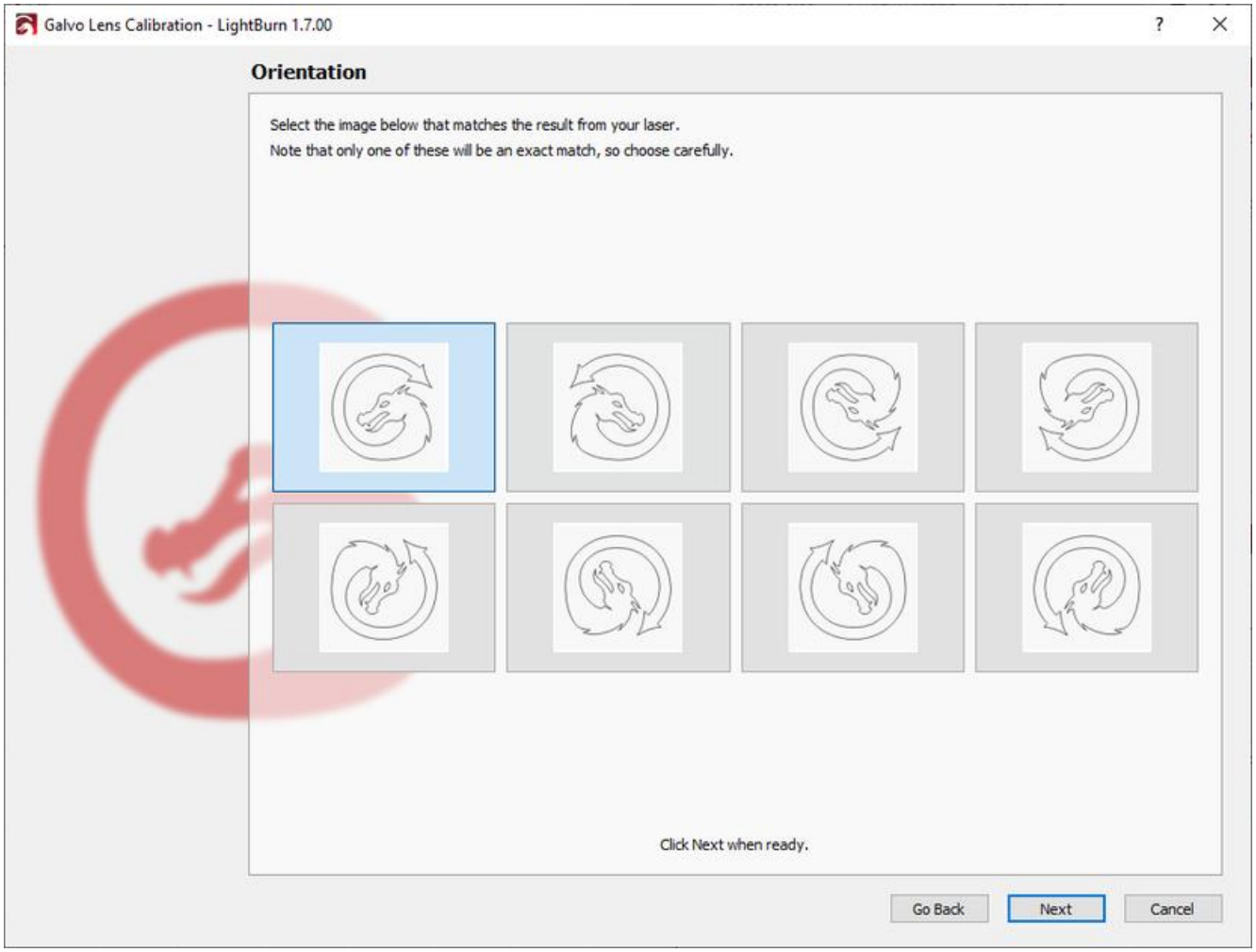
Enter appropriate settings for your laser to mark the material you selected, with strong contrast. If you are unsure what settings to use, try running a Material Test first, or consulting your laser's manufacturer for suggested settings.

The Frame, Start, and Stop buttons work exactly as they do for all other output.

Use the Frame button to verify that the **size of the box to output** is valid for your laser's working area. Mark the design if it fits, or return to the previous step using the **Go Back** button, if you need to adjust the size.

Click **Next** to continue to the next step, or **Cancel** to exit the process (progress will not be saved).

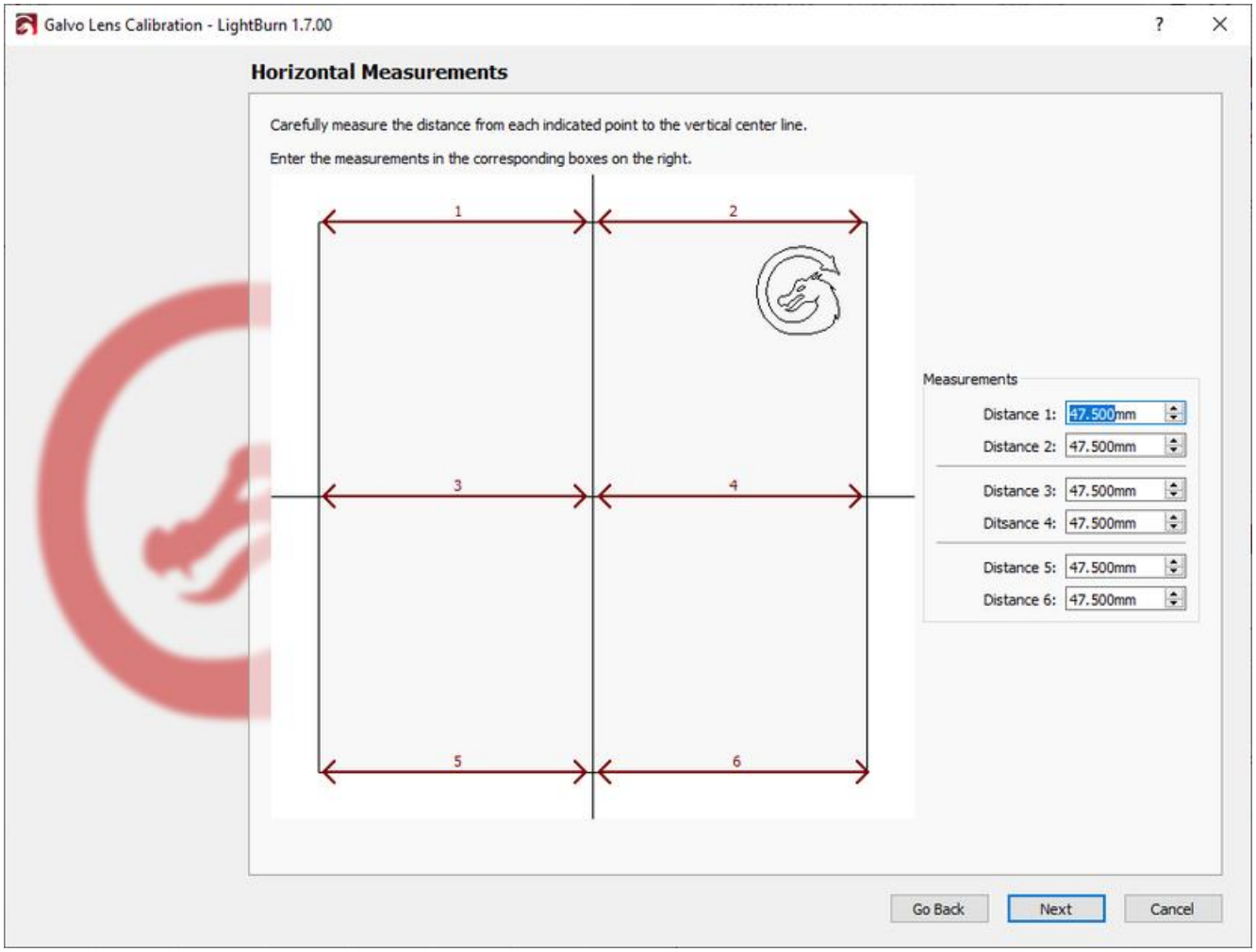
Orientation



Select the LightBurn Dragon that matches the orientation of the output from your laser. Only one orientation will match exactly — take care to select the correct one.

Click **Next** to continue to the next step, **Go Back** to return to a previous step and adjust settings, or **Cancel** to exit the process (progress will not be saved).

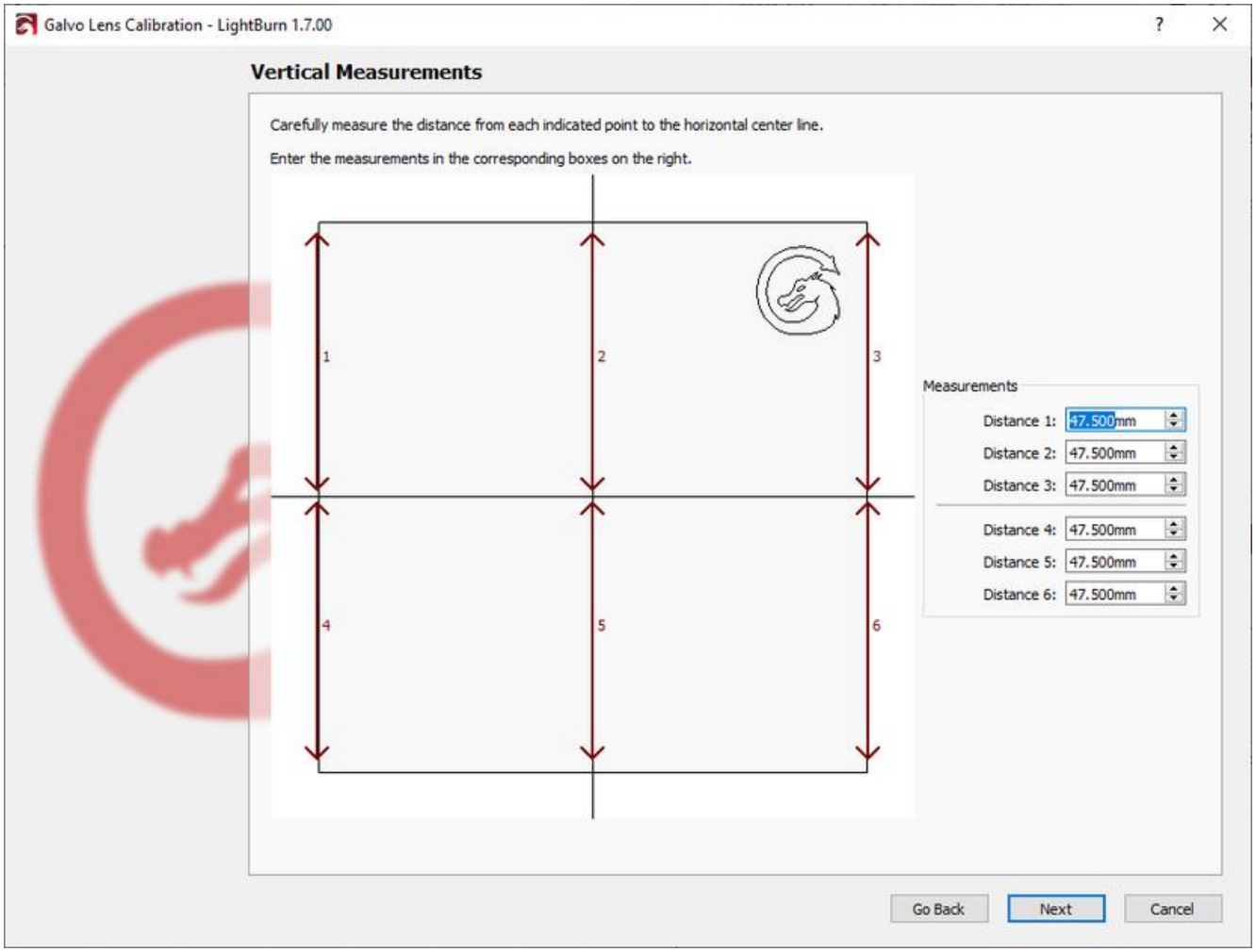
Horizontal Measurements



Measure and enter the horizontal distances between lines, as indicated in the graphic. Take care to measure the distances as accurately as possible — the accuracy of the lens correction depends on the accuracy of these measurements.

Click **Next** to continue to the next step, **Go Back** to return to a previous step and adjust settings, or **Cancel** to exit the process (progress will not be saved).

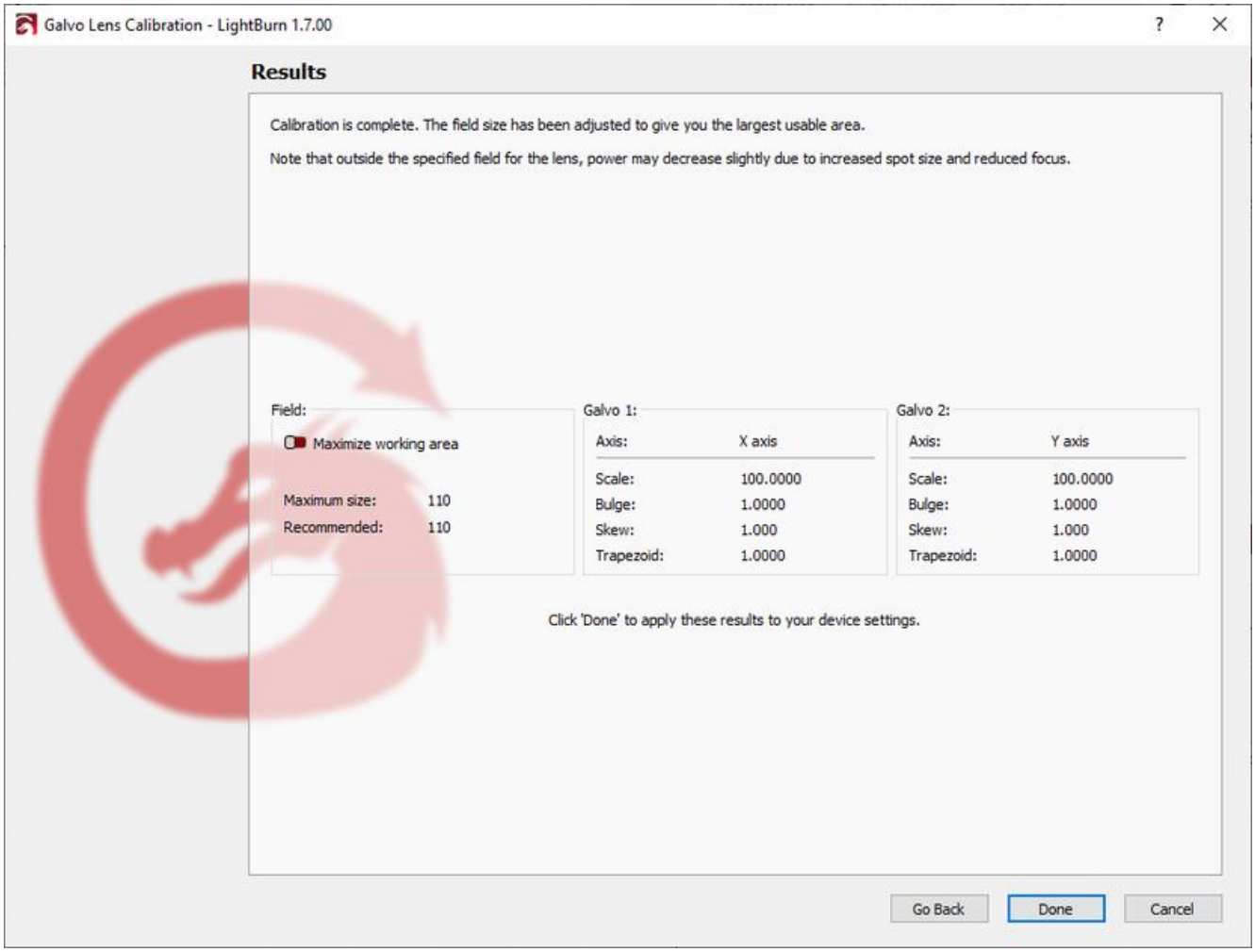
Vertical Measurements



Measure and enter the vertical distances between lines, as indicated in the graphic. Take care to measure the distances as accurately as possible — the accuracy of the lens correction depends on the accuracy of these measurements.

Click **Next** to continue to the next step, **Go Back** to return to a previous step and adjust settings, or **Cancel** to exit the process (progress will not be saved).

Results



The final page displays the resulting corrections that will be applied to each Galvo.

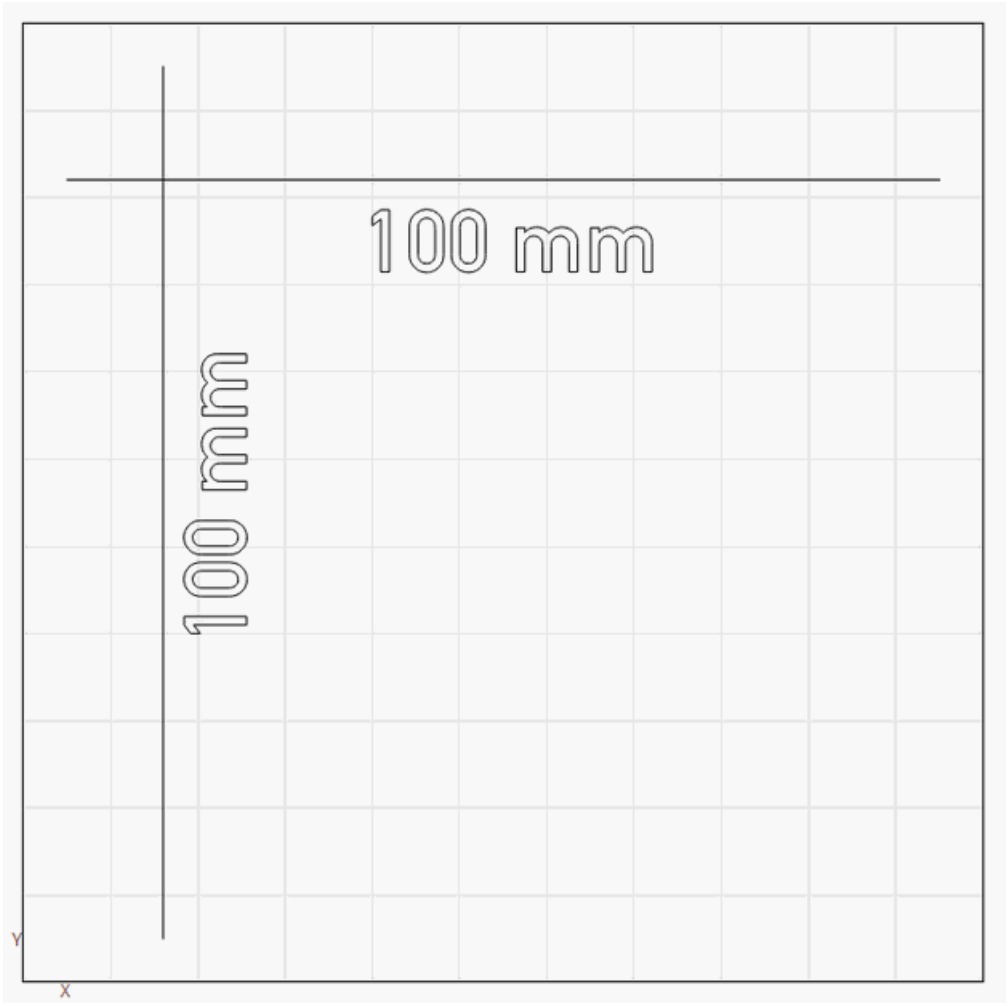
Toggle on **Maximize working area** to increase your laser's **Field** to the maximum size possible, if a larger possible size is indicated by the measurements you entered. Note that you may see a reduction in power output in areas outside the dimensions specified by the lens' manufacturer.

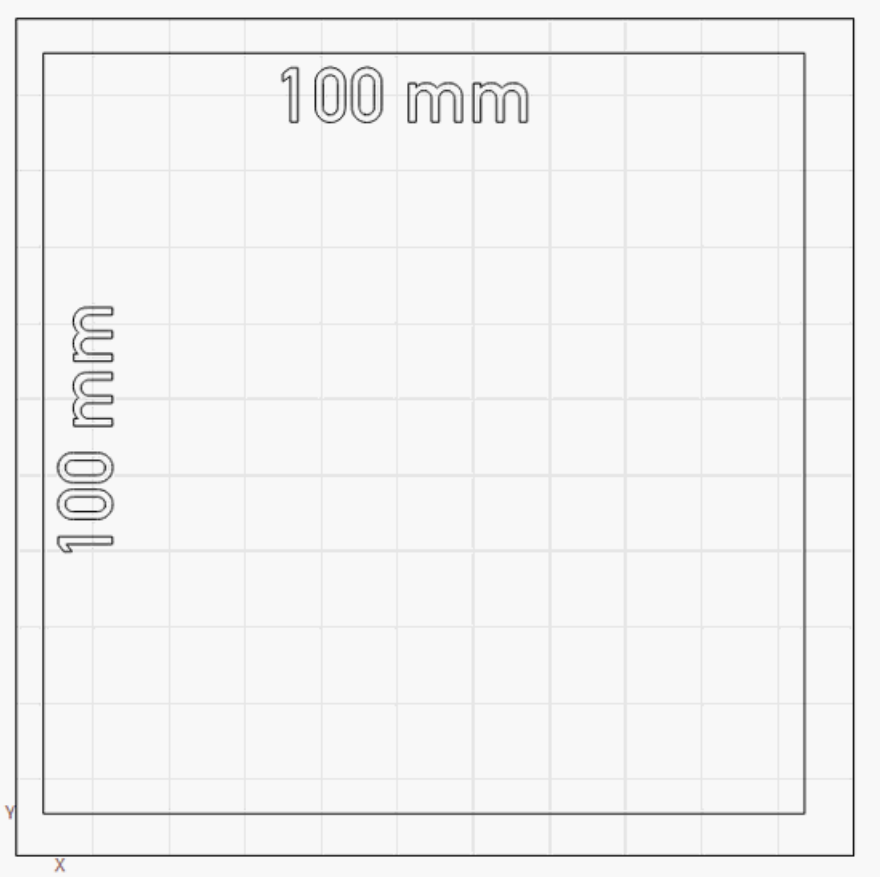
Click **Done** to exit the process and automatically apply the corrections to your Device Settings, **Go Back** to return to a previous step and adjust settings, or **Cancel** to exit the process (progress will not be saved).

Calibrate Axis

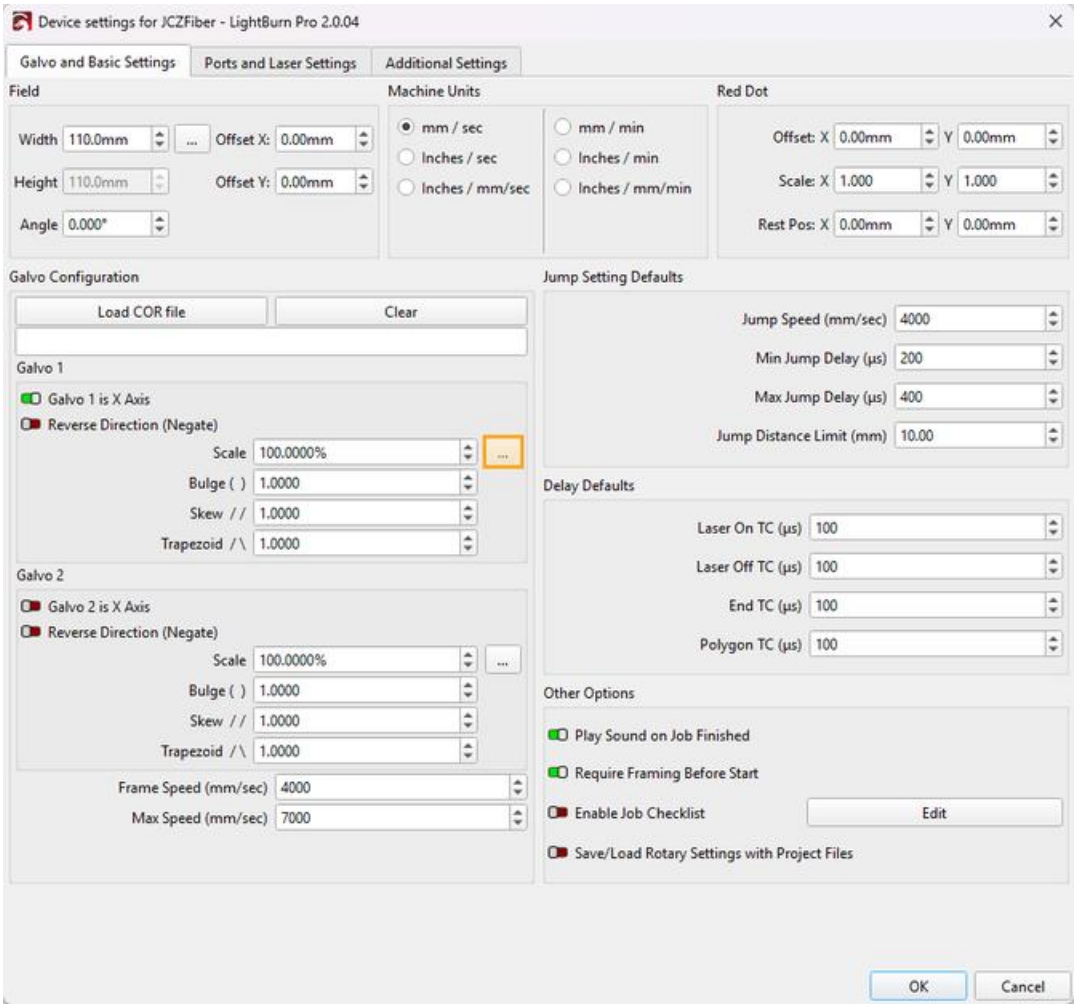
Calibrate your X and Y axis scaling after running the Nine Point Correction wizard to fine tune your Galvo laser's output to match the requested artwork dimensions.

1. Select the Draw Line tool and in the LightBurn Workspace draw a 100 mm line along the X axis and another along the Y-axis, or the Draw Primary Shapes tool and draw a box with equal dimensions — for example 100 mm.

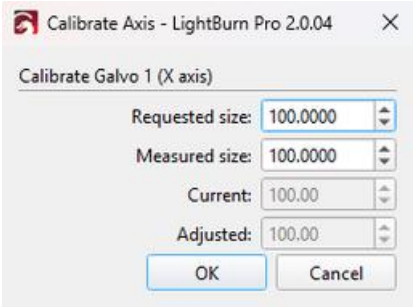

100 mm
100 mm
Y
X



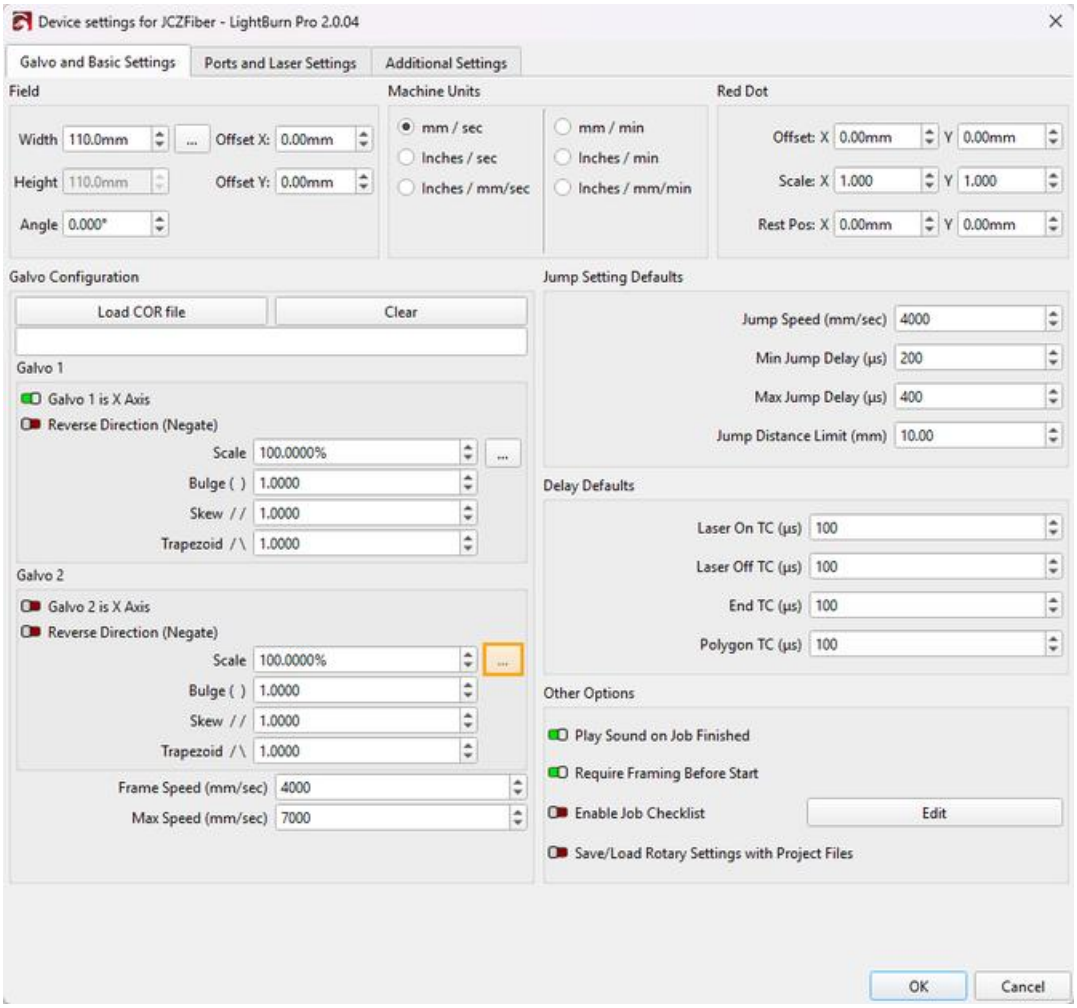
2. Open the Galvo Device Settings window again and in the **Galvo 1** section select the **...** button to open the **Calibrate Axis** window. Depending on which galvo is set as the X Axis, you'll see either **Calibrate Galvo 1 (X axis)** or **Calibrate Galvo 1 (Y axis)**.



3. Measure the marked material along the appropriate axis, and update the Calibrate Axis **Requested size** and **Measured size** values.
4. Select **OK** to apply the new scaling values to the first axis.



5. Repeat the process for the other axis in the **Galvo 2** section.



6. Select **OK** to apply the new scaling values to the second axis.
7. If needed, rerun the **Calibrate Axis** process another time to further refine or verify your scaling as needed.

Related Topics

- Add Galvo
- Changing a Lens
- Focusing
- Galvo Basic Device Settings

Dual Laser Control



LightBurn provides special setup options for machines with two laser tubes. These settings will only be available when using DSP devices.

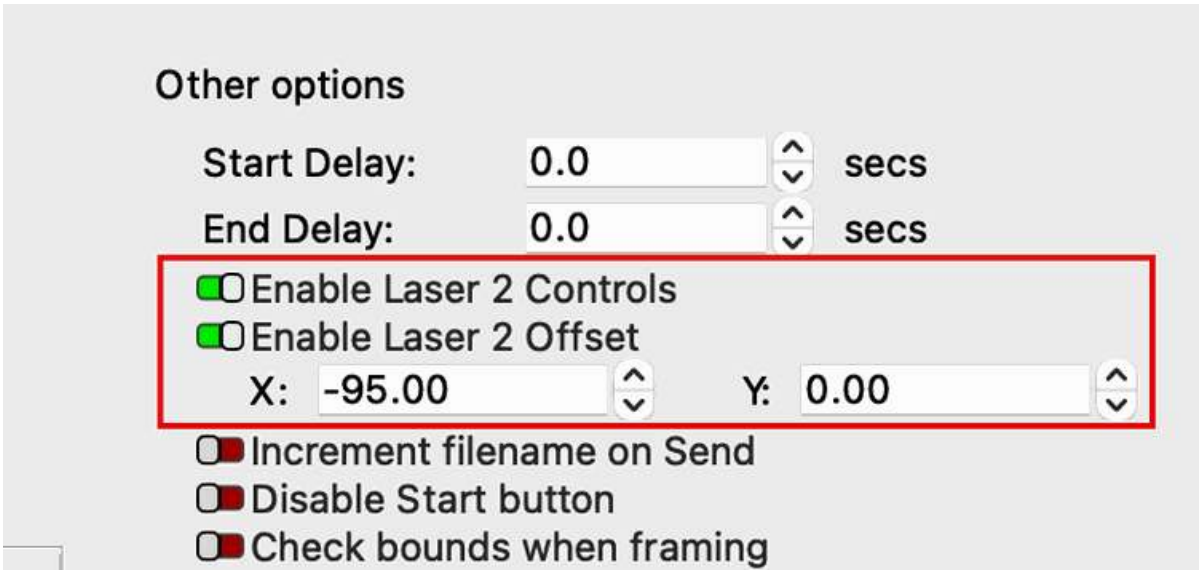
Enable Laser 2 Controls

Three options for working with dual lasers are available in the Basic Settings tab of Device Settings.

Turn on **Enable Laser 2 Controls** to expose switches in the Cuts / Layers Window and Cut Settings Editor that will allow you to enable or disable each tube for a given layer and adjust their power settings separately.

Enable **Only allow one laser at a time** to automatically disable the alternate tube when you enable Laser 1 or Laser 2. In other words, when you enable Laser 1 on, LightBurn will automatically disable Laser 2, and vice versa.

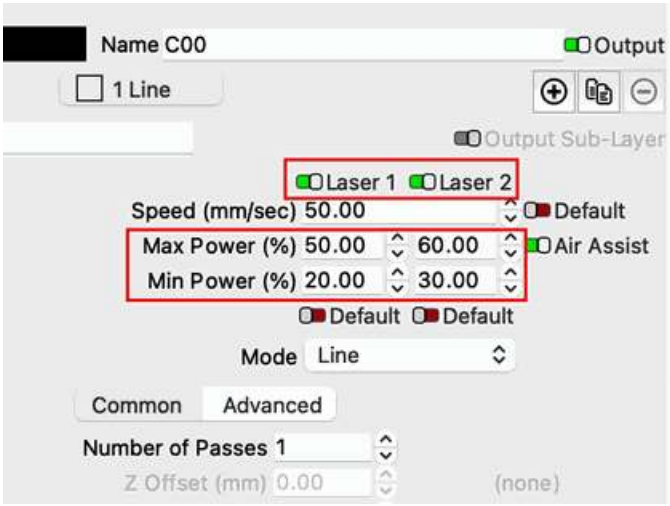
Turn on **Enable Laser 2 Offset** to specify an offset to account for the distance between the second and first laser head, if your laser has dual heads in addition to dual tubes.

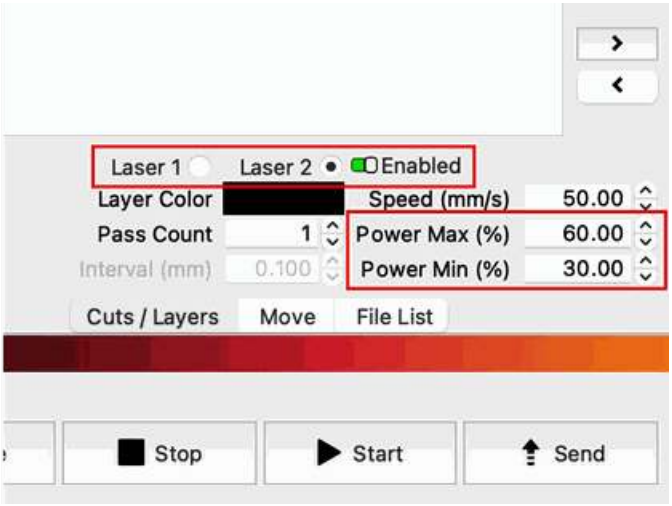


Individual Laser Controls

The switches in the Cuts / Layers Window and Cut Settings Editor are used to enable one or both lasers on a per-layer basis. Each laser's **Max Power** and **Min Power** settings can be managed individually, but all other settings are shared when both laser lasers are enabled at the same time.

In the Cut Settings Editor, both lasers' switches and power settings are visible simultaneously. In the Cuts / Layers Window, select the button for the laser you wish to adjust to show the switch and cut settings for that laser.





Both Lasers Enabled at Once

The option to enable both lasers at once is intended for lasers with dual heads. With both lasers enabled for the same layer, the lasers will run simultaneously in tandem, meaning two copies of the same output will be cut or engraved at once, without any offset applied to the output from the second laser.

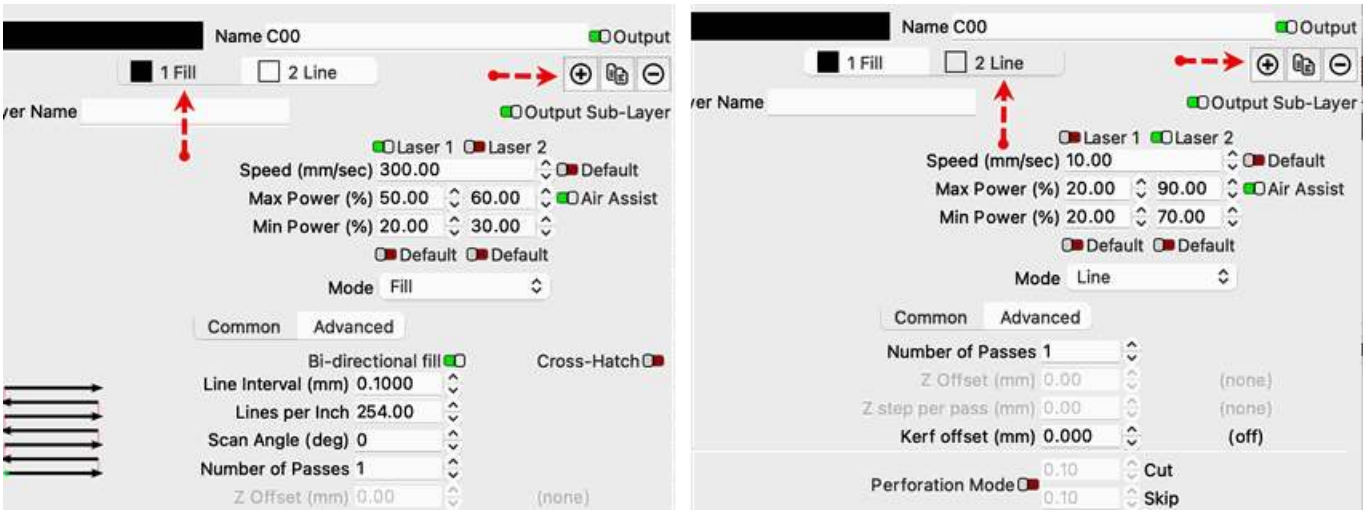
Make sure your laser heads are physically spaced far enough apart from one another that each has room to cut or engrave the entire graphic, without overlapping or running into the side of your machine. This means that any graphic you run with both lasers enabled must be narrower than the distance between the two heads.

Only One Laser Enabled at a Time

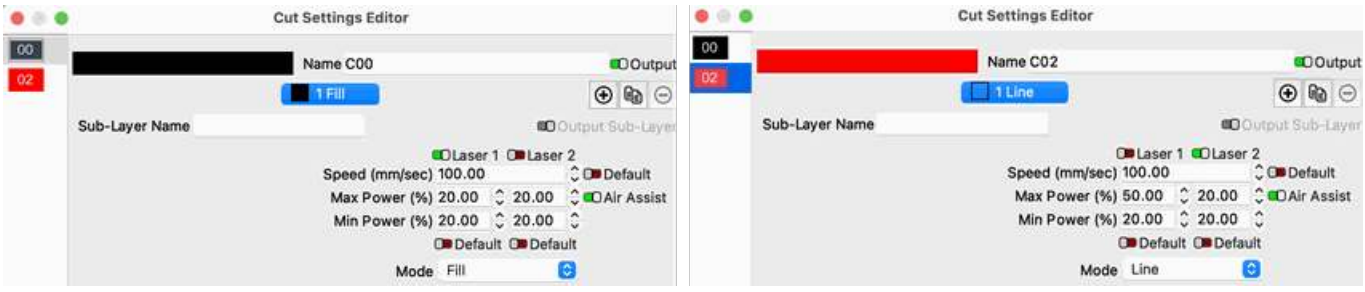
If each of your laser tubes is optimized for different processes, you may choose to enable one laser for engraving and the other laser for cutting. Any layer or Sub-Layer can be set to output using only Laser 1 or only Laser 2.

To have one laser engrave a graphic and the other cut the outline of the same graphic, create a Sub-Layer for the layer the graphic is assigned to, and enable only the appropriate laser for the operation assigned to each Sub-Layer.

> **Tip**
>
> Enable Only allow one laser at a time in your Device Settings to guarantee you never accidentally enable both lasers for a given layer.



If you are engraving and cutting separate graphics in the same job, set each to their own layer, and enable only the appropriate laser for the operation assigned to each layer.



If each of your laser tubes also has its own head, in order for output from Laser 2 to align to output from Laser 1, you must measure and apply an accurate Laser 2 Offset.

See our Set Laser 2 Offset Guide for more information.

Machine Management

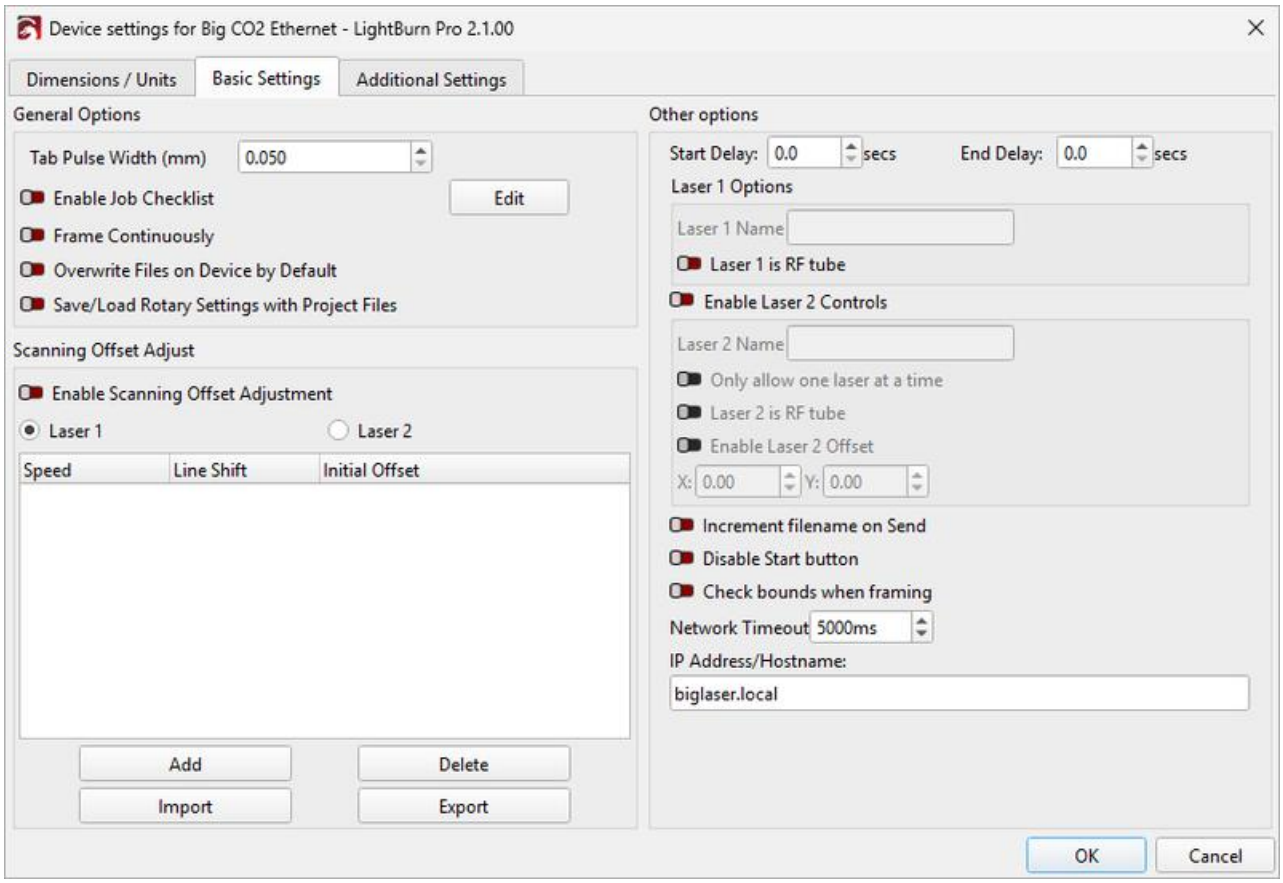
Device Settings

The **Device Settings** window contains device-specific settings for your laser. The settings available change based on the device selected — not all settings described below will necessarily be shown in your instance of LightBurn.

Access Device Settings by selecting Laser Tools → **Device Settings** or by clicking  in the Main Toolbar.

> **Have Multiple Lasers?**
>
> Select the correct device in the Devices dropdown in the Laser Window.

Device Settings Window



The Device Settings window

The **Device Settings** window is split into several tabs, not all of which will be available for all machines.

Dimensions / Units

Per-device unit and dimension settings.

Read More

Basic Settings

The most commonly used device settings.

Read More

GCode

Modify GCode output for GCode Devices.

Read More

Additional Settings

Settings for simulating previews and estimating time.

Read More

Custom GCode

Advanced GCode customization for the Custom GCode device type.

Read More

Galvo and Basic Settings

The most commonly used device settings for Galvo lasers.

Read More

Ports and Laser Settings

Configure a Galvo laser's ports and laser source.

Read More

External Axes Settings

Configure motorized X, Y, and Z axes for EZCad3 lasers.

Read More

Additional EZCad3 Settings

Additional configuration for EZCad 3 lasers. Most users should not change these manually.

Read More

Troubleshooting

- If you aren't seeing a setting you expected to see, check which tab you're in and make sure it's applicable to your laser.

Related Topics

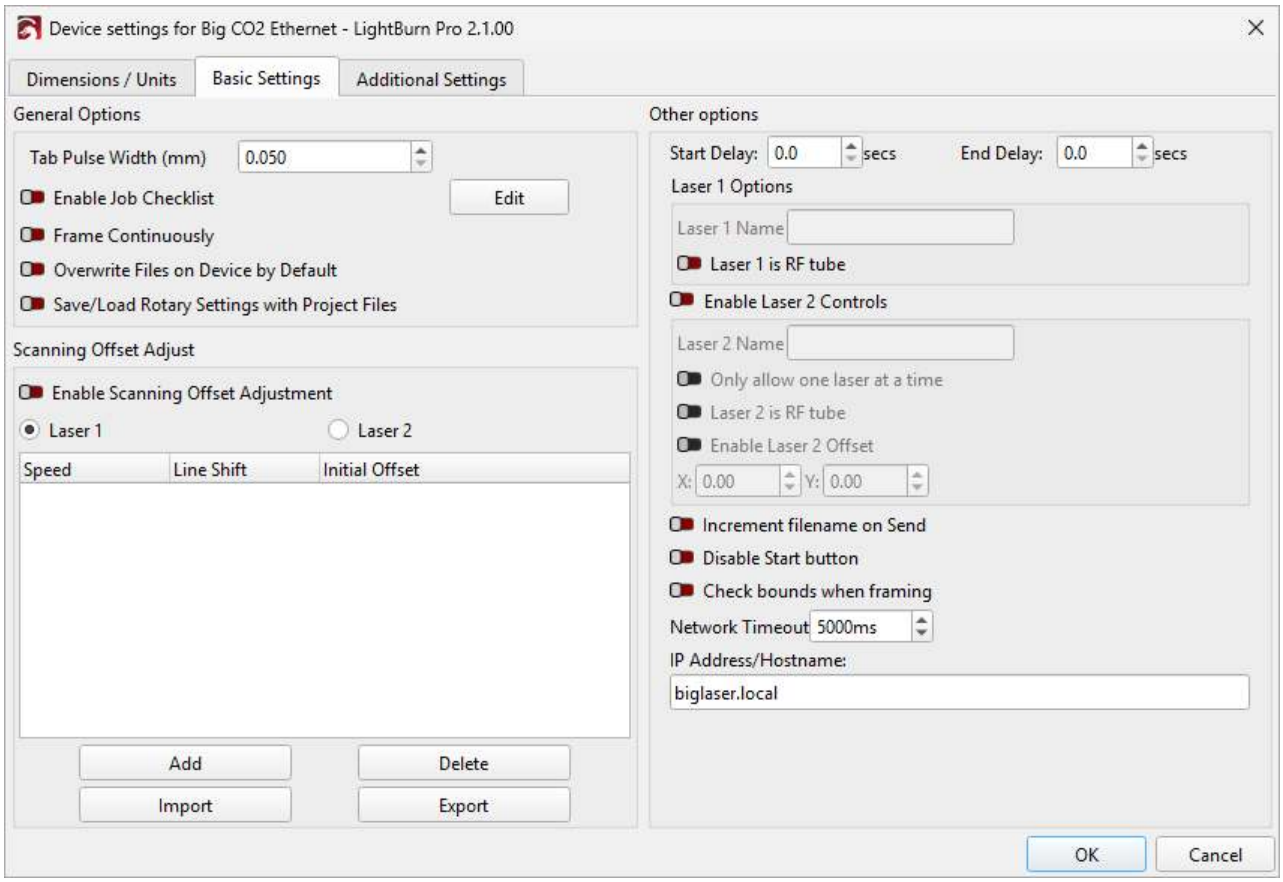
- Settings/Preferences
- Machine Settings
- Devices

Basic Settings

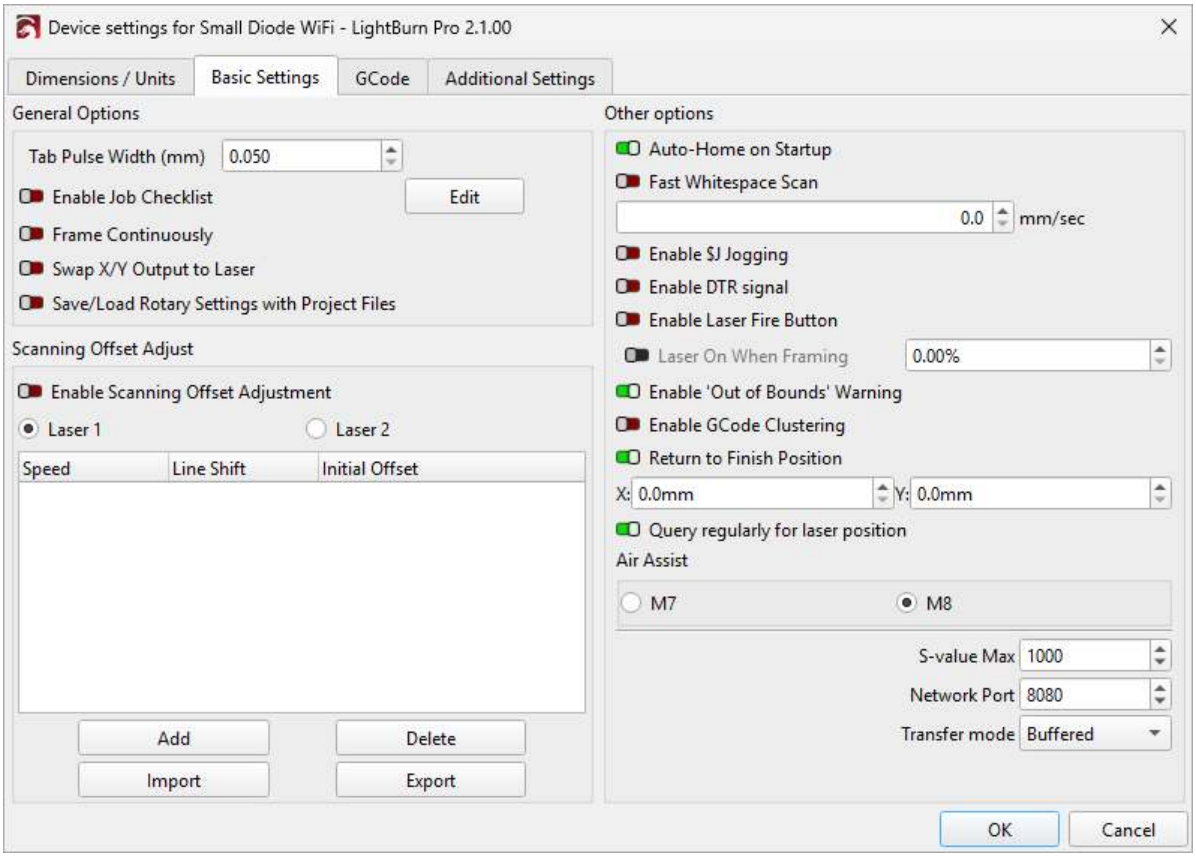
The **Basic Settings** tab of **Device Settings** contains the most commonly used settings, and is available regardless of device type.

The image below shows the Device Settings window for a DSP device — several alternate options will be presented if you are using a different type of device.

Click any option in the images below to jump directly to the relevant section for that option, or scroll down for a full list of options and descriptions, for both DSP and GCode-based lasers.



The Basic Settings tab of Device Settings on a DSP laser



The Basic Settings tab of Device Settings on a GCode laser

General Options

Tab Pulse Width

When using Tabs, the **Tab Pulse Width** allows for additional control of the Tab Cut Power setting. Change this length to increase or decrease the width of the pulses used to simulate reduced power cuts.

Enable Job Checklist

When this setting is enabled, LightBurn will display a text box with a list of items whenever you **Start** or **Send** a job to your laser. This is typically used as a reminder of things that need to be done before starting the laser, such as turning on an exhaust fan or putting on protective glasses.

To create a custom checklist, click the **Edit** button.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=oRA-tJgANMc

Frame Continuously

This setting alters the behavior of the Frame button. When enabled, LightBurn will frame your project repeatedly until stopped, instead of a single time.

Continuous Framing is often useful for adjusting material placement within the laser.

Overwrite Files on Device by Default

When enabled, LightBurn will automatically overwrite pre-existing files of the same name when sending a project to your laser's controller. Keep this setting disabled to receive a warning message whenever LightBurn will need to overwrite a file in order to send a new one.

Save/Load Rotary Settings with Project Files

When enabled, LightBurn will include rotary settings when saving a project, and will load those settings when re-opening the file. This can streamline job setup when working with rotaries.

Scanning Offset Adjust

This setting is used to compensate for minor delays in your power supply causing the laser's firing point to be a little behind where it should be when scanning back and forth at high speeds. For more information, see Scanning Offset Adjustment. For an example of what Scanning Offset problems might look like, see Troubleshooting: Blurry Edges.

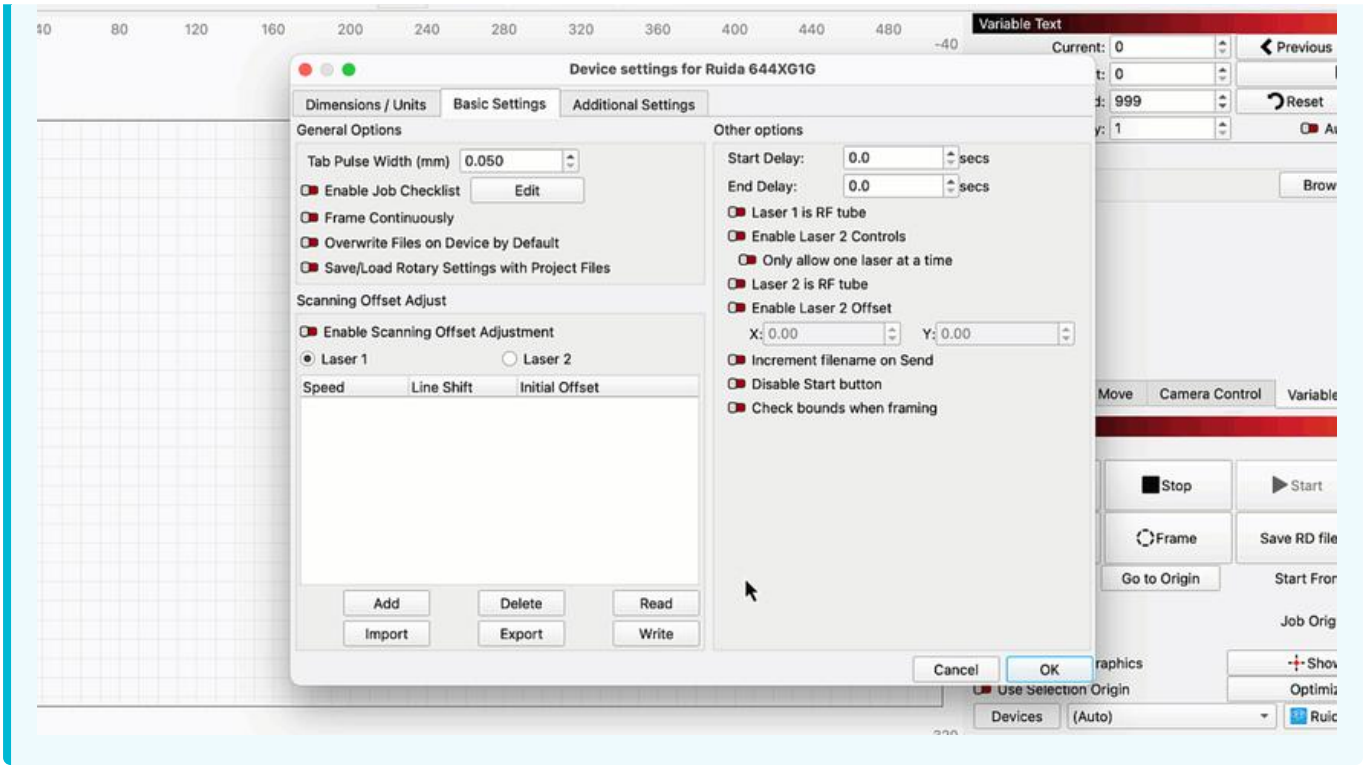
Use the **Enable Scanning Offset Adjustment** switch to toggle between enabling or disabling your saved offset values. If your machine has multiple laser sources, click the buttons next to **Laser 1** and **Laser 2** to switch between viewing and applying settings for each source.

The **Add**, **Delete**, **Import**, and **Export** buttons allow you to create, edit, load, and save your Scanning Offset settings. You can use this to Import settings provided by your machine's manufacturer, share adjustments with other users, or back up your settings before making changes.

Read and Write Buttons

Some Ruida controllers allow the ability to directly **Read** Scanning Offset Adjustments stored in the controller's settings, or **Write** new settings from LightBurn.

These buttons only appear when LightBurn is directly connected to a Ruida controller that supports the Read/Write functions.



Other Options

GCode Devices

These settings are only available for GCode-based lasers.

Auto-Home on Startup

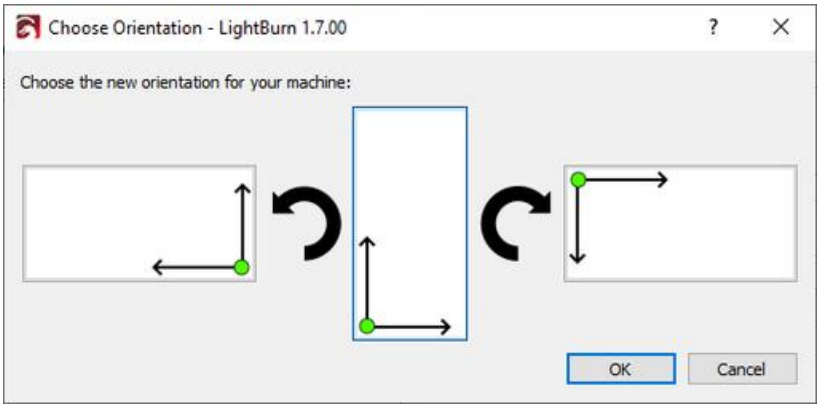
If your laser has homing switches, enable this to automatically home the machine on startup. For lasers without homing switches, disable this setting to prevent GRBL Error 5.

Swap X/Y Output to Laser

Enable this setting to rotate your LightBurn Workspace to match your machine's physical orientation, and swap X/Y output to your laser accordingly.

The green dot in the dialog window indicates the location of the machine's 0,0 point (the location it homes to) in the machine's physical orientation. Select the image that matches your machine's orientation, then click **OK**. LightBurn will automatically adjust your Origin and rotate your Workspace and output to your machine to account for the new orientation.

This option is intended for machines with Y dimensions that are longer than their X dimensions.



Fast Whitespace Scan

This setting boosts your laser's travel speed through blank areas of an engraving, if the speed you set here is faster than your laser's engraving speed.

This can save a lot of time if you're engraving a design slowly to get a good burn, particularly if there are large blank areas. This setting applies to both images and filled vector engravings.

> **Note**
>
> Since Marlin treats G0 and G1 moves identically, this value is used to specify the speed for rapid moves. If you do not set this value, LightBurn will use the same speed as the G1 moves.

Enable $J Jogging

Newer versions of GRBL (1.1f and later) can use this custom jogging mode. Enable this setting to add a Continuous Jog switch to the Move Window. After also enabling Continuous Jog in the Move Window, you'll be able to hold an arrow key to begin jogging in a direction, and release to stop (cancel) the move.

$J Jogging (Continuous Jog) has several benefits over normal jogging:

- It does not affect the GCode parser state.
- If soft limits are enabled, jog commands that would go out of bounds are ignored without triggering an error or alarm.
- It allows for cancelling a jog move.

Enable DTR Signal

This setting controls whether LightBurn sends a **Data Terminal Ready** signal to the controller to start serial communication. Many programmable hobby-level controllers, including Arduino-based systems, use the DTR pin to reset the controller. LightBurn usually sets this value for you, but if your GCode controller won't communicate, toggling this setting may help.

Enable Laser Fire Button

Enable this setting to add a **Fire** button and power setting to the Move Window to allow you to turn on the laser at low power for focusing, Framing, and positioning. When the Fire button is enabled, you can hold the `Shift` key while Framing to enable the beam.

This setting should only be used for diode lasers, which generally don't have a red dot pointer for Framing. This should never be used for a CO_2 laser, which has an invisible beam that could blind you or start a fire.

Laser on When framing

This setting turns on the laser at the power level you specify while Framing.

> **Safety Warning**
>
> This changes the default behavior of the laser and can cause the laser to fire when a user is not expecting it. Enable this setting with caution, with a very low power level, and only when using a diode laser.

Enable 'Out of Bounds' Warning

Enable this setting to have LightBurn warn you if a job will cause your machine to travel out of bounds. This setting requires that your device has been properly homed and is reporting coordinates correctly, and that your Workspace size is set accurately in LightBurn.

Enable GCode Clustering

GCode Clustering condenses the commands sent to your device. For devices that support it, this reduces the bandwidth needed for communication and allows the controller's motion planner to see further ahead. For devices that don't support this feature, enabling it will result in a GRBL error. More information is available in this post in our forum.

Return to Finish Position

This setting tells LightBurn to send the laser to the specified position after a job is run. This requires that the job is running in **Absolute Coords** or **User Origin** mode.

Air Assist

This setting determines whether LightBurn uses an M7 or M8 command to enable air assist.

S-value Max

This setting tells LightBurn how to represent 100% power for GRBL and Smoothieware controllers. Current versions of GRBL default to **1000** for this, while older versions use **255**. This value needs to match what your controller expects. Read more in GRBL: Low or No Power Output.

Baud Rate

This setting controls the speed at which LightBurn attempts to communicate with your laser's controller.

LightBurn defaults to 115,200 baud for GRBL controllers and 250,000 baud for Marlin controllers. Some Marlin controllers use 115,200 baud, and some GRBL controllers go as low as 9600 baud. If you are having trouble communicating with your controller and are certain the controller and firmware are supported, contact the manufacturer for the correct baud rate for the board, which may be different from the default value.

Network Port

The network port a device uses to connect. The default value is 23, but some devices use a different port, such as 8080.

Transfer Mode

LightBurn defaults to the **Buffered Transfer mode** for communication with GRBL devices. This is faster and more reliable than the **Synchronous** Transfer mode. However, there are a few devices that are incompatible with Buffered Transfer mode. These devices will stutter, stop, or even restart when buffered communication is selected. If you notice this happening, try switching the Transfer mode to Synchronous.

Query Regularly for Laser Position

When enabled, LightBurn will regularly check the laser's current position and update it in the Move Window.

DSP Devices

These settings are only available for DSP lasers.

Start and End Delay

Start Delay allows you to set a delay before the job starts cutting, and **End Delay** allows you to delay the end of the job after the cut is complete. This is typically used to allow the exhaust fan to start up and spin down.

This feature is only available on Ruida controllers, and only works if the controller's wiring is compatible. Your success in enabling it will be dependent on your specific hardware.

Laser 1 Name

If you have two laser tubes enabled, use this to set a name for Laser 1.

Laser Is RF Tube

Enable this switch if your laser has an RF tube.

If your laser has a single tube, only the adjust the **Laser 1 is RF Tube** switch. If your laser has two tubes, you can enable this switch for either Laser 1 or Laser 2 individually, as appropriate.

This will disable an automatic warning message, intended for CO_2 lasers, which is sent when you start a job which uses an RF tube and includes a layer whose Min Power is set to zero.

Enable Laser 2 Controls

This setting enables controls for a second laser tube within the same machine. Enable this setting only if your machine has two laser tubes.

Laser 2 Name

If you have two laser tubes enabled, use this to set a name for Laser 2.

Only Allow One Laser at a Time

Enable this switch to automatically disable the alternate tube when you enable Laser 1 or Laser 2.

In other words, when you enable Laser 1, LightBurn will automatically disable Laser 2, and vice versa.

Enable Laser 2 Offset

If you are using a dual head laser, use this setting to specify the position of the second laser head relative to the main one.

See Dual Laser Control for more information on dual laser setup.

Increment Filename on Send

This option will automatically increment file names when sending jobs to your laser, to make identifying separate runs easier and maintain your file history. Your file name must contain a digit in order to auto-increment.

Disable Start Button

Use this setting to disable the **Start** button.

On some lasers, streaming a job with the **Start** button can be unreliable, so this feature allows you to ensure that all jobs are sent to the laser using the **Send** button, which transfers the entire job to the laser before it can be started.

Check Bounds When Framing

Enable this setting to have LightBurn check that a job won't travel out of bounds when Framing.

Network Timeout

This setting determines how long LightBurn will attempt to communicate with a networked device before reporting a communication failure. Increasing the value from the default of 5000 ms (5 seconds) can help with slower connections.

IP Address/Hostname

The IP address or hostname to use when connecting to a networked device. This value is set when adding a device, and can be viewed or edited here.

Troubleshooting

- If you aren't seeing a setting you expected to see, check which tab you're in and make sure it's applicable to your laser.

Related Topics

- Settings/Preferences
- Machine Settings

- Devices

Dimensions / Units

Per-device unit and dimension settings.

Read More

Basic Settings

The most commonly used device settings.

Read More

GCode

Modify GCode output for GCode Devices.

Read More

Additional Settings

Settings for simulating previews and estimating time.

Read More

Custom GCode

Advanced GCode customization for the Custom GCode device type.

Read More

Galvo and Basic Settings

The most commonly used device settings for Galvo lasers.

Read More

Ports and Laser Settings

Configure a Galvo laser's ports and laser source.

Read More

External Axes Settings

Configure motorized X, Y, and Z axes for EZCad3 lasers.

Read More

Additional EZCad3 Settings

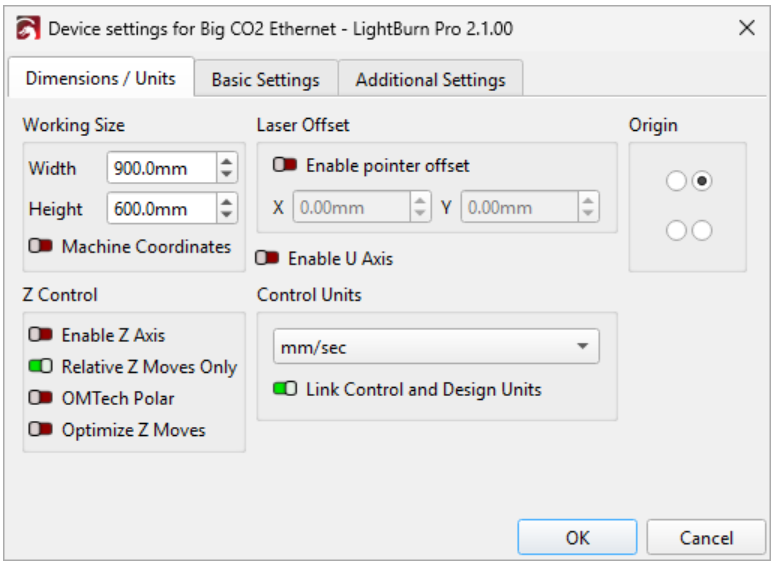
Additional configuration for EZCad 3 lasers. Most users should not change these manually.

Read More

Dimensions / Units

The **Dimensions / Units** tab of **Device Settings** contains settings related to units and measurement and is available regardless of device type.

The image below shows the Dimensions / Units tab for a DSP device. Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a full list of options and descriptions.



The Dimensions / Units tab of Device Settings

Working Size

The maximum X (**Width**) and Y (**Height**) travel for your laser. This sets the size of the work area in LightBurn, but does not affect the laser itself.

CNC Machine

Enable this setting if working with a machine with a rear-right origin and negative workspace coordinates. CNC machine users may also want to try MillMage.

Machine Coordinates

Enable this setting when working with a CNC machine that uses G54 offsets.

LightBurn works best with the G54 Workpiece Coordinate Offset set to 0,0,0. You can set this in the console by entering `G10 L2 P1 X0 Y0` and pressing enter, while connected to your machine. If you do need to use the G54 offset, enabling **Machine Coordinates** will fix some problems, but not all. In particular, User Origin workflows and Continuous Jogging may not work correctly with a G54 offset.

Origin

The location of your laser's **Origin**, or home. If this is set incorrectly, the orientation of your laser's output will be mirrored or inverted.

For help with troubleshooting your laser's Origin, please see Troubleshooting: Wrong Orientation.

Laser Offset

If your laser has a red dot pointer for Framing and positioning, this setting allows you to compensate for misalignment between the pointer and the actual cutting location. Enable this setting to shift the laser's output relative to the red dot location.

See Laser Offset Setup for instructions on calculating the offset.

Enable U Axis

Enable this setting to apply per-layer U Axis offsets. This is not supported by all controllers.

Device Units

Use this section to tell LightBurn whether your device expects to receive metric or imperial units. Most machines will use metric. Not all device types will have this section.

Control Units

This section sets the **Control Units** for your currently active device, which are the units used for speed and distance when controlling your device.

If you prefer to design in inches but use metric units for speed, choose **Inches / mm/sec** for CO_2 and Galvo lasers or **Inches / mm/min** for diode lasers.

Link Control and Design Units

By default, Control Units and Design Units are linked, so switching between millimeters and inches for either one will switch both. To unlink the two types of units, turn off the **Link Control and Design Units** toggle. To relink them, enable the toggle.

Z Control

This section contains settings for your laser's Z Axis, which controls the distance between the laser and the workpiece. The options available will vary depending on what laser you're using.

Enable Z Axis

Enable this to allow LightBurn to control the Z Axis of your machine.

> **Warning**
>
> Enabling this setting means that LightBurn will always emit Z values for a running job. If using this setting, you must either enable **Relative Z moves only** or set a **Material** height value in the **Cuts / Layers** window.
>
> **If you do not take either of these steps, the default Material height of 0 may result in your laser head crashing into the workpiece.**

Reverse Z Direction

If your machine's Z Axis moves in the opposite direction of what you expect, toggle this switch to flip the direction for Z moves.

Relative Z Moves Only

This setting treats the machine's height at the start of the job as 0, and uses that height as the starting point for all Z moves. Material Height is ignored when **Relative Z Moves** are enabled.

This is the simplest way to work, as you set your focus according to the manufacturer's directions and LightBurn will treat that height as the correct height to start from.

> **Info**
>
> For DSP devices, LightBurn must be connected to your machine to use Relative Z moves.

Optimize Z Moves

By default, LightBurn always retracts the Z back to the starting height after completing a shape with a Z Offset. This is done to prevent the laser head from crashing into the workpiece.

With **Optimize Z Moves** enabled, LightBurn will only issue Z moves when the Z height actually changes. If you know your material is flat and that none of the Z moves will result in the laser moving low enough to run into anything on your work table, this can save a lot of time.

OMTech Polar

Enable this if using an OMTech Polar device.

Troubleshooting

- If you aren't seeing a setting you expected to see, check which tab you're in and make sure it's applicable to your laser.

Related Topics

- Settings/Preferences
- Machine Settings

- Devices

Dimensions / Units

Per-device unit and dimension settings.

Read More

Basic Settings

The most commonly used device settings.

Read More

GCode

Modify GCode output for GCode Devices.

Read More

Additional Settings

Settings for simulating previews and estimating time.

Read More

Custom GCode

Advanced GCode customization for the Custom GCode device type.

Read More

Galvo and Basic Settings

The most commonly used device settings for Galvo lasers.

Read More

Ports and Laser Settings

Configure a Galvo laser's ports and laser source.

Read More

External Axes Settings

Configure motorized X, Y, and Z axes for EZCad3 lasers.

Read More

Additional EZCad3 Settings

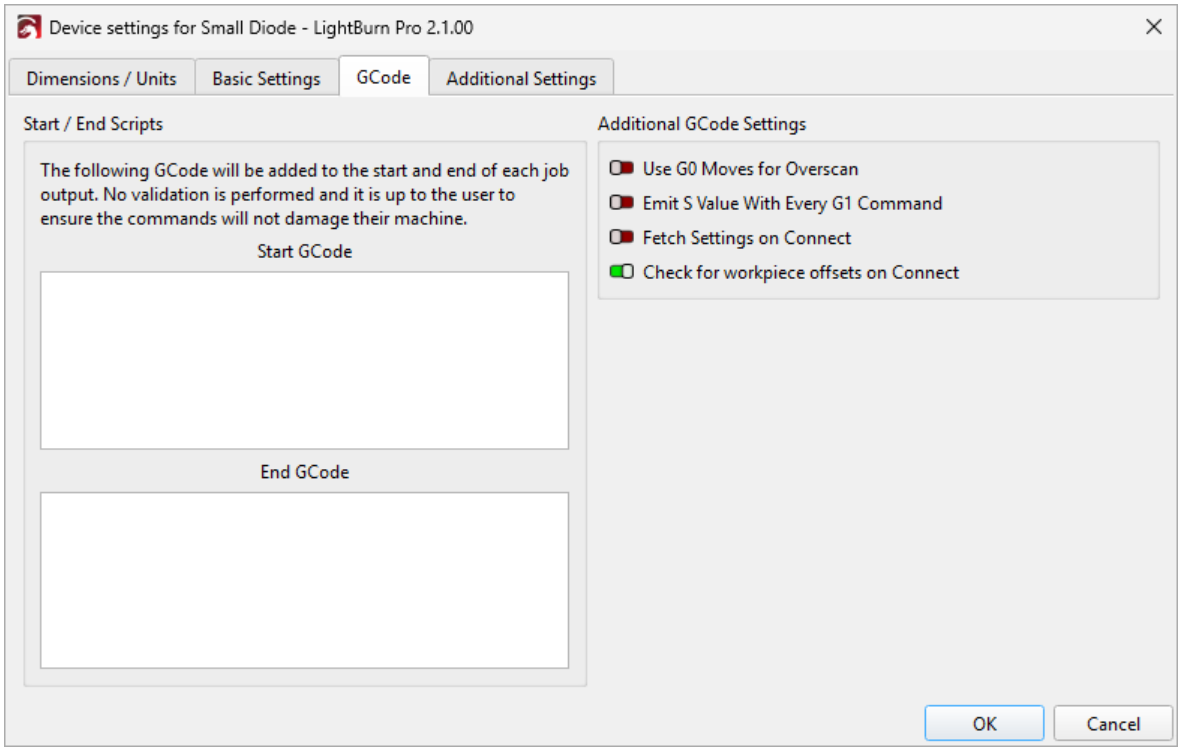
Additional configuration for EZCad 3 lasers. Most users should not change these manually.

Read More

GCode

The **GCode** tab of **Device Settings** allows for modification of the GCode output for GCode devices. Please see Device Settings: Custom GCode if you are using a Custom GCode device.

Click an option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



The GCode tab of Device Settings

Start / End Scripts

Some devices may need additional commands before or after running a job. GCode entered into these boxes will be added to the start or end of each job LightBurn sends to your device.

> **Warning**
>
> LightBurn does not validate whether these commands are valid or safe. Any modifications here are done at your own risk.

Use G0 Moves for Overscan

Enable this setting to use G0 moves for overscan instead of G1. Some controllers, such as FabCreator Smoothieware boards, have a non-zero power value for their minimum output, causing them to burn during the overscan portion of an engraving. If you find that your laser doesn't shut off completely for overscan moves, try enabling this setting.

Emit S Value With Every G1 Command

Enable this setting to have LightBurn specify the output power of the laser with every move command, rather than only when the power level changes. Enable this setting if your laser does not turn back on after travel moves.

Fetch Settings on Connect

Enable this setting to have LightBurn read your laser's firmware settings from the controller each time it connects.

Check For Workpiece Offsets on Connect

Enable this setting to have LightBurn check whether there are workpiece offsets present when connecting to the controller.

For devices that don't comply fully with GRBL standards, you may need to disable this settings. It will also disable the G54 offset warning.

Troubleshooting

- If you aren't seeing a setting you expected to see, check which tab you're in and make sure it's applicable to your laser.

Related Topics

- Settings/Preferences
- Machine Settings
- Devices

Dimensions / Units

Per-device unit and dimension settings.

Read More

Basic Settings

The most commonly used device settings.

Read More

GCode

Modify GCode output for GCode Devices.

Read More

Additional Settings

Settings for simulating previews and estimating time.

Read More

Custom GCode

Advanced GCode customization for the Custom GCode device type.

Read More

Galvo and Basic Settings

The most commonly used device settings for Galvo lasers.

Read More

Ports and Laser Settings

Configure a Galvo laser's ports and laser source.

Read More

External Axes Settings

Configure motorized X, Y, and Z axes for EZCad3 lasers.

Read More

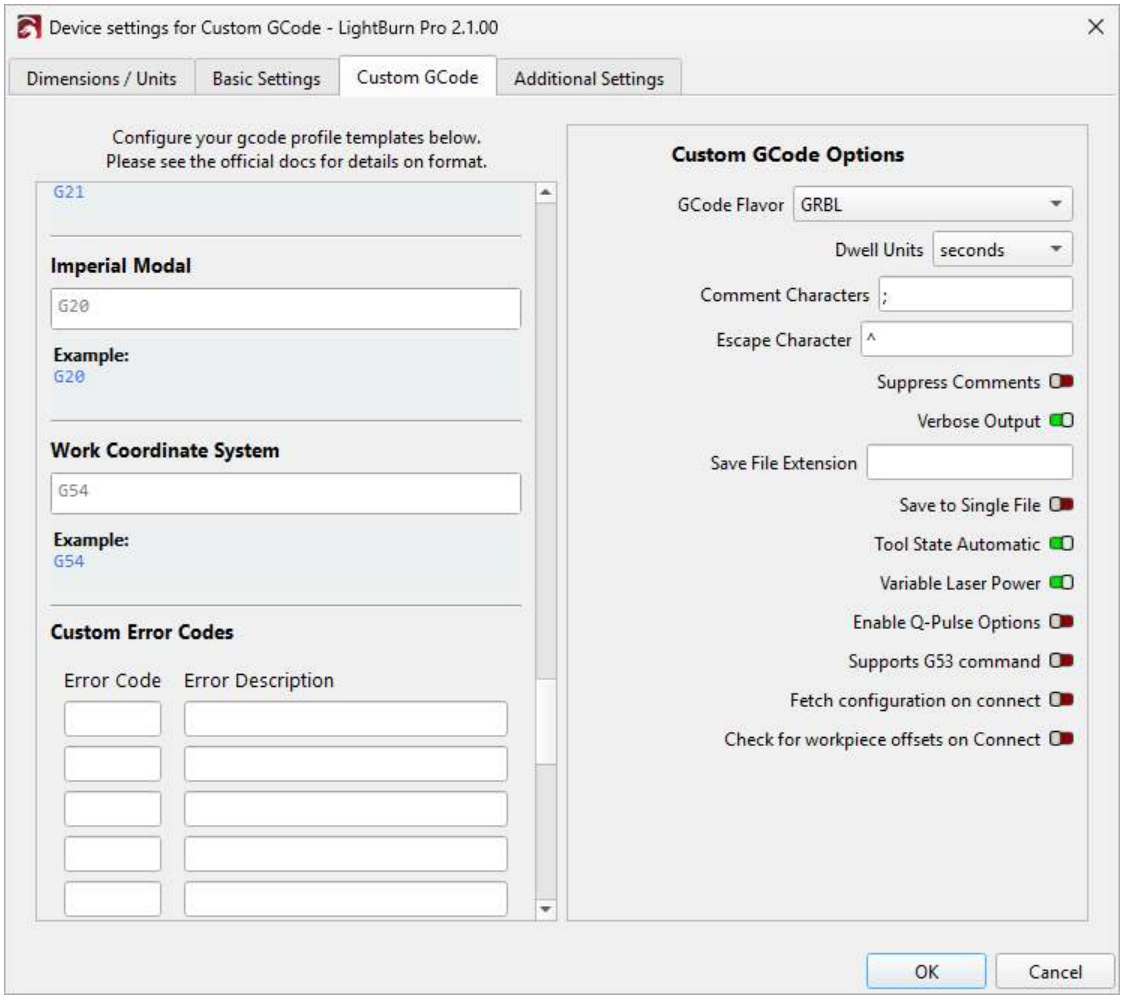
Additional EZCad3 Settings

Additional configuration for EZCad 3 lasers. Most users should not change these manually.

Read More

Custom GCode

The **Custom GCode** tab of **Device Settings** allows for advanced GCode customization when using the Custom GCode device type.



The Custom GCode tab of Device Settings

Custom GCode Options

In this section, you can set some baseline options for your custom GCode profile.

Name	Description
GCode Flavor	What flavor of GCode to use as a base
Dwell Units	Whether pause times are given in seconds or milliseconds
Comment Characters	What character(s) are used to indicate comments
Escape Character	The character used to escape text sequences
Suppress Comments	Enabling this will strip all comments from the GCode LightBurn generates

Name	Description
Verbose Output	Enabling this output will include additional detail in the generated GCode that can help with debugging
Save File Extension	Specify the file extension to be used, if you need something other than the default of `.gc`
Save to Single File	Enabling this will cause LightBurn to always export GCode as a single file, rather than showing the export file dialog box
Tool State Automatic	Enabling this will cause the tool to automatically turn on during a cut move
Variable Laser Power	Enabling this will allow you to control the laser's power beyond on and off
Enable Q-Pulse Options	Enabling this will allow you to control Q-Pulse settings. Only relevant with fiber laser sources
Supports G53 Command	Specify whether or not the device supports the G53 command. Disable for WeCreat devices
Fetch configuration on connect	Specify whether or not to fetch the device's configuration on connection. Disable for WeCreat devices
Check for Workpiece Offsets on Connect	Specify whether or not to check for workpiece offsets when connecting to the machine

GCode Commands

This section lists each command that can be customized. Leaving a command blank will leave it unchanged from the default based on your Custom GCode Options.

- The title indicates what command you're working with.
- The text box contains the default value in gray text, if there is a default. If there is no default value, the text box is blank.
- Below the text box is a preview of what the output for the command would look like, based on current settings.

Hover over a section for more detailed information. For information on variables to use in the commands, see Custom GCode Variables.

New in LightBurn 2.1

- **G20** (set units to imperial) and **G21** (set units to metric) modals are now editable GCode blocks. They default to simply `G20` and `G21`.
- If your device uses custom error codes and/or alarm codes, you can enter those codes along with a description under **Custom Error Codes** and **Custom Alarm Codes**.
- Added **Work Coordinate System** to select which Work Coordinate System to use.

Custom GCode Variables

These variables can be used in your GCode templates. Some variables are only available for MillMage.

Variable	Description	Availability
`x`	X Axis position	
`y`	Y Axis position	
`z`	Z Axis position	
`r`, `a`, `u`, `rot`, `rotation`	Rotary axis position	
`x_probe`	X probe offset	MillMage-only
`y_probe`	Y probe offset	MillMage-only
`z_probe`	Z probe offset	MillMage-only
`s`, `speed`	Movement speed	
`feed`, `feedrate`	Movement speed	MillMage-only
`p`, `power`	Laser Power	
`rpms`, `spindlespeed`	Spindle speed or RPM/s	MillMage-only
`dwell`	Dwell time	
`tool`	Tool number	

Custom GCode Profile Templates

Template	Description
Home	Machine homing routine.
Focus Z	Z axis laser focus (if applicable).
User Start Script	Custom script to run at the start of the job.
User End Script	Custom script to run at the end of the job.
Tool On	Enable the tool.
2nd Tool On	Enable the 2nd tool (if applicable).
Tool Off	Disable the tool.
Fire On	Fire the laser.
Fire Off	Stop firing the laser.
Pierce	Fire the laser initially without moving, to help pierce through difficult materials.
Dwell	Pause movement for a specified time.
Air On	Turn on air assist.

Template	Description
Air Off	Turn off air assist.
Layer Start	Commands to run before the start of each layer.
Cut Move	Command to move while cutting. Typically uses `G1`.
Raster Move	Command for moving while engraving a raster image.
Rapid Move	Command to move without cutting. Typically uses `G0`.
Safe Rapid	Rapid move at a safe height above the material surface.
Abort	Immediate abort/stop command.
Pause	Immediate pause command.
Resume	Command to resume after pausing.
Job End	Job completion command.
Get Position	Command to get current position.
Get Status	Command to get current status. Often the same as Get Position.
Metric Modal	Set the machine to metric units.
Imperial Modal	Set the machine to imperial units.
Work Coordinate System	Set the work coordinate system to use, such as G54, G55, G56, etc.
Custom Error Codes	Specify custom error codes, if your device uses them.
Custom Alarm Codes	Specify custom alarm codes, if your device uses them.

Custom GCode Macros

Relative and absolute mode are controlled by the software, so if you need to change it within your GCode template, use the following macros to prevent the software overriding your changes. You must always use `|RESTORE|` when you're done to ensure a return to the previous mode.

Macro	Description
`|REL|`	Enter relative mode
`|ABS|`	Enter absolute mode
`|RESTORE|`	Restore previous mode

Optional and Required Values

Surrounding the letter before a variable with `[]` will tell LightBurn that this portion of the command has not changed since the last emit. Putting a `*` in front of any variable will tell LightBurn to skip the entire line if it has not been changed.

- `G1 X{x} Y{y} Z{z}` will always output the X, Y, and Z values.

- `G1 [X]{x} [Y]{y} [Z]{z}` will skip any values that have not changed.
- `M106 S{*power}` this will output the `M106` command only if it has been changed.

Reserved / Prohibited Characters

Certain characters — `[]`, `<>`, and `{}` — are reserved for use by LightBurn and MillMage and are only permitted as shown above in Optional and Required Values.

New in LightBurn 2.1

To escape text sequences in LightBurn 2.1 or newer, surround the text in the escape character specified in Custom GCode Options.

Non-Printable / Non-Typeable Characters

Control characters can be output by using `\x` followed by the two digit hex value of the character. For instance, `\x18` represents an immediate job abort on a GRBL device.

Troubleshooting

- If you aren't seeing a setting you expected to see, check which tab you're in and make sure it's applicable to your laser.
- See Device-Specific Troubleshooting for help with device-specific configuration.

Related Topics

- Settings/Preferences
- Machine Settings
- Devices

Dimensions / Units

Per-device unit and dimension settings.

Read More

Basic Settings

The most commonly used device settings.

Read More

GCode

Modify GCode output for GCode Devices.

Read More

Additional Settings

Settings for simulating previews and estimating time.

Read More

Custom GCode

Advanced GCode customization for the Custom GCode device type.

Read More

Galvo and Basic Settings

The most commonly used device settings for Galvo lasers.

Read More

Ports and Laser Settings

Configure a Galvo laser's ports and laser source.

Read More

External Axes Settings

Configure motorized X, Y, and Z axes for EZCad3 lasers.

Read More

Additional EZCad3 Settings

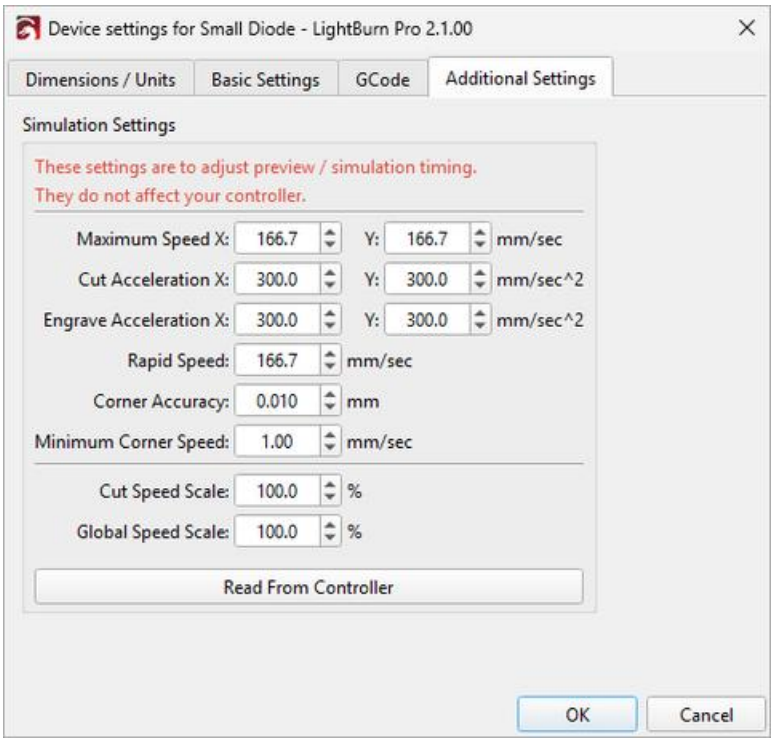
Additional configuration for EZCad 3 lasers. Most users should not change these manually.

Read More

Additional Settings

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=s1jCqB233N0

The **Additional Settings** tab of **Device Settings** controls the settings used in Preview for simulating a preview and estimating time. Changing these settings will not change anything about how the job actually runs, but can make time estimates more accurate.



The Additional Settings tab of Device Settings

Read From Controller

Use the **Read From Controller** button to sync these settings with your laser's internal settings, providing much more accurate time estimates without manual adjustment.

Manual Adjustment

> **Note**
>
> Y Axis Speed and Acceleration values are often set lower than X. Due to the greater mass of the Y Axis gantry, many lasers must travel slower in the Y dimension to avoid slipping.

Setting	Description
Maximum Speed X/Y	The maximum speed at which the laser can travel on the X and Y axes, individually.

Setting	Description
Cut Acceleration X/Y	The maximum rate at which the laser can speed up or slow down during Line operations.
Engrave Acceleration X/Y	The maximum rate at which the laser can speed up or slow down during Fill operations.
Corner Accuracy	A value used to estimate cornering speed. Corner Accuracy is not read from the controller. Read more about how this value works for lasers with GRBL controllers here.
Minimum Corner Speed	The slowest possible speed the laser will turn corners at.
Cut Speed Scale	Scales cut speed estimates by this percentage when simulating job times. Not read from controller. Use this value to manually fine-tune estimates.
Global Speed Scale	Scales all estimates by this percentage when simulating job times. Not read from controller. Use this value to manually fine-tune estimates.

Troubleshooting

- If you aren't seeing a setting you expect to see, check which tab you're in and make sure it's applicable to your laser.

Related Topics

- Settings/Preferences
- Machine Settings
- Devices

Dimensions / Units

Per-device unit and dimension settings.

Read More

Basic Settings

The most commonly used device settings.

Read More

GCode

Modify GCode output for GCode Devices.

Read More

Additional Settings

Settings for simulating previews and estimating time.

Read More

Custom GCode

Advanced GCode customization for the Custom GCode device type.

Read More

Galvo and Basic Settings

The most commonly used device settings for Galvo lasers.

Read More

Ports and Laser Settings

Configure a Galvo laser's ports and laser source.

Read More

External Axes Settings

Configure motorized X, Y, and Z axes for EZCad3 lasers.

Read More

Additional EZCad3 Settings

Additional configuration for EZCad 3 lasers. Most users should not change these manually.

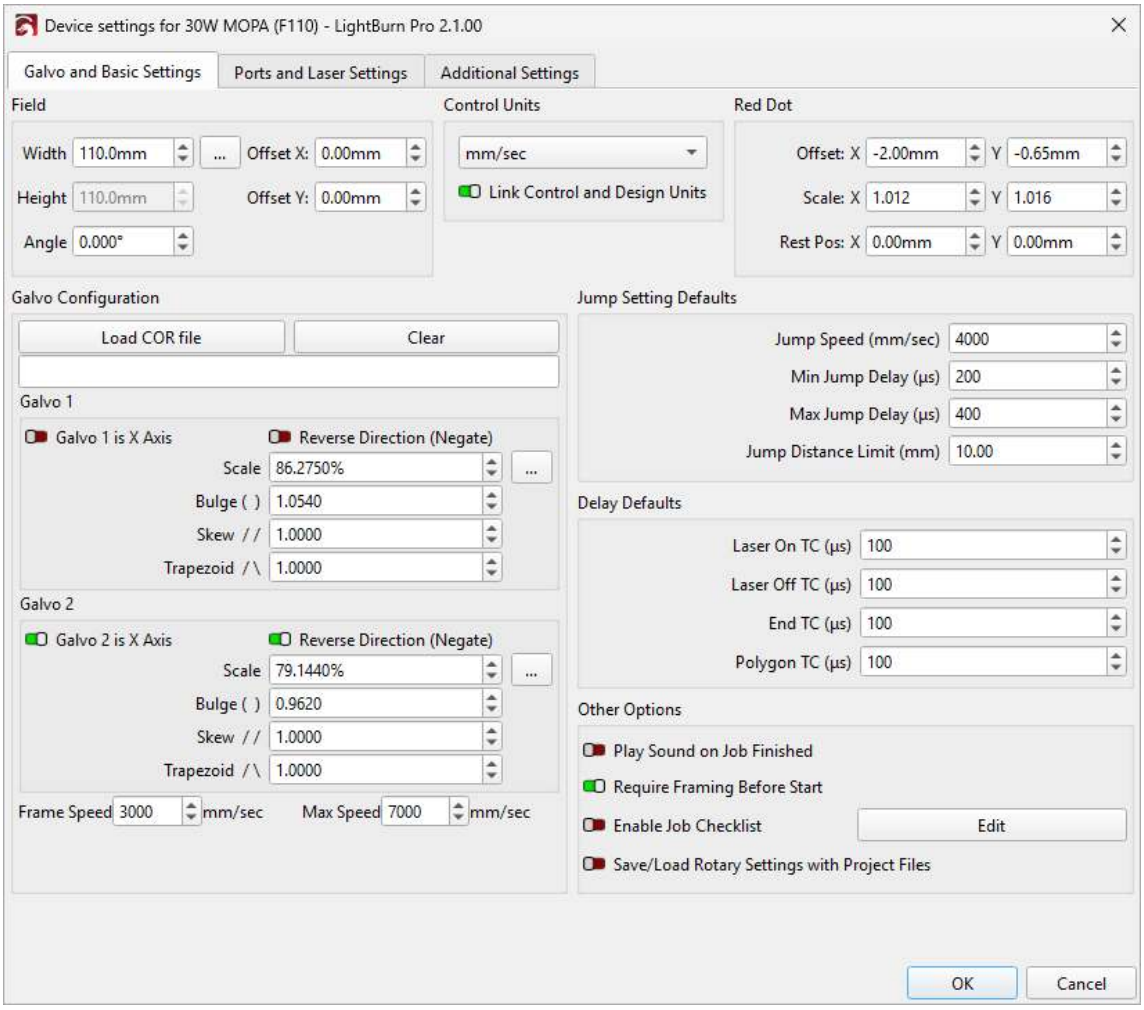
Read More

Galvo Basic Settings

The **Galvo and Basic Settings** tab of **Device Settings** is the Galvo laser version of Basic Settings and contains the most commonly used settings for Galvo lasers.

> **Info**
>
> If you used the EZCAD importer when setting up your Galvo laser, these settings will be pre-configured for you.

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



The Galvo and Basic Settings tab of Device Settings

Field

These settings refer to the field — the area that the lens installed in a Galvo laser can physically cover when focused. Changing lenses requires going through setup and calibration again, and we recommend setting up additional devices for each lens you use with your Galvo laser.

Setting	Description
Width/ Height	The size of the field for the currently installed lens. Sets the size of the Workspace in LightBurn.
Offset X/Y	Fine-tune the position of the field by offsetting it if needed.
Angle	Rotational correction. Use this to compensate for the Galvo head being rotated relative to the physical workholding.

Control Units

This section sets the Control Units for your currently active device, which are the units used for speed and distance when controlling your device.

If you prefer to design in inches but use metric units for speed, choose **Inches / mm/sec** for CO2 and Galvo lasers or **Inches / mm/min** for diode lasers.

Red Dot

The red dot laser is separate from the marking laser and is used for framing and positioning. It fires through the same optics as the marking laser, but may require fine-tuning for alignment. For additional help, see our video on Galvo Laser Red Dot Alignment below.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=2Ec4-F49aV0

Setting	Description
Offset X/Y	Fine-tune the position of the framing output.
Scale X/Y	Fine-tune the size of the framing output.
Rest Pos X/Y	Set the resting position of the red dot when not in use.

Galvo Configuration

Load COR File

This section allows you to import, apply, and clear EZCAD lens correction files, also known as COR files.

- If your EZCAD folder contains a COR file, it will automatically load when using the EZCAD importer during Setup.
- If you skipped the importer, or you create a new COR file, you can load it here.
- If you don't have a COR file, you can enter lens correction settings manually in the Per Galvo Settings below.
- To remove a COR file, click the **Clear** button.

Per Galvo Settings (Galvo 1/Galvo 2)

Each Galvo head is responsible for an axis and can be fine-tuned individually.

Setting	Description
Galvo # is X Axis	Specifies which axis is responsible for horizontal motion. If your output is rotated 90 degrees, switch which Galvo is toggled on.
Reverse Direction (Negate)	Reverses the direction for the selected Galvo. If output to an axis is mirrored or inverted, toggle this setting.
Scale	Percentage by which to scale output for a given axis.
...	Calibrate the scaling for each axis by measuring lasered output to ensure that the requested size of the artwork matches the measured size. See Calibrate Lens Calibration for more information.
Bulge	Adjusts for distortion near the center of the engraving field.
Skew	Adjusts for distortion near the edges of the field.
Trapezoid	Adjusts corner tapering to fix keystoning.

Frame Speed

The travel speed for the red dot laser while framing.

Max Speed

The maximum travel speed for the laser during cuts.

Jump Setting Defaults

Jump Settings control how the laser moves between cuts. Each jump causes the mirror to shake, with higher speeds and longer distances increasing the shaking. If you have wobbly lines at the start of a marking move, you may need to increase the delay and/or reduce the speed of jumps.

Setting	Description
Jump Speed	The speed at which the Galvo mirrors move between cuts. Higher values will make your job run faster at the cost of increased vibration for the mirrors.
Min Jump Delay	The minimum time the laser will wait to allow the Galvo mirrors to stop shaking.
Max Jump Delay	The maximum time the laser will wait to allow the Galvo mirrors to stop shaking.
Jump Distance Limit	Any jump above this limit will use the **Max Jump Delay** setting above. Jumps below this limit will use a delay value between the minimum and maximum settings.

Delay Defaults

Delay settings control the delay between starting (or stopping) mirror movement and firing the laser. **Laser On/Off TC** and **End TC** control the delays for the start and end points of a path. **Polygon TC** controls the delays for other corners within a path.

Setting	Description
Laser On TC	Microseconds the laser will wait to fire after the mirrors begin to move. Too high a value will leave gaps, while too low a value may cause excessive burning at the start of a cut.
Laser Off TC	Microseconds to delay turning off the laser at the end of a marking move. Too high a value may cause excessive burning at the end of a cut, while too low a value may cause gaps.
End TC	Microseconds to pause the laser's movement after completing a cut.
Polygon TC	Microseconds the Galvo head will pause when turning corners. Too long a delay may cause excessive burning, while too short a delay can cause rounded or cut-off corners.

Other Options

Setting	Description
Play Sound on Job Finished	If enabled, the computer will play a sound when your current run is complete.
Require Framing Before Start	If enabled, the **Start Button** in the Laser Window will open the Live Framing window instead of starting the laser directly.
Enable Job Checklist	If enabled, LightBurn will display a window with user-configured text before running the laser. The user will need to press **OK** to confirm they're ready before the laser will start. This is often used for a checklist of reminders before running the laser. Use the **Edit** button to create or edit the checklist.
Save/Load Rotary Settings with Project Files	If enabled, LightBurn will include rotary settings when saving a project, and will load those settings when re-opening the file. This can streamline job setup when working with rotaries.

Troubleshooting

- If you aren't seeing a setting you expected to see, check which tab you're in and make sure it's applicable to your laser.

Related Topics

- Settings/Preferences
- Devices

- Galvo Lens Calibration

Dimensions / Units

Per-device unit and dimension settings.

Read More

Basic Settings

The most commonly used device settings.

Read More

GCode

Modify GCode output for GCode Devices.

Read More

Additional Settings

Settings for simulating previews and estimating time.

Read More

Custom GCode

Advanced GCode customization for the Custom GCode device type.

Read More

Galvo and Basic Settings

The most commonly used device settings for Galvo lasers.

Read More

Ports and Laser Settings

Configure a Galvo laser's ports and laser source.

Read More

External Axes Settings

Configure motorized X, Y, and Z axes for EZCad3 lasers.

Read More

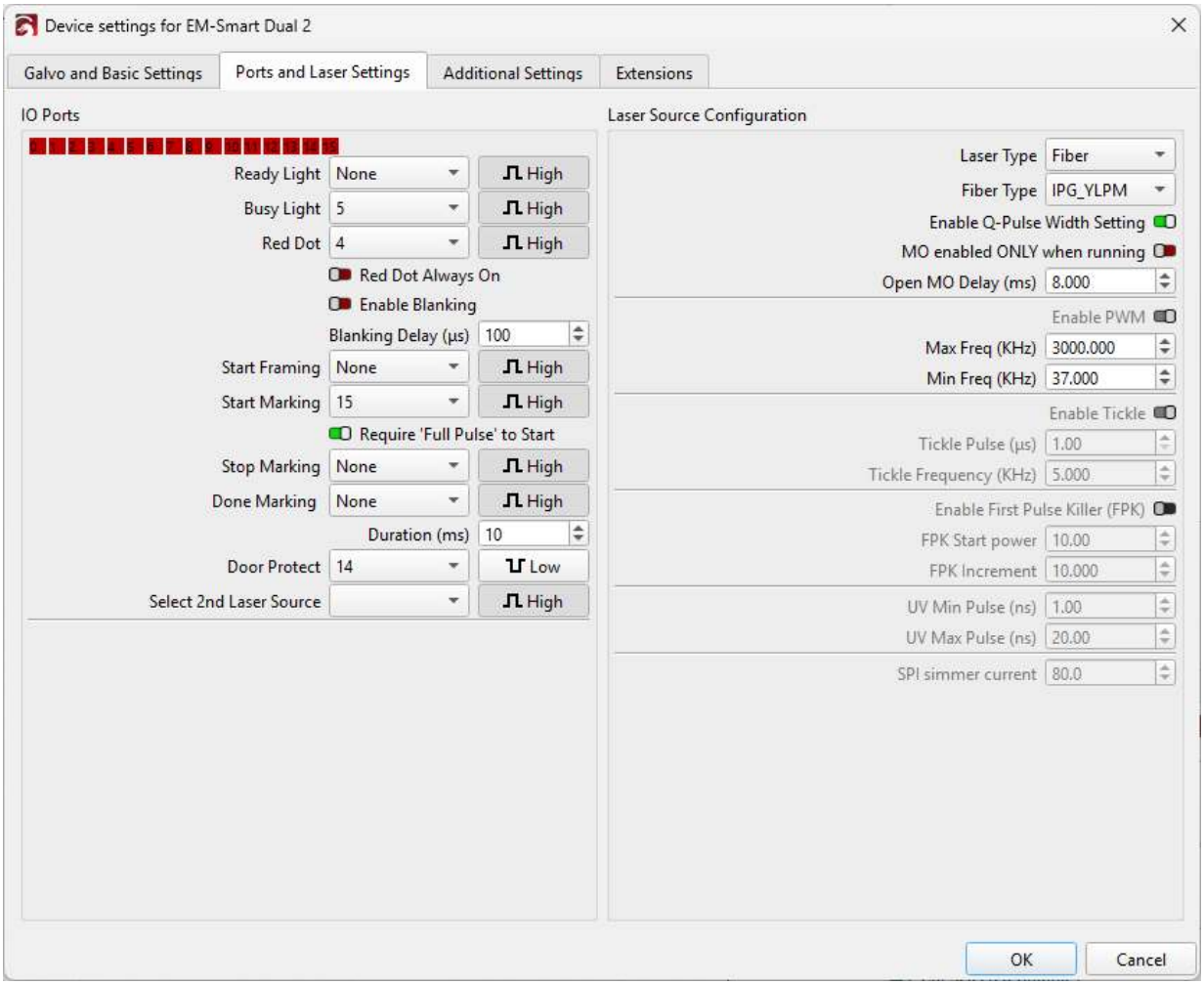
Additional EZCad3 Settings

Additional configuration for EZCad 3 lasers. Most users should not change these manually.

Read More

Ports and Laser Settings (Galvo)

The **Ports and Laser Settings** tab of Device Settings allows you to configure your laser's Input and Output (IO) ports and laser source.



The Ports and Laser Settings tab of Device Settings

IO Ports

This section controls external switches or features attached to the laser's control board. The available options will depend on your particular laser.

> **Low vs High**
>
> Most boards use a High signal to indicate that a port is active, but some instead use a Low signal as the trigger. If most or all of your ports are green by default, you'll probably need to change the trigger type to Low using the button next to the port selection.

The numbered boxes at the top show the current state of each port. Red indicates that the port is currently Low (off), while green indicates that the port is currently High (on).

Use the dropdowns to select a port to assign to a feature, and use the **High**/**Low** button to set the trigger type. If the port is set to **None**, the feature is disabled or unavailable.

Setting	Description
Ready Light	Select the appropriate port to enable on/off control of LEDs mounted on the Galvo head to illuminate your work.
Busy Light	If you have a light hooked up to indicate the laser is in use, select the appropriate port to allow LightBurn to enable it when the laser is in use.
Red Dot	If you are using an external switch to enable the red dot laser, select the appropriate port.
Red Dot Always On	Enable this setting to force the red dot laser to be always on.
Enable Blanking	If enabled, attempts to turn off the red dot laser between drawing shapes. Not supported on all devices. Disable if your laser will not frame when you select Contour Framing.
Start Framing	Select the appropriate port to enable an external switch to start framing.
Start Marking	Select the appropriate port to enable an external switch – typically a foot switch – to start an engraving job. If not using a foot switch, it's best to set this to **None** to prevent accidental firing as a result of static or electrical noise.
Require 'Full Pulse' to Start	Enable this toggle to require the external switch to activate and be released before starting a job.
Stop Marking	Select the appropriate port to use an external switch to stop running an engraving job.
Done Marking	Select the appropriate port to send a signal when an engraving job is complete.
Door Protect	Select the appropriate port to look for a signal from an enclosure's safety interlock to prevent unprotected laser firing.
Select 2nd Laser Source	Select the appropriate port to allow the selection of a second laser source.
Enable Analog Output (Laser 2)	Enable an analog output signal on the second laser source. Only available if a valid port is selected for the second laser source.
Focus Light	If you have a second red-dot laser for focusing, select the correct port to enable it.

Laser Source Configuration

> **Warning**
>
> Laser Source settings should be imported from EZCAD or copied very carefully if entering manually. Incorrect settings may damage your laser.

Setting	Description
Laser Type	The source for the Galvo laser. Currently, **CO_2**, **Fiber**, **UV**, or **SPI**.

Setting	Description
Fiber Type	For Fiber lasers, the specific manufacturer or type of Fiber laser. Certain options enable additional settings.
Enable Q-Pulse Width Setting	MOPA Fiber lasers only. Shows the Q-Pulse option in Cut Settings. This setting allows fine adjustment of the laser's pulse timing. Smaller timing values increase the density of the pulse's impact.
MO Enabled ONLY When Running	MOPA fiber lasers only. This is similar to EZCad's Laser Leak Handle. If enabled, LightBurn only enables the oscillator when a job is running, not immediately on connection. Enable this if your laser marks or discolors the workpiece while idle.
Open MO Delay	MOPA fiber lasers only. Time in miliseconds to wait after enabling the master oscillator (MO) before marking, giving the laser source time to stabilize. Increase this value if the start of the first mark is weak or missing.
Enable PWM	Always enabled for Fiber and SPI lasers. Toggle on or off for CO_2 Galvo lasers, to enable or disable PWM. When enabled, adjust the valid range of minimum and maximum frequencies in the boxes below.
Enable Tickle	For CO_2 Galvo lasers only. CO_2 lasers require constant low power to fire consistently at low powers. Enable **Tickle** to provide this constant low power. Appropriate settings for Tickle Pulse and Frequency should be provided by your manufacturer.
Enable First Pulse Killer (FPK)	For CO_2 Galvo lasers only. CO_2 lasers require a higher power draw when initially turned on. FPK settings allow you to tune that initial pulse. Check with your manufacturer for settings, but this can be found through trial and error if needed.
UV Min/Max Pulse	For UV Galvo lasers only. Minimum and maximum **Q-pulse** values to use.
SPI Simmer Current	For SPI lasers only.

Z-Axis Control

Z-Axis Control is no longer part of the Ports and Laser Settings tab. See External Axes Settings for configuration and Galvo Move Window for controlling the Z axis. This currently works for EZCad3 and BSLFiber devices.

Troubleshooting

- If you aren't seeing a setting you expected to see, check which tab you're in and make sure it's applicable to your laser.

Related Topics

- Settings/Preferences

- Devices

Dimensions / Units

Per-device unit and dimension settings.

Read More

Basic Settings

The most commonly used device settings.

Read More

GCode

Modify GCode output for GCode Devices.

Read More

Additional Settings

Settings for simulating previews and estimating time.

Read More

Custom GCode

Advanced GCode customization for the Custom GCode device type.

Read More

Galvo and Basic Settings

The most commonly used device settings for Galvo lasers.

Read More

Ports and Laser Settings

Configure a Galvo laser's ports and laser source.

Read More

External Axes Settings

Configure motorized X, Y, and Z axes for EZCad3 lasers.

Read More

Additional EZCad3 Settings

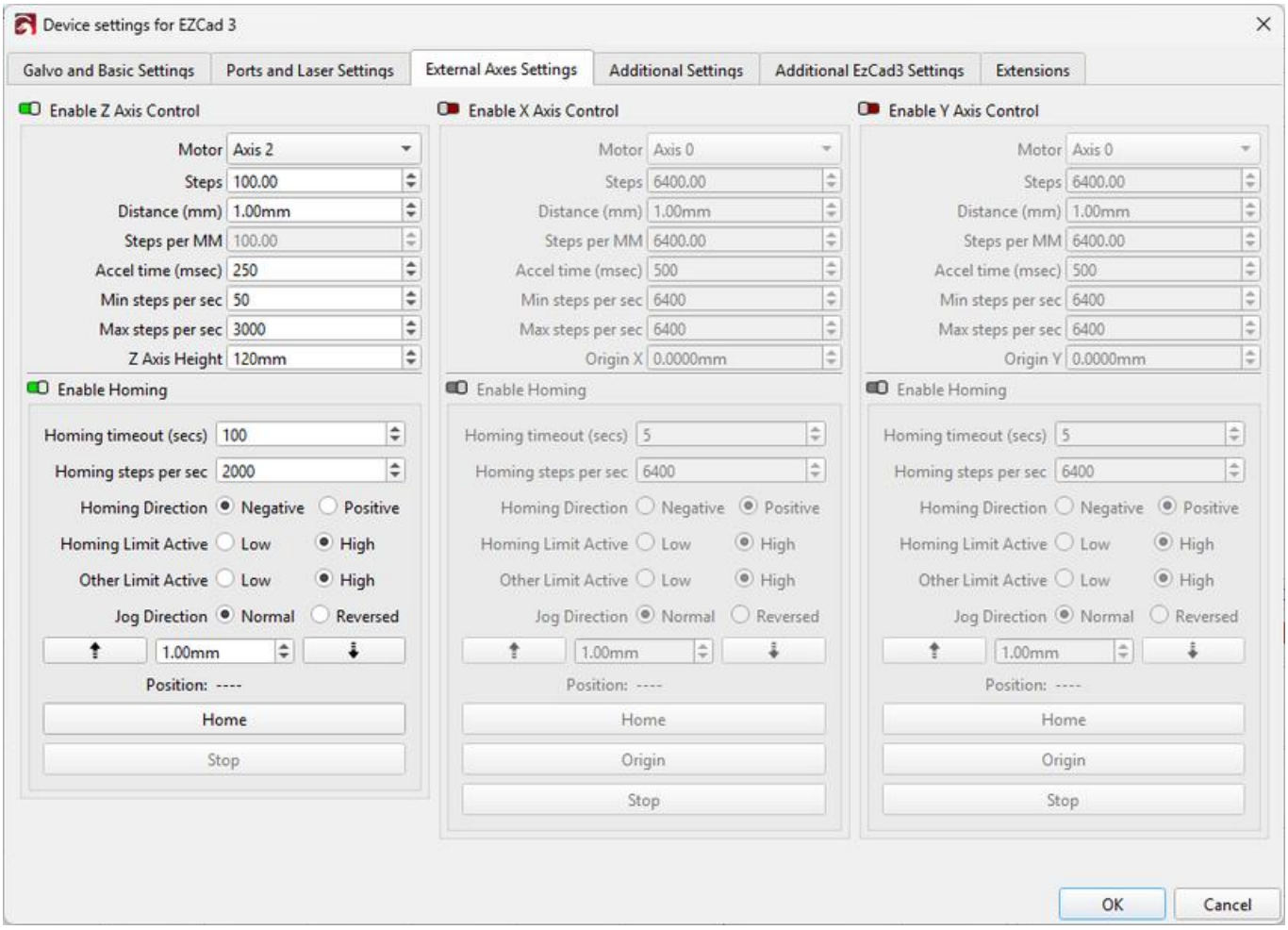
Additional configuration for EZCad 3 lasers. Most users should not change these manually.

Read More

External Axes Settings (EZCad3 / BSLFiber)

The **External Axes Settings** tab of **Device Settings** allows you to enable and configure motors controlling your laser's X, Y, and/or Z axes. This tab is currently only available for EZCad3 and BSLFiber lasers. The Z axis is always available, but only EZCad3 will have the option for X and Y axes as well.

The **X Axis**, **Y Axis**, and **Z Axis** are all configured independently, and each one has a separate homing configuration. We'll use the Z axis below as an example.



The External Axes Settings tab of Device Settings

Z Axis Control / X Axis Control / Y Axis Control



Enable Z Axis Control

Toggle this on to allow LightBurn to control your Z axis (or other selected axis). If the axis isn't motorized or is controlled in some other way, this should be disabled.

Motor

Select which motor controls this axis. If you're not sure which motor to select, pick one and attempt to (carefully) jog it a short distance to see if the correct motor moves.

Steps and Distance (mm)

These two settings work together so you can set how many steps your stepper motor takes to move a set distance. For instace, 100 steps* and 1 mm* will tell LightBurn it has to rotate the motor 100 steps to make the axis travel 1 mm.

Steps Per MM

Shows the computed Steps / Distance in steps/mm. This is not set manually - it comes from the two settings above.

Accel Time (msec)

The time it takes your motor to accelerate, in milliseconds.

Min Steps Per Sec

The minimum number of steps per second LightBurn should command your stepper motor to turn at. This sets a minimum travel speed.

Max Steps Per Sec

The maximum number of steps per second LightBurn should command your stepper motor to turn at. This sets a maximum travel speed.

Z Axis Height

Sets the total travel distance of the Z axis. This setting is only for the Z axis.

The Z axis needs to be enabled with Z Axis Height set to something other than 0 for the EZCad3 Move Window to be available.

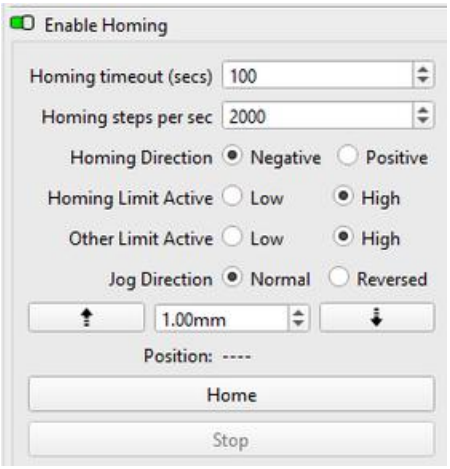
Origin X / Origin Y

Sets the coordinate used for the origin in X and Y. Origin X and Origin Y are the distance from home on the relevant axis that puts the work table directly under the laser.

If either of these axes are enabled, before beginning a job:

1. The axis will home (if it hasn't already homed).
2. The axis will travel to the origin point.
3. The job will begin.

External Axis Homing



Homing Timeout (secs)

Maximum length of time to attempt homing. Setting a limit on homing reduces the chances of damaging your machine if homing is unsuccessful, by preventing the motor from running endlessly after hitting the end of its travel.

If this is too short, your axis may not reach its home position before time runs out. If this happens, try setting a longer timeout.

Homing Steps Per Sec

How many steps your motor should spin each second while homing. Increase this value to move faster during homing, or decrease it to move slower.

Homing Direction

Sets the direction for the laser to move towards the homing switch. If the axis moves away from the homing switch when you try to home it, reverse this value.

Homing Limit Active

Use the **Homing Limit Active** setting to tell LightBurn whether to look for a high or low signal when the limit switch is reached. If this is set wrong, LightBurn will see the axis as always being in contact with the homing switch except when it actually is.

If an axis immediately reports that it's homed without actually moving to the switch, switch this value.

Other Limit Active

Similar to Homing Limit Active, but for a limit switch at the other end of the axis.

Jog Direction

If the jog controls move the axis the opposite direction you expect, switch the **Jog Direction** to reverse the direction the axis moves.

Jog Test

Use the **Jog Controls** here to test your jogging settings.

Position

Shows the current position of the selected axis.

Home

Use the **Home** button to test your homing settings.

Origin (X and Y axes only)

Use the **Origin** button to move the axis to its configured origin.

Stop

Use the **Stop** button to stop moving the selected axis.

Dimensions / Units

Per-device unit and dimension settings.

Read More

Basic Settings

The most commonly used device settings.

Read More

GCode

Modify GCode output for GCode Devices.

Read More

Additional Settings

Settings for simulating previews and estimating time.

Read More

Custom GCode

Advanced GCode customization for the Custom GCode device type.

Read More

Galvo and Basic Settings

The most commonly used device settings for Galvo lasers.

Read More

Ports and Laser Settings

Configure a Galvo laser's ports and laser source.

Read More

External Axes Settings

Configure motorized X, Y, and Z axes for EZCad3 lasers.

Read More

Additional EZCad3 Settings

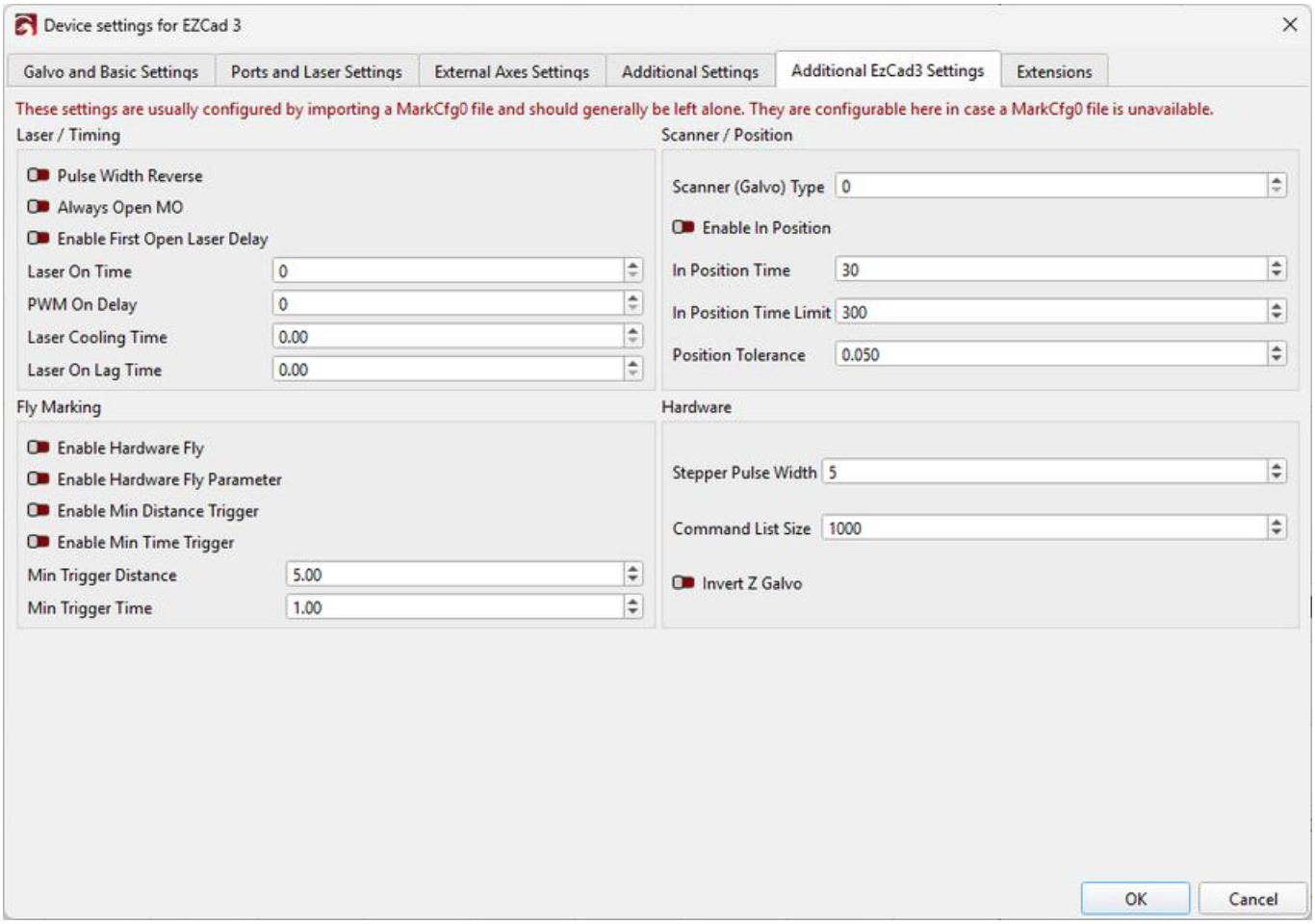
Additional configuration for EZCad 3 lasers. Most users should not change these manually.

Read More

Additional EzCAD3 Galvo Settings

The **Additional EzCad3 Settings** tab of LightBurn's Device Settings allows you to manually configure your EzCad3 Device if a `MarkCfg0` file is unavailable.

> **Most users should not change these settings**
>
> The settings here are normally configured when you import a `MarkCfg0` file while adding your EzCad3 laser to LightBurn, but we've made the settings manually configurable in case the configuration file is unavailable. If you're not sure what you're doing, you should probably leave this tab alone unless told otherwise by support.



The Additional EzCad3 Settings tab of Device Settings

Dimensions / Units

Per-device unit and dimension settings.

Read More

Basic Settings

The most commonly used device settings.

Read More

GCode

Modify GCode output for GCode Devices.

Read More

Additional Settings

Settings for simulating previews and estimating time.

Read More

Custom GCode

Advanced GCode customization for the Custom GCode device type.

Read More

Galvo and Basic Settings

The most commonly used device settings for Galvo lasers.

Read More

Ports and Laser Settings

Configure a Galvo laser's ports and laser source.

Read More

External Axes Settings

Configure motorized X, Y, and Z axes for EZCad3 lasers.

Read More

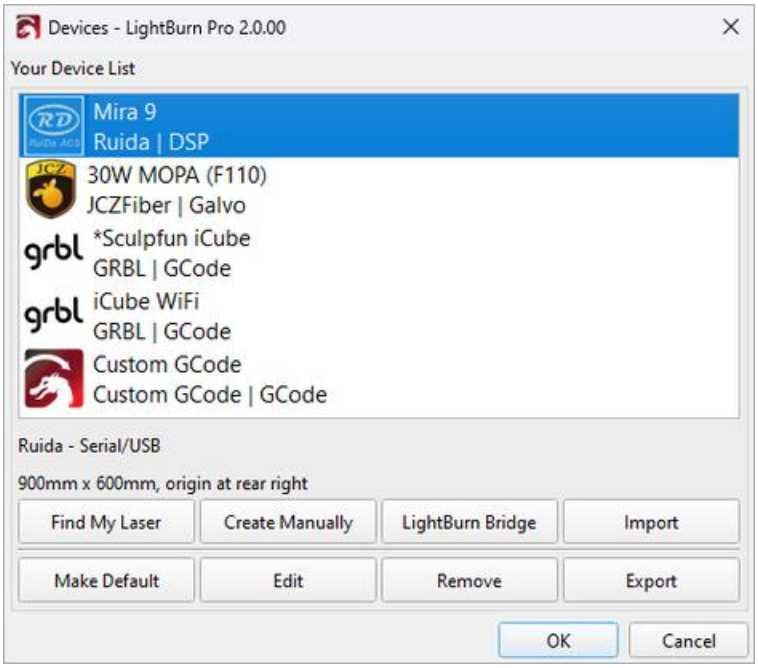
Additional EZCad3 Settings

Additional configuration for EZCad 3 lasers. Most users should not change these manually.

Read More

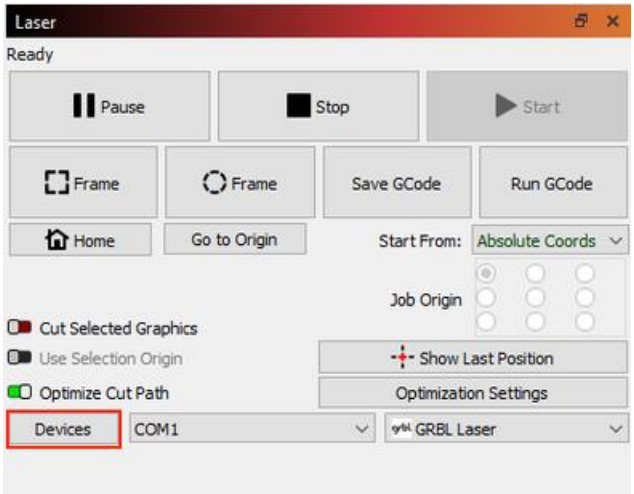
Devices

The **Devices** window shows all the devices you've added to LightBurn. Use this window to add new devices, **Import** and **Export** devices, and manage existing devices.



Accessing the Devices Window

To open the **Devices** window, click the Devices button in the Laser Window.



Managing Devices

Device List

The main section of the **Devices** window lists all the devices you've added to LightBurn. Click a machine to select it and see a device summary below the Device List. Some buttons in this window are only available with a device selected.

On a fresh install, this list will be empty. LightBurn will ask you to set up a device in order to continue. There are a few ways to do this:

- Use Find My Laser to attempt to automatically detect a laser connected over USB.
- Use Create Manually to manually set up a laser that can't be automatically detected.
- Import a pre-configured device profile from a `.lbdev`, `.lbzip`, or `.lbvendor` file.

You must add at least one device in order to use LightBurn. If you don't have a laser yet, you can choose **Create Manually** and select **No Machine** to create a device to act as a placeholder.

Find My Laser

Starts the Device Discovery Wizard to attempt to automatically detect a laser connected over USB. If your laser is not detected, use Create Manually.

Create Manually

Starts the New Device Wizard to manually set up a laser. If you don't have a laser yet, a **No Machine** option is available.

LightBurn Bridge

Starts the Bridge Discovery Wizard to set up a LightBurn Bridge device.

Import

Imports a pre-configured device profile to recreate a device exported from another LightBurn installation. Select a `.lbzip`, `lbdev`, or `.lbvendor` file containing one or more device profiles to import them into your current LightBurn installation. See Migration for more information.

Export

> **What about `.lbdev` files?**
>
> Guides and videos based on older versions of LightBurn may still refer to `.lbdev` files, which contain a single device profile. As of LightBurn 2.0, this button now creates a User Bundle for the device. User Bundles can be imported just like the `.lbdev` files, but have additional uses described in our Migration guide.

Exports a `.lbzip` User Bundle file containing the device profile for the selected device. Use this to back up a device or recreate the device on a different computer. See Migration for more information.

Make Default

Sets the currently selected device as the default. If you have multiple devices, use this to set which device is active when LightBurn launches. The current default device has an asterisk before the name.

Edit

Opens the New Device Wizard to go back through the device creation process on the selected laser. You can also double-click any device in the list to re-open the New Device Wizard for that device.

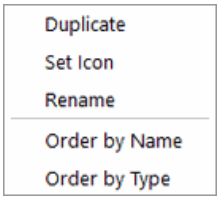
See Device Settings for help with settings not covered in the device creation process.

Remove

Removes the selected laser from LightBurn. Make sure you have the right device selected!

Devices Context Menu

Right-click any device in the list to open a context menu with additional options.



Duplicate

Opens the **New Device Wizard** and populates each step with identical settings from the selected device, with the exception of appending **(Duplicate)** to the name. You can adjust any setting, including the name, during this process.

Set Icon

Opens your system's file browser, where you can select a custom icon for the selected device from your computer. The icon must be 1024 x 1024 pixels or fewer in size.

Rename

Rename the selected device.

Order by Name

Rearranges the list of devices in alphabetical order.

Order by Type

Rearranges the list of devices by type. Sections of types are arranged in alphabetical order.

OK and Cancel

Click **OK** to save any changes you've made and exit the **Devices** window.

Click **Cancel** or X-out of the window to discard any changes you don't want to save.

Troubleshooting

- If the button you want is disabled, make sure you have a device selected.
- If you haven't added any lasers yet and try to close the window, LightBurn will open a window asking you to either go back and create a device profile, create a dummy machine, or quit LightBurn. Use the **Create a dummy machine** button if you don't have a laser to add yet.

Related Topics

- Migration
- Device Settings
- Add a Laser

LightBurn Device Types

While we can't support every device, LightBurn supports a large and growing number of device types, including Ruida, GCode, xTool, Epilog, and more. Whether you're using Find My Laser or Adding a Laser Manually, LightBurn's supported device types are listed below.

We have device-specific directions for some machines, such as xTool and ACCELaser devices. Click the links below to go right to those pages.

Device Type	Supported Connection Types	License Compatibility	Notes
No Machine	No Connection	Core, Pro	Placeholder device to use when you don't have an actual laser to set up.
Custom GCode	USB, Network	Core, Pro	Create a custom GCode device, which allows you to modify GCode for devices that don't perfectly match one of our other device types and require generating customized GCode. Offers several flavors, which act as your starting template: GRBL, GRBL M3, grblHAL, FluidNC, GRBL-STM, Smoothie, Marlin, LinuxCNC, UCCNC, Duet / RepRap, Duet / RepRap HTTP, Creality Falcon HTTP, WeCreat, WeCreatHTTP, Masso, Generic.
ACCELaser	USB, Network	Pro	Support for the ACCELaser HD1 Pro and HD1 Ultra. Additional setup documentation coming soon.
BSLFiber	USB	Pro	Support for BSLFiber/SeaCAD galvo lasers.
CanCam	USB	Core, Pro	Support for CanCam devices.
Cubiio 2	Network	Core, Pro	Support for the Cubiio 2 device.
Emblaser 1 A3	USB	Core, Pro	Support for the Emblaser 1 A3.
Emblaser 1 A4	USB	Core, Pro	Support for the Emblaser 1 A4.
Emblaser 2 / Core	USB, Network	Core, Pro	Support for the Emblaser 2 and Emblaser Core.
Emblaser Pro	USB, Network	Core, Pro	Support for the Emblaser Pro.
Epilog (Labs)	Network	Pro	Support for some Epilog lasers. This is a labs feature - it's still in development and subject to change. Please use with caution and report unexpected behavior.
EZCad 2 (JCZFiber)	USB	Pro	Support for EZCad2 galvo lasers.
EZCad 3 (Labs)	USB	Pro	Support for EZCad3 galvo lasers. This is a labs feature - it's still in development and subject to change. Please use with caution and report unexpected behavior.

Device Type	Supported Connection Types	License Compatibility	Notes
Fabkit	USB	Core, Pro	Support for the Fabcreator Fabkit.
Gerbil-STM	USB	Core, Pro	Support for Gerbil-STM devices.
GRBL	USB, Network	Core, Pro	Support for devices using GRBL firmware. The most common device type for GCode devices, used in a lot of lower-cost or DIY devices.
GRBL-LPC	USB, Network	Core, Pro	Support for devices using GRBL-LPC firmware, which is a specific implementation of GRBL for 32-bit NXP LPC microcontrollers. It's extremely rare for current devices.
GRBL-M3 (1.1e or earlier)	USB	Core, Pro	Support for devices using older versions of GRBL firmware. If your device uses GRBL 1.1e or earlier and you can't upgrade to a newer version, use the GRBL-M3 device type.
iLaser	USB	Core, Pro	Support for the iLaser.
LinuxCNC	USB	Core, Pro	Support for LinuxCNC devices. For laser cutters, this will usually be a CNC router with a laser diode added on.
Marlin	USB	Core, Pro	Support for devices running Marlin firmware.
Ruida	USB, Network, LightBurn Bridge	Pro	Support for devices with Ruida controllers.
Smoothieware	USB	Core, Pro	Support for devices running Smoothieware firmware.
SnapMaker (Marlin)	USB	Core, Pro	Support for SnapMaker devices, which use a variant of Marlin firmware.
TopWisdom	USB	Pro	Support for devices wth TopWisdom controllers.
Trocen-AWC	USB, Network	Pro	Support for devices with Trocen controllers.
xTool	USB, Network	Core, Pro	Support for xTool devices.
ZhiYuan	Network	Pro	Support for devices with ZhiYuan controller.

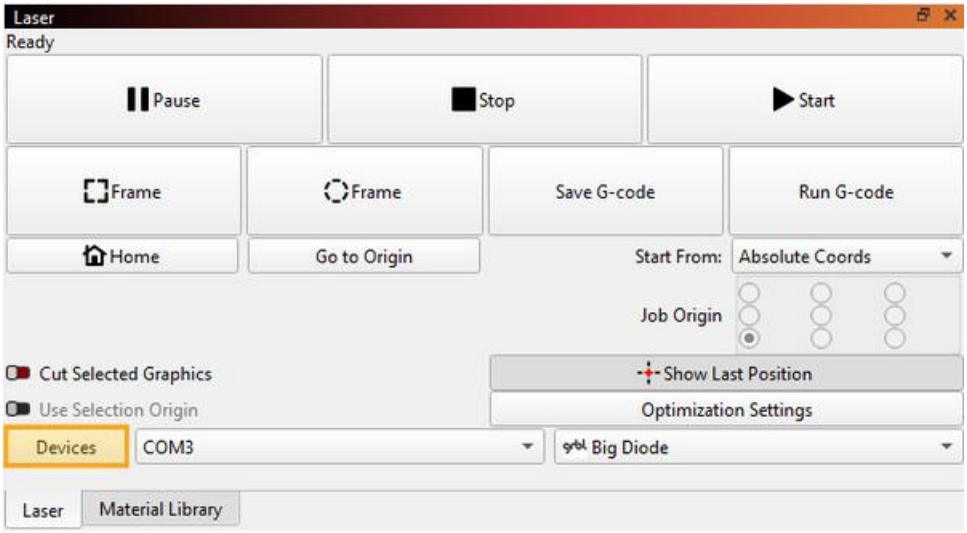
ACCELaser

ACCELaser HD1 Series machines are flying-galvo style lasers that come with special cut/engrave parameters and additional features like LiDAR depth scanning.

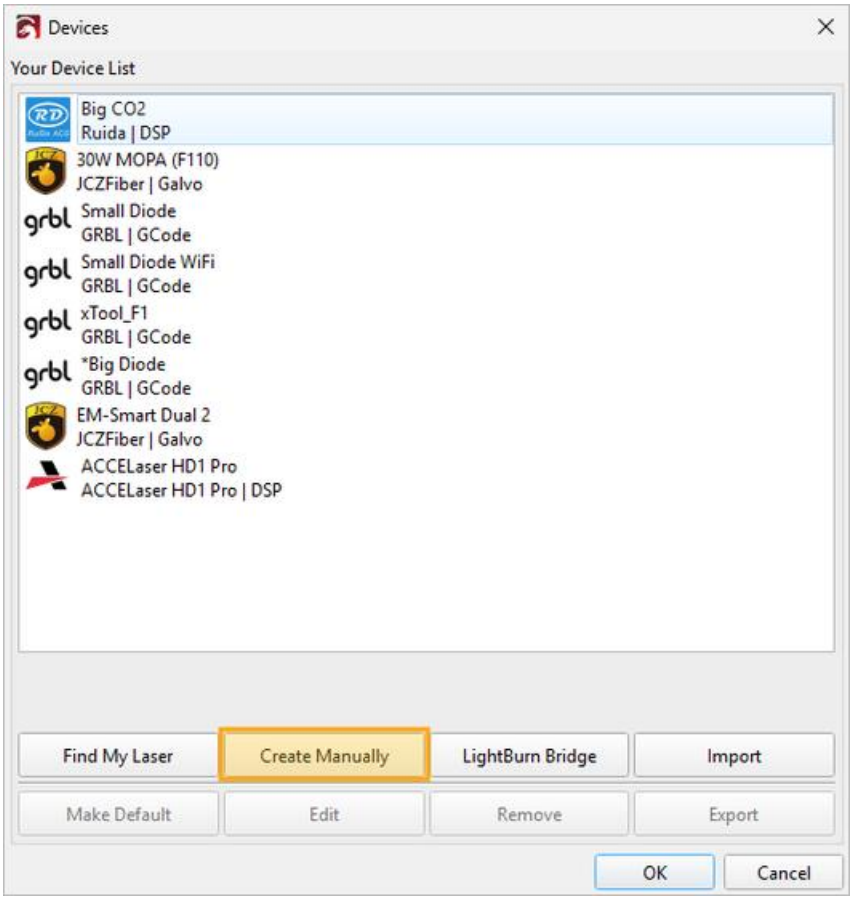
This page provides information on setting up and operating an ACCELaser machine with LightBurn.

Adding an ACCELaser Device to LightBurn

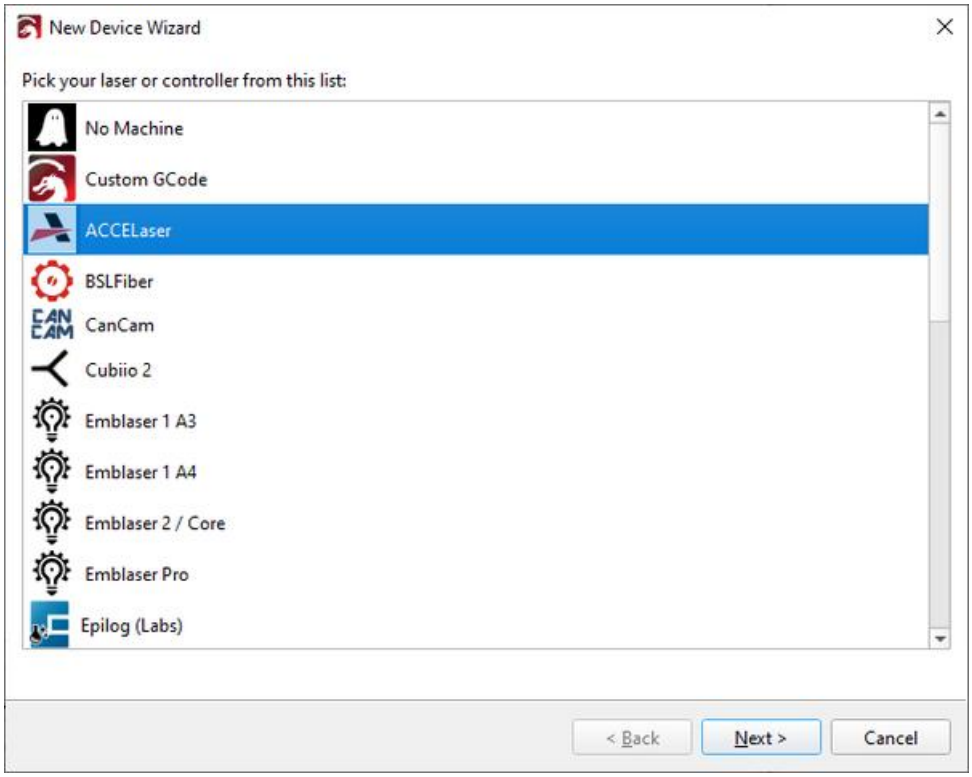
1. Use the ACCELaser Handy companion app to connect your machine to your local WiFi network.
2. Connect your ACCELaser machine to your computer by USB.
3. In LightBurn, click the **Devices** button (highlighted in the screenshot below) in the Laser window to open the Devices window.



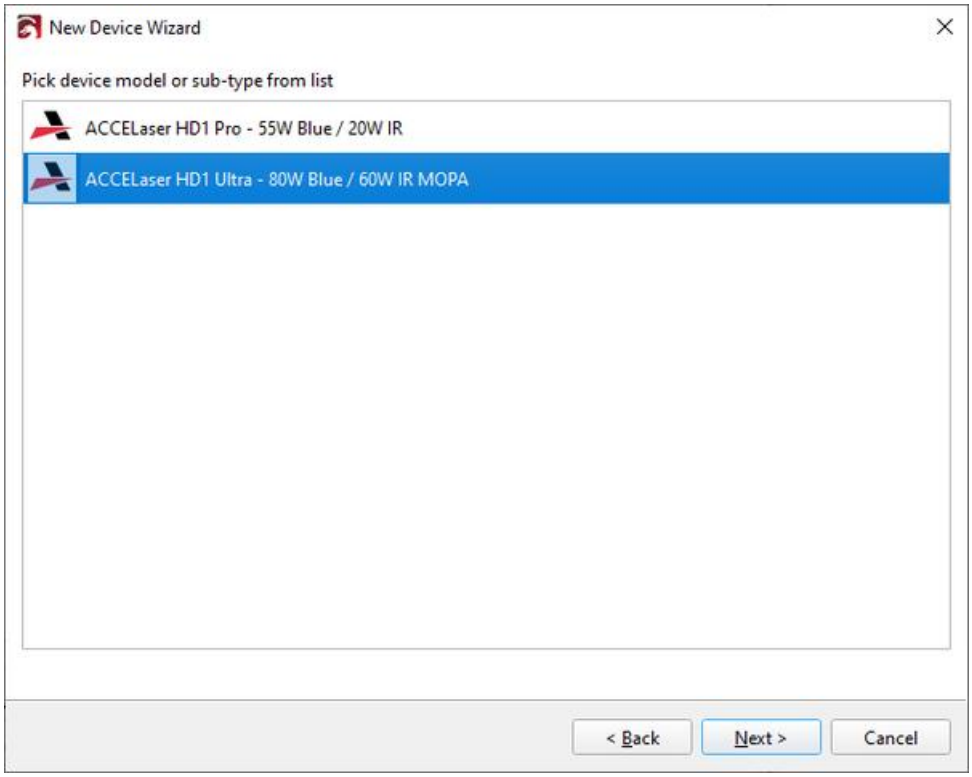
4. In the **Devices** window, click the **Create Manually** button under the device list to open the **New Device Wizard**.



5. In the **New Device Wizard**, scroll down and select **ACCELaser** from the list of device types. Click Next to continue.



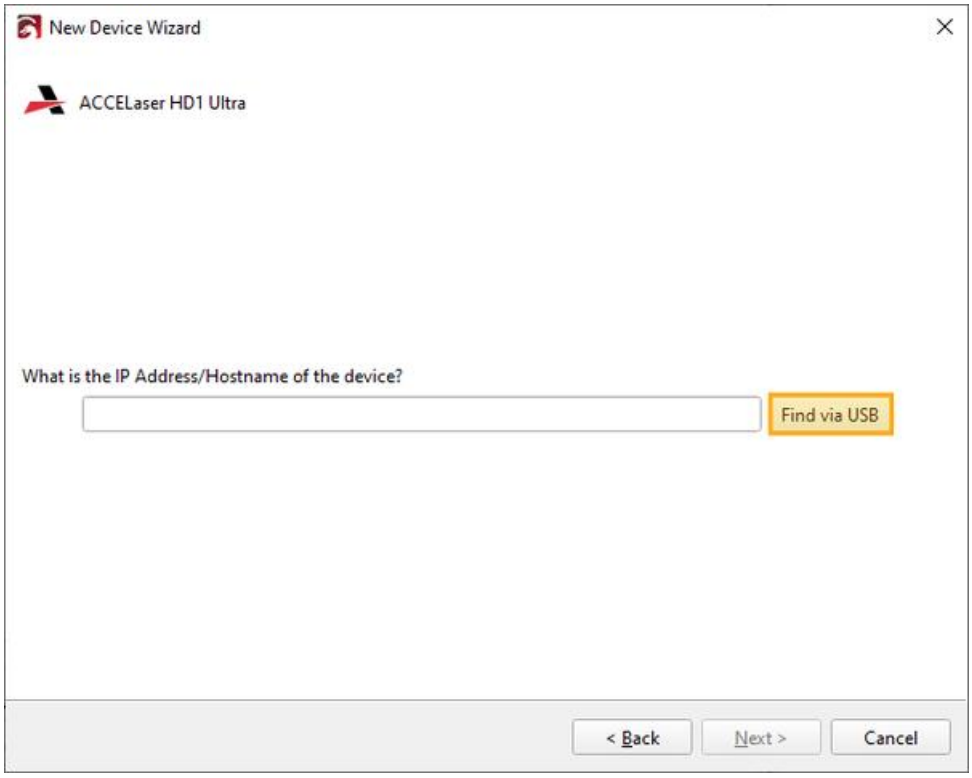
6. Select your HD1 model from the list and click Next to continue.

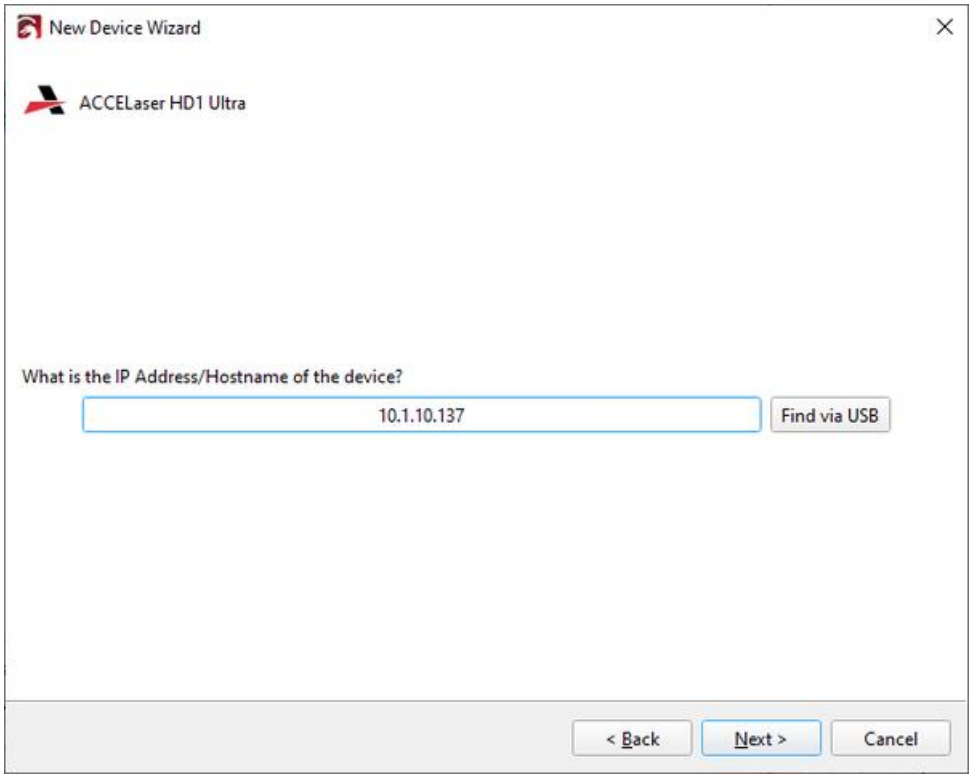


7. Click **Find via USB** to automatically detect your machine's IP address and enter it in the field.

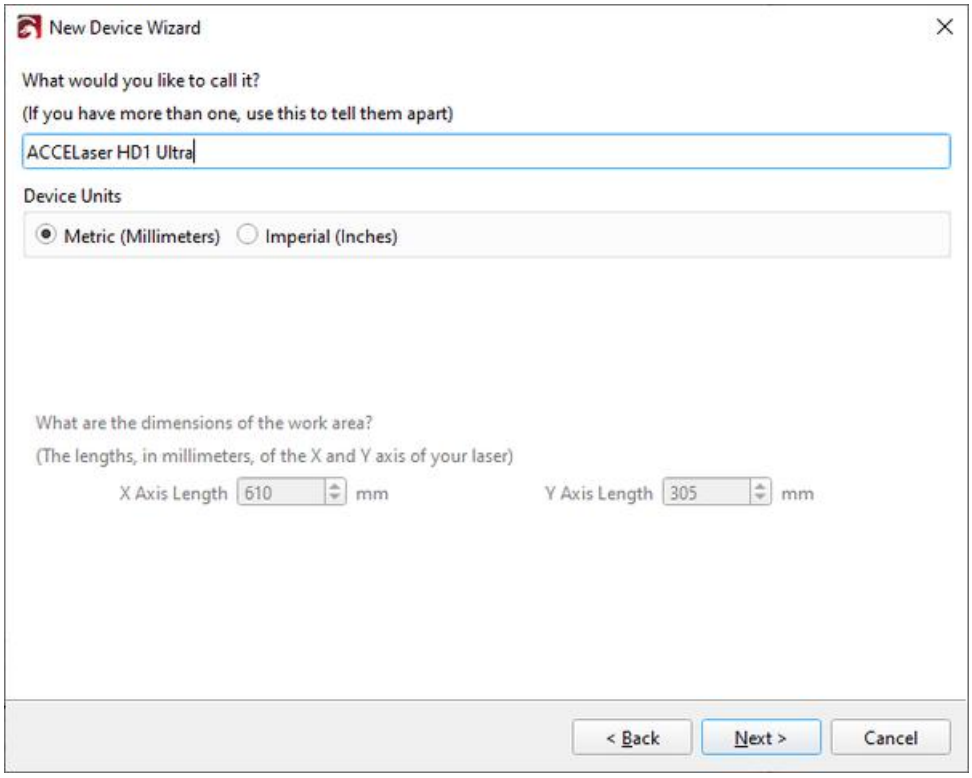
   Alternatively, you can find identify your device's IP address using the ACCELaser Handy app, and enter it manually.

   Click Next when done.

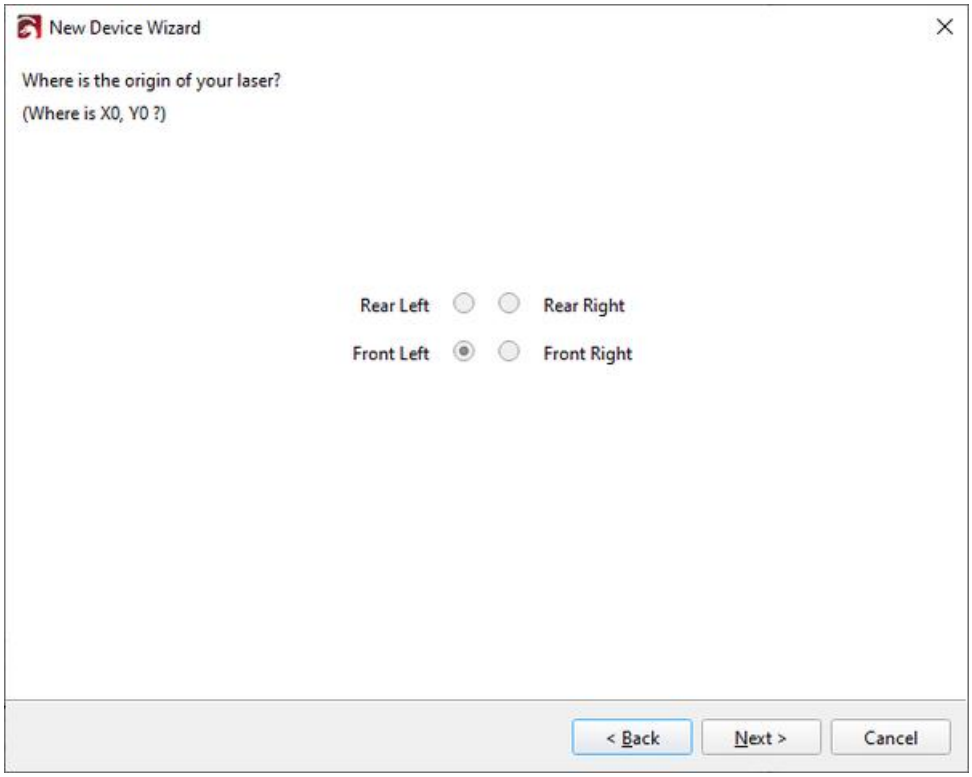




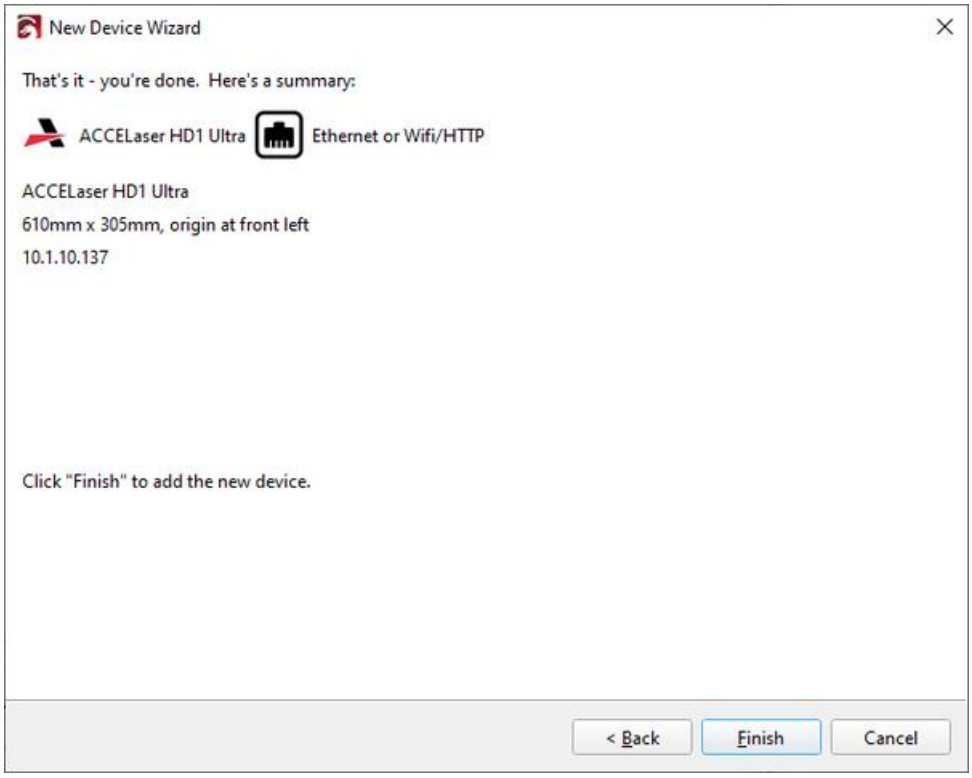
8. Your device's working dimensions are entered automtically. You can choose to rename your device from the default model name, and set your **Device Units** to either **Metric** or **Imperial**. You can adjust units at any time later in the Device Settings window.



9. The device origin is automatically set to the front left. Click Next to continue.



10. Make sure the summary on the final screen seems correct — since most parameters were entered automtically, you'll only need to verify the IP address if you entered it manually. Click **Back** to go back and make changes, or **Finish** to save your changes.



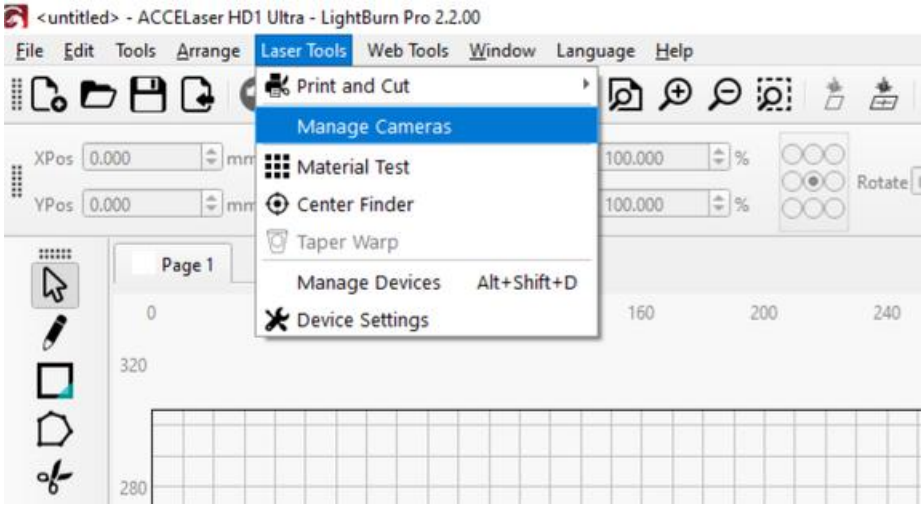
Cameras

ACCELaser HD1 series machines include two cameras:

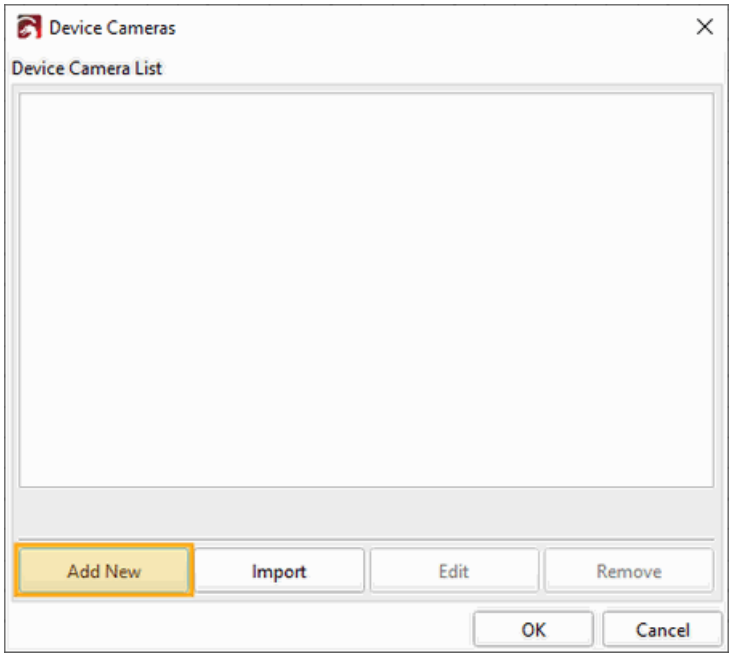
- **Overhead Camera**. Mounted to the top of the laser's lid, capable of capturing the entire work area. This option is useful for quick captures where high percision is not necessary, such as placing graphics you plan to cut from a sheet of material.
- **Head Mounted Camera**. Mounted on the laser head, capable of capturing smaller areas in finer detail. To capture larger areas, the Head Mounted camera can jog to take multiple captures, which are stitched together in your Overlay. This option is useful when you require greater precision, such as engraving accurately onto an object. See Head Mounted Camera for more information.

Adding the Cameras

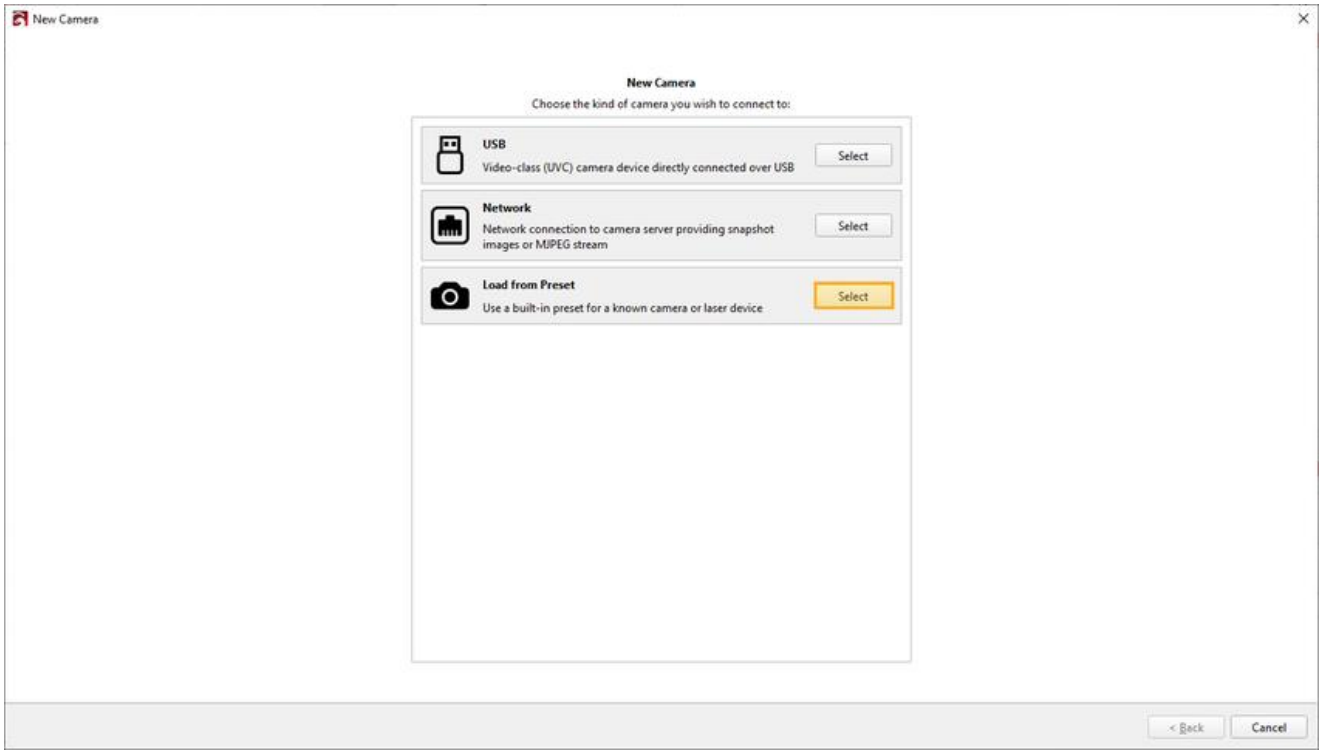
1. Go to Laser Tools → Manage Cameras.



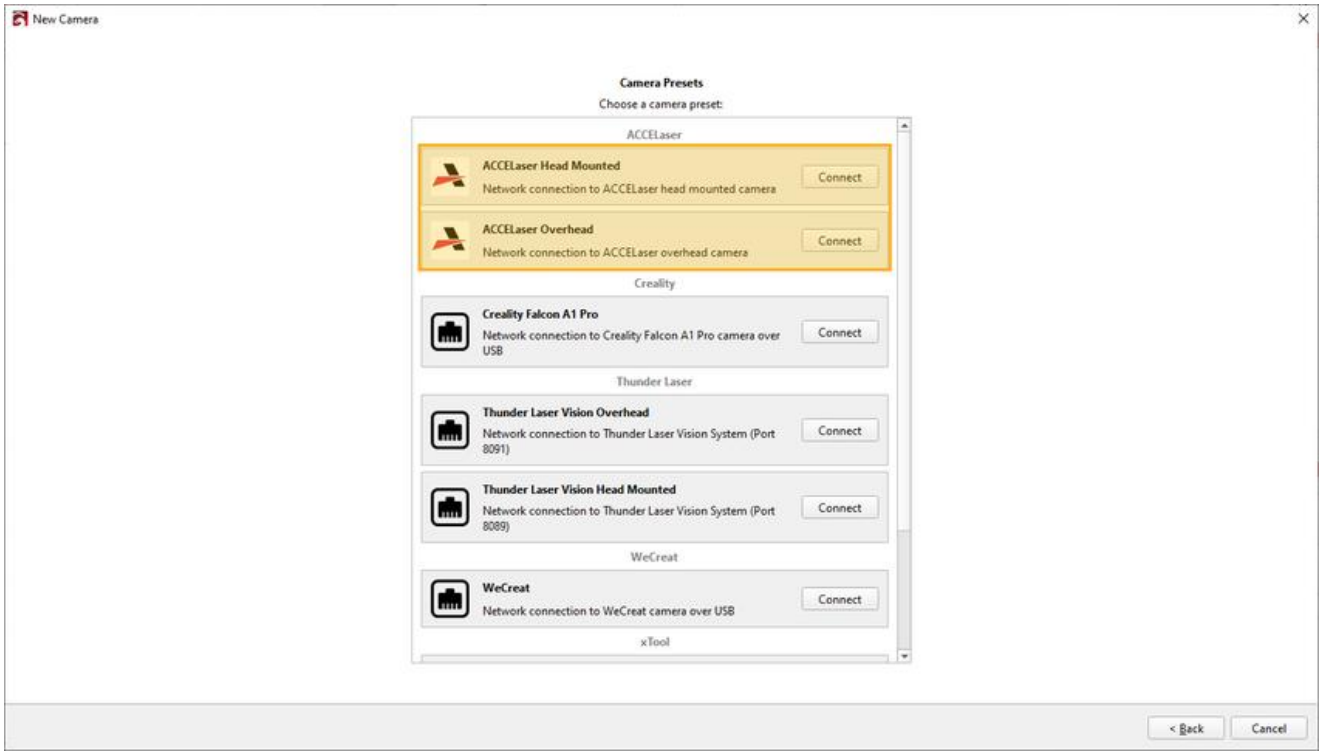
2. Click **Add New**.



3. Choose **Load from Preset**.



4. Select **ACCELaser Head Mounted** or **ACCELaser Overhead**.



5. Repeat steps 1-3 process, then select the preset for the second camera.

Using the Cameras

Overhead Camera

1. Place your material on the bed.
2. Home or move the laser head to the top-left corner of the work area, so it doesn't block the Overhead Camera's view.
3. In the Cameras / Overlays Window, click Update Overlay for the **Overhead Camera**. An Overlay of your entire work area appears in your Workspace.
4. Position your graphics over your material.

Head Mounted Camera

1. Place your material on the bed.
2. Optionally, capture an Overhead Camera overlay to see the general location of the material you want to capture more precisely.
3. Select the area you want to capture. You can select the graphics you plan to engrave, or create a Rectangle set to a Tool Layer, place it over your material, and select it.
4. In the Cameras / Overlays Window, click the **Capture Selected** button for the **Head Mounted Camera**. The laser head jogs over the selected area, taking multiple captures that are stitched together into a detailed Overlay in your Workspace.
    - To capture the entire work area instead, click the **Capture Workspace** button.
5. Use the detailed Overlay to precisely position your graphics.

See our full Head Mounted Camera and Cameras / Overlays Window documentation for more information.

LiDAR

ACCELaser HD1 machines use a LiDAR ranging module on the laser head to measure the distance to your material. You can use it to automatically focus your laser to your material's surface, or to scan the surface of a curved object, or objects with multiple surfaces at varying heights, and create a height map the laser follows as it engraves.

Focus Z

Use **Focus Z** to automatically focus to the surface of material.

1. Move the laser head over your material. If you have already captured a camera overlay, you can use Position Laser to quickly jog over the material's location.
2. Click **Focus Z** in the Move Window or Cameras / Overlays Window.

> **Warning**
>
> Make sure the laser head is positioned over your material before using Focus Z.

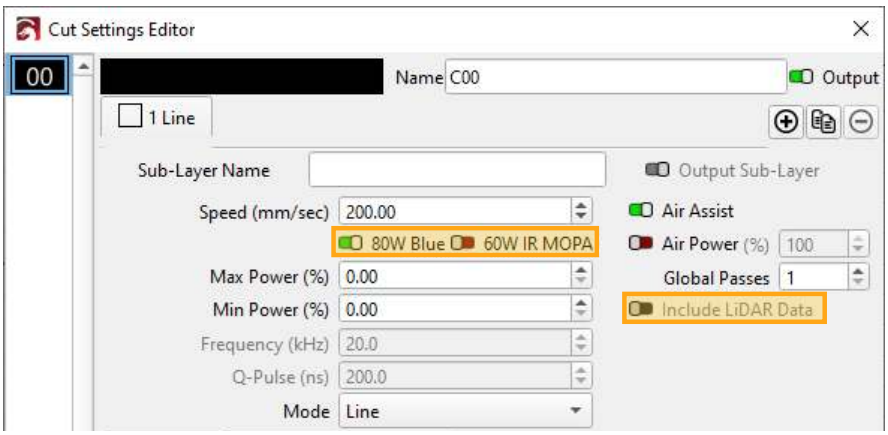
Depth Scanning

Use Depth Scanning to create a height map of the surface of your object or material. After capturing this data, the laser can dynamically adjust its focal distance and stay in focus across curved or uneven surfaces.

1. Place your material on the bed and capture a Camera Overlay.
2. Select the area you want to scan. You can select the graphics you plan to engrave, or create a Rectangle set to a Tool Layer, place it over your material, and select it.
3. In the Main Toolbar or the Cameras / Overlays Window, click the Scan Selected button.
   - Optionally, before scanning, you can click Settings and adjust the Scan Interval. Smaller Intervals lead to higher resolution height maps, at the cost of longer scanning times.
4. To scan the entire bed instead, click Scan Full Workspace.
5. When the scan is finished, an overlay of the height map appears in your Workspace.
6. Enable Include LiDAR Data in the Cut Settings Editor for all layers you want to engrave on the scanned surface.

See our full Depth Scanning / LiDAR documentation for more information.

Cut/Engraving Settings



Most Cut/Engraving settings work exactly the same as they do for any other type of laser. See our general Cut Settings Editor documentation for information on these settings.

Source Selection

Beneath the Speed field, you can select which laser source to use to process graphics on the current layer.

- **Blue** diode laser
- **IR** (or **IR MOPA**) fiber laser

The infrared source depends on your model — the HD1 Pro uses a 20W fiber laser, while the HD1 Ultra uses a 60W MOPA fiber laser.

Include LiDAR Data

Enable **Include LiDAR Data** to have the laser dynamically adjust focal height to follow the height map created by a Depth Scan. Use this option for curved or uneven objects, or for large-area jobs where your material may not be perfectly flat.

You must run a Depth Scan of the work area before enabling this option.

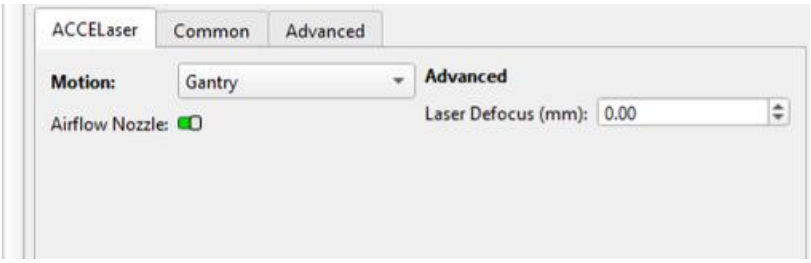
Motion

Motion parameters control how your laser processes your graphics.

- Gantry utilizes the system that physically drives the laser head around the work area only — the galvo motion system is not used. This option is best for air-assisted cutting operations using Line Mode.
- Hybrid Motion synchronizes both the gantry and galvo motion systems to allow you to achieve higher speeds than are possible with a traditional gantry laser, without the size limitations of traditional galvo laser. This option is usually best for engraving and scoring.
- Galvo utilizes the mirrors only. The field size is limited to 40mm x 40mm. This option is fastest for engraving objects that fit entirely within the field size, allowing you to utilize the maximum speeds the mirrors are capable of.

Each option comes with additional customizations. See the sections below to learn more about a given Motion option.

Gantry



Use the **Gantry** option when you need to cut through materials at relatively slow speeds using **Air Assist**.

The gantry system's maximum speed is limited to 400 mm/second.

Click an option in the image below to learn more about specific Gantry options.

Airflow Nozzle

Enables use of the Airflow Nozzle — this is a nozzle specially designed for better cutting airflow. You must also enable Air Assist in standard settings to use this nozzle.

Over Line

Entering an **Over Line** value extends travel distance past the ends of scan lines. This function works similarly to **Overscanning** for standard gantry lasers. Learn more about Overscanning here.

This option is only available if your layer is set to Fill Mode — utilizing the gantry motion system only is an uncommon use case. In general, you should select **Hybrid Motion** or **Galvo** for Fill operations instead.

Laser Defocus

Adjusts the focal height of this layer by the amount entered in the field. Negative values move the focal point lower, into the material, while positive values move it higher, above the surface.

When cutting thicker materials, use a negative value to adjust focus into the material.

Hybrid Motion

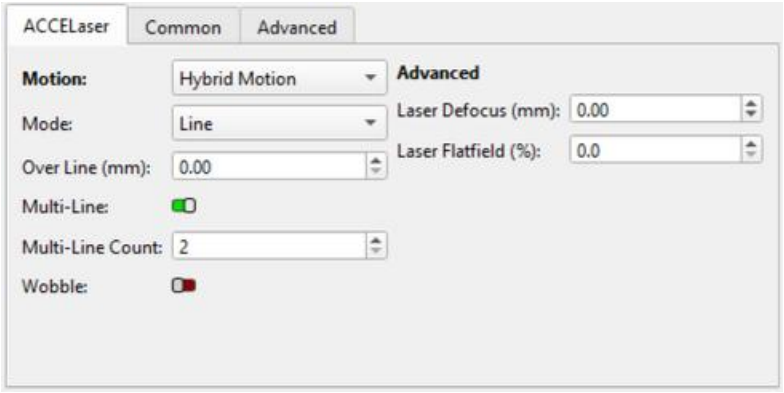
Hybrid Motion utilizes both the gantry and galvo motion systems simultaneously. This is the best general-use option for Fill Mode and Image Mode engravings.

The Line and Stitch options each increase processing times beyond what a standard gantry laser is capable of.

The mode dropdown menu lets you select from additional sub-options for your chosen **Motion** type.

Mode

Line



Speed is limited to 400 mm/sec, but you can increase effective speed by enabling Multi-Line.

For the best quality and stability in most Fill and Image jobs, use **Line** mode with Multi-Line disabled.

Multi-Line

Enable **Multi-Line** to have the laser process multiple scan lines at one time. This effectively increases the speed of the job by a multiple of the Multi-Line Count.

Multi-Line Count

This field controls the number of lines the laser will process at one time. **2** or **3** are common values that will increase speed significantly, though higher values may reduce quality on some materials. We recommend a maximum value of **3**.

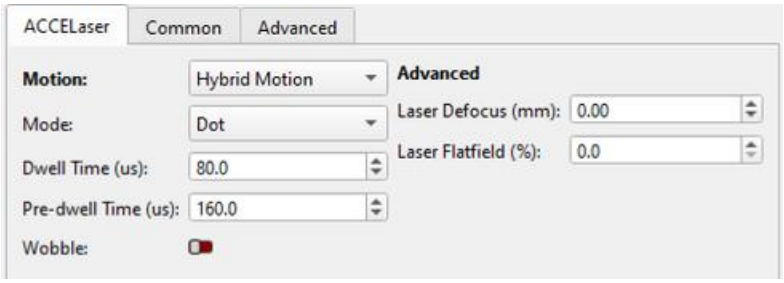
Multi Line effective speed limit is equal to 400 mm/sec (maximum gantry speed) times the number lines.

2 lines = 400 mm/sec x 2 = 800 mm/sec

3 lines = 400 mm/sec x 3 = 1200 mm/sec

> **Tip**
>
> Multi-Line is the recommended option for Fill and Image engraving on wood and plywood. A **Multi-Line Count** of **2** (800 mm/sec) offers a good balance of speed and quality. When using Multi-Line, Fill Shapes Individually is usually faster than **Fill All Shapes at Once**.

Dot



Dot mode fires the laser at individual points instead of along continuous lines. It offers the highest precision of any mode, and is ideal for fine detail image engraving on materials like black acrylic or painted aluminum.

The **Speed** setting has no effect in Dot mode. Instead, two dwell settings determine speed.

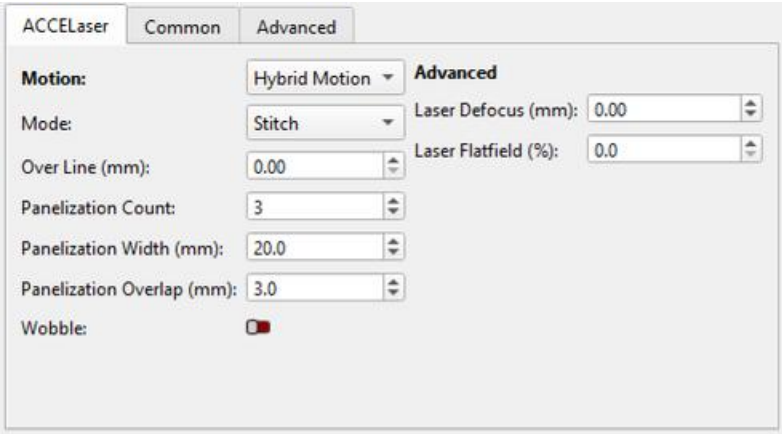
Dwell Time

How long the laser fires at each point, in microseconds. Start with a value of at least 40 µs. Higher **Dwell Times** lead to the laser outputting more power at each point it fires.

Pre-dwell Time

How long the laser waits at each point before firing, in microseconds. Start with a value of at least 160 µs. Longer **Pre-dwell Times** give the galvo mirrors more time to settle before firing, and can increase job quality.

Stitch



Your engraving is divided into multiple panels, each small enough to fit within the 40 x 40 mm galvo field. The galvo engraves each panel at full speed, then the gantry moves to the next panel.

Stitch is the fastest **Fill** mode, with speeds up to 4000 mm/sec, but small misalignments between panels can leave visible seams. It works best on materials where seams are hard to see, like cork and stone. On wood, seams are usually visible, so use Multi-Line instead.

Panelization Count

The number of overlapping passes used to blend panel edges. Values of 1–3 help compensate for stitching errors.

Panelization Width

The width of each panel. This must be less than 40 mm to fit within the galvo field.

Panelization Overlap

How much adjacent panels overlap. A value of around 3 mm helps compensate for stitching errors.

Over Line

Entering an **Over Line** value extends travel distance past the ends of scan lines. This function works similarly to **Overscanning** for standard gantry lasers. Learn more about Overscanning here.

Oscillation

Enable **Oscillation** to have the laser move rapidly back and forth along the path as it travels, retracing each short segment of the line several times before moving on. This concentrates more energy along the path without slowing the beam itself, which is useful for cutting thin metals.

- **Oscillation Count**: The number of times the laser passes over each segment before advancing.
- **Oscillation Length**: The length of each segment the laser moves back and forth over.

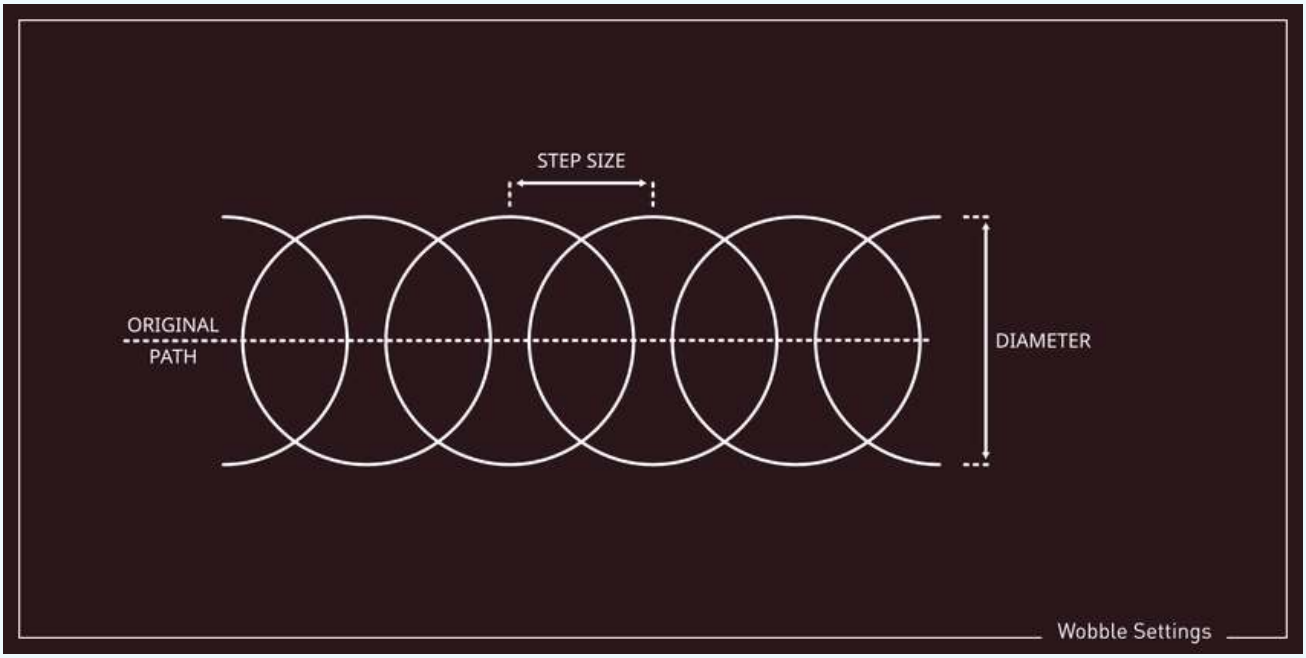
> **Tip**
>
> **Oscillation** and **Wobble** are useful techniques for cutting through metal.

Wobble

Enable **Wobble** to tell the laser to trace small circles along the path of each line, rather than following the line directly. This creates a thicker line and forces the beam to spend longer in each section of the line. Wobble is especially useful for black marking and color marking on metal with the infrared laser.

- **Step**: How far apart the circles are.
- **Size**: The diameter of the circles.

Wobble Diagram



Laser Defocus

Adjusts the focal height of this layer by the amount entered in the field. Negative values move the focal point lower, into the material, while positive values move it higher, above the surface.

Laser Flatfield

Laser Flatfield keeps the laser's focal point consistent across the entire galvo field when engraving flat material.

Without compensation, the angle at which the galvo mirrors direct the beam can cause loss of focus on flat surfaces at the edges of the galvo's working field. HD1 series lasers use a dynamic focus module to compensate, continuously adjusting focus as the beam moves so the focal point stays on a flat plane.

Enter a percentage to control the amount of correction applied. At **0%**, no flatfield correction is applied.

Galvo

Galvo is limited to a 40x40 mm working area, but allows the galvo mirrors to move at their full speed of up to 4000 mm/sec. Use it for any job that fits entirely within this area, like small metal tags or jewelry.

If your graphics are larger than 40 x 40 mm, the laser will engrave them in sections, using the gantry to move between each one. This may leave visible seams on large objects — use **Hybrid Motion** instead.

Line

Processes graphics with continuous lines, the same as standard Fill and Line operations.

Dot

Fires the laser at individual points instead of along continuous lines, for the finest detail. See **Dot** under Hybrid Motion for more information.

Oscillation

Enable **Oscillation** to have the laser move back and forth along the path as it travels, retracing each short segment of the line several times before moving on. This concentrates more energy along the path without slowing the beam itself.

- **Oscillation Count**: The number of times the laser passes over each segment before advancing.
- **Oscillation Length**: The length of each segment the laser moves back and forth over.

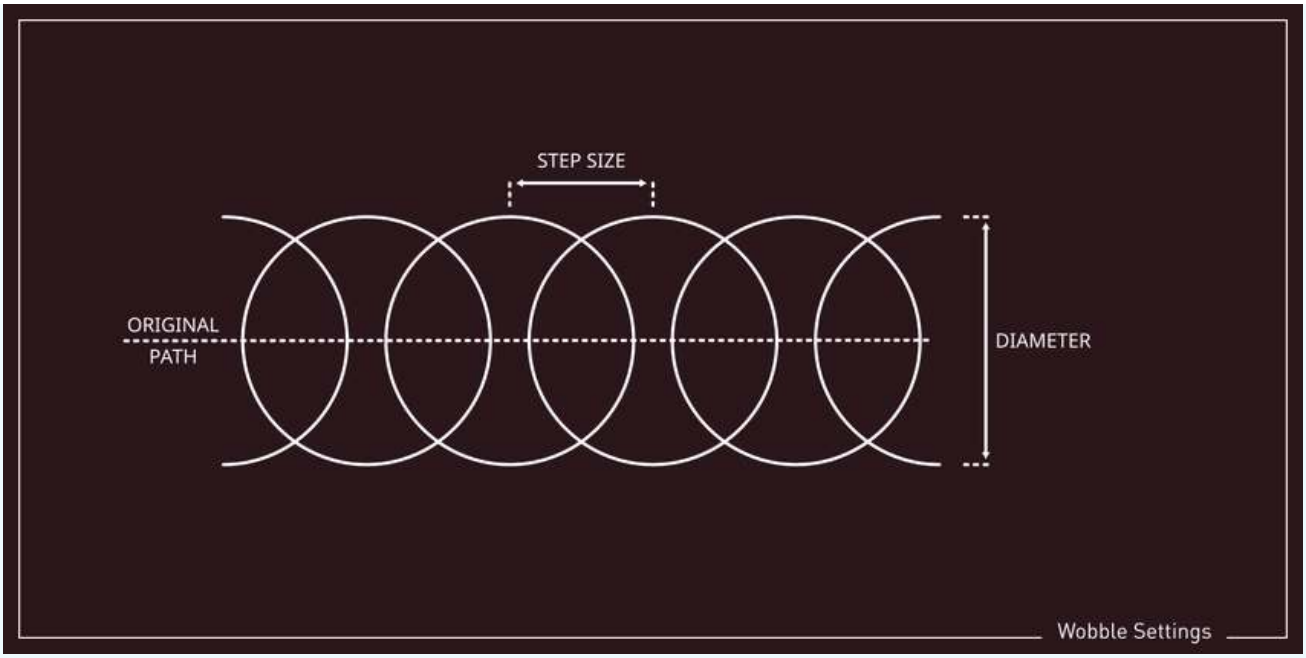
> **Tip**
>
> **Oscillation** and **Wobble** are useful techniques for cutting through metal.

Wobble

Enable **Wobble** to tell the laser to trace small circles along the path of each line, rather than following the line directly. This effectively creates a thicker line and causes the beam to spend longer and output more energy in each section of the line.

- **Step**: How far apart the circle centers are.
- **Size**: The diameter of the circles.

> **Wobble Diagram**



Laser Defocus

Adjusts the focal height of this layer by the amount entered in the field. Negative values move the focal point lower, into the material, while positive values move it higher, above the surface.

Laser Flatfield

Laser Flatfield keeps the laser's focal point consistent across the entire galvo working field when engraving flat material.

Enter a percentage to control the amount of correction applied. At **0%**, no flatfield correction is applied.

Without compensation, the angle at which the galvo mirrors direct the beam can cause loss of focus on flat surfaces at the edges of the galvo's working field. HD1 series lasers use a dynamic focus module to compensate, continuously adjusting focus as the beam moves so the focal point stays on a flat plane.

EZCad3

> **Labs Feature**
>
> Labs features are new tools in development that we couldn't wait to get into the hands of our users. Please experiment with caution, and report any unexpected results on our forum.

LightBurn 2.2 includes support for EZCad3 lasers.

Adding an EZCad3 Laser to LightBurn

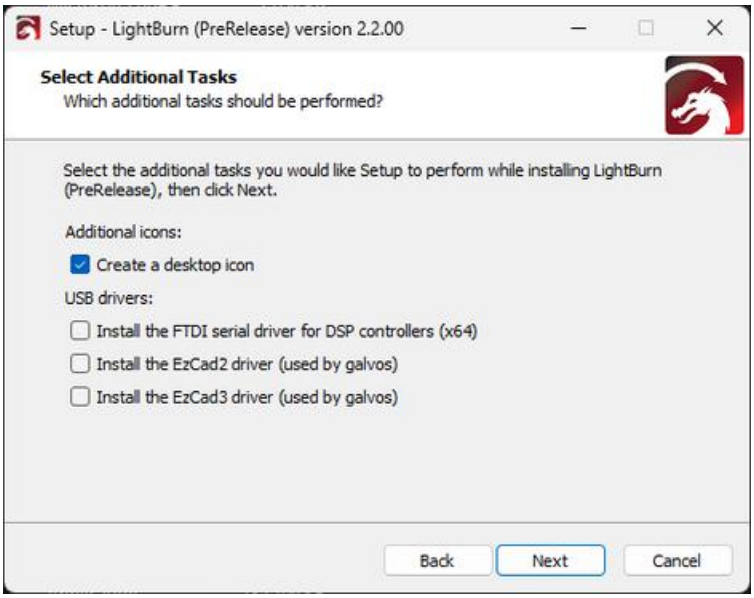
Directions for installing EZCad3 lasers are in our guide to Adding a Galvo Laser to LightBurn.

The installation process is similar to other galvo lasers. The guide linked above includes information on where to find your configuration files and the necessary drivers.

> **EZCad3 and non-fiber laser sources**
>
> EZCad3 support has been tested on lasers with fiber sources. If you have an EZCad3 laser using UV or another source, please use the latest version of LightBurn 2.2 and reach out to support if you see unexpected behaviors

EZCad3 LightBurn Drivers

LightBurn 2.2 has a checkbox to install the EZCad3 driver as part of the installation process. If you need to install the EZCad3 driver later, you can re-run the LightBurn installer and check the driver installation box - you won't lose any settings.

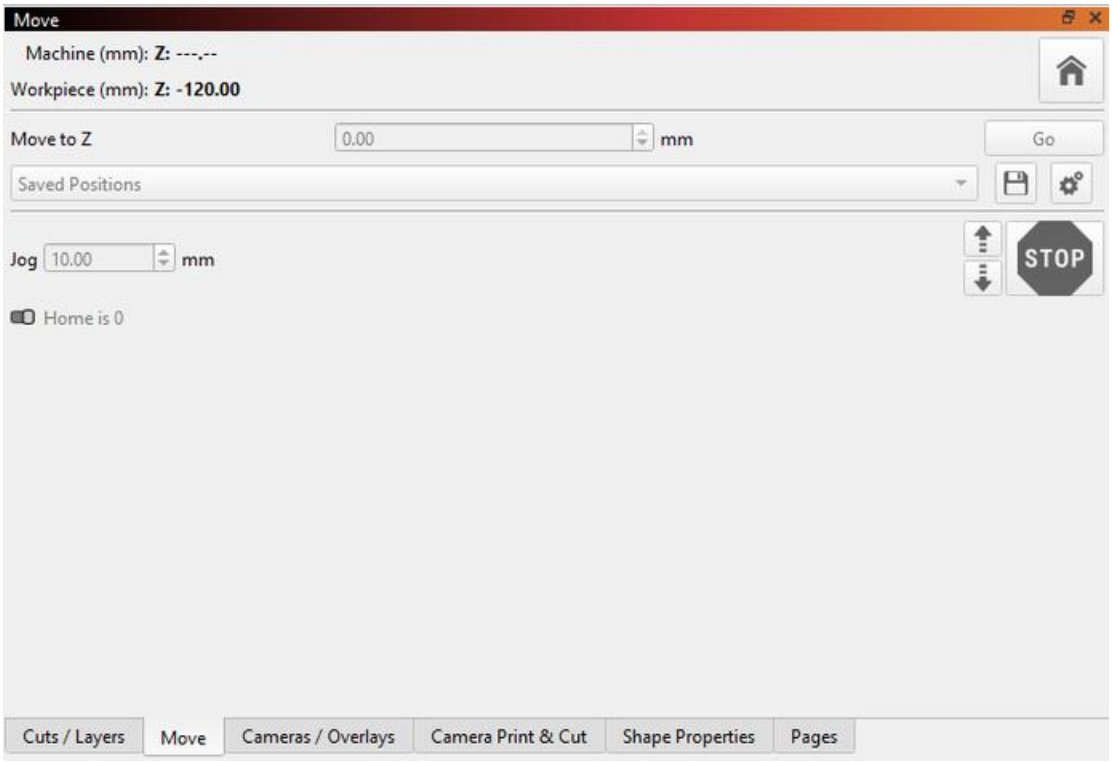


LightBurn UI/Behavior Changes for EZCad3 Devices

- New Move Window for EZCad3 ↓

- External Axes Settings tab in Device Settings ↓
- Additional EZCad3 Settings tab in Device Settings ↓

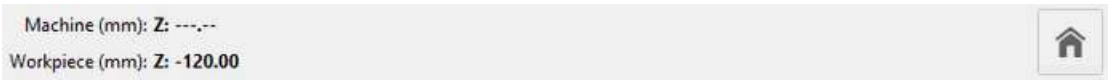
New EZCad 3 Move Window



After adding an Adding an EZCad3 device with a motorized Z axis to LightBurn, you'll have an alternate version of the **Move Window** available for controlling the Z axis.

Enabling the Move Window for EZCad3

You need to set the **Z Axis Height** (the distance the Z axis can travel) to be able to open the EZCad3 Move Window. This can be done when you're adding your laser or in the External Axes Settings. If you don't see the Move Window, make sure the Z Axis Height is set, and then go to Window → **Move Window**.



Position Display

When your laser is connected, the **Machine: Z** and **Workpiece: Z** sections of the Move window will show the current position of the Z axis. The **Machine** section shows the position relative to the machine itself, and the **Workpiece** section shows the position relative to your material.

Home Z Axis

The **Home Z Axis** button will home the Z axis using a limit switch to find the origin, similar to homing the X and Y axes on a gantry laser.



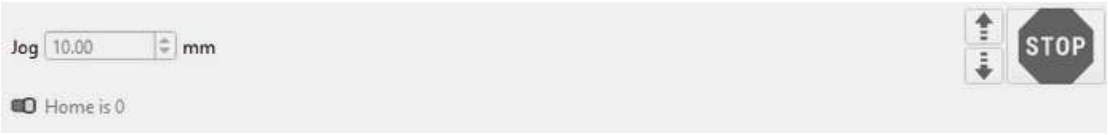
Move to Z

Enter a number into the **Move to Z** section and click **Go** to command your laser to move the Z axis to the position you've set. You can use this to go to an exact height without needing to jog precisely.

Saved Positions

Use this section of the window to choose, save, and manage **Saved Positions**.

- Select a saved position from the dropdown and click **Go** to move the Z axis to the selected position.
- Click the **Save** icon to save the curent position into the list of **Saved Positions**.
- Click the gear icon to **Manage Saved Positions**.



Jog Distance

The **Jog Distance** is the distance your Z axis will move when you use the Jog Controls. Set it with the numeric entry box on the left. This allows you to move large distances when needed, or reduce the distance when you need to jog very precisely and get a position exactly right.

Jog Controls

Use the Up, Down, and Stop buttons on the right to jog your laser. The up and down arrow buttons will jog your laser up and down by the distance shown in jog distance, and the stop button will stop the jog if you need to interrupt it.

External Axes in Device Settings

The **External Axes** tab in Device Settings is available for EZCad3 lasers.

Use this tab to enable and configure external axes, allowing control of a Z axis for raising and lowering the laser head as well as control of the X and/or Y axis to enable moving the workpiece you're engraving on.

See the Device Settings - External Axes page for more information.

Additional EZCad3 Settings in Device Settings

The **Additional EZCad3 Settings** tab in Device Settings contains additional configuration settings specific to EZCad3 lasers. These are generally set when you import a config file, and should generally not be modified by users unless support or your manufacturer tell you to do so.

See the Device Settings - Additional EZCad3 Settings page for more information.

Epilog

> **Labs Feature**
>
> Labs features are new tools in development that we couldn’t wait to get into the hands of our users. Please experiment with caution, and report any unexpected results on our forum.

With the release of LightBurn 2.2, LightBurn now works directly with certain Epilog lasers.

See the list of compatible Epilog lasers ↓

How to add your Epilog laser to LightBurn ↓

UI changes for Epilog lasers ↓

Adding an Epilog Device to LightBurn

> **Info**
>
> Epilog's built-in cameras are not yet supported in LightBurn.

All supported Epilog lasers communicate with LightBurn over a network connection. You can either create your device manually or use LightBurn's new network option for Find My Laser. See below for detailed directions.

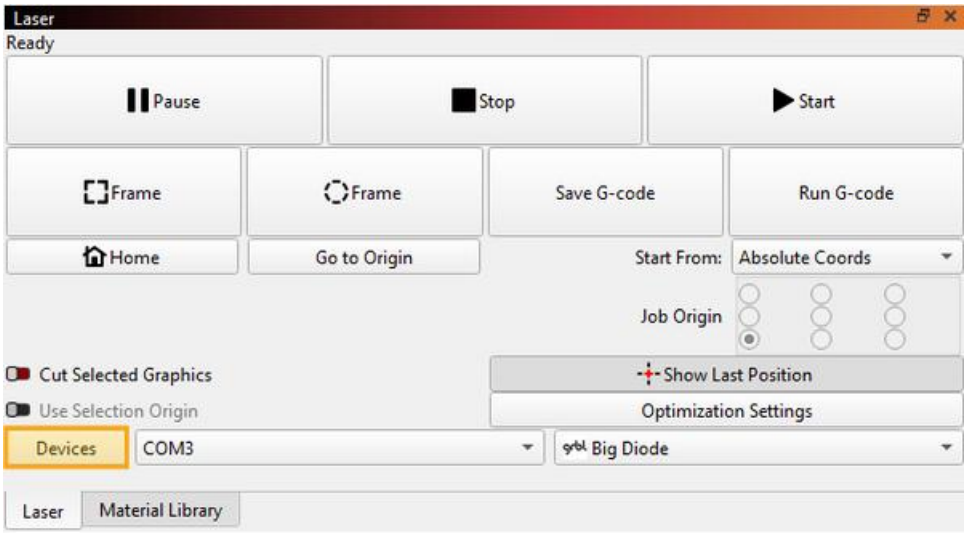
Before you begin, make sure you:

- Are using LightBurn Pro (all trials are LightBurn Pro)
- Have connected your Epilog laser to the network

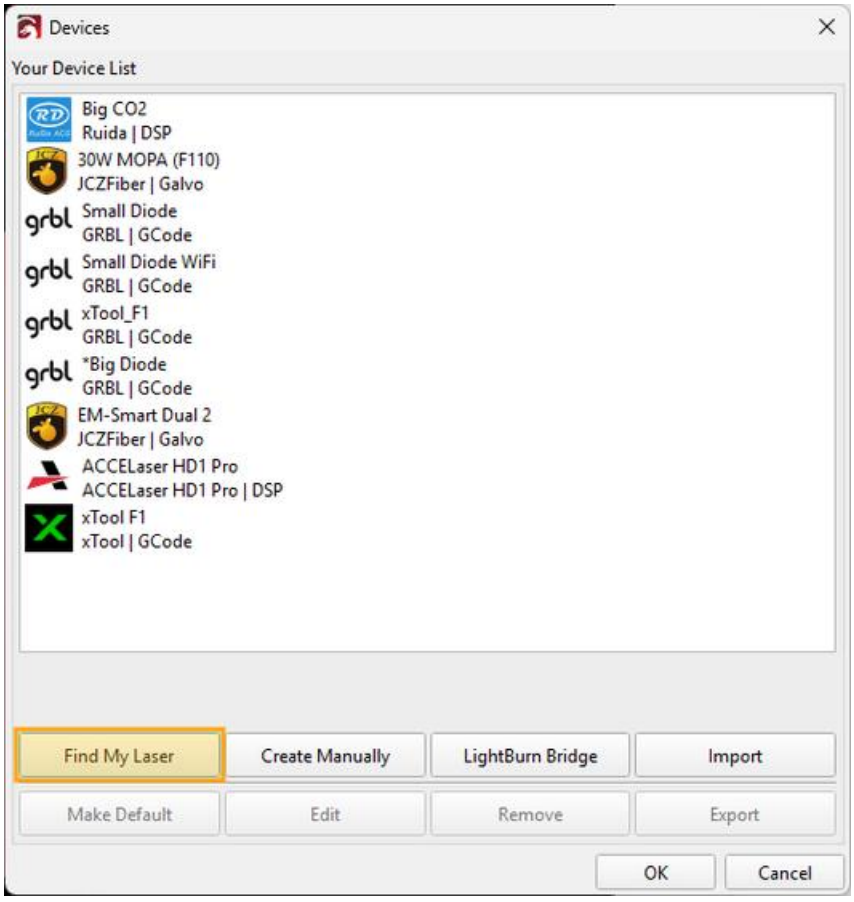
If you're creating your laser manually, you'll need your laser's IP address. You can get this from the on-device menus.

Using Find My Laser with an Epilog Laser

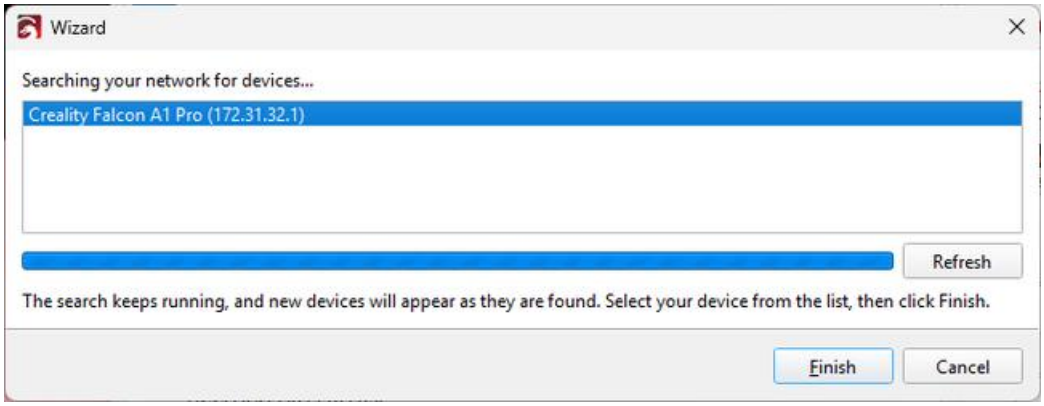
1. In LightBurn, click the **Devices** button (highlighted in the screenshow below) in the Laser window to open the Devices window.



2. In the **Devices** window, click the **Find My Laser** button under the device list to open Find My Laser.



3. Make sure your device is connected to the network. Click the **Search Network** button.
4. If LightBurn detects your device, it will show up in the list. Select it and click **Finish**.



Completing Setup

Find My Laser Worked

Great! You can jump ahead to naming your device and continue with the rest of setup.

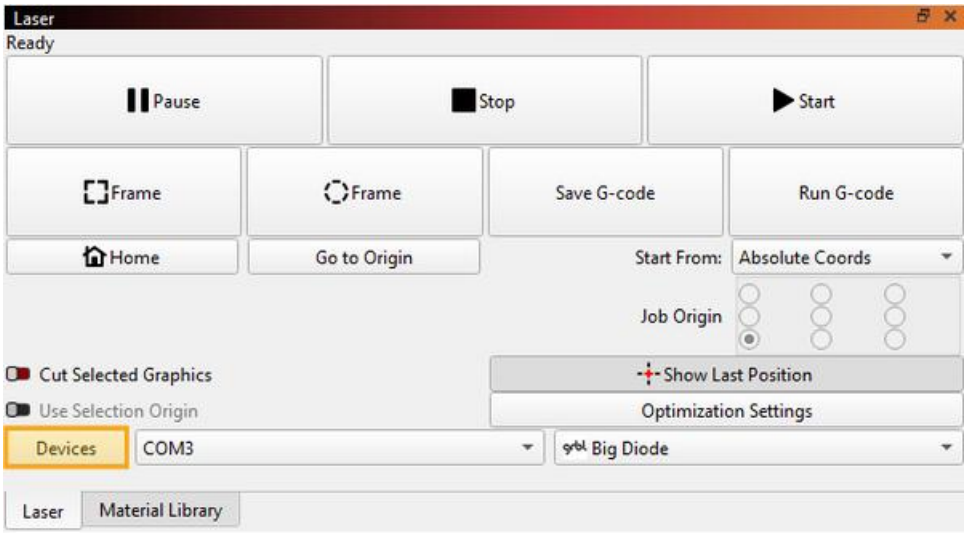
Find My Laser Didn't Work

If **Find My Laser** didn't work for you, you'll need to complete the setup manually. Continue on to Create Your Laser Manually.

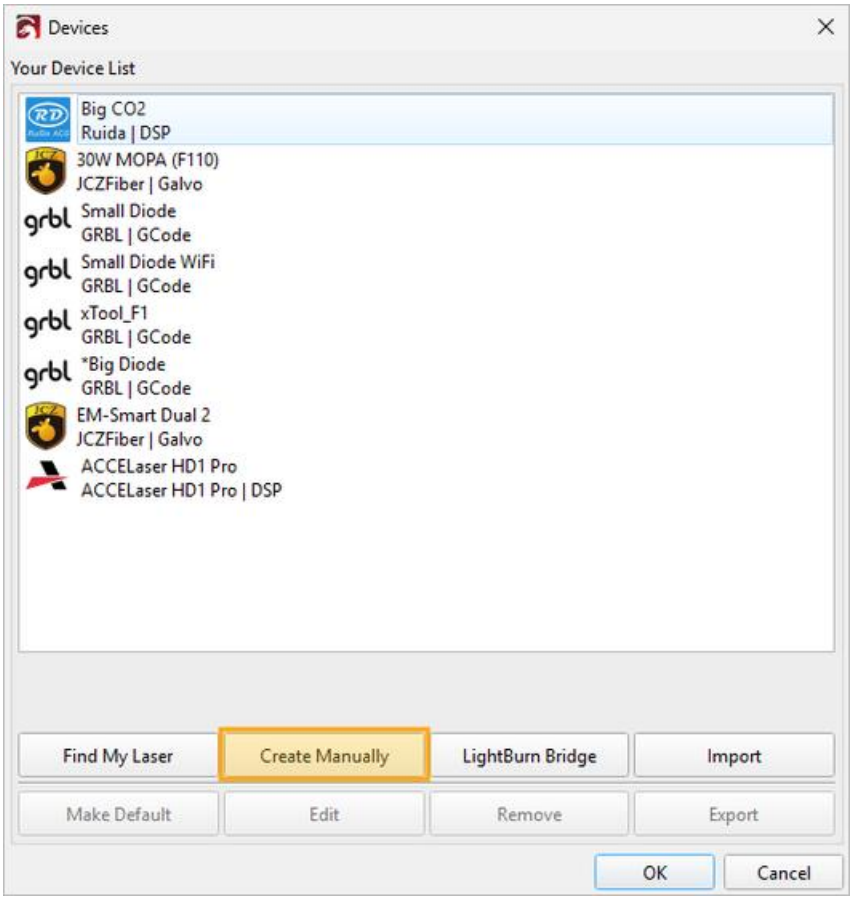
Adding Your Epilog Laser Manually

Before you begin, find your laser's IP address using the menus on the laser's controller.

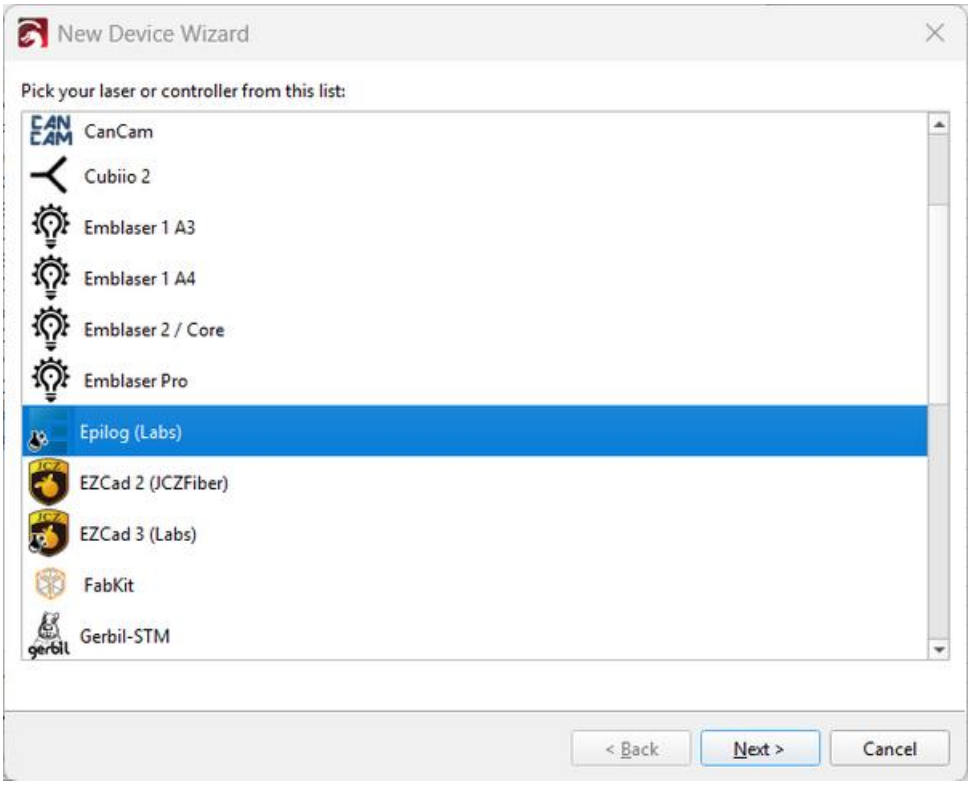
1. In LightBurn, click the **Devices** button (highlighted in the screenshot below) in the Laser window to open the Devices window.



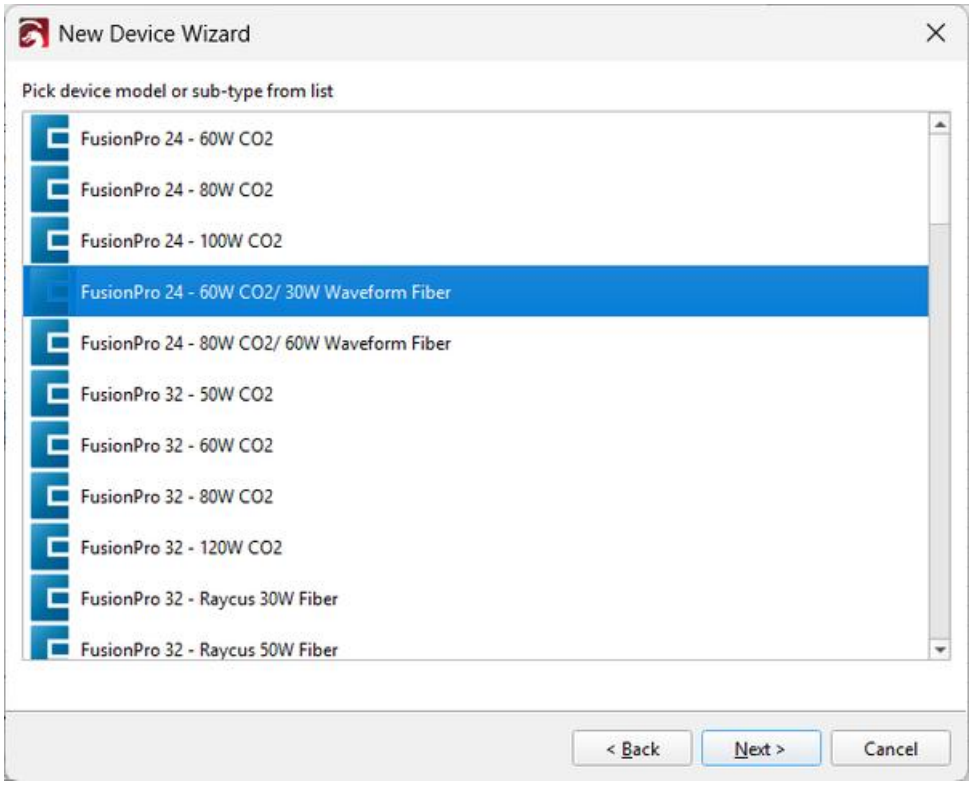
2. In the **Devices** window, click the **Create Manually** button under the device list to open the **New Device Wizard**.



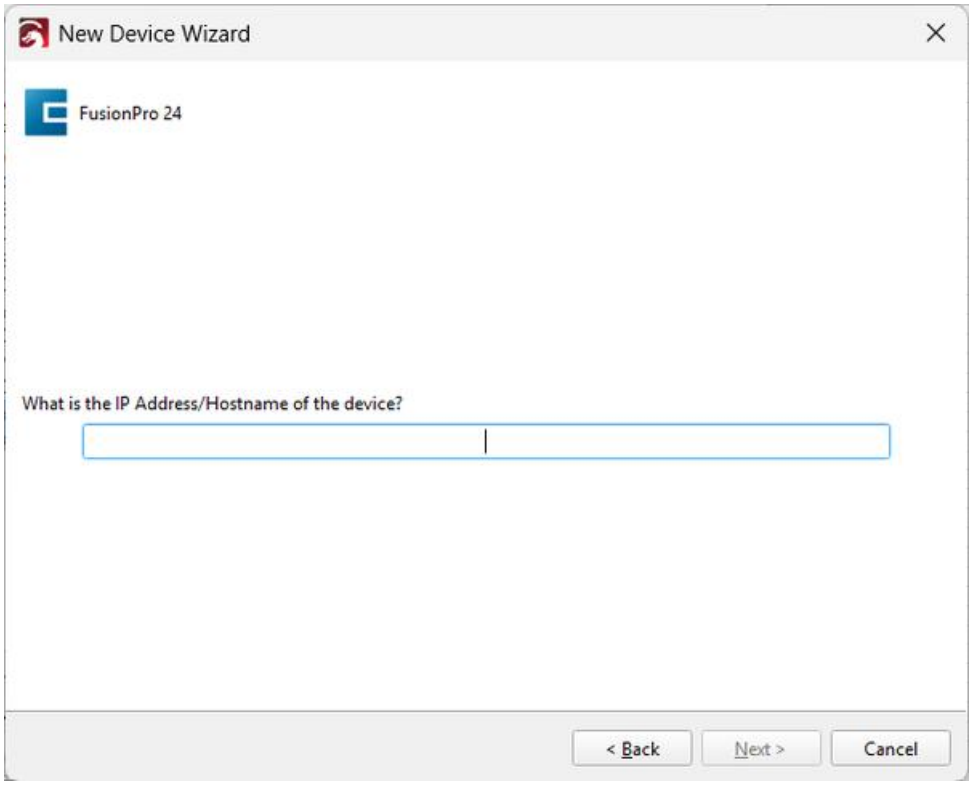
3. In the **New Device Wizard**, select **Epilog** from the list of device types. Click Next to Continue.



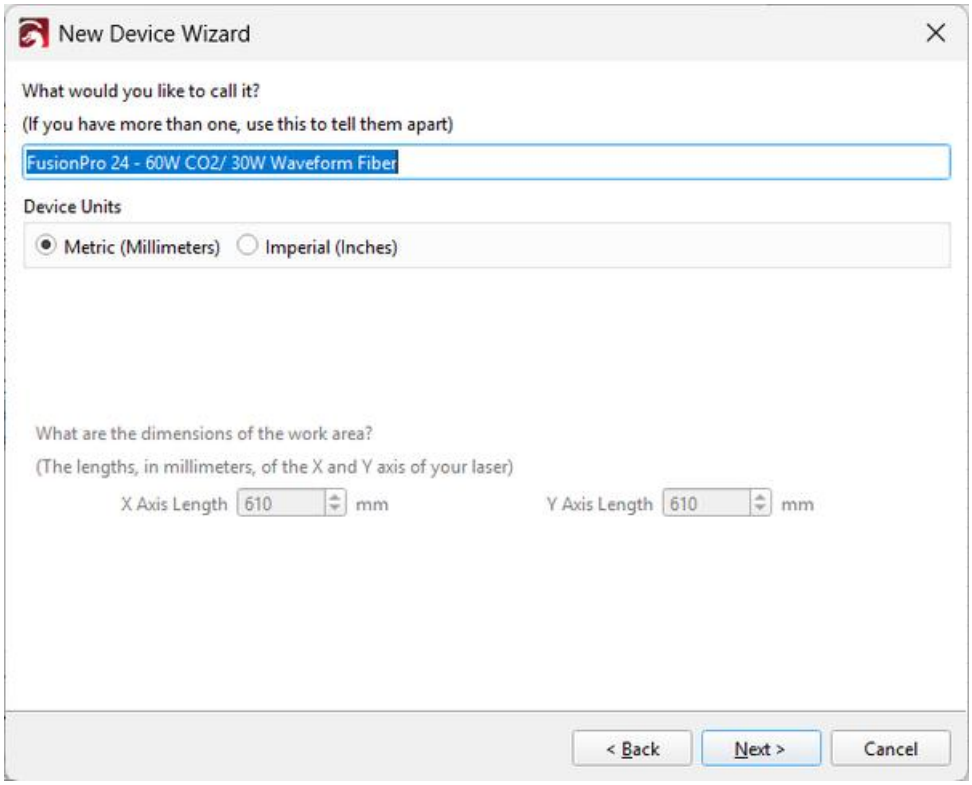
4. Select your specific Epilog model from the list.



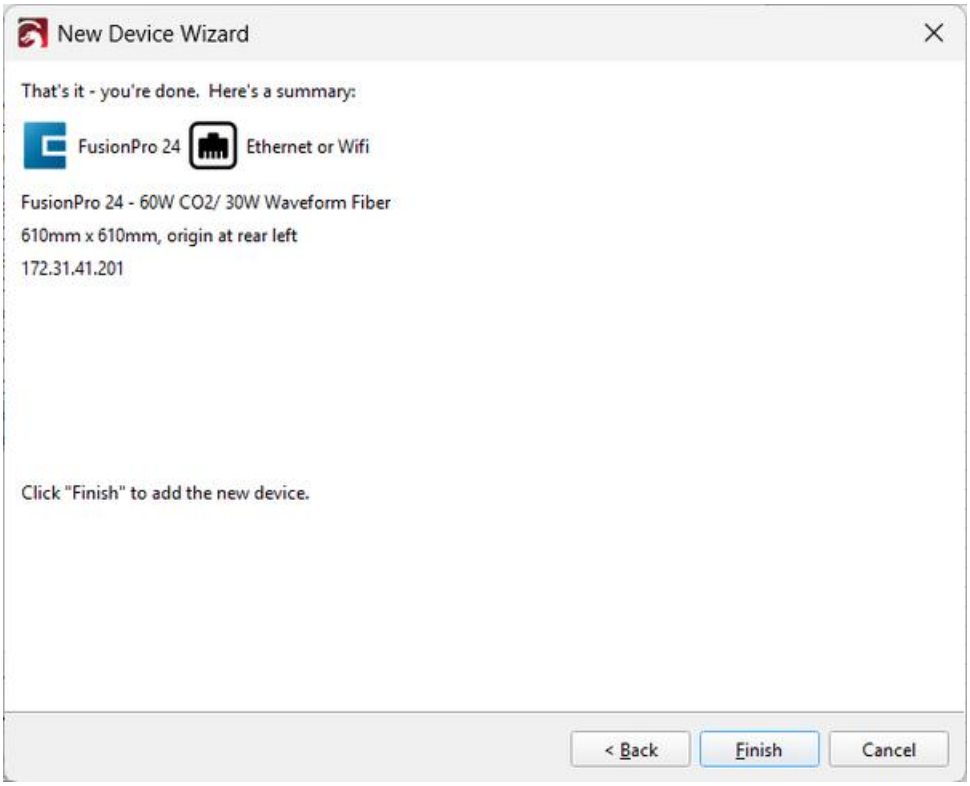
5. Enter your IP address to tell LightBurn where to find your laser on your network. Click Next to continue.



6. Name your device something recognizable. The dimensions should be set automatically. Leave the **Device Units** in millimeters - if you prefer to design in inches, you can set that separately later. Click Next to continue.



7. Make sure the summary on the final screen seems correct. Click **Back** to go back and make changes, or **Finish** to save your changes.



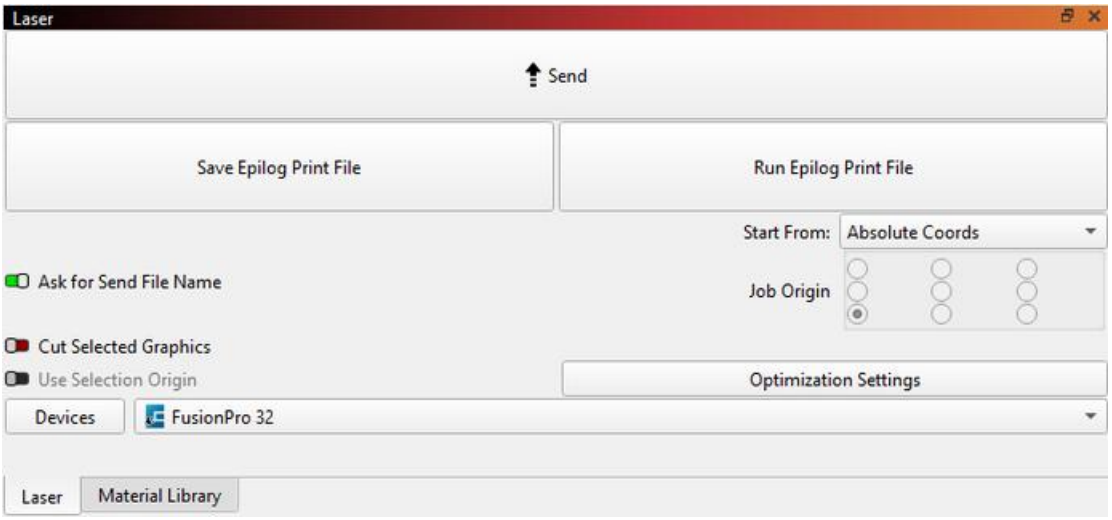
LightBurn UI/Behavior Changes for Epilog

When using an Epilog device, you'll see the following changes:

- Custom Laser Window for Epilog Devices
- Epilog Settings tab in Device Settings
- Save Epilog Print File option in File Menu

Custom Laser Window for Epilog Devices

When you use LightBurn with an Epilog laser, you'll see a custom version of the Laser Window.



Send

Click the **Send** button to send the current design to your laser. If Ask for Send File Name is enabled, LightBurn will ask you for a file name.

Save Epilog Print File

Saves the cut data for the current design to an Epilog Print File (`.prn`) that you can upload to your laser later.

Run Epilog Print File

Loads an Epilog Print File and runs it on the laser immediately.

Ask for Send File Name

When **Ask for Send File Name** is enabled, LightBurn will ask you to name the file that is sent to the laser before sending it.

Start From and Job Origin

These two controls determine where the job is placed on the bed of your laser.

See Coordinates and Job Origin for more information.

Cut Selected Graphics

This toggle switch tells LightBurn to only send the portion of your design that is currently selected when you click **Start**, **Send**, or **Save Machine File**.

See Cut Selected Graphics for more information.

Use Selection Origin

Only available in combination with **Cut Selected Graphics**, the **Use Selection Origin** button tells LightBurn to calculate origin relative to selected graphics only, and not the whole design.

See Cut Selected Graphics – Use Selection Origin for more information.

Optimization Settings

This button opens the **Optimization Settings** window, where you can adjust various options that control the order in which your graphics are cut or engraved by the laser.

See Optimization Settings for more information.

Devices

Opens the **Devices** window, where you can create, remove, or edit device profiles for lasers you want to use with LightBurn.

> **Right-click to reset the USB connection**
>
> Right-clicking the Devices button closes and re-opens the connection to the currently selected device to reset it. This currently does not work with BSL devices.

See Devices for more information.

Device Selection Dropdown

If you've created multiple device profiles for multiple lasers, select the laser you are currently working with the dropdown menu.



Epilog Settings Tab in Device Settings

When an Epilog laser is selected, you'll have an extra tab in the Device Settings window for **Epilog Settings**.

Firmware Version

Use the **Firmware Version** dropdown to select which firmware version your Epilog laser is using. This information can be found from your laser's on-device control panel menus.

Save Epilog Print File

Go to File → **Save Epilog Print File** to create a print file with a `.prn` extension to send to your Epilog laser manually.

Epilog Laser Compatibility

Epilog Laser support in LightBurn as of October 16, 2026

Supported in LightBurn

- FusionPro 24
- FusionPro 32
- FusionPro 36
- FusionPro 48
- Fusion Ascent 12
- Fusion Ascent 24
- Fusion Ascent 36
- Fusion Edge 12
- Fusion Edge 24
- Fusion Edge 36
- Fusion Maker 12
- Fusion Maker 24
- Fusion Maker 36

Not Yet Supported in LightBurn

- Fusion Galvo G100 4″x4″
- Fusion Galvo G100 6″x6″
- G2 Galvo
- Fusion 32
- Fusion 40
- Fusion M2 32
- Fusion M2 40
- FiberMark Fusion 32
- FiberMark 24
- FiberMark 24 S2
- Zing 16
- Zing 24
- Helix 24
- Mini 18
- Mini 24

- Legend EXT 36

xTool

> **Labs Feature**
>
> Labs features are new tools in development that we couldn’t wait to get into the hands of our users. Please experiment with caution, and report any unexpected results on our forum.

LightBurn 2.2 works with more xTool devices than previous LightBurn versions, with better support for features like dual laser sources. This page provides additional information about which machines and features are supported, and how to set up your laser in LightBurn.

> **Info**
>
> xTool's built-in cameras are not yet supported in LightBurn.

See the list of compatible xTool lasers ↓

How to add your xTool laser to LightBurn ↓

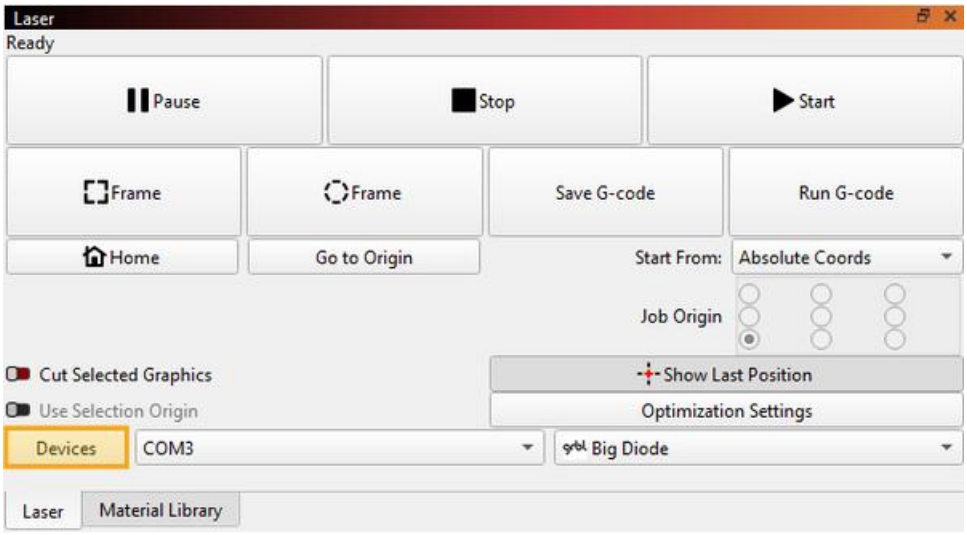
Adding an xTool Device to LightBurn

Setting up an xTool device in LightBurn has some slight differences from other devices.

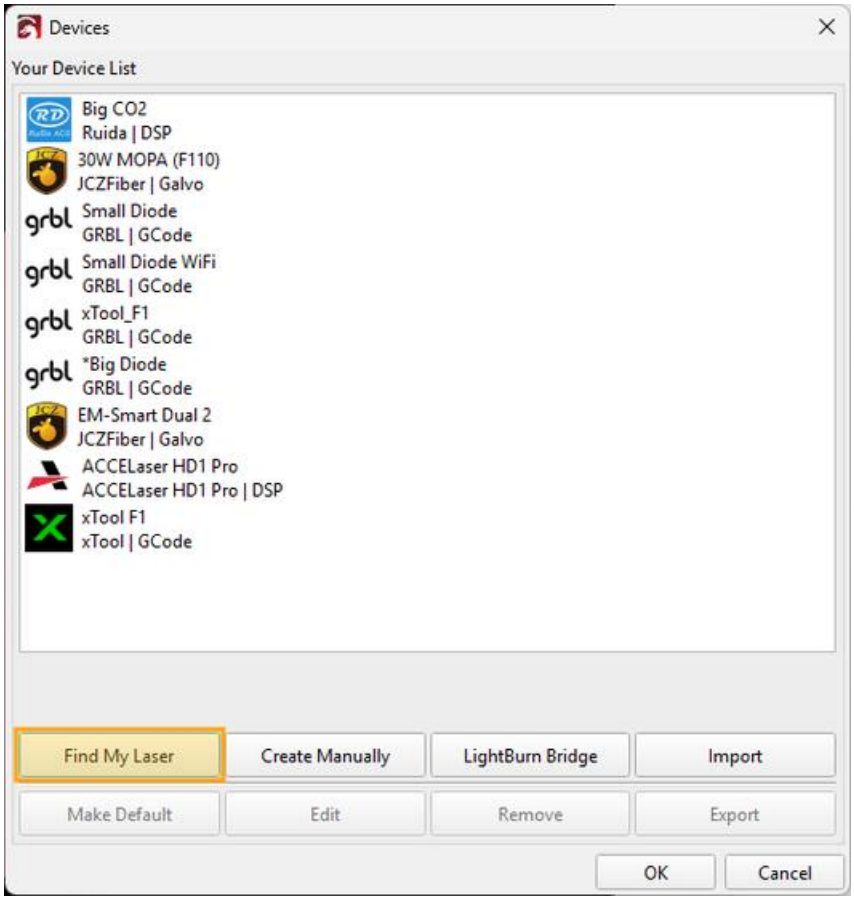
- Aside from the xTool S1 and xTool D1/D1 Pro, all xTool devices use network connections even if you're connecting with a USB cable. In LightBurn 2.2 RC 6 and newer, Find My Laser has the ability to detect lasers over the network, and we also have directions below to find your IP address. If you're adding your machine manually, LightBurn needs this IP address to know where to find your laser on your network.
- After setting up your xTool device, you'll need to check that it's configured correctly in Device Settings.

Using Find My Laser with xTool Devices

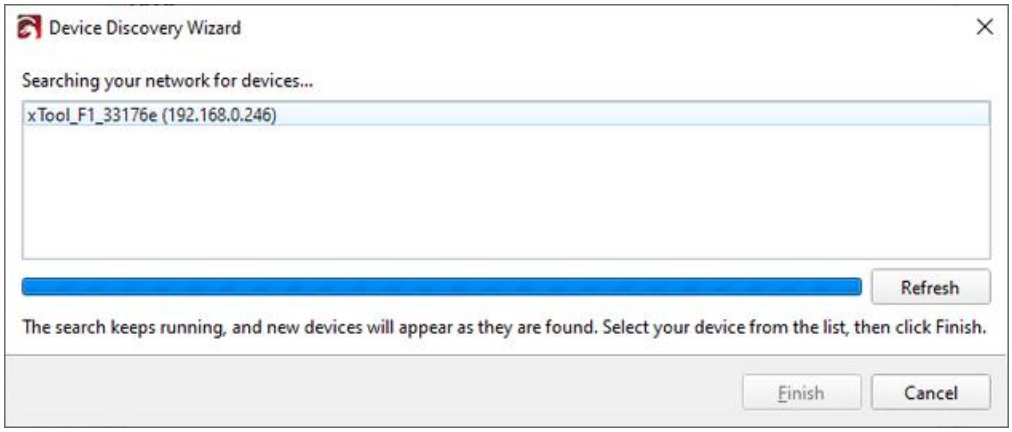
1. In LightBurn, click the **Devices** button (highlighted in the screenshow below) in the Laser window to open the Devices window.



2. In the **Devices** window, click the **Find My Laser** button under the device list to open Find My Laser.



3. Make sure your device is connected to the network or directly to the computer with a USB cable. For the xTool S1 or xTool D1/D1 Pro, click the **Find My Laser** button. For all other xTool devices, click the **Search Network** button.
4. If LightBurn detects your device, it will show up in the list. Select it and click **Finish**.



Find My Laser Worked

Great! You can jump ahead to naming your device and continue with the rest of setup.

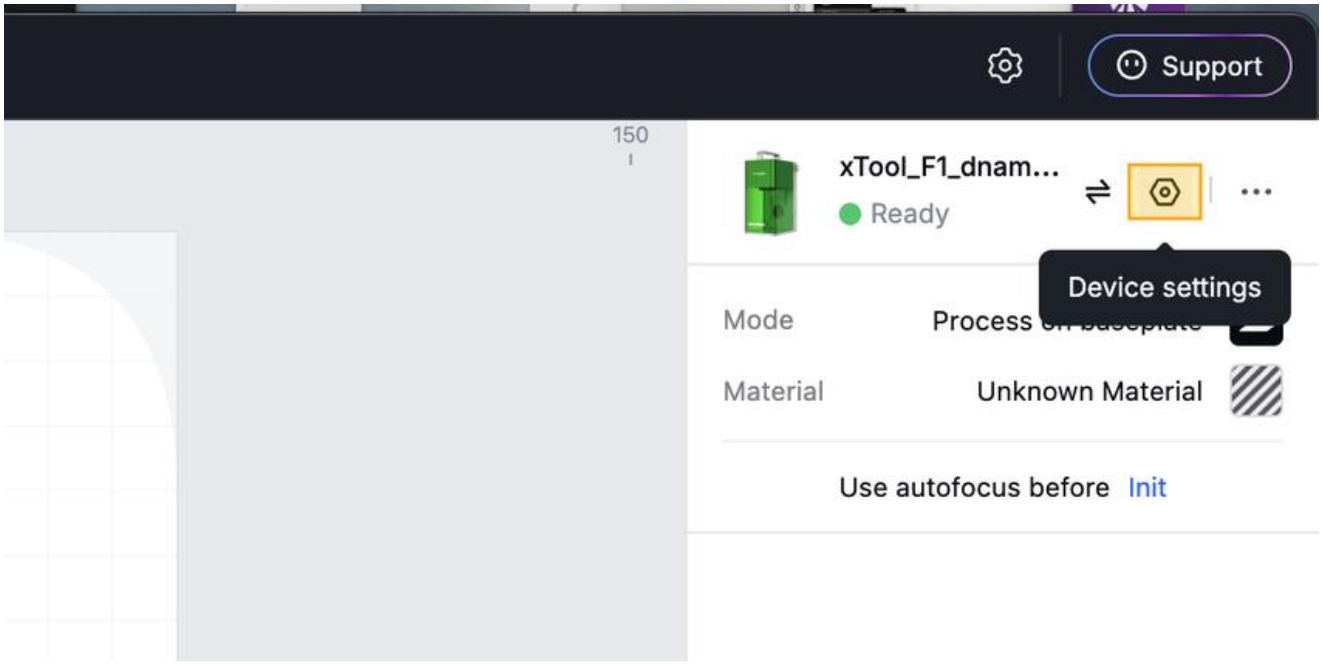
Find My Laser Didn't Work

If **Find My Laser** didn't work for you, you'll need to complete the setup manually. Continue on to Finding Your IP Address

Adding a Laser Manually

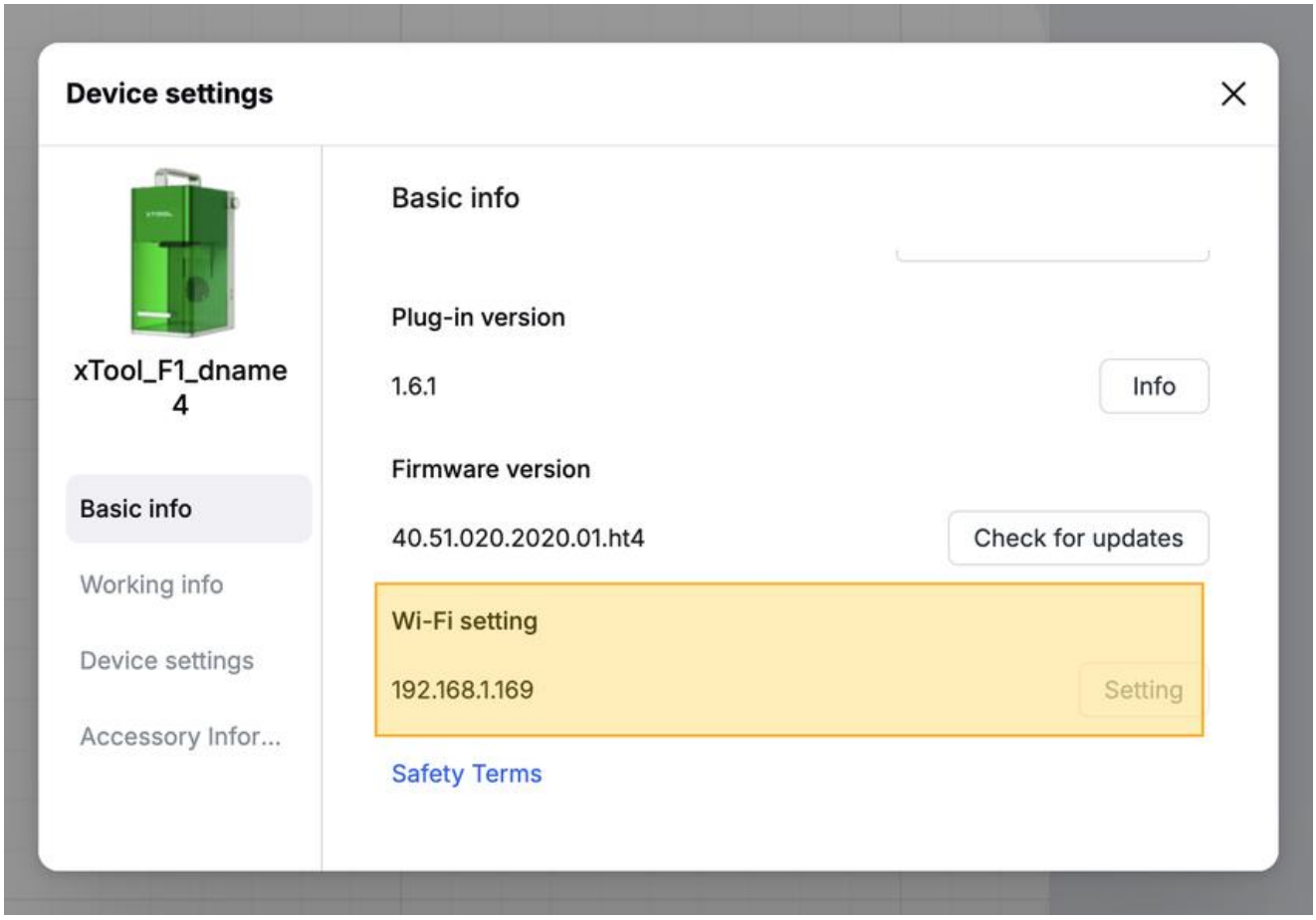
Part 1: Use xTool Studio to Find Your IP Address

1. Connect to your laser in xTool Studio. If this is your first time, you'll need to connect over USB. If you've connected before, you'll be able to connect however you usually do.
2. Open the **Device Settings** panel in xTool Studio by clicking on the nut icon highlighted in the screenshot below.



3. In the **Basic Info** section, scroll down to find the **Wi-Fi Setting** section. You should see an IP address there. In the screenshot below, you can see the IP address of 192.168.1.169 highlighted.

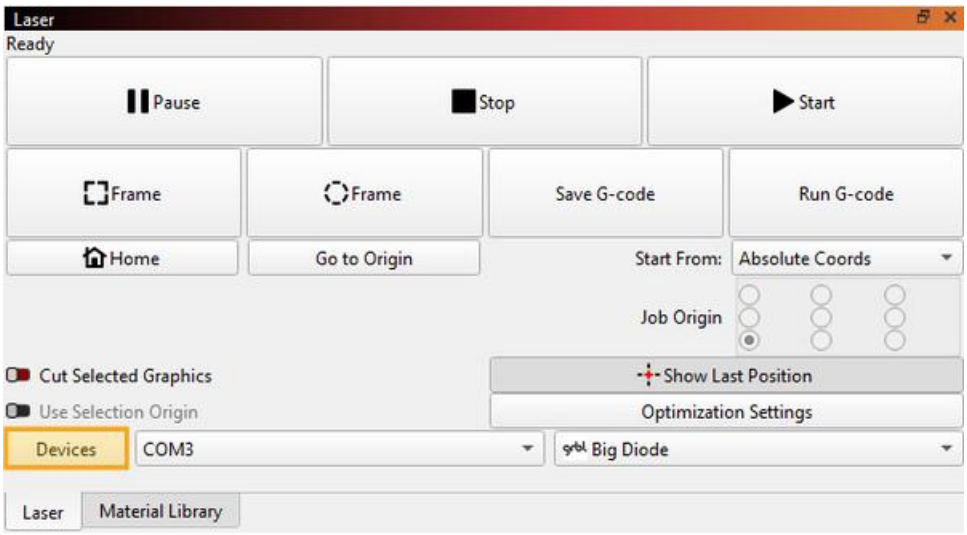
Copy the IP address or write it down.



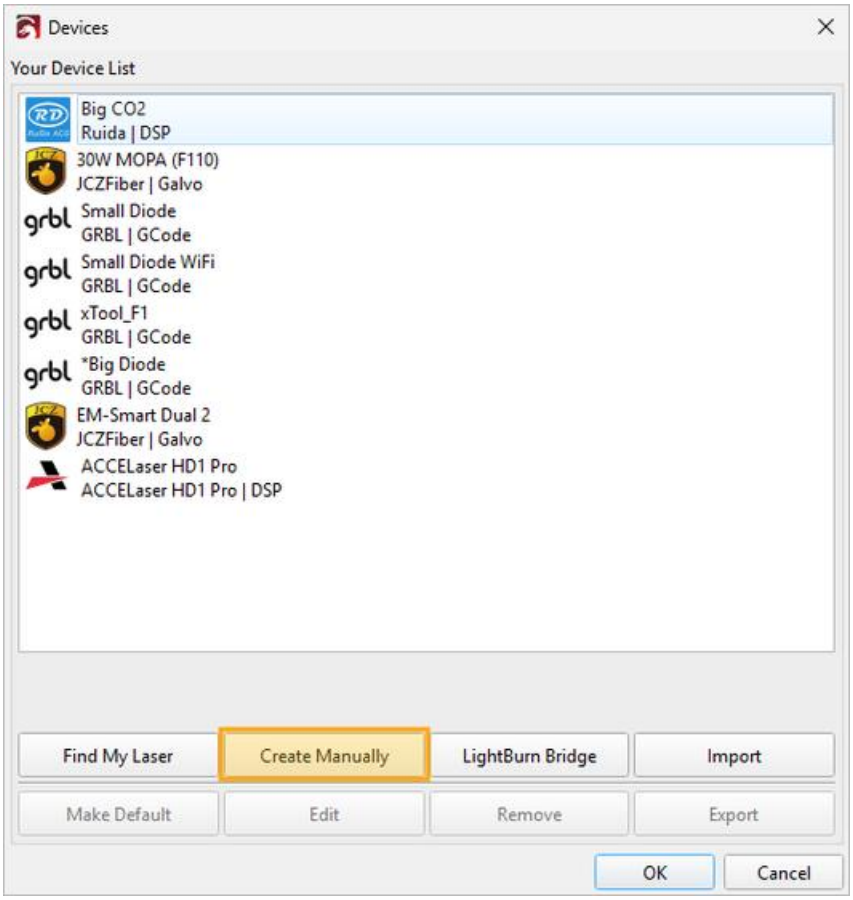
4. Head over to LightBurn to add your device.

Part 2: Add your xTool Device in LightBurn.

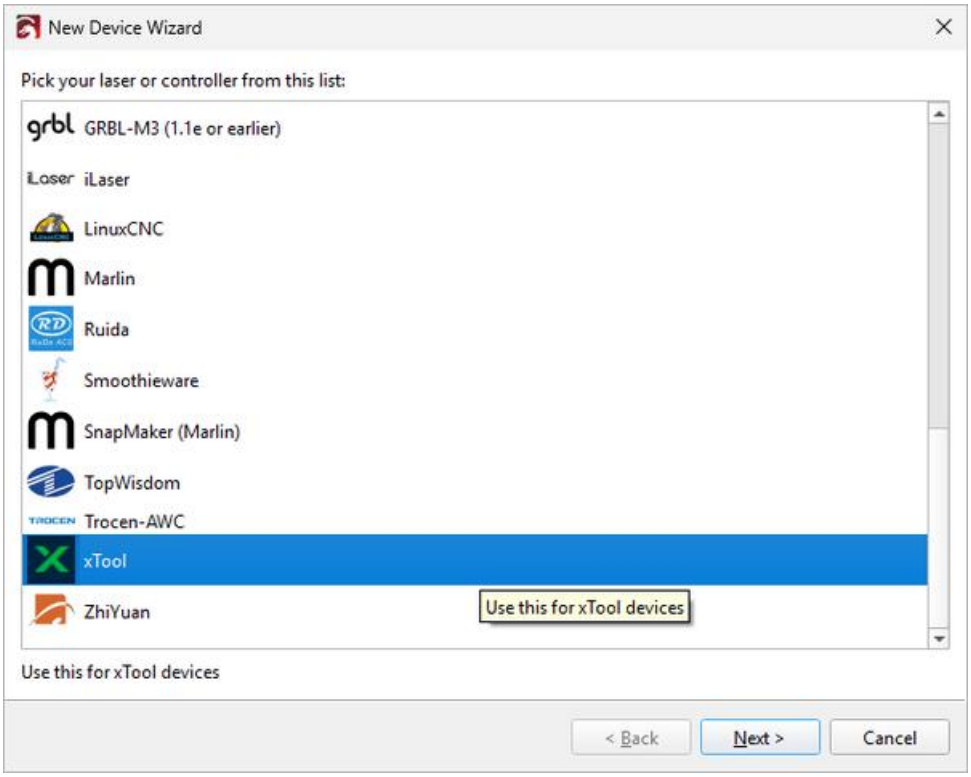
1. In LightBurn, click the **Devices** button (highlighted in the screenshot below) in the Laser window to open the Devices window.



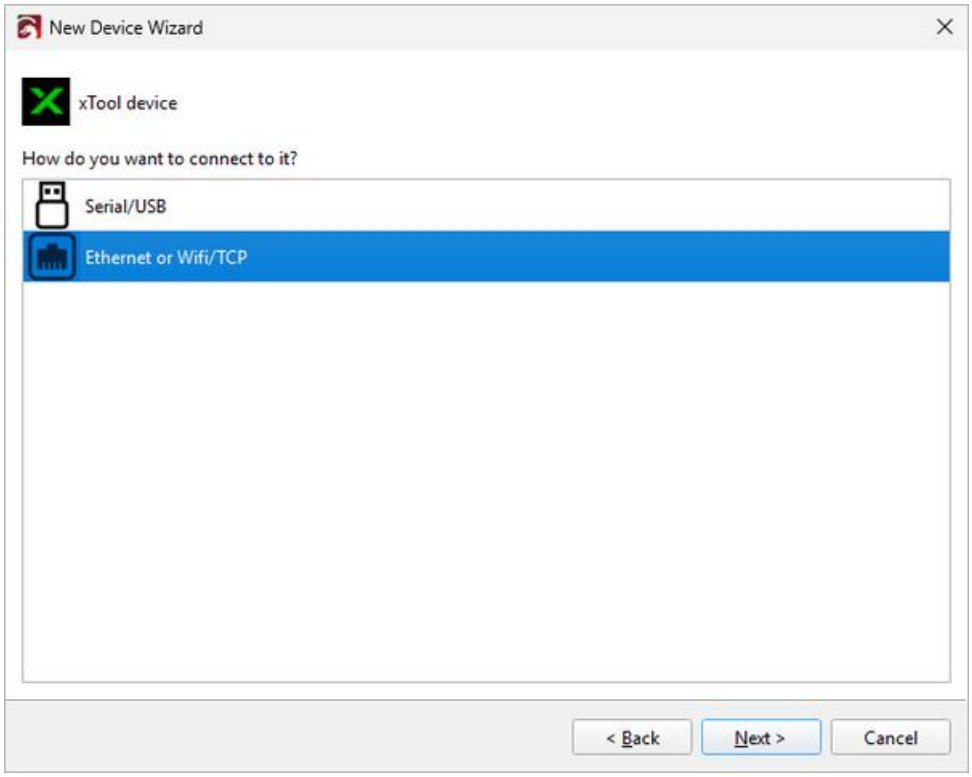
2. In the **Devices** window, click the **Create Manually** button under the device list to open the **New Device Wizard**.



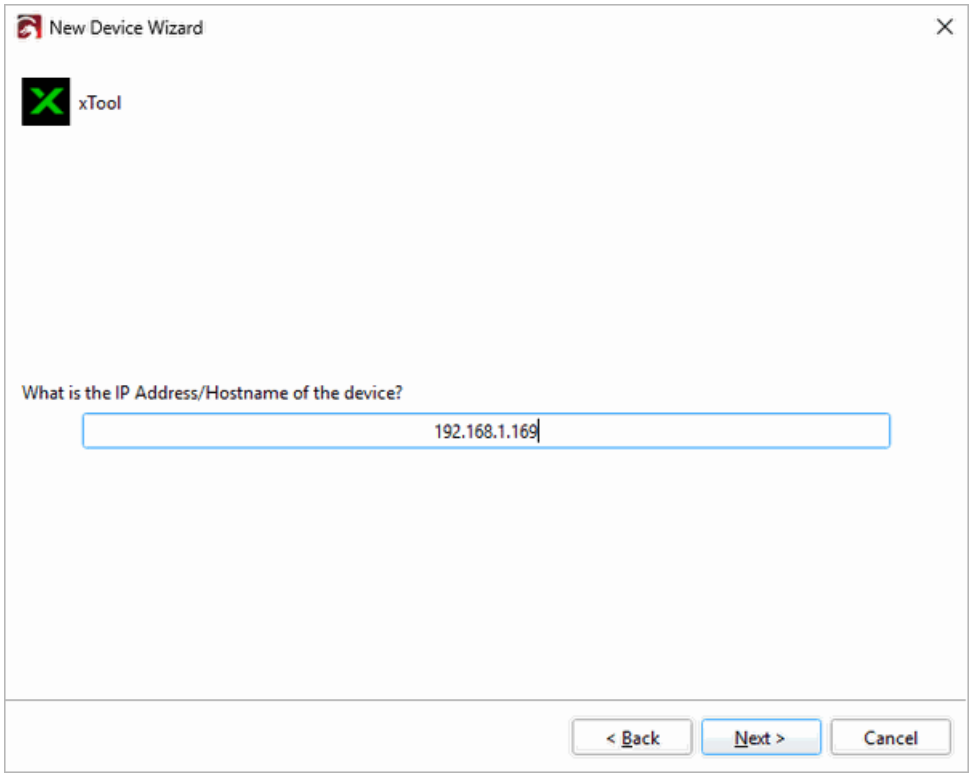
3. In the **New Device Wizard**, scroll down and select **xTool** from the list of device types. Click Next to continue.



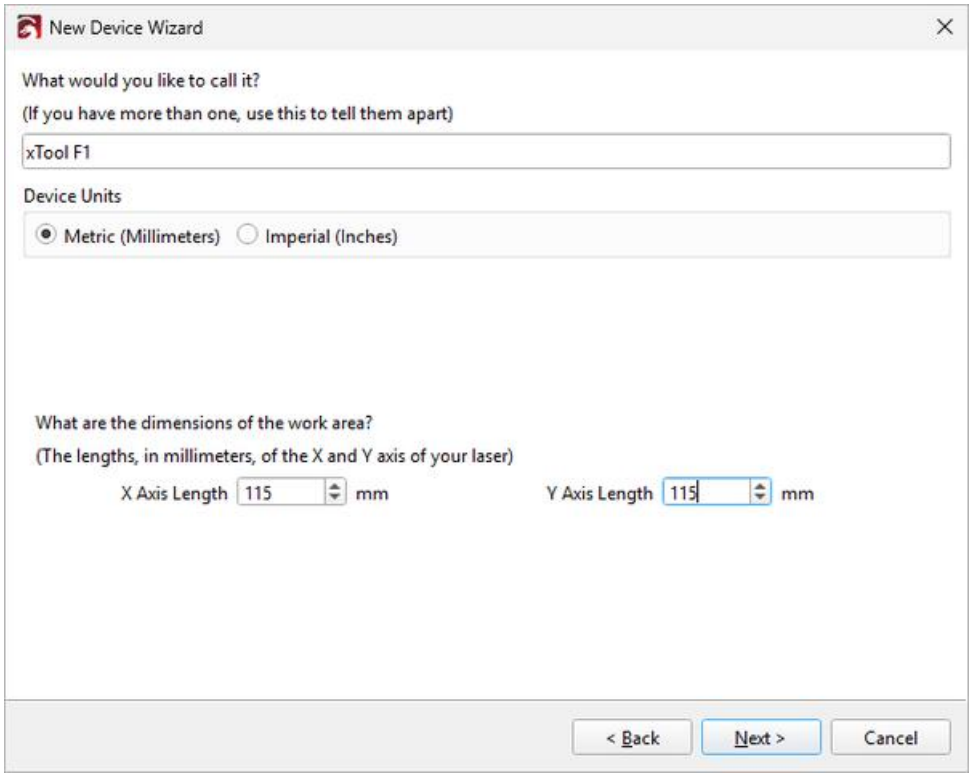
4. Select **Serial/USB** if you're connecting an xTool S1, D1, or D1 Pro with a USB cable. Select **Ethernet or WiFi/TCP** for all other devices or connection types. Click Next to continue.



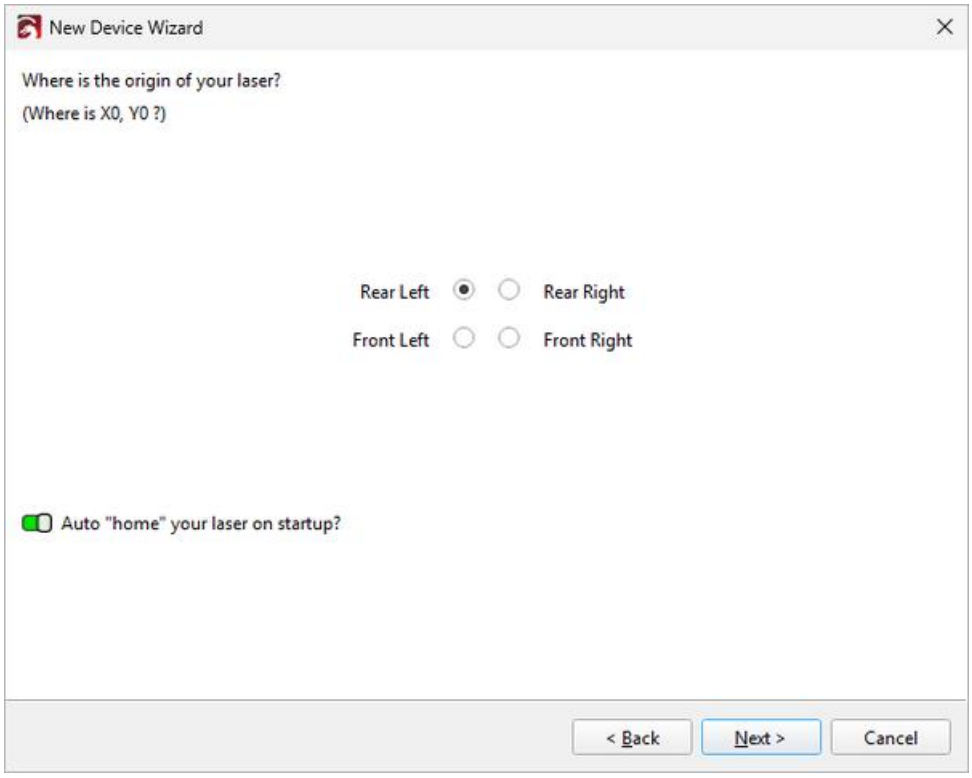
5. Enter the IP address you found in Part 1 to tell LightBurn where to find your laser. If you're connecting an xTool S1, D1, or D1 Pro over USB, LightBurn will skip this step. Click Next to continue.



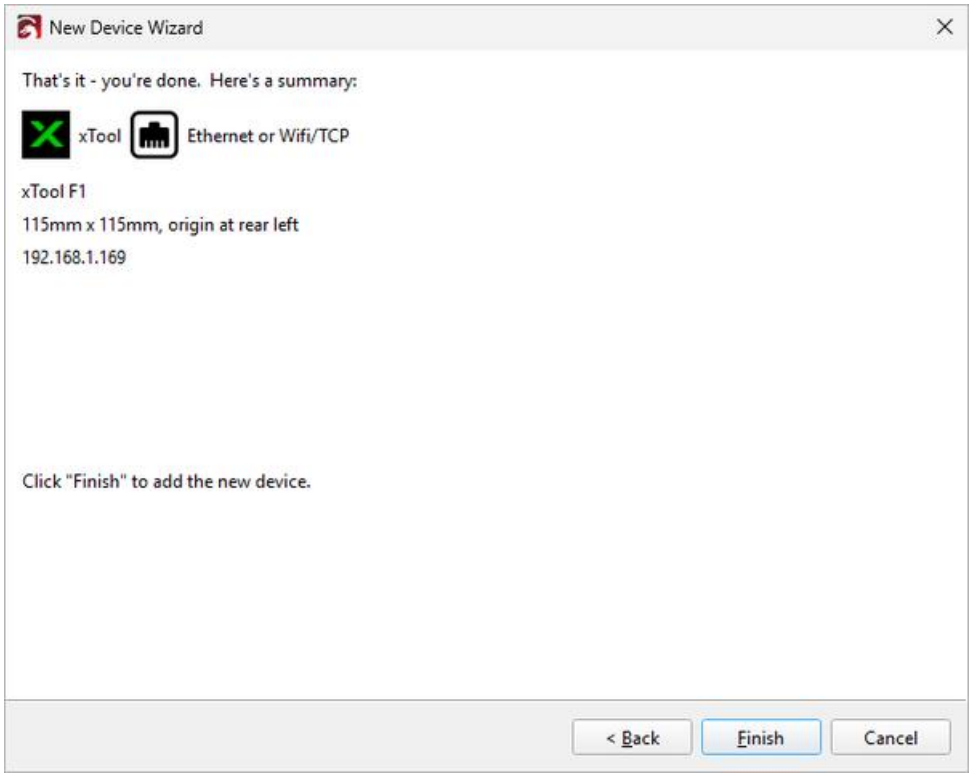
6. Name your device something recognizable and set the dimensions of the work area to match your laser. Leave the **Device Units** in millimeters - if you prefer to design in inches, you can set that separately later. Click Next to continue.



7. Set the origin of your laser to the appropriate corner. Click Next to continue.



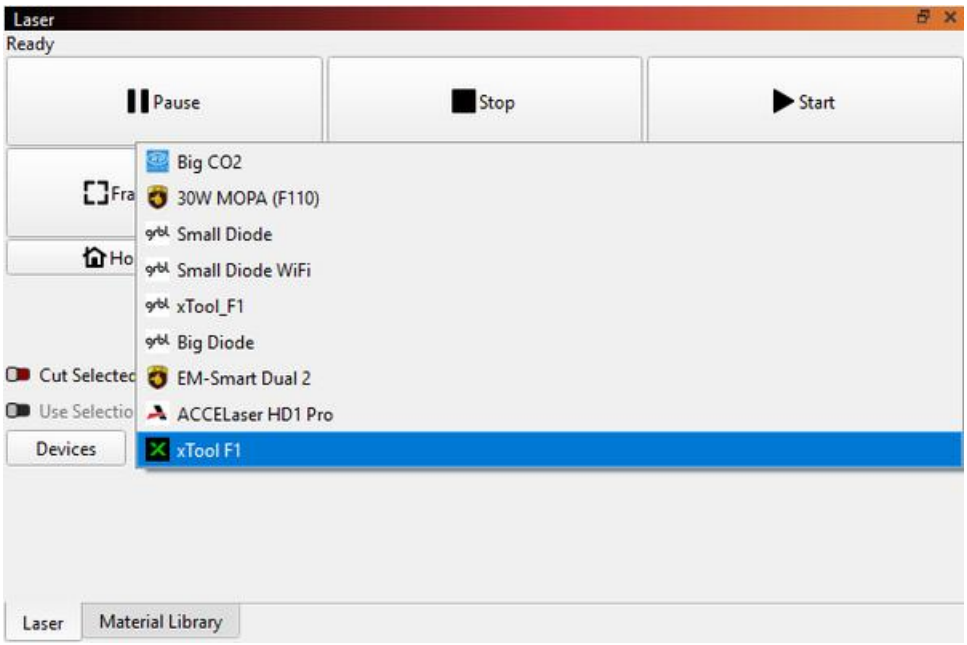
8. Make sure the summary on the final screen seems correct. Click **Back** to go back and make changes, or **Finish** to save your changes.



Check Your Device Settings

Whether you used **Find My Laser** or added your laser manually, you'll need to check your device configuration.

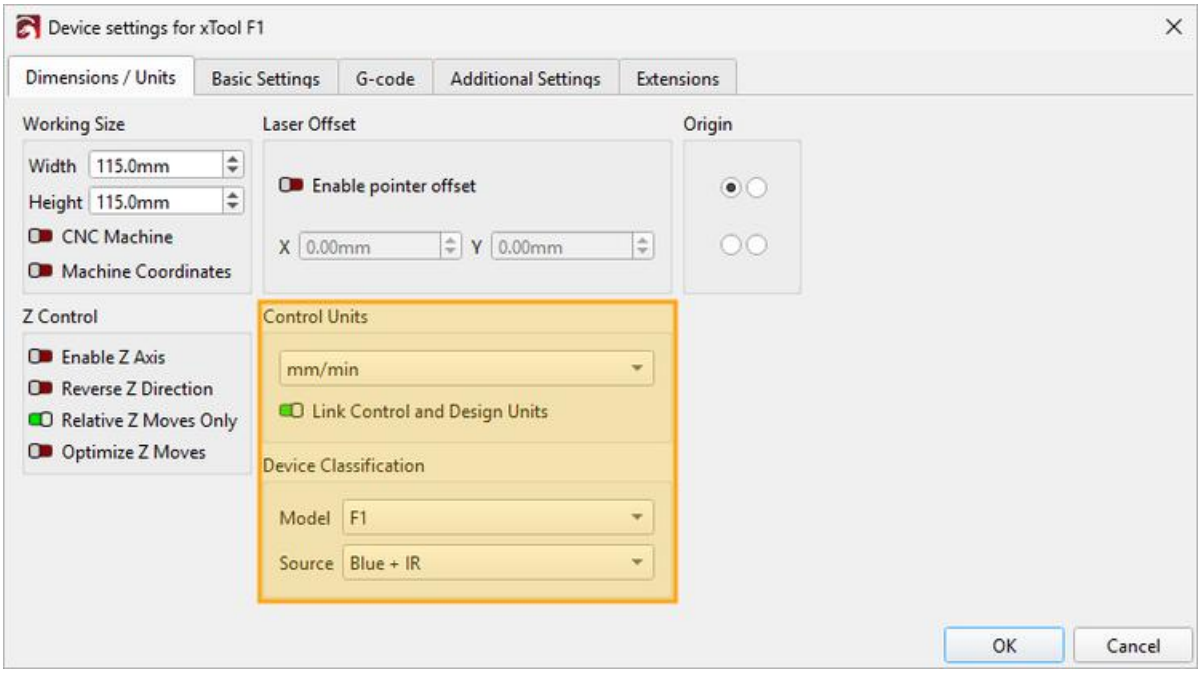
1. If you have multiple lasers configured in LightBurn, make sure your xTool Device is selected in the dropdown in the **Laser** window.



2. Open Device Settings by going to Laser Tools → **Device Settings** or clicking the wrench icon in the toolbar.



3. In the Dimensions / Units tab, find the **Device Classification** section. When you add your laser, LightBurn will attempt to automatically identify it and select the correct device type. If you see unknown or the wrong device, select your xTool model from the **Model** dropdown.



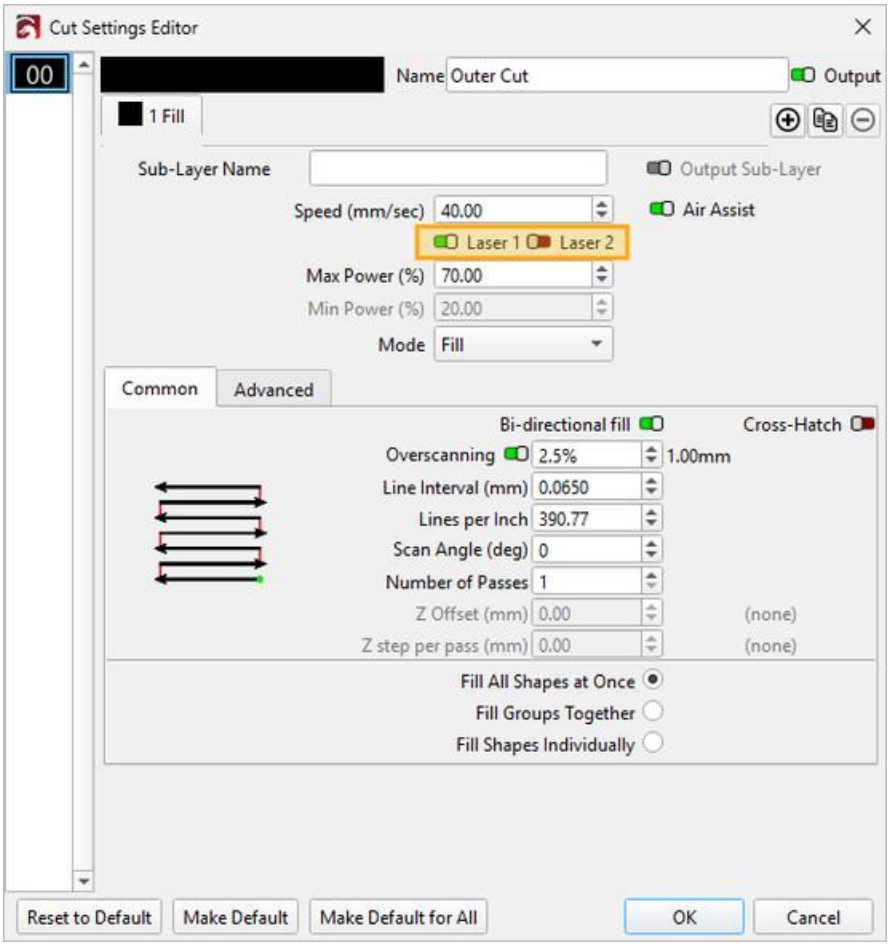
4. Adjust your **Control Units** if needed. These are the units that control how you set your speed.
   - xTool's material settings are given in **millimeters/second**, so if you want to use those recommendations more easily, choose **mm/sec**.
   - In general, the default of **millimeters/minute** has been the more common way to describe diode laser cut settings. If you want to convert a speed from mm/sec to mm/min, divide the value by 60, so 120 millimeters/minute becomes 2 millimeters/second.
   - If you usually design in inches, turn off the **Link Control and Design Units** toggle under **Control Units**. In the Settings Window you can set your **Design Units** to inches. Alternatively, you can set your **Control Units** to **in / mm/sec** or **in / mm/min** to design in inches and control your laser in millimeters.

> **Why is xTool F1 in your device list twice?**
>
> In one of the screenshots on this page, there are two entries for the xTool F1. One has an xTool icon, and the other has a grbl icon. The one with the grbl icon was set up in an earlier version of LightBurn, before the improved xTool support, and will not benefit from the improves in 2.2. If you've already created an xTool device in an older version of LightBurn, you should recreate it using the new setup method.

Selecting the Laser Source for xTool Lasers in LightBurn

Some xTool lasers, like the F1 and F2, are dual-source lasers and can output two different wavelengths of light to work with different materials. LightBurn 2.2's new xTool device type adds native support for switching between the laser sources, making it much easier to use both sources than in previous version of LightBurn.

To switch between laser sources on a dual source laser, double click an entry in the Cuts / Layers Window to open the Cut Settings Editor. In this window, you can choose between **Laser 1** and **Laser 2** to switch which laser source will be used for that layer. This also allows you to switch between the two sources in a single job. See the screenshot below for the location of the laser source toggle.



xTool and Galvo features

LightBurn's 3D Sliced Engraving and Split Marking features are not currently available for xTool lasers. We are working on making these features available for xTool galvo lasers such as the xTool F1 and F2 as well.

xTool Firmware

The new xTool device type in LightBurn 2.2 may require you to update your device's firmware.

> **Why does the firmware version matter?**
>
> Without going into too much detail, the newer firmware changes the laser's communication method. The newer firmware listed below is compatible with LightBurn and with xTool Studio versions newer than 1.6.

As of October 16, 2026, the following firmware versions are required for the lasers shown below. If you want to go back to the older firmware at any point, you can find directions on xTool's Support Site.

Laser	Firmware Version
xTool F1	40.51.020.04 or higher
xTool F1 Lite	40.55.020.04 or higher
xTool F1 Ultra (Class 4)	40.52.016.05 or higher
xTool F1 Ultra (Class 1)	40.53.007.05 or higher
xTool P2S	40.22.011 or higher

Detailed xTool Device Support List

We've been working with xTool to expand and improve support for xTool devices. The table below summarizes xTool device support as of October 6, 2026.

Device	Supported	Notes
D1	Yes	If connecting with a USB cable, does not require an IP address.
D1 Pro	Yes	If connecting with a USB cable, does not require an IP address.
D1 Pro 2.0	Yes	If connecting with a USB cable, does not require an IP address.
S1	Yes	If connecting with a USB cable, does not require an IP address.
F1	Yes	
F1 Lite	Yes[1]	
F1 Ultra	Yes	Manually editing GCode is no longer required to use the xTool F1 Ultra with LightBurn.
F2	Yes[1]	
F2 Ultra	Yes[1]	
F2 Ultra Single	Yes[1]	
F2 Ultra UV	Yes[1]	
P1	No	Unsupported
P2	Yes[1]	Some more recent firmware for the xTool P2 does not currently support camera use in LightBurn.

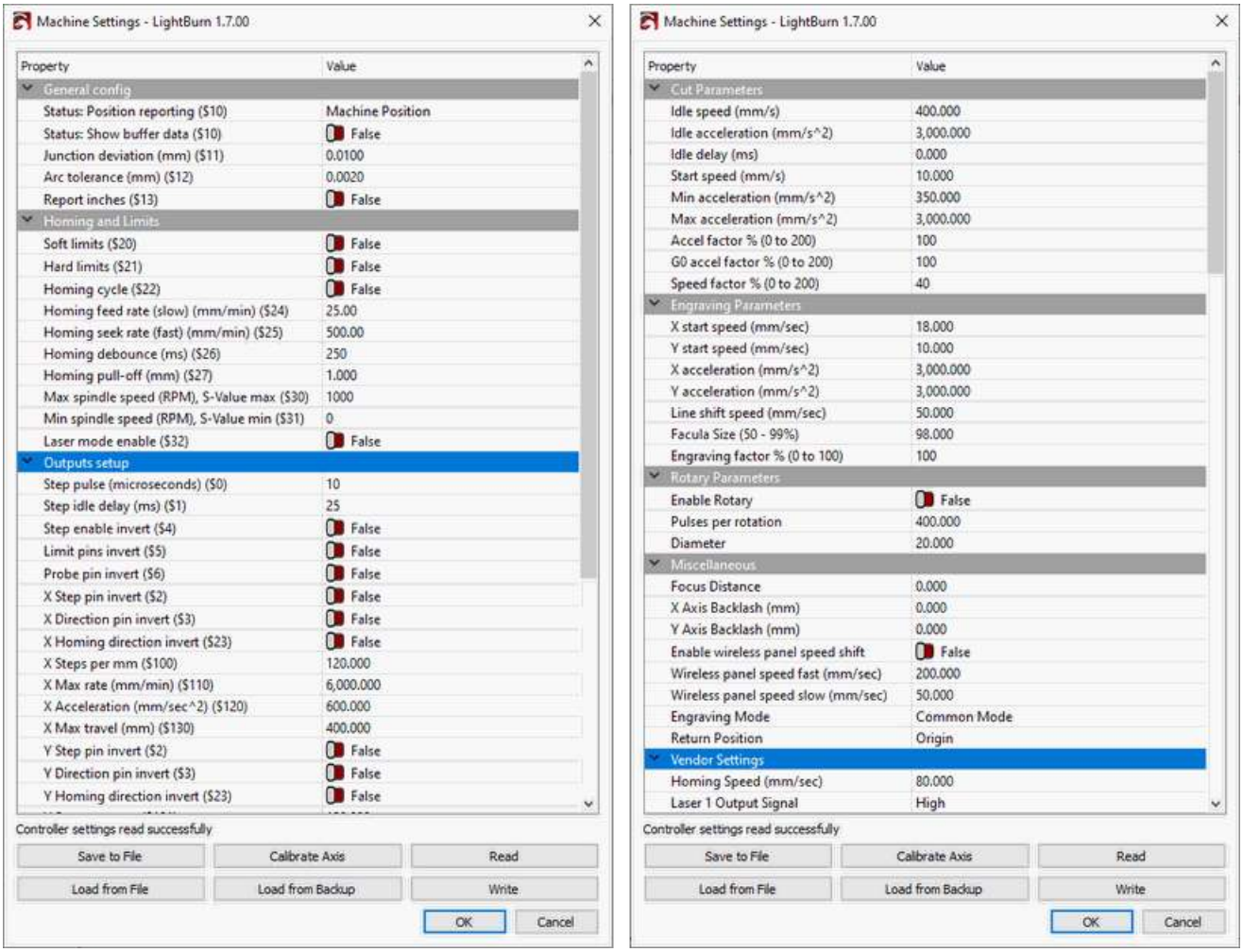
Device	Supported	Notes
P2S	Yes	Some more recent firmware for the xTool P2S does not currently support camera use in LightBurn.
P3	No	Unsupported
M1	No	Unsupported
M1 Ultra	No	Unsupported
M2	No	Unsupported
Retail Marker	No	Unsupported
MetalFab	No	Unsupported

1. At the present time, this device should be functional, but needs further testing.

Machine Settings

The **Machine Settings** window allows you to view and edit firmware settings on lasers with Ruida, TopWisdom, or GRBL-based controllers.

To open the Machine Settings window, go to Laser Tools → **Machine Settings**.



Left: The Machine Settings window for a GRBL controller.

Right: The Machine Settings window for a DSP controller.

Working With Machine Settings

The settings available are determined by your laser controller's firmware. In general, if your laser is working correctly, you can (and should) leave the settings here alone. For additional information on specific settings, consult your laser's manufacturer, or see the documentation for your laser's controller.

Outputs Setup / Vendor Settings

This section will display differently depending on whether you are connected to a DSP or GRBL laser. All settings in this section are hidden by default, and you'll see a warning message if you click to expand and view them. These settings are typically set by the manufacturer, and should generally be left alone unless instructed otherwise by support.

Save to File

Saves your current controller firmware settings to a `.lbset` file. We strongly recommend using this feature to back up firmware settings before making any modifications.

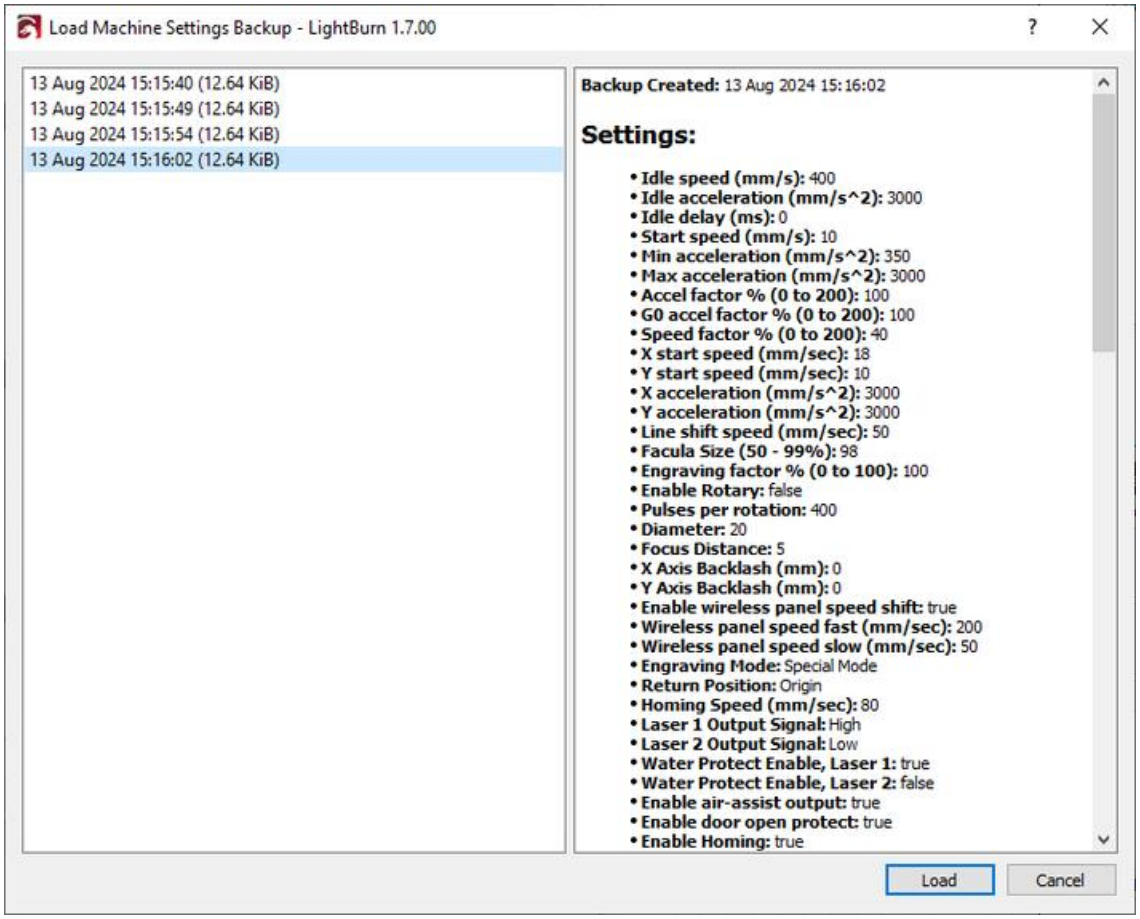
Load from File

Loads a `.lbset` file and overwrites all firmware settings with its contents. In some cases, your laser manufacturer may make this file available as a backup.

> **Warning**
>
> This will overwrite all values. Make sure you back up your existing settings first using the Save button, and only load parameters from a known and trusted source.

Load From Backup

Use **Load from Backup** to restore automatically-generated backups of firmware settings. LightBurn will automatically store backups each time you Write new settings to your controller.



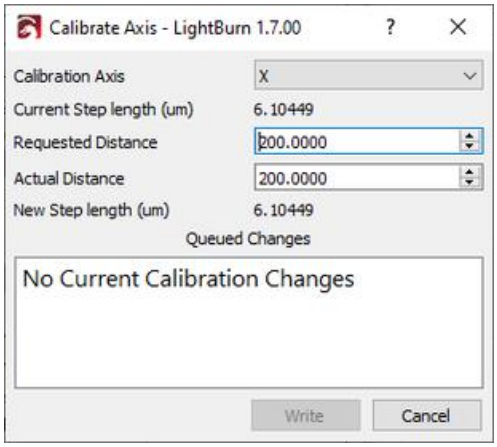
Read

Reload the settings from the controller. Any changes that haven't been written to the controller will be lost.

Write

Writes all currently modified settings back to the controller, saving your changes. Changes are not saved unless you click **Write**.

Calibrate Axis



The **Calibrate Axis** tool is a shortcut to help you calculate the **Steps/mm** or **Step Length** value for each axis, if they need adjustment — if the current values are inaccurate, output may be stretched or squished.

See Explainer: Steps/MM for more information.

To use Calibrate Axis:

1. Choose an axis to calibrate in **Calibration Axis**.
2. Enter a distance to travel in **Requested Distance**. Larger distances can be helpful when measuring with less precise tools.
3. Mark the start point, move the laser the specified distance, and mark the end point. The easiest way is to burn a line the specified length, although you can also use the Jog functions in the Move Window or built into some controllers.
4. Measure the actual distance traveled and enter it into **Actual Distance**.
5. Click on **Write** to save the value to the controller.

This may need to be done multiple times, especially for less precise measuring tools, to reach an acceptable level of accuracy.

Troubleshooting

- If LightBurn is unable to read the **Machine Settings**, make sure your laser is connected and turned on.
- See Troubleshooting: Connection Issues for help connecting to your laser.

Related Topics

- Device Settings
- Console Window

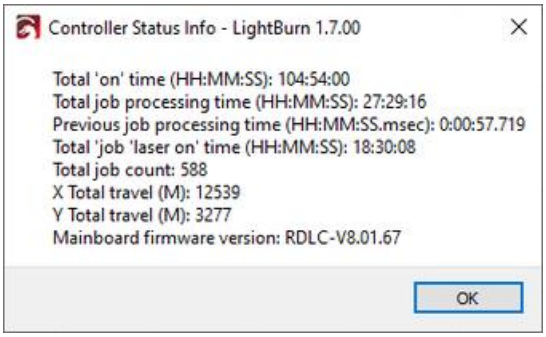
Get Controller Info

The **Controller Status Info** window provides information about your connected laser's controller. It primarily shows usage statistics, but also contains information about the controller's firmware version.

To view Controller Info, go to Edit → **Get Controller Info**.

> **Note**
>
> Get Controller Info is only available for lasers with Ruida controllers.

Controller Status Info



Name	Description
Total "on" time	Total time the laser has been on for.
Total job processing time	The total time the laser has spent processing jobs sent to it.
Previous job processing time	The time the laser spent processing the most recent job sent to it.
Total job "laser on" time	The total time the laser has spent running jobs.
Total job count	The total number of jobs run on the laser.
X Total travel	The total horizontal distance traveled in meters.
Y Total travel	The total vertical distance traveled in meters.
Mainboard firmware version	The firmware version installed on the laser controller.

Troubleshooting

- If LightBurn is unable to open the **Controller Status Info** window, make sure the laser is connected and turned on.
- See Troubleshooting: Connection Issues for help connecting to your laser.

Related Topics

- Device Settings

- Machine Settings
- Devices

Console Window

The **Console Window** displays messages from your laser's controller and commands sent to it by LightBurn. You can also input direct commands to your laser through this window.

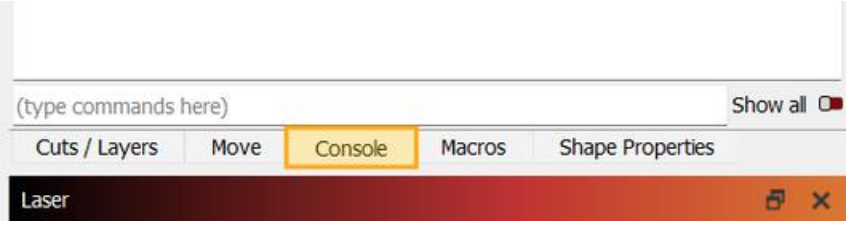
The Console Window functions are only available when LightBurn is directly connected to a GCode-based laser.

> **Note**
>
> Users with DSP or Galvo lasers with Ruida, TopWisdom, Trocen, EZCAD2, EZCAD3 or BSL boards cannot use the Console Window to communicate with their laser's controllers.

Accessing the Console Window

If you have a GCode-based device profile selected, the **Console Window** will be located in the upper right side of LightBurn, docked behind the Cuts / Layers Window, by default.

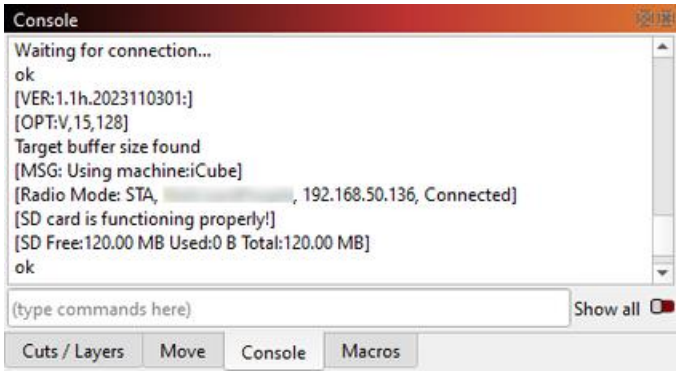
If you have closed the Console Window, go to Window → **Console** to re-enable it. To restore it and all other windows to their default positions, go to **Window** → Reset to Default Layout.



> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Startup Message

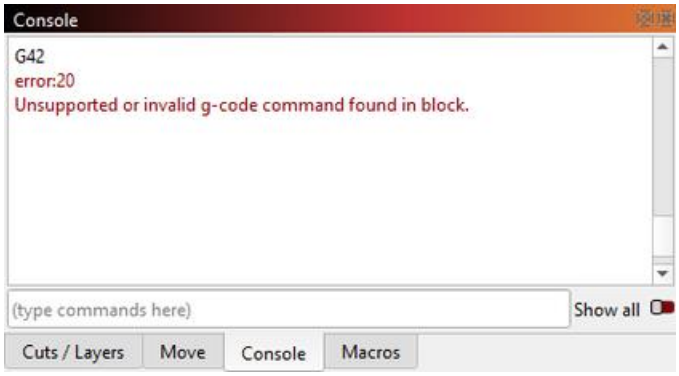
When you first connect to your laser you will see a response message in the **Console Window**, indicating that LightBurn is successfully communicating with the controller.



The startup response message displays the type and specific version of firmware your laser's controller is running. Most GCode-based lasers run GRBL firmware, version 1.1f or later.

If you do not see a similar message from the controller when you start up and attempt to connect to your laser, see Troubleshooting: Connection Problems.

Error Messages

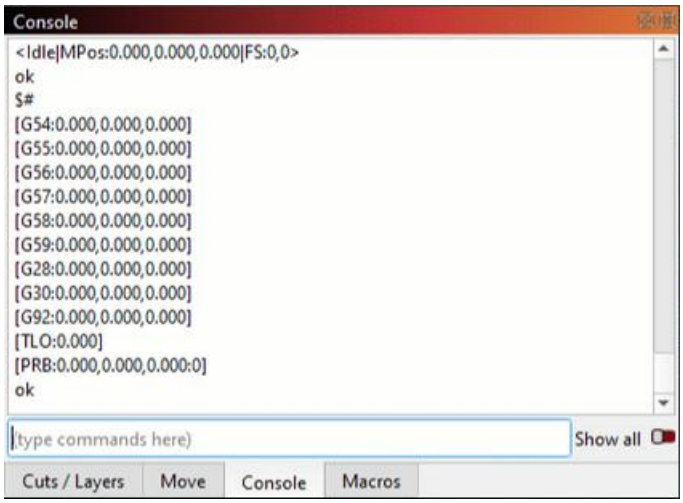


The **Console Window** will sometimes display alarm or error messages sent to LightBurn by the laser's controller, indicating that there was a problem with the machine's operation.

See Troubleshooting: GRBL Errors to find solutions to common alarms and errors.

Common GRBL Commands

Use the text box in the **Console Window** to directly enter commands to your laser's controller. These commands are only valid for lasers running GRBL firmware.



Request Information

Use the following commands to prompt your laser to return information on its settings or status.

- `$I` : display the controller's firmware version.

- `$$` : display GRBL settings. See the official GRBL documentation for more information on each setting.
- `$#` : display stored offsets in the machine's coordinate systems.
- `?` : display status and current position.

Alter Settings and Status

Use the following commands to adjust firmware settings or change machine status.

- `$X` : unlock machine if it has been locked due to an alarm or error.
- `$[setting]=[value]` : adjust a specific GRBL setting by entering `$`, followed by the number of the setting to be adjusted, `=`, and the new value to assign to the setting.
  - Example command: `$110=6000`
- `RST=#` : reset all offsets stored in the machine's coordinate systems.
- `RST=$` : restore all GRBL settings to "factory" default.
- `RST=*` : reset all EEPROM data used by GRBL, including all offsets stored in the machine's coordinate systems and GRBL settings.

> **Warning**
>
> Do not reset these settings unless directed by to do so by your machine's manufacturer, or LightBurn support.
>
> In most cases, resetting GRBL settings restores all values to those set by your machine's manufacturer, but can sometimes restore them to stock GRBL settings, which will necessitate significant reconfiguration in order to operate your laser.

Show All

Enable the **Show All** switch to display all communications between LightBurn and your controller. When Show All is disabled, the **Console Window** will display startup messages, simple commands, alarm or error messages, and all responses to status queries, but hide extensive commands such as those used to jog or fire your laser.

Macros

> **Where did the macro buttons go?**
>
> Prior to LightBurn 2.0, LightBurn had 6 slots available for macros. LightBurn 2.0 has a new interface for creating any number of macro buttons you want, which can be found in the Macros Window.

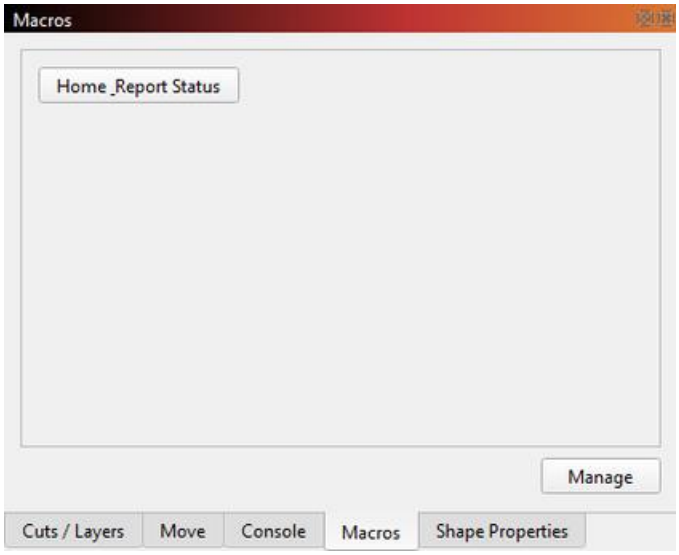
LightBurn allows the creation of **Macro** buttons to store and access frequently used commands instead of typing them into the **Console Window** manually each time. For more information, please see our page on the Macros Window.

Related Topics

- Identify Your Laser
- Troubleshooting: Connection Problems
- Troubleshooting: GRBL Errors
- Macros Window

Macros Window

The **Macros Window** allows you to create buttons to store frequently used commands for GCode-based lasers. These functions will only work when LightBurn is directly connected to a GCode-based laser.



Click on macro buttons in the Macros Window to use them.

Accessing the Macros Window

If you have a GCode-based device profile selected, the **Macros Window** will be located in the upper right side of LightBurn, docked behind the Cuts / Layers Window by default.

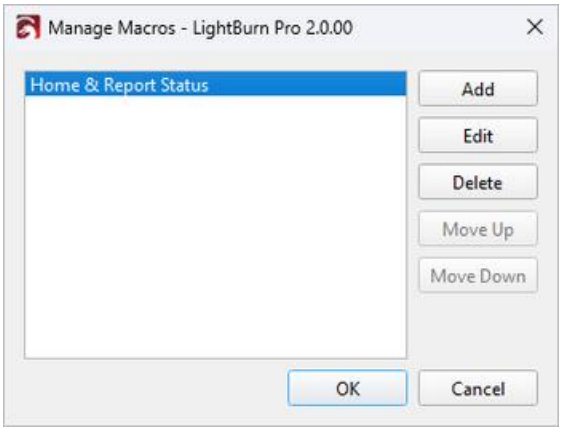
If you have closed the Macros Window, go to Window → **Macros** to re-enable it. To restore it and all other windows to their default positions, go to **Window** → Reset to Default Layout.

> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

Creating and Managing Macros

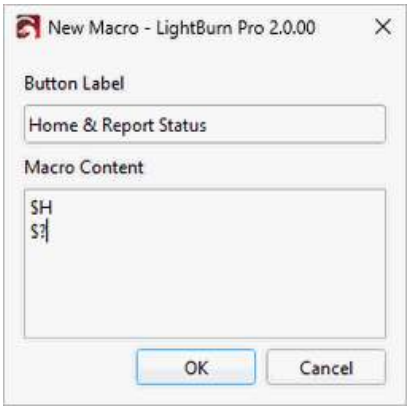
To create, edit, and rearrange macros, click the **Manage** button in the bottom right of the **Macros Window** to open the **Manage Macros** dialog.

This window allows you to add, edit, delete, and rearrange macros.



Add a Macro

1. Click **Add** to open the **New Macro** dialog.
2. Enter a name for the button under **Button Label**.
3. In the **Macro Content** section, enter one or more commands.
4. Click **OK** to save the macro or **Cancel** to discard changes.



Edit a Macro

Edit a macro by selecting it in the **Manage Macros** window and clicking **Edit**. From here, the process is the same as adding a new macro.

Delete a Macro

Delete a macro by selecting it in the **Manage Macros** window and clicking **Delete**.

Rearrange Macros

Change the order in which macros appear by selecting the macro you want to move and clicking **Move Up** or **Move Down**, or by clicking, holding, and dragging a macro from the list to a new position.

Troubleshooting

- If you're unable to use or manage macros, make sure you've selected a GCode device profile.
- If your macros don't work as expected, please check that you've entered them correctly. We recommend testing your commands in the Console Window first.

Related Topics

- Console Window
- Troubleshooting: GRBL Errors
- Troubleshooting: Connection Problems

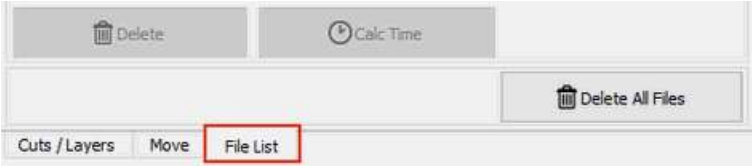
File List Window

Use the **File List Window** to browse and manage files stored on your laser's controller. This window is only available for users with DSP controllers that allow saving files to the controller.

Accessing the File List Window

If you have a DSP device profile selected, the **File List Window** is located in the upper right side of LightBurn, docked behind the Cuts / Layers Window, by default.

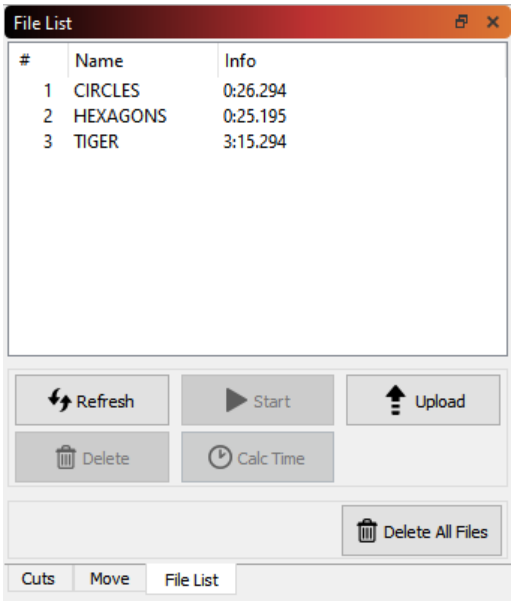
If you have closed the File List Window, go to Window → **File List** to re-enable it. To restore it and all other windows to their default positions, go to **Window** → Reset to Default Layout.



> **Tip**
>
> The layout of LightBurn is highly customizable. For more information on enabling and disabling windows and toolbars, or rearranging the default layout, see Customizing the LightBurn Window.

File List Window Options

Click any option in the image below to jump directly to the relevant section for that option, or scroll down for a list of options and descriptions.



Select File

Select a file from the list to enable all relevant buttons.

Refresh

Queries the DSP controller to update the list of files stored on the laser file system.

Start

Commands the laser to run the selected file.

Upload

Opens your system's file browser, where you can select and send any files previously saved to your computer in machine-specific format to the laser's controller.

> **Upload Current Project**
>
> Use the Send button in the Laser Window to upload the current project in the LightBurn Workspace.

Delete

Removes the selected file from the DSP controller file system.

> **Warning**
>
> Deleted files cannot be recovered.

> **Tip: Ruida Controllers**
>
> Start deleting from the bottom of the list. Starting from the top of the list may lead to extended file compaction time.

Calc Time

Press the **Calc Time** button to begin computing the job completion time for the selected file. The **Calc Time** button is available only on supported DSP controllers (Ruida). This can take a considerable amount of time for engraving jobs, as the controller is simulating the job through the onboard DSP microprocessor. Once complete, the calculated completion time will be shown in the **Info** column.

Delete All Files

Removes all files from the DSP controller file system.

> **Warning**
>
> Deleted files cannot be recovered.

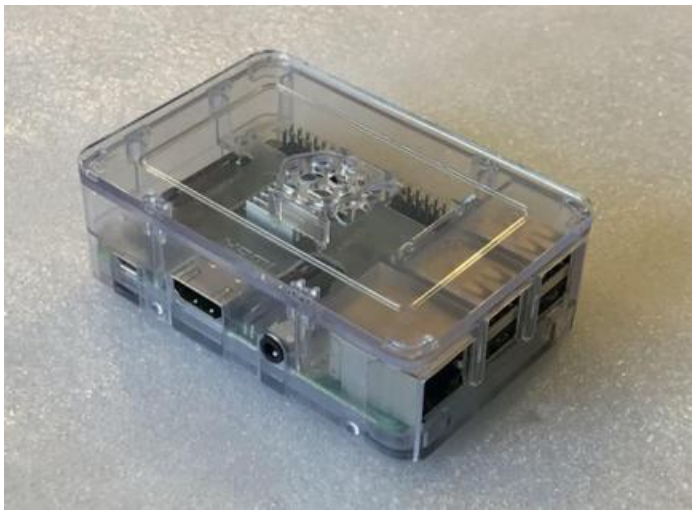
Related Topics

- File Management

- File Menu
- Laser Window
- Identify Your Laser

LightBurn Bridge

The LightBurn Bridge was created by LightBurn to simplify Ethernet connections to Ruida controllers, especially for computers without Ethernet ports or where running a cable between the laser and computer is impractical. The LightBurn Bridge connects to your laser with an Ethernet cable and to your computer using Wi-Fi, and acts as a relay between the two. There are no drivers to install, and the Bridge typically sends data much faster than a USB connection.



LightBurn Bridge

> **Warning**
>
> LightBurn Bridge is intended to be an appliance. The software is thoroughly tested, but we do not support modifications to the hardware or firmware. If you alter the system to add functionality beyond what is covered here or in the LightBurn Bridge Guide, we will not be able to provide support.

Basic Setup

The steps below cover basic setup for the LightBurn Bridge. For additional help or more advanced configuration, please see the LightBurn Bridge Guide.

Hardware Setup

Before you can configure the Bridge in LightBurn, you will need to connect the hardware and set the appropriate IP address on your Ruida controller:

- Use the Ruida control panel on your laser to set the IP address to **10.0.3.3**.
- Use an Ethernet cable to connect the Bridge to the laser's Ethernet port.
- Power up the Bridge with its power adapter.

Add Device to LightBurn

In LightBurn's Devices window, click **LightBurn Bridge** to open up the **Bridge Discovery Wizard**, which will walk you through connecting LightBurn to the Bridge. If you need to go back later, you can double-click the laser in the Devices window to re-open the wizard and change the LightBurn Bridge configuration, or you can adjust your device's configuration from Device Settings, just as if you were connecting directly to the laser.

Troubleshooting

- If you're not sure which corner to use for the origin, power on your laser. The corner the head goes to is your origin.
- The LightBurn Bridge is only for use with Ruida controllers. Other lasers will not work with it.
- If you're on macOS 15 or higher, please see our directions on allowing access to the local network.

Related Topics

- Devices
- LightBurn Bridge Guide

Camera Overview

> **Significant Changes in LightBurn 2.2**
>
> LightBurn 2.2 includes significant UI improvements to the camera interface as well as updates to make it easier to calibrate your camera. If you are on an older version of LightBurn, use the dropdown at the top of the page to select the appropriate version of the camera pages.

Camera Overview

If your laser has a LightBurn-compatible camera installed, you can use it to position designs on your material and to trace artwork or objects placed in your laser. Before you can use the camera features, you'll need to Calibrate and Align your camera.

Camera Changes in LightBurn 2.2

- The **Cameras Window** is now the Cameras / Overlays Window.
- New Camera Calibration features, including an improved capture experience when using **Auto-Capture**.

Skip to the Cameras / Overlays Window

Getting Started

Before you can use a camera in LightBurn, it has to be calibrated and aligned. If you move your camera, you'll need to redo the camera alignment, but the camera calibration doesn't need to be redone.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=videoseries

The video above uses an older version of LightBurn and may not match perfectly with what you see on your screen.

Importing an Existing Setup

If you've already set up your camera on a different computer or using an older version of LightBurn and exported a camera file, you can import your existing calibration and alignment into LightBurn 2.1.

If you're happy with your existing calibration and alignment, great! If not, you may want to try redoing either (or both!) with our improved processes.

New Camera Setup

If you haven't already configured a camera, or you just want to start from scratch, you'll need to do the following:

1. Select and install a camera, if you haven't already done so.

2. Add Your camera to LightBurn.
3. Calibrate your camera lens to correct for the camera's distortion. We have presets for the LightBurn cameras, so if you've purchased your camera directly from us, you can use the appropriate preset.
4. Align your camera to your workspace. Unlike Lens Calibration, which is based only on the camera itself, Alignment depends on the camera's location in the machine, so you'll have to redo this if you move the camera.

Cameras / Overlays Window

You'll use the **Cameras / Overlays Window** to add cameras, work with camera overlays, and check camera status. For more details, see Cameras / Overlays Window. You also might want to jump to Camera Lens Calibration.

Camera Types

You can set up multiple cameras for your laser, and depending on your laser, you may want to. For larger lasers in particular, combining an **Overhead Camera** for an overview of the entire workspace with a **Head Camera** for a more accurate and detailed view of the area you're going to be working in can be very effective.

Overhead Camera

An **Overhead Camera** is one that's mounted to your laser's frame (often the lid) and doesn't move around while in use. Overhead cameras are intended to show an overview of the laser's entire work area, letting you preview your entire workspace at once. When you choose a camera to use as an Overhead Camera, it's important to choose something with a wide enough field of view to show your entire workspace, and high enough resolution to get a useful amount of detail.

On larger lasers, it can be difficult to get a wide enough field of view to cover the entire bed, so some users set up two Overhead Cameras, each one covering half the bed. If you do this, you'll calibrate and align each camera separately, and enable both in the Overlay.

You can also use an Overhead Camera to get an overview of the entire workspace and combine it with a Head Camera to get a close-up view and refine the overlay for the specific part of the bed you're working in.

Head Camera

A **Head Camera** (formerly referred to as a Head-Mounted Camera in LightBurn) is one that's mounted to the head of your laser and moves around with it. Head Cameras aren't as effective as Overhead Cameras for getting an overview of the entire work area, but they're great for getting a sharper, more detailed view of a specific part of your workspace. The difference is particularly significant towards the edges of the workspace, where an Overhead Camera's view will be more distorted, and with particularly large lasers, where an Overhead Camera may not have enough resolution to show much detail.

You can get an overlay of the entire workspace with a Head Camera by having LightBurn stitch together captures of the entire workspace, but this is slower and less effective than with an **Overhead Camera**.

LiDAR

LiDAR Ranging, or Depth Scanning, uses a sensor to create a height map of the surface of an object loaded into your laser. This is currently only supported on the ACCELaser HD1. See LiDAR for more information.

Cameras / Overlays Window

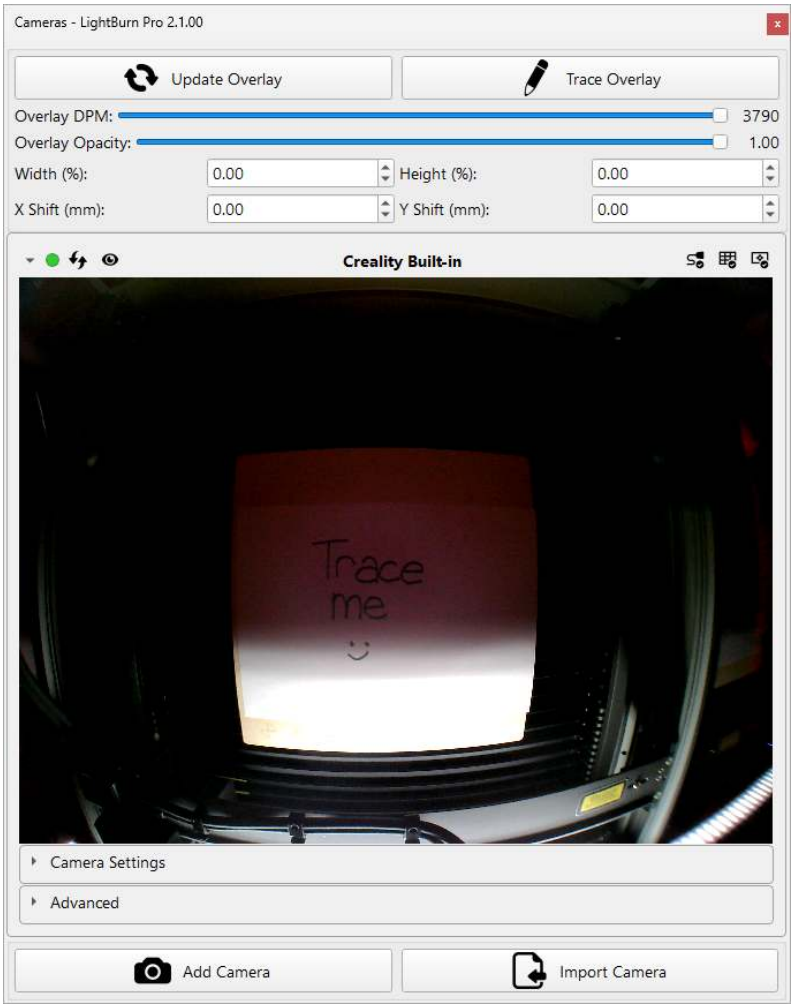
About The Cameras / Overlays Window

> **Info**
>
> The **Cameras** window is now the **Cameras / Overlays** window.

Use the **Cameras / Overlays** window to view and manage connected cameras. To enable the Cameras / Overlays window, go to Window → **Cameras**. If you don't have any cameras available, you'll need to Add a Camera.

If you have multiple lasers, cameras are specific to each device, so changing devices will change which cameras you havee available.

Click any option in the image below to jump directly to more information about it, or scroll down for a list of options and descriptions.



Using the Cameras / Overlays Window

Each camera you've added for your currently selected laser will show up in the Cameras / Overlays window, with the following sections:

- Camera Overlay Toggle & Resolution
- Overlay Tools & Opacity Slider
- Camera Preview
- Camera Settings
- Adjust Overlay
- Advanced

At the bottom of the window, you'll find:

- Home button to home your laser.
- Add Camera button.
- Manage Cameras button.
- Import Camera button.

Keep reading for more on each of these.

Camera Overlay Toggle & Resolution

Each camera section starts with a toggle switch to enable or disable the camera in the overlay on the workspace. If the toggle is disabled, the camera will not appear in the overlay.

This section also tells you whether the camera is an Overhead or Head camera, and shows you the resolution in terms of pixels/mm. The higher this value is, the more detail your camera can show.

Overlay

Each camera you add will have a section with buttons to control and manage the Camera Overlay, as well as a slider to control the opacity.

Update Overlay

Depending on what type of camera you have, your Update Overlay options will change.

Overhead Camera: Update Overlay

For Overhead cameras, the **Update Overlay** button will capture an image from your connected overhead camera(s) and update the overlay in the workspace. If you have multiple overhead cameras, all of them will be included (unless they're disabled). Use this to quickly update the Workspace with an overview of the bed of your laser.

Head Camera: Single, Selected, and Workspace Capture

Head Cameras have three different ways to update the Overlay: Single, Selected and Workspace.

Single

The **Single** capture option focuses the Z axis and captures a single image at the laser head's current location.

Selected

The **Selected** capture option requires you to have at least one shape selected. It will move the laser head over the selected shape(s) and stitch captures together for the area around your selected shapes.

Workspace

The **Workspace** capture option moves the laser head over the entire Workspace and stitches captures together to create an overview of the entire laser bed, similar to what you would get from an Overhead Camera.

Trace

Click **Trace Overlay** to open LightBurn's Trace Image tool and trace directly from the camera.

Clear

Click **Clear** to clear the overlay for that particular camera.

Opacity

Use the **Opacity** slider to change the transparency of the overlay for that particular camera. Reducing the opacity can sometimes make it easier to see what you're drawing.

> **Info**
>
> In LightBurn 2.2, the opacity slider for each camera only goes down to 10% opacity. To hide the overlay entirely, turn off the Camera Overlay Toggle

Camera Preview

Each connected camera has a preview with a status bar. The status bar shows you the current state of the camera and gives you quick access to tools like Calibration and Alignment. You can find more info about each item in the status area in the table below.

Camera Status Area

Name	Description
Arrow	Use this to expand or collapse individual cameras.
Streaming Indicator (Green Dot)	If a camera is connected and streaming video to LightBurn, a green circle appears in the status area.
Update Overlay	Click this button to update the Camera Overlay.
Included in Global Overlay	Use this button to toggle whether or not the camera should be included in the global Camera Overlay. If a camera is disabled, the eye will be crossed out with a red background.

Name	Description
Camera Connection Indicator	Has a red background if the camera is disconnected. Click to open the Edit Camera window.
Camera Calibration Indicator	Has a red background if the camera is uncalibrated. Click to open Camera Calibration.
Camera Alignment Indicator	Has a red background if the camera is not aligned to the workspace. Click to open Camera Workspace Alignment.

Context Menu

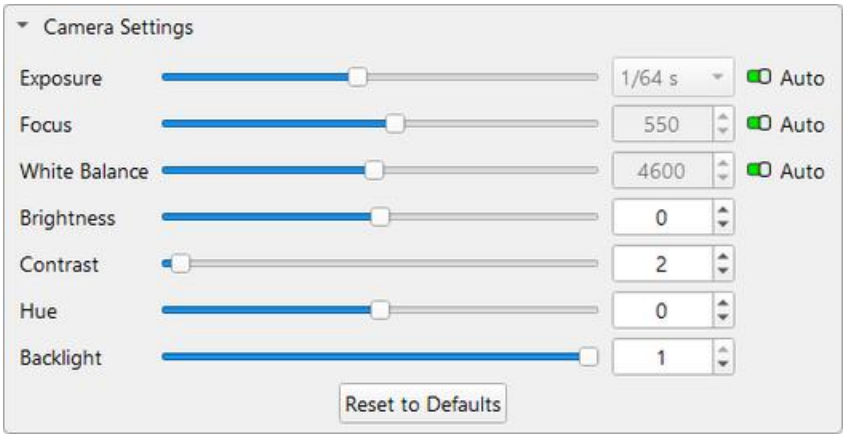


Right-click in the Camera section to open the context menu. The first line on the menu tells you which camera you're working with. In addition:

- **Edit** opens the Edit Camera Window for the selected camera.
- **Remove** will remove the selected camera.
- **Export** will export a LightBurn Camera File so you can back up your camera settings or copy them to a different LightBurn installation.

Camera Settings

The **Camera Settings** section contains settings for adjusting your camera image.



The default settings here are chosen to work for most setups, but you can adjust your settings using the sliders or numeric entry boxes. For **Exposure**, **Focus**, and **White Balance**, you can also enable or disable automatic adjustment here.

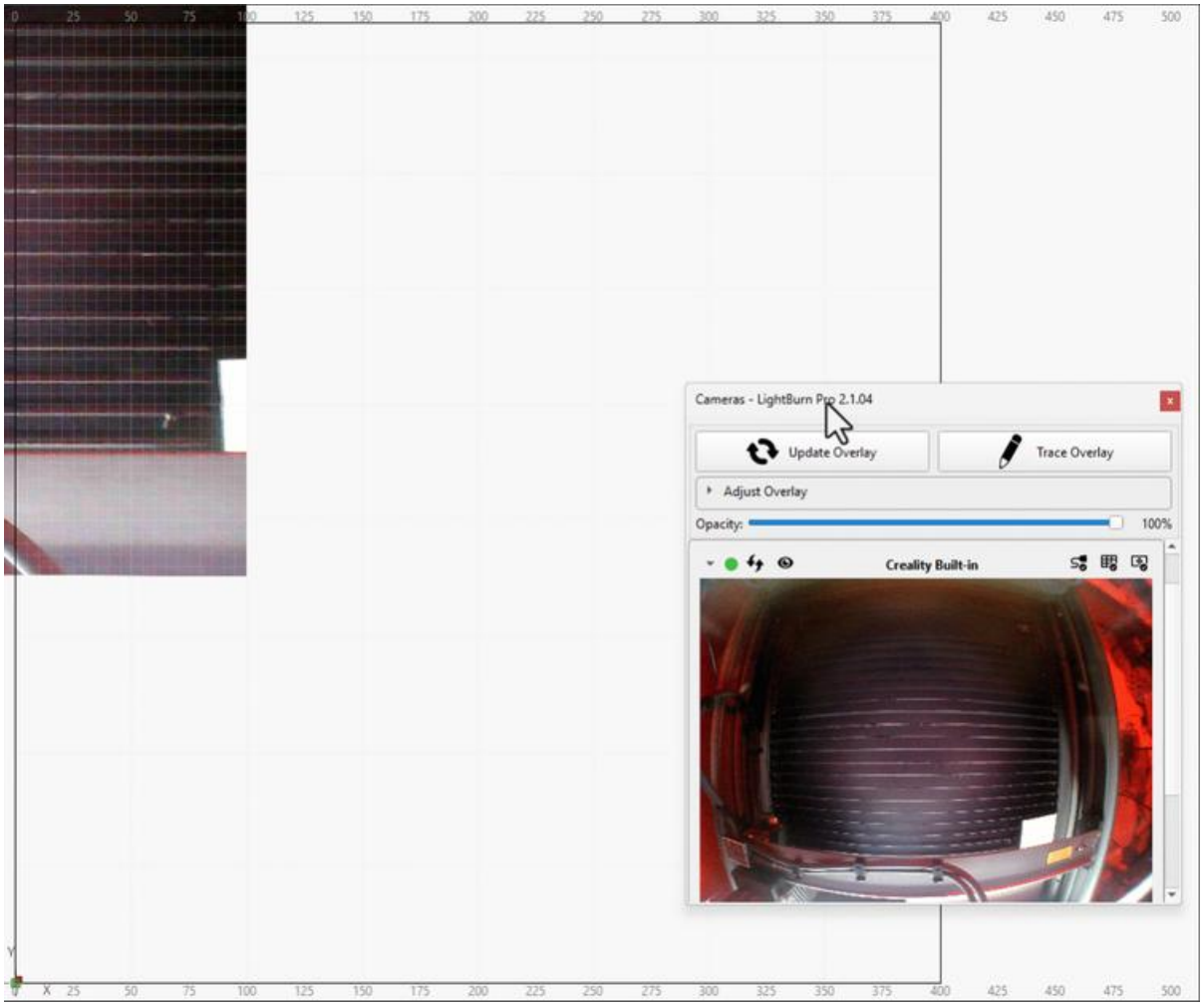
Adjust Overlay

Overlay Width / Height

Adjust the size of the camera overlay.

X Shift / Y Shift

Shift the camera image in the X and/or Y directions, if the entire image is shifted over from where it should be.



Advanced

The **Advanced** section contains less commonly used camera settings.



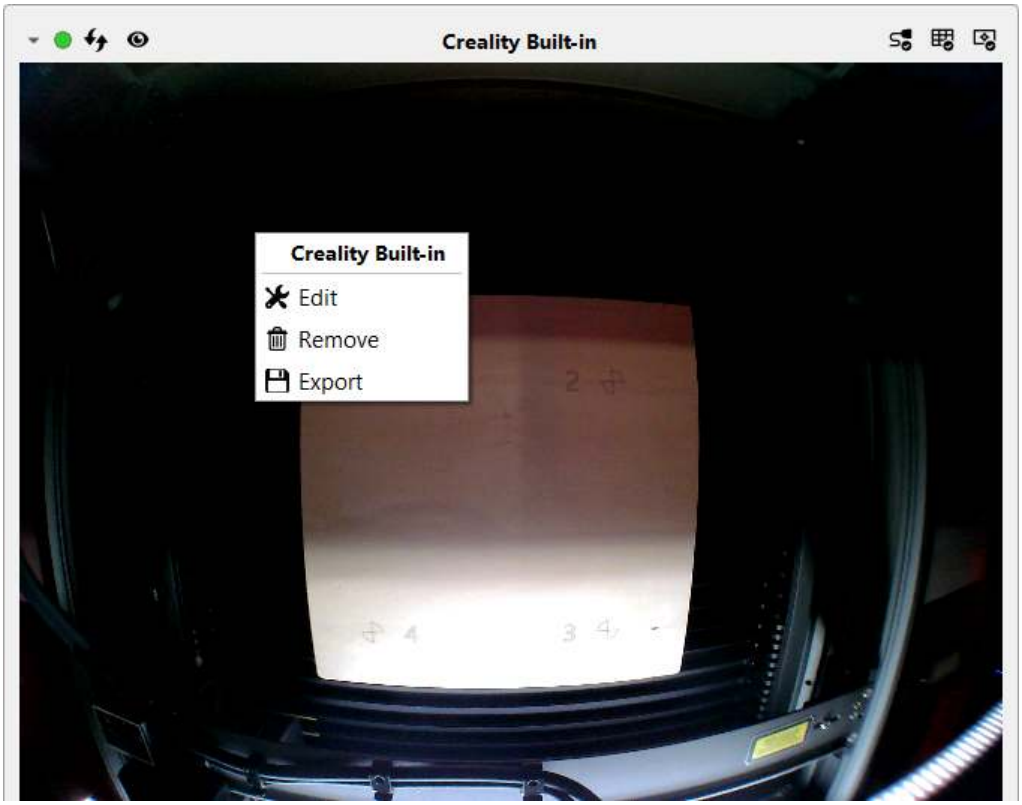
Preview FPS

Use the dropdown to set the framerate of your camera. Higher numbers capture more images per second, giving you a more responsive preview, but also increasing the demand on your computer.

If your camera seems to be slowing down your computer, try lowering the framerate. On the other hand, if you've got a powerful computer and want a more responsive preview, you can try increasing the framerate.

Capture Image

Use either of these buttons to capture the image from your camera and insert it into your LightBurn Workspace. The **Calibrated** button will insert the image after applying your lens calibration, and **Uncalibrated** will ignore your lens calibration.



Add Camera

Click the **Add Camera** button to open the New Camera Wizard and set up a new camera in LightBurn.

Manage Cameras

Click the **Manage Cameras** button to open the Device Cameras window.

Import Camera

Click the **Import Camera** button to import a pre-configured camera with a LightBurn Camera File (`.lbcm`).

Context Menu



Clicking anywhere in the **Cameras / Overlays Window** other than inside a camera preview will open a context/right-click menu.

- **Camera Selection Help** will open the Camera Selection Help window to help you find an appropriate camera.
- **Add Camera** will open the wizard for Adding a Camera.
- **Import Camera** will open a dialog box to Import a Camera

Camera Installation & Focusing

If your laser didn't come with a camera and you're installing one yourself, you'll need to make sure you mount it correctly. If you don't have a camera yet, see Camera Selection for help.

Mounting Overhead Cameras

The best place to mount an overhead camera is directly above the center of the bed of the laser, with the bed completely visible. The best way to do this is usually to mount the camera on the inside of the lid, so the placement is correct when the lid is open.

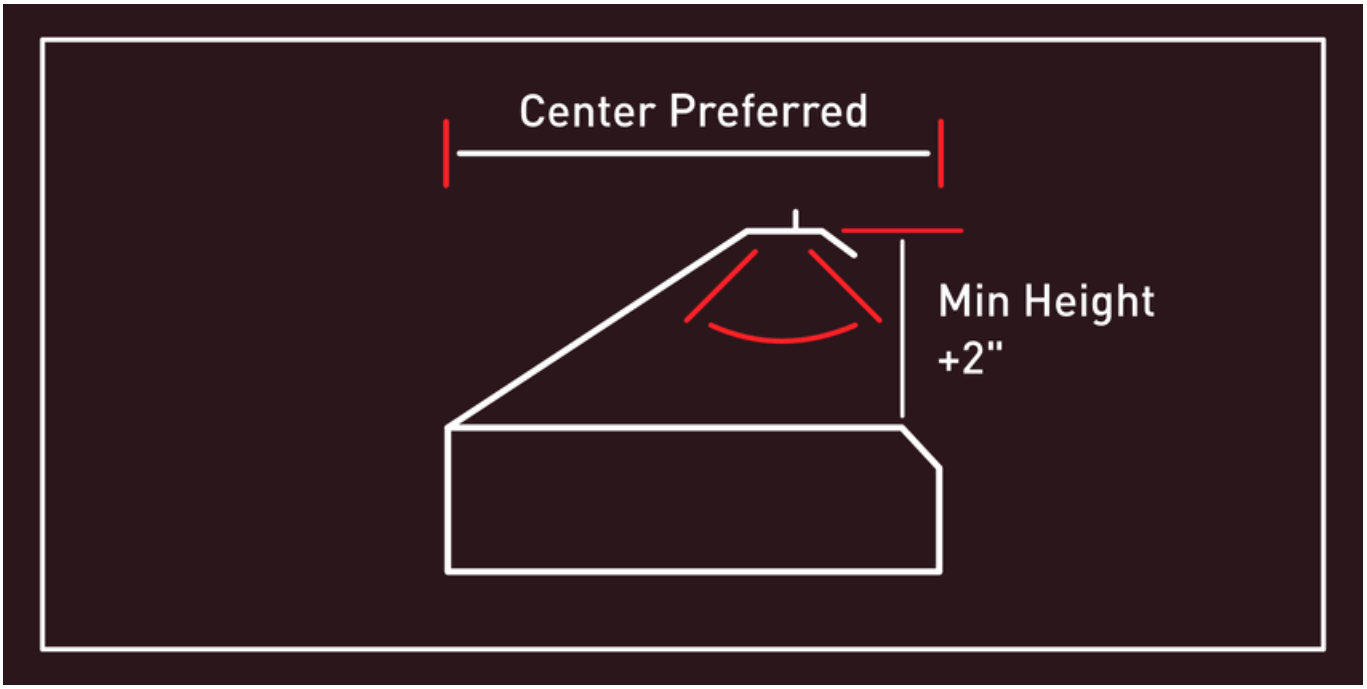
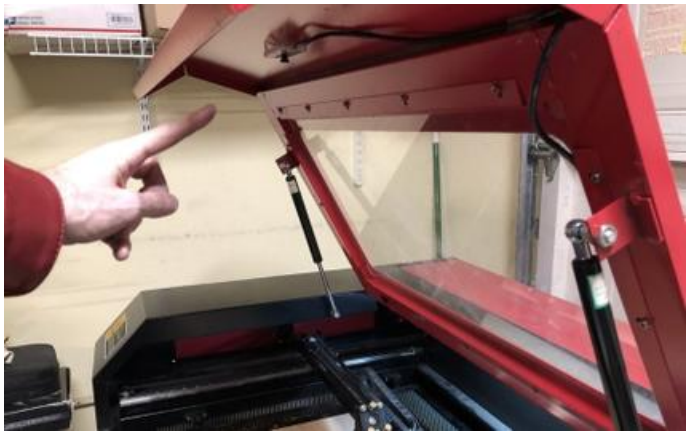


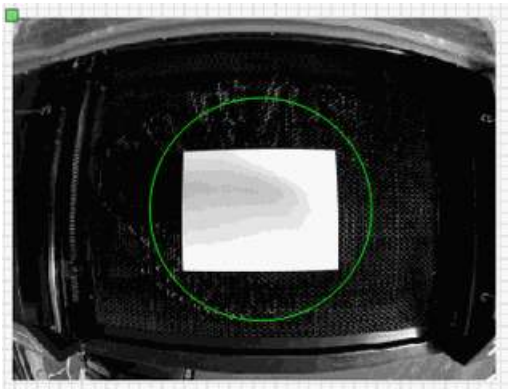
Diagram showing ideal placement



Camera installed in a laser

Mounting tips:

- The camera should be focused as well as possible. Most LightBurn cameras are manually focused by gently twisting the lens.
- The camera must be solidly mounted to be in **exactly** the same position when using it as it was when you performed the alignment process. If you mount it to the lid of your laser, make sure the lid opens to the same place every time. You can use a cord or rod to ensure the lid is always at the same position when opened.
- If your camera is in a mount, make sure it doesn't move within the mount. A small piece of EVA foam or even tissue can hold the camera securely in the mount.
- Mount your camera so it isn't in the path of the laser head, and run the cable so it is not in the path of the beam. Ramming the laser head into your shiny new camera is a great way to increase your blood pressure.
- Focus the camera so as much of the bed is in focus as possible. Some cameras, like the 5mp-60, have a narrow focus depth, and if mounted high, can be tricky to focus. If this happens, focus on a circle that is roughly halfway from the center of the bed to the edge of the image, like this:



If you can't get the entire area in focus, try to focus approximately where the green circle is

Info

USB Cameras should plug in to your computer as directly as possible, without using an excessively long cable. USB 2.0 cables have a maximum length of 16 feet (5 meters). If you absolutely must use a longer cable, a powered USB 2.0 amplifier may help extend the distance, but is not guaranteed.

If USB cable length limitations are a problem, consider the possibility using a network camera, which requires LightBurn 2.1 or newer.

Info

To use LightBurn's camera functionality, you must use Absolute Coords mode, so LightBurn can accurately position the work. If you use an older Trocen controller, they do not allow setting this mode from software, so you must change the working mode through the controller menu.

Mounting Head Cameras

Mounting a camera to the head of the laser can be as simple as using zip ties or a hose clamp to hold the camera in place, or you can use a 3D printer or the laser cutter you already have to create a custom mount. The inexpensive USB endoscope cameras commonly used as head-mounted cameras usually have 5/16" (about 8.1 mm) diameter.

Your camera should be:

- At an appropriate height to focus on the surface of your material.
- Mounted as vertically as you can manage. It's okay if it's not absolute perfect - the Alignment process corrects for your camera being crooked, but the less correction is needed, the better.
- Mounted securely enough not to shift or vibrate when your laser moves.
- Installed in a way that the USB wire doesn't interfere with the laser's movement or get pinched between moving parts.

You can download a simple endoscope camera mount from our resources page, and we have more coming soon.

Read more on selecting a head-mounted camera

Camera Selection

New Camera Options

LightBurn 2.1 and newer allow for a wider range of cameras to be used, including some web-based IP cameras and a wider range of manufacturers' built-in cameras.

IP/Network cameras need to be able to support sending MJPEG or individual frames. RTSP cameras will not work. See Camera Troubleshooting for more information.

Can I Use a Camera I Already Have?

Possibly. Interactions between cameras and computers are complex, and although a wide range of cameras are compatible with LightBurn, we can't guarantee your specific camera will work. If you want to try using your own camera, your best option is a fixed-zoom USB camera that can be detected as a webcam by your computer. If you're using a network camera, look for a non-RTSP IP camera that can send MJPEG or individual frames.

Selecting a Camera

You can add an Overhead camera or a Head camera to LightBurn. Neither type is inherently better or worse, and combining the two can be particularly effective, especially on larger lasers, where it may be helpful to use an overhead camera to capture the entire bed, then capture specific areas of interest with a head camera.

Overhead Cameras	Head Cameras
Mount to a fixed point, such as the top or lid of your machine, and don't move around.	Mount to the head of your laser, and move around inside your laser.
Captures a view of the entire laser bed.	Captures a view of a small section of the laser bed around the head of your laser.
Uses a lens with a wide field of view to capture a large area relatively close to the camera. The wide field of view requires a lot of correction for distortion, especially around the edges of large machines.	Uses a lens with a narrower field of view. Requires less correction for distortion and gives more accurate results around the edges of the laser bed.
To get good detail on larger machines, requires a relatively high camera resolution for a webcam.	Low resolution cameras are fine, because each pixel only represents a small area.
Within reason, the size and weight of the camera isn't important since it isn't moving around. Something around the size of a typical webcam is completely fine.	Head cameras need to be small and light. USB endoscope cameras are a popular choice for this.
Many lasers come with overhead cameras pre-installed.	Head cameras are rarely pre-installed, although some lasers do include them.
Get help selecting an Overhead Camera ↓	Get help selecting a Head Camera ↓

Selecting a Head Camera

With growing interest in Head Cameras for compatibility with our new Camera Print & Cut feature, we've had more people asking for guidance on what to purchase.

If your laser didn't come with a head-mounted camera pre-installed, we recommend an inexpensive **USB Endoscope Camera** with a long, flexible USB cable. These cameras are also sometimes listed as a "USB Inspection Camera". Avoid cameras with built-in screens or rigid or semi-rigid cables that will interfere with your laser's motion. As of October 2026, these cameras are typicallly available for about $15-30 or €20-30 from common online retailers.



Example image of the type of endoscope camera to look for. This one has a semi-rigid cable, but you should look for one with a flexible cable.

If you want to explore other options, any small, light camera you can mount securely to your laser head and connect to over USB or WiFi should work. Keep in mind that WiFi cameras that use RTSP will not work - IP cameras need to output individual frames or an MJPEG feed.

For more information on mounting your new camera, see Camera Installation - Head Cameras.

Selecting an Overhead Camera

To select a camera, you'll need to know two things:

1. The size of your laser's work area, in millimeters.
2. The distance between the height from which you want to mount the camera and the surface of your laser's work area.

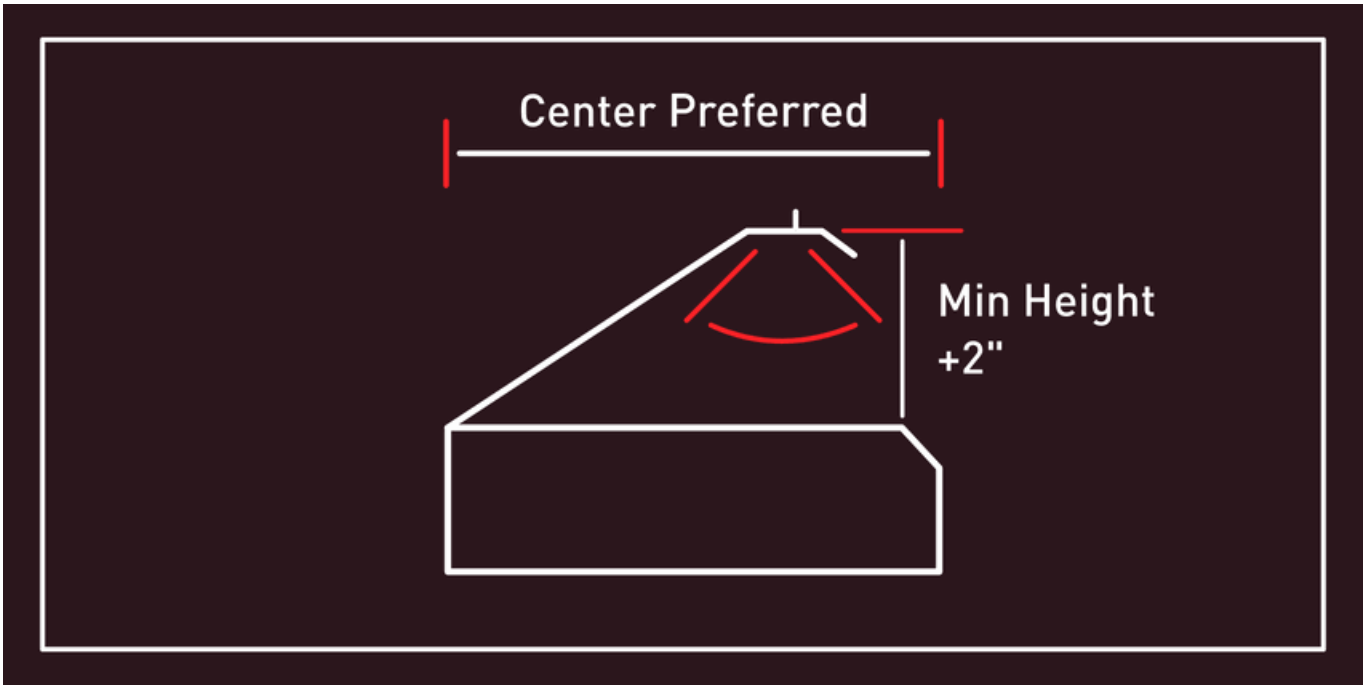


Diagram showing ideal mounting location

If that seems like a lot, don't worry - we've got tools to make it easy!

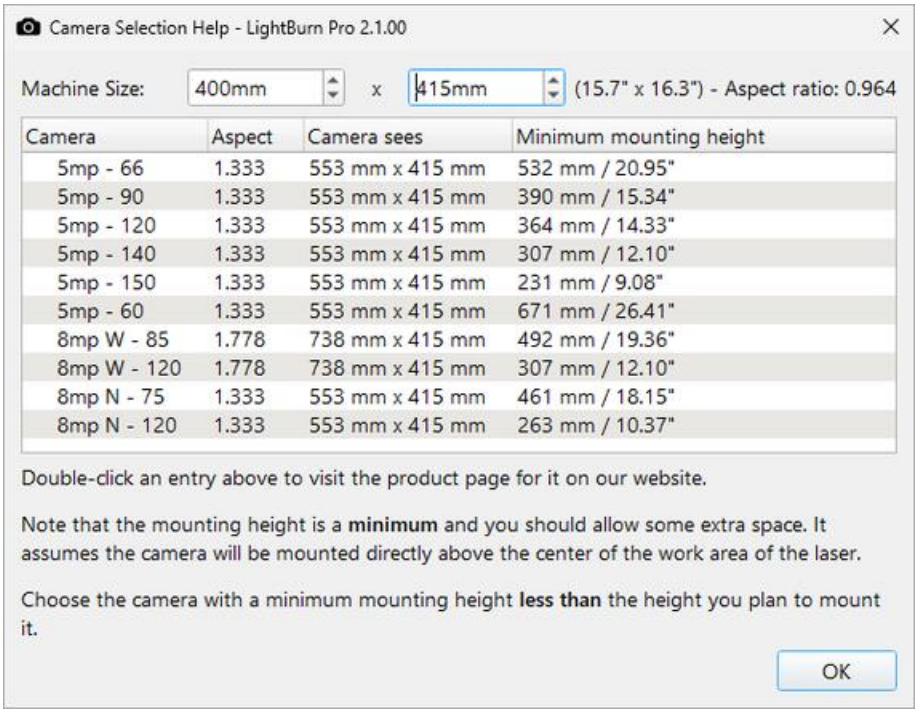
Camera Selection Help Window

The **Camera Selection Help** window helps you select an appropriate camera for your laser. The information provided is specifically for the USB cameras sold on LightBurn's Website, which are known to work well with LightBurn and have calibration presets built into the software for easier Lens Calibration.

Go to Help → **Camera Selection Help** to open the window.

To select a camera:

1. Enter the machine size into the **Camera Selection Help** window.
2. Look through the list to find a camera with a minimum mounting height about 2 inches less than the height you intend to mount at. This leaves room for adjustment.
3. Double-click an entry in the list to visit the product page to purchase it.



Info

We are phasing out sales of official LightBurn cameras. If the camera suggested by our built-in helper is no longer available, try searching for a third party camera with the same lens angle, or as close as possible. See the calculator below for help verifying that a given camera will work for your laser.

Sourcing a Third Party Camera

With sales of official LightBurn cameras winding down, we have resources here to help you find a suitable third-party camera for your laser, including a camera lens selection calculator.

Camera Specs

LightBurn is compatible with a broad range of third party cameras. Here are some guidelines on technical requirements:

- Fixed focus
- 12 FPS or higher is sufficient
- 8MP to 12MP resolution is ideal
  - >16MP hits USB 2.0 bandwith limits
  - >20MP hits defraction limits
- Global shutter camera recommended

Lens Selection Calculator

Use the **Browse Lens Options** tab below to enter your laser's work area dimensions and intended camera mounting height, then see recommended camera options listed by lens angle and aspect ratio.

If you cannot source a camera with the exact specifications listed in row labeled "Best fit", try sourcing a similar camera, then using the **Verify Camera** tab to make sure it will work with your laser. Enter a camera's known resolution and lens angle, along with your laser's work area dimensions and intended mounting height, then check if the camera will be capable of capturing your laser's entire bed.

Identifying Lens Angle and Resolution in Product Listings

- Lens angles are typically listed in degrees representing diagonal field of view. You may see the terms "Optical Lens" or "FOV".
- Resolution is typically expressed in `*####x####` format, for example `1920x1080`. You may also see an aspect ratio written in `#:#` format, for example `16:9`.

Units:
Laser work area

Width
mm
Height
mm
Mounting height

The height above your laser's work area at which you plan to mount your camera
mm
The ideal lens has a minimum mounting height about 50 mm (2 in) below your intended height.

Lens angles listed here are **diagonal FOV** — the most common convention used in camera specs. If your camera's spec sheet lists **HFOV** or **VFOV** instead, use the Verify my camera tab and select the matching measurement axis.
Camera sensor

Width
px
Height
px
Camera FOV spec

FOV measurement axis

Most specs list diagonal FOV. Only switch if your spec sheet explicitly says HFOV or VFOV.

Diagonal FOV
°

Mounting height

The height above your laser's work area at which you plan to mount your camera
mm

Troubleshooting

Some computers may have difficulty connecting with older or less expensive USB cameras because of incompatibilities between newer USB 3.0 ports and older USB 2.0 cameras. This is more common with USB C ports and newer computers such as M1 or newer Macs. If you have this problem, try using a USB 2.0 hub or adapter between the camera and the computer.

See Camera Troubleshooting for more information.

Related Topics

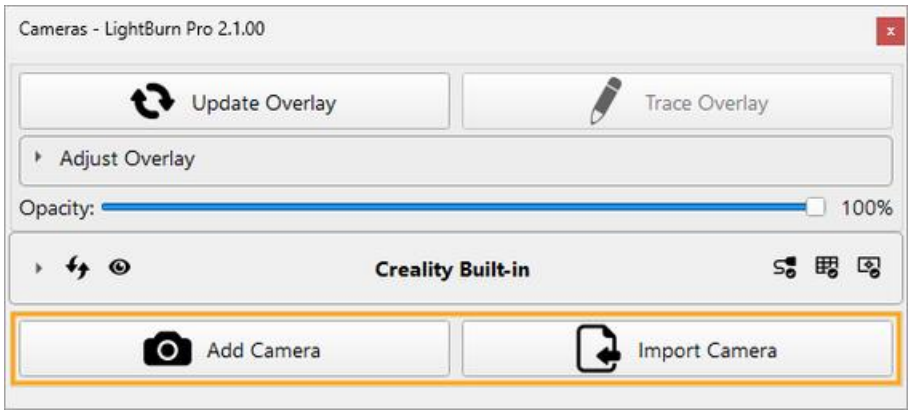
- Camera Installation and Focusing
- Calibrate Camera Lens
- Calibrate Camera Alignment
- Cameras / Overlays Window

Add a Camera

Before you can use a camera with LightBurn, you'll need to add it. If you've already configured your camera on an older version or a different computer, you can export a LightBurn Camera File (`.lbcm`) and import it. Otherwise, you'll need to create a new camera.

Quick Start

If you want to dive right in with adding a camera, you can use the **Add Camera** or **Import Camera** buttons in the Cameras / Overlays Window, and follow along with the onscreen directions.



Add Camera

Use this if you don't have an existing `.lbcm` file to import.

1. Select your camera type.
2. Select your camera source or preset.
3. Follow the on-screen directions to adjust settings and calibrate and align your camera, if needed.
4. Click **Finish** to save your settings.

Import Camera

Use this if you're importing an existing LightBurn Camera (`.lbcm`) file.

1. In the dialog box that opens, select your `.lbcm` file and click **Open**. If you are importing from an older version and need help with the warning, jump to the full directions.
2. Follow the on-screen directions to adjust settings and calibrate and align your camera, if needed.
3. Click **Finish** to save your settings.

Skip to Lens Calibration

Need more help? Continue for the full directions.

Add a New Camera

Import a Camera

Create a New Camera

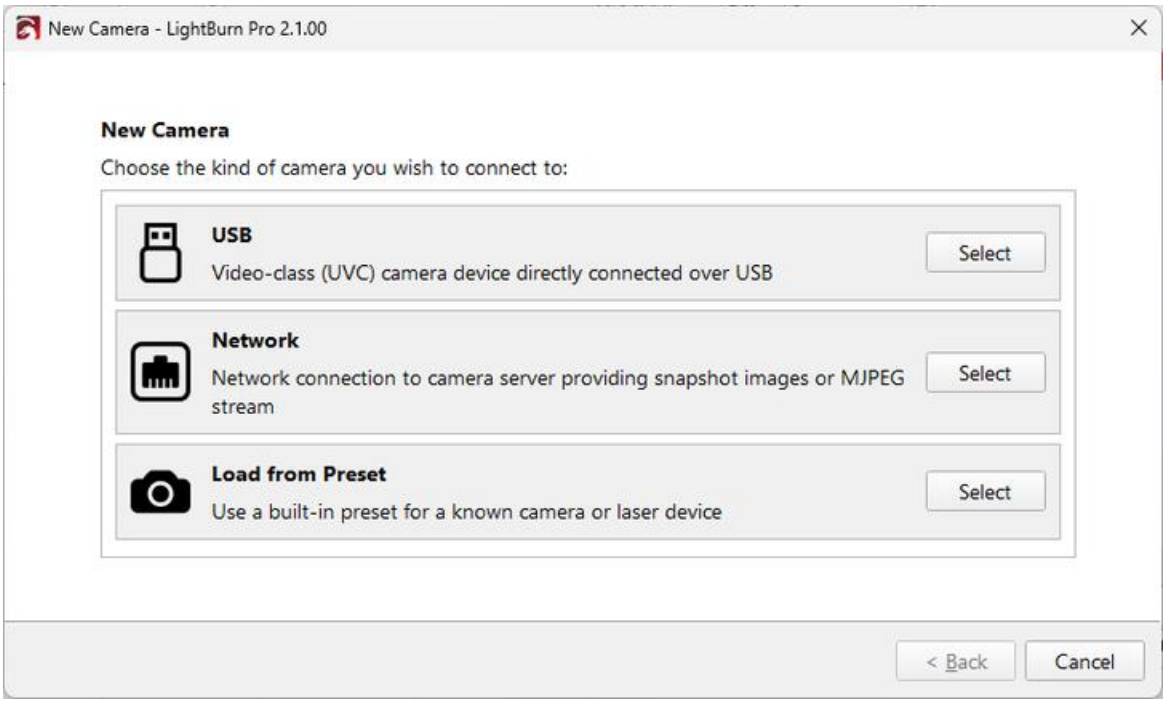
Click the **Add Camera** button in the Cameras / Overlays window to start the process of adding a new camera to LightBurn.

> **Tip**
>
> You can also right-click in the **Cameras / Overlays** window or go to **Laser Tools → Manage Cameras → Add New**.

1. Select Camera Type

LightBurn 2.2 offers three options for camera type. Select the one that matches your setup.

- **USB** for typical USB cameras with a direct wired USB connection to your computer.
- **Network** for cameras you connect to over a wired or wireless network.
- **Load from Preset** to load a preset, if one exists for your device. Currently, LightBurn has presets for the following camera setups:
  - Creality Falcon A1 Pro
  - Thunder Laser Vision (both Overview and Head-Mounted)
  - WeCreat
  - xTool P2
  - xTool P2S



2. Find Camera Source

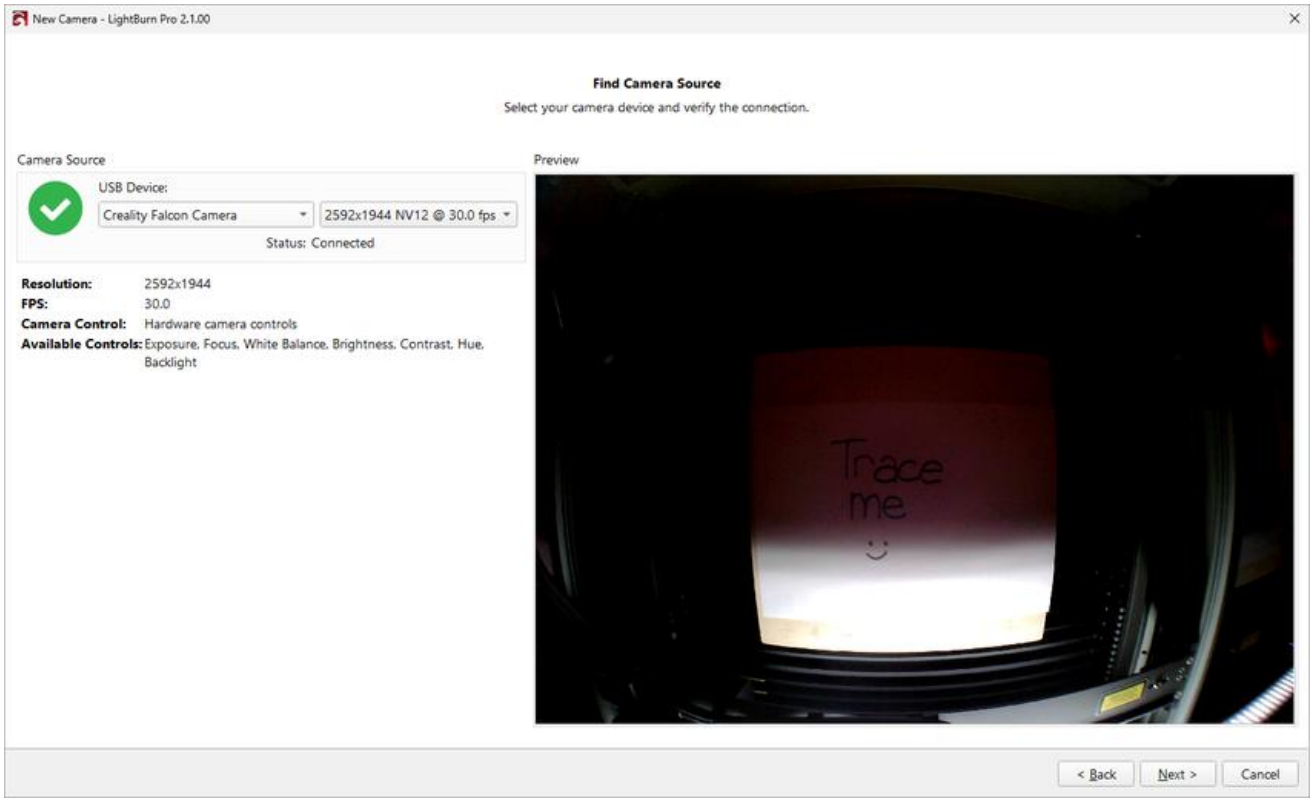
For USB or network cameras, you'll start by selecting your camera, or telling LightBurn how to find it. This is slightly different depending on the type of camera you're using.

If you're loading a preset, the process is similar but looks a little different - after selecting camera type you'll go to the Edit Camera screen, which shows you the camera name, type, source, calibration, and alignment all in one place.

After selecting or connecting to a camera, the left side of the window will show additional information about the camera, and the right side of the screen shows you a preview of the camera feed.

Click the appropriate tab below to view more detailed directions for connecting to USB or network cameras.

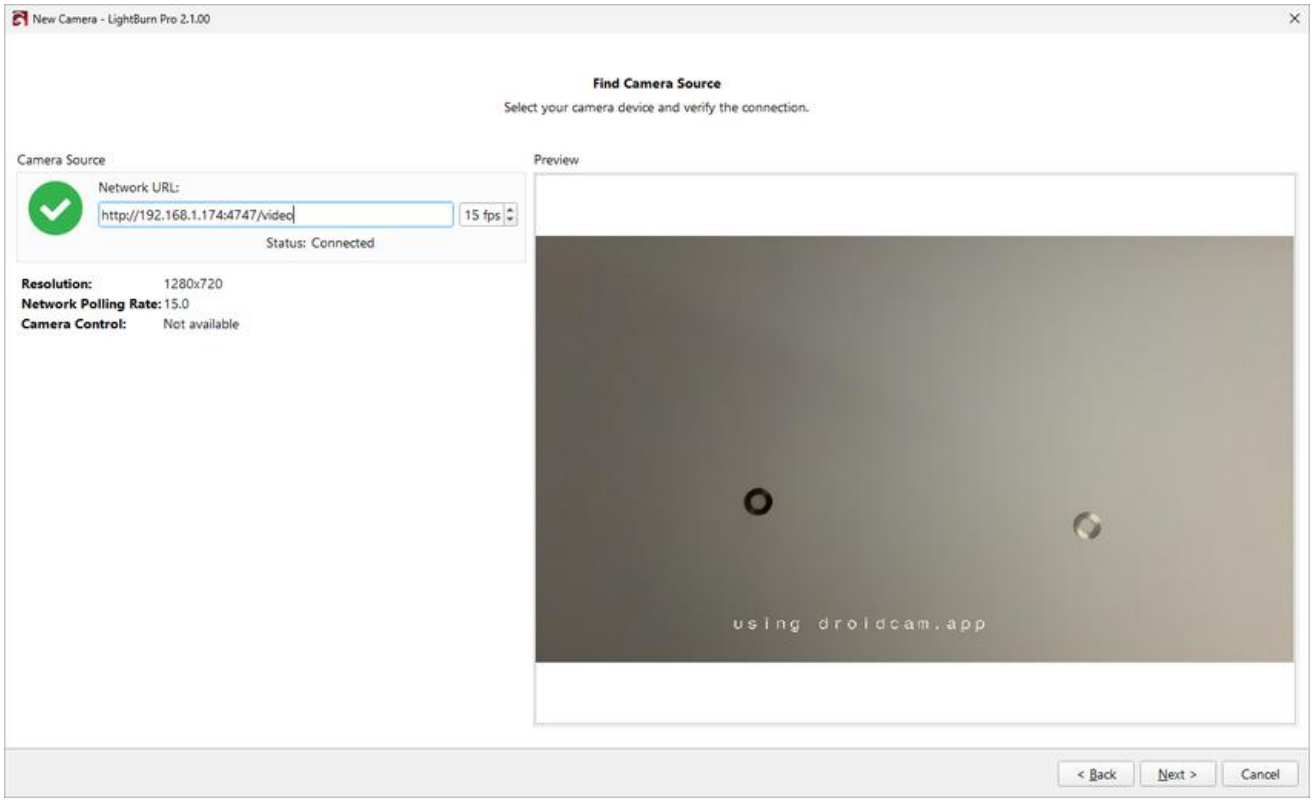
USB Camera



To select your USB camera, click the dropdown under **USB Device** and select the camera you want to use. You may have additional options for framerate or resolution in the second dropdown. Higher resolutions will give you more detailed images, and higher framerates will give you more responsive previews, but at the cost of requiring a more powerful computer.

If you don't see the camera you want to use, make sure it's connected. Check our Camera Troubleshooting or email support for more help.

Network Camera



To connect to your network camera, type the URL for the camera feed into the box under **Network URL** and set the framerate you want to use. Higher framerates will give you more responsive previews, but at the cost of requiring a more powerful computer.

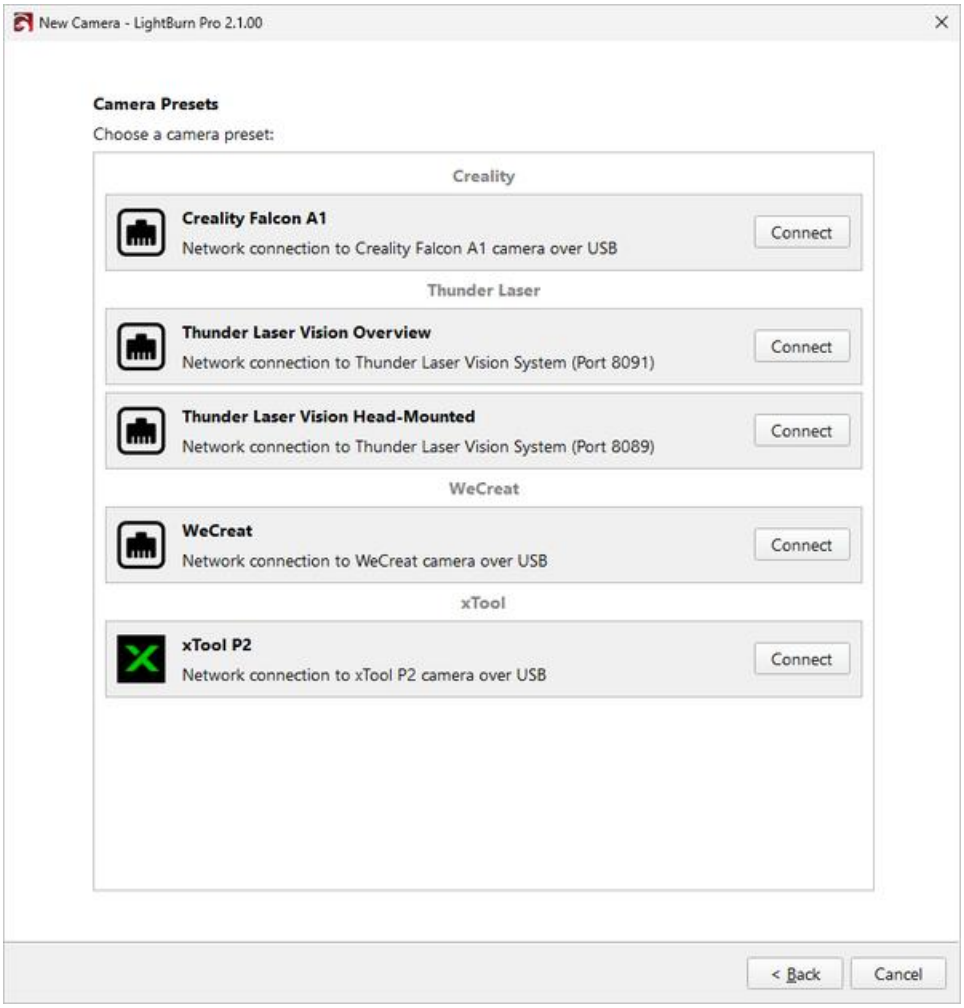
If you can't connect to your camera, check that you've entered the URL correctly, and that your video feed is accessible. Check our Camera Troubleshooting for additional help with getting the correct URL or email support for help.

Load From Preset

Select the camera that matches your hardware and click **Connect** to continue to the Edit Camera screen, where you'll make sure the computer can find the camera, preview the camera feed, and manage lens calibraton and workspace alignment.

Camera presets currently support:

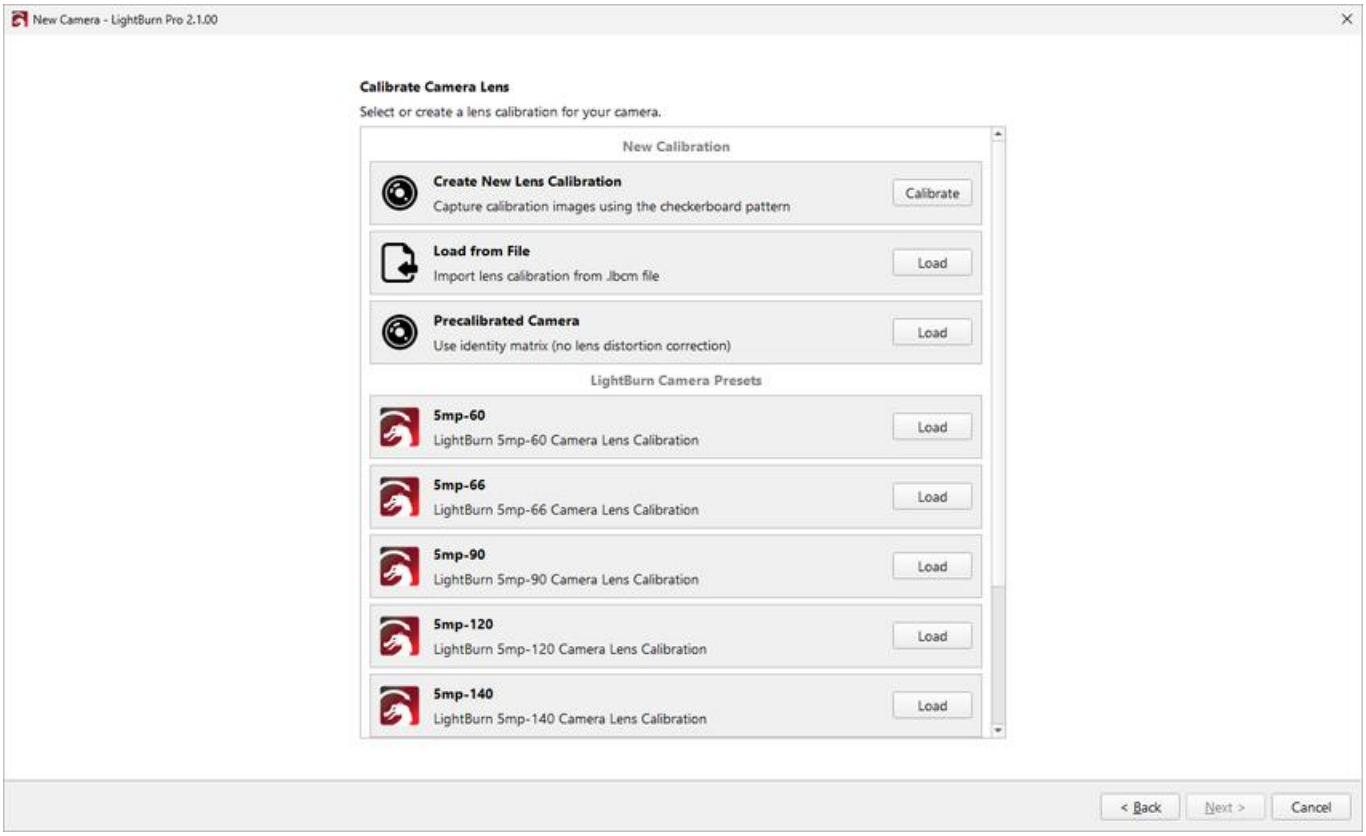
- Creality Falcon A1 Pro
- Thunder Laser Vision
- WeCreat
- xTool P2



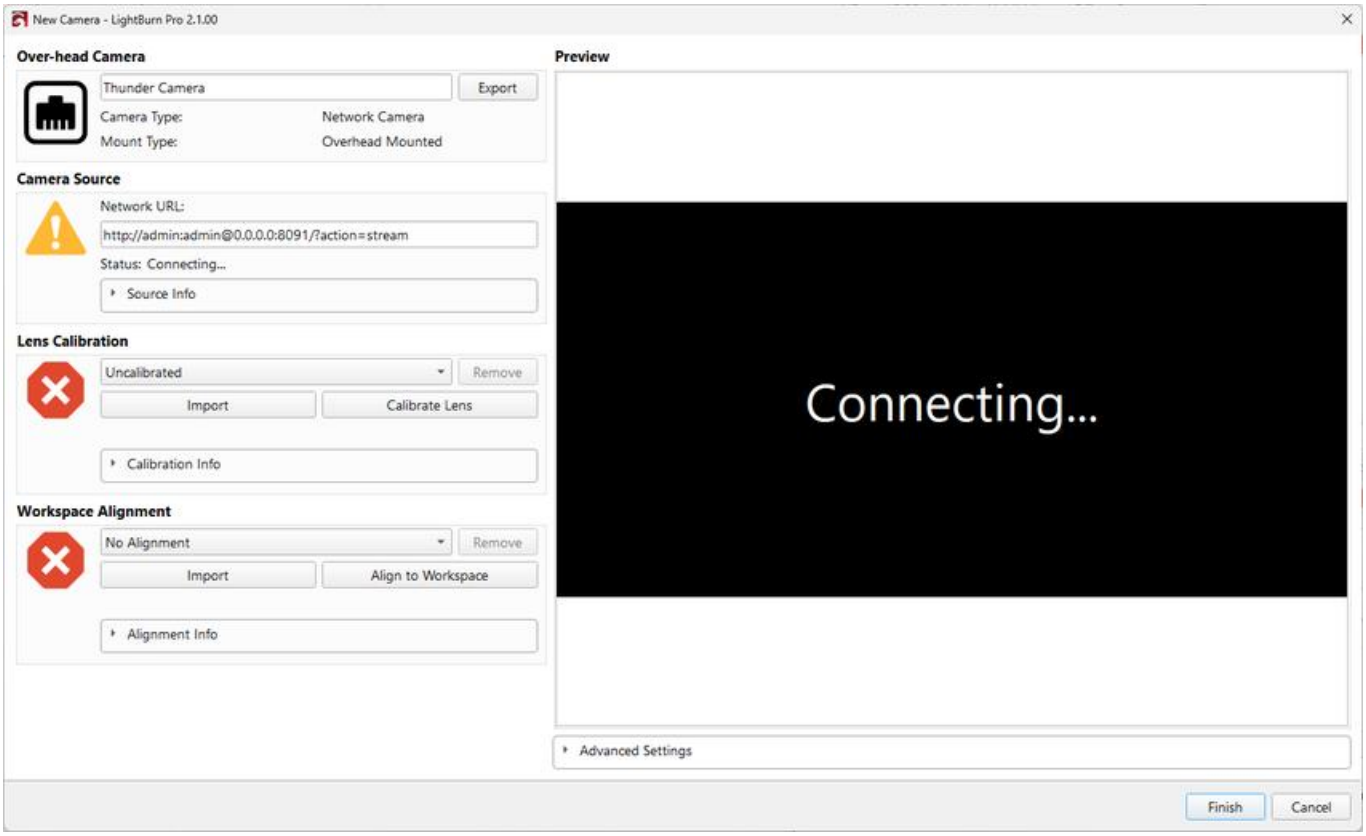
3. Calibrate and Align Camera Lens

After selecting your camera type and finding your camera source, LightBurn will walk you through the steps to calibrate your camera lens and align your camera to your workspace.

This will look a little different depending on how you added your camera:



After adding a USB or Network camera, LightBurn takes you to the **Calibrate Camera Lens** screen.



After loading a Camera Preset, LightBurn takes you to the **Edit Camera** screen, where you can manage lens calibration and workspace alignment.

There are a few paths you can take for lens calibration.

- **Create New Lens Calibration** will walk you through calibrating your camera lens yourself.
- **Load From File** will import a camera calibration from a `.lbcm` file, which you can create by exporting the camera settings from an existing camera setup.
- **Precalibrated Camera** is for cameras that are already calibrated and don't require additional distortion correction.
- **LightBurn Camera Presets** will automatically apply the appropriate settings for cameras purchased directly from LightBurn.

Regardless of which one you choose, you can follow the on-screen directions or jump ahead to our Camera Lens Calibration page.

Skip to Lens Calibration

Import an Existing Camera

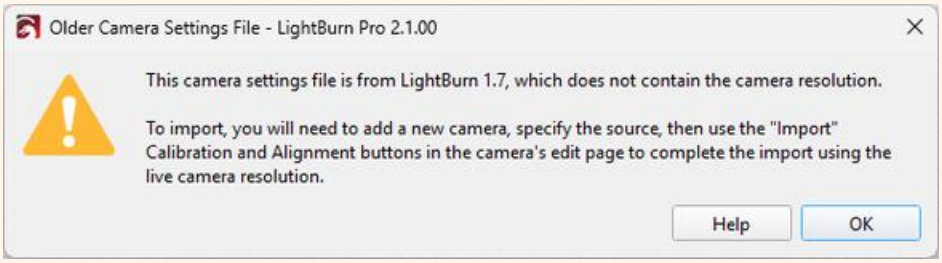
If you've already set up your camera before and exported your configuration, then congratulations! You get to do this on easy mode.

Click the **Import Camera** button to get started.

Are you importing a camera from an older version of LightBurn?

If you're importing a camera into LightBurn 2.1 from LightBurn 1.7 or older, you'll see the following **Older Camera Settings File** warning.

> This Camera settings file is from LightBurn 1.7, which does not contain the camera resolution.
>
> To import, you will need to add a new camera, specify the source, then use the "Import" Calibration and Alignment buttons in the camera's edit page to complete the import using the live camera resolution.



You can still import your camera, but there is an extra step to make it work:

1. Create the camera, following the directions in Step 1 and Step 2 of Create a New Camera. Don't worry about Calibration and Alignment, you'll import those.
2. Once you've added the camera and connected to it, you'll be able to import the Lens Calibration and Camera Alignment from your camera settings file, as described in Import Calibration and Import Alignment.

Why did this happen?

You're seeing this message because the older camera settings file you're using doesn't have information about the camera resolution, so LightBurn can't import it in one step. Once you create the new camera in LightBurn 2.1 and connect to it, LightBurn has the information it needs to be able to import the Lens Calibration and Workspace Alignment.

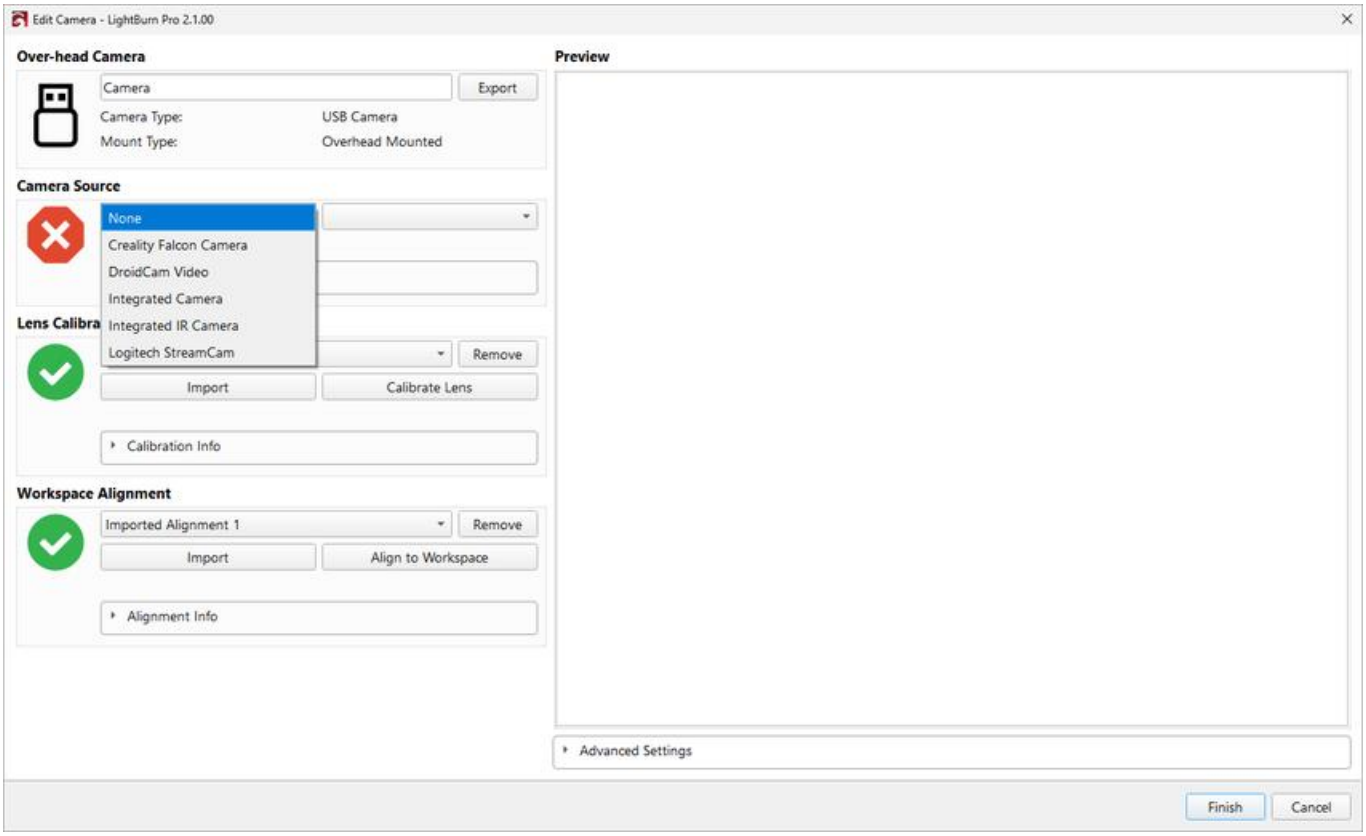
1. Select Your LightBurn Camera File

A LightBurn Camera File has the `.lbcm` extension and contains information about your camera.

In the file picker that opens when you click **Import Camera**, find and select your `.lbcm` file.

2. Edit Camera

After importing your camera, you'll be taken to the Edit Camera window. You'll probably want to change the name of the camera in the text box to something a little more descriptive than "Camera", especially if you'll be using more than one.



Because you've imported your existing camera data, **Lens Calibration** and **Workspace Alignment** should both have green checkmarks and show the imported calibration data. If you got the **Old Camera Settings File** warning described above, you'll need to select your camera source from the dropdown under **Camera Source**. Otherwise, that should also have a green check mark and show the camera name.

The screenshot here shows the Edit Camera window after getting the Old Camera Settings warning - to fix it, you would select the appropriate camera option under **Camera Source**.

3. You're (Probably) Done

You can certainly redo your Lens Calibration and/or Workspace Alignment if you want, for instance if you weren't happy with your results previously, but your camera should be set up and ready to go as soon as you click the **Finish** button.

Edit Camera

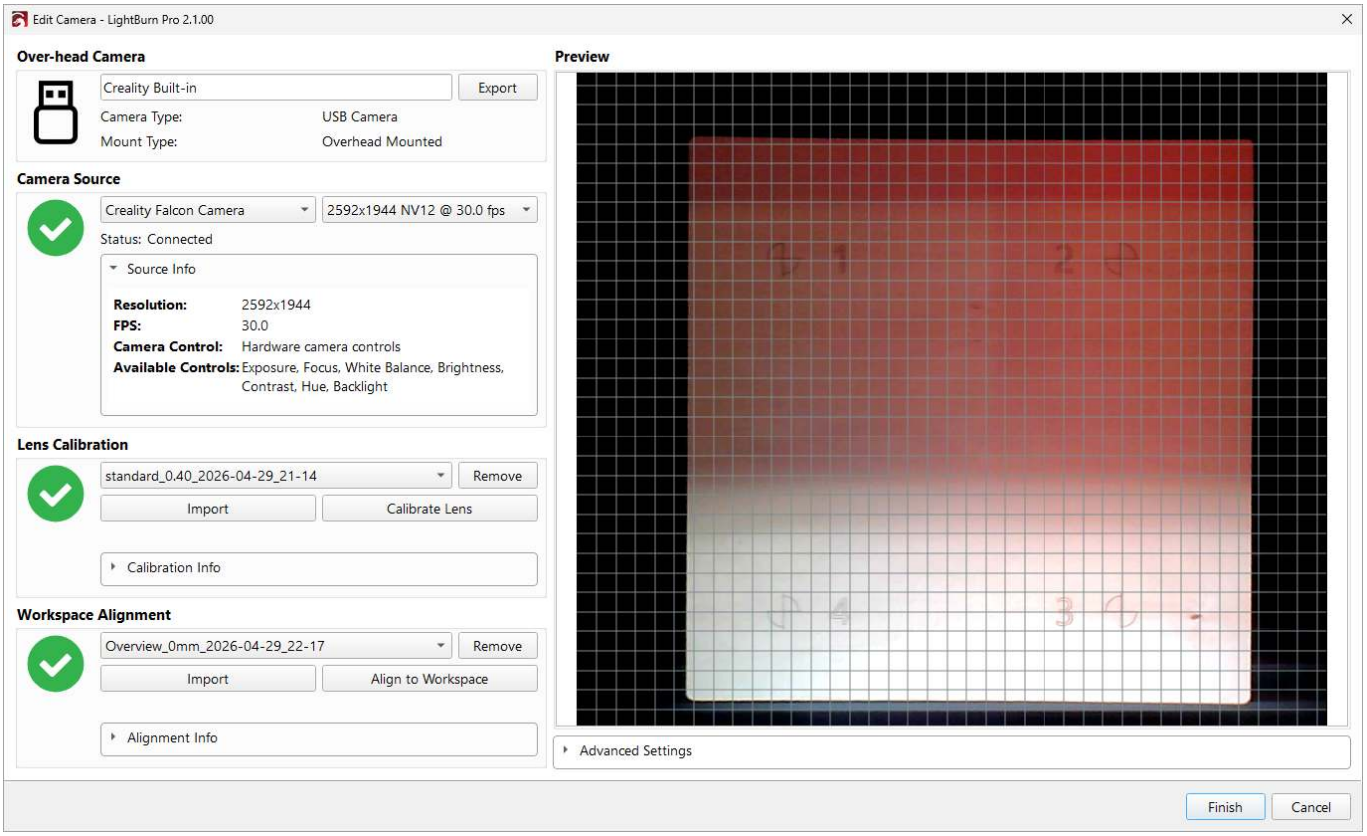
After adding a camera to LightBurn, you can manage individual cameras from the **Edit Camera** window. While the Cameras / Overlays Window gives you an overview of all of your connected cameras and their image settings, the Edit Camera window allows you to manage individual cameras in more detail.

Opening the Edit Camera Window

There are a few ways to access this window:

- The **Edit Camera** window will automatically open when you import a camera from a **LightBurn Camera File** (`.lbcm`).
- In the Laser Tools menu, select **Manage Cameras**, then select the camera you want to edit and click the **Edit** button.
- In the Cameras / Overlays Window, click the **Manage Cameras** button.

Click the part of the window you have questions about to go directly to information about it, or scroll down for more information.



Edit Camera Settings and Options

Camera Overview

The first section on the left has an overview of the camera name, type, and mounting method (overhead or head-mounted).

Camera Name

Change the name of the camera here to change how it's displayed in the Cameras / Overlays Window.

Export

Click the **Export** button to export a **LightBurn Camera File** (`.lbcm`). You can use this to back up your camera settings or copy them to another LightBurn installation.

Camera Source

The Camera Source section lets you manage where the camera feed comes from.

Source

Use the source dropdown (for USB cameras) or the text entry box (for network/IP cameras) to set where LightBurn pulls the camera feed from.

Even if the camera itself hasn't changed, you may need to update this if:

- You've disconnected a USB camera and reconnected it, especially to a different USB port.
- The URL or IP address for a network camera has changed.

Resolution/Frame Rate

Your camera may offer multiple options for resolution and framerate. A higher resolution gives you a more detailed image, and a higher frame rate means your camera is more responsive, but also puts more demand on your computer.

Connection and Source Info

This section shows whether your camera is connected and provides more information about the camera itself.

The **Source Info** box can be expanded by clicking on it, and will summarize:

- Camera resolution
- Camera framerate
- Available camera controls

Lens Calibration

Use the Lens Calibration section to create and manage your Camera Lens Calibration

Lens Calibration Dropdown

Use this dropdown to choose between your current lens calibration model and an uncalibrated version of your camera feed.

Remove

Delete your current lens calibration from LightBurn.

Import

Import a lens calibration from a **LightBurn Camera File** (`.lbcm`). This is the same file you would use to import a camera, but if you import it here, it'll only update the Lens Calibration. LightBurn will warn you if your Lens Calibration is not the same one you used when completing your Camera Workspace Alignment.

Calibrate Lens

Click the **Calibrate Lens** button to open the Lens Calibration wizard and start the calibration process.

Calibration Info

The **Calibration Info** box can be expanded by clicking on it, and will summarize:

- Calibration Name
- Camera Name
- Calibration Type
- Original Camera Resolution

Workspace Alignment

Use the Workspace Alignment section to create and manage your Camera Workspace Alignment.

Workspace Alignment Dropdown

Use this dropdown to choose between your current workspace alignment and an unaligned version of your camera feed.

Remove

Delete your current workspace alignment from LightBurn.

Import

Import a workspace alignment from a **LightBurn Camera File** (`.lbcm`). This is the same file you would use to import a camera, but if you import it here, it'll only update the workspace alignment. LightBurn will warn you if your workspace alignment is not the same one you used when completing your Camera Lens Calibration.

Align to Workspace

Click the **Align to Workspace** button to open the Alignment wizard and start the alignment process.

Alignment Info

The **Alignment Info** box can be expanded by clicking on it, and will summarize:

- Alignment Name
- Calibration Name (for the Lens Calibration model used when performing the workspace alignment process)
- Mount Type (overhead or head-mounted)
- Material Height
- Image DPM (dots per meter)
- Original Resolution

Camera Preview

The **Camera Preview** section shows you the camera view based on your current settings.

Advanced Settings

The **Advanced Settings** section includes settings most user won't need to touch, but they're available if you need them.

USB Device Identifier

You can select between using the device ID or device name as an identifier for a USB camera.

Nr Warm-up Frames

You can select a number of warm-up frames, which will be ignored when the camera starts up. The default of 3 frames will work well for most people.

Reframe Calibration Field of View:

> **Labs Feature**
>
> Labs features are new tools in development that we couldn’t wait to get into the hands of our users. Please experiment with caution, and report any unexpected results on our forum.

Interpolate Alignment Height

> **Labs Feature**
>
> Labs features are new tools in development that we couldn’t wait to get into the hands of our users. Please experiment with caution, and report any unexpected results on our forum.

With **Interpolate Alignment Height** enabled, if you've aligned your camera at different **Material Heights** and/or **Support Heights**, LightBurn will automatically select the alignment that matches your current Material Height, if it exists. If there isn't already an alignment for the current Material Height, LightBurn will calculate the expected alignment for the current Material Height based on your existing alignments.

Camera Lens Calibration

> **Significant Changes in LightBurn 2.2**
>
> LightBurn 2.2 includes significant UI improvements to the camera interface as well as updates to make it easier to calibrate your camera. If you are on an older version of LightBurn, use the dropdown at the top of the page to select the appropriate version of the camera pages.

Before you can use your camera, you'll need to calibrate it. All camera lenses distort the camera view, and the distorted view won't show your workspace accurately. LightBurn's **Camera Calibration Wizard** will walk you through the process of capturing a series of photos so LightBurn can calculate and correct for this distortion.

Camera Lens Calibration only needs to be done once per camera - moving the camera doesn't require a new calibration. Camera Alignment, which compensates for the angle and location your camera is mounted at, is a separate process that will need to be redone every time the camera is moved.

> **Info**
>
> The video below is for an older version of LightBurn, but still has a lot of good information about Camera Calibration, so we're still sharing it here. Please understand that it won't be an exact match for what you see on your screen in LightBurn 2.2.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=dSBjoaAu5zI

> **What if I use more than one camera?**
>
> If you have two or more overhead cameras in your laser to better cover a large bed, calibrate each camera separately.

Did you already go through the Camera Lens Calibration process and export a **LightBurn Camera File**? If so, you can import the file and reuse that calibration.

Jump to importing a Lens Calibration

Jump to the Lens Calibration Pattern

Open The Camera Calibration Wizard

There are a few ways to open the Camera Lens Calibration Wizard outside of creating a new camera:

- Click the **Calibration** button for the camera you want to align in the Cameras / Overlays Window.
- Click the **Calibrate Lens** button in the Edit Camera window.

Before You Begin

1. Make sure you have the AprilTag calibration pattern ready. You can download and print it, engrave it onto a flat, rigid sheet of material, or load it in full screen on a phone or tablet (set the screen to always on).
2. Make sure the camera is connected and available in the Cameras / Overlays window.
3. Make sure the bed of your laser has good, consistent lighting and the camera is in focus. If you're using an official LightBurn camera, most can be manually focused by gently twisting their lenses.
4. Make sure your camera is mounted in a way that holds it perfectly still.

> **Camera Placement**
>
> You do not need to have your camera mounted in the machine as long as it's perfectly still. You will need to have it mounted in the correct placement for the Alignment process, though, so it may make sense to make sure it's installed now.

Preparing the Calibration Pattern

In order to work properly, the calibration pattern must be:

- Undistorted
- Perfectly flat
- Clear with good contrast.
- An appropriate size

If you print the pattern, mount it to something stiff and flat, such as cardboard, foam board, wood, or a clipboard. You can also display it on a phone or tablet screen. If you want to do this, you can scan the QR code in the **Calibration Wizard** to come back to this page and

Calibration Pattern Sizing

Because of the wide range of cameras available, there's no one right size that will work well for everybody for the calibration pattern. Instead, here are some guidelines to help you find an appropriate size:

- You should be able to fit the entire pattern into the camera frame comfortably. For cameras with a narrow field of view, such as the endoscope cameras often used as Head Cameras, this might require making your pattern very small. You can scale the PDF when printing it, or consider displaying it on a phone screen.
- While big laser beds need larger calibration patterns, don't make the pattern unnecessarily large. You should be able to hold or place it comfortably in front of the camera. Typically, you shouldn't need to print larger than an A4 or US Letter size page. A tablet screen is good for large laser beds as well.
- The camera should be able to focus clearly on the pattern when it's close enough to fill most of the camera's field of view.

Download the Calibration Pattern

To quickly display the calibration pattern on a tablet or other device, you can scan this QR code to go straight to the PDF version of the calibration pattern. Otherwise, you can open the pattern as a PDF or download it in LBRN2 format with the buttons below.

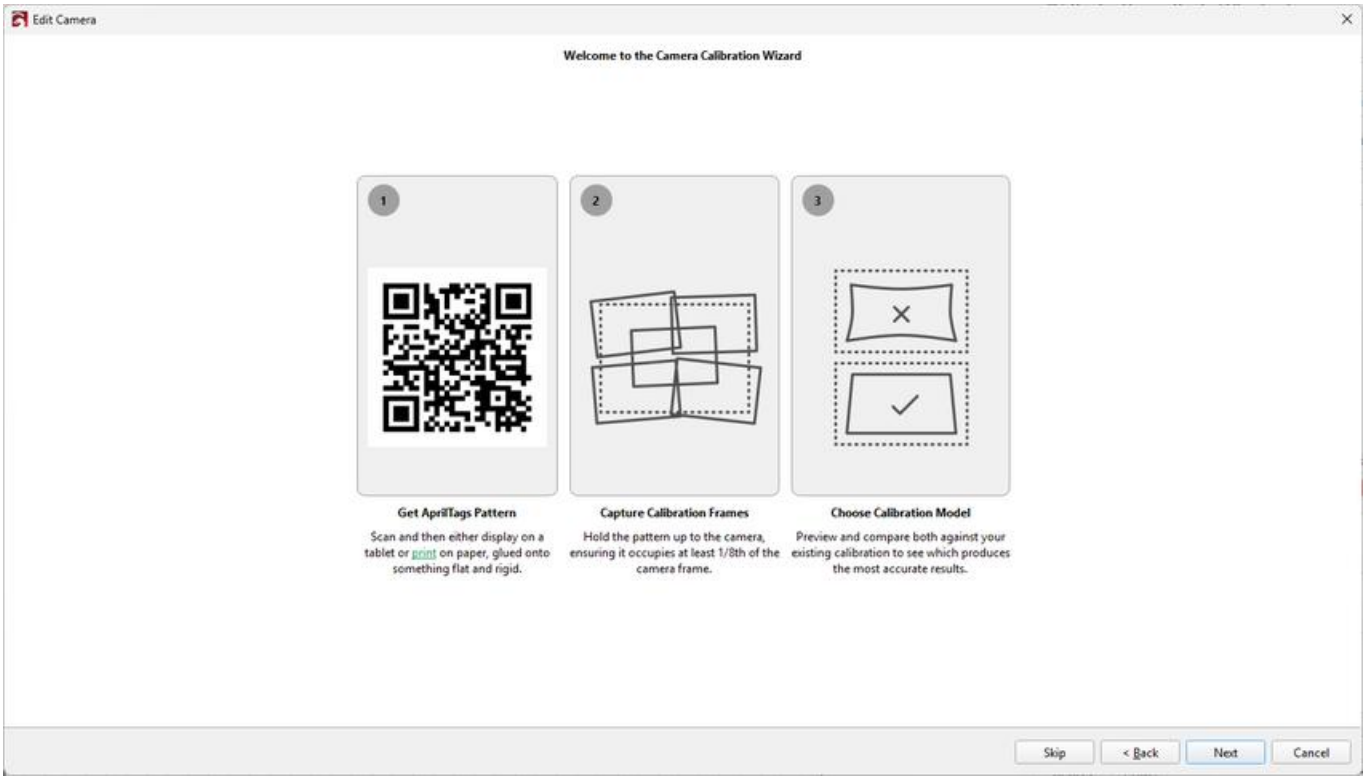
Download PDF

Download LBRN2

Quick Start

Need more help? Jump ahead to the full directions.

Camera Lens Calibration Walkthrough



The Camera Calibration Wizard has on-screen directions to walk you through the process. Here's a quick overview if you want to just jump in and get started.

1. Prepare the calibration pattern as described above.
2. Open the Camera Calibration Wizard for the camera you want to calibrate and click **Next** to continue.
3. Capture images of the calibration pattern following the directions in the wizard. You can use the jogging controls to move your laser out of the way. Click **Next** to continue.
4. Select your Calibration Model. You can preview the results of each model by selecting it. You want straight lines to appear straight, and should pay attention to what the corners of the bed look like.
5. If you're happy with the results, you can move on to Camera Workspace Alignment. Otherwise, you can try again.

Camera Workspace Alignment

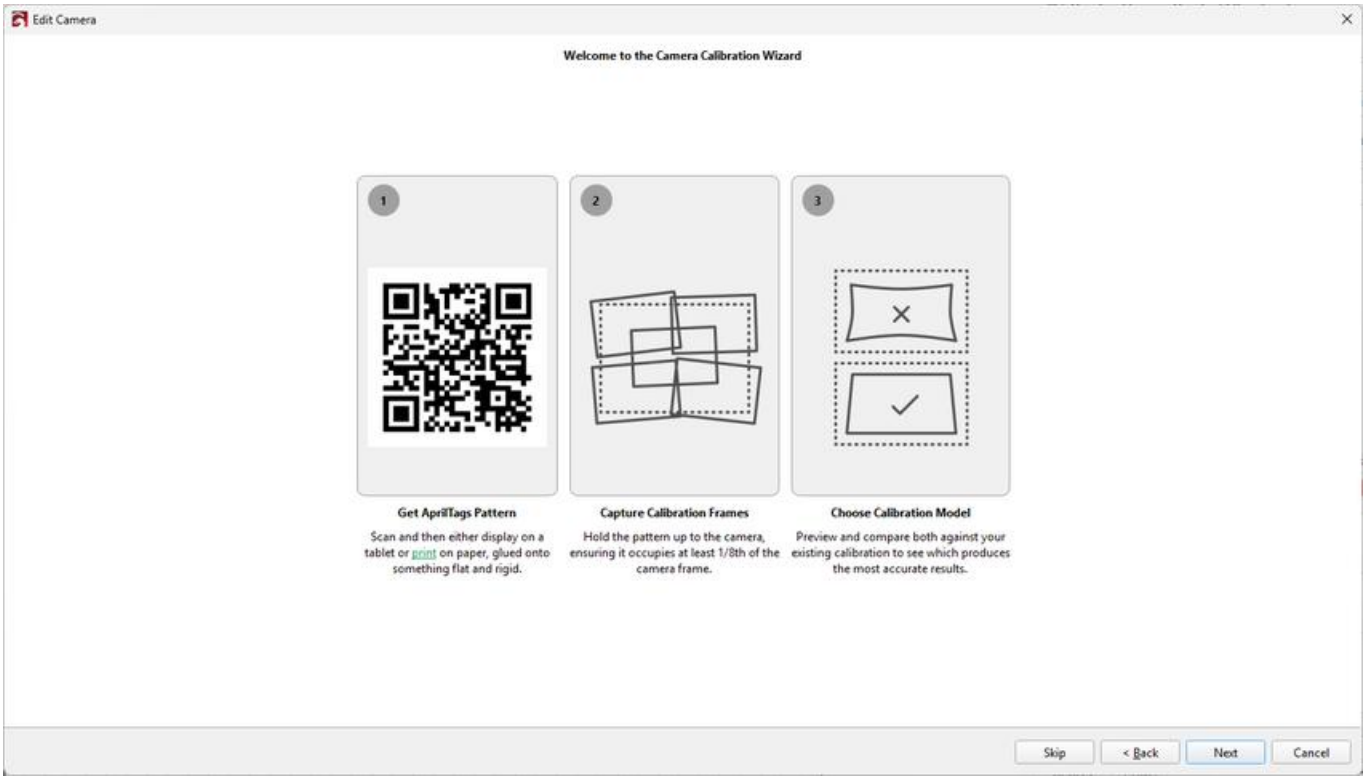
Walkthrough

If you want more detailed help with Camera Lens Calibration, you're in the right place.

1. Prepare the Calibration Pattern

As described above, make sure you have the AprilTag pattern printed or loaded up on a tablet screen. LightBurn detects the tags and uses their location, size, and distortion to calculate the distortion of the camera lens.

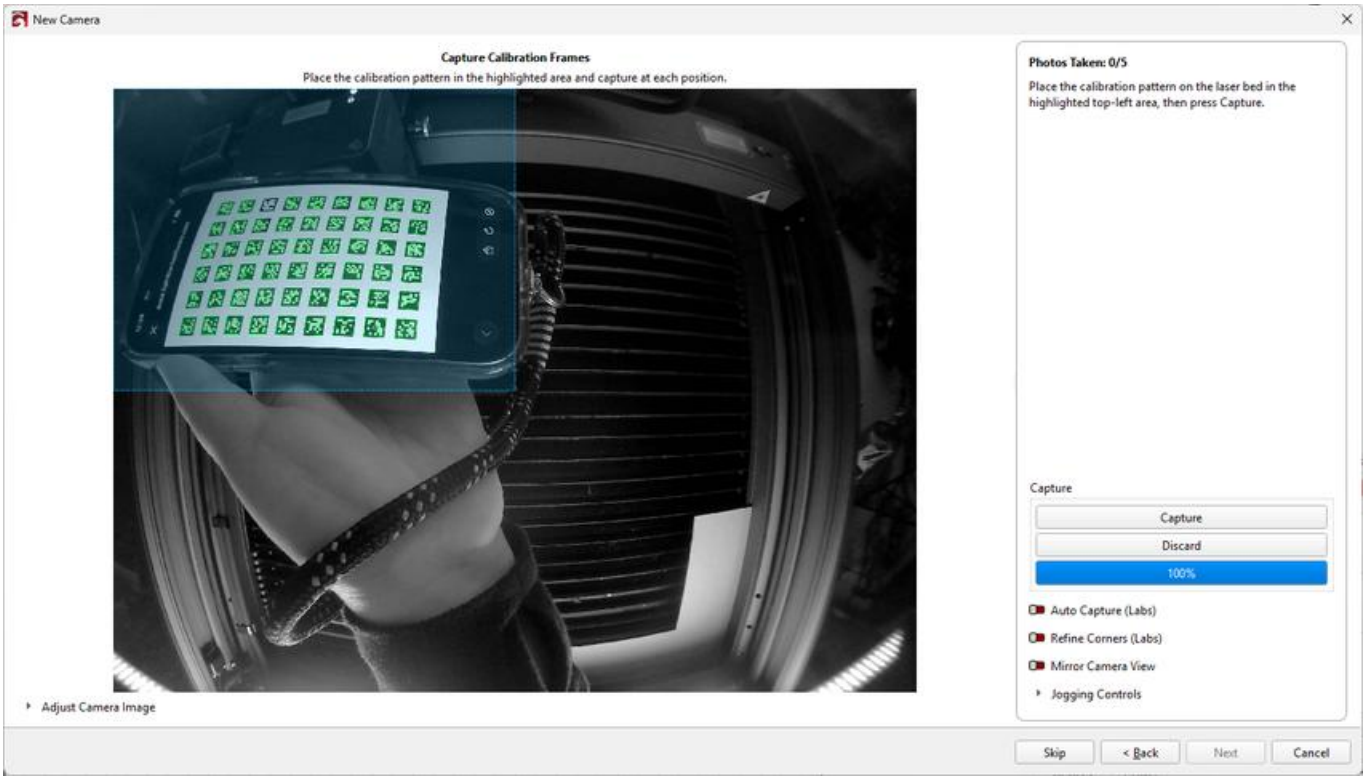
2. Open the Camera Lens Calibration Wizard



There are a few ways to open the Camera Lens Calibration Wizard outside of creating a new camera:

- Click the **Calibration** button for the camera you want to align in the Cameras / Overlays Window.
- Click the **Calibrate Lens** button in the Edit Camera window.

3. Capture Calibration Frames



The next screen will walk you through capturing **Calibration Frames**, which are images showing the calibration pattern in different parts of your camera's view. This is the information LightBurn needs to calculate distortion.

LightBurn's on-screen directions will show you where to hold the calibration pattern, and green highlights will appear on the AprilTags to show you when they are detected. If you're using **Auto Capture**, LightBurn will automatically capture the frames when the pattern is in a good place and the tags are focused. If you're not using Auto Capture, you'll need to use the **Capture** button to manually capture specific frames.

Camera Calibration With Auto Capture Disabled

1. Place your calibration pattern in the camera's field of vision in the highlighted box (the blue overlay shown in the top left in the screenshot above). Detected tags will be highlighted in green. You can use the **Jogging Controls** in the bottom right to move the laser head out of your way.
2. Click the **Capture** button to capture a frame. If you're unhappy with the frame you captured, you can discard it with the **Discard** button.
3. Move the calibration pattern to fit in the next box and repeat the process.

Camera Calibration With Auto Capture Enabled

1. Enable the **Auto Capture** option in the bottom right corner of the Calibration Wizard.
2. Place your calibration pattern in the camera's vield of vision, following the on-screen directions to guide your placement. For the first few captures, the placement won't be very specific, but after that, LightBurn will start giving you specific ways to hold the calibration pattern.

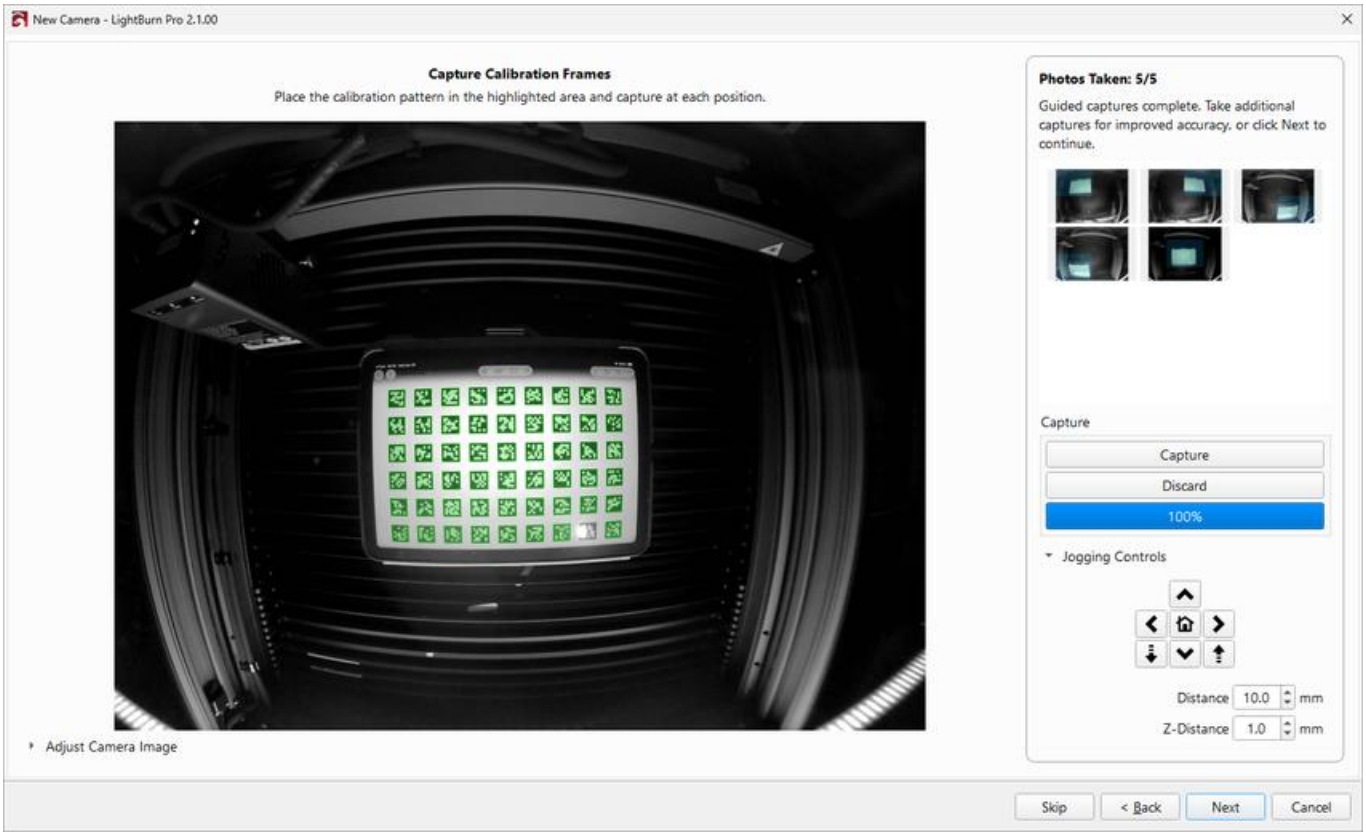
3. If you can't achieve one of the calibration frames LightBurn requests, you can choose **Skip** and LightBurn will recommend a different placement.
4. As LightBurn gets closer to being ready to capture a frame, the highlighting on the tags will shift from red to green.
5. When a suitable calibration frame is detected, the highlights will be bright green and LightBurn will automatically capture the frame and move on to the next position.

Capture Complete

After you've captured the required frames, your view of the calibration tags should be more accurate than at the beginning, but don't worry if it's not quite right yet. You can continue to capture additional frames if you want, but in the next step you'll choose a calibration model to complete the process.

Calibration Frame Capture Options

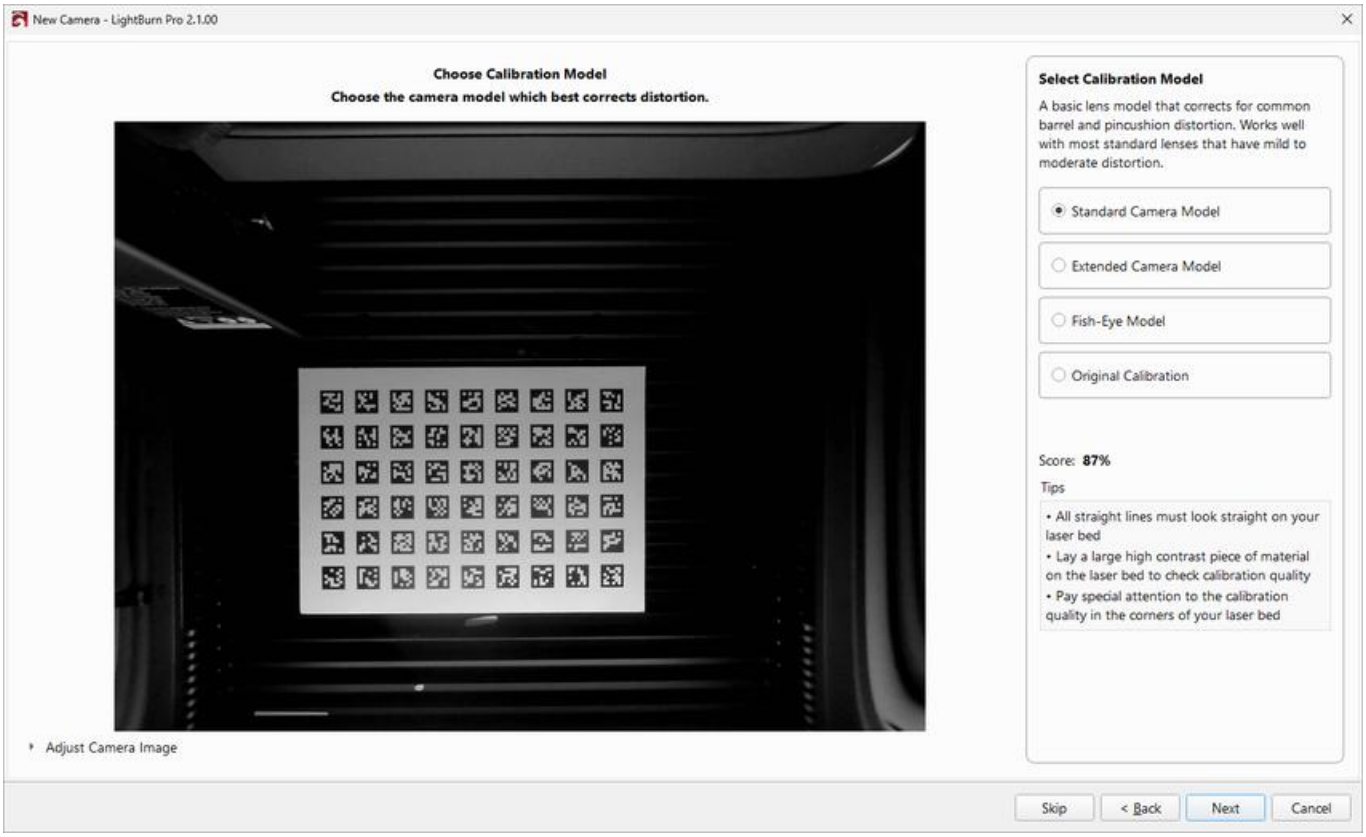
- Enable **Auto Capture** to use LightBurn 2.2's new capture method and have LightBurn automatically capture images.
- Enable **Refine Corners** to have LightBurn attempt to make the corners more accurate than a standard calibration.
- Enable **Mirror Camera View** to have LightBurn flip the camera image, if your camera view is backwards.
- Use the **Jogging Controls** to move your laser head if it's in your way.
- Use **Adjust Camera Image** to adjust your camera image, including brightness and contrast.



At this point, the view of the calibration tags should be more accurate than at the beginning, but will likely not look entirely correct. You can keep going to take more calibration frames, or click **Next** to continue to the next step.

4. Choose Calibration Model

There are 4 different mathematical models LightBurn can apply for the corrections.



Click the model options on the right to preview what the camera image looks like under each. A large, high contrast pattern (for instance, the calibration pattern) can be helpful for checking the accuracy of the calibration. Select the model that looks the most accurate and click the **Next** button to continue. The **Extended Camera Model** works well with most lasers, but it's worth trying all of them.

If none of the models look accurate, click **Back** to capture additional samples or restart the process.

> **Tip**
>
> Look for lines you know are straight in the real world. If they're wavy or curved in the capture, that means there's distortion in the selected model.

Each model includes a description with more information.

> **Calibration Scores**

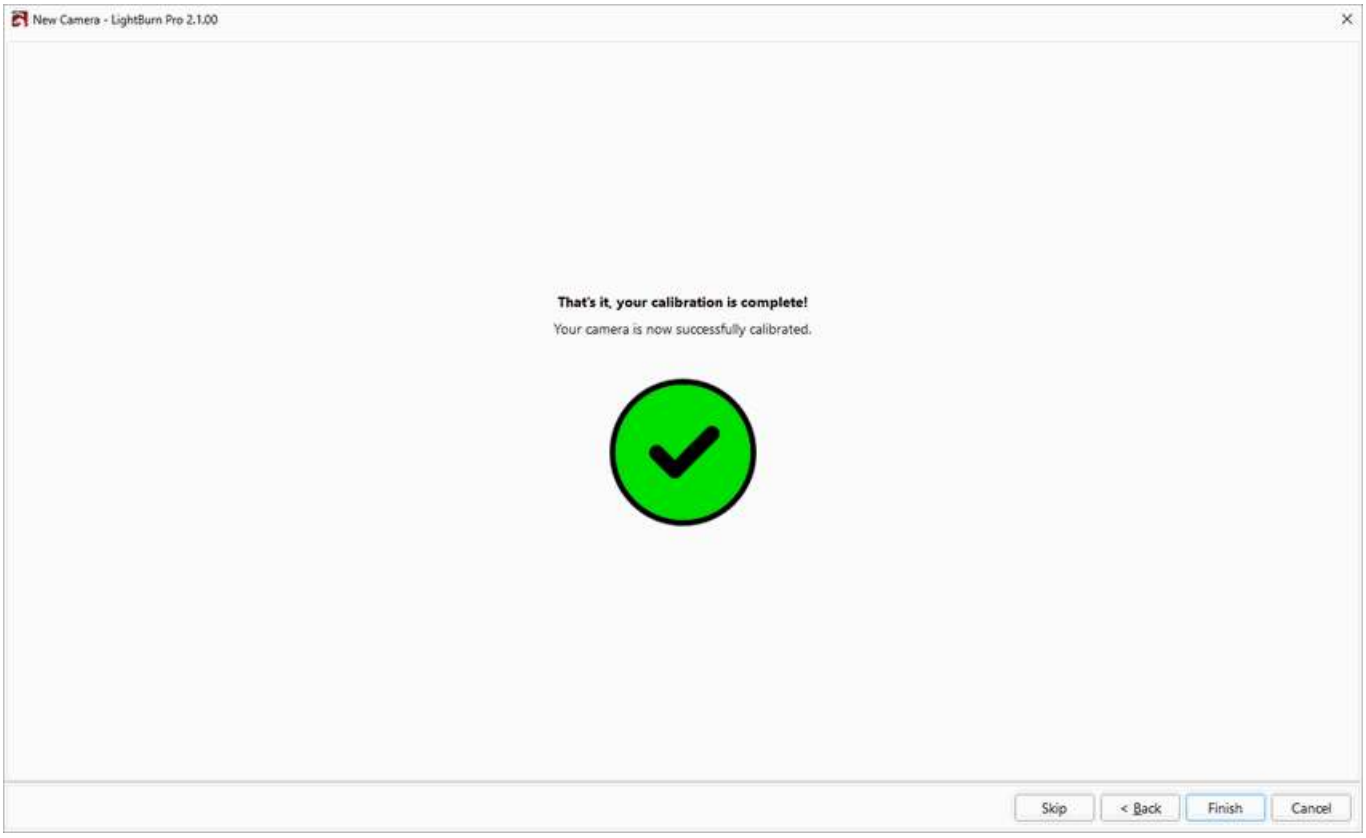
The percentage score shown for each calibration model can be a helpful indicator, but a higher score doesn't guarantee a better calibration. Pick the model that looks most correct, even if it doesn't have the highest score. The **Extended Model** often produces the best results for cameras with very distorted lenses. The **Fisheye Model** is designed for fisheye lenses, which are a very specific style of lens, not just any camera with a very wide field of view.

Why isn't a high score always better?

The score comes from the average error between where LightBurn expects the calibration tags to be according to the calibration model and where they actually are. A calibration that only covers the center of the image tends to score higher because the center only has a little bit of distortion. Once you include the edges in the calibration, the model has to account for more real-world imperfection, so the score tends to drop, even though the calibration itself can be more accurate.

5. Success!

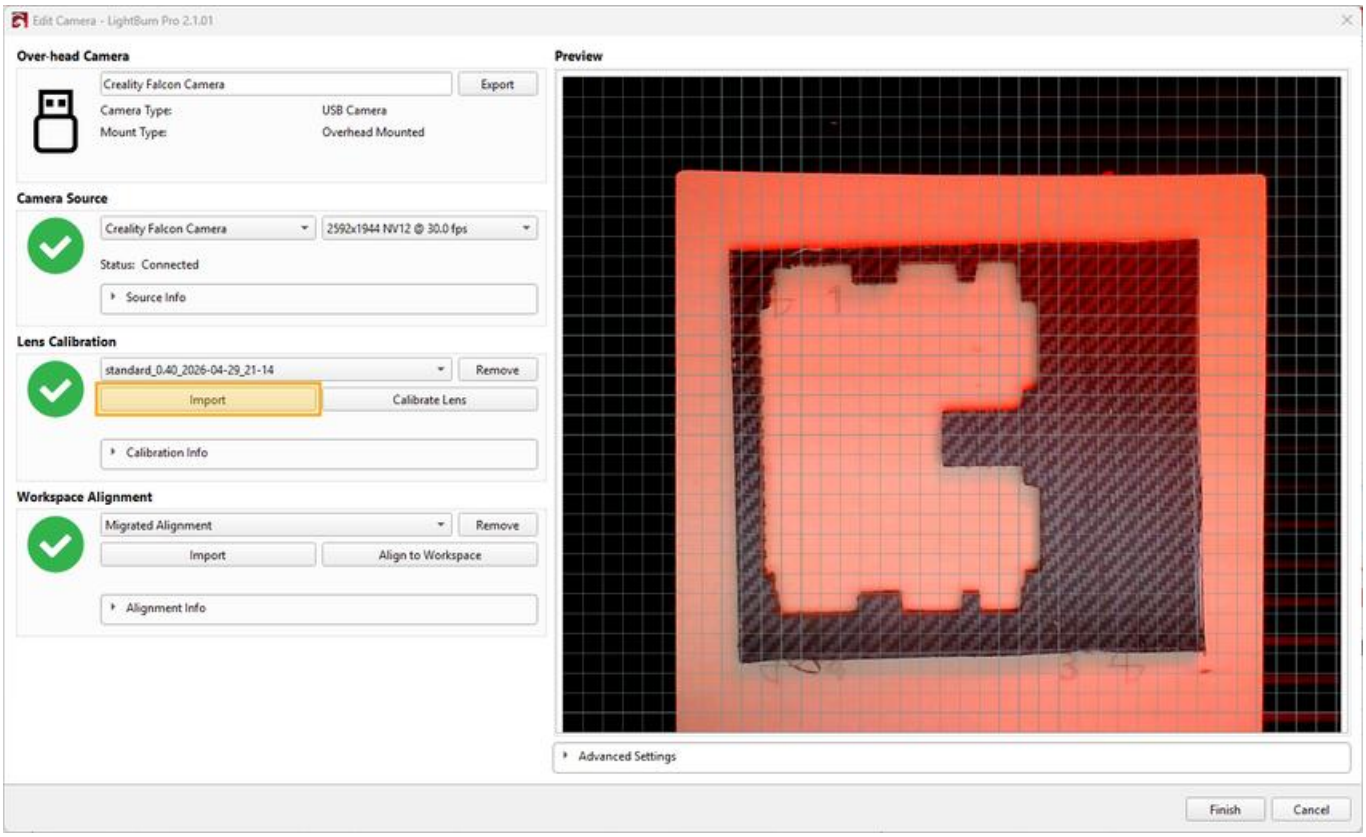
On the final screen, click **Finish** to close the process and save your changes. Next, you'll need to do Camera Alignment to align your camera to your workspace.



Import Existing Calibration

If you've already calibrated your camera and exported a **LightBurn Camera File**, you can import and reuse that calibration. Camera Lens Calibration only depends on the camera, not the mounting within the device, so even if you've moved the camera, you can still reuse the lens calibration.

1. Open the Edit Camera window in one of the following ways:
   - In the Laser Tools menu, select **Manage Cameras**, then select the camera you want to edit and click the **Edit** button.
   - In the Cameras / Overlays Window, right-click on the camera you want to edit to open the context menu and select **Edit**.
2. Select the **Import** button in the **Lens Calibration** section. See the screenshot below for the location.
3. Select your `.lbcm` file and click **OK**.



What's Happening

Feel free to skip this section if you're not interested.

All cameras distort the image they capture at least a little, and the larger the area they can capture, the more serious that warping is. Wide-angle or fisheye lenses that can capture large workspaces from relatively close will be particularly badly distorted.

The Camera Lens Calibration process is required so that LightBurn can figure out the distortion pattern of your particular camera and correct for it in the software. The AprilTag calibration pattern you use for this process is important for the improved process, and allows for a simplified calibration process compared to the older dot pattern.

Camera Alignment

> **Significant Changes in LightBurn 2.2**
>
> LightBurn 2.2 includes significant UI improvements to the camera interface as well as updates to make it easier to calibrate your camera. If you are on an older version of LightBurn, use the dropdown at the top of the page to select the appropriate version of the camera pages.

After calibrating your camera's lens to correct for distortion, you'll need to go through the **Camera Alignment** process to tell LightBurn where your camera is positioned relative to your machine's work area.

Unlike the lens calibration, which is only based on the camera and can be reused when you move the camera, the Camera Alignment needs to be redone each time you move the camera, although it can be reused if you're migrating to a new computer and haven't changed the position of the camera in your laser.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=bwlfvLYgIMA

> **What if I use more than one camera?**
>
> If you have two or more overhead cameras in your laser to better cover a large bed, align each camera separately. To make this easier, you can select the **Two Zones** preset when you get to Pattern Setup. You can edit the size with the controls at the bottom of the window or by dragging the edges around.

If you're using a head-mounted camera instead of the more common overhead cameras, skip ahead to Head-Mounted Camera Alignment. The process is similar, but there are some differences.

Skip to Head-Mounted Camera Alignment

Open the Camera Alignment Wizard

There are a few ways to open the Camera Alignment Wizard outside of creating a new camera:

- Click the **Alignment** button for the camera you want to align in the Cameras / Overlays Window.
- Click the **Align to Workspace** button in the Edit Camera window.

Before You Begin

1. Select the correct camera in the Cameras / Overlays window.
2. Calibrate your camera lens.
3. Make sure your camera is mounted where and how you want it mounted for actual use.
4. Make sure you have laser-engravable material to burn the test pattern on. The material should be big enough to burn a pattern that fills most of your workspace.
5. Place your material into the laser and focus your laser to the material.

6. Make sure your Start From is set to Absolute Coords
7. If you have **Material Height** and/or **Support Height** available in the Cut Settings Editor or the Cuts / Layers Window, make sure they're set correctly for the material you'll be using.

Quick Start

Need more help? Jump ahead to the full directions.

Camera Alignment Walkthrough

1. Select your alignment method. **Automatic alignment** is faster and easier, but if you don't have good lighting or can't get a good high-contrast mark, you may need to go with **Manual Alignment** instead. The Alignment Wizard will show you an overview of the steps you'll need to take.
2. Set up your alignment pattern, following the on-screen directions, then click **Next** to move to the next step.
3. Make sure your material is centered in the laser bed and the laser is focused. Burn your alignment pattern, following the on-screen directions.
   - You can adjust your **Speed** and **Power** directly from the Alignment Wizard, use **Import** to select a setting from your Material Library, or click **Settings** to open the Cut Settings Editor.
   - Use **Preview** and **Frame** to make sure your alignment pattern will burn where and how you expect, and start your job as usual with the **Start** or **Send** buttons.
   - The **Machine Control** section in the bottom left will let you jog and home your machine.
4. After burning your pattern, **do not move the material**. The location of the pattern needs to remain unchanged.
5. Follow the on-screen directions to complete the alignment procedure, then click **Next** to continue.
6. For the Automatic Alignment process, you may need to adjust camera settings and lighting to make sure at least 4 tags are detected.
7. For the Manual Alignment process, you'll need to click to indicate the centers of the crosshairs in order.
8. Check that the alignment appears correct in the preview, and either click **Next** to save the alignment or **Discard** to try again.
9. Success!

More Camera Information

Walkthrough

More detailed help on completing Camera Workspace Alignment.

1. Prepare Material

As described above, you'll need to have a suitable large piece of material, appropriate cut settings for a crisp, dark mark, and your camera securely mounted in the location you want to use it from.

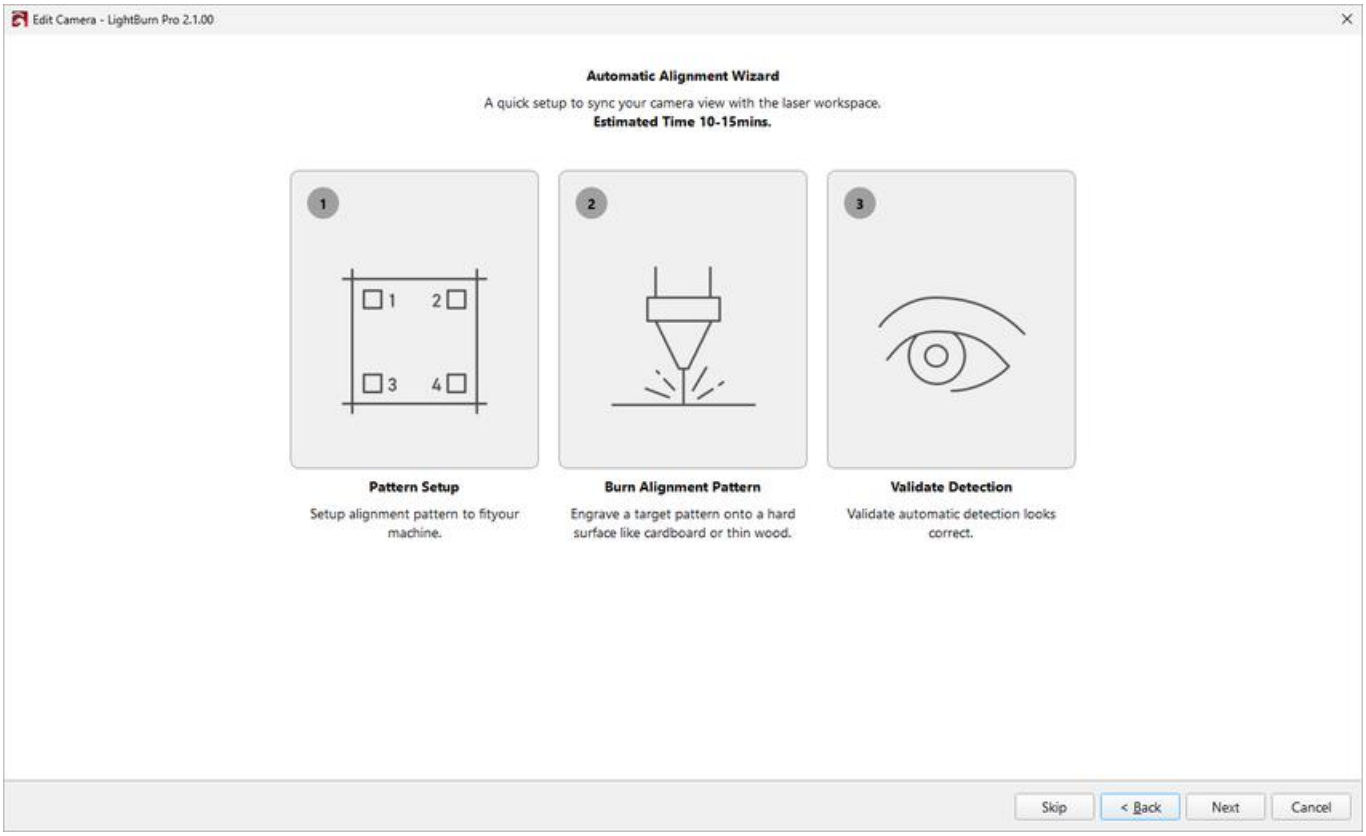
2. Open the Camera Alignment Wizard

There are a few ways to open the Camera Alignment Wizard outside of creating a new camera:

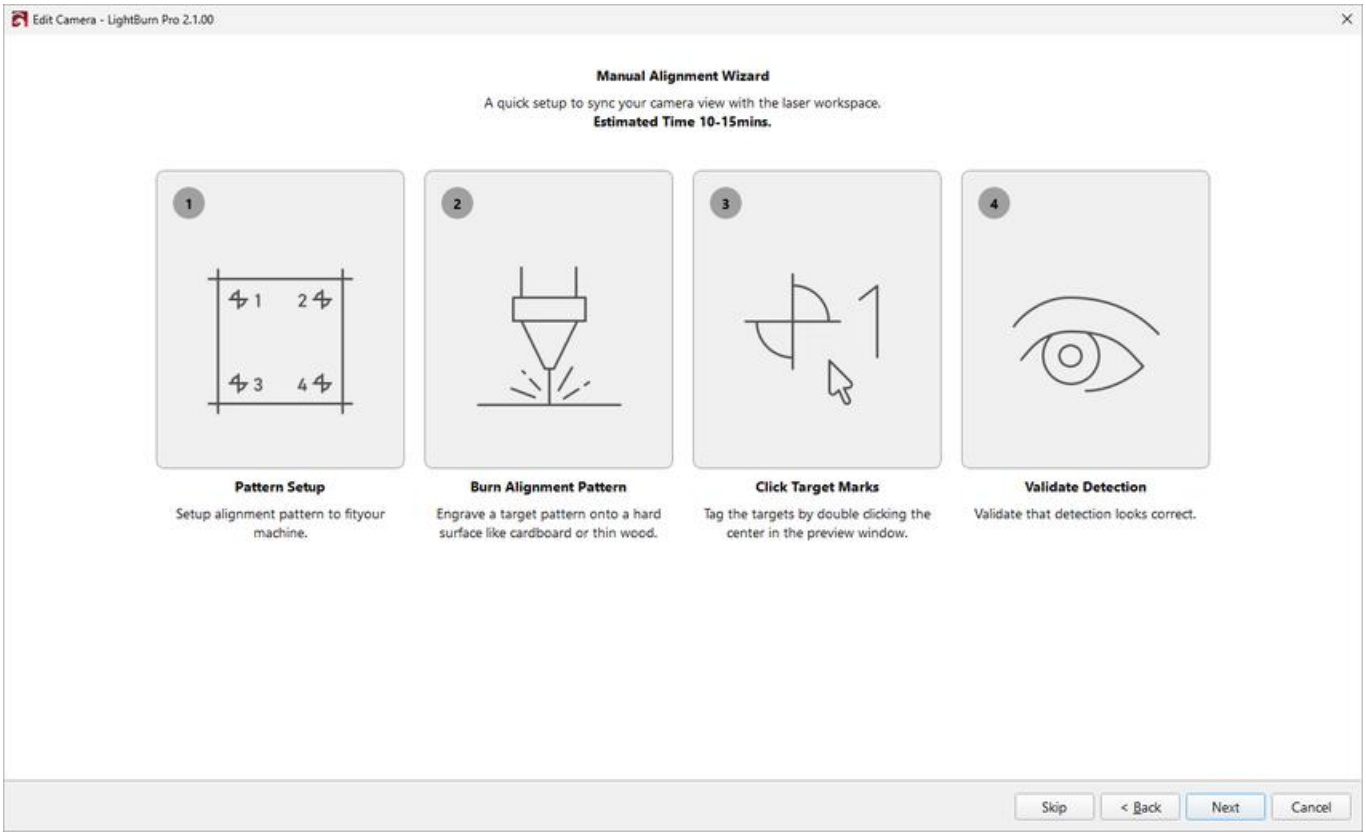
- Click the **Alignment** button for the camera you want to align in the Cameras / Overlays Window.
- Click the **Align to Workspace** button in the Edit Camera window.

3. Select your Alignment Method.

You can use automatic or manual Camera Alignment. The automatic method is quicker and easier, but requires good lighting and high contrast between the marked pattern and the background material.



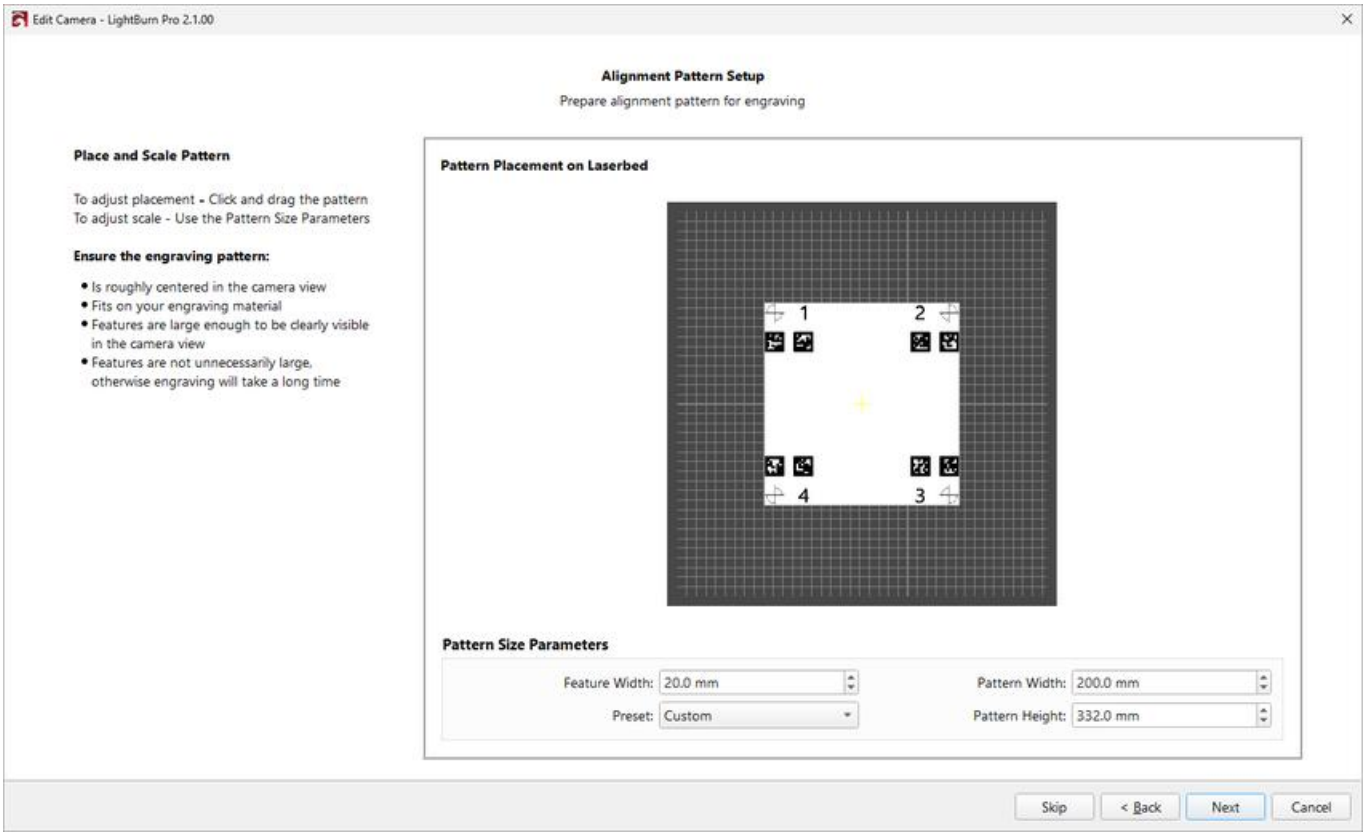
Overview screen for automatic alignment.



Overview screen for manual alignment.

4. Set Up Your Alignment Pattern

Next, you'll set up your alignment pattern. The screenshot below is from the Automatic Alignment wizard, but the only difference for Manual Alignment is that there are no AprilTags - all the settings work the same between the two.



Place and Scale Pattern

When creating the pattern, it should be:

- Roughly centered in the camera view.
- Sized to fit on your engraving material.
- Created with features large enough to be clearly visible in the camera, but not so large they take an excessively long time to engrave.

Fill Features

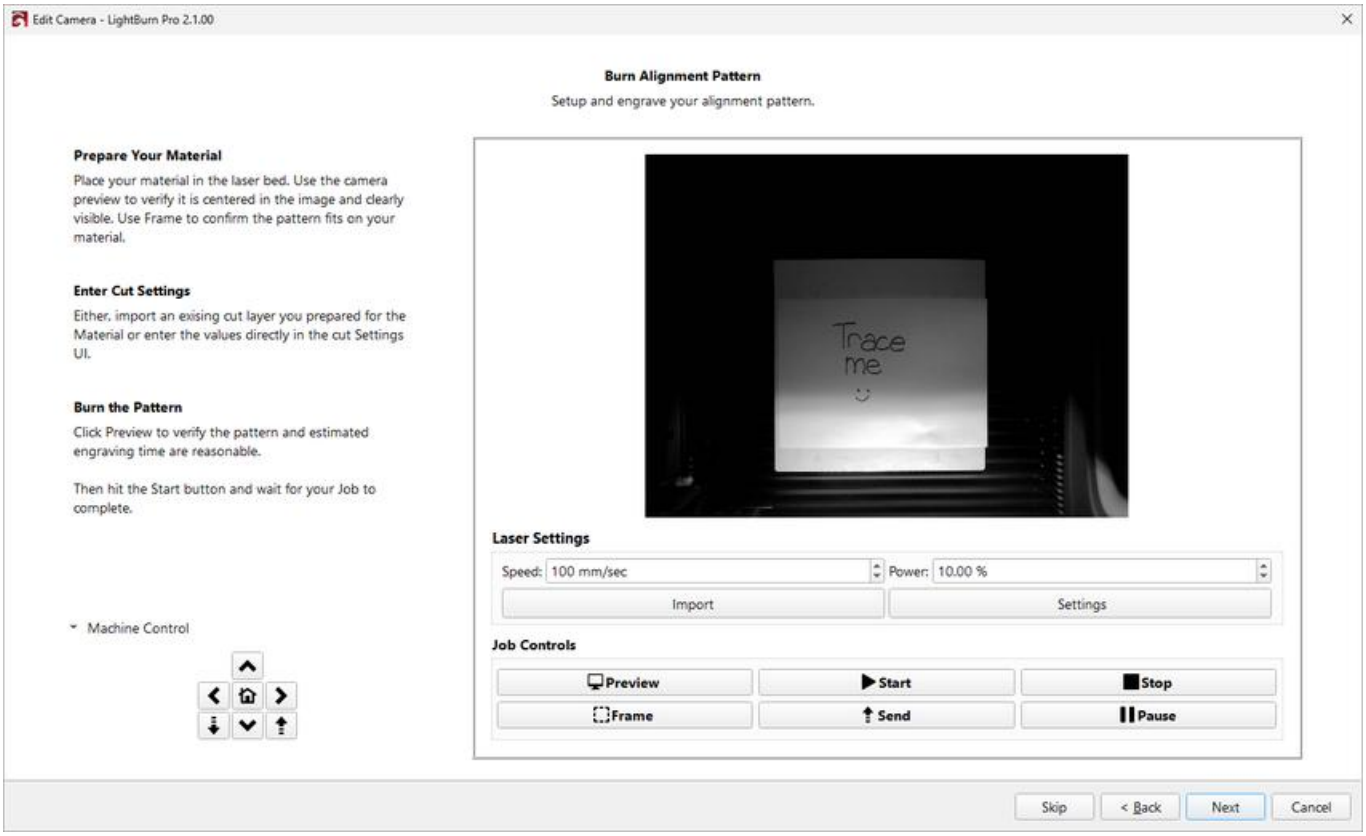
Enable **Fill Features** (new in LightBurn 2.2) to make the numbers and crosshair markers easier to see.

Pattern Size Parameters

- **Feature Width:** The size of the calibration features.
- **Pattern Width / Pattern Height:** The width and height of the overall pattern.
- **Preset:** You can choose one of several presets as a starting point. To go back to the defaults, you can select **Center** from the dropdown.

5. Burn Alignment Pattern

Place your material in the laser cutter and get ready to burn the alignment pattern.



Use the camera preview to make sure the material is near the center of the image and clearly visible. Use **Frame** to make sure the pattern fits the material, and use **Preview** to see what will actually be burned. Use the **Machine Control** section in the lower left to move your laser head around if needed.

Adjust your cut settings to engrave a dark, crisp mark without cutting through the material. You can adjust your settings in a few ways:

- Adjust speed and power right in the Alignment Wizard window, just below **Laser Settings**.
- Click the **Import** button to load a layer from your Material Library.
- Click **Settings** to adjust your layer settings in the Cut Settings Editor.

Run the job using **Start** or **Send**, however you'd usually control your laser.

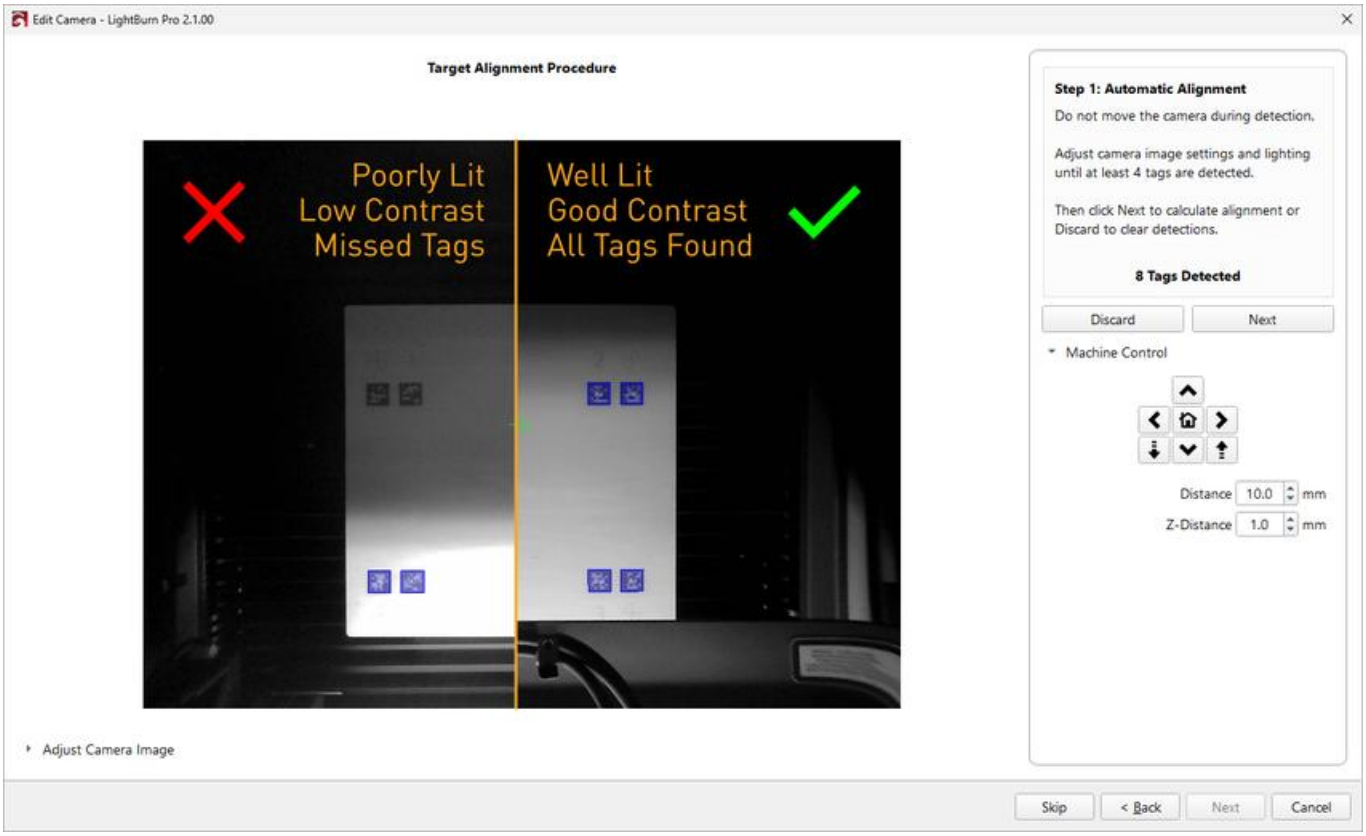
Do not move the pattern after you burn it, or you will need to burn a new alignment pattern.

6. Target Alignment

Depending on whether you chose **Automatic Alignment** or **Manual Alignment**, you'll see one of two screens.

Automatic Alignment

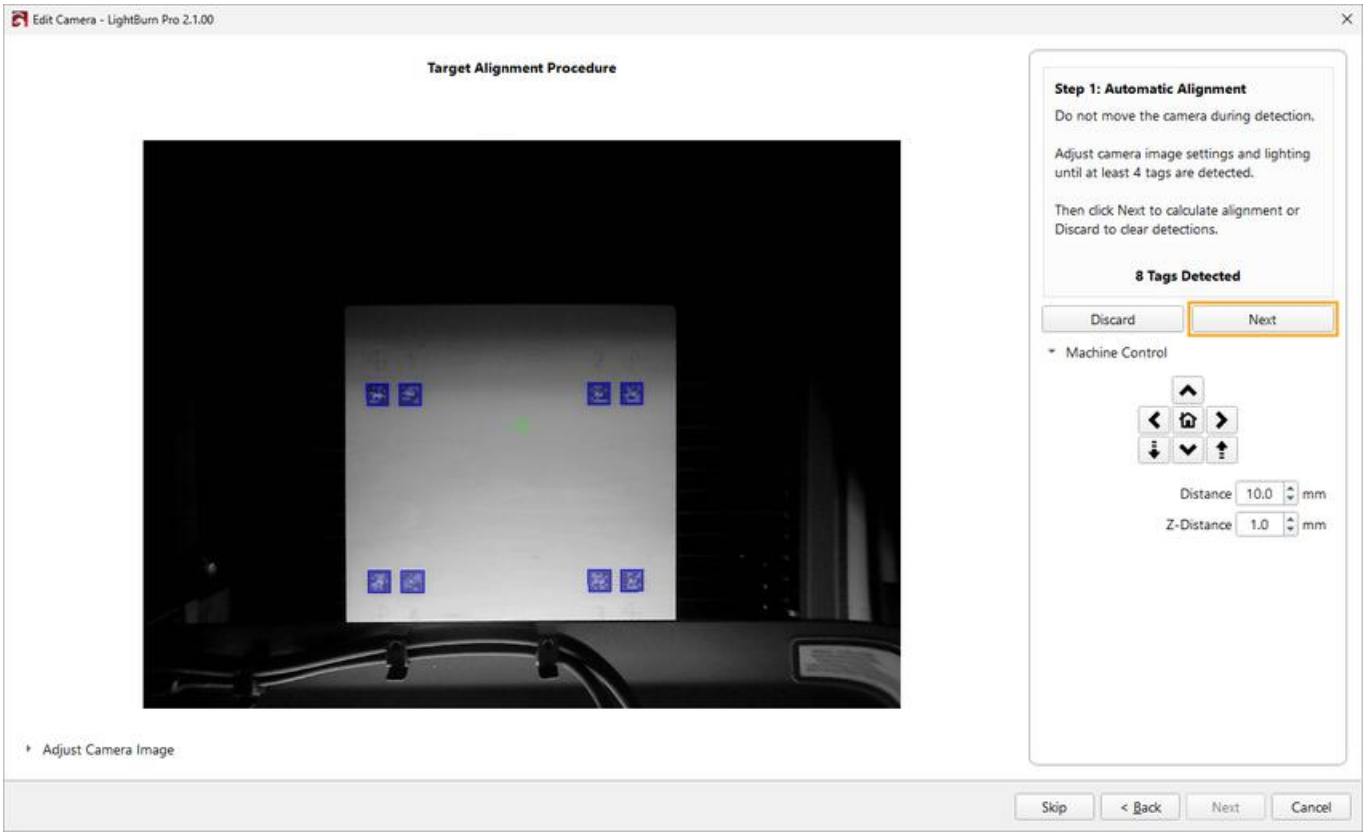
Automatic Alignment uses the camera to read the AprilTags you've engraved and automatically calculate the placement of the camera relative to the material.



Getting good results with automatic alignment

Because LightBurn is attempting to read the alignment pattern for you, it's important that you have good lighting and clean, high contrast engravings. See the screenshot above for an example of poor lighting/contrast on the left, and improved lighting/contrast on the right. If LightBurn is not detecting the tags, you may need to improve the lighting or adjust the camera image with the **Adjust Camera Image** controls in the lower left corner.

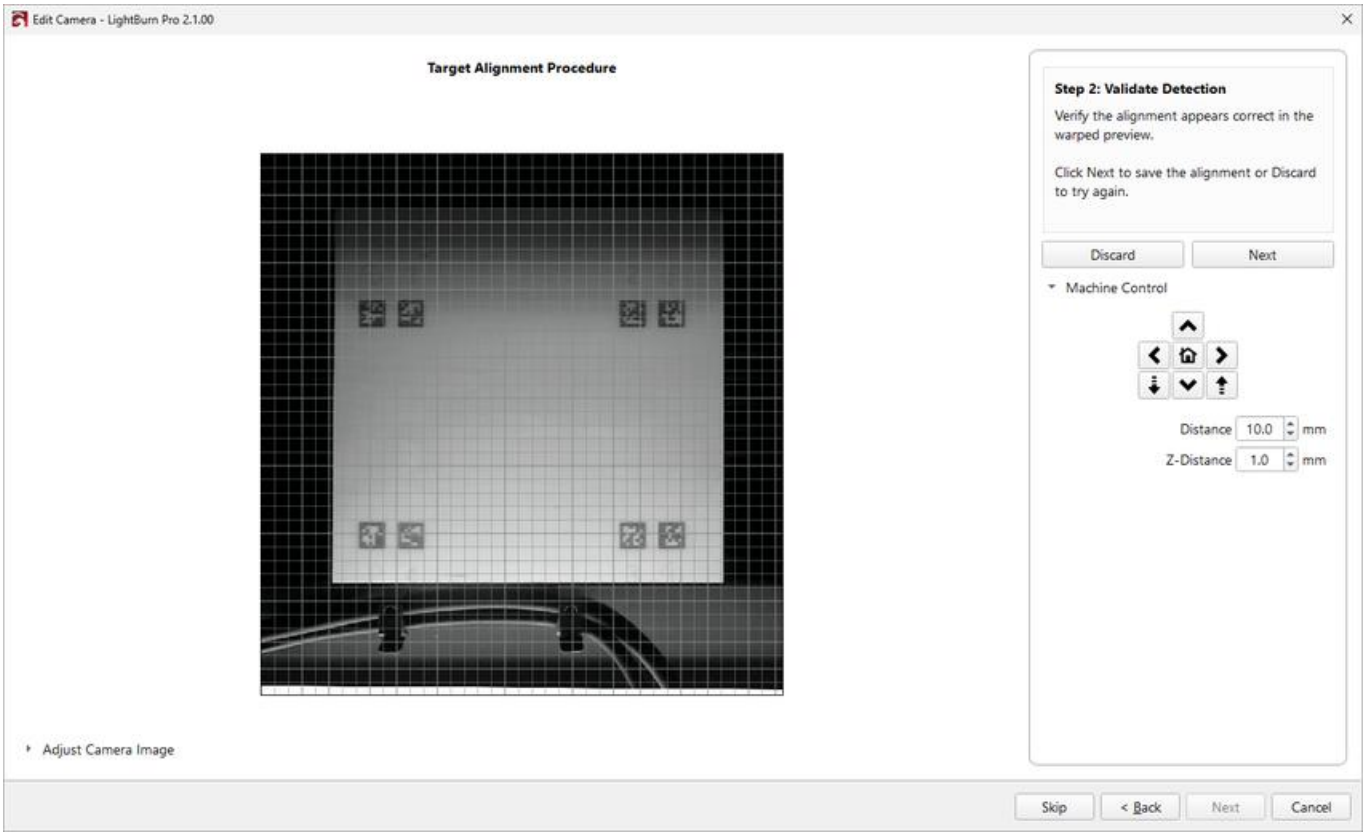
To start, LightBurn will attempt to detect the tags burned into the material. For automatic alignment to work, LightBurn needs to be able to detect at least 4 of the 8 tags you've engraved. You may need to use the **Machine Control** section or **Adjust Camera Image** section to make sure the image is clear and unobstructed.



If LightBurn Can't Automatically Detect the Tags

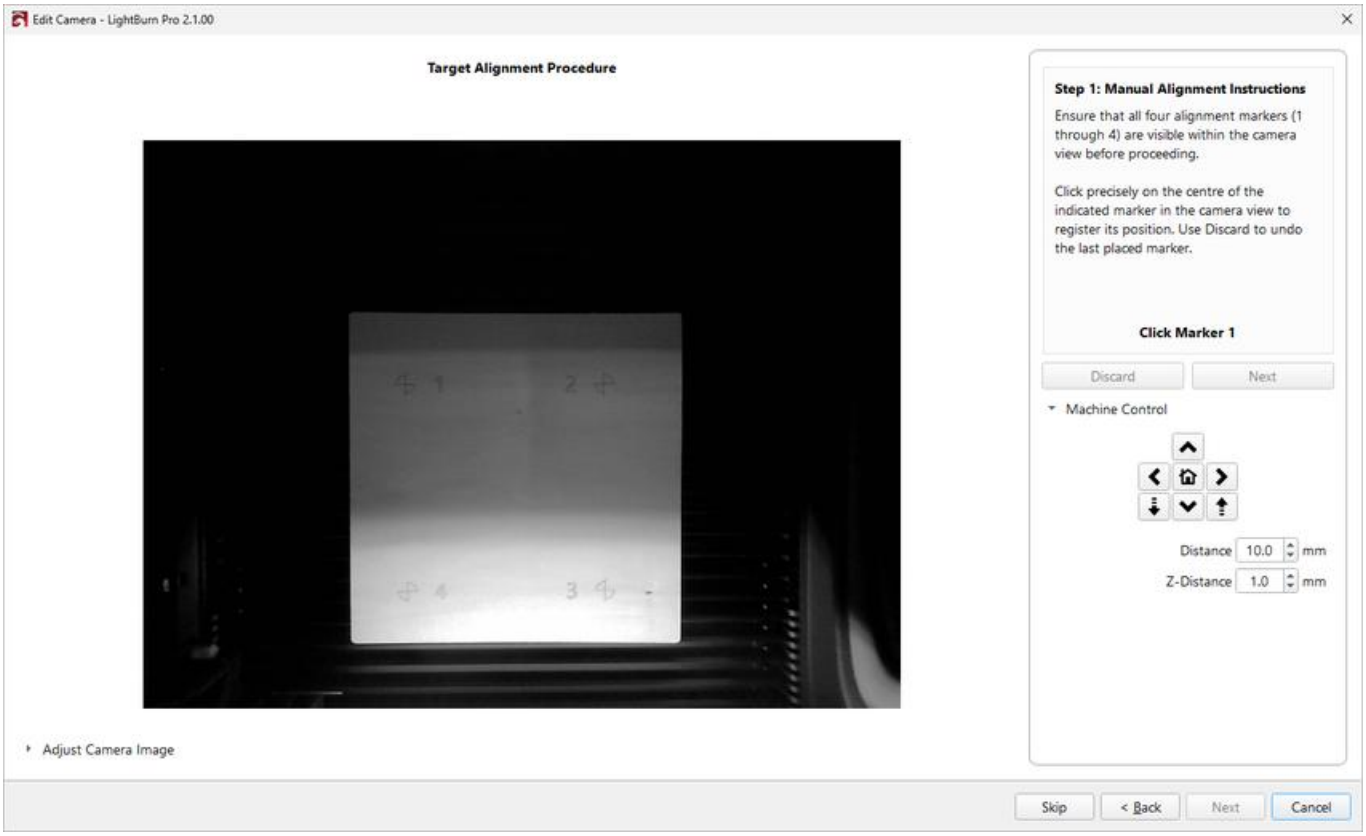
If LightBurn can't detect the alignment tags automatically, you can use the **Revert to Manual Alignment** button to switch over to the Manual Alignment process. You don't need to reburn the alignment pattern to do this - both the automatic and manual alignment patterns have crosshairs in the same place.

After LightBurn detects the alignment tags, you'll check the results, making sure that the alignment looks correct. Straight lines should appear straight, and the view of the alignment pattern in your laser should match the overlay shown in the **Alignment Wizard** window. If you're satisfied, click **Next** and then **Finish** to save your alignment. Otherwise, you can try using the manual alignment process.



Manual Alignment

If you're unable to use Automatic Alignment, or just don't want to, you can also use the Manual Alignment process. Manual Alignment does the same calculations as Automatic Alignment, but requires you to identify the center of four crosshairs engraved onto the material yourself. If you aren't able to get a clear, well-lit, high-contrast alignment pattern or if your camera is particularly low resolution, you may need to use Manual Alignment.

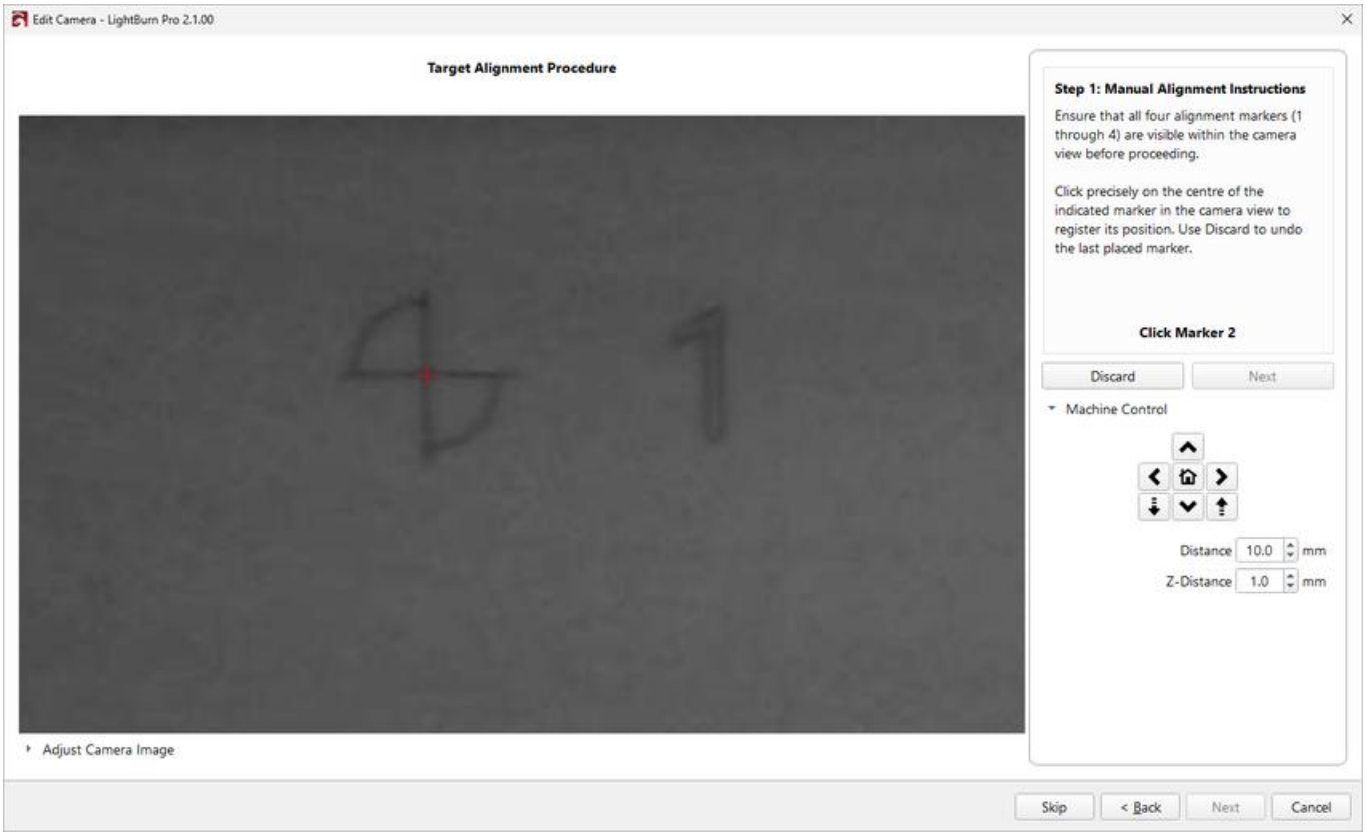


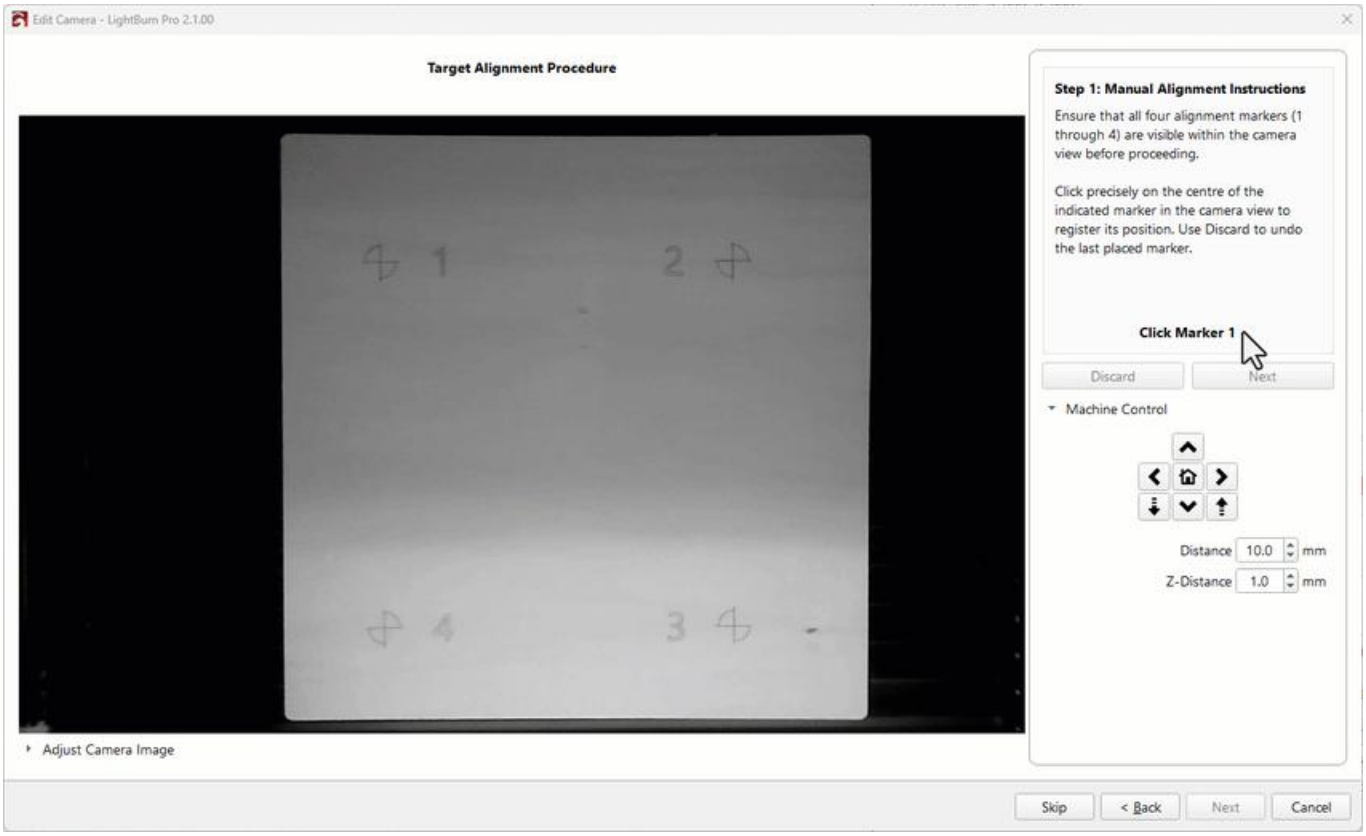
Tip

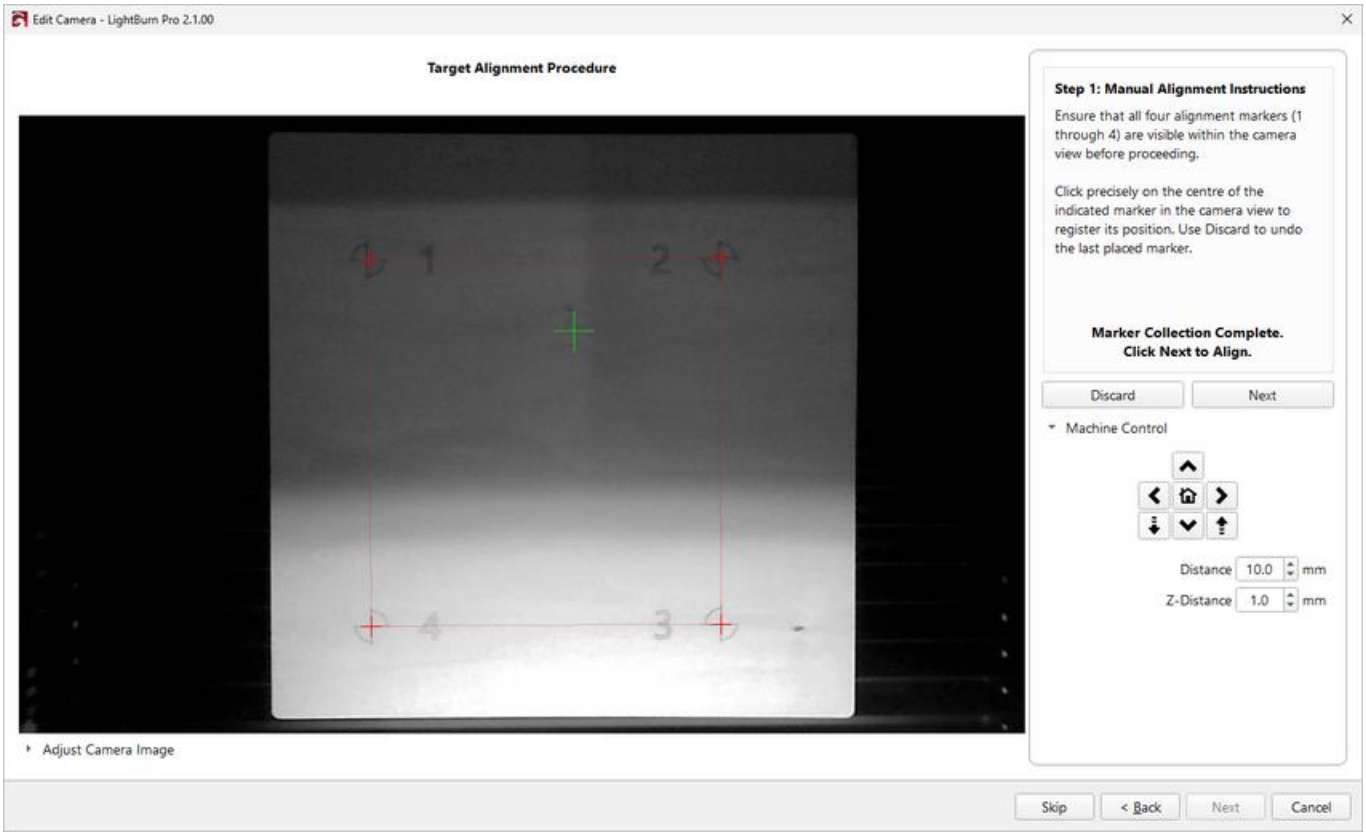
Trying to do this looking at the entire camera image will be both difficult and inaccurate. Use the scroll wheel to zoom in and out on the camera preview so you can see and select the center of the marks more easily.

LightBurn will start by showing you the camera view. You'll need to identify the exact center of the crosshairs - where the two lines meet. Click the center of each target in order. Press the **Discard** button to remove and redo a center mark.

Repeat for all four corners in order, following the on-screen directions.

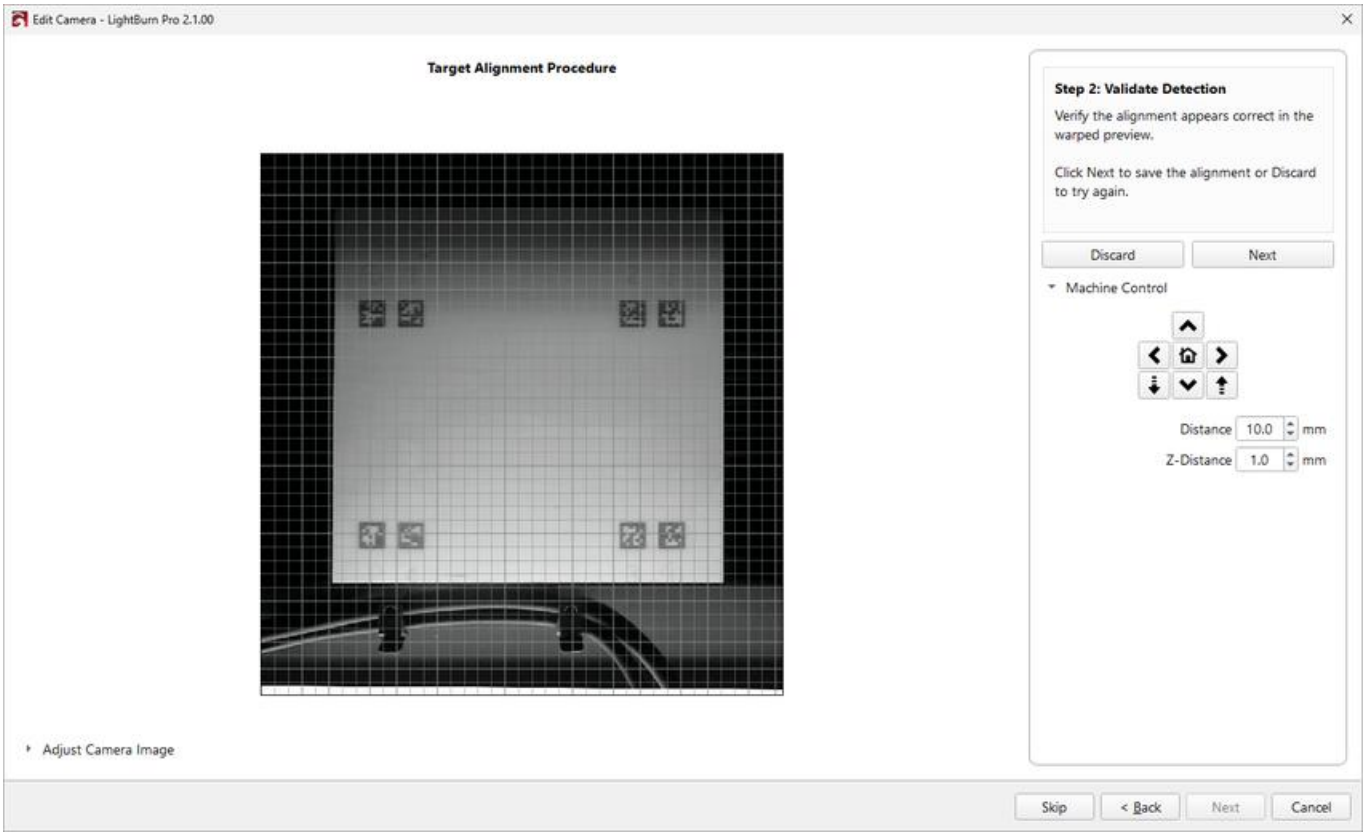

Edit Camera - LightBurn Pro 2.1.00
Target Alignment Procedure
Step 1: Manual Alignment Instructions
Ensure that all four alignment markers (1 through 4) are visible within the camera view before proceeding.
Click precisely on the centre of the indicated marker in the camera view to register its position. Use Discard to undo the last placed marker.
Click Marker 2
Discard
Next
Machine Control
Distance 10.0 mm
Z-Distance 1.0 mm
Adjust Camera Image
Skip
< Back
Next
Cancel


Edit Camera - LightBurn Pro 2.1.00
Target Alignment Procedure
Step 1: Manual Alignment Instructions
Ensure that all four alignment markers (1 through 4) are visible within the camera view before proceeding.
Click precisely on the centre of the indicated marker in the camera view to register its position. Use Discard to undo the last placed marker.
Click Marker 1
Discard
Next
Machine Control
Distance 10.0 mm
Z-Distance 1.0 mm
Adjust Camera Image
Skip
< Back
Next
Cancel



When you're done selecting all four manual targets, you should see something similar to the screenshot above, with red marks at the center of each crosshair, and red lines connecting each corner. Click **Next** to continue onto validation.

After you select the targets, you'll check the results, making sure that the alignment looks correct. Straight lines should appear straight, and the view of the alignment pattern in your laser should match the overlay shown in the **Alignment Wizard** window. If you're satisfied, click **Next** and then **Finish** to save your alignment. Otherwise, you can click **Discard** and try again.



Multiple Alignments for Different Material Heights

If have **Material Height** and/or **Support Height** available in the Cuts / Layers Window, each alignment you do will have the material thickness included in the name of the alignment. Creating a new alignment will replace an older alignment with the same Material Height, but won't overwrite alignments for different Material Heights. You can switch between these alignments in the Alignment Dropdown.

> **Don't want to redo the alignment for every material height you use?**
>
> You can use the Interpolate Alignment Height feature to have LightBurn auto-select from your existing alignments and approximate alignments for other material heights based on your existing alignments.

Head Mounted Camera

> **Significant Changes in LightBurn 2.2**
>
> LightBurn 2.2 includes significant UI improvements to the camera interface as well as updates to make it easier to calibrate your camera. If you are on an older version of LightBurn, use the dropdown at the top of the page to select the appropriate version of the camera pages.

> **Info**
>
> In previous versions of LightBurn, these were referred to as head-mounted cameras.

Head cameras are attached to a laser's head and travel with it as it moves around its work area. After properly Calibrating and **Aligning** your Head camera, LightBurn will be able to stitch together captures taken by the camera at multiple locations in the work area, producing an Overlay for your Workspace that you can use to position graphics over material.

Before configuring your head camera, make sure the camera is securely mounted to your laser head, with no ability to move or wiggle as the laser jogs. Before use, make sure the camera's wires don't interfere with the laser's movement.

Selecting a Head Camera

With growing interest in Head Cameras for compatibility with our new Camera Print & Cut feature, we've had more people asking for guidance on what to purchase.

If your laser didn't come with a head-mounted camera pre-installed, we recommend an inexpensive **USB Endoscope Camera** with a long, flexible USB cable. These cameras are also sometimes listed as a "USB Inspection Camera". Avoid cameras with built-in screens or rigid or semi-rigid cables that will interfere with your laser's motion. As of October 2026, these cameras are typicallly available for about $15-30 or €20-30 from common online retailers.



Example image of the type of endoscope camera to look for. This one has a semi-rigid cable, but you should look for one with a flexible cable.

If you want to explore other options, any small, light camera you can mount securely to your laser head and connect to over USB or WiFi should work. Keep in mind that WiFi cameras that use RTSP will not work - IP cameras need to output individual frames or an MJPEG feed.

For more information on mounting your new camera, see Camera Installation - Head Cameras.

Open the Camera Alignment Wizard

There are a few ways to open the Camera Alignment Wizard outside of creating a new camera:

- Click the **Alignment** button for the camera you want to align in the Cameras / Overlays Window.
- Click the **Align to Workspace** button in the Edit Camera window.

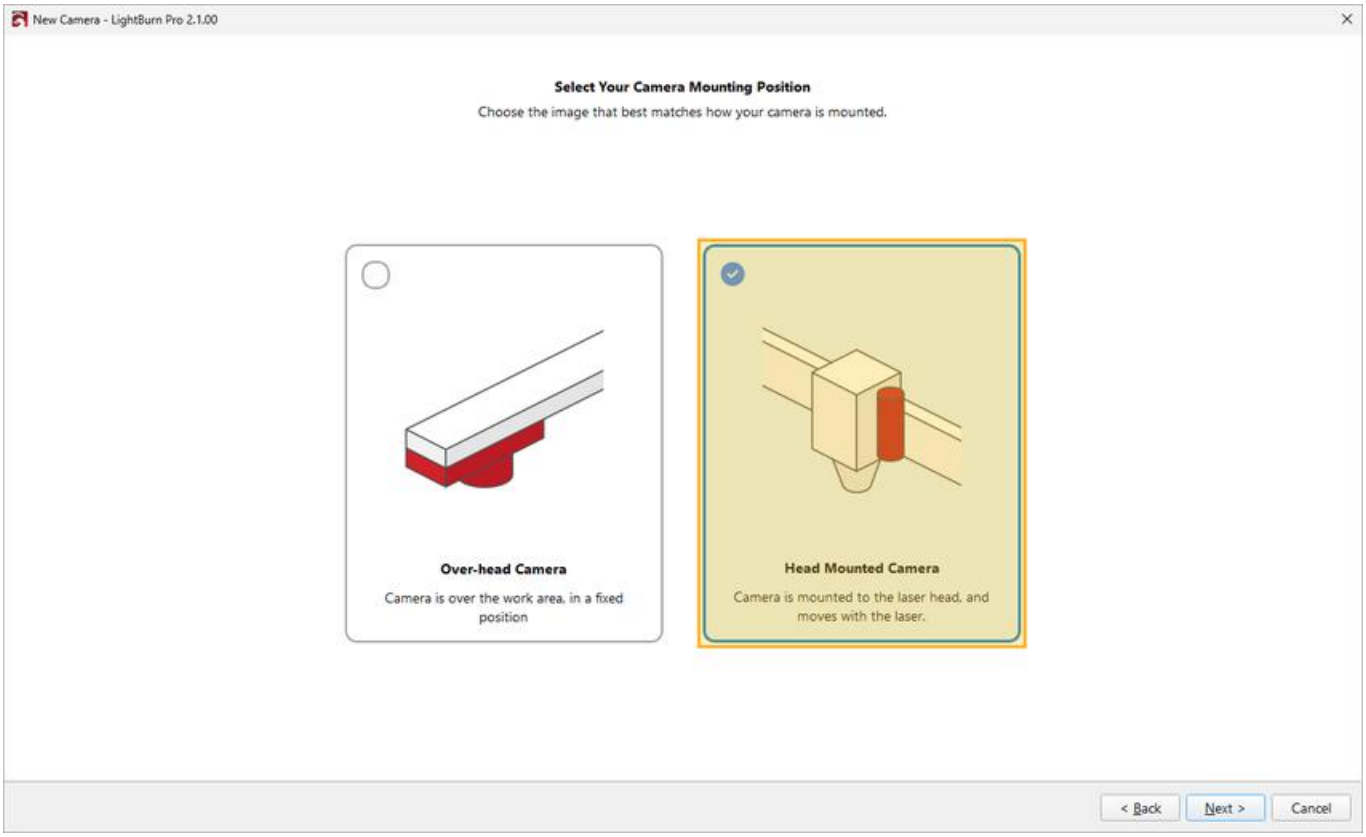
Before You Begin

1. Add your camera to LightBurn.

2. Calibrate your camera or import an existing Camera Lens Calibration from a LightBurn Camera File.
3. Make sure your camera is mounted appropriately and wires are out of the way.
4. Make sure you have laser-engravable material to burn the test pattern on. Choose a light-colored material that works well for engraving on your laser.
5. Make sure you've tested the material you'll use and have cut settings ready for a high-contrast, crisp engraving. You can enter the settings manually, import from the Material Library, or use an existing layer in your current file.
6. Place your material into the laser and focus the laser so you're ready to engrave.

Head Camera Alignment

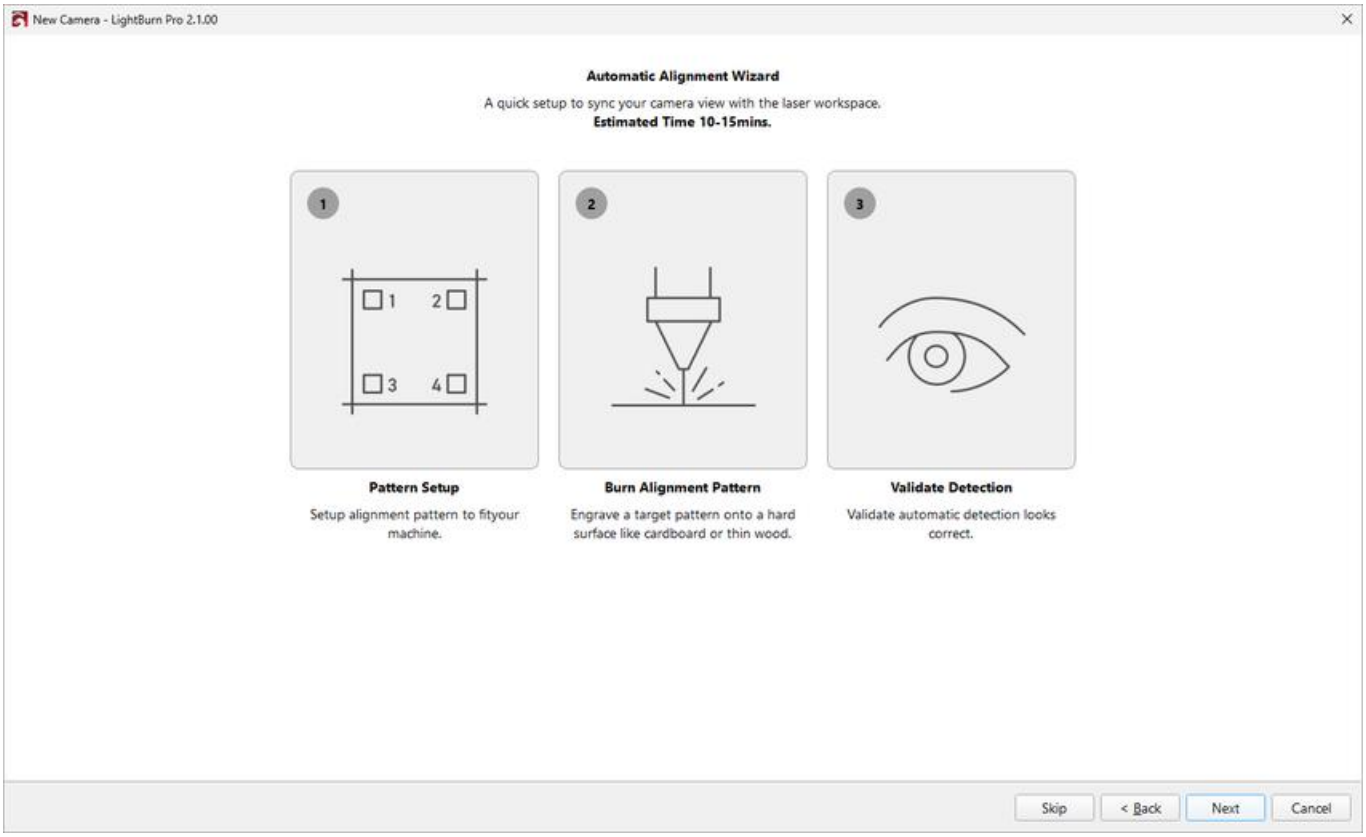
After calibration, when it's time to perform the Camera Workspace Alignment, choose the **Head Camera** option.



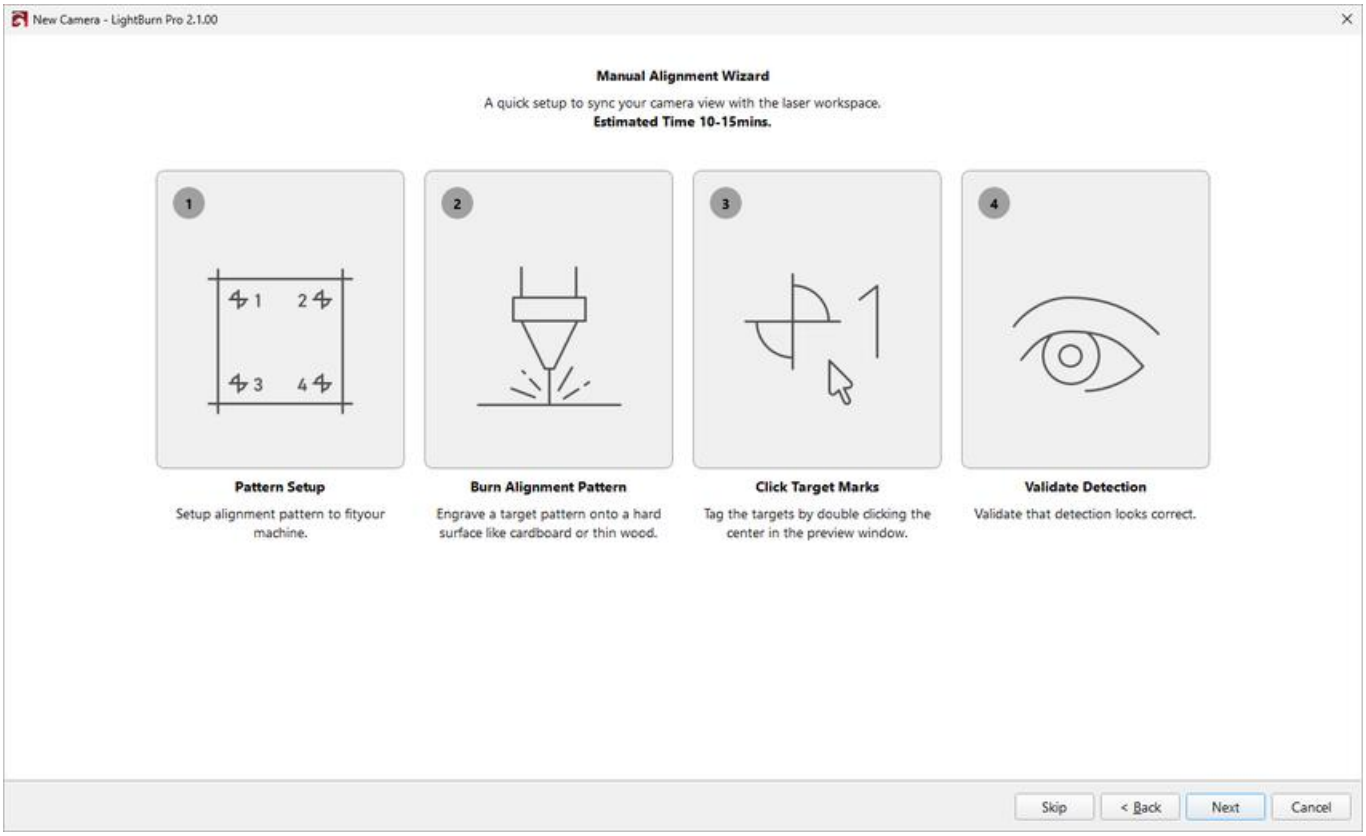
The Camera Alignment Wizard will walk you through the process, or you can continue reading for more detailed instructions. Click **Next** to continue to the next screen and select your alignment method.

1. Select Camera Alignment Method

You can use automatic or manual Camera Alignment. The automatic method is quicker and easier, but requires good lighting and high contrast between the marked pattern and the background material. Select the method you want to use and click **Next**. You'll see an overview of the process you've chosen. Click **Next** again to continue to set up your alignment pattern.



Overview screen for automatic alignment.



Overview screen for manual alignment.

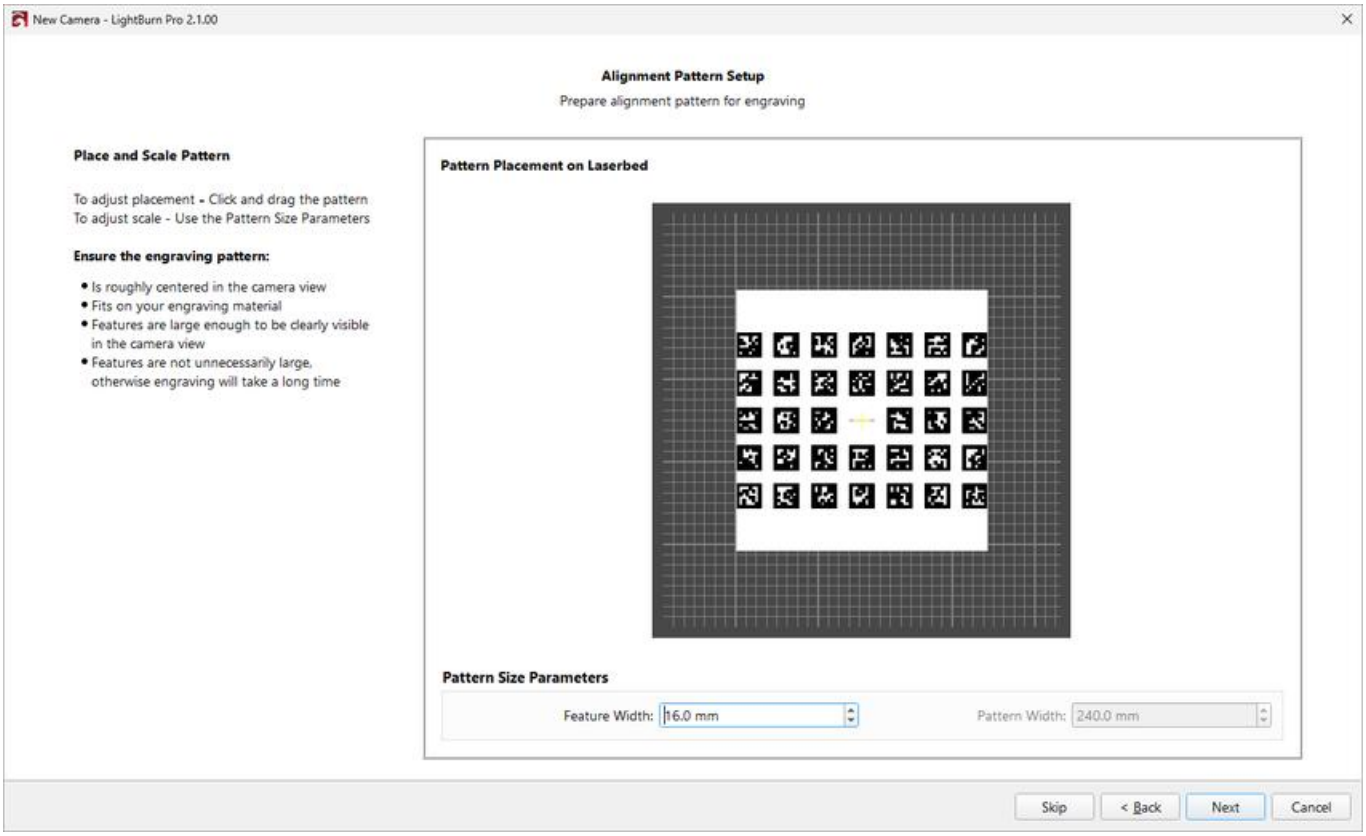
2. Alignment Pattern Setup

The automatic alignment process and the manual alignment process use different patterns, but the process is similar. Regardless of which pattern you're using, you can adjust the **Feature Width** under **Pattern Size Parameters** to adjust the size of the markings and the overall pattern size. The pattern should be:

- Roughly centered in the camera view.
- Sized to fit on your engraving material. Adjust the overall size by changing your **Feature Width**.
- Created with features large enough to be clearly visible in the camera, but not so large they take an excessively long time to engrave.

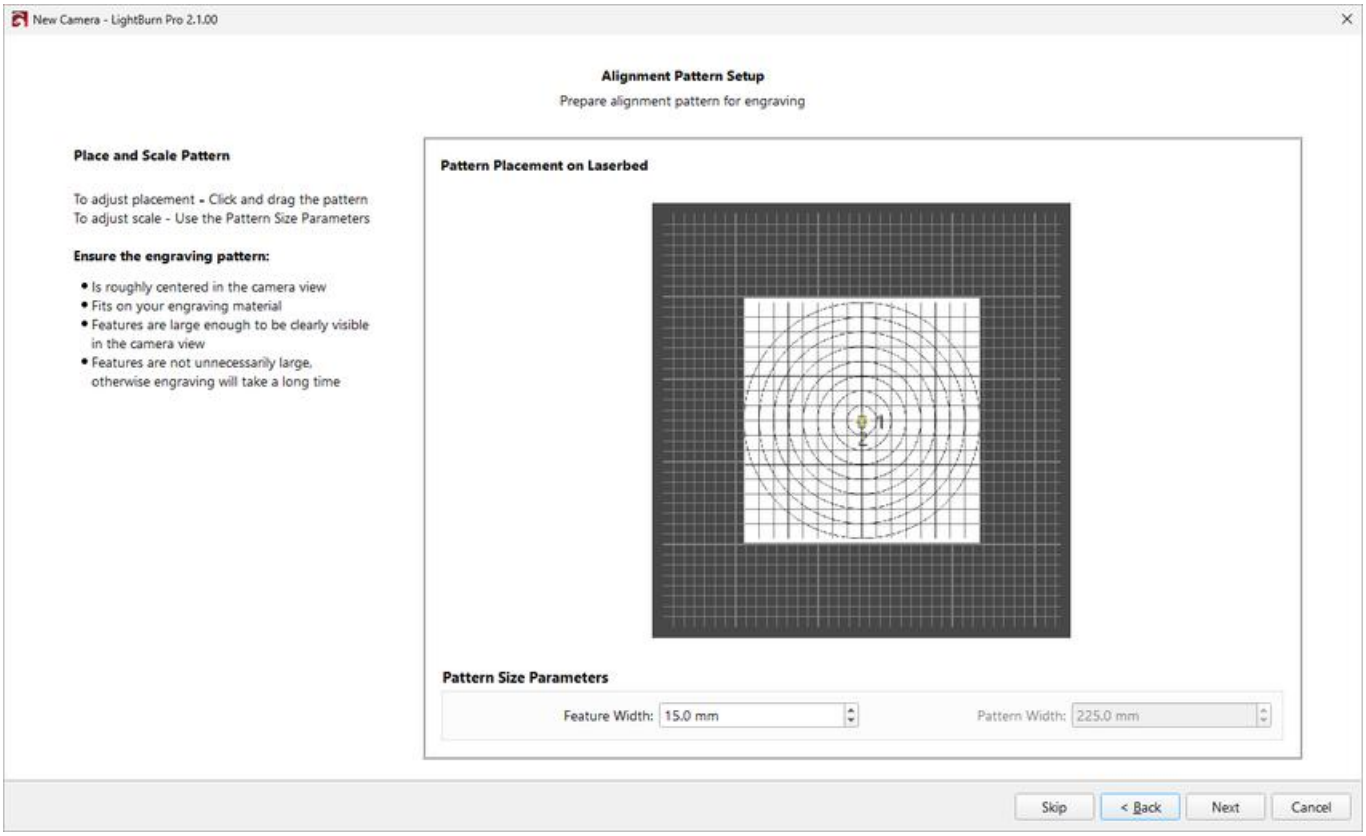
Automatic Alignment Pattern

The automatic alignment pattern uses AprilTags to allow LightBurn to automatically detect the position of the camera relative to the workspace.



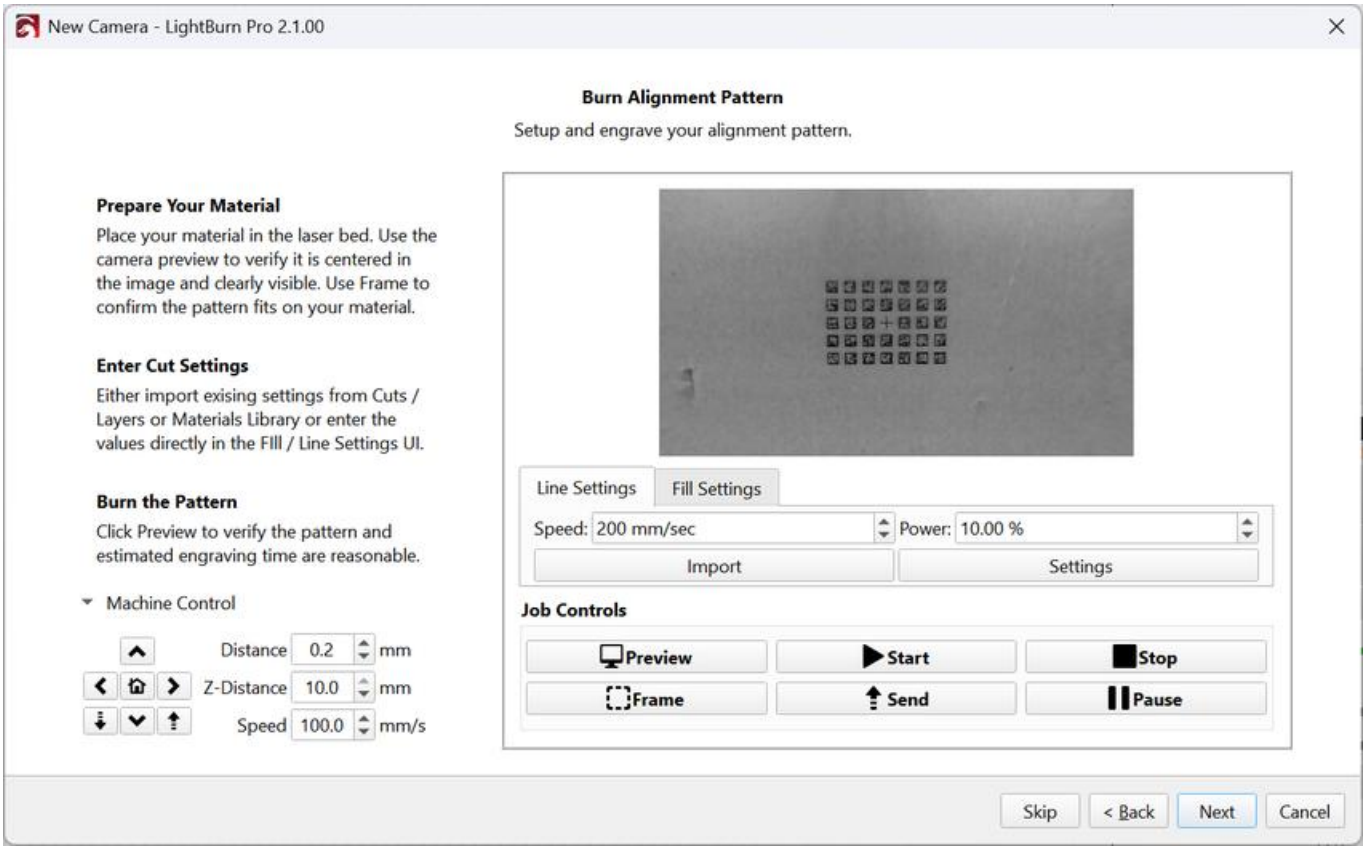
Manual Alignment Pattern

The manual alignment pattern engraves a set of markers for you to identify, and uses them to guide you through aligning the camera to the workspace yourself.



3. Burn Alignment Pattern

This step is the same between automatic and manual alignment, even though the actual pattern is different. Place your material in the laser cutter, make sure it's focused, and get ready to burn the alignment pattern.



Use the camera preview to make sure the material is near the center of the image and clearly visible. Use **Frame** to make sure the pattern fits the material, and use **Preview** to see what will actually be burned. Use the **Machine Control** section in the lower left to move your laser head around if needed.

Adjust your cut settings to engrave a dark, crisp mark without cutting through the material. You can adjust your settings in a few ways:

- Adjust speed and power right in the Alignment Wizard window, just below **Laser Settings**.
- Click the **Import** button to load a layer from your Material Library.
- Click **Settings** to adjust your layer settings in the Cut Settings Editor.

Run the job using **Start** or **Send**, however you'd usually control your laser.

Do not move the pattern after you burn it, or you will need to burn a new alignment pattern.

4. Target Alignment

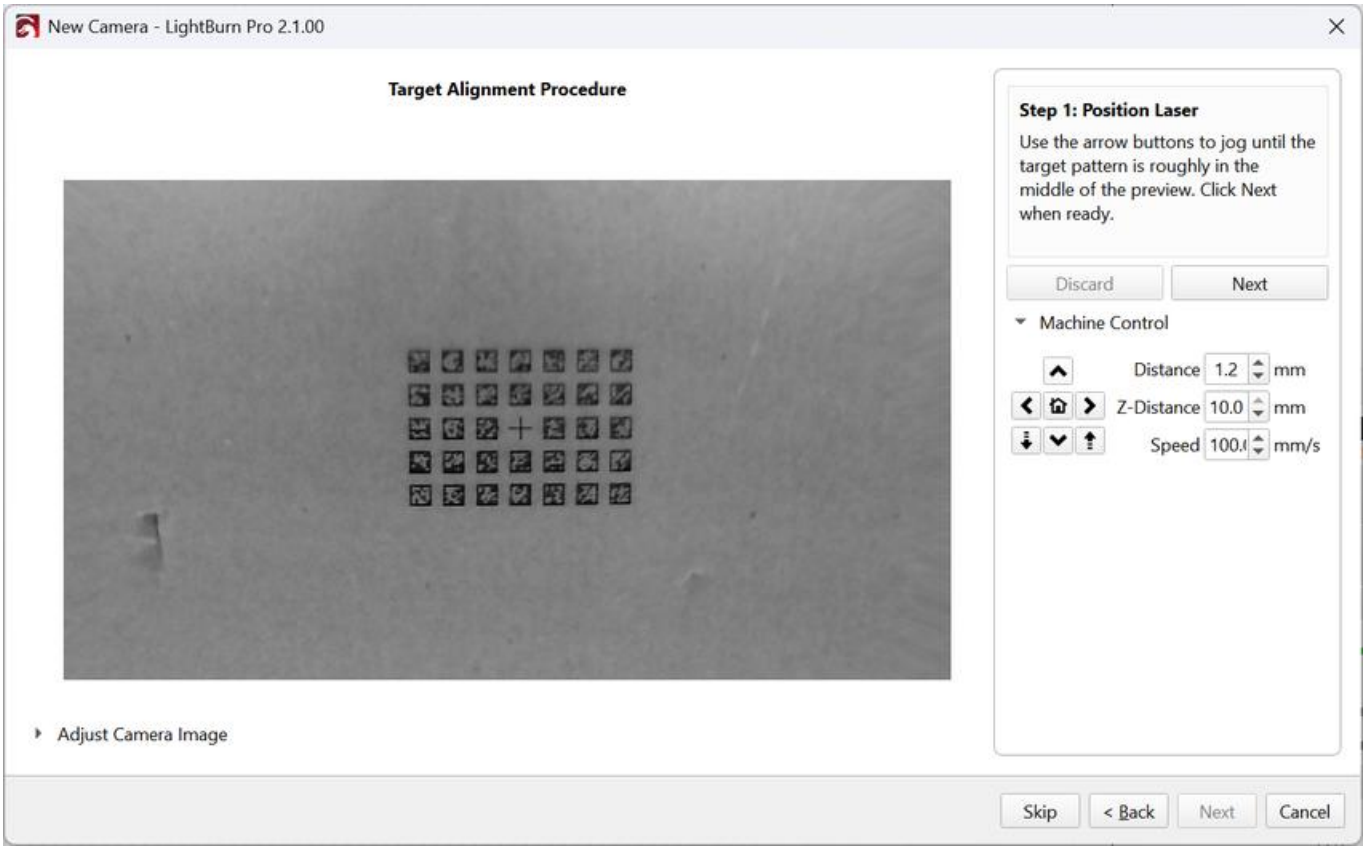
This process is different depending on whether you chose **Automatic Alignment** or **Manual Alignment**.

Automatic Target Alignment

Automatic Alignment uses the camera to read the AprilTags you've engraved.

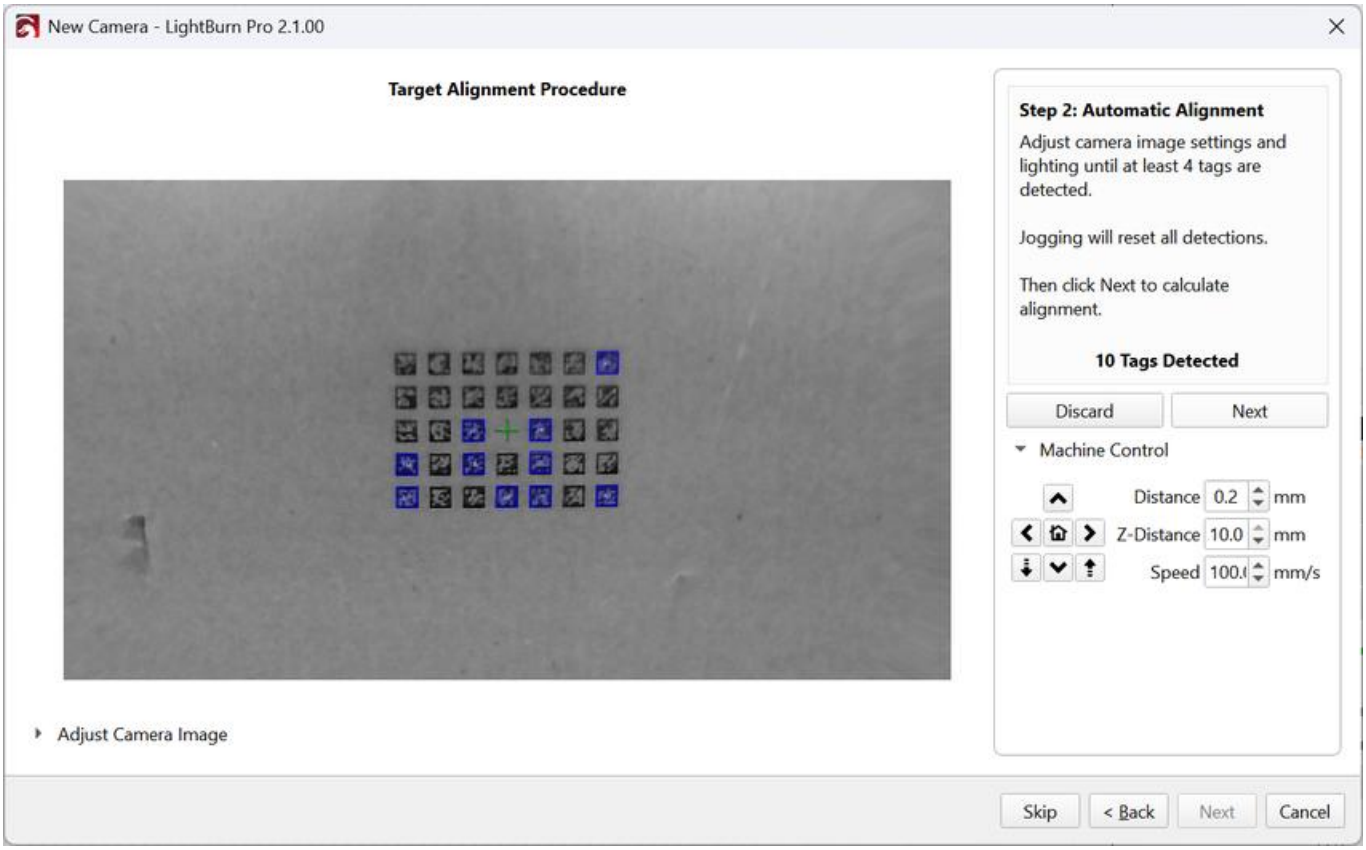
1. Position Laser

Start by using the arrow buttons to jog until the target pattern is roughly in the middle of the camera preview, then click **Next**.



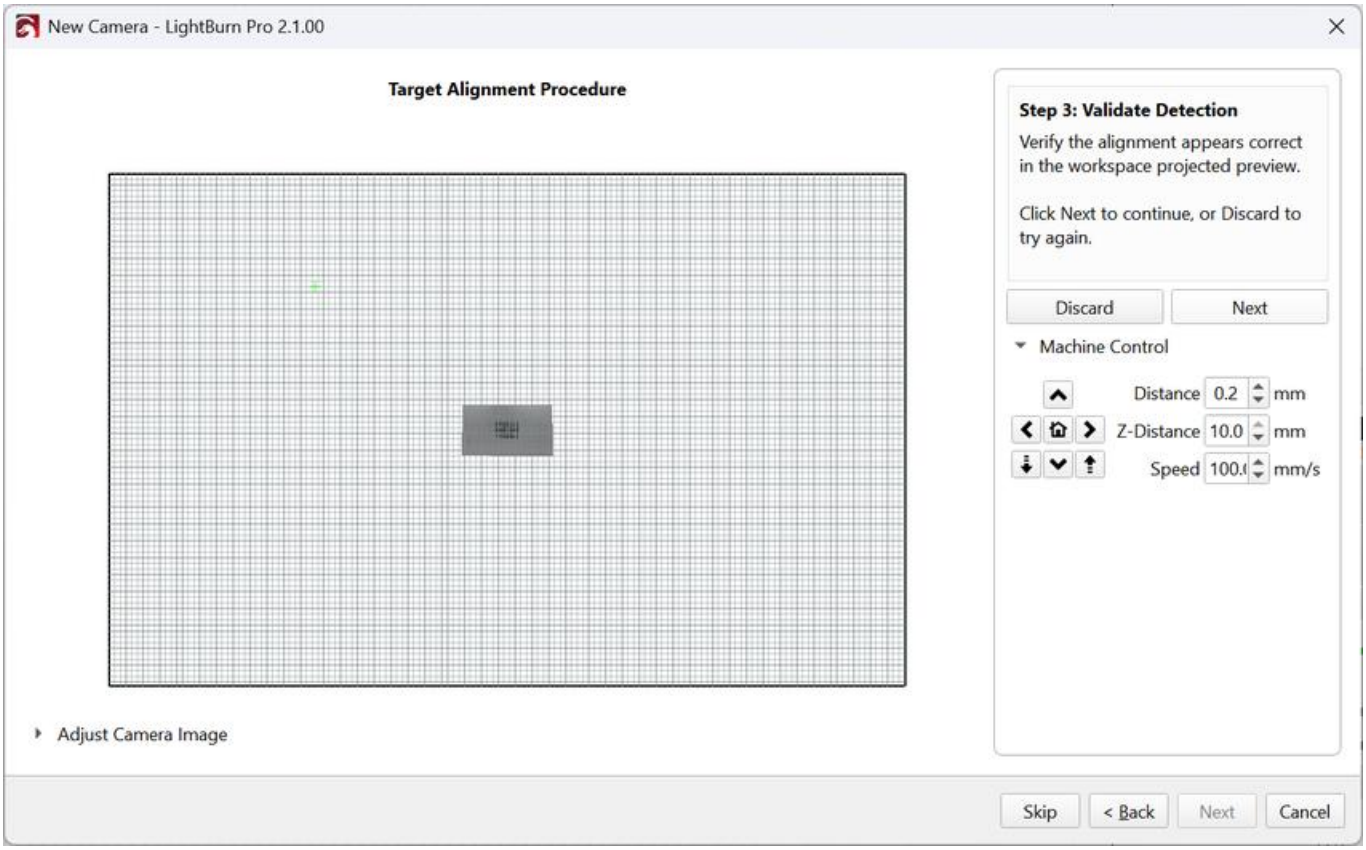
2. Detect Tags

Adjust the camera image settings and lighting until at least 4 tags are recognized. Recognized tags are highlihgted in blue. If you jog the laser, LightBurn will reset the detection of the tags. Click **Next** to calculate the alignment.



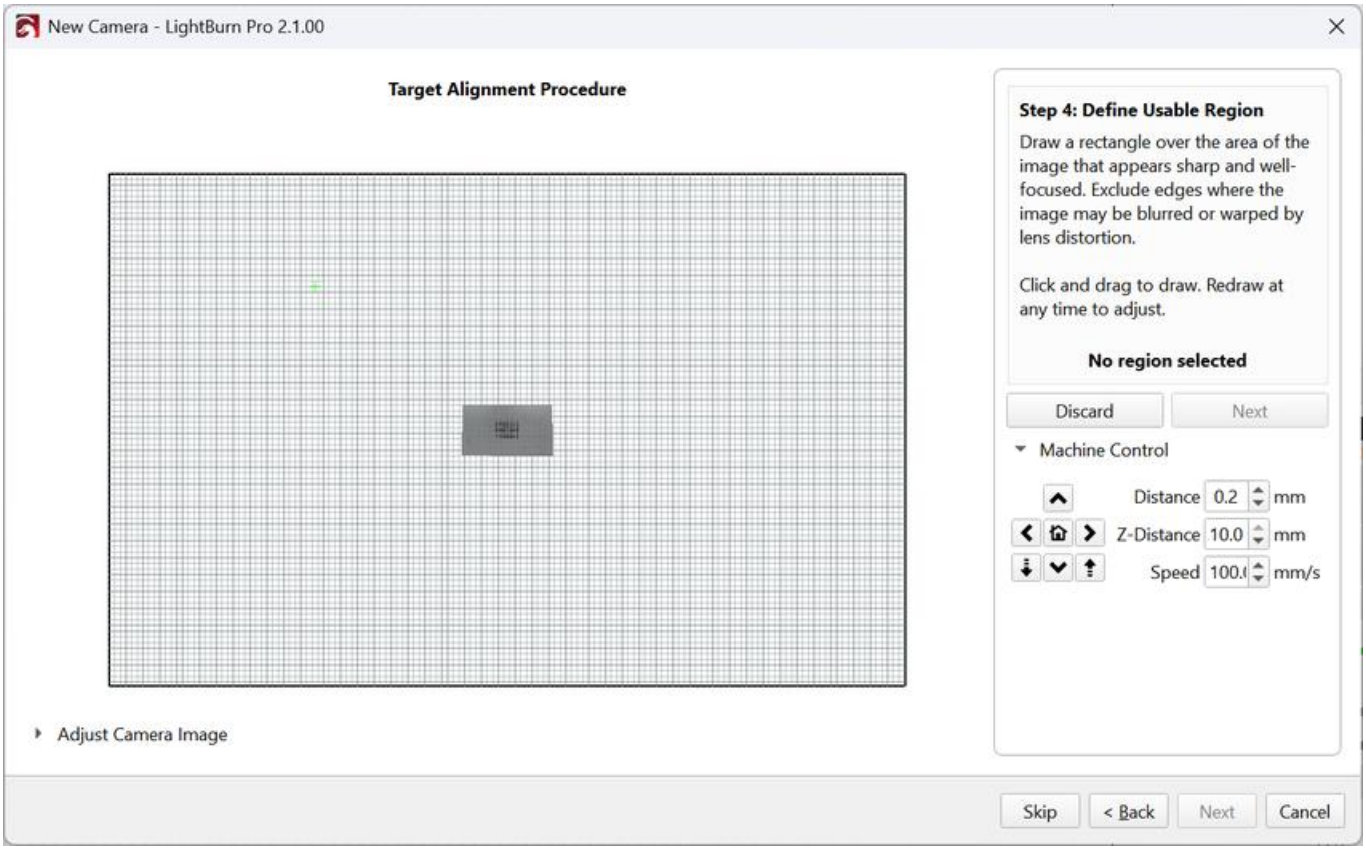
3. Validate Detection

Make sure the alignment appears correct in the projected camera preview. Click **Next** to continue, or **Discard** to try again.



4. Define Usable Region

Click and drag to draw a rectangle over the portion of the image that appears sharp and well-focused. Leave out edges where the image is blurred or warped by lens distortion. You can redraw as much as you want to adjust this. Click **Next** to save the camera alignment, then **Finish** on the final screen.

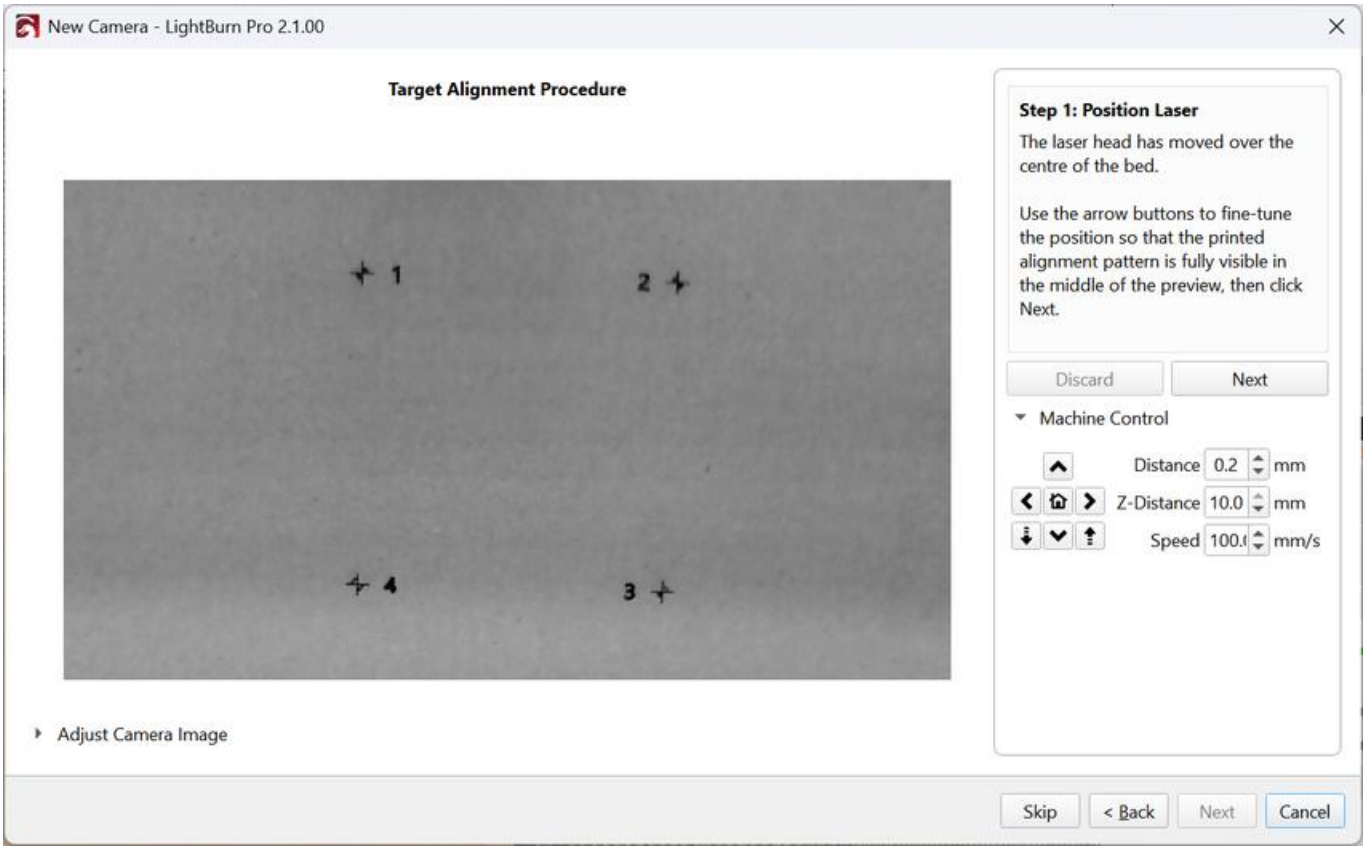


Manual Target Alignment

Manual Target Alignment will walk you through identifying the centers of the targets you've engraved and identifying the portion of the camera's view that's clear and well-focused.

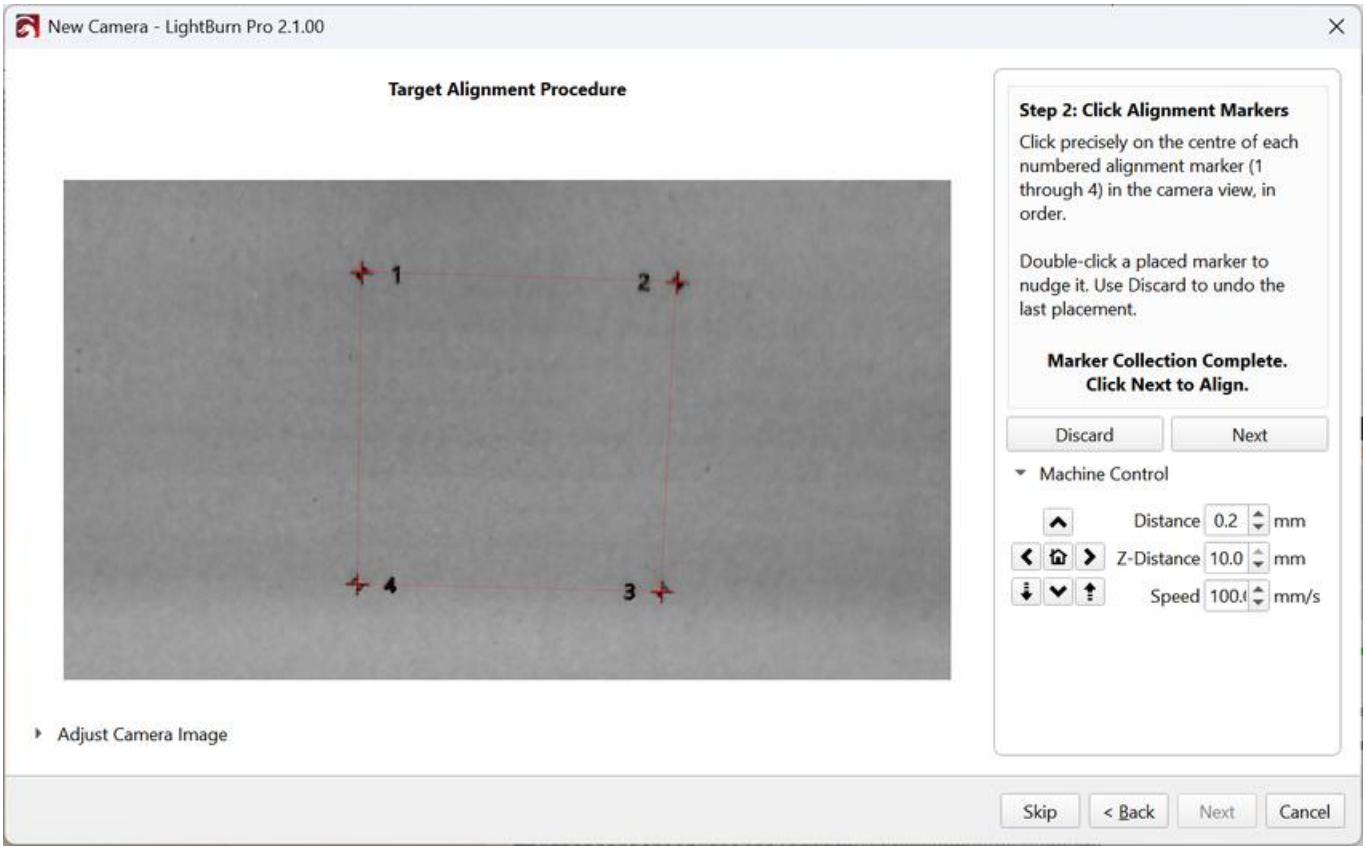
1. Position Laser

Start by using the arrow buttons to jog until the target pattern is roughly in the middle of the camera preview, then click **Next**.



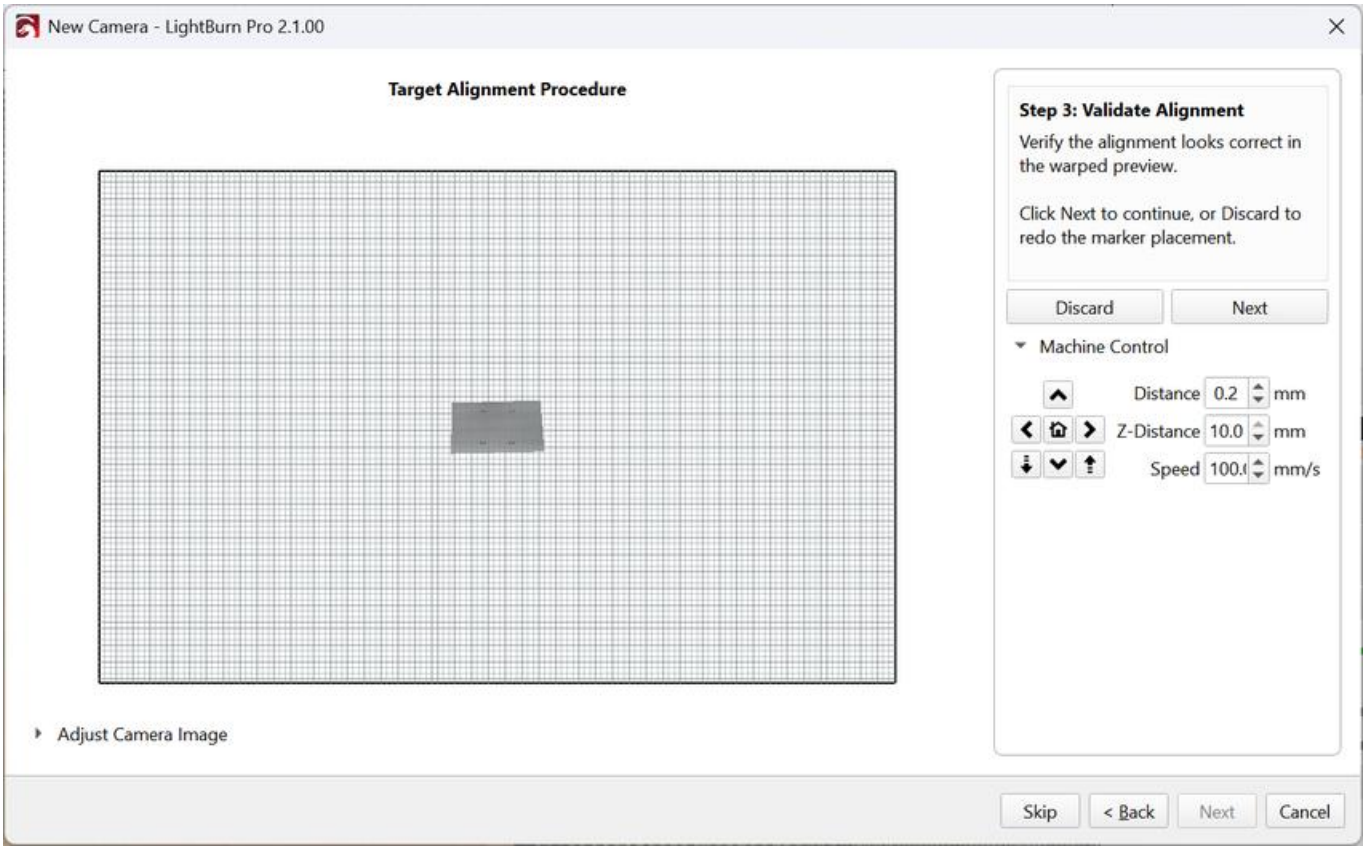
2. Click Alignment Markers

Click precisely on the center of each numbered alignment marker in order, as directed in the instructions. Double-click a placed marker to nudge it if you're slightly off, or click **Discard** to undo the last placement. When you've selected all four, a red box will connect all four markers. Click **Next** to continue and calculate the alignment.



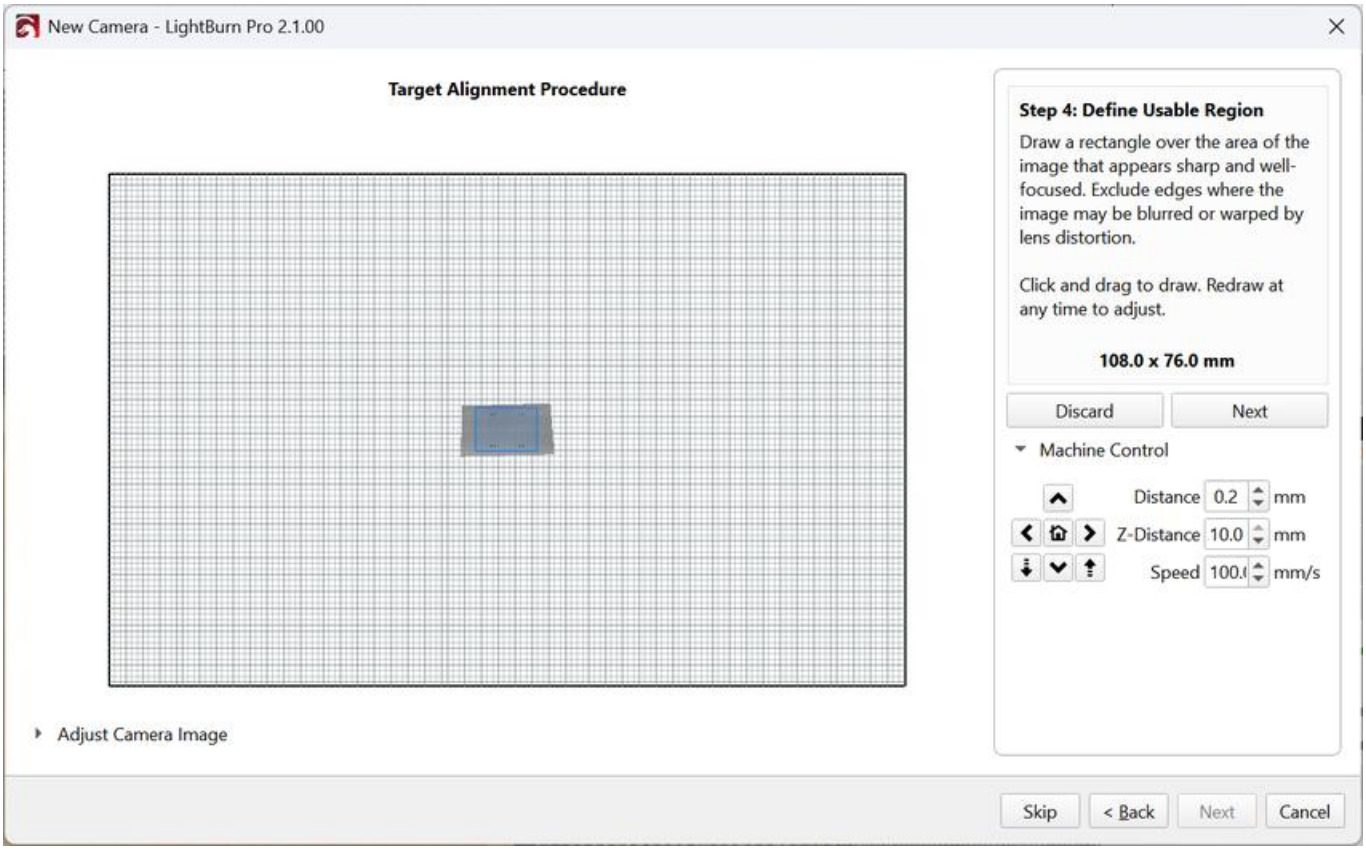
3. Validate Alignment

Make sure the alignment appears correct in the projected camera preview. Click **Next** to continue, or **Discard** to try again.



4. Define Usable Region

Click and drag to draw a rectangle over the portion of the image that appears sharp and well-focused. Leave out edges where the image is blurred or warped by lens distortion. You can redraw as much as you want to adjust this. Click **Next** to save the camera alignment, then **Finish** on the final screen.

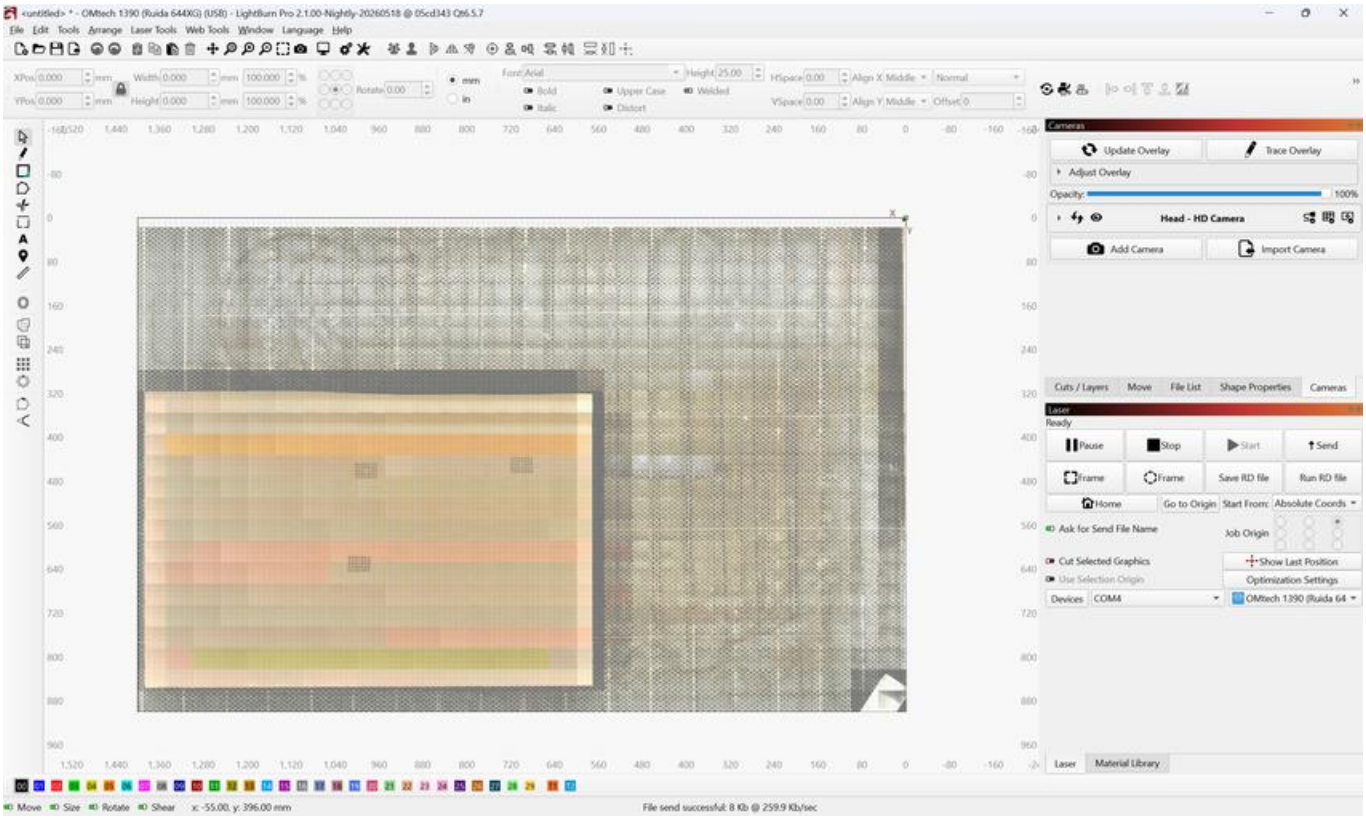


Capturing an Overlay

LightBurn 2.2's new [Cameras / Overlays] window has three options for capturing overlays with your Head Camera.

- Single, which captures a single frame at the laser's current location using your Head Camera.
- Selected, which captures the area around selected shapes using your Head Camera.
- Workspace, which captures the entire workspace.

You can read more about Head Cameras and Overlays in the Cameras / Overlays window.



Troubleshooting

- If the Overlay is misaligned, try rerunning the entire process, taking care to mark the points in the pattern as precisely as possible, and to not move the pattern at all from its original, engraved position.

Related Topics

- Cameras / Overlays Window
- Calibrate Camera Lens
- Calibrate Camera Alignment
- Camera Selection Helper
- Camera Installation and Focusing

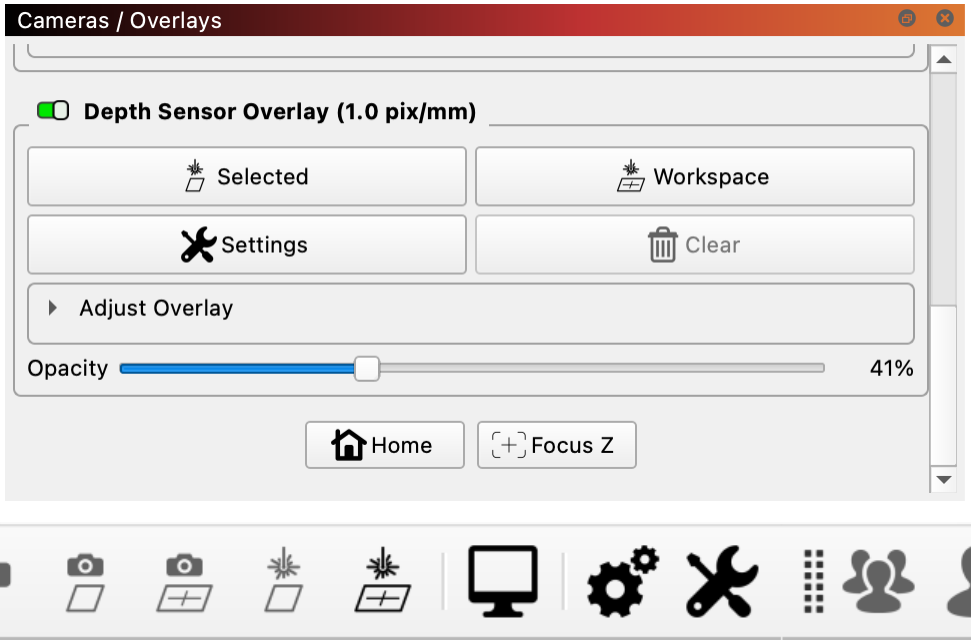
Lidar Cameras

> **Supported Lasers**
>
> LightBurn currently supports LiDAR ranging for ACCELaser HD1 series lasers only. If you have a different model of laser, you will not be able to use this feature.

Depth Scanning, also called **LiDAR Ranging**, uses a sensor to create a height map of the surface of an object or material loaded into your laser.

Using the height map data, the laser is able to dynamically adjust focal distance and engrave on curved surfaces, or objects with multiple surfaces at varying heights.

Using Depth Scanning



Depth Scan controls are available in the Cameras / Overlays Window and Main Toolbar. You can choose to scan the laser's entire bed, or an area defined by objects currently in your selection.

After scanning, you'll see an overlay of the height map data in your Workspace.

After performing a LiDAR scan, make sure to enable **Include Lidar Data** in the Cut Settings Editor. With this switch enabled, the laser will dynamically adjust focal height to account for the scanned height of the surface.

> **Warning**
>
> Before an ACCELaser H1 series laser performs a LiDAR scan, it first focuses to the center of your Selected Area or Full Workspace. Make sure your object is properly centered in the scan area — if nothing is detected on the bed at that location, the head may crash into objects as it attempts to scan.

Depth Sensor Overlay

Toggle this switch to show or hide the height map overlay in your Workspace.

Scan Selected Area

Commands the machine to run a LiDAR scan over an area defined by the position and boundaries of shapes currently selected in your Workspace.

This option will be grayed out unselectable if you have no objects in your Selection prior to opening the LiDAR window.

Typically, you'll want to capture a Camera Overlay, create a Rectangle set to a Tool Layer, and then place it over the material you plan to cut or engrave.

Scan Full Workspace

Commands the machine to run a LiDAR scan over your laser's entire work area.

Clear

Clears the data captured from the previous LiDAR scan.

LiDAR Settings

Click the **Settings** button to open a window with scan settings options.



Scan Interval

Scan Interval controls the size in X and Y of each pass of the LiDAR ranging process.

Smaller Intervals will lead to higher resolution height maps, at the cost of longer scanning times.

Smooth

Interpolates scanning data from adjacent areas to smooth height map data between segments.

Adjust Overlay

Overlay Width / Height

Adjust the size of the height map overlay.

X Shift / Y Shift

Shift the overlay image in the X and/or Y directions, if the entire overlay is shifted over from where it should be.

Overlay Opacity Slider

This slider controls the transparency of the height map overlay. Reduce opacity for a clearer view of your Camera Overlay, or graphics in your Workspace.

Save Background Capture

Use **Save Background Capture** to save a bitmap image of a camera overlay produced by a camera mounted over your laser's work area.

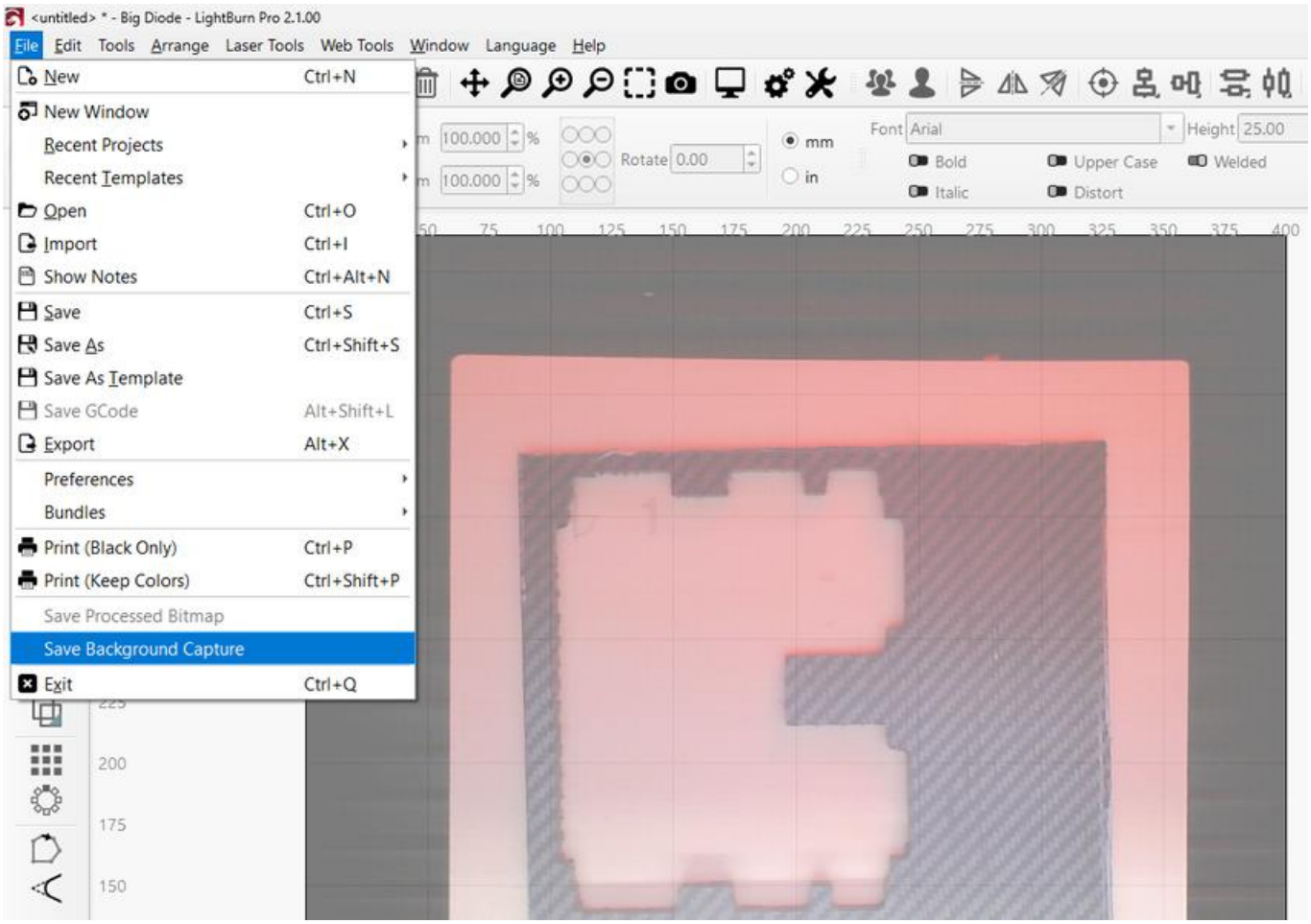
Saving Background Captures

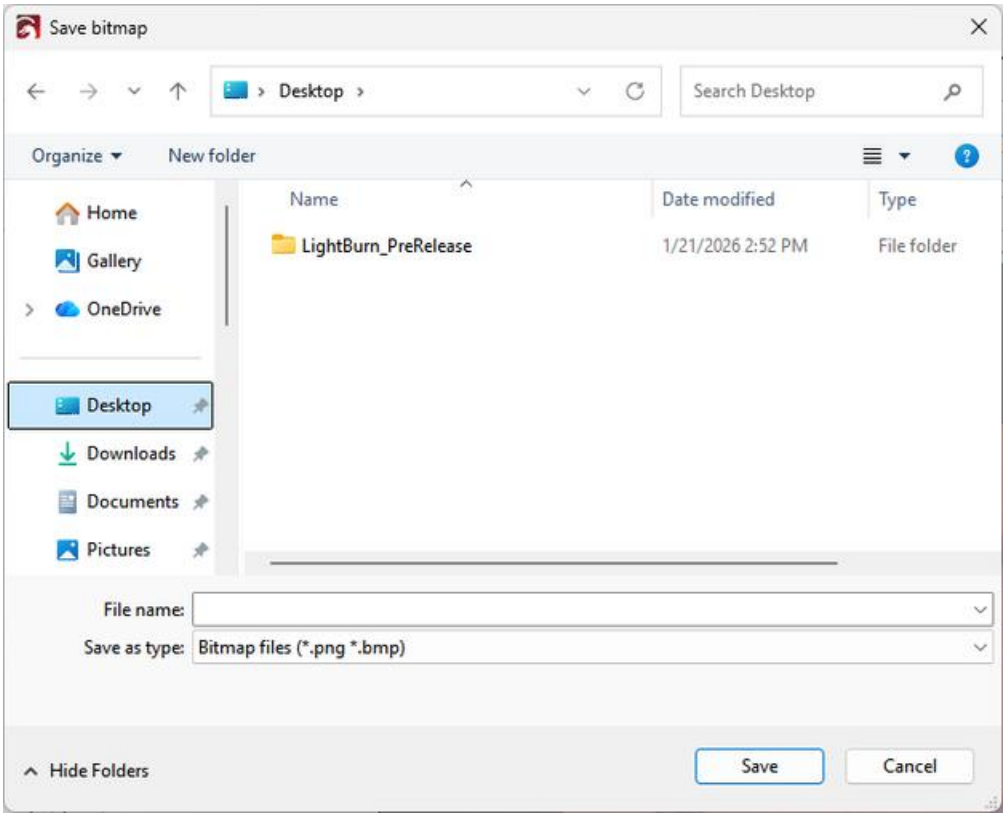
In order to use **Save Background Capture** you must first properly Calibrate and Align your camera.

Click **Update Overlay** in the Cameras / Overlays Window to produce an overlay.

Go to File → **Save Background Capture** to save the camera overlay.

A dialog window will open, allowing you to name and navigate to the location you'd like to save the capture.





Related Topics

- Cameras / Overlays Window
- Calibrate Camera Lens
- Calibrate Camera Alignment
- Camera Installation and Focusing
- Save Processed Bitmap

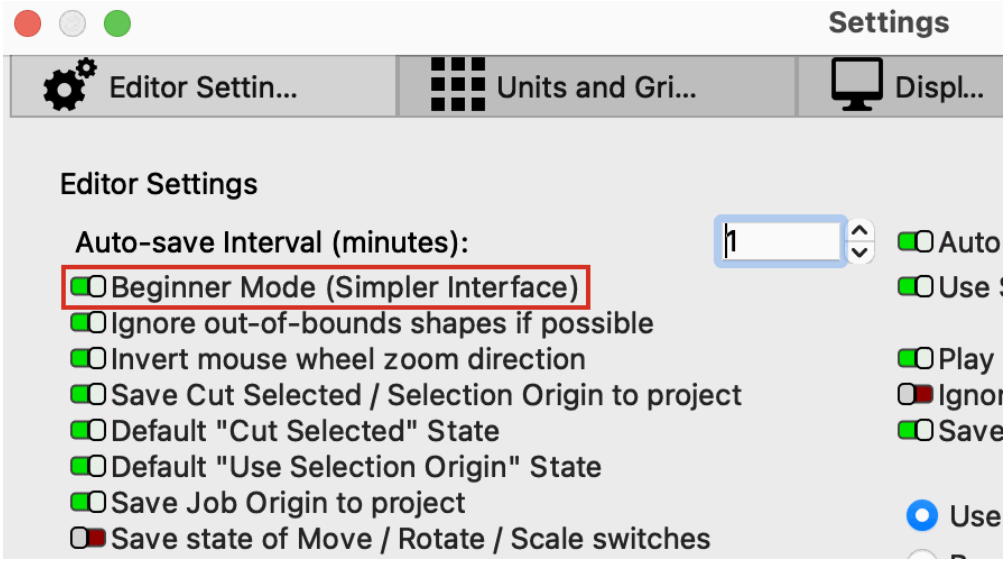
Settings and Preferences

Beginner Mode

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=QatjALP3QIk

Beginner Mode simplifies the LightBurn interface by removing several options that new users are unlikely to use.

You can enable or disable Beginner Mode at any time in the **Editor Settings** tab of the Settings window.



When Beginner Mode is active, you'll see a message in the title bar, next to the location where your currently selected device profile is indicated.



Note

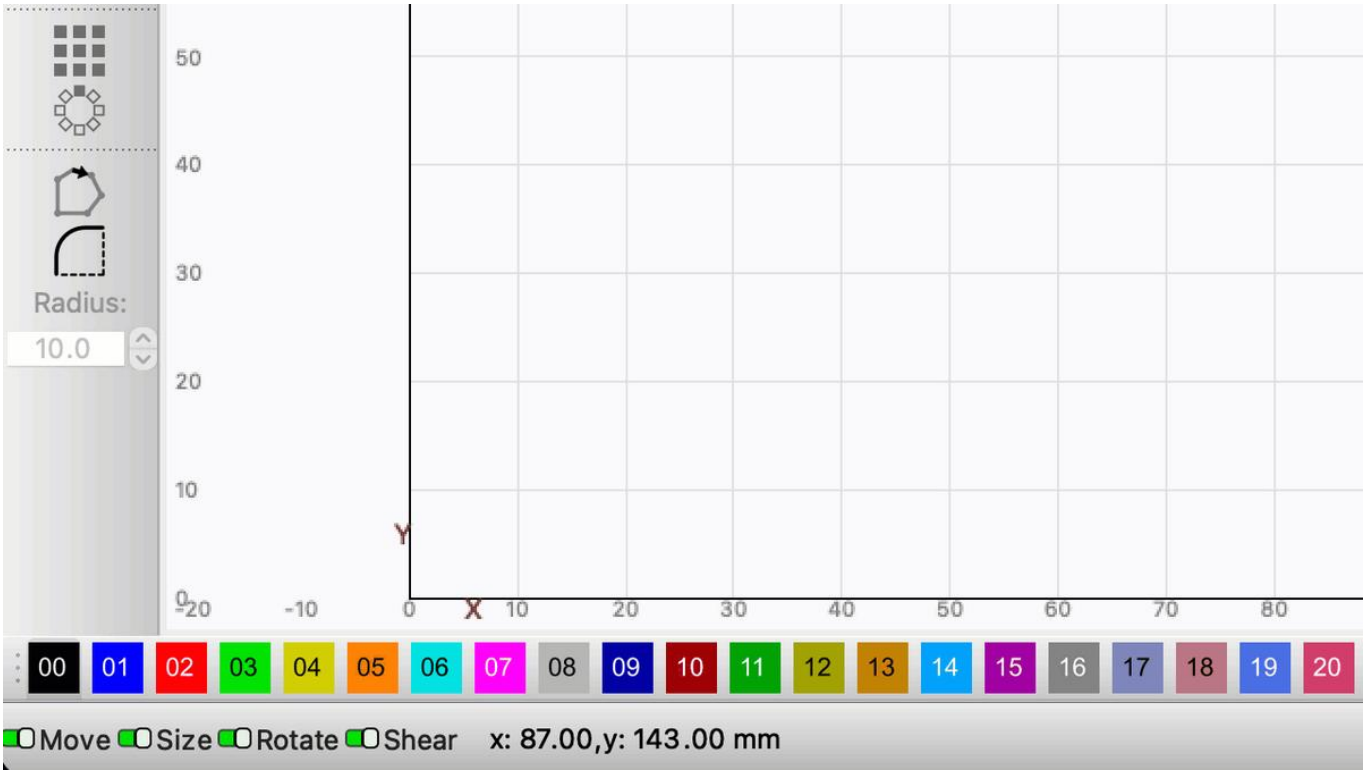
Although Beginner Mode can make it easier to get started, it also means that your interface will look a little different than most video tutorials or screenshots you'll see.

If you're searching for a button or features and you don't see it, it might be because of Beginner Mode, but it could also be because you're using a machine that doesn't support the feature you're looking for. LightBurn's interface adapts to only show you the features that are compatible with your machine type.

Beginner Mode Changes

Transform Control Toggles

- The toggles for Move, Rotate, Shear and Scale are hidden.

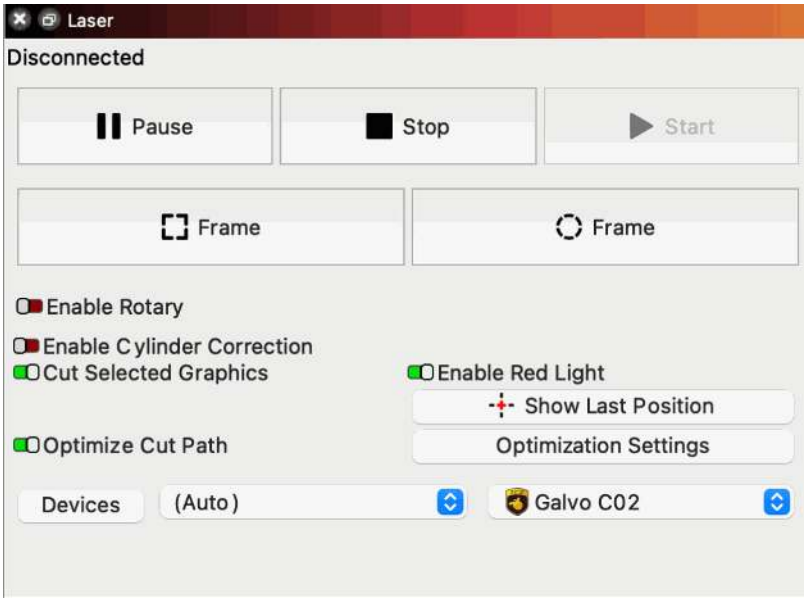


Laser Window

The following buttons and options are hidden in the Laser Window:

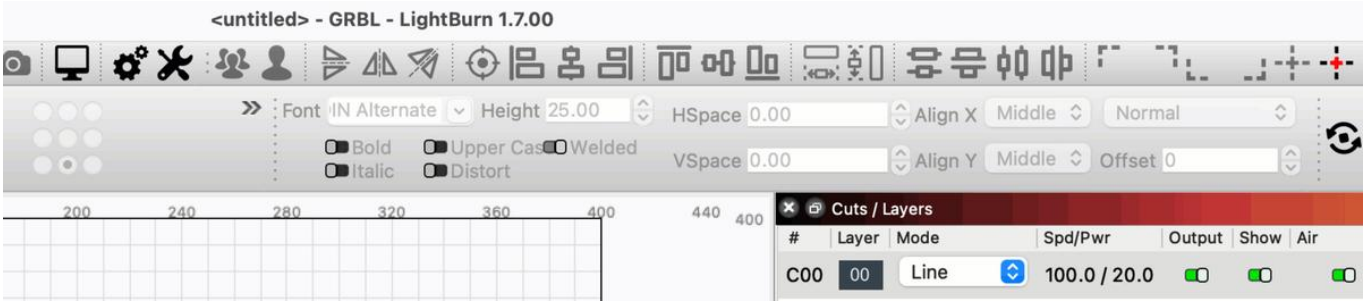
- Save Machine File
- Run Machine File
- Optimization Settings
- Cut Selected Graphics
- **Enable Rotary** and **Enable Cylinder Correction**

> **Note**
>
> These options only appear if **Show rotary enable on main window** and **Show cylinder correction enable on main window** in your Settings. Enabling **Beginner Mode** hides them, but disabling Beginner Mode does not show them, by itself.



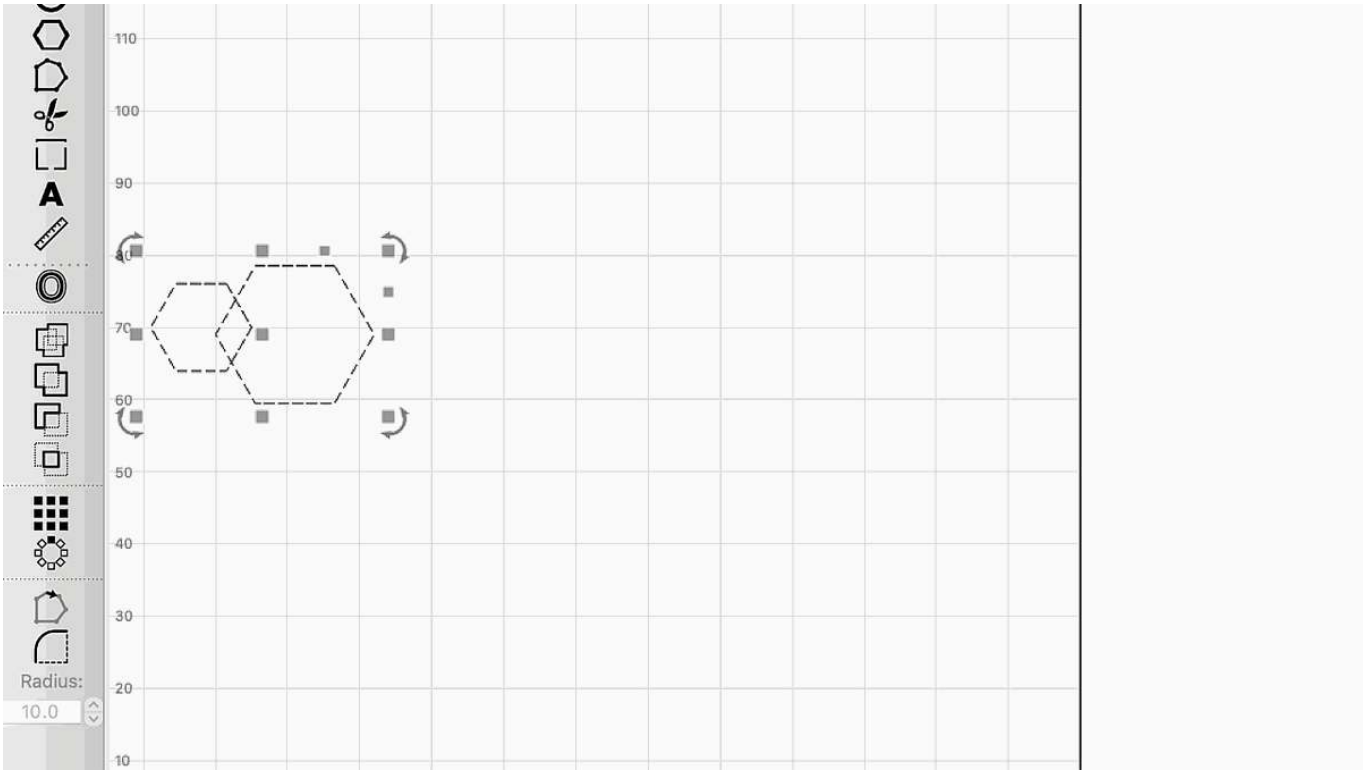
Arrange Toolbar

- The simplified version of the Arrange Toolbar is automatically selected.
- The Move Selected Objects tools appear in a dropdown menu.
- The Distribute tools are hidden.
- Mirror Across Line is hidden.
- The **Align** tools are replaced with the Align Tools Helper. Click the Align left icon to open a dialog window where you can select any Align option and see the result before clicking **OK** to confirm the changes, or **Cancel** to discard them.



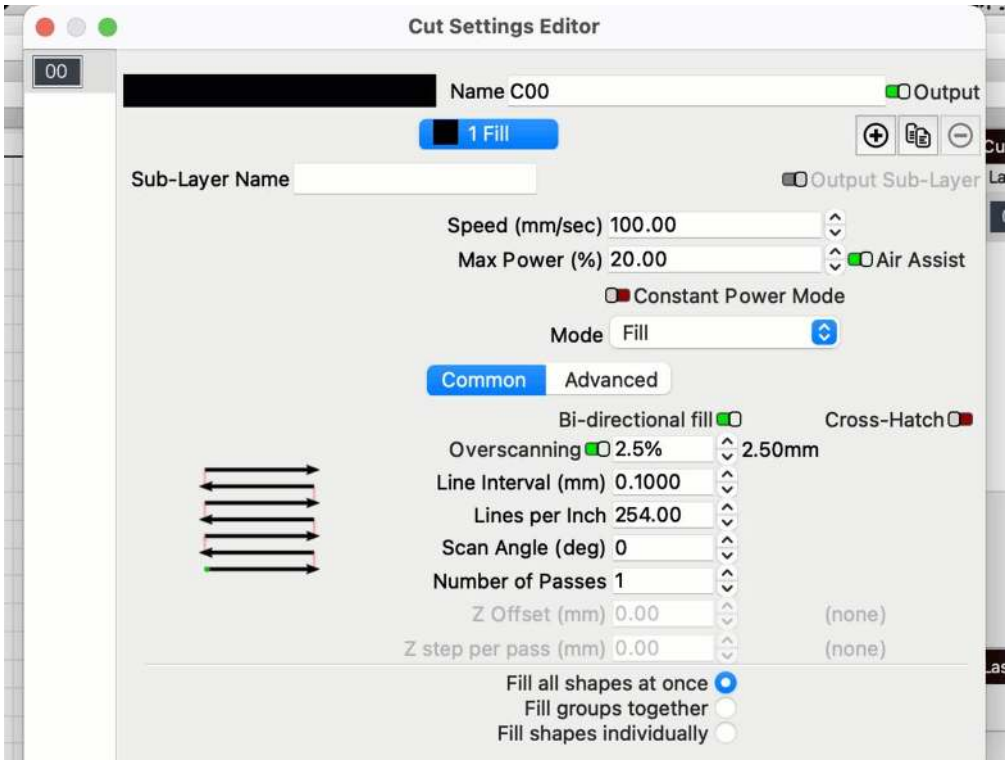
Boolean Assistant

- In the Modifiers Toolbar, the Boolean tools are replaced with the **Boolean Assistant**. Click the **Boolean Union** icon to open a dialog window where you can select any Boolean option and see the result before clicking **OK** to confirm the changes, or **Cancel** to discard them.



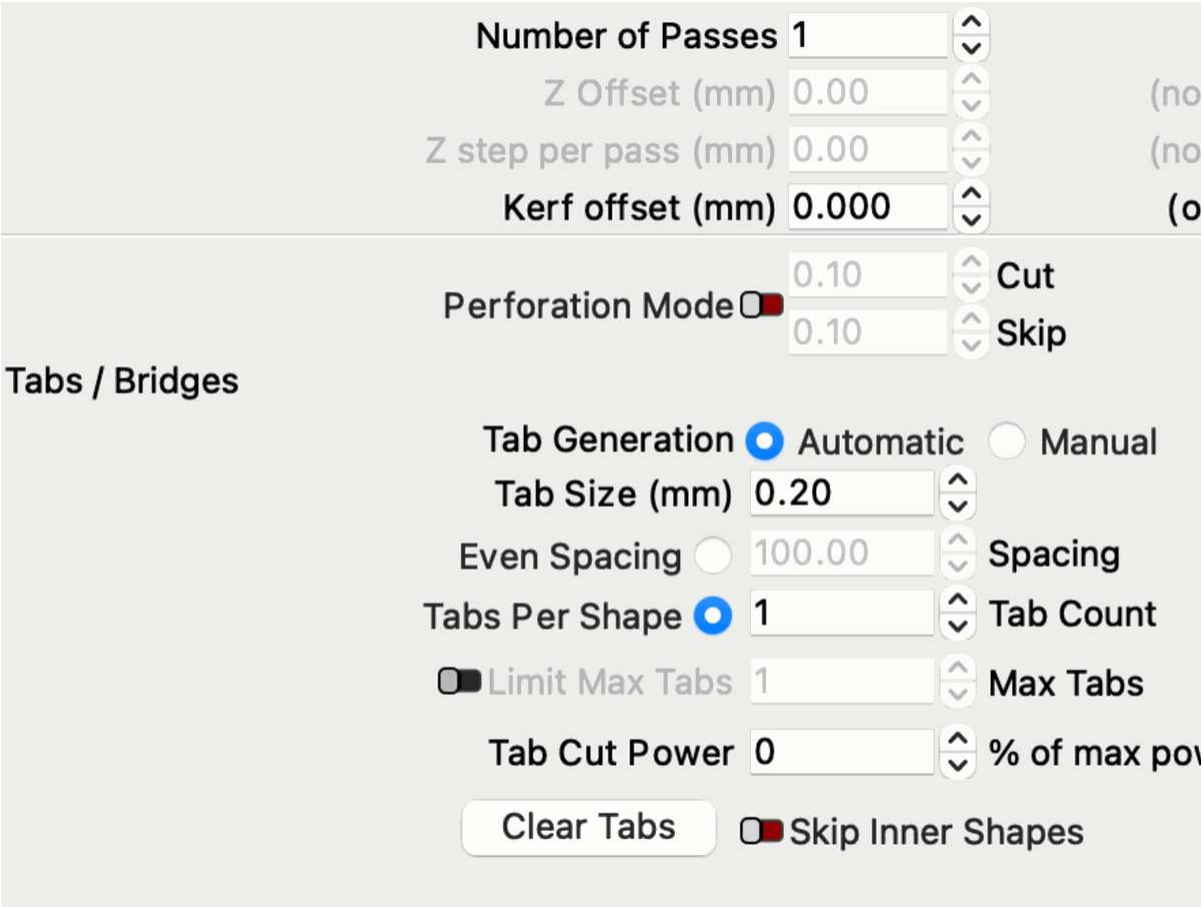
Cut Settings Editor

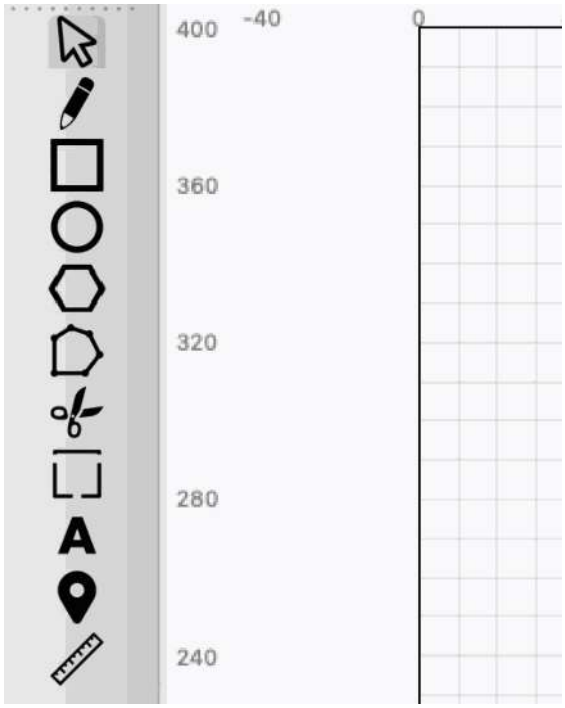
- All advanced setting are hidden in the Cut Settings Editor. Advanced settings are ignored in cases where enabling them is optional, and defaults are applied in cases where a selection is required.
- **Fill** layers are always set to **Bi-directional fill** and **Fill all shapes at once**. **Scan Angle** is set to 0°.



Tabs

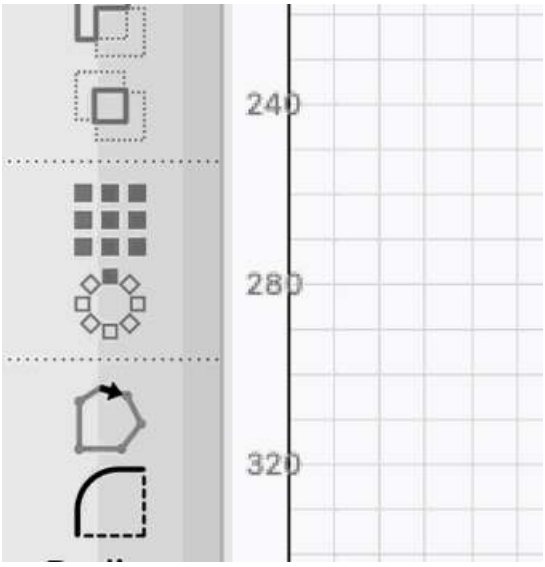
- The **Tabs / Bridges** section of the Cut Settings Editor for **Line** layers is hidden.
- The Add Tabs tool is removed from the Creation Toolbar.





Set Start Point

- The Set Start Point tool is hidden in the Modifiers Toolbar.



Related Topics

- Get Started
- Main Window

Settings / Preferences

The **Settings** window contains global user preferences for LightBurn — these settings apply regardless of what lasers you have set up, and which one you currently have selected.

The Settings window is split into five tabs:

Editor Settings

Units and Grids

Display

Import / Export

Camera

Accessing the Settings Window

To open the **Settings** window, click the gears icon in the top toolbar, or go to Edit → **Settings**.

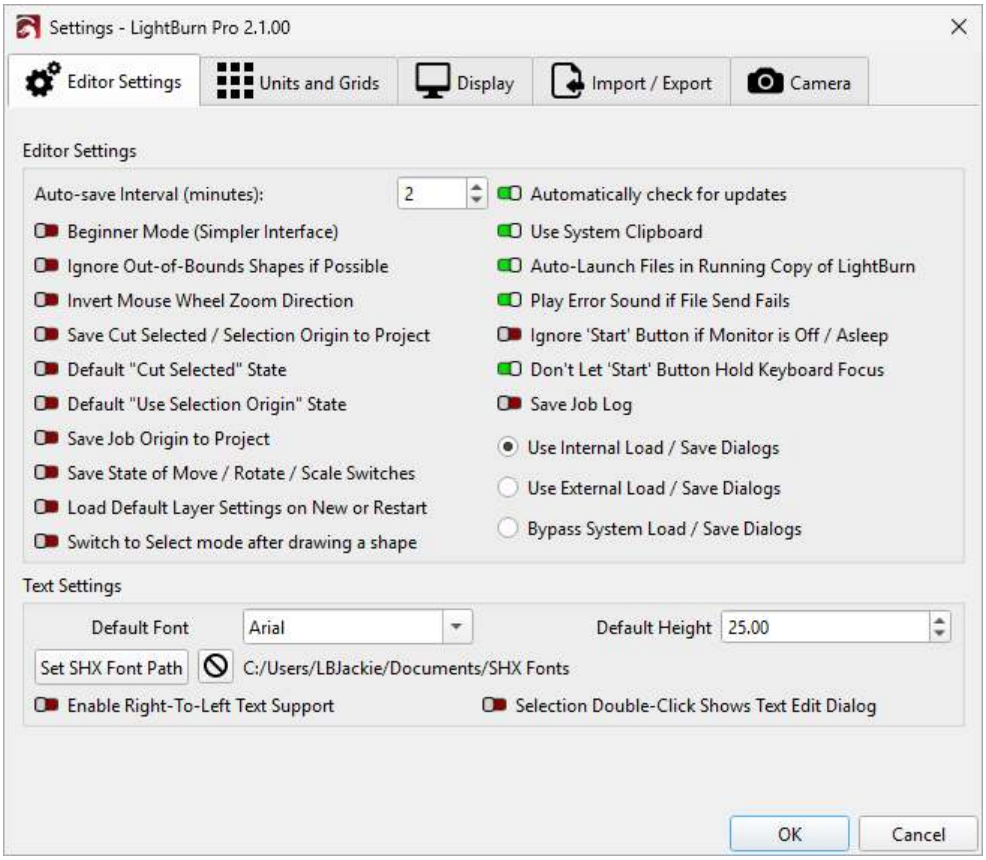
If you're on macOS, go to **LightBurn** → **Preferences** instead.

Settings Window

Click any option in the images below to jump directly to the relevant section for that option, or scroll down for an alphabetized list of options and descriptions for each tab.

Click a tab to jump to the relevant section for that tab and the settings it contains.

Editor Settings



Automatically check for updates

Permits (or denies) LightBurn the ability to automatically check for updates, and inform you of them.

> **Beta, or Release Candidate User?**
>
> Beta and Release Candidate versions of LightBurn will ignore the **Automatically check for updates** toggle. Users testing early builds will always be prompted to update to the latest testing release available to ensure that the version of LightBurn being tested is the most current candidate.

Auto-save Interval (minutes)

Sets the frequency of auto-saves, in minutes. Setting the **Auto-save Interval** to 0 disables auto-saving.

Auto-saves are stored in the same location as you saved the original file, with `_backup` appended to their name. If you've never named a file, the auto-save will be in your computer's Documents folder.

Each auto-save overwrites the previous auto-save, and when you manually save the file, the auto-saved copy is deleted.

Auto-launch files in running copy of LightBurn

When opening projects through your system's file explorer, automatically open any LightBurn project files on your computer within a running copy of LightBurn, rather than opening an additional copy of the program.

Beginner Mode

Enables a simplified version of the LightBurn interface which removes several options that new users are unlikely to use.

See Beginner Mode for more information.

Default "Cut Selected" State

Automatically enables **Cut Selected Graphics** each time you restart LightBurn.

Default "Use Selection Origin" State

Automatically enables **Use Selection Origin** each time you restart LightBurn.

Ignore out-of-bounds shapes if possible

When Starting or Sending projects to your laser, tells LightBurn not to send graphics positioned outside the grid in your Workspace.

Graphics that are partially inside and partially outside the grid will be ignored.

Ignore Start button if monitor is off or asleep

Prevents you from accidentally starting the laser by pressing the wrong key when waking the computer. This is particularly useful on lasers that don't have enclosures with safety interlocks, such as most diode and Galvo lasers.

Don't Let Start Button Hold Keyboard Focus

Prevents you from accidentally starting the laser by pressing the wrong key while using other software with LightBurn running in the background. This is particularly useful on lasers that don't have enclosures with safety interlocks, such as most diode and Galvo lasers.

Invert mouse wheel zoom direction

Overrides the default scrolling behavior within LightBurn to match preferences for macOS or Windows.

Load default layer settings on new or restart

Cut settings are normally pre-filled with the last settings used for a given layer. If **Load default layer settings on new or restart** is enabled, any time you open up a new project or restart LightBurn, all settings for all layers will be restored to defaults.

If you've never set up your own defaults, all layers will be restored to LightBurn's universal defaults.

See Working With Default Layer Settings for more information.

Load / Save dialogs

These settings permit you to use non-native system dialogs to perform **Save/Open/Import/Export** operations with LightBurn.

- **Use internal Load / Save dialogs** uses LightBurn's internal code to load and save files.
- **Use external Load / Save dialogs** opens a second application to load and save files.
- **Bypass system Load / Save dialogs** bypasses your system's Load / Save dialogs.

Try external Load / Save dialogs or Bypass system Load / Save dialogs if LightBurn crashes when performing Save/Open/Import/Export operations.

Play error sound if file send fails

Produces an audible chime if uploading a file to a DSP controller fails.

Save Cut Selected / Selection Origin to project

Stores the status of the **Cut Selected Graphics** and **Use Selection Origin** switches to any project you save.

Save Job Log

Instructs LightBurn to save a log of projects that are run, on what laser profile, and at what time.

It will save a file called LightBurnJobs.csv in your system's Documents folder.

Save Job Origin to Project

Stores your selected Job Origin with any project you save.

Save state of Move / Rotate / Scale switches

Stores the state of the Transform Control Toggles so they persist between restarts of LightBurn.

Switch to Select mode after drawing a shape

Enable this to automatically switch back to Select mode after drawing a shape. When disabled, the drawing tool will remain active after completing a shape.

Text Settings

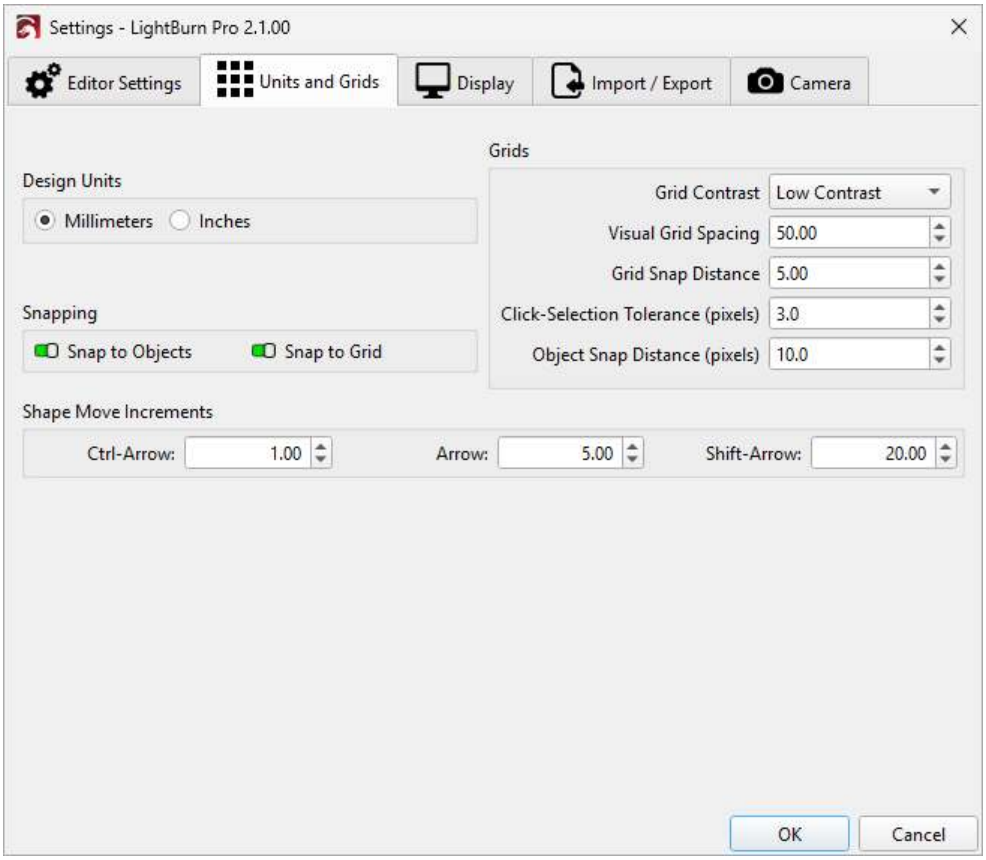
See Create and Edit Text — Text Settings for information on these settings.

Use System Clipboard

Enabling this makes Copy and Paste operations slightly slower, but allows LightBurn to:

- Copy and Paste across different runs of LightBurn, or between two running copies of the app
- Paste images copied from other software or web browsers
- Paste text directly into the Workspace, auto-creating a Text Object

Units and Grids



Click Selection Tolerance (pixels)

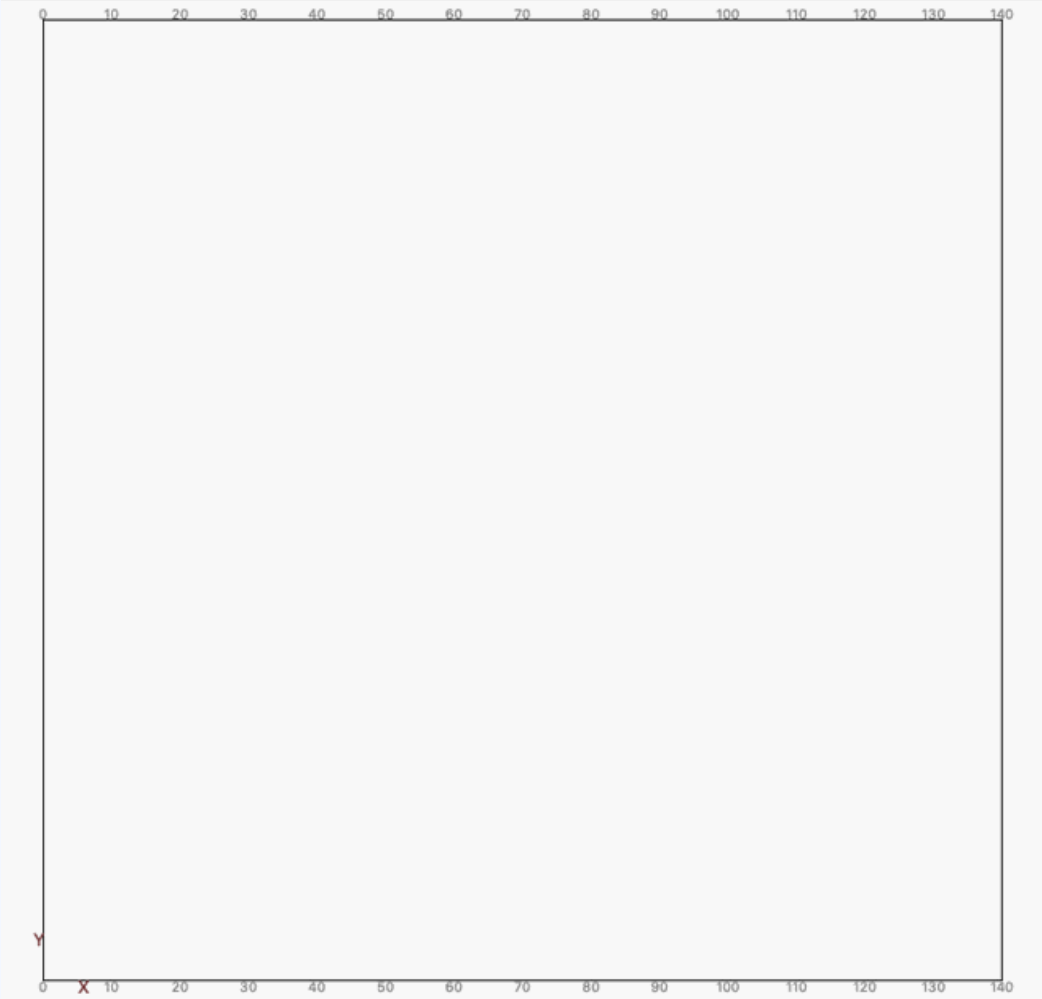
Determines how close your cursor must be to a line or vertex, in screen pixels, to click it.

Increase this number if you have trouble selecting things, decrease it if you find yourself selecting things you didn't mean to.

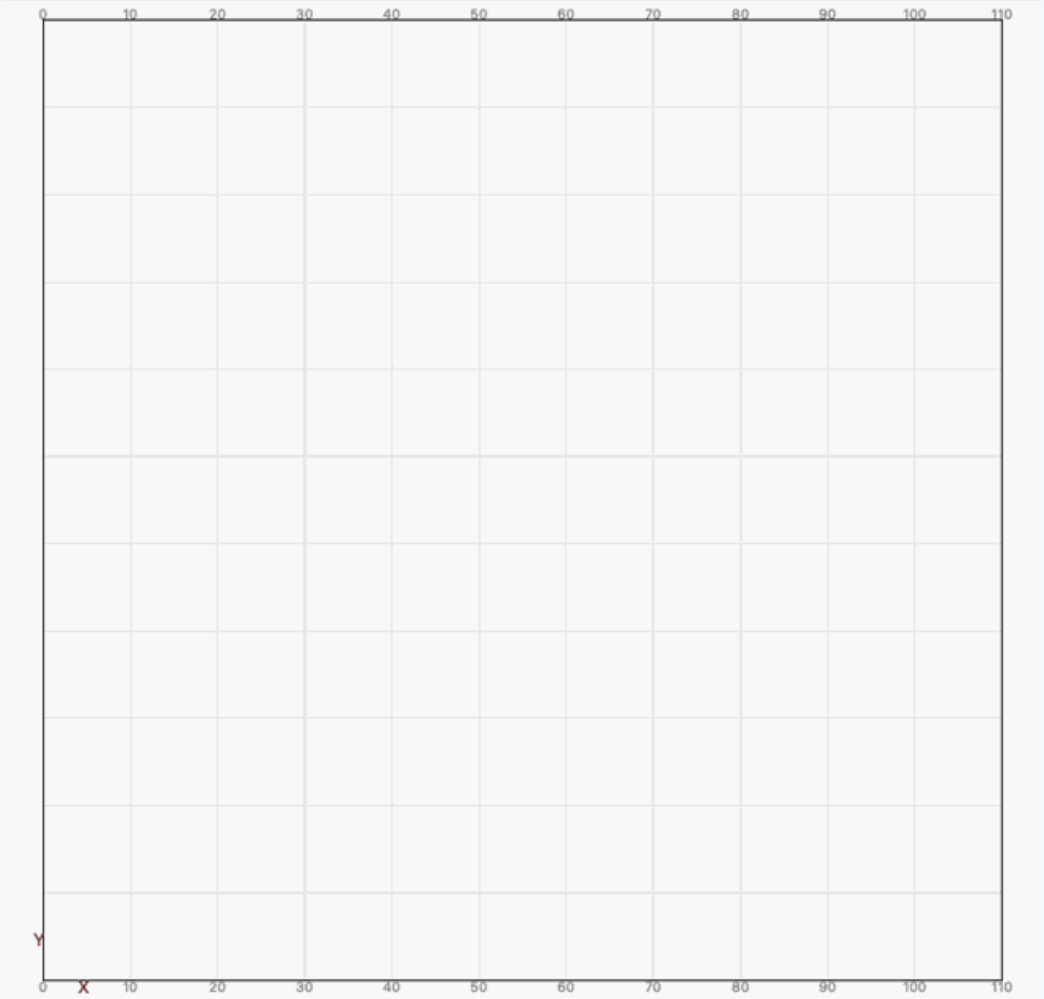
Grid Contrast

Adjust the brightness of the grid in your Workspace, relative to the background.

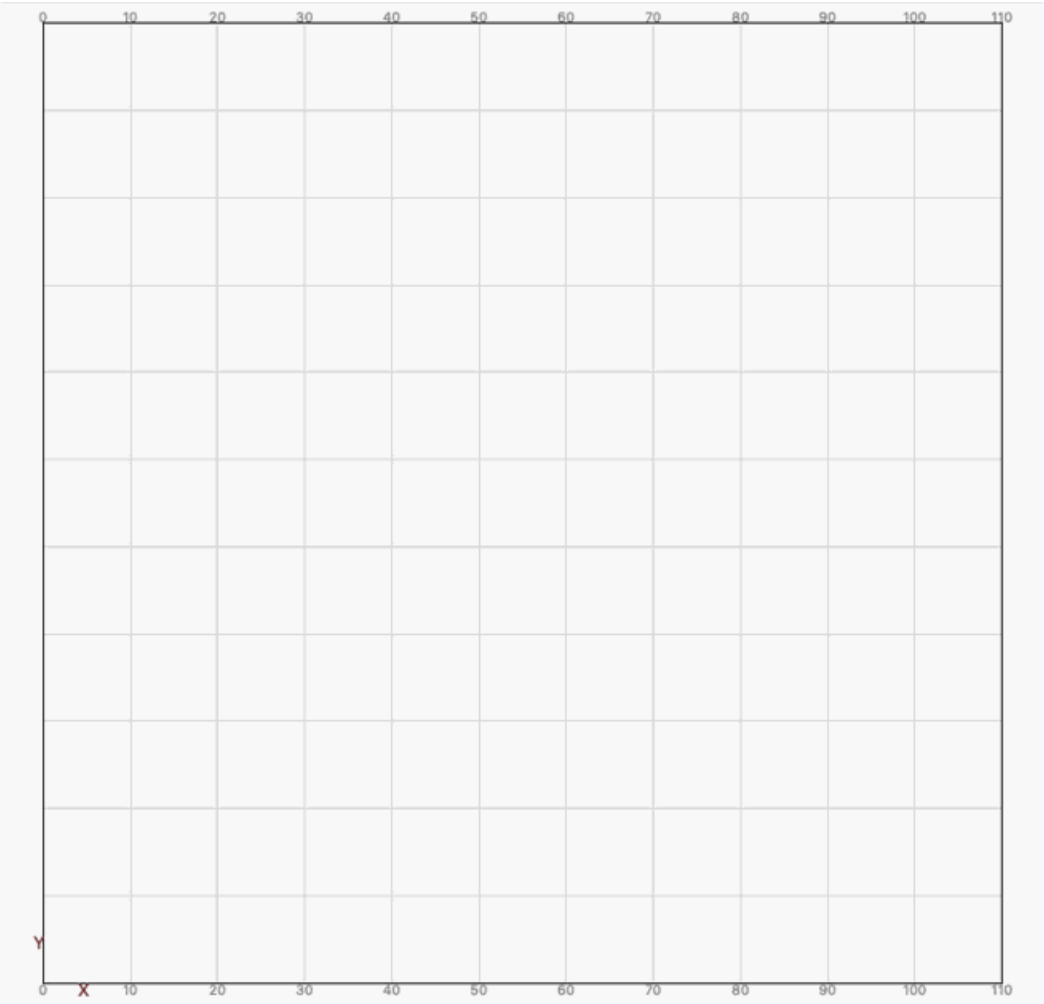
Grid Contrast Comparison



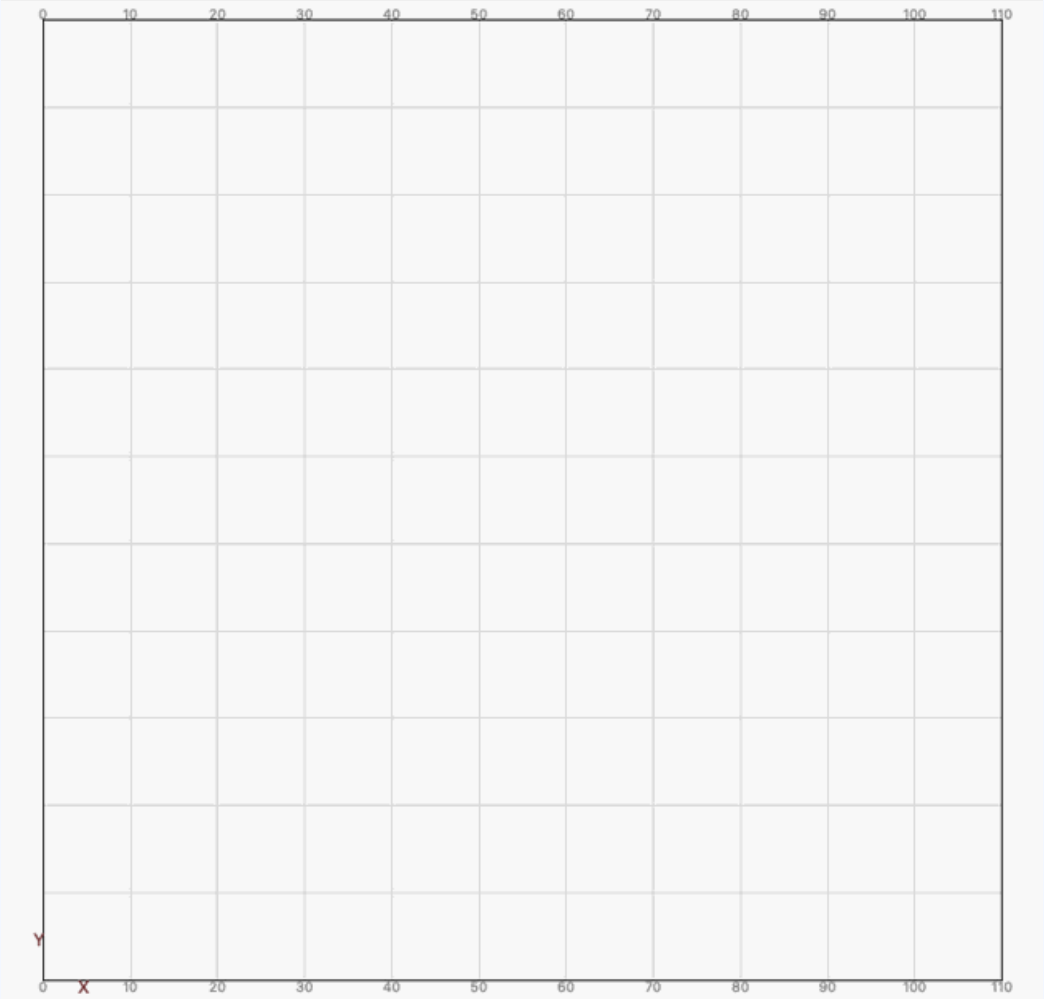
Grid Contrast off



Low contrast grid



Medium contrast grid



High contrast grid

Grid Snap Distance

Determines how close lines and other primitives must be to a grid line to snap to it.

Object Snap Distance (pixels)

Controls how close, in screen pixels, your cursor has to be to an object node or center to activate the Object Snapping behavior.

Shape Move Increments

When moving objects with the arrow keys in the Workspace, these values control the distance to move the selection.

Assign individual values to movements when pressing:

- The `Left`, `Right`, `Up`, and `Down` arrow keys
- `Ctrl`/`Cmd` while pressing an arrow key
- `Shift` while pressing an arrow key

Snapping

LightBurn has two snapping behaviors which you can enable or disable here.

Snap to Objects will snap your pointer location to the nearest object center or vertex when creating new objects, drawing lines, or moving objects, making it easier to connect and align shapes.

Snap to Grid will snap your cursor position to the nearest grid location, as specified by the Grid Snap Distance value. Note that the Grid Snap Distance and the Visual Grid Spacing do not have to be the same.

Hold `Ctrl`/`Cmd` while moving an object to toggle the status of the snapping behaviors — if they are currently enabled, they will be temporarily disabled, and if they're disabled, they'll be temporarily enabled.

See Snapping for more information.

Design Units

Determines the units LightBurn will use for design purposes. This is not necessarily the same as the units used for cutting speed, which can be changed in the Dimensions / Units tab of **Device Settings**.

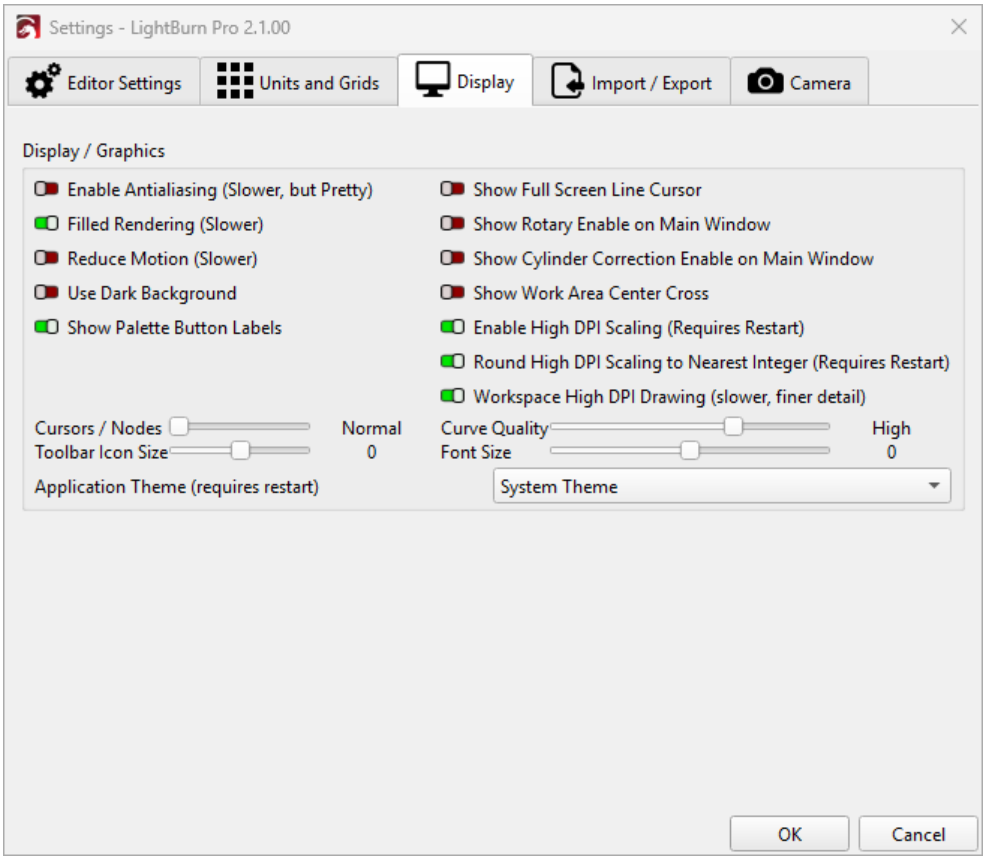
> **Control and Design Units**
>
> By default, control units — the units used for speed and distance when controlling your device — are linked with design units — the units used when designing in LightBurn. Changing either one between millimeters and inches will automatically change the other.
>
> To unlink or relink the two types of units, use the Link Control and Design Units Toggle in **Device Settings**.

Visual Grid Spacing

Visual Grid Spacing determines the visual size of grid squares at the standard zoom level. As you Zoom In, you'll see a secondary 10 x 10 grid of squares within each larger square. Each smaller square is 1/10th the size of the Visual Grid Spacing value.

Note that this setting is independent of the Grid Snap Distance, which controls how close lines and other shapes must be to a grid line to snap to it.

Display



Application Theme

Use this dropdown to switch between LightBurn's light and dark themes. The default **System Theme** setting will match your operating system's theme setting.

> **Tip**
>
> When using a dark **Application Theme**, enable Use Dark Background to darken your Workspace along with LightBurn's UI.

> **Note**
>
> Switching themes requires a restart to take effect.

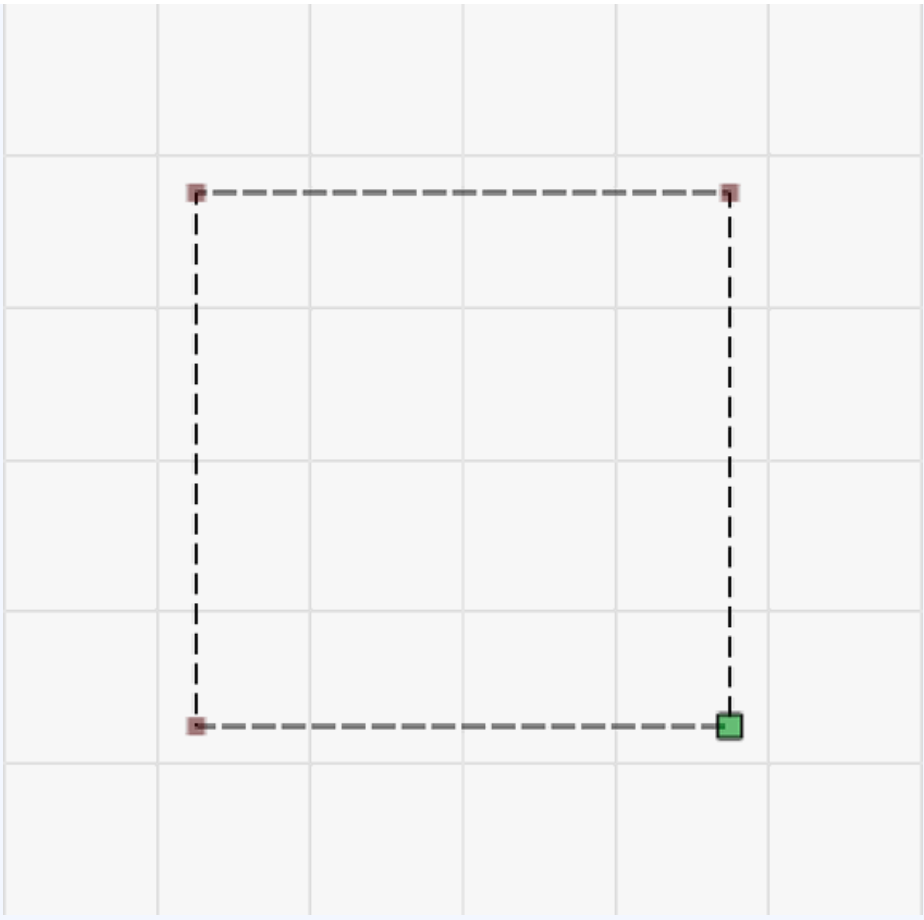
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=m7P--QrhG1s

Cursors / Nodes

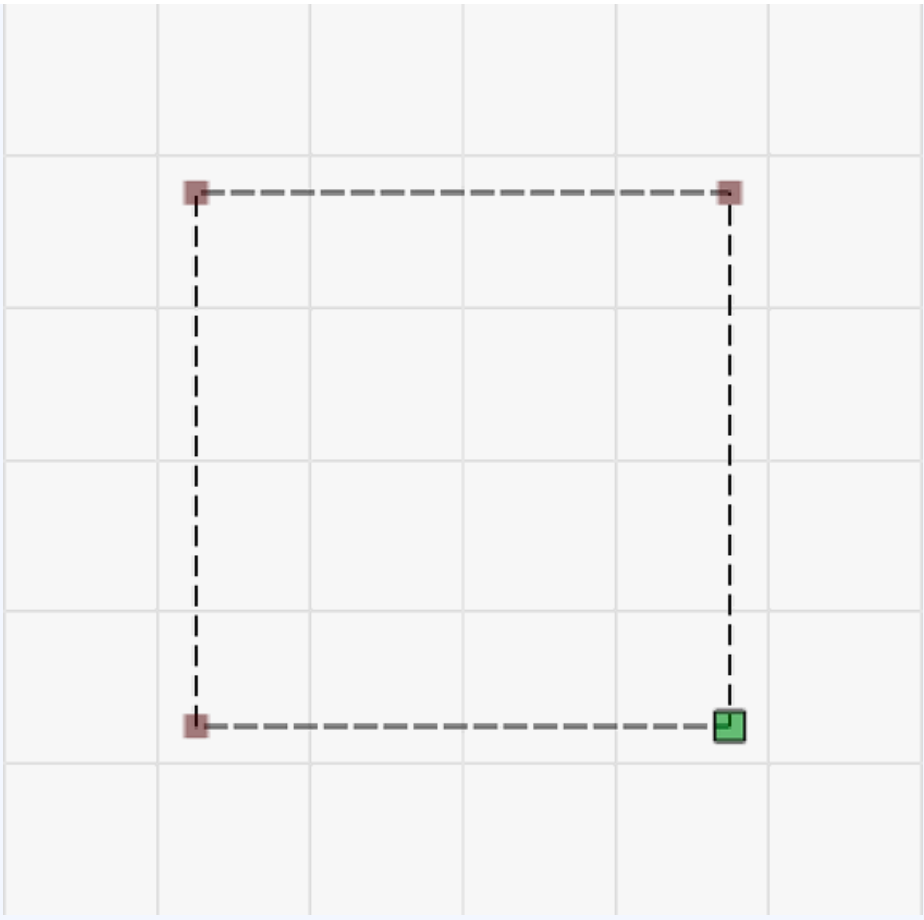
Adjusts the size of special cursors used when Snapping, applying radii, Bending text, and more, as well as the size of individual nodes when using Edit Nodes.

Adjust the slider to select between **Normal**, **Larger**, and **Largest**.

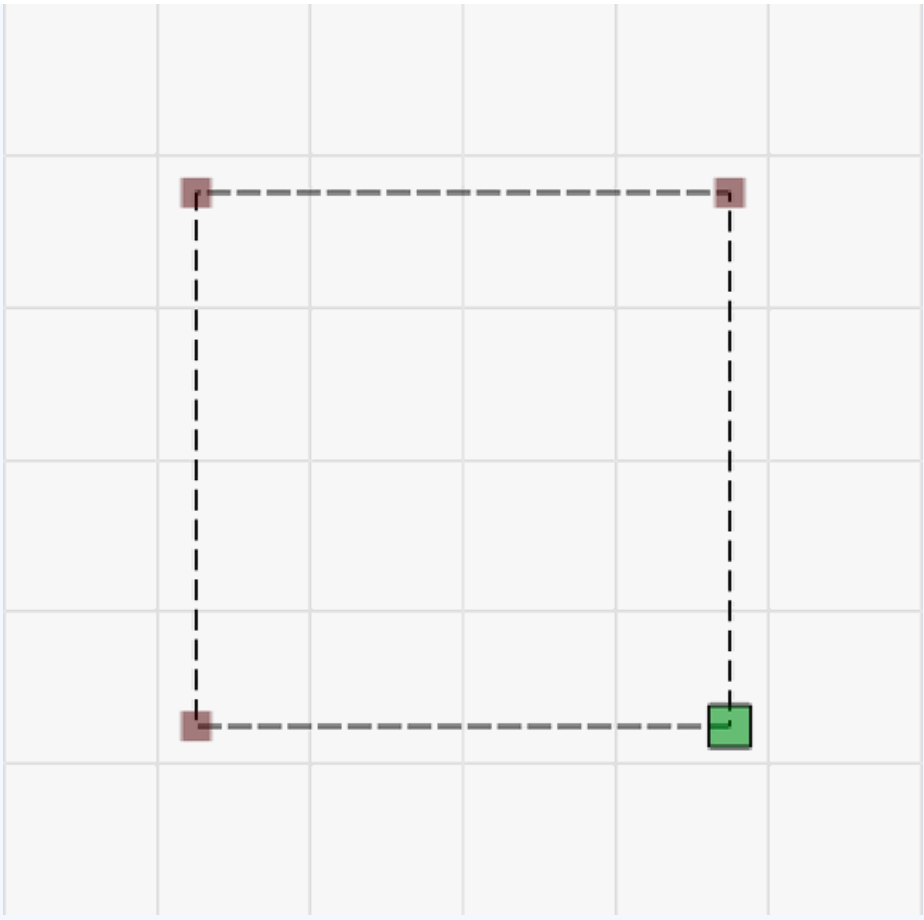
> **Node Size Comparison**



Normal nodes



Large nodes



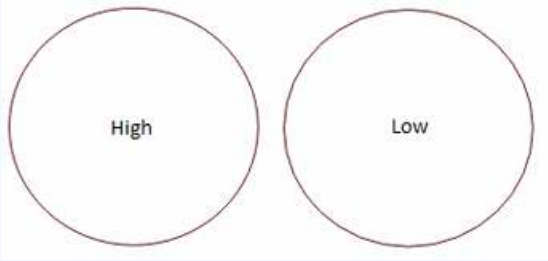
Largest nodes

Curve Quality

Controls the precision LightBurn uses in displaying curves on the screen. All curves are displayed as a series of short lines, but the higher this value is, the shorter those lines will be.

Reduce this value to improve performance if your system is slow.

Curve Quality comparison

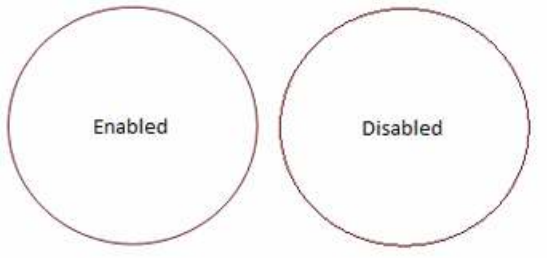


Enable Antialiasing (slower, but pretty)

Antialiasing smooths the appearance of lines on your screen. Enabling anti-aliasing will make your design rendering look much nicer, but increases demand on your machine.

If you notice your computer lagging, especially on older computers or when working with very complex designs, try disabling this setting.

Antialiasing comparison



Enable High DPI Scaling

High DPI scaling can make the LightBurn interface easier to use on high resolution screens. If the interface is unreasonably small on a high resolution screen, try enabling this and restarting LightBurn to scale the LightBurn UI to match your operating system's scaling settings.

Tip

If enabling High DPI Scaling causes blurriness or other visual artifacts, try enabling **Round High DPI Scaling to Nearest Integer**.

Workspace High DPI Drawing

When Workspace High DPI Drawing is enabled, the Workspace and Preview Window will show artwork at the true display resolution. This increases detail, but can make small details harder to see on very high DPI displays, such as Apple Retina displays.

If small details are hard to see on a high resolution display or you need to speed up LightBurn's rendering when working with large, complex designs, try disabling this setting.

Filled Rendering (slower)

Renders shapes that are assigned to layers set to Fill Mode as solid shapes.

This takes longer than rendering the wire outlines, and it can hide graphics behind Filled shapes that may still be sent to the laser, so we recommend using this only for spot-checking your work, and not for general editing.

See View Style for more information.

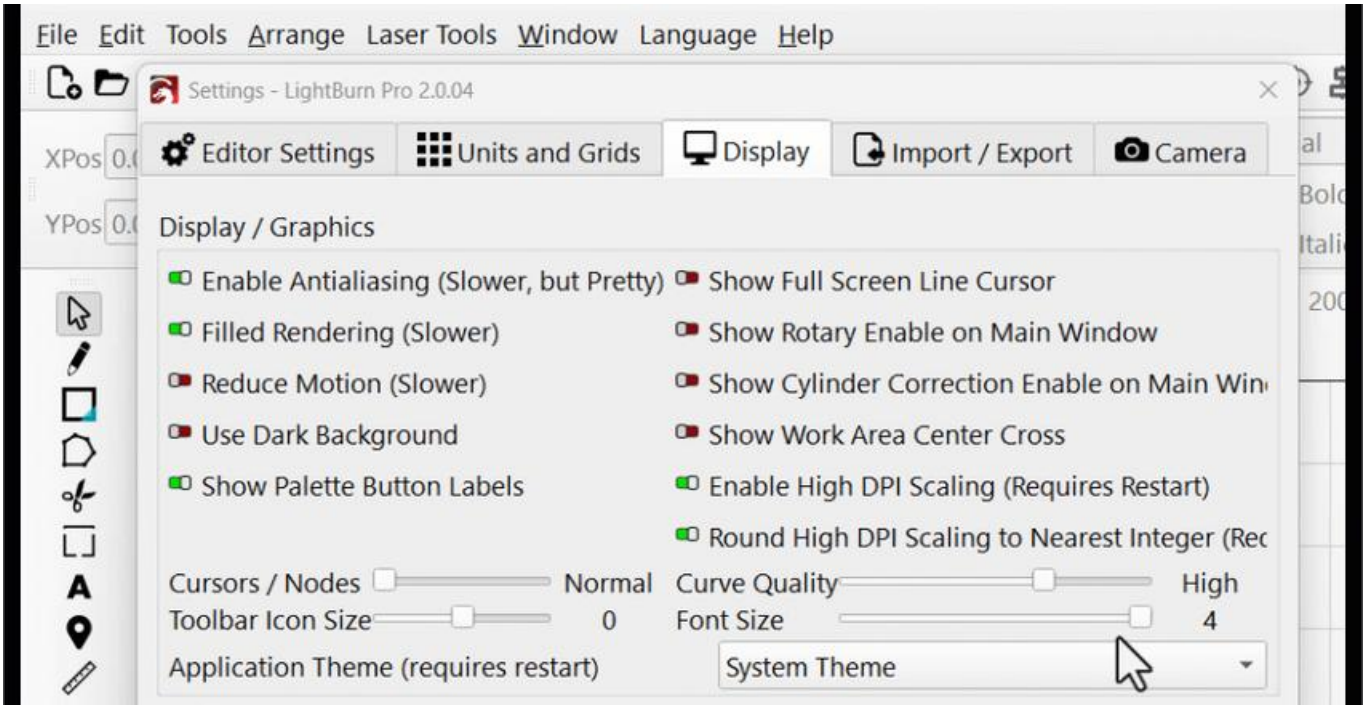
Filled vs. Wireframe comparison



Font Size

Increases or decreases the font size for LightBurn window text anywhere in the program. This allows for manual overrides for high DPI screens, UI scaling, and visually impaired users. Not all UI elements may fit on screen if this is set too high.

Adjust the slider to adjust your font size. The default font size is **0** in the center, with **-4** being the smallest font setting and **4** being the largest.



Warning

Setting this slider too high may result in clipping of UI elements. If you scale font up or down such that you cannot reach the menu items to re-open the **Settings** window and shrink it, use the method listed here to reset LightBurn.

Perform Startup Display Size Checks

On small or low resolution screens, LightBurn's default display settings can cause parts of the interface to be cut off. If this setting is enabled, LightBurn will do a check each time it starts up to look for potential problems with LightBurn and your display settings and offer to fix them automatically.

If you don't want LightBurn to perform this check, disable the toggle. This is enabled by default.

Reduce Motion (slower)

Instructs LightBurn to try to reduce motion or other flickering effects in the rendering.

If you notice your computer lagging, especially on older computers, or when working with very complex designs, try disabling this setting.

Show cylinder correction enable on main window

Adds a toggle to the Laser Window to enable or disable Cylinder Correction mode.

Show full screen line cursor

Draws a moving crosshair, centered on your cursor, that spans the entire edit screen.

Show Palette Button Labels

Shows numbered labels on the Color Palette swatches, making it easier to tell similar colors apart.

Show rotary enable on main window

Adds a toggle to the Laser Window to enable or disable Rotary Mode.

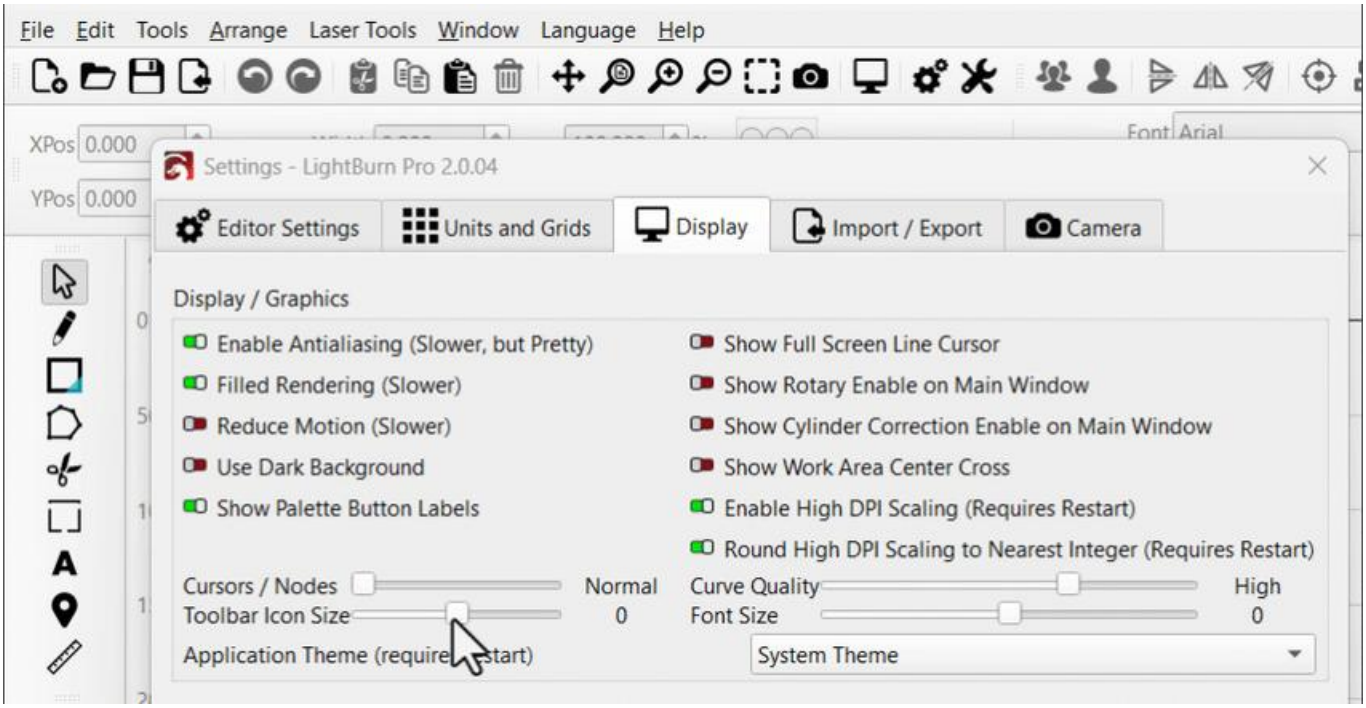
Show work area center cross

Draws an immovable crosshair through the center of your Workspace, spanning the entire grid.

Toolbar Icon Size

Changes the scaling of icon elements in the LightBurn UI. This allows for manual overrides for high DPI screens, UI scaling, and visually impaired users. Not all UI elements may fit on screen if this is set too high.

Adjust the slider to adjust the size of icons and other graphical elements in LightBurn. The slider ranges from **-5** (smallest) to **5** (largest), with **0** in the center being the default sizing. Any scaling from Enable High DPI Scaling is applied on top of this adjustment.



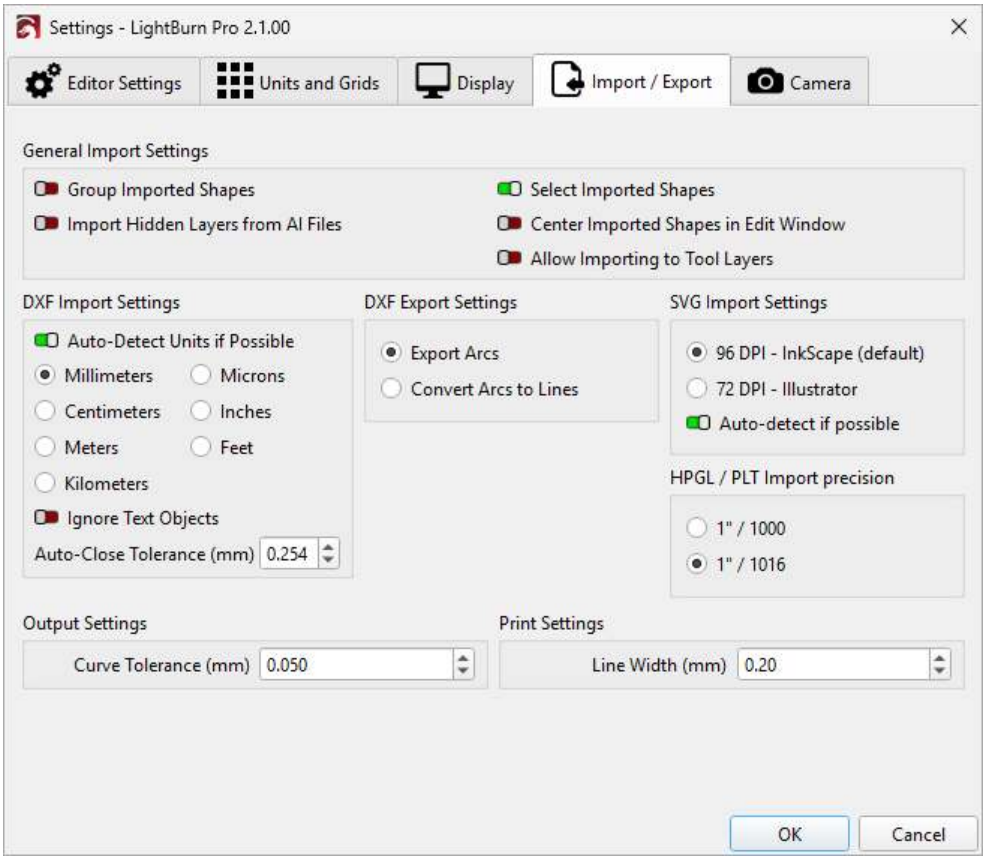
Warning

Setting this slider too high may result in clipping of UI elements. If you scale icons up or down such that you cannot reach the menu items to re-open the **Settings** window and shrink them, use the method listed here to reset LightBurn.

Use Dark Background

Shows the Workspace using a dark color scheme.

Import / Export



General Import Settings

Group Imported Shapes

Automatically Groups shapes imported from a single file together, so they don't get scattered in a complex design.

Import Hidden Layers From AI Files

Enables importation of layers hidden in the source file, when importing from AI (Adobe Illustrator) files.

Select Imported Shapes

Automatically selects shapes upon importing them.

Center Imported Shapes in Edit Window

Automatically centers shapes in the Workspace when importing files.

Allow Importing to Tool Layers

Enables importing graphics directly onto Tool layers in LightBurn.

DXF Import Settings

Enable **Auto-detect units if possible** to have LightBurn automatically detect the units used to produce the DXF, if it can.

Select the appropriate unit of measurement for your DXF if the units are not contained in the file, or the DXF may import at an incorrect scale.

Auto-close tolerance (mm) determines how close together imported nodes must be for LightBurn to automatically close them, placing them on a shared path.

Ignore Text Objects ignores text objects included in a DXF file. This can be useful for leaving out dimensions and other notes and only importing relevant shapes.

DXF Export Settings

When exporting in DXF format, choose whether to **Export Arcs** or **Convert Arcs to Lines**.

When Convert Arcs to Lines is selected, arcs in your LightBurn designs will be exported as a series of small line segments.

SVG Import Settings

Select whether to import SVGs at:

- **96 DPI** — the default for Inkscape
- **72 DPI** — the default for Adobe Illustrator

If your SVGs import at the incorrect scale, try the alternate setting to the one you currently have selected.

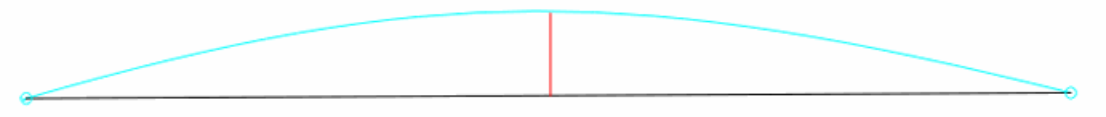
HPGL / PLT Import precision

Determines the precision of files imported from HGPL or PLT format.

Output Settings

Curve Tolerance specifies the maximum amount of error allowed when projects are output to your laser. The lower the value, the more accurate the curve will be. A value of 0 would be "perfect", but would create very dense data, as some lasers can only process line segments. Most users will be fine leaving this at the default value of 0.05 mm.

Technical Details



In the above image, the blue curve between the two points is the ideal shape. The black line is a straight line between them, and the red line shows the error (how far the line is from the curve). LightBurn measures this error, and if it's equal to or lower than the Curve Tolerance value, it outputs the straight line. If not, the curve is subdivided into two linear segments and the process repeats with each new segment. Those segments are shown below in violet, along with their new error values. You can see that the two new lines do a much better job of approximating the original curve.



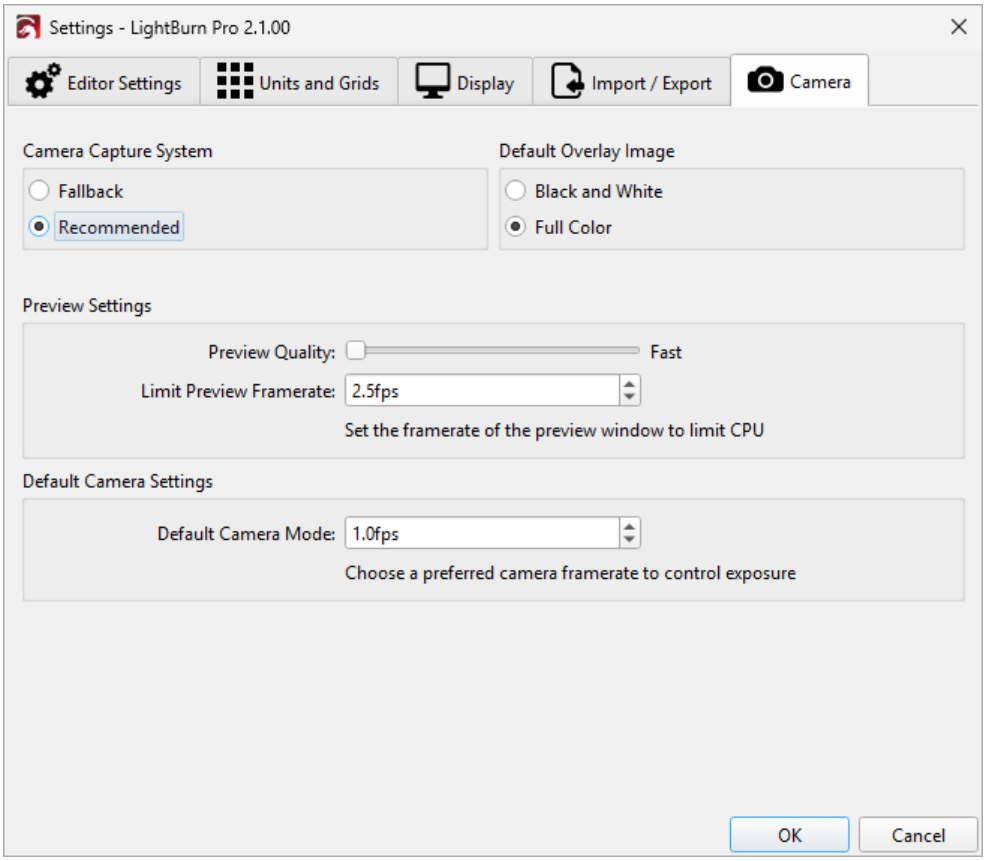
Most people will probably never need to change this — the default is 0.05 mm, which is about ½ the width of a typical beam. Note that this is the maximum error value allowed, typical output will be better than this, and this only affects curves, not straight lines or vertices — which are exact.

Print Settings

Line Width

Sets the line width to be used when Printing a LightBurn file. If your laser cutter uses a printer driver for control and requires a specific line width, you can use this to send artwork to your machine from LightBurn.

Camera



Camera Capture System

Windows Only

For most users, the default **Recommended** setting should work. If you are having trouble accessing your camera, try the **Fallback** option for an alternate camera capture method.

Default Overlay Image

Select whether to show overlay captures from your camera in **Full Color** or **Black and White**.

Preview Settings

Preview Quality

Use this slider to adjust the quality of your camera preview, finding a good balance between refresh speed and smoothness.

Limit Preview Framerate

Sets the limit of the framerate of the preview image in the Cameras / Overlays Window. Lower this setting to improve CPU performance.

Default Camera Settings

Default Camera Mode

Sets the limit of the framerate to use when the camera captures an overlay. Lower this setting to control exposure.

Managing Preferences

The **Preferences** submenu provides several tools to manage your LightBurn Preferences. Access the Preferences submenu by going to File → **Preferences**. For information about adjusting Preferences themselves, see Settings/Preferences.



Importing and Exporting Preferences

The following tools are useful for backing up, restoring, saving, importing, and migrating LightBurn Preferences between computers.

> **Tip**
>
> User Bundles streamline the process of migrating between computers, allowing you to export not only your Preferences, but also your Custom Hotkeys, Material Test Presets, Image Presets, Devices, Material Libraries, and Art Libraries, all in one bundle.

Import/Export Prefs

The **Export Prefs** command allows you to save a copy of your LightBurn Preferences as a `.lbprefs` file. You can import this file to another LightBurn installation using the **Import Prefs** command.

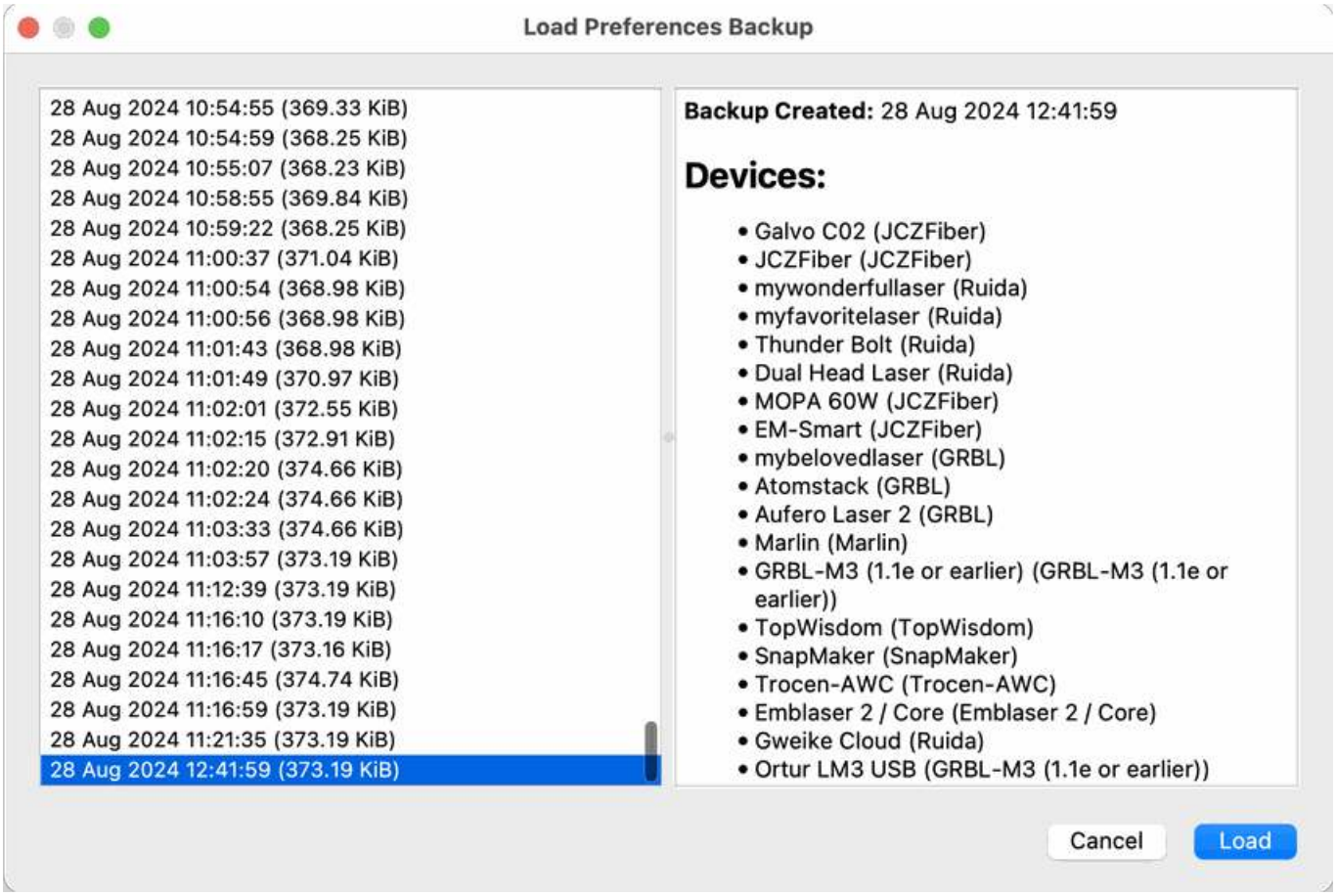
Open Prefs Folder

The **Open Prefs Folder** command opens the folder containing your LightBurn Preferences.

Load Prefs Backup

The **Load Prefs Backup** command allows you to import preferences from a timestamped list of automatic backups. Click a backup in the list to see the list of Devices available in that backup, on the right side of the window. Use the list of devices and the timestamp showing when the backup was made to find and select the backup you want to use.

Click **Load** to restore the settings from the selected backup, or **Cancel** to close the window and maintain your current settings.



Edit Hotkeys

Opens the Hotkey Editor window, where you can customize keyboard shortcuts.

Related Topics

- LightBurn Migration
- Settings/Preferences
- User Bundles
- Hotkey Editor

User Bundles

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=S0QrokNepCQ

User Bundles simplify the process of migrating LightBurn between computers. Go to File → **Bundles** → **Export Bundle** to create a bundle, and import a bundle by dragging the `.lbzip` file into the LightBurn Main Window or using **File** → **Bundles** → **Import Bundle**.

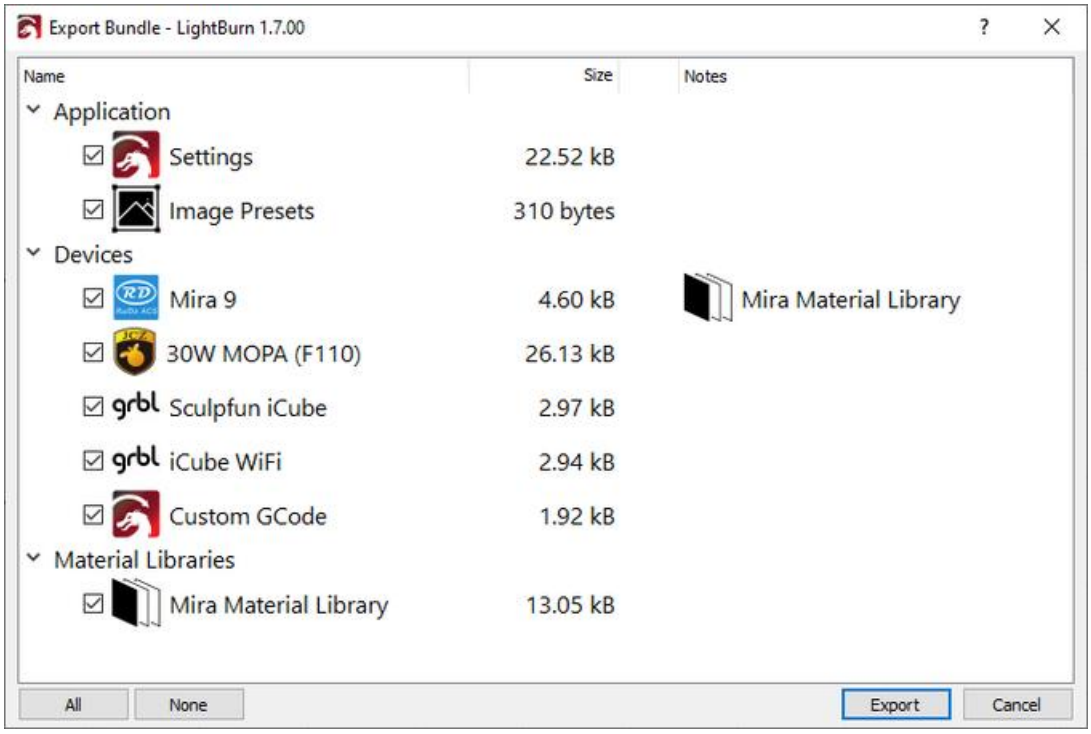
A bundle can include any of the following. You can export everything, or select only the components you want to include:

- LightBurn Settings
- Custom Hotkeys
- Material Test Presets
- Image Presets
- Devices (including cameras)
- Material Libraries
- Art Libraries

It's important to keep in mind that this list does not include your project files. If you want these available on your new computer, be sure to copy or move them separately.

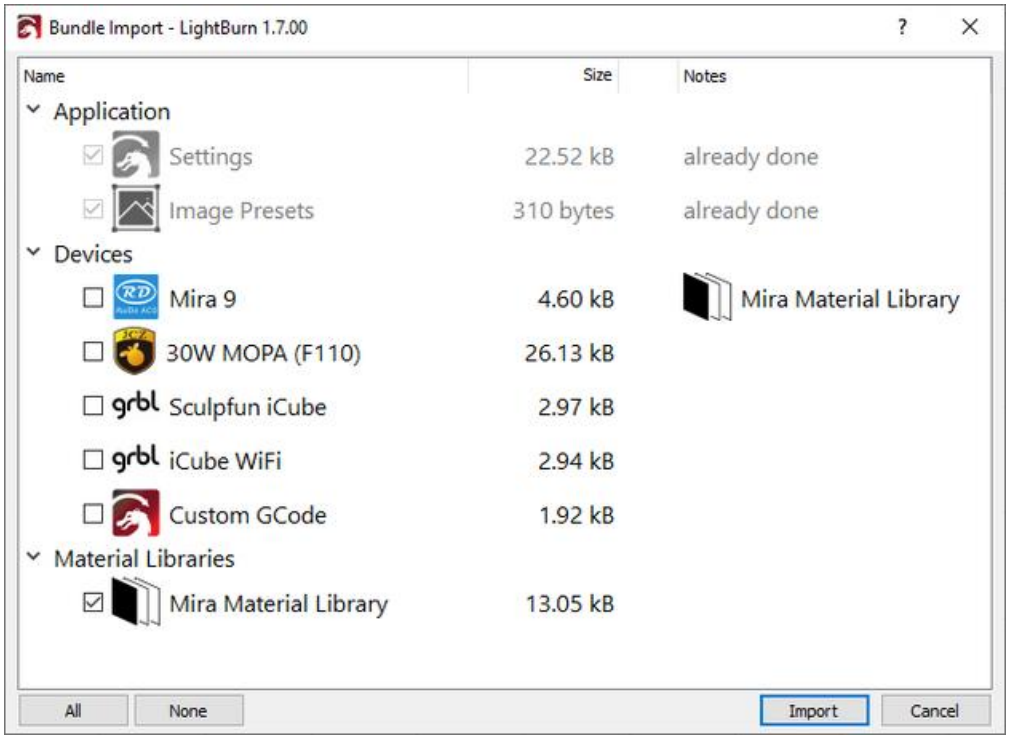
Working With Bundles

Export Bundle



1. In an existing and already configured LightBurn installation, go to **File → Bundles → Export Bundle**.
2. Make sure that all components of the current installation that you want to include are selected and click **Export**. Save the file somewhere you can find it again easily.

Import Bundle



1. Move the `.lbzip` file created by **Export Bundle** to the new computer.
2. Open LightBurn on the new computer. Go to File → **Bundles** → **Import Bundle** and select the file. Click **Okay**.
3. Make sure that all components you want to import are selected, and uncheck any components you don't want to import. Components that are already detected on the computer will be grayed out and unavailable, with some exceptions:
   - If you choose to a import a Device that conflicts with one that already exists, a duplicate will be created, and the existing profile preserved.
   - If the new computer has an existing Material Library or Art Library with the same name but different contents as the one contained in the bundle, a new copy of the library will be created, without overwriting the original.
4. Click **Import** to complete the process.

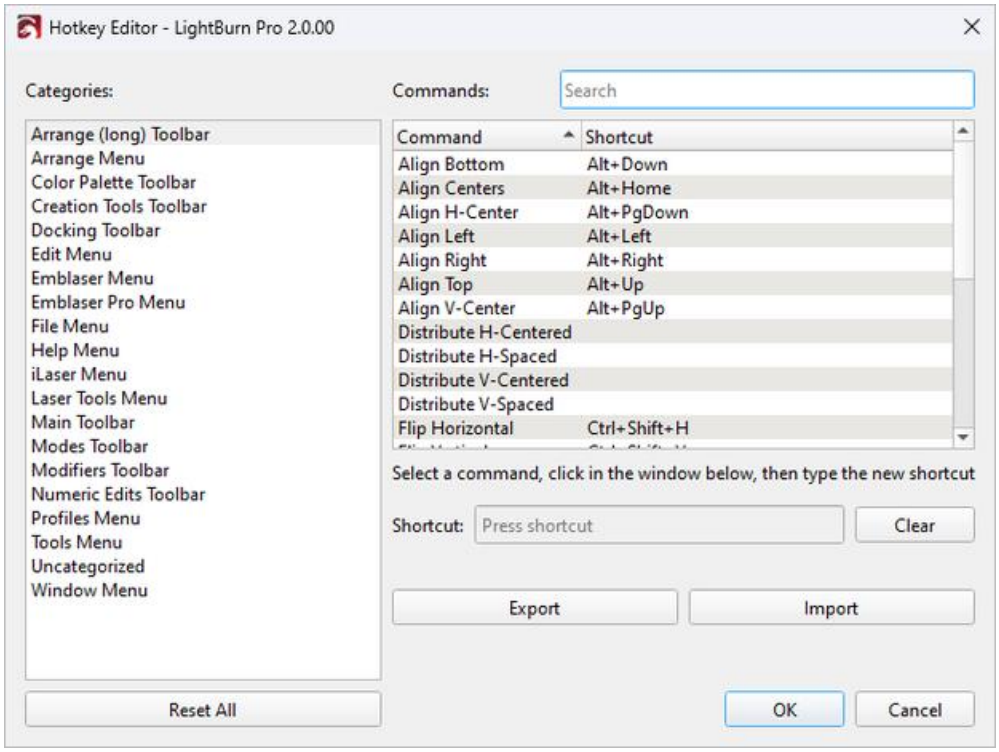
> **Tip**
>
> You can also import a bundle by dragging the `.lbzip` file into LightBurn.

Related Topics

- Migration Guide
- Managing Preferences

Edit Hotkeys

Use the **Hotkey Editor** to customize LightBurn's hotkeys. From this window, you can add new keyboard shortcuts and edit or delete existing ones. Access the Hotkey Editor by going to **File → Preferences → Edit Hotkeys**.



You can also download printable hotkey cheatsheets.

Printable hotkeys for Windows

Printable hotkeys for macOS

Using the Hotkey Editor

To customize hotkeys:

1. Find the command you want to modify using the categories on the left or the search bar at the top of the window.
2. Select the command.
3. Click in the **Shortcut** text box.
4. Type the new hotkey, or press **Clear** to erase the hotkey.
5. Click **OK** to save your changes when you're finished.

From this window, you can also use **Reset All** to return hotkeys to their defaults, or use the **Export** and **Import** buttons to back up your custom hotkeys or restore a backup. Custom hotkeys can also be imported and exported as part of a User Bundle.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=UNj3y0aJ_w8

Related Topics

- Settings/Preferences
- User Bundles

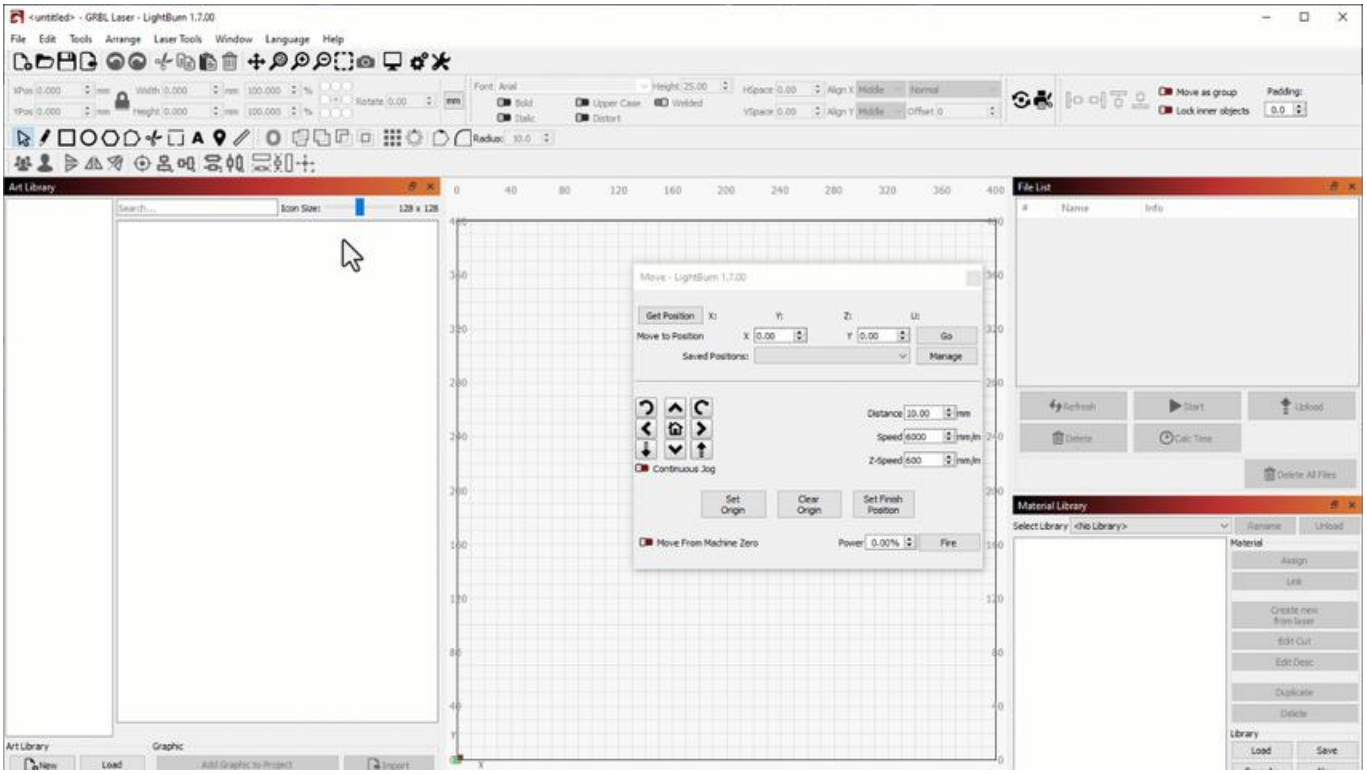
Reset to Default Layout

Reset to Default Layout restores the layout of the full LightBurn window to the same format it was in when you first installed and opened LightBurn. If you've made lots of changes to your layout that you want to reverse all at once, or you've lost sight of the Laser Window, Cuts / Layers Window, Color Palette, or any other default windows or toolbars, use this option to get them back.

Resetting the layout doesn't reset anything else — your preferences, devices, and libraries are all preserved.

Go to Window → **Reset to Default Layout** to re-open and re-dock all default windows in their original order, while closing any that are not visible by default.

Holding `Shift` while clicking to open LightBurn will also reset the layout.



Related Topics

- Workspace
- Main Window
- Customizing the LightBurn Window
- Reset to Default Settings

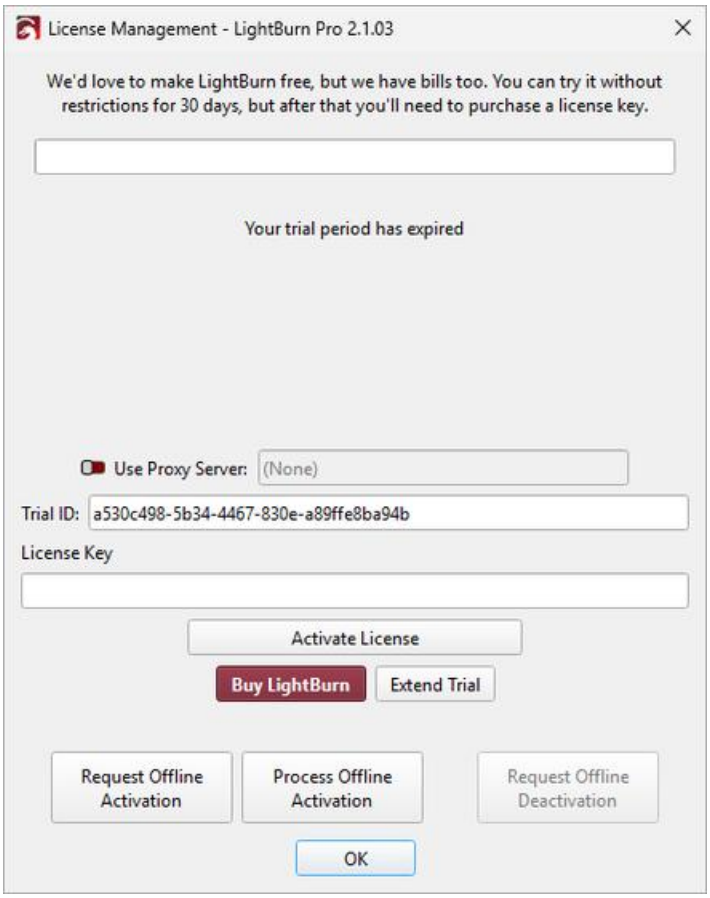
Help and Software Features

License Management

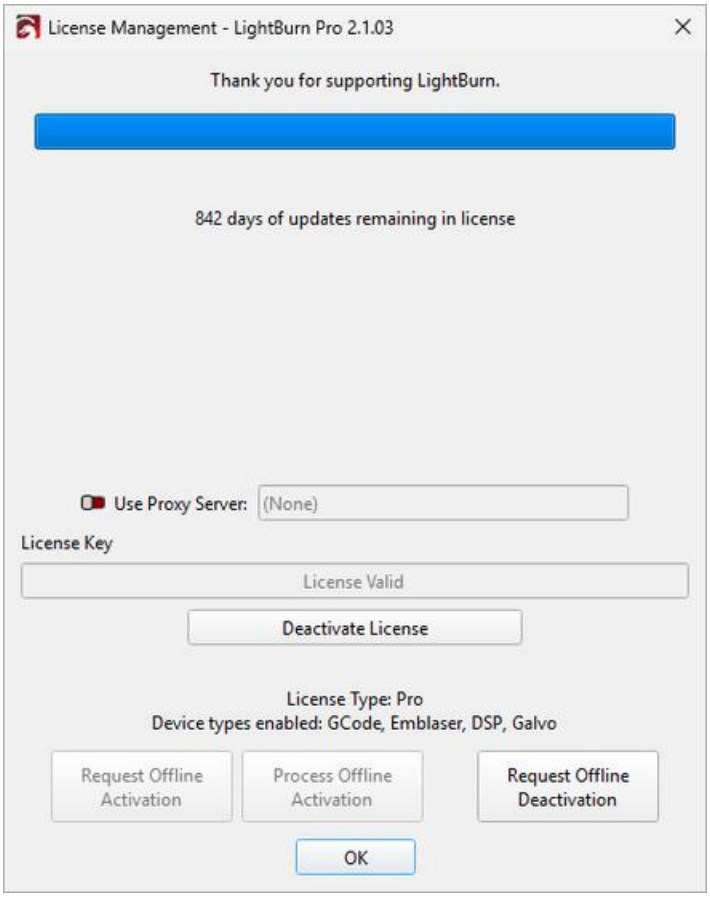
If your LightBurn installation has not been activated, you'll see the **License Page** as soon as you open LightBurn. From this page, you can:

- Enter and activate a license key you've already purchased
- Start a free 30 day trial
- Purchase a license key from our website
- Extend your free trial
- Deactivate a license to free up an installation for a different computer

You can get back to this screen in LightBurn at any time by going to Help → **License Management**. For more information on License Management, including offline activation and deactivating a computer, please see our License Management Guide.



The License Page before activation.



The License Page after activation.

You can manage your license from our License Portal.

License Portal

Activate Trial

Click the **Activate Trial** button to activate a free 30 day trial of LightBurn.

Extend Trial

Click **Extend Trial** to automatically extend your trial period. You can extend your trial up to 3 times, for an additional 15 days each time. Trial keys over a year old are extended for 30 days, but may only be eligible for a single auto-extension.

Activate License

If you already have a license key, enter it into the **License Key** field and click **Activate License**. Be sure to enter your license key exactly as provided, including the dashes. We recommend copying the license key directly from your original order email and pasting it into the License Key field.

Use Proxy Server

See System Locked and Floating Licenses: Network Restrictions for information on connecting to LightBurn's licensing server through a proxy.

Troubleshooting

If you receive an error message when trying to activate your license, see Troubleshooting: License Activation and Management.

Request Offline Activation and Deactivation

If you will be using LightBurn on a computer without an internet connection, please see our Offline Activation instructions.

Deactivate License

Click **Deactivate License** to deactivate LightBurn on your computer, freeing up an activation to be used on another computer.

See our License Management Guide for information on deactivating a computer that you no longer have access to.

After Activation Information

License Expiration

After you've activated your license, you'll see the number of days remaining on your license at the top of the **License Page**.

After your key has expired, you'll still be able to use the last version of LightBurn released before it expired forever, but you'll need to Renew your key in order to receive future updates.

Devices Supported

Beneath the **Deactivate License** button, you'll see the list of device types supported by your key. To add support for new device types to your key, see our website.

Check for Updates

Check for Updates checks whether there is a more recent version of LightBurn available than the version currently running, and whether it was released within your license's valid update period. If a valid newer version is available, you can begin downloading it from the Check for Updates dialog window.

Check for Updates Responses

Go to Help → **Check for Updates** to tell LightBurn to check for available updates. There are three possible responses.

Version Up to Date



The currently running version is up to date. No further action is required.

Newer Version Available



A newer version is available. Click **Yes** to automatically begin downloading the newer version. Click **No** to remain on the current version.

License Expired



A newer version is available, but your license has expired. To use the newest version, you must renew your license. You will be able to continue receiving updates for one year after you renew your license.

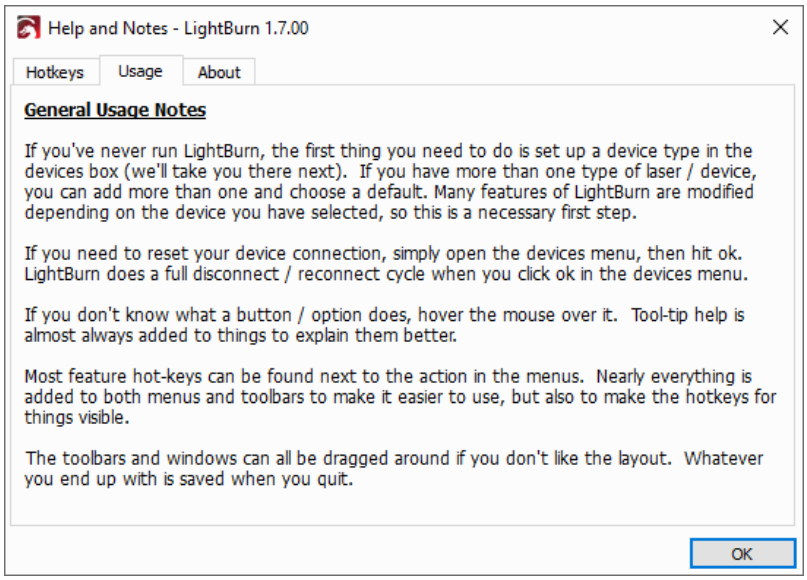
> **Note**
>
> If you choose not to renew, you will always be able to continue using the most recent version of LightBurn that was released within your license's valid update period.

Related Topics

- How the License Works
- Install LightBurn

Help and Notes

Open the **Help and Notes** window by going to Help → **About**, Help → **Quick Help and Notes**, or pressing `F1`.



This window has three tabs:

- **Hotkeys** summarizes some of the most frequently used mouse and keyboard shortcuts. See Hotkey Editor for information on setting custom hotkeys.
- **Usage** contains tips to help you get started.
- **About** contains information about the version of LightBurn installed on your computer.

Related Topics

- Help Menu

REST API

LightBurn's REST API provides a way to communicate between LightBurn and other applications. The API is intended to allow users and developers to create integrations and improve interoperability with other software for advanced workflows, automate processes, monitor status, and exchange data between LightBurn and third-party applications. The API is intended for use on a single computer or within your local network. Public documentation for the LightBurn REST API is available at https://github.com/LightBurnSoftware/LightBurn-REST with more detailed documentation coming soon.

As a reminder, LightBurn does not endorse, recommend, support, or encourage unattended use of lasers or other machinery. LightBurn's API is not a replacement for direct operator supervision.

Using the REST API in LightBurn

LightBurn 2.2 enables the REST API automatically. By default, LightBurn will listen for API commands on port 19520. If you're not sure what that means, don't worry - most users will never need to change it.

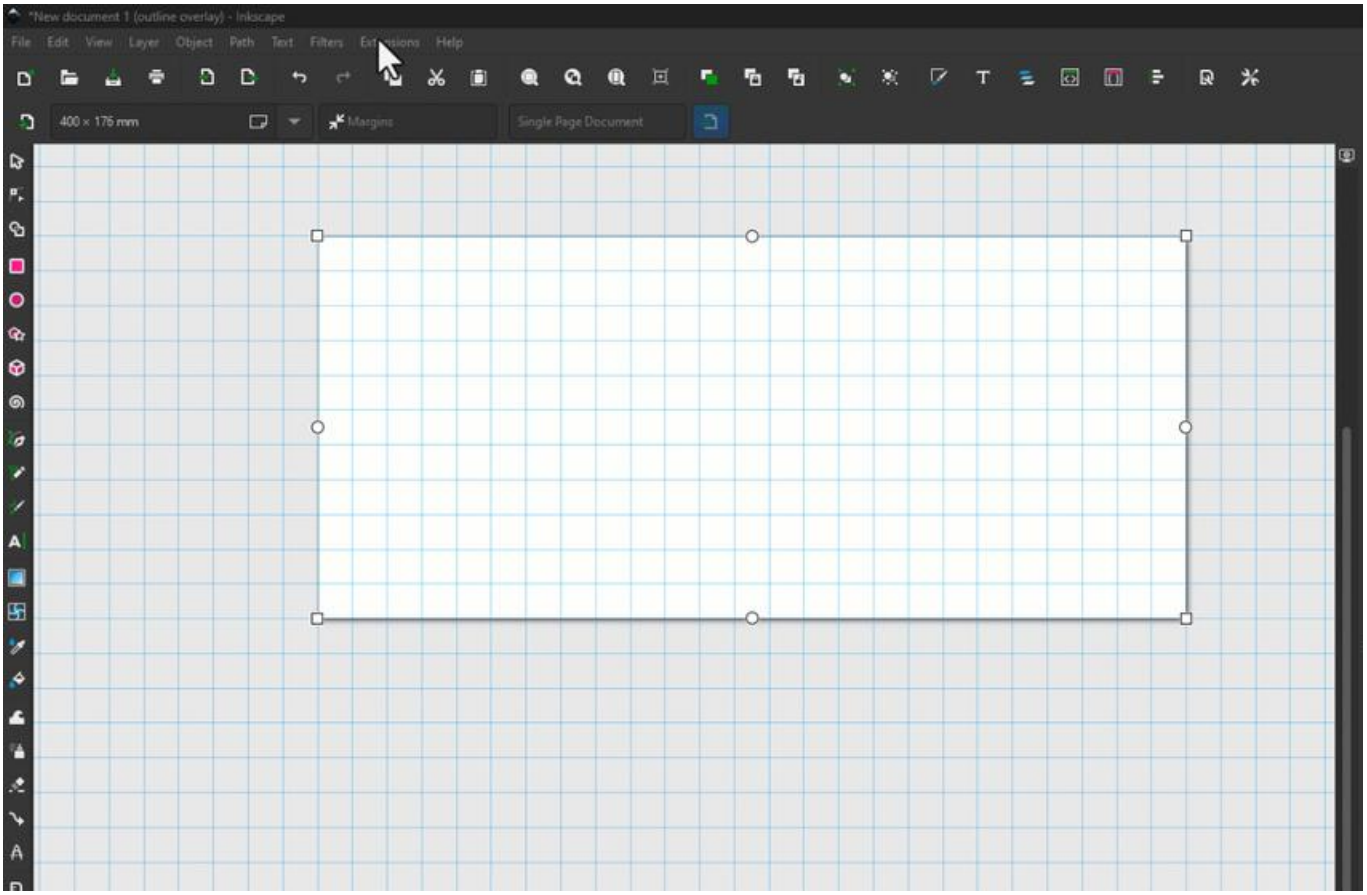
Enabling API Access From Other Computers

You can use the API to access LightBurn from other computers on the same network by enabling the **Allow API Access From Network** toggle in Settings / Preferences → **Extensions Tab**.

Managing API Authorizations

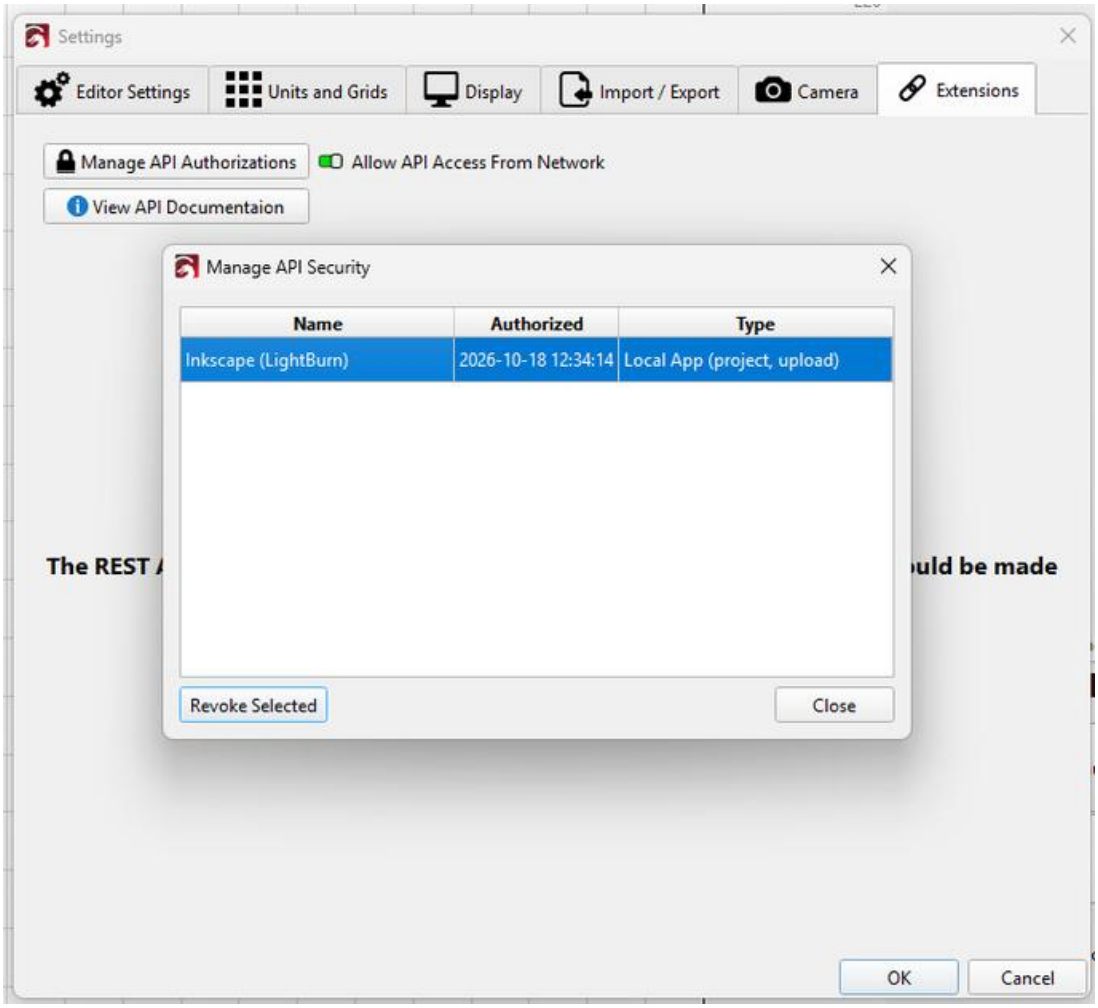
Granting Authorizations

The first time you allow a new extension to connect to LightBurn using the API, you will need to approve that access in a pop-up window. You will have 30 seconds to authorize the access or it will be automatically rejected. If you're connecting over your network from a separate computer, the authorization has to be granted from the computer LightBurn is running on.



Removing Authorizations

You can use the **Manage API Authorizations** button in Settings / Preferences → **Extensions** to open the **Manage API Authorizations** window to view and, if needed, remove API authorizations.



Each API authorization currently registered in LightBurn on your computer will be listed, showing:

- Name
- Date and time of authorization
- Type of API access

If an authorization is no longer needed or is causing problems, select it in the **Manage API Authorizations** window and click **Revoke Selected** to remove it.

Working With Multiple LightBurn Instances

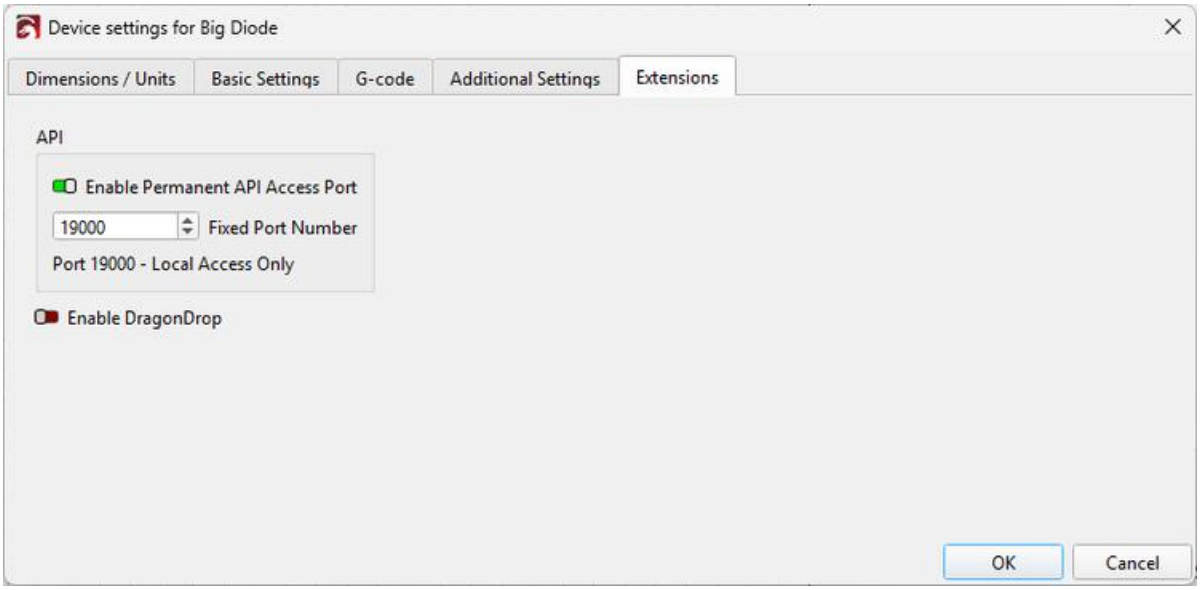
If you have multiple intances (windows) of LightBurn open and are connecting to the default switching API port (19520), whichever one was accessed most recently is the window that will be listening. For more advanced control using device-specific listening ports, see Advanced API Access with Multiple Instance of LightBurn.

Advanced API Access With Multiple Instances of LightBurn

While the default for the REST API is for the most recently opened window to be the one listening, you can set and enable device-specific listening ports. This isn't necessary for most users, but it allows further control over which window is listening for API requests if needed. A device-specific listening port will listen from the first LightBurn instance that opens that specific device profile.

You might, for instance, want to set a device-specific listening port so that you can open LightBurn and connect to an external tool using the REST API, then continue to connect from that same window while working on a design in a different window.

To set a device-specific listening port, open the **Extensions** tab of the Device Settings window.



To enable device-specific listening ports in LightBurn:

- Open the **Device Settings** window.
- Enable the toggle switch for **Enable Permanent API Access Port**.
- Change the per device port number if needed.
- If you want to allow API access to LightBurn from other computers on the same network, go to Settings/Preferences and switch to the **Extensions** tab. Enable the **Allow API Access From Network** toggle.

Disabling

Trying the REST API

We have two sample integrations for LightBurn's REST API. You may find these helpful as tools in your workflow, but they can also be useful for verifying that the API is working correctly or as examples when creating your own integrations.

Inkscape Extension

Inkscape extensions to draw the machine workspace in Inkscape and send your design from Inkscape back into LightBurn.

Installation and Use

FreeCAD Extension

A FreeCAD workbench that generates 2D gear profiles and sends them directly to LightBurn or MillMage for laser cutting or CNC milling.

Installation and Use

Detailed Developer Documentation

If you want to create integrations using the REST API, detailed developer documentation is available at https://github.com/LightBurnSoftware/LightBurn-REST.

FreeCAD Sample Workbench

A FreeCAD workbench that generates 2D gear profiles and sends them directly to **LightBurn** or **MillMage** for laser cutting or CNC milling.

Supported gear types

Type	FreeCAD gear	Notes
Involute spur gear	`CreateInvoluteGear`	Bore + keyway support
Involute rack	`CreateInvoluteRack`	Linear, no bore
Timing belt pulley	`CreateTimingGear`	gt2, gt3, gt5, gt8, htd3, htd5, htd8

Requirements

- FreeCAD 1.0+
- The **FCGear** (freecad.gears) addon — install via Tools → Addon Manager

Install

> **Warning**
>
> This is an early demo, installed by hand (steps below), not through the FreeCAD Addon Manager. More features are planned.

SpurLine must end up as a `SpurLine` folder inside FreeCAD's user `Mod` directory. Follow the steps below, then restart FreeCAD. There are two variations of Step 2 - use whichever you prefer.

1. Find your FreeCAD `Mod` directory/folder

OS	Path
Windows	`%APPDATA%\FreeCAD\Mod`
Linux	`~/.local/share/FreeCAD/Mod`
macOS	`~/Library/Application Support/FreeCAD/Mod`

Not sure about this? You can check in FreeCAD:

1. In FreeCAD open the **Python console** (**View → Panels → Python console**)
2. Run `import FreeCAD; print(FreeCAD.getUserAppDataDir())`
3. You'll see the FreeCAD user directory. If `Mod` exists, it will be in there. If not, this is where you should create it.

2. Copy or Symlink the necessary files

Copy the folder

You can also just copy the folder into `Mod\SpurLine`. Run the following from the repository root:

Linux / macOS:

```
cp -r examples/spurline ~/.local/share/FreeCAD/Mod/SpurLine
```

Windows (PowerShell):

```
Copy-Item examples\spurline "$env:APPDATA\FreeCAD\Mod\SpurLine" -Recurse
```

Symlink the folder

Best if you'll edit the code. Changes apply on the next restart. Run the following from the repository root:

Linux / macOS:

```
ln -s "$PWD/examples/spurline" ~/.local/share/FreeCAD/Mod/SpurLine
```

Windows (PowerShell, as Administrator):

```
New-Item -ItemType SymbolicLink `
-Path "$env:APPDATA\FreeCAD\Mod\SpurLine" `
-Target "$PWD\examples\spurline"
```

3. Restart FreeCAD

After restarting, select **SpurLine** from the workbench dropdown.

Configuration

No manual configuration is required. When you click **Send to LightBurn** or **Send to MillMage** for the first time, SpurLine identifies itself as `"FreeCAD (SpurLine)"` and requests access via the local REST API. A consent dialog appears in LightBurn / MillMage — approve it and the shared secret is stored in FreeCAD's preferences for future sends.

Each call to `/api/connect` produces a new consent prompt and a distinct secret. If your stored secret becomes invalid (e.g. the server was reinstalled), use **SpurLine → Reset authorizations** from the menu to clear it, then click **Send** again to re-authorize.

The default ports are **19520** (LightBurn) and **19521** (MillMage). If you have changed the port in the app's Settings, use **SpurLine → Set port...** to match it (you pick which application).

Project structure

The REST API spec lives at the repo root in `docs/openapi.yaml`.

```
examples/spurline/
├── InitGui.py                          ← FreeCAD workbench entry point
├── Init.py                             ← Headless init (no-op)
├── package.xml                         ← Addon metadata
├── freecad/spurline/
│   ├── commands/
│   │   ├── cmd_involute.py             ← Toolbar command: involute gear
│   │   ├── cmd_rack.py                 ← Toolbar command: rack
│   │   ├── cmd_timing.py               ← Toolbar command: timing pulley
│   │   ├── cmd_extract.py              ← Toolbar command: extract from selection
│   │   ├── cmd_port.py                 ← Menu command: set port
│   │   └── cmd_settings.py             ← Menu command: reset authorizations
│   ├── ui/
│   │   ├── gear_panel.py               ← Task panel for gear creation
│   │   └── extract_panel.py            ← Task panel for profile extraction
│   ├── core/
│   │   ├── profile_extractor.py        ← Create gear → 2D Part.Face
│   │   ├── selection_extractor.py      ← Existing shape → 2D Part.Face
│   │   └── sheet_composer.py           ← Grid layout + DXF/SVG export
│   ├── api/
│   │   └── client.py                   ← REST client (connect + file upload)
│   └── prefs/
│       └── preferences.py              ← Port + shared secret storage
└── resources/icons/                     ← Toolbar SVG icons
```

Typical workflow

Create a new gear profile

1. Switch to the **SpurLine** workbench
2. Click a gear type in the toolbar (involute, rack, or timing pulley)
3. Set teeth count, module, bore diameter, and number of copies
4. Optionally click **Preview in FreeCAD** to inspect and measure the profile
5. Click **Send to LightBurn** or **Send to MillMage**

Extract profiles from existing objects

1. Open a document with existing objects (FCGear gears, STEP imports, any Part)
2. Select one or more objects in the model tree
3. Click **Extract Profile from Selection** in the toolbar
4. A blue cutting plane appears on the selected objects
5. Use the **Cutting Plane** controls to set the orientation (Top / Front / Right) and drag the offset slider to position the slice

6. Set copies per object and optional bore/keyway overrides
7. Click **Preview in FreeCAD** or **Send to LightBurn / MillMage**

Multi-slice export

To export cross-sections at regular intervals through an object:

1. Follow the steps above to select objects and set the cutting plane orientation
2. Switch to the **Multi-Slice** tab
3. Choose a mode: **Fixed distance** (set mm spacing) or **Slice count** (set number of evenly-spaced slices)
4. Click **Preview All Slices** to inspect, or **Send to LightBurn / MillMage** to upload each slice as a separate DXF file

License

GPL-3.0 — see LICENSE.

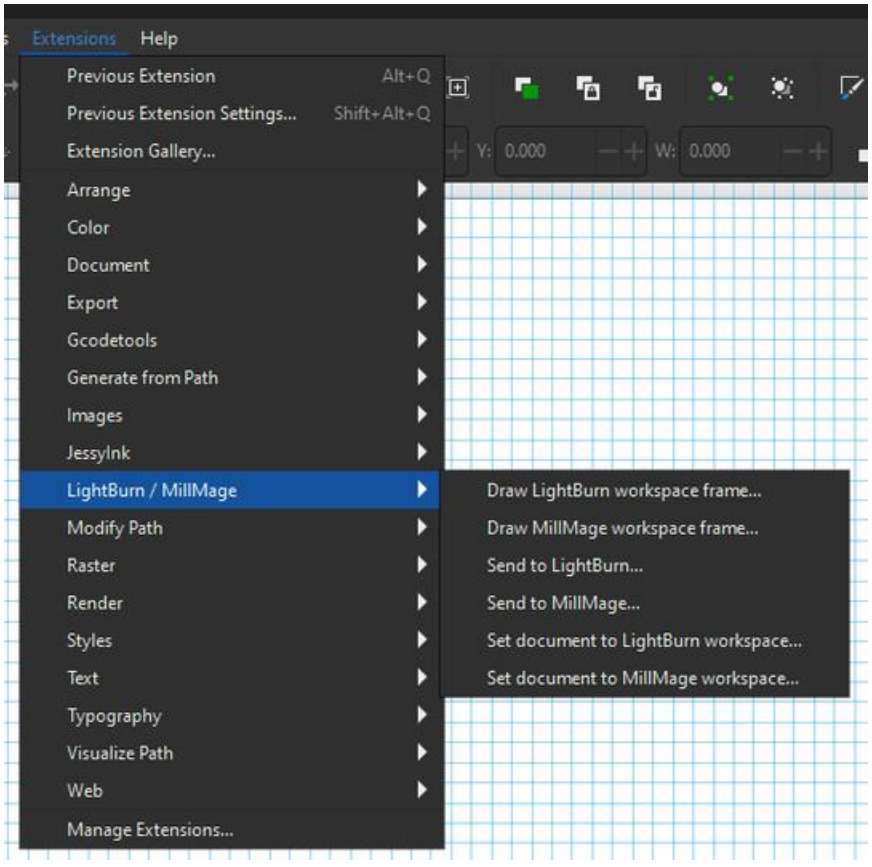
Inkscape Sample Extension

> **Warning**
>
> This is an early demo, installed manually and not through an addon manager. More features are planned.

Our Inkscape sample extension talks to LightBurn / MillMage over the REST API to draw the machine workspace as a guide and send your drawing into it — de-duplicating coincident edges so shared cut lines aren't cut twice.

After installation, six new commands appear under **Extensions → LightBurn / MillMage**:

- Set document to LightBurn/MillMage workspace — sizes the current document to the machine's workspace. (see Document size).
- Draw LightBurn/MillMage workspace frame — fetches the machine's workspace size and draws it as a dotted, brand-coloured background rectangle.
- Send to LightBurn/MillMage — uploads the drawing.



Installation

The extension is five files that must stay together. Copy them into a new `lightburn-send` folder inside your Inkscape **user extensions** directory, then restart Inkscape.

1. Find your user extensions directory

You can also find your user extensions directory in Inkscape under **Edit → Preferences → System → User extensions**.

OS	Path
Windows	`%APPDATA%\inkscape\extensions`
Linux	`~/.config/inkscape/extensions`
macOS	`~/Library/Application Support/org.inkscape.Inkscape/config/inkscape/extensions`

2. Create the new `lightburn-send` folder/directory and copy the files there

You can do this manually if you want, or using the commands shown below.

Windows (PowerShell):

```
$dst = "$env:APPDATA\inkscape\extensions\lightburn-send"
New-Item -ItemType Directory -Force $dst | Out-Null
Copy-Item examples\inkscape\*.inx, examples\inkscape\*.py $dst
```

Linux / macOS:

```
dst=~/.config/inkscape/extensions/lightburn-send
mkdir -p "$dst"
cp examples/inkscape/*.inx examples/inkscape/*.py "$dst"
```

These commands copy the six `.inx` files plus the Python modules `lightburn_send.py`, `lightburn_frame.py`, `lightburn_newdoc.py`, `lightburn_client.py`, `lightburn_common.py`, and `dedupe.py` into the `lightburn-send` folder.

> **Tip**
>
> If you're editing the code, symlink the folder instead so changes are picked up: `ln -s "$PWD/examples/inkscape" "$dst"`.
>
> If the sentence above doesn't mean anything to you, ignore it - you probably don't need to do this.

3. Restart Inkscape

The commands appear under **Extensions → LightBurn / MillMage**.

Using the Extension

1. Make sure LightBurn / MillMage is **open** (the API listens whenever the app is running).
2. In Inkscape: **Extensions → LightBurn / MillMage → Send to LightBurn** (or **Send to MillMage**).

3. The first send triggers a consent dialog in the app — approve it. The shared secret is stored under `…/lightburn-rest/inkscape-secrets.json` and reused on later sends (LightBurn and MillMage each get their own).

The correct port is used automatically — **19520** for LightBurn, **19521** for MillMage. If you've customised the REST API port, override it on the **Advanced** tab.

Other options:

- **De-duplicate shared segments** - Removes duplicated segments if checked, skips de-duplication if unchecked.
- **Selection only** - sends only the selected objects if checked, or the entire document if unchecked.

> **Info**
>
> The consent dialog requests two scopes: **project** (read the workspace size) and **upload** (send files). If you paired an earlier build that only asked for `upload`, thenext run re-pairs automatically.

Send

1. Collects the selected shapes (or the whole document).
2. **De-duplicates exact-coincident line segments** — when two shapes share an edge, only the first keeps it; the laser cuts that edge once.
3. Positions the geometry into the workspace, in priority order:
   1. A **workspace frame** if present
   2. Otherwise a **document sized to the workspace**
   3. Otherwise it drops the content at the view center at native size.
4. Uploads the result as SVG via `POST /api/file/upload`. The open document is never modified.

The de-dupe is intentionally simple (see `dedupe.py`): it matches straight line segments by endpoint, direction-independent. **Curves always pass through unchanged** — coincident curves and partial/collinear overlaps are a known limitation, to be revisited with a geometric union if needed.

Document Size

Run **Set document to LightBurn workspace** on a fresh, blank document to size its page to the machine's workspace at **1 mm = 1 user unit**. Drawing at real size then sends at real size. (Inkscape extensions act on the open document and can't open a new window — make the new document first, then run this.)

The command also tags the document (a namespaced attribute on the root, plus the workspace size in mm), so **Send positions your work relative to the page edges** — no frame needed. Add a frame too if you want a movable workspace that isn't the whole page.

Workspace frame

Run **Draw LightBurn workspace frame** to fetch the machine's workspace size and drop a dotted, brand-coloured rectangle (LightBurn red, MillMage purple) at the back of the canvas It's a guide — never sent.

- **Position your work**: move and resize the frame; your designs' position and size relative to the frame are mapped into the real workspace on send. Bottom-left of the frame is the workpiece origin.
- **Scale is uniform** (the frame represents the machine at a locked aspect). If you stretch the frame off-aspect, Send uses its width for a single scale and says so — geometry is never distorted.
- The frame is tagged with a hidden identifier (a namespaced attribute, not its colour) and its workspace size in mm, so it's re-found instantly after saving and the Send path needs no extra API call. Re-running the command refreshes an existing frame rather than adding another.

Resetting authorization

If a command fails with an authorization error that doesn't clear on its own, tick **Reset authorization (pair again on this run)** on the **Advanced** tab of any of the three commands. It forgets the secret saved for that host and port, so the run that follows pairs from scratch — approve the fresh consent dialog in the app. Only the host you're running against is cleared; other hosts, and the other product, keep their own authorization.

The plugin already drops a secret by itself when the app answers 401/403, so the checkbox is for the cases that survive that — a half-written secrets file, a pairing the app no longer recognises, or a machine that was reimaged. The store is a plain JSON file at `…/lightburn-rest/inkscape-secrets.json`; deleting it clears every host at once.

Sending over the network

By default everything talks to `localhost`. To drive a LightBurn / MillMage instance on **another computer**, enable network access in that app's settings, then set the **Host** field (IP or hostname) on the **Advanced** tab of any command. The first send pops up the consent dialog **on the host machine** — approve it there.

Each host gets its own stored secret. Every command (set document, draw frame, send) honours the Host override.

Device safety check

When you size a document or draw a frame, the **device profile name** is recorded alongside the workspace size. On Send, the plugin checks the listening instance's device and workspace against what the document was built for. If they differ — e.g. focus switched to a different machine profile — Send **aborts** and tells you both the expected and the live device/size, so you don't scale a design to the wrong machine. Switch back to the right instance, or tick **Send anyway** on the Advanced tab to override. (Inkscape can't show a mid-run prompt, hence the checkbox.)

Requirements & testing

- Inkscape 1.0+ (uses the bundled `inkex` Python module). No extra packages. Verified against Inkscape 1.4 / inkex 1.4.
- The de-dup core has a standalone self-check: `python dedupe.py`.
- Sample drawings live in `test-svgs/`: two squares sharing an edge, and two triangles sharing a diagonal. Open one in Inkscape and send it — the shared edge should be dropped from the second shape, and each shape's colour preserved.

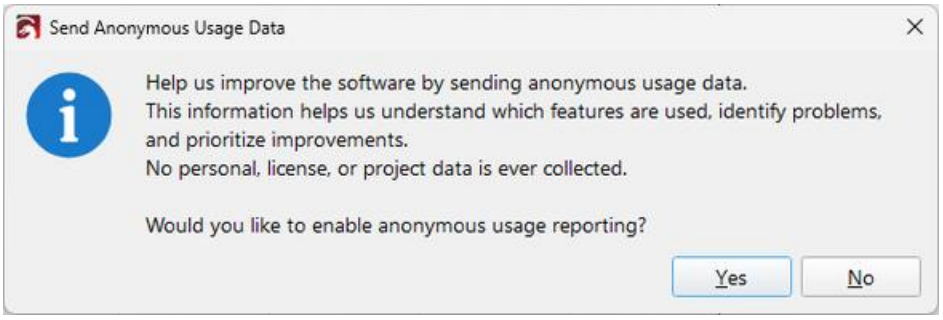
License

GPL-3.0 — see LICENSE. The extension builds on Inkscape's `inkex` module, which is GPL (v2-or-later), so this example is GPL too. The non-`inkex` modules (`dedupe.py` , `lightburn_common.py` , `lightburn_client.py`) are pure standard-library and independent of `inkex` .

View Usage Data

LightBurn's anonymous usage data collection is entirely opt-in for full release versions of the software. If you don't chooose to share this information with us, it won't be sent.

When you first open a full release version of LightBurn, you'll be asked if you would like to enable anonymous usage reporting.



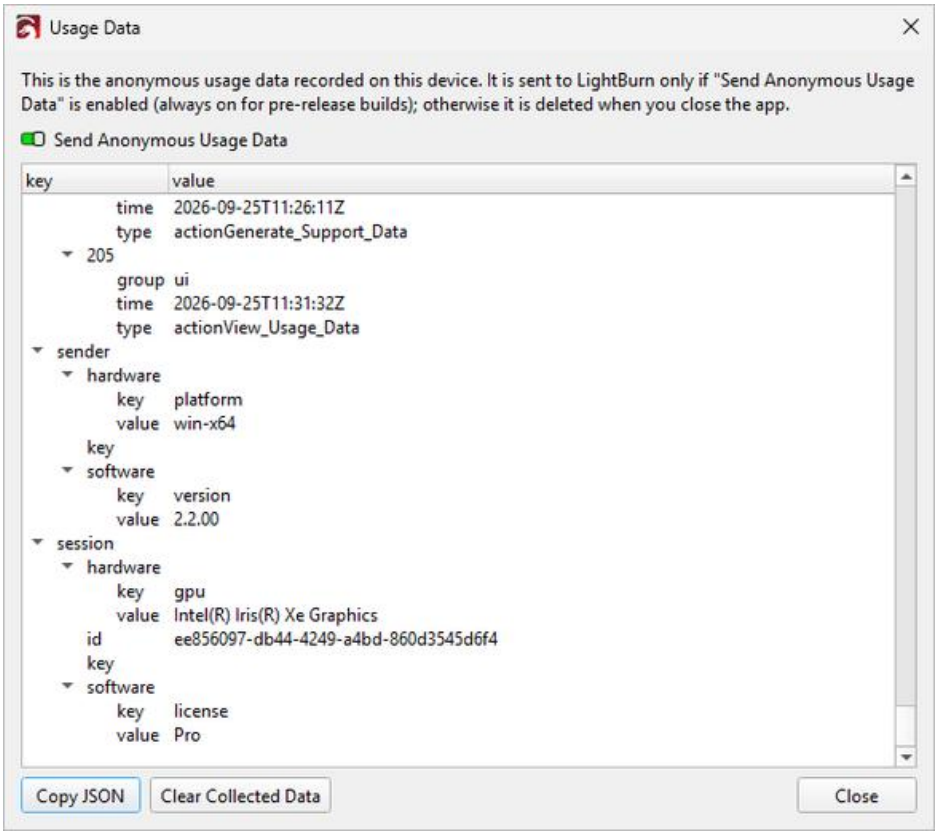
> Help us improve the software by sending anonymous usage data. This information helps us understand which features are used, identify problems, and prioritize improvements. No personal, license, or project data is ever collected. Would you like to enable anonymous usage reporting?

If you click **Yes**, anonymous usage reporting will be enabled, and can be viewed, managed, and disabled at any time from Help → **View Usage Data**.

If you click **No**, anonymous usage reporting will be disabled.

Usage Data Window

The **Usage Data** window allows you to view, manage, enable, and disable the reporting of anonymous usage data, which we use to help us identify problems and improve the software. Reporting data is completely optional and opt-in for full release versions of LightBurn. For Release Candidate / Early Access versions, anonymous usage data is always reported and can't be disabled. If you don't want to provide this data, please wait for full release versions.



You can manage data reporting and view the collected data by going to Help → **View Usage Data**.

Send Anonymous Usage Data

If enabled, anonymous usage data will be reported to LightBurn to help with improvements to the software. If this is disabled, data from each session is purged when LightBurn is closed cleanly (not a crash). Anonymous usage data reporting is always enabled for Release Candidate / Early Access builds.

The data described below is collected during a session, but never leaves your computer without your permission. If you've opted in to anonymous usage data reporting, it will be sent once a day while LightBurn is open, and also when you close LightBurn. If you encounter a bug and need to send in a Bug Report, you have the option to send this anonymous usage data along with the bug report.

Copy JSON

Copies the collected usage data as a JSON file so you can view it in other software if you'd prefer.

Clear Collected Data

Clears the collected usage data.

Data Collected

What We Don't Collect

We do not collect Personally Identifing Information (PII), including but not limited to:

- Your name.
- Your location.
- Your actual license key.
- The names you assign to your devices and the IP address or COM port they connect to. Neither of these are needed for aggregating data or guiding software improvements, and could potentially contain identifying information.

What We Do Collect

We collect anonymized data to aggregate and use for software improvements.

- Anonymous, randomly generated unique user ID (UUID) for the installation and for each run.
- LightBurn version.
- LightBurn license type (Core or Pro).
- Hardware (OS, CPU type, and GPU type)
- A timestamped list of events, such as what tools you've used.
- Unit and language settings.
- A list of your devices, containing the following for each device:
  - A globally unique identifer (GUID) to follow a device acros sessions. This is randomly generated, and not derived from the machine, a serial number, or anything about the user.
  - The device protocol, such as Ruida, GRBL, or JCZFiber.
  - The connection type, such as Serial or TCP.
  - For devices created using certain profiles, such as Emblaser, a string containing the vendor and model.
  - The device's workspace size.
  - The type of controller the device uses, such as GCode, DSP, or Galvo.
  - The per-device units and control units settings. This is omitted if the device has never stored this setting.
  - Device classification, for devices with adaptive protocols such as xTool. This is omitted if the device protocol doesn't support it, which is the case for most devices.

Enable Debug Log

Use **Enable Debug Log** to begin recording a log saved in the **My Documents** folder on Windows or **Documents** on Mac. You can send a Debug Log to LightBurn Support to aid in troubleshooting or bug fixing.

> **What is a Debug Log?**
>
> LightBurn's **Debug Log** contains timestamped information about operations performed in LightBurn, any errors that occur, and the general state of the software. In general day-to-day use, there's no reason to have it active, but it can be very helpful to our developers and support team for tracking down bugs and identifying the cause when LightBurn isn't behaving as expected.

Enabling and Disabling the Debug Log

Go to Help → **Enable Debug Log** to begin recording a log of the actions you take in LightBurn. A check mark will appear next to it when it's enabled, and logging will continue until you select Enable Debug Log again to disable it, or restart LightBurn.

No check mark — Debug Log not enabled

Check mark — the Debug Log is enabled

The Debug Log will automatically save to **My Documents** on Windows or **Documents** on Mac, with the file name `LightBurnLog.txt`.

The Debug Log is cumulative — each time you use Enable Debug Log it will append to any existing log, so it's a good idea to delete it after you're finished and have sent the log to LightBurn Support.

Enabling Debug Logging at Startup

If LightBurn is crashing on startup, you might not be able to enable the Debug Log from within the software. If that happens, you can enable Debug Logs by adding `-d` when launching the software. If you don't already know how to do that, don't worry. We've got directions to do this on both Windows and macOS.

Windows

If you're not comfortable with using our directions to launch LightBurn from the command line, jump to our alternate directions.

You can launch LightBurn with Debug Logs active from the command line by doing the following:

1. Open up a **Command Prompt** window:
    1. Click the **Start** menu, then type "CMD".
    2. Windows will automatically find the **Command Prompt** app as the best match.

2. Type the following into the command prompt and press `Enter`:

```
cd "C:\Program Files\LightBurn”
```

3. Next, type the following into the command prompt and press `Enter` to launch LightBurn with the Debug Log activated:

```
LightBurn -d
```

Click here for an alternate method

1. **Find the app shortcut:** Open the Start menu and start typing "LightBurn". You should see the LightBurn icon come up. Don't click on it just yet.
2. **Open the app location in Windows Explorer:** Right-click on the LightBurn entry and select Open File Location from the menu that comes up.
3. **Open the properties window:** Right-click on LightBurn in the new window that opened and select Properties.
4. **Edit the target location:** Find the line labeled Target: and add `-d` at the end, outside the quotation marks around `"C:\Program Files\LightBurn\LightBurn.exe"`.
5. **Open LightBurn:** Open LightBurn however you usually would.

After getting the Debug Log to send to support, make sure you disable logging by removing the `-d` you added.

macOS

On macOS, you can use Terminal to launch LightBurn with Debug Logs activated by following these directions:

1. Open up a **Terminal** window.
    1. Press `Cmd`+`Space` then type "Terminal".
    2. Click Terminal to open it.
2. Type the following into the Terminal window and press `Enter`.

```
cd /Applications/LightBurn.app/Contents/MacOS
```

3. Next, type the following into the Terminal window and press `Enter`:

```
./LightBurn -d
```

LightBurn will launch with Debug Logs active. You can then send the logs to LightBurn Support.

Related Topics

- Generate Support Data
- Get Controller Info

- Troubleshooting

Debug Drawing

Debug Drawing shows the bounds of shapes being drawn in your Workspace. It is mostly intended for use by LightBurn developers as a debugging tool.

Do not use this tool unless directed to do so by a LightBurn developer or support agent.

Go to Help → **Debug Drawing** to enable the tool. You'll see several sub-options appear in the same menu — each will have a checkmark next to them by default.

Click any sub-option to disable the debug display for that option. Click Debug Drawing again to collapse the options and turn off debugging.

A bounding box will appear around each shape in your project, and data will appear in the top left corner of your Workspace — but it likely won't mean much to you if you aren't a LightBurn developer.

Related Topics

- Generate Support Data
- Enable Debug Log
- Generate Path From Cut

Explainers

Explainers

Our explainers go into more detail about advanced topics to help you use LightBurn more effectively.

Laser Types

Read more about different types of lasers and accessories.

Read More

Layer Modes

Read more about the four different Layer Modes in LightBurn: **Fill**, **Line**, **Offset Fill**, and **Image**.

Read More

Speed vs. Power

Read more about how speed and power interact in your cut settings in LightBurn.

Read More

Images Vs. Vectors

Read more about the differences between vector paths and images, and the best ways to use each.

Read More

Explainers

Open vs. Closed Shapes

Read more about the differences between open and closed shapes, why the difference matters, and how to fix them.

Read More

How to Invert a Vector Engraving

Read more about how vector paths interact with each other and learn how to reverse which part of a vector engraving gets engraved.

Read More

Template Files

Read more about Template Files and how they work.

Read More

Overscanning

Read more about what overscanning is, why it's important, and how to make overscanning adjustments.

Read More

Steps/MM

Read more about how and why the steps/mm setting may need to be adjusted if your laser output looks stretched or squished.

Read More

Explainers

About Galvos

Read more about Galvo lasers and how they work.

Read More

Laser Types

There is a diverse array of laser cutting and engraving machines, defined by various characteristics. These characteristics — and how they interact with each other — dictate the qualities of results, safety, speed, and ease-of-use for that machine.

If you're looking for a new laser, see our Hardware Compatibility matrix for information on finding LightBurn-compatible hardware.

LightBurn Compatible Lasers

How a Laser Project is Made

To understand the parts each of the characteristics play, let's briefly touch on how a project goes from idea to physical product.

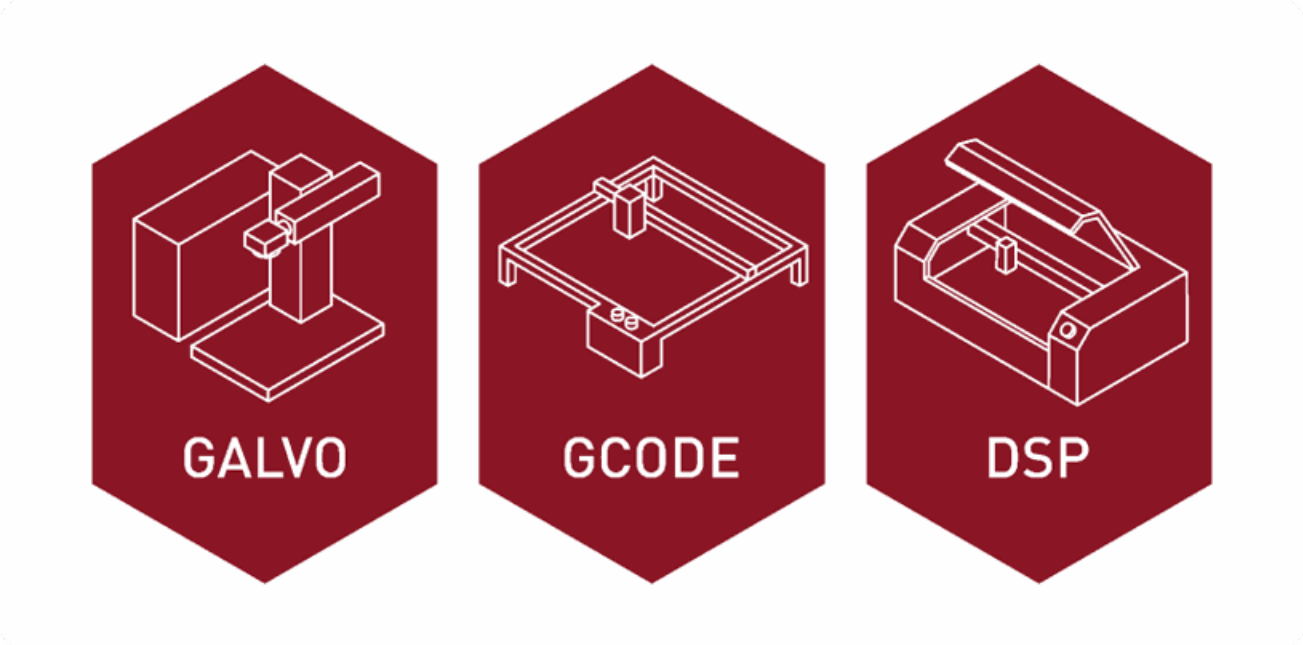
A **design** is created by a user in **design software**, and turned into machine **instructions** (code) through a **control software**.

LightBurn operates as both a design and control software.

The instructions are delivered to a computer within the laser machine called a **controller**. The controller translates the instructions into electrical signals that control the machine's **motion system**. The movement of the motion system delivers the laser beam from the **laser source** (where the beam is generated) to the directed locations on the material.

Each part of this process is explored in more detail throughout the rest of this article.

Controllers and Firmware



The controller — and the firmware it runs — determines whether or not a machine is compatible with LightBurn, and what type of license you need.

A controller is essentially a computer within your laser, and the firmware it runs determines the type of instructions it can translate into signals it delivers to the motion system.

LightBurn needs to know what type of firmware your laser's controller is running to know what "language" to speak to the machine. In most cases, LightBurn can automatically identify the firmware if you set up your machine using Find My Laser, but you'll need to be able to identify the controller yourself in order to complete a manual setup.

LightBurn is currently compatible with three main categories of controllers: GCode, DSP, and Galvo.

- To connect to a GCode-based laser, any LightBurn license will work. If you only have a GCode-based laser, you'll probably want a LightBurn Core license.
- To connect to a DSP or Galvo laser, you'll need a LightBurn Pro license. LightBurn Pro licenses also support GCode-based lasers.

Below is a brief description of each type, as well as a list of common controllers/firmware used by each. If you're not sure what controller or firmware your laser uses, consult your machine's manufacturer, or contact us at support@lightburnsoftware.com.

GCode

Most entry-level diode lasers use GCode-based controllers.

Supported controllers/firmware: GRBL, Smoothieware, Marlin, FluidNC, grblHAL, xTool

DSP

DSP controllers are common in more industrial-grade machines. If your machine is a CO_2 laser in a metal case and has an LCD display, it's probably a DSP model.

Supported controllers: Ruida, Trocen, TopWisdom

Galvo

Galvo lasers use a fixed scanning head mounted to an arm, and project the beam from above. If your laser uses EZCAD2, EZCAD3, or SeaCAD as its default software, it is this type of laser.

Supported controllers: EZCAD2, EZCAD2 Lite, EZCAD3, BSL

> **Does LightBurn support EZCAD3 lasers?**
>
> LightBurn 2.1 adds support for EZCAD3 lasers.

Motion System

The motion system consists of the mechanical components that direct the beam from the laser source to the material. These include parts such as motors, mirrors, lenses, and axes.

The choice of motion system determines which LightBurn tools are available for your machine, as well as the maximum job size, and the speed of the laser.

> **Note**
>
> Many tools in LightBurn are only available for machines with a particular type of motion system.

There are two common motion systems:

- **Gantry motion systems** have frames, wheels, and motors that move a laser head around a work area. Some Gantry lasers use a system of mirrors to reflect a beam from its source, while others hold the laser source and move it around directly. Gantry systems are relatively slow due to the mass of the components moving around, but can have very large working areas and fire the beam straight down at the material.
- **Galvo motion systems** use tiny moving mirrors to bounce the beam to different locations on a large lense, which focuses the beam and points it back down at the work. Because the mirrors are so lightweight, they can bounce the light around at very high speeds, but Galvos are limited by the size of the lense to relatively small jobs.

> **Note**
>
> All Gantry lasers supported by LightBurn have GCode-based or DSP controllers.
>
> Almost all Galvo lasers supported by LightBurn use EZCAD2, EZCAD3, or BSL controllers and require a LightBurn Pro license. However, there are a limited number of GCode-based Galvo lasers that are compatible with LightBurn Core, and require special setup — consult your laser's manufacturer for more information.

Laser

The laser beam is generated by a source that dictates its power and wavelength.

These properties determine which materials the beam will be able to mark, engrave, or cut through.

Source

- **Diode**: similar to an LED light, a semiconductor is pumped with electrical current to produce light. These are generally lower power, with a very fine focal dot which forms a rectangular shape. These generally make nice engraves, but poor cuts.
- **Diode stack/array**: combines the power of many diodes into one beam, to overcome some of the downsides of single-diode beams.
- **Glass tube**: contains a gas (CO_2) that is excited by a DC current to produce a beam. Is cheaper than a Metal RF tube, but needs water-cooling, wears out quicker, and produces a (comparatively) slow "pulsating" beam which has a wider focal dot, and thus is less suited to engraving. The focal dot is round.
- **Metal RF tube**: contains a gas that is excited by a radio frequency to produce a beam. Is more expensive than a Glass tube, but can be air-cooled, lasts much longer, and produces a more rapid beam-pulse, with a finer focal dot, and is more suited to engraving than Glass tubes. The focal dot is round.
- **Solid State Drive**: Use a solid (e.g. ruby) as the lasing medium, instead of a gas.

- **Fiber**: combines several beams into one using fiber optics, to generate more power output.

Wavelength

The wavelength of the light generated by the source dictates what materials the beam can interact with, or will pass through.

- **Blue Diode** — 400-500 nm: interacts with dark-colored materials such as wood, veg-tan leather, and black acrylic, but struggles with light-colored or transparent materials. Can't interact with metals or glass. Produces dark engravings that are great for photos and images.
- **CO_2 Glass/Metal Tube** — 10600 nm: great for cutting most laserable materials, including wood, veg-tan leather, fabric, plastics, transparent and light-colored materials, as well as engraving glass and some metals, and removing rust.
- **Near Infra Red (IR) Diode** — 750-1400 nm (commonly 1064 nm): mostly used for engraving some metals.
- **Infra Red (IR) Fiber** — 1030-2100 nm: great for engraving most metals, removing rust and oxides from metals, and even cutting some.
- **Ultra Violet (UV) (Solid State Drive)** — 150-400 nm — used for cutting and engraving most materials, including glass.

Power

The power dictates the strength of the beam. Lower powered beams may not be able to cut a material that a higher powered version of the same wavelength can, or may require more passes to achieve the same result (increasing job time).

Generally it's best to purchase the highest-powered laser possible, as you can lower power in your settings, to produce "softer" results, where required.

Connection

The connection is the method by which LightBurn transfers the instructions to the machine. The current methods supported are as follows:

- **USB cable**: limited to short distances, USB cables and prone to errors, but are common in many machines. They don't work well when connected via hubs, splitters, or extension cables.
- **Network/Data Cable**: assentially an internet cable, these can be used over long distances and retain great signal strength.
- **LightBurn Bridge**: for use with Ruida controllers where a network cable is impractical, LightBurn Bridge relays a signal over TCP.
- **Wifi**: available on some GRBL controllers, this communication method is wireless.

Accessories

There are various accesories and non-critical or optional features available from manufacturers for machines. Some are explored below.

- **Homing/limit switches**: critical for acurate job-placement and replicability, these small buttons are triggered when the axis of a gantry machine reached its "home" position, and allow the machine to position the laser according to a known physical location.
- **Cameras**: allow the user to visually place their design onto locations on their physical material. There are two main kinds:
  - **Stationary cameras**: by far the more popular, these cameras are mounted in a position over the laser's work area that remains the same between jobs and enables a full picture of a machine's bed.
  - **Head-mounted cameras**: these cameras are attached to the head of a laser, near to the beam. These allow close and safe vicsibility of the laser as it works, along with greater accuracy of design placement, at the cost of having to move the head into a location in order to see it.
- **Rotaries**: allow cylindrical objects to be engraved. There are two main types, chucks and rollers.
  - **Roller-based rotaries**: carry an object by rotating wheels beneath it. This style of rotary is not suited to objects that aren't perfectly cylindrical or are prone to slipping, and are tricky to dial in for full-wrap engraves.
  - **Chuck-based rotaries**: physically hold the workpiece in jaws, and rotate around a known axis by a known number of steps (pulses of the motor).
- **Pass-through gates**: these are doors that allow material larger than the laser's bed to be placed in the machine, so that a job can be completed in sections.
- **Exhaust**: a critical safety feature of all laser-machines, these systems ferry the smoke and fumes produced by a job away from the user.
- **Enclosures**: another critical safety feature, appropriate enclosures protect a user's eyes (by shielding the user's eyes from the beam), and lungs (by containing the fumes so the exhaust can work effectively).

Layer Modes

In LightBurn, there are four **Layer Modes** that determine how your laser will cut or engrave graphics in your design. Below, we'll explain the effect and purpose of each type of Layer Mode.

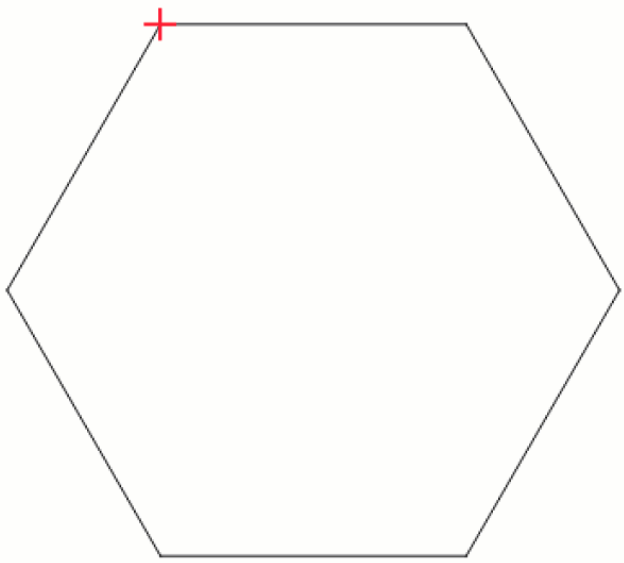
You can use the Color Palette to assign different colors/layers to your graphics, and adjust the mode of those layers in the Cuts / Layers Window or Cut Settings Editor.

> **Tip**
>
> Use the Preview window to view a simulation of the output to your laser before starting the job.

Line Mode

Line Mode tells your laser to trace a path around the contours of vector graphics.

If you command the laser to move quickly, or with low Power, it will likely just etch the surface (sometimes called scoring). If you move slowly and with high power, especially with a CO_2 laser, the laser will cut deeper, or even completely through the material. The only difference between surface marking and cutting with Line Mode is the Power and Speed you set.



See Cut Settings Editor — Line Mode for information on additional settings specific to Line Mode.

Fill Mode

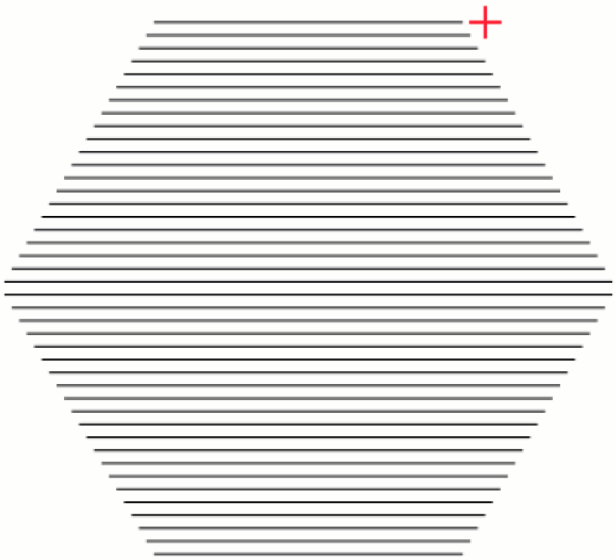
> **Closed Shapes Only**

This feature only works with shapes that are closed. A shape is closed when it is a complete, continuous loop whose start and end points are the same. When a shape's start and end points are different, it is open.

See Open vs. Closed Shapes for more information.

Fill Mode tells your laser to etch parallel lines within the boundaries of vector graphics.

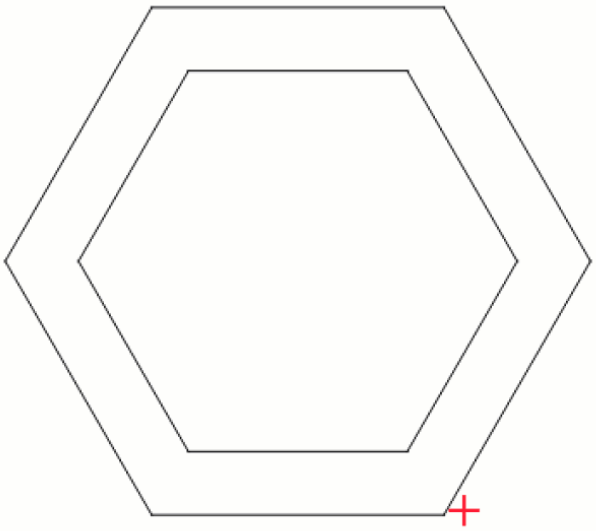
The distance between each line is determined by the Line Interval setting, and the Speed and Power settings control the darkness or depth of the etch. Use low Intervals, low Speeds, and high Powers for deep engravings or dark marks, or tell your laser to move quickly at low Power to lightly etch or mark the surface.



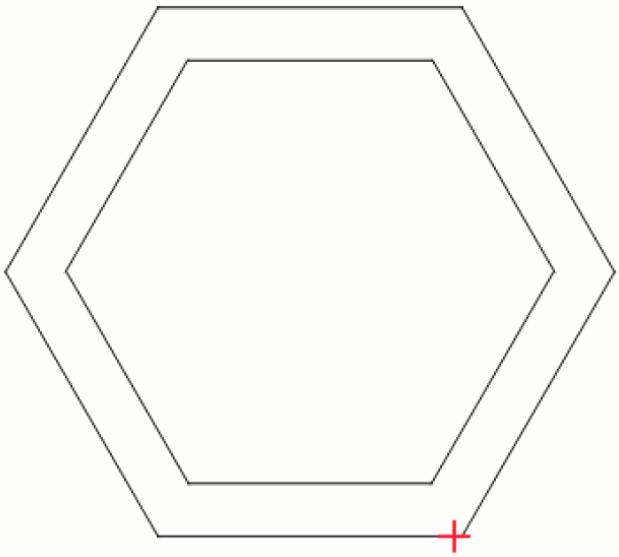
How Graphics Set to Fill Mode Interact

Graphics set to **Fill Mode** interact with one another differently depending on whether they are set to the same or different layers.

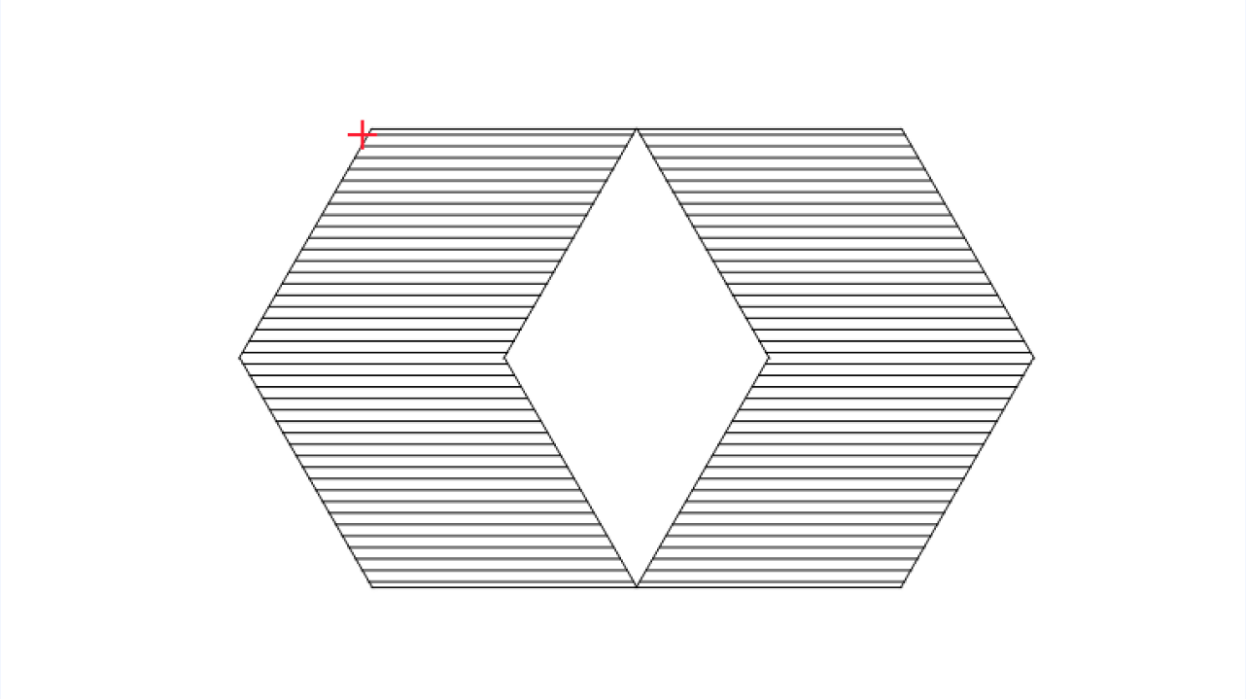
If two shapes are set to the same layer, and one is inside of the other, the area between the two outlines will be filled.



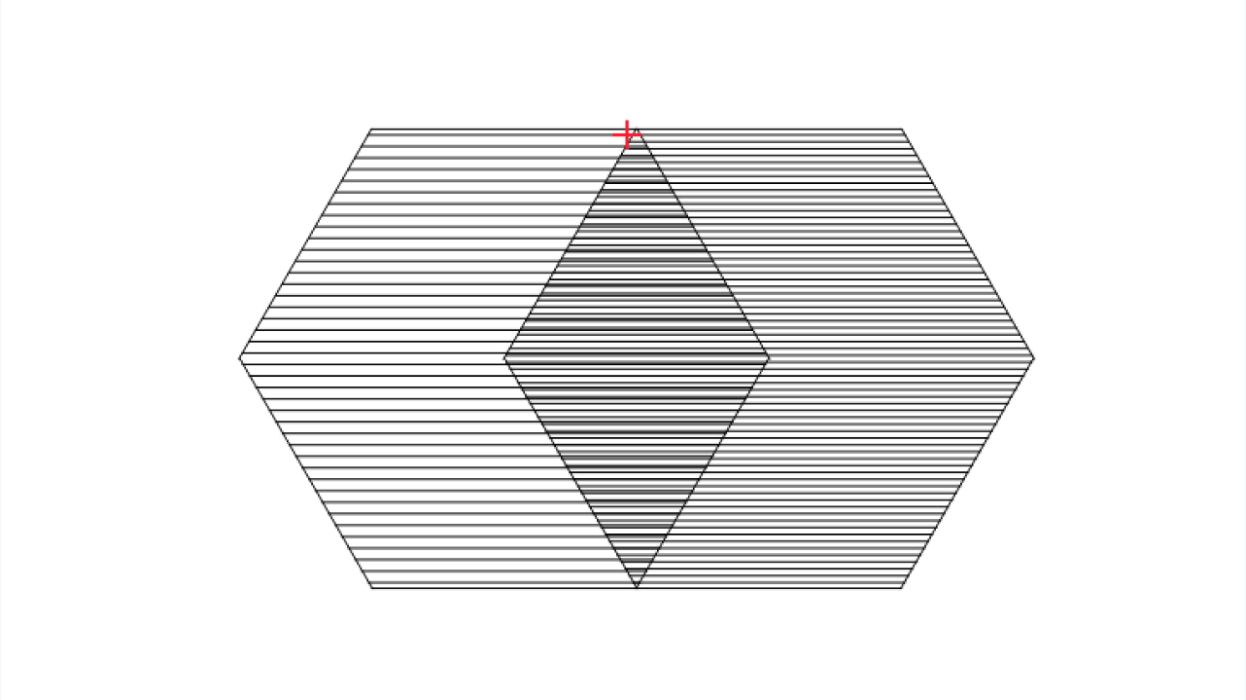
If those same two shapes are set to different layers, the entirety of the outer shape and the entirety of the inner shape will both be engraved, meaning the area within the inner shape will be engraved twice.



The same logic holds true when shapes only partially overlap — if the shapes are assigned to the same Fill Mode layer, the area where they overlap will not be engraved. If they are set to different Fill Mode layers, the area where they overlap will be engraved twice.



Partially overlapping shapes set to the same layer



Partially overlapping shapes set to different layers

See Cut Settings Editor — Fill Mode for information on additional settings specific to Line Mode.

Offset Fill Mode

Closed Shapes Only

This feature only works with shapes that are closed. A shape is closed when it is a complete, continuous loop whose start and end points are the same. When a shape's start and end points are different, it is open.

See Open vs. Closed Shapes for more information.

Offset Fill mode works similarly to Fill Mode, but instead of telling your laser to scan parallel lines within the boundaries of a vector graphic, it tells your laser to etch concentric lines that follow the outline of the shape.

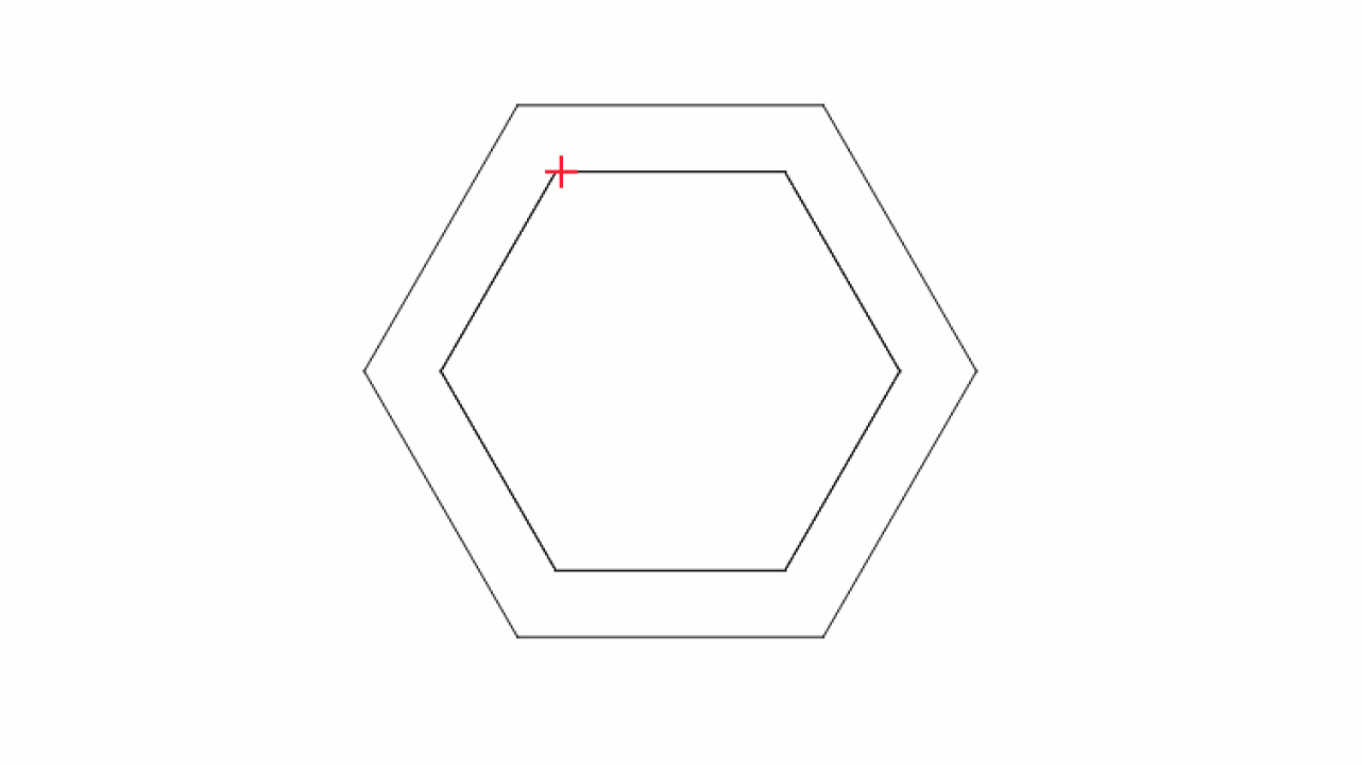
As with both Line and Fill Mode, Speed and Power control the depth and darkness of the etch, and, as with Fill Mode, the Line Interval setting controls the spacing of the lines.

Offset Fill is intended for designs with a lot of empty space within graphics, to cut down on the amount of travel time necessary to fill large, hollow shapes. If that description fits your design, Offset Fill will most likely be much faster than Fill Mode. If that doesn't apply to your graphics, it's usually best to use Fill instead of Offset Fill. Offset Fill is generally not a good choice for complex shapes.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=haLj85zVjBs

Offset Fill Computation Time

Offset Fill is very computationally expensive, and the more complex the design, or the smaller the Line Interval set, the more potential there is for the computation to cause LightBurn to hang. Sometimes graphics with stray nodes or unusual paths can confuse the calculation interminably.



See Cut Settings Editor — Offset Fill Mode for information on additional settings specific to Line Mode.

Image Mode

Several additional settings that affect the quality of the engraving are applied on a per-image basis, using the Adjust Image tool or Shape Properties Window.



See Cut Settings Editor — Image Mode for information on additional settings specific to Image Mode.

Speed vs. Power

Understanding the relationship between **power** and **speed** is an essential part of being able to produce high quality cuts and engravings, as well as optimizing your settings to finish jobs faster.

Always pay attention to units of distance and time when entering Speed settings

When entering **Speed** values recommended by your laser's manufacturer or other LightBurn users, make sure to use the same units of distance and time as those from the recommendation, or to convert the values to your preferred units.

A given number of **millimeters per second** is much faster than that same number in **millimeters per minute**. Mixing up units can lead to reduction in power output due to unexpectedly high speeds, or excessive power output — and even fire — due to unexpectedly slow speeds.

Change your Control Units in the **Dimensions / Units** tab of the **Device Settings** window. LightBurn automatically converts any existing values when you switch between units.

Speed vs. Power Basics

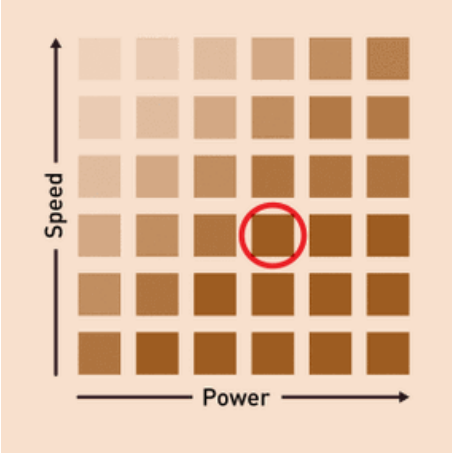
The two primary settings controlling the depth of a cut or the darkness of an engraving are power and speed.

Power controls the laser's intensity. Higher powers allow you to cut deeper or make darker marks for engravings, but at the cost of potentially charring or melting edges, producing excessive smoke, and making a wider kerf.

Speed controls how fast the laser moves. Lower speeds will deliver more power and heat to the material, as the laser takes longer to pass over any one point. Higher speeds can complete a job faster, but will require higher power. High speeds can also exacerbate or make any existing mechanical issues more apparent.

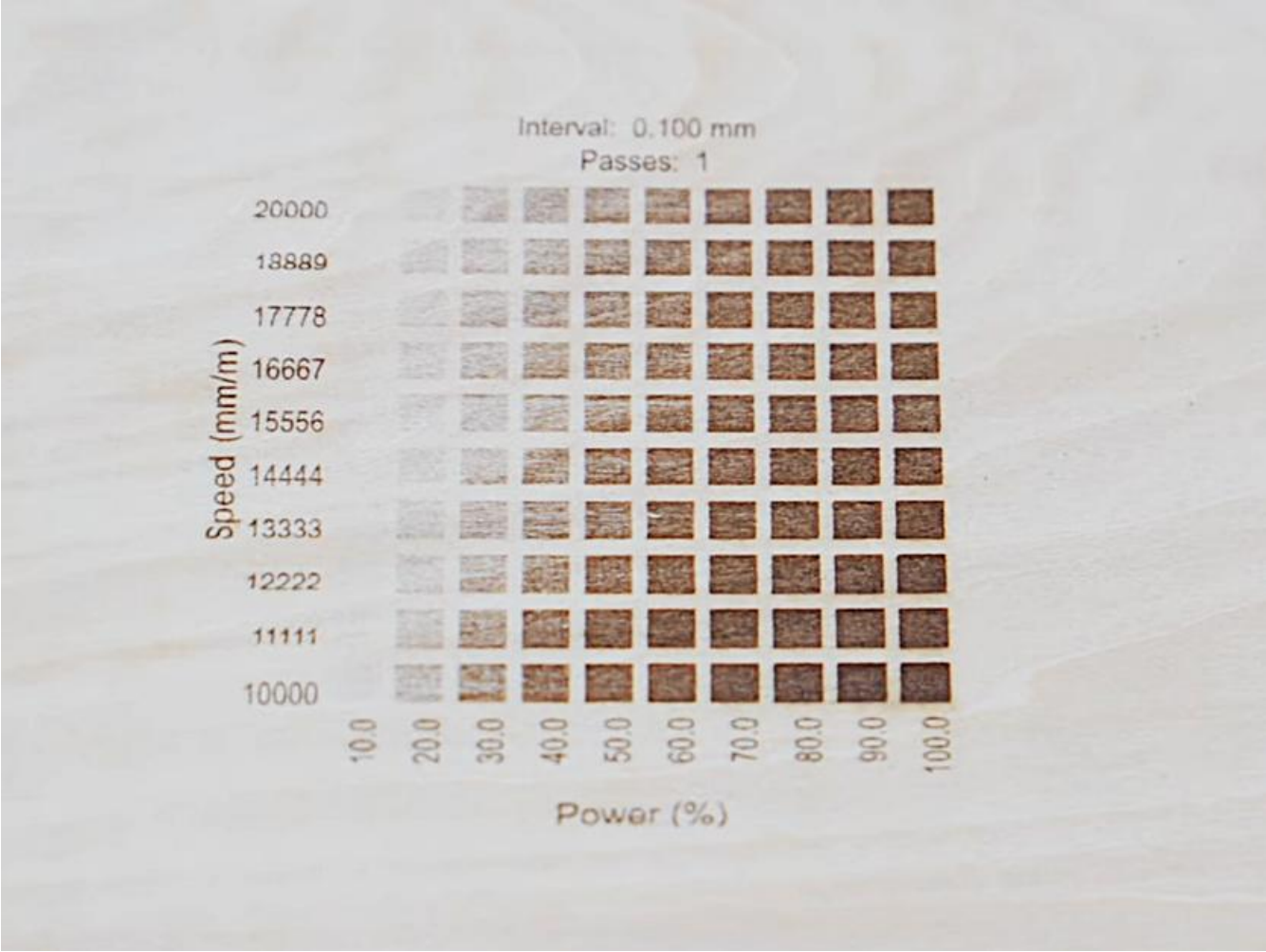
This image shows a simplified example of a test grid similar to what LightBurn's Material Test Generator produces. This example helps show the relationship between speed and power:

- In the top left corner, where the speed is high and the power output is low, the mark is faint and barely visible.
- The marks get darker both going to the right (increasing power) and going down (decreasing speed).
- In general, to make darker marks or deeper cuts, you can increase power, decrease speed, or do both.



You might also have noticed that the bottom right corner isn't noticeably darker than the circled mark. Beyond a certain point, increasing power or decreasing speed has limited usefulness. In this example, the circled mark would get similar results to a slower and higher power cut, but faster and with reduced power consumption.

For a more realistic example, see the picture below showing an actual material test. As with the example above, the marks get darker and deeper with increasing power and decreasing speed.



Optimizing Cut Settings

The best power and speed combination will depend heavily on not only your laser and material, but what you're aiming to achieve.

- For image engraving, especially with CO_2 lasers, going slowly at a low power typically gives the best results. See 5 Steps to Perfect Image Engravings for more information.
- When cutting, it's best to use LightBurn's Material Test Generator to find a setting that cuts through the material cleanly without scorching. Look for the lowest power and highest speed that cuts through reliably.
- If you're having issues with scorching or melting the edges of your material, try using multiple passes rather than continuing to increase power.

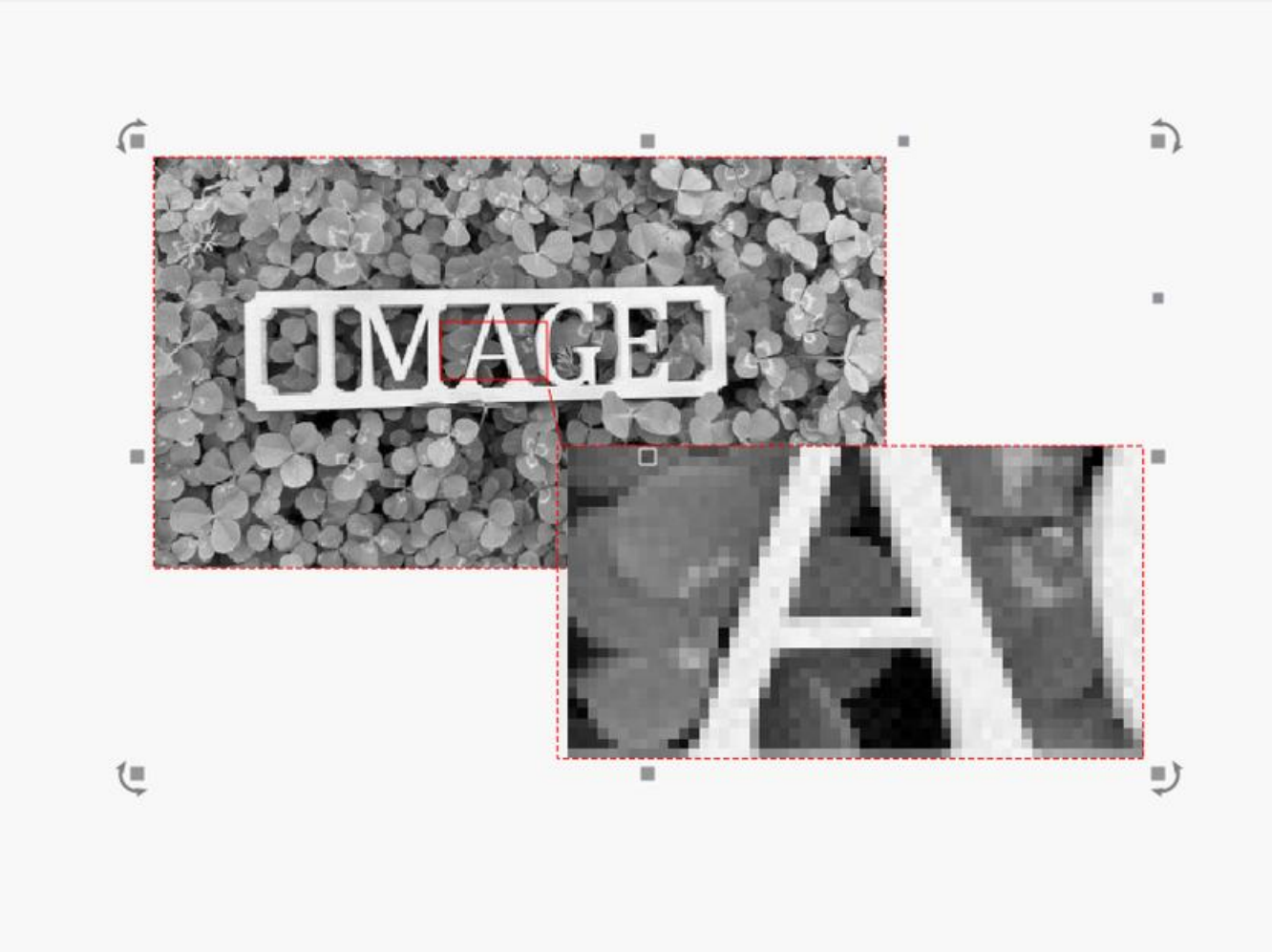
Related Topics

- Material Test Generator
- Cut Settings Editor
- 5 Steps to Perfect Image Engraving

Images vs. Vectors

Understanding the difference between **images** and **vectors** is essential for creating and editing custom projects. It influences the tools and methods used in LightBurn, as well as the settings, speed, and quality of jobs.

There are two main ways a computer can describe something visually:

Images	**Vectors**
	
Images are made up of pixels — tiny blocks of color in a grid. They're sometimes called bitmaps or rasters, Images include digital photographs, flatbed scans, digital drawings, and computer-generated renders.	**Vectors** a LightBurn how far y
Common file types: `.png`, `.jpg` / `.jpeg`, `.bmp`, `.gif`, `.tif` / `.tiff`	**Common**

Both Images and Vectors can either be made within LightBurn, or imported from files made in other software. See Importing for more information about file types that are compatible with LightBurn, and read on for more information about images and vectors.

Images

Image Uses and Laser Behavior



Images can only be used for engraving, not cutting. When a laser produces an image, it does so by scanning side-to-side across the material surface. As it moves, it can create shading by either switching the beam on and off to form little dots, or by varying the power. Producing an image in this way generally takes a considerable amount of time, as each line is usually a fraction of a millimeter apart, and it takes hundreds of lines to form most images.

Image Appearance and Quality

When an image is used, LightBurn assigns it to a layer set to Image Mode. When selected, images appear with a border in the color of the layer they are assigned to. If you zoom in on an image, or make an image larger, the blocks of color will eventually become noticable, resulting in an undesirable "pixelized" quality.

Image Operations

There are various types of image processing modes, the choice of which has a significant effect on the final result.

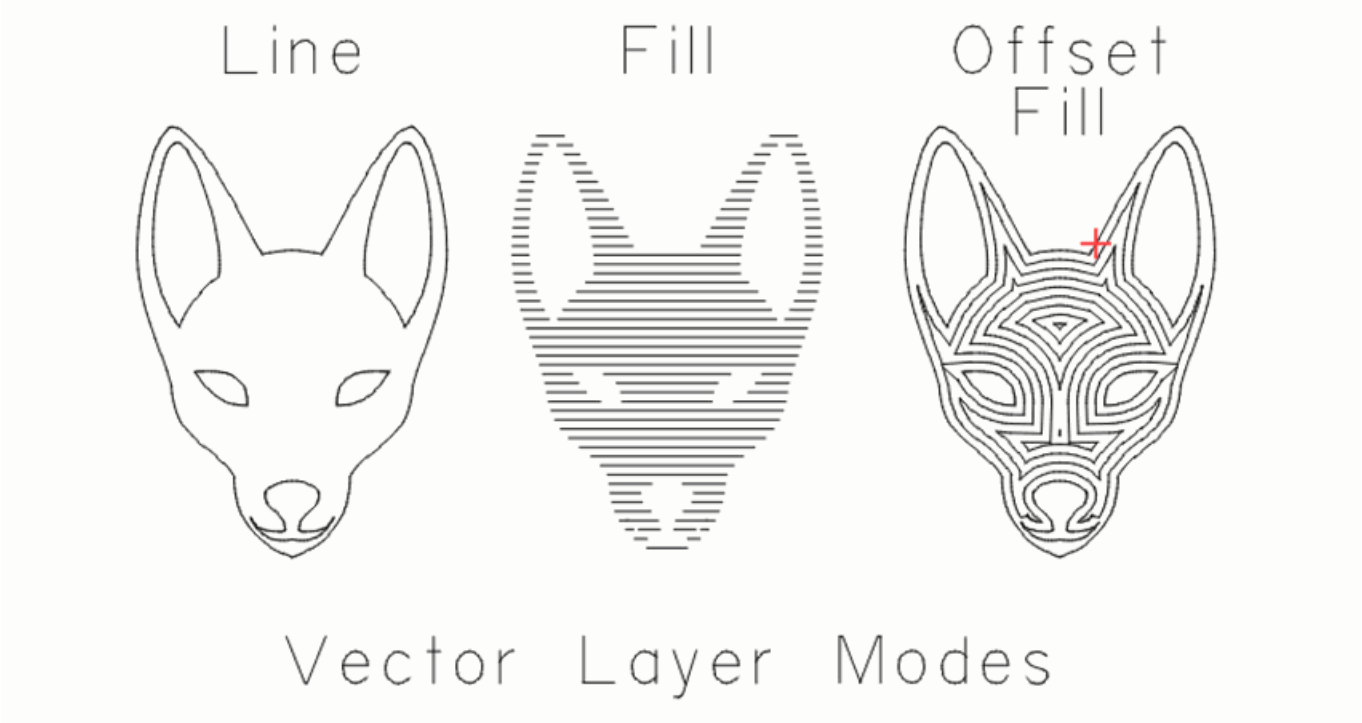
Vectors

Vector Uses and Behavior

You can use Vectors to:

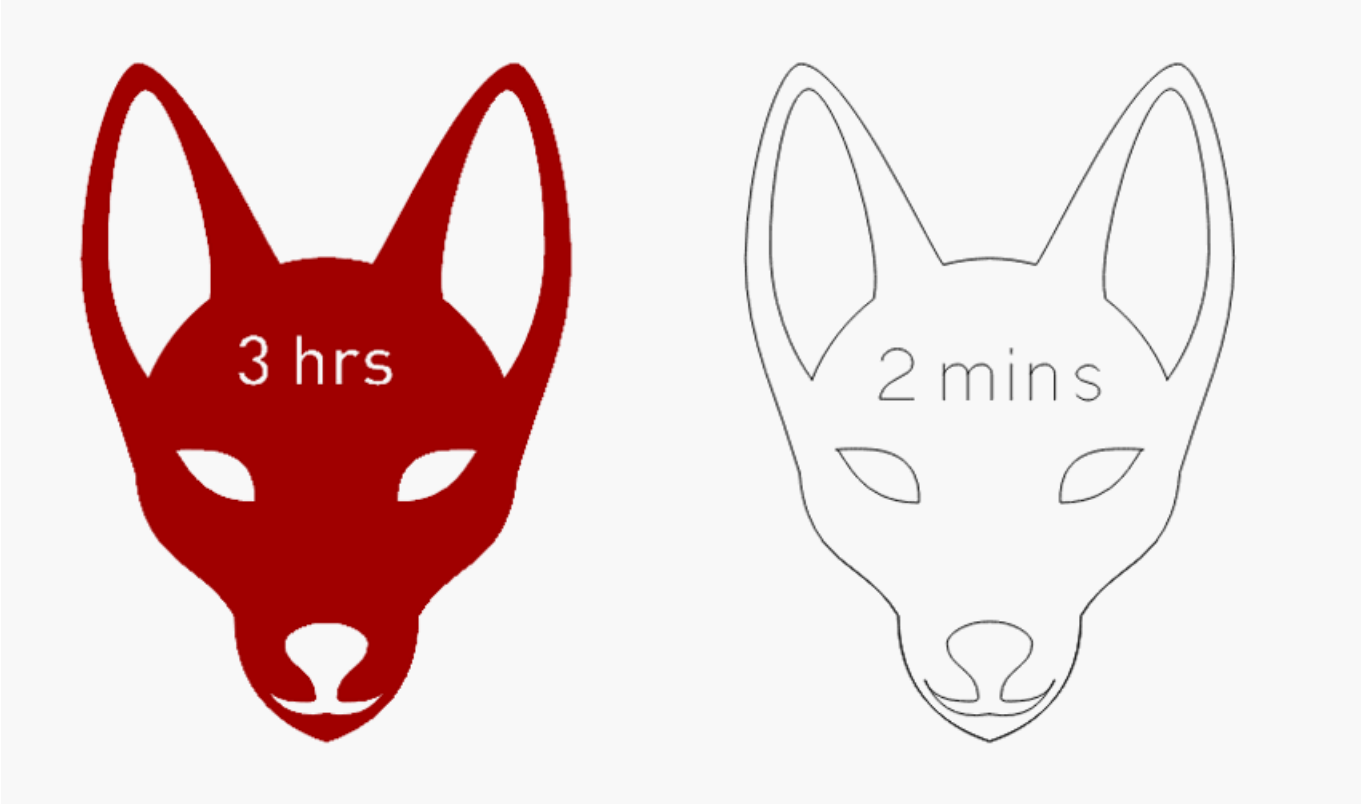
- Cut through a material
- Score/mark lines and perimeters
- Engrave areas

To use a vector to cut, score, or make a line or perimeter, you must assign the shapes to a layer set to Line Mode. This tells the laser to trace/follow the paths and shapes of the vector.



Vectors used to engrave an area must be closed shapes and assigned to a layer set to either Fill or Offset Fill. Engravings are produced in a similar fashion to images — with many lines next to each other — however, there is no variation in power, nor dot-dithering, so the result is a flat color without gradients or shading.

Filling areas is generally much more time consuming than marking or scoring them, so it's a good idea to consider if a design can be produced using Line layers.

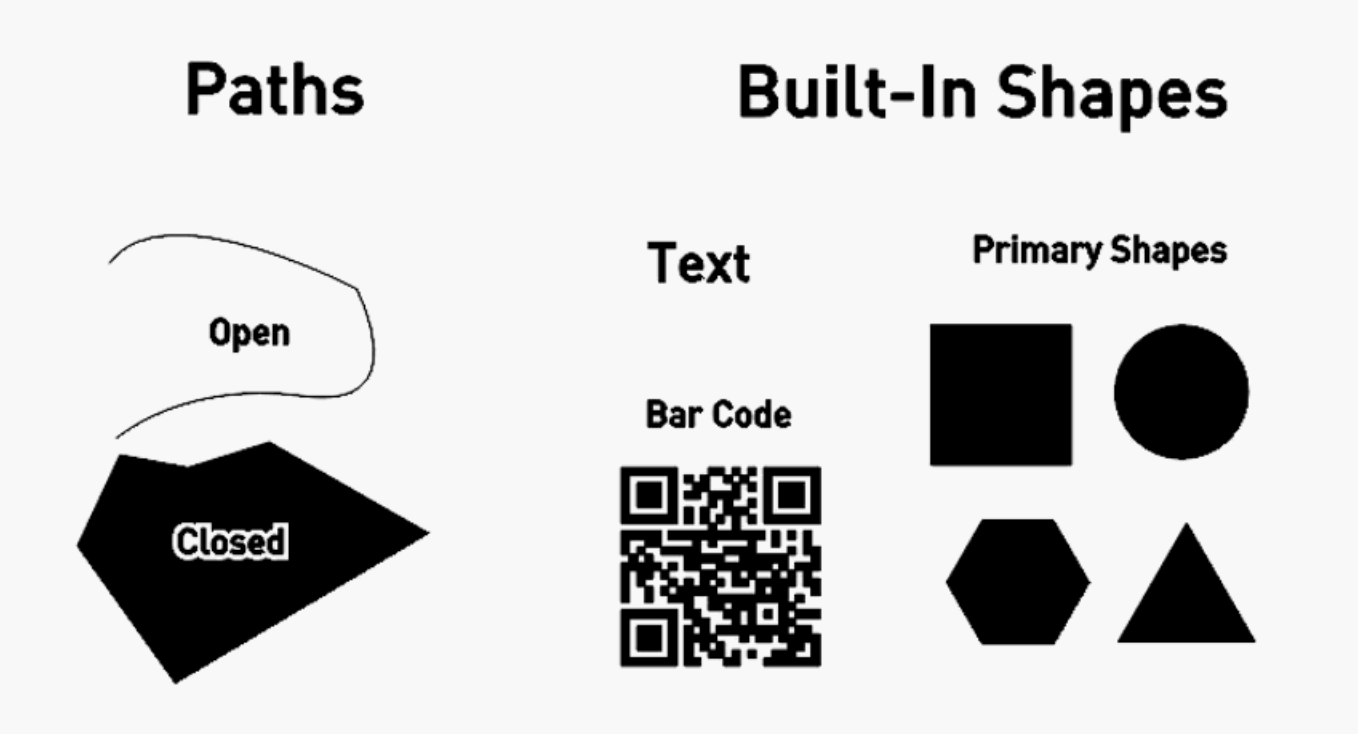


Wireframe vs Filled View

By default, Fill Mode layers will appear as outlines, which makes it easier to spot and select vectors that are behind other vectors.

Select a View Style from the Window Menu, or toggle between View Styles by pressing `Alt`/ `Option`+`Shift`+`W`.

Creating and Editing Different Types of Vectors



The most simple form of vectors in LightBurn are **paths**, created using Draw Lines and edited using Edit Nodes. These paths can be open or closed.

LightBurn also offers more intelligent vectors in the form of **built-in shapes**. These have properties that remain editable so long as the object isn't converted to a path. Built-in shapes include Text, Bar Codes and Primary Shapes such as Ellipses, Rectangles, and Polygons.

Converting Between Images and Vectors

There are a few ways to convert graphics between images and vectors. However, these processes are often not required, and may reduce quality.

You can turn images into vectors using Trace Image. This is often used by people wishing to engrave "clip art". However, this process results in an estimation of the image, and often the results from using Threshold image processing mode are better. Trace Image is best used when you cut, edit, or scale up an image that is too small to result in a nice engraving at the current size — vectors can be upscaled infinitely without losing quality.

You can turn vectors into images using Convert to Bitmap. This is often used by people wishing to repair a vector that was not well optimized for lasering. However, this conversion process eliminates the ability to scale infinitely and edit the vector directly, so it is usually better to repair the vector.

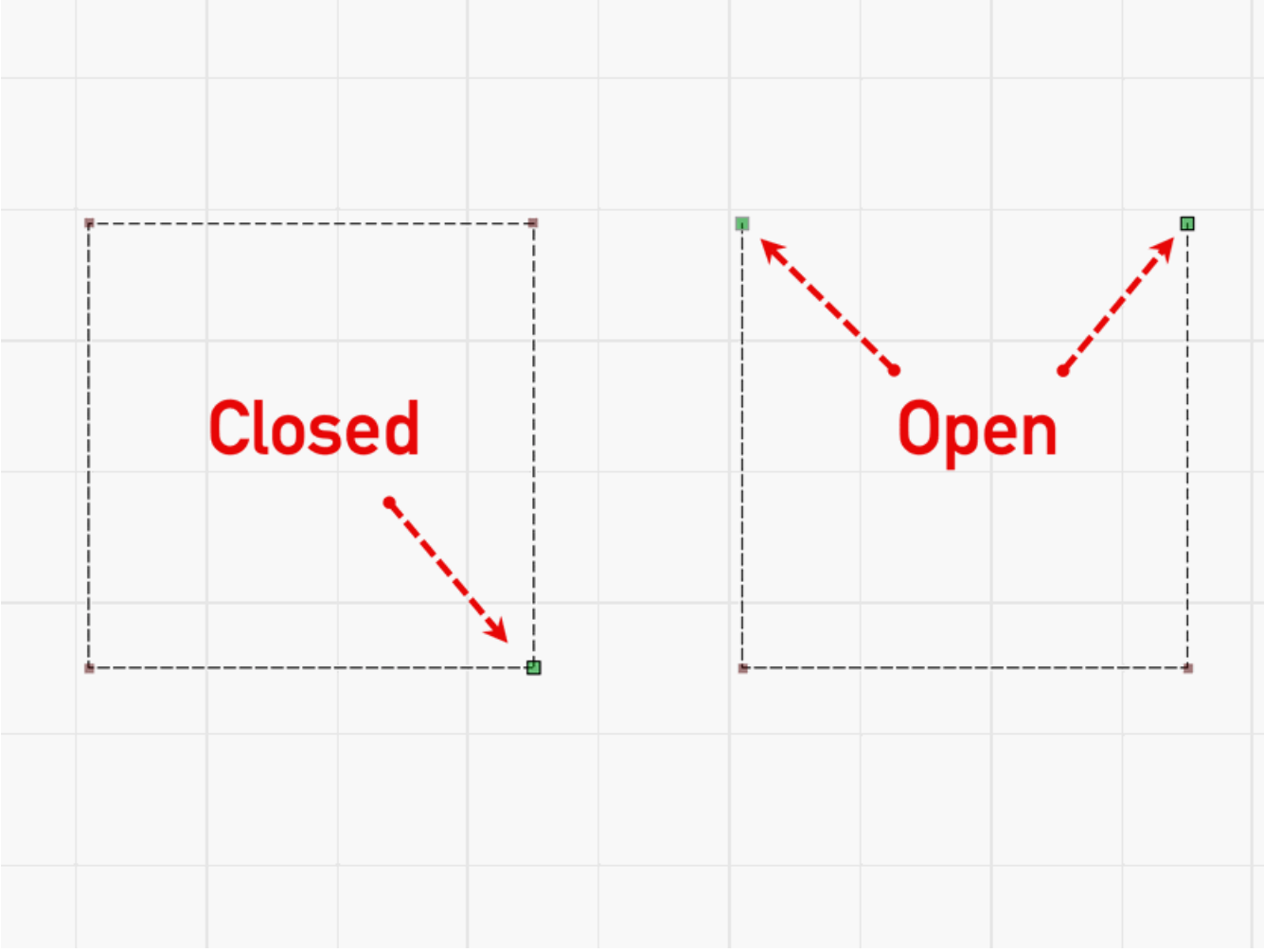
Open vs. Closed Shapes

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=lQ-qhpQypCk

All shapes in LightBurn are made up of lines, curves, and nodes. Sometimes those nodes are connecting points between lines or curves, and sometimes they represent the start or end point of a segment or sequence of segments.

A shape is closed when it is a complete, continuous loop whose start and end points are the same. When a shape's start and end points are different, it is open. A shape can look closed but be open if the start and end points are very close, but not the same.

You can view a shape's nodes by selecting it and enabling the Edit Nodes tool. Start and end nodes are green, and slightly larger than connecting nodes.



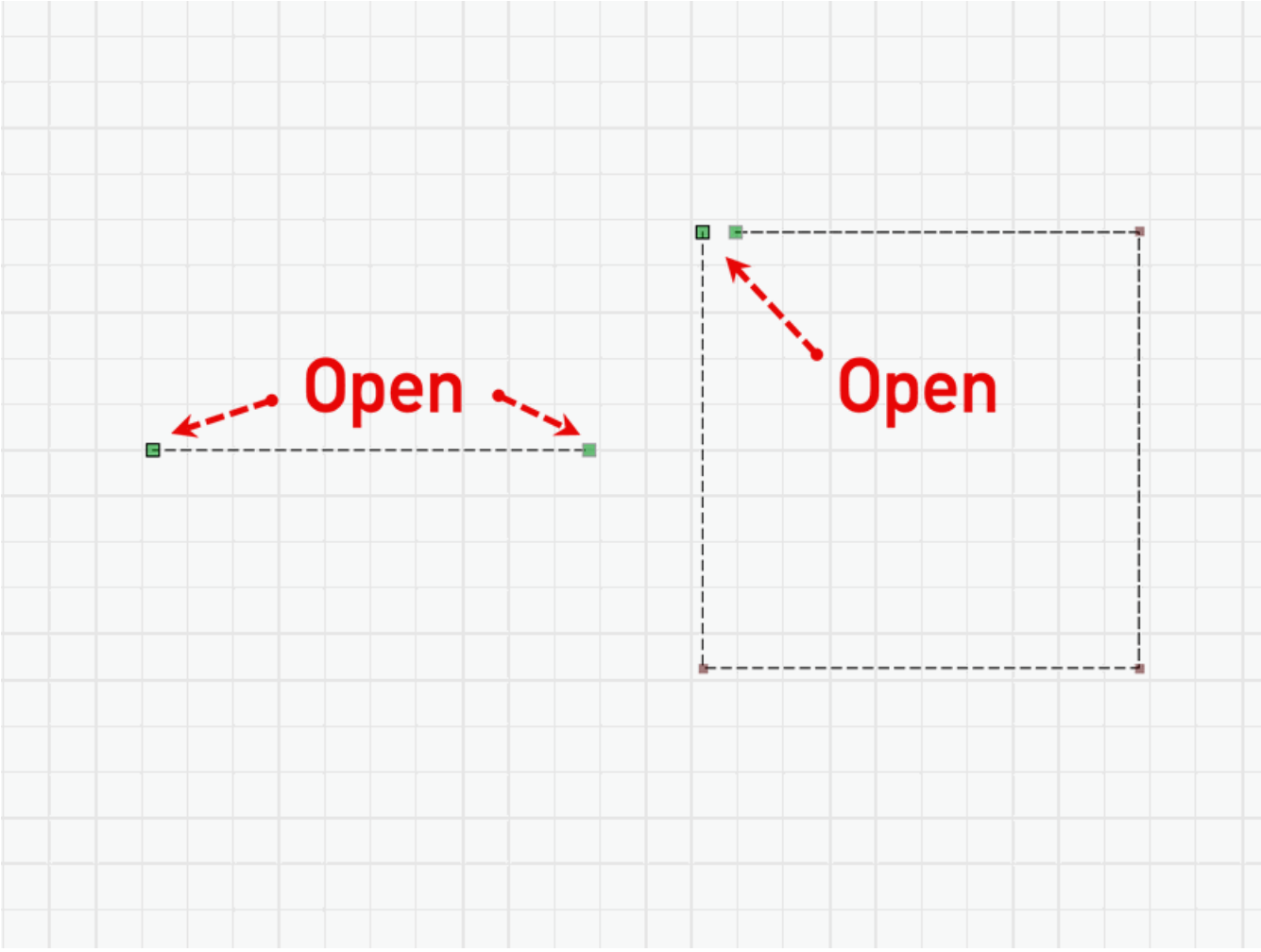
When and Why Shapes Must Be Closed

Some functions require LightBurn to determine the inside and outside of a shape, and in order to figure that out the shape must be closed.

The most commonly used function that requires closed shapes is Fill Mode, which commands your laser to engrave or mark an area that falls within the outline of a single specified shape, or between the outlines of two shapes. Without knowing what area is inside a shape, LightBurn can't determine what to Fill.

This is most obvious to the human eye when looking at line segments. If you draw a simple line segment with a node on each end, it makes no sense to say there is an inside and an outside to it. Following from that, if you assign that line segment to a layer set to Fill Mode, it will not be possible for LightBurn to determine an area to be Filled.

In other cases, as with shapes that have only small gaps, it may be relatively clear to a human what is meant to be the inside and outside, but software must interpret things literally, based on formal rules. That means if there is any gap at all, any open shape is the same to LightBurn as a line segment, for the purpose of determining an inside and outside.



Some other functions that require the use of closed shapes are:

- Offset Fill Mode
- Weld
- Boolean Tools
- Cut Shapes

- Resize Slots
- Kerf Offset
- Cut Inner Shapes First

Identifying Open Shapes

In some cases, as with simple line segments or shapes with large gaps, it's easy to see that a shape is open, but in others it may not be as obvious.

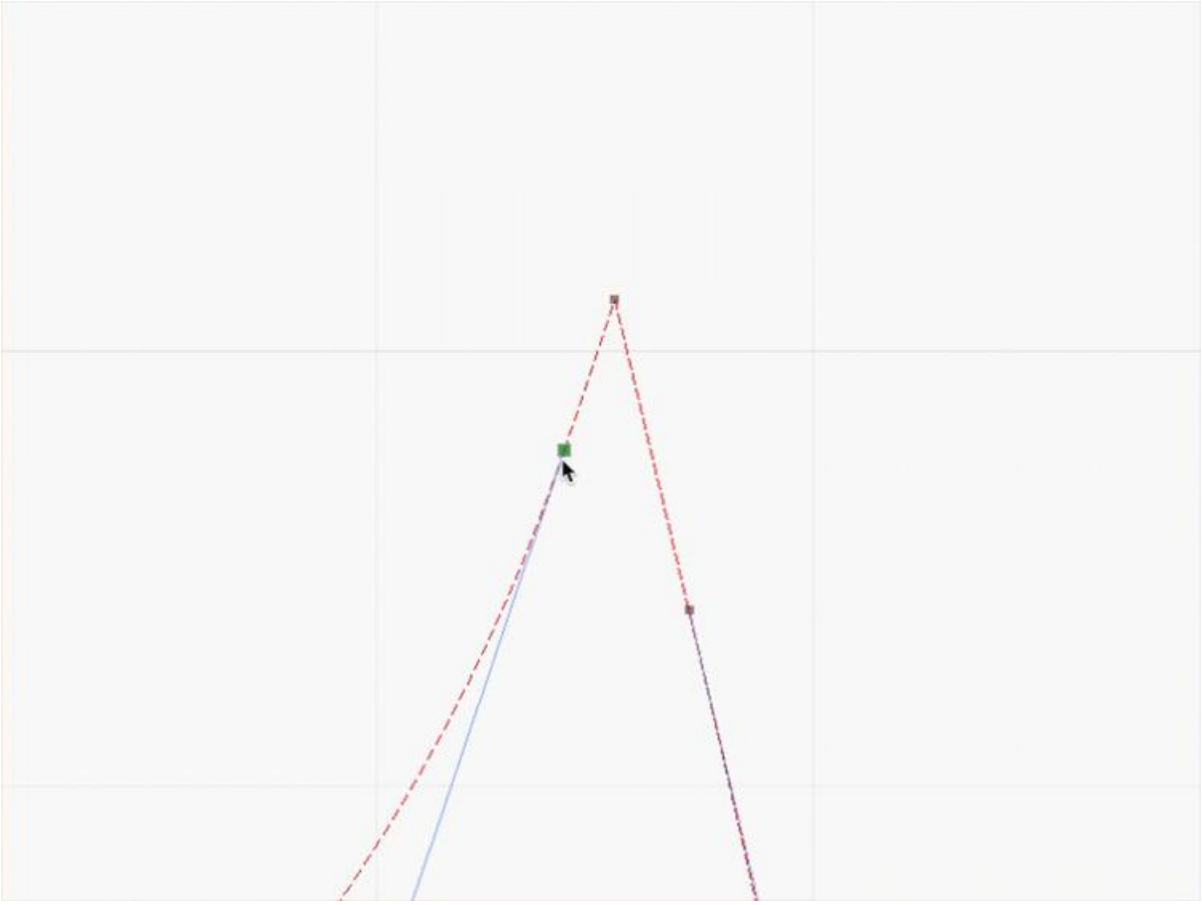
At times, the gaps can be very small, often so small that they are not visible without zooming in very close. Be aware: it is even possible for a start and end point to be directly on top of one another, but not connected, making a shape open.

LightBurn provides several methods for identifying whether a shape is open or closed.

Edit Nodes

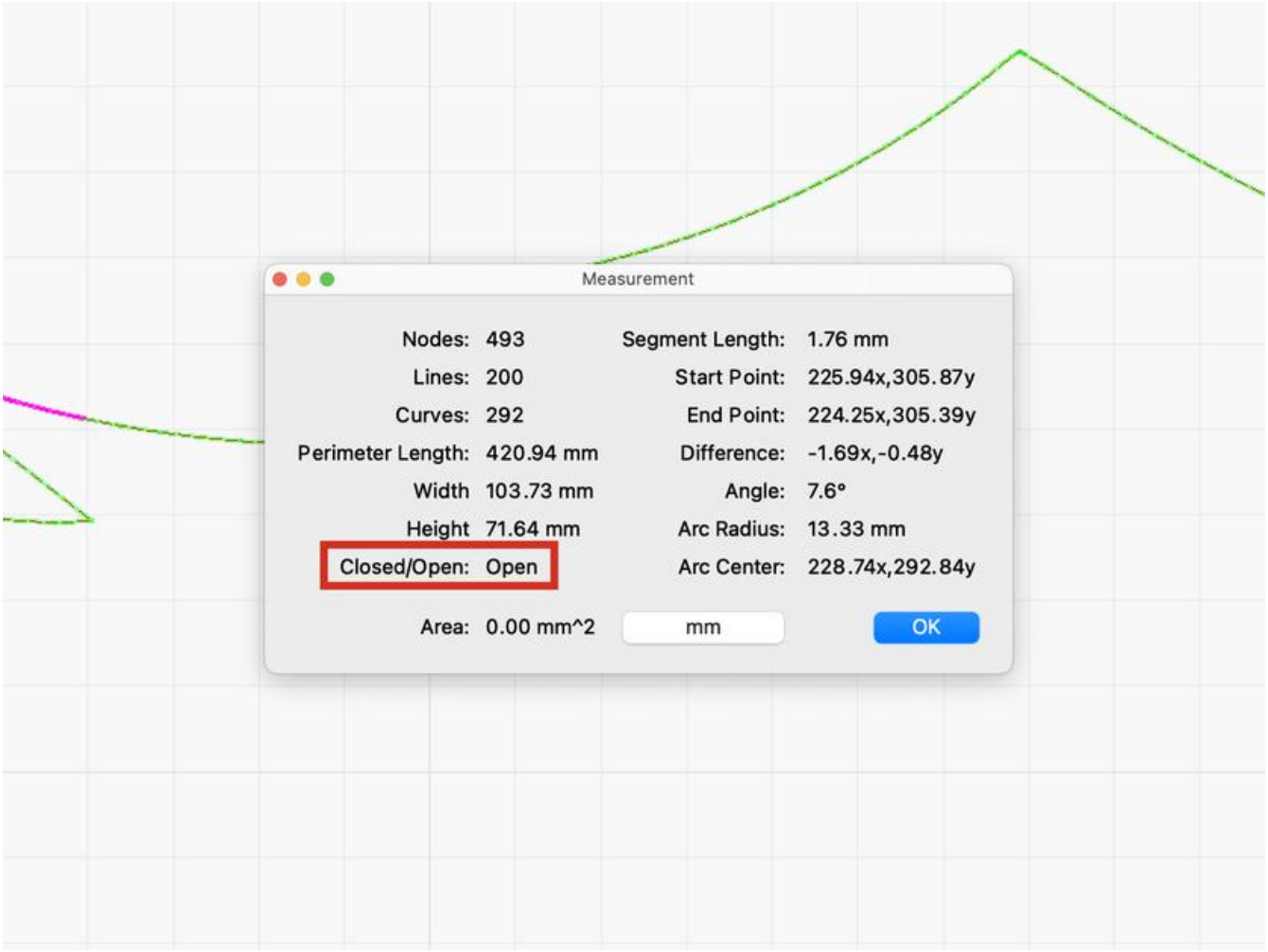
As mentioned above, it is always possible to select a shape and the Edit Nodes tool to identify where its start and end nodes are, and whether they are the same, making a closed shape, or different, making an open shape.

In some cases, you may need to zoom in to see gaps.



Measure Tool

Select the Measure tool from the Creation Toolbar (or press `Alt`/`Option`+`M`) and hover over a segment of any shape to see a readout of its attributes. In the bottom right, you'll see the **Closed/Open** descriptor, and a value telling you whether that shape is closed or open.

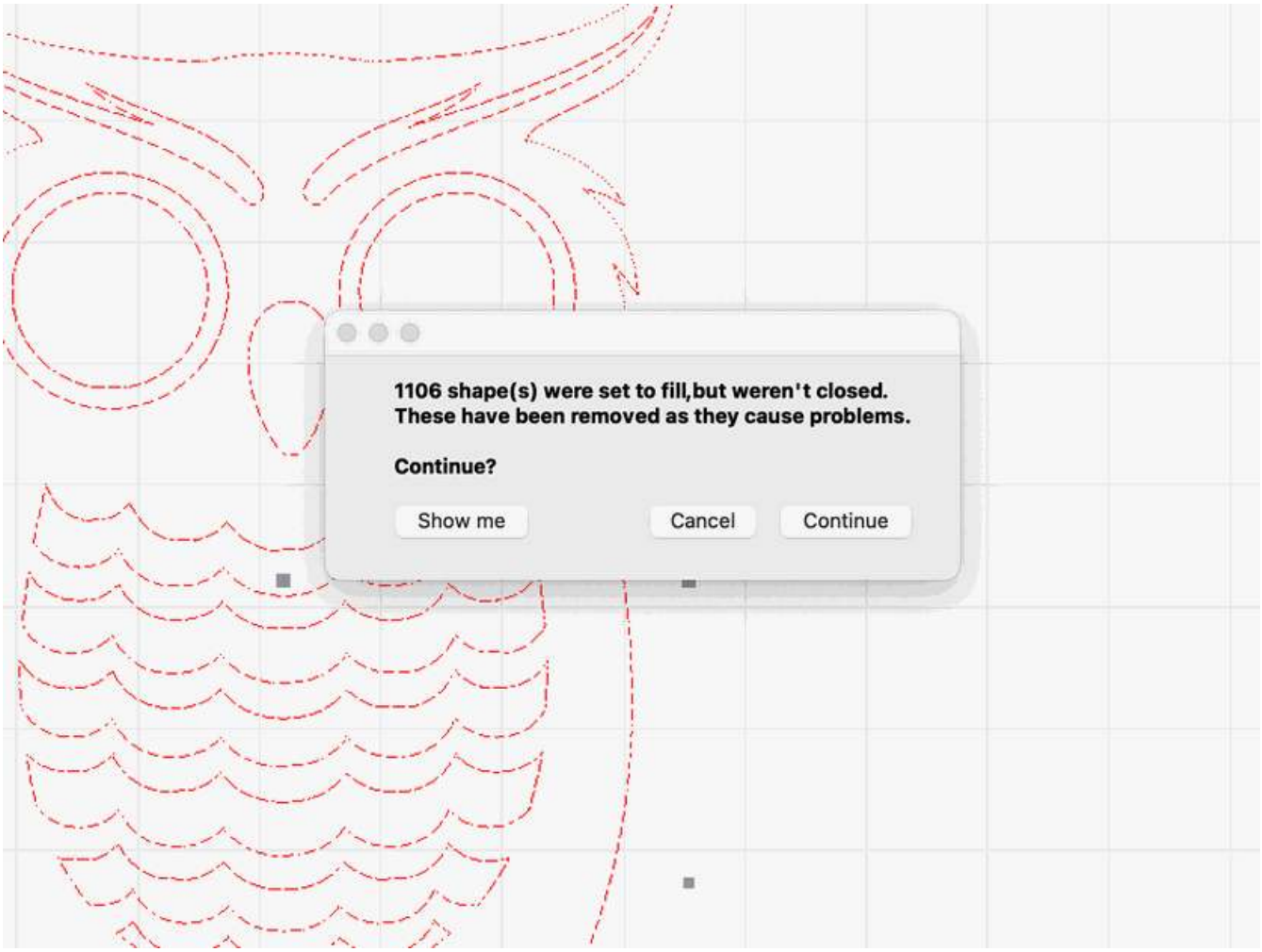


Warning Message

If you have open shapes set to a **Fill** or **Offset Fill** layer, and attempt to Preview or Start a job, LightBurn will warn you that "### shape(s) were set to fill, but weren't closed. These have been removed as they cause problems. Continue?"

The warning will give you three options:

- **Show Me** will close the warning and automatically select the open shapes set to fill.
- **Cancel** will close the warning and take no further action.
- **Continue** will continue with the job, skipping the open shapes.

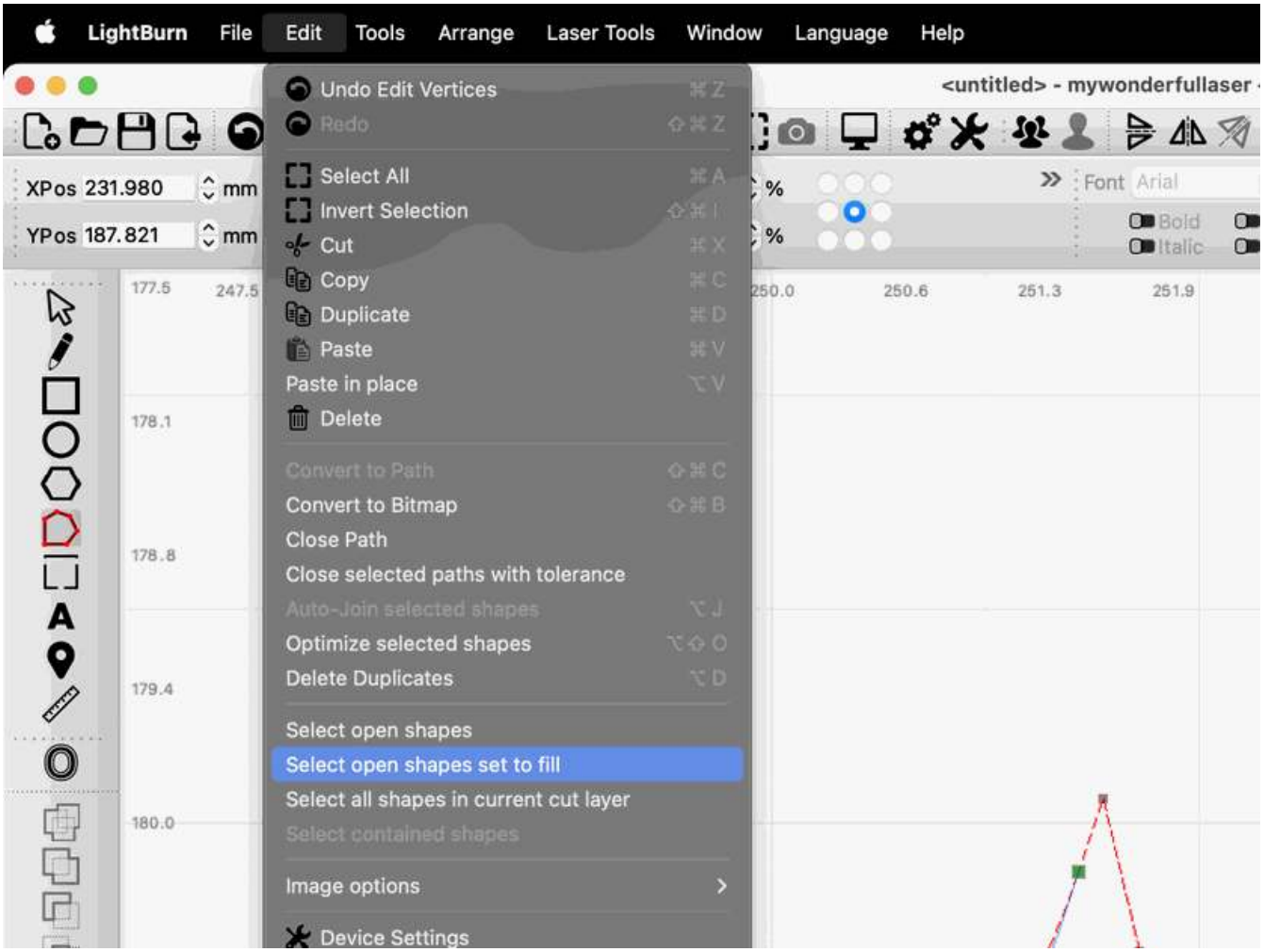


Select Open Shapes

From the Edit Menu, you can click the **Select open shapes** option to automatically select any open shapes in your Workspace.

Select Open Shapes Set to Fill

Also found in the **Edit Menu**, this option works the same as the **Show Me** button from the pop up warning, automatically selecting only those shapes in your Workspace that are set to a **Fill** or **Offset Fill** layer.



Correcting Open Shapes

Change Layer or Mode

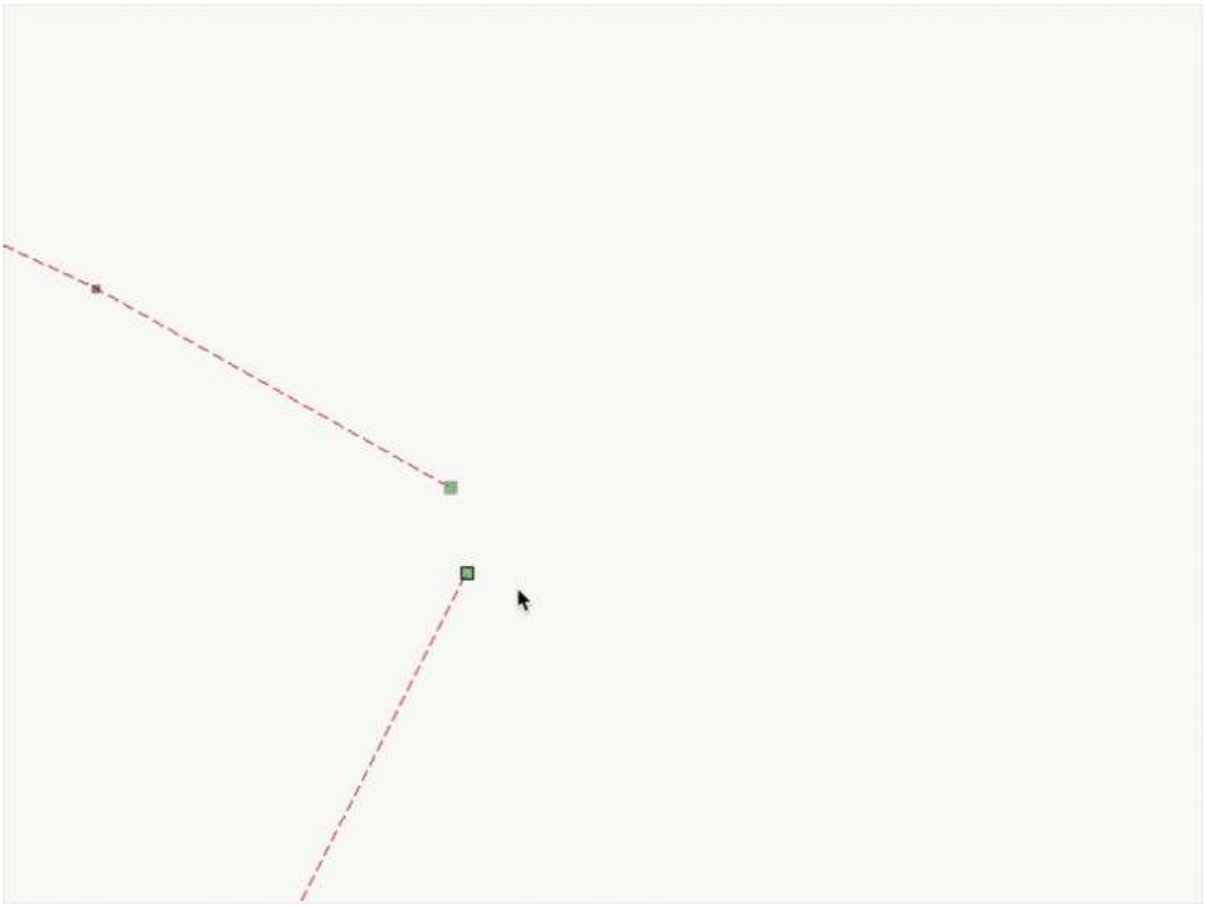
Before attempting to close an open shape, it's important to decide whether that is the best step to take to get the result you're looking for.

In some cases, especially when working with a project you did not design yourself that has shapes with start and end points that are very far apart, the best course of action is to assign those shapes to a layer set to Line Mode. Open shapes like these are often meant to be set to a speed and power appropriate to score a material along outlines defined by those shapes, and not to be filled at all.

To see how your result will look, you can set the open shapes to a Line layer, and use the Preview window to check whether the simulation matches your desired output.

Edit Nodes

The Edit Nodes tool is not just an option for identifying open shapes, you can also use it close them. To join open shapes with the Edit Nodes tool, select and drag a start point onto an end point (or vice versa).



Because this option requires you to close open shapes manually, in cases where you have many shapes to close it is often easier to use a tool that does so automatically.

Close Path

Found in the Edit Menu, the Close Path option will connect a shape with one start and one end node by creating a new line between them. The start and end nodes must be within .5 mm of one another, so this option is only for shapes with very small gaps. It cannot be used to closed shapes made up of more than one disconnected line segment.

Close Selected Paths With Tolerance

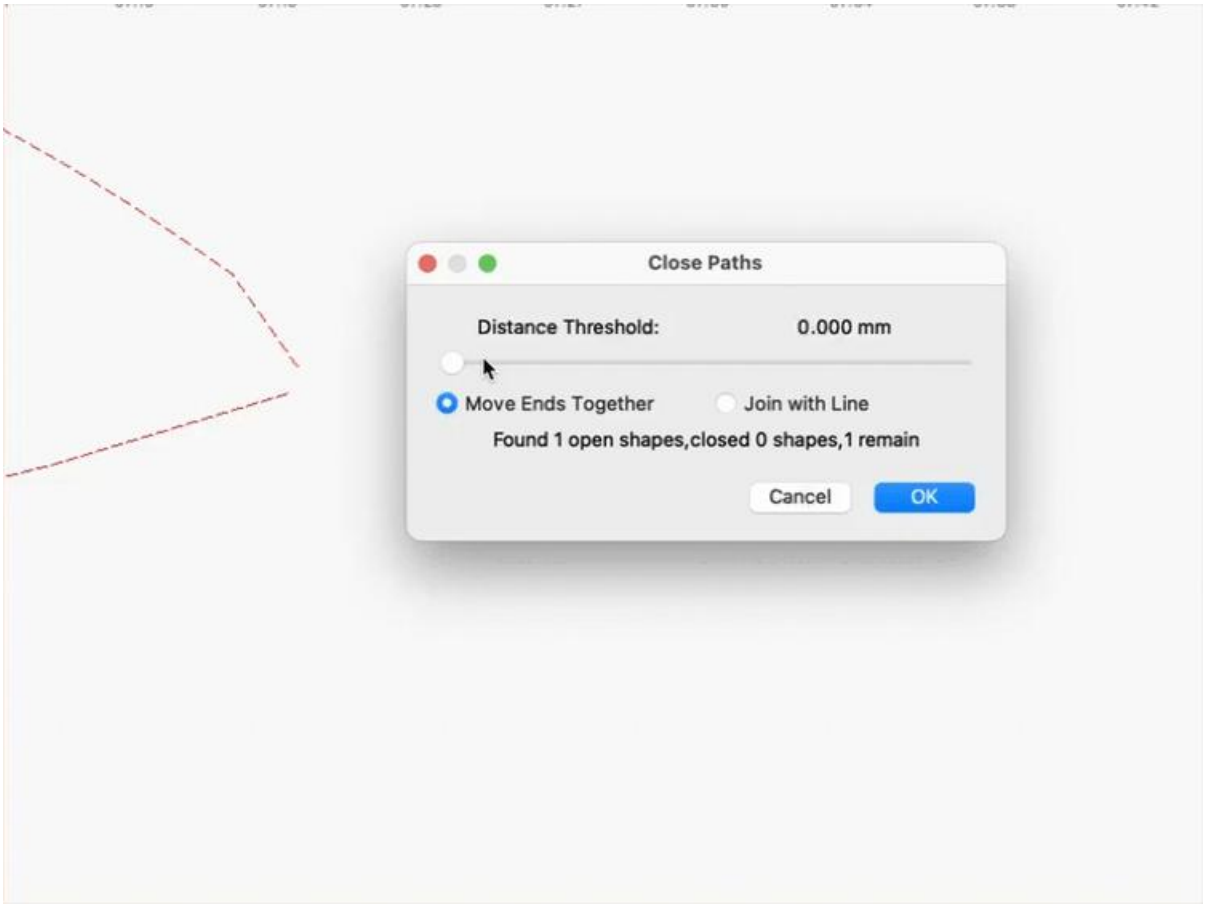
Close selected paths with tolerance is also found in the **Edit Menu**, and, like **Close Path**, can only connect a shape with one start and one end node.

Unlike Close Path, this option opens a pop up menu that allows you to specify the **Distance Threshold** (tolerance), between start and end nodes you'd like to close.

You can also choose whether you'd like to join the start and end nodes by bringing the existing lines together with **Move Ends Together**, or by creating a new line segment between them using **Join with Line**.

At the very bottom of the window, you'll see a readout of how many open shapes were found, how many shapes have been closed, and how many open shapes remain in your Workspace.

Click **OK** to confirm the changes and close the selected shapes, or **Cancel** to discard them and leave the shapes open.



Auto-Join Selected Shapes

Auto-Join selected shapes (`Alt`/`Option`+`J`) is also found in the **Edit Menu**, and is able to join many disconnected segments at once. In order for Auto-Join to work, the start and end points of two segments must be very close together, within .05 mm of one another.

In some cases where Auto-Join fails to join segments with start and end points within the necessary .05 mm tolerance, it can help to first go to the Arrange Menu and select Break Apart (`Alt`/`Option`+`B`), before again trying Auto-Join.

Related Topics

- Edit Nodes
- Edit Menu
- Measure Tool
- Close Path
- Close selected paths with tolerance
- Auto-Join selected shapes

How to Invert a Vector Engraving

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=EAlw371Ey0M

A common question we're asked is how to change which part of a design is engraving, swapping negative and positive space. With a little understanding of how vector shapes interact, it becomes easy to invert a vector engraving and swap which areas are engraved and which are not.

What to Do

To invert the filled and unfilled areas of a vector engraving, , you need to add an outline around it, and make sure the outline is set to the same layer as the original vector graphic.

How It Works

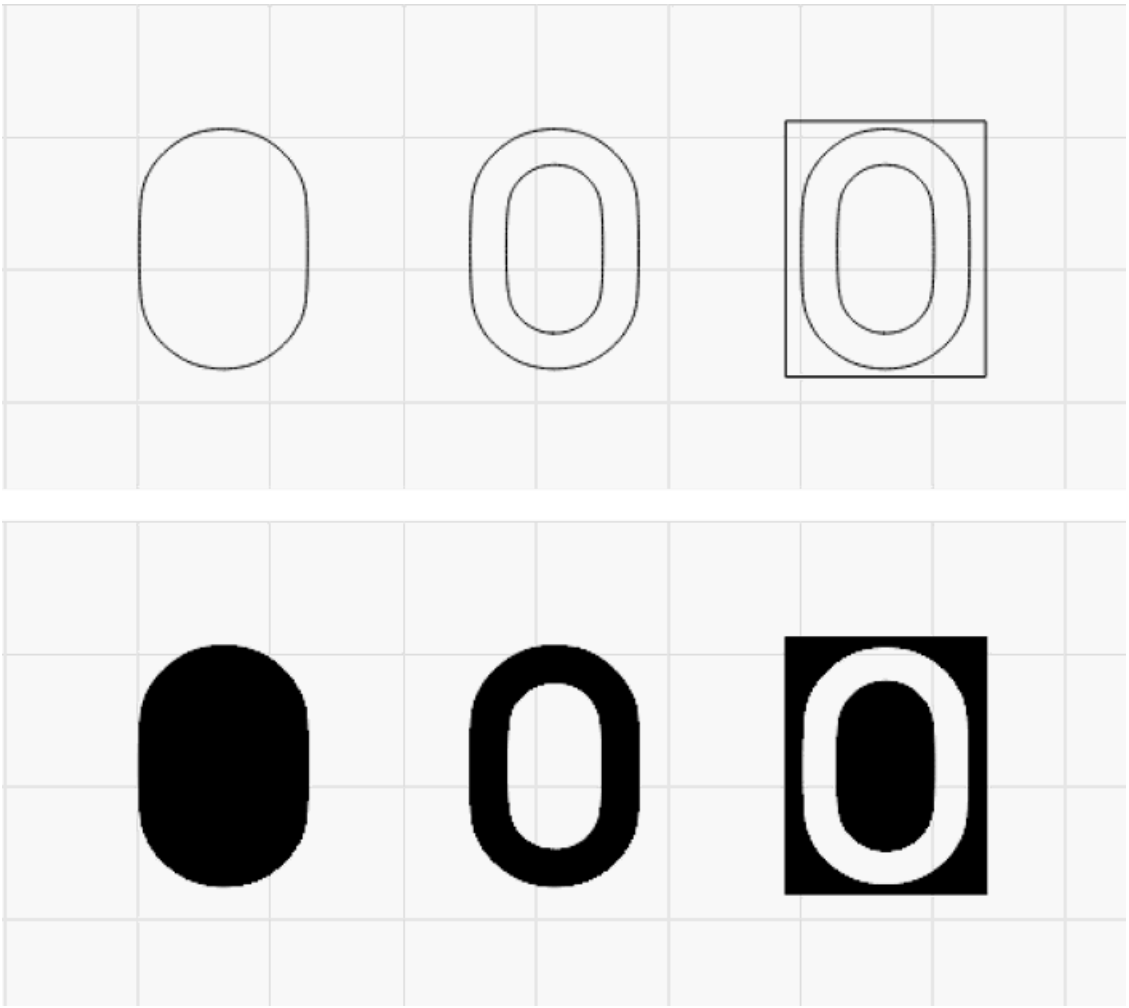
This behavior is due to the way LightBurn calculates what to engrave when you assign graphics to a layer set to Fill Mode or Offset Fill Mode.

When you assign a closed shape to a layer set to Fill or Offset Fill Mode, LightBurn tells your laser to engrave everything inside that shape.

That calculation changes when you put one closed shape around another. Now, instead of telling your laser to engrave everything inside each shape, LightBurn tells it to engrave only the space between the outlines of both.

The below example shows three different collections of graphics — first in Wireframe View Style, and then, below, in **Filled View Style**. Filled View Style shows the areas of your graphics that will be engraved as a solid color, while Wireframe only shows the graphics' outlines.

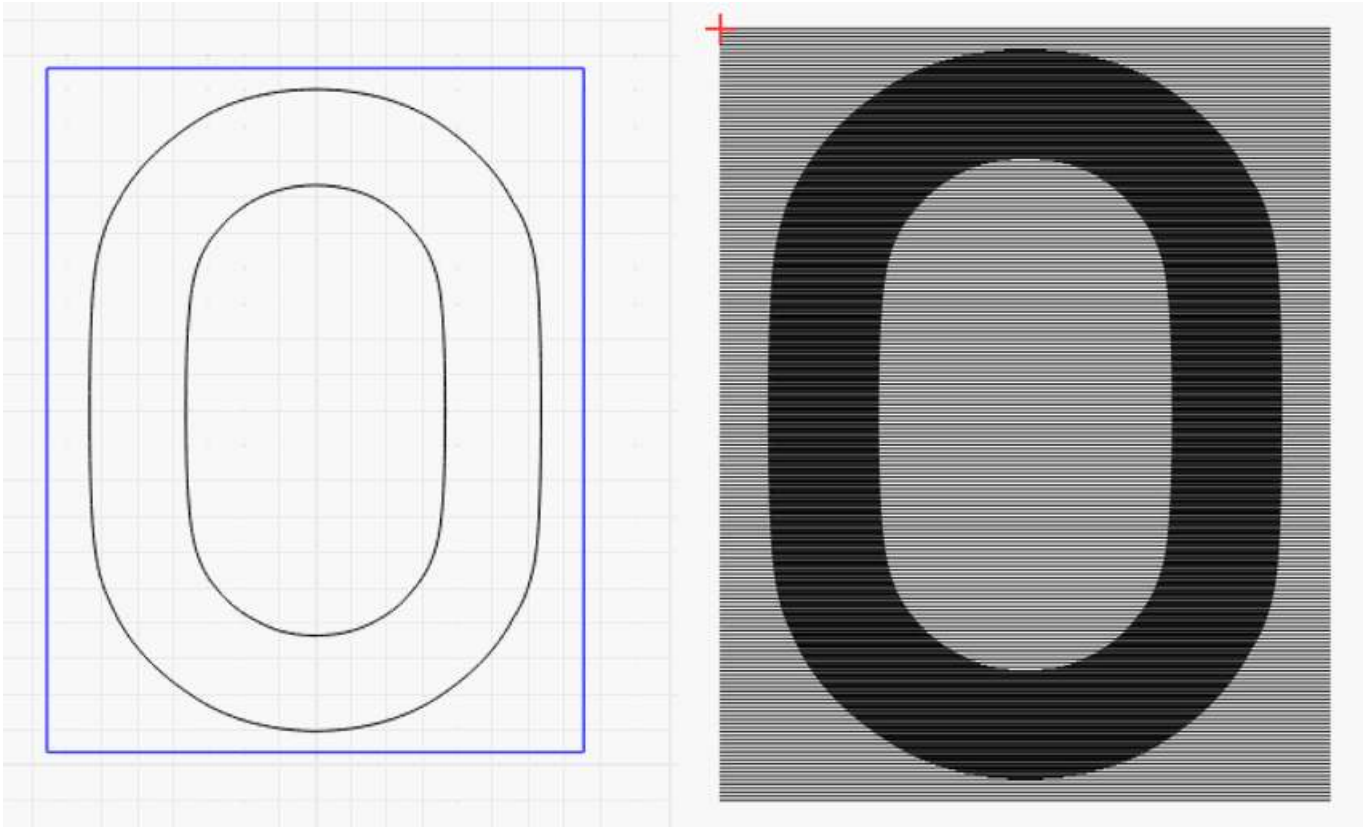
> **Switching View Styles**
>
> Press `Alt`/`Option`+`Shift`+`W` to quickly toggle between Wireframe and Filled View Style.



- The first is the outline of the letter O. Because this is just a single outline, LightBurn will tell your laser to engrave the entire area inside the outline.
- The second is the letter O with a center ring. Now, because there are two closed shapes set to a shared Fill layer, LightBurn will tell your laser to engrave everything between both.
- In the third, we've put a box around the complete letter O. Because of this new outline, the space that LightBurn considers to be between outlines is flipped, and nothing between the rings of the O will be engraved. Instead, everything between the outer ring and the rectangle, and everything inside the inner ring, will be engraved.

Each additional outline you add around a vector graphic will effectively toggle the area to be engraved, from within certain shapes, to between certain shapes, and so on.

An important note: shapes only negate each other if the inner and outer shapes are set to the same layer — if they're set to different layers, LightBurn will tell your laser to engrave the entire area inside the inner shapes, and the entire area inside the outer shapes, meaning some areas will be engraved twice.



Adding Outlines

There are several options for adding outlines to graphics.

- The Offset tool can create outlines at a specified distance around your shapes.
- You can use any of the Draw Shapes tools to manually create a new shape around your original graphics.
- Create Rubber-Band Outline creates an outline around your shapes, defined as the shortest possible path that can contain them all, as if a rubber-band were stretched around them.
- You can Duplicate an existing shape, and then increase its size using the Size handle, or in Numeric Edits Toolbar.

Template Files

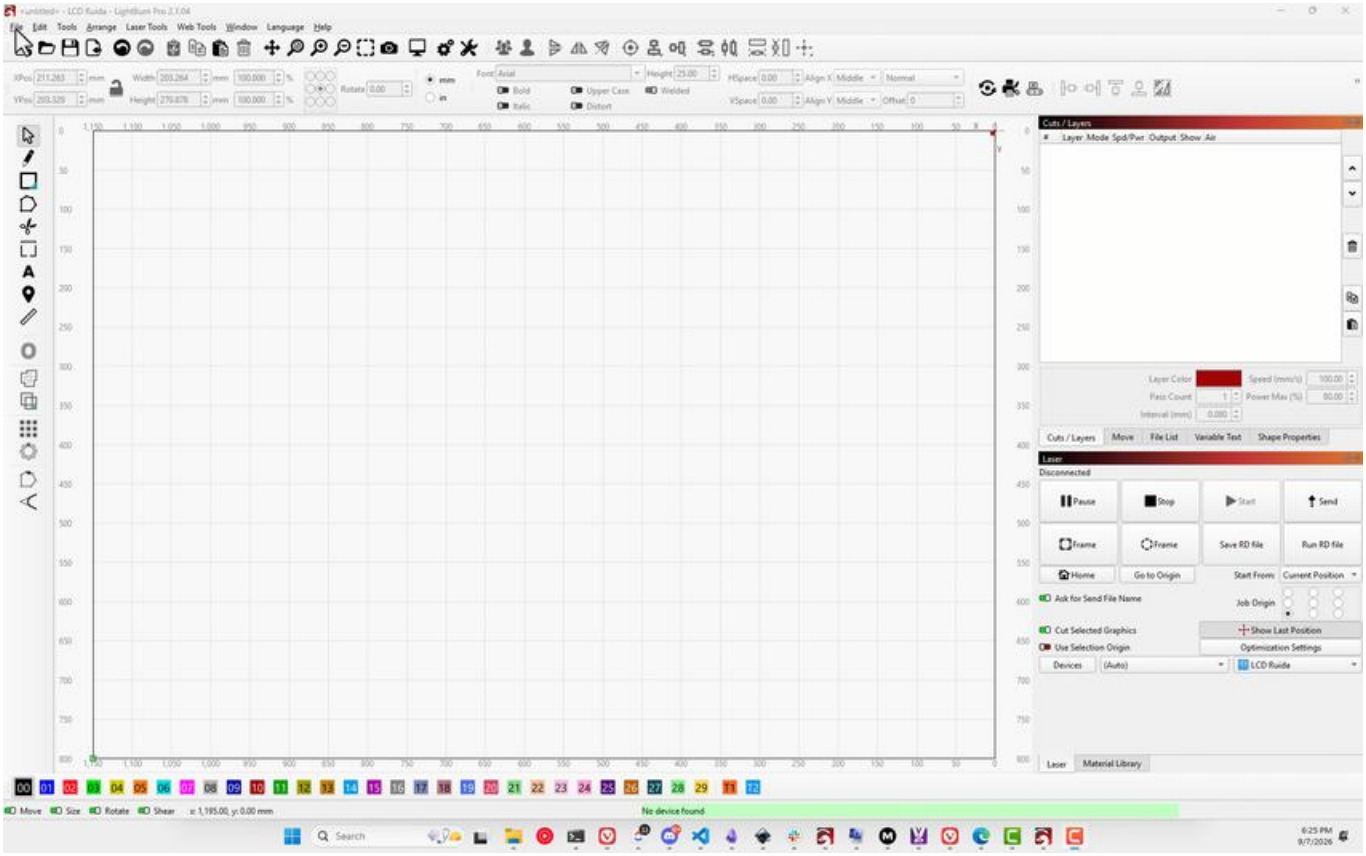
LightBurn 2.1 introduces Template Files, which we demonstrated in our Laser Engraved Pencil Project.

If you do a lot of customization or modification of a base file, you probably have a workflow something like this:

- Open a file with the base design you're going to modify.
- Make your changes to the base design.
- Save as a new file, so you don't overwrite your existing file.
- Do your cutting and engraving.
- Close the file.

It's a good workflow, but at some point you've almost certainly accidentally saved over your base design, possibly with changes you don't want applied to all your future design. Template Files offer a simple way from preventing that from happening — whenever you open a Template File, it creates a new, unsaved LightBurn File with the contents of your template file - including all the layer settings in the original template, even if the original Template File doesn't have any objects on that layer.

The GIF below shows an example of how you might use a Template File.



- Open the template file. Our example has a puzzle made with the puzzle generator linked on our Resource List, with the layers already configured how I want them. If you look at the top of the

window, you'll see that this creates a new, file, so it's impossible to accidentally save over the template.

- Make your changes to the base design. Layer 00 already has the image settings I want to use for this project, so it saves me the hassle of reconfiguring the Image Settings.
- Save the file. You can't accidentally overwrite the template!

Templates for Education

Whether you're in a classroom or a less formal educational setting, getting a group of new users laser cutting for the first time is easier if you can simplify things a bit and make their files more consistent. Having each new user open a Template File you've already created means you can provide pre-existing portions of the projects, pre-configured layer settings, or both.

Related Topics

- File Management
- Projects

Ovescanning

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=oNW8lTxuul4

When your laser engraves graphics set to Fill or Image Mode, the line-by-line, back-and-forth movement across the graphics is sometimes called scanning.

Overscanning refers to additional moves added to the beginning and end of each line to give the laser space to speed up before firing, and slow down afterward, which in turn prevents the edges of engraved areas from burning darker than the centers.

> **Note**
>
> Only gantry-style lasers with GCode-based or DSP controllers use Overscanning.
>
> Galvo lasers compensate for overburning at edges using Delay settings.



The extra Overscanning movements are indicated in red. Enable Show traversal moves in the Preview window to see these movements.

How Overscanning Works

Engravings made without **Overscanning** are prone to overburning at the edges due to the relationship of Speed and Power output in laser engraving.

The longer your laser spends passing over any part of an engraving, the more effective power it will output over that area. That means that the same Power setting will burn more at a slower Speed than a faster one.

When your laser approaches the end of a scan line, it needs to decelerate so it can change direction, and, after turning around, accelerate to reach the Speed you told it to engrave at. It can only slow down and speed up so quickly — just how quickly depends on the laser, but all lasers have physical limits.

That means if you tell your laser to engrave an area, and it's not allowed to travel beyond the bounds of that area, it will be engraving as it slows down and speeds up, and so the edges will absorb more power than the center, and burn darker.

By giving your laser extra room to accelerate and decelerate with the laser off, you can ensure that it only fires once it's reached the full Speed you set it to engrave at, resulting in an even burn across the entire engraved area.

Applying Overscanning

DSP controllers apply **Overscanning** automatically, so DSP users never have to do so manually, and do not have the option to disable it.

Users with GCode-based lasers have the option to enable or disable Overscanning in the Cut Settings Editor, and to specify the distance.

Overscanning distance is calculated as a percentage of Speed. You'll see the amount of required distance for your combination of Speed and Overscanning percentage to the right of the percentage field.

If you increase Speed, Overscanning percentage, or both, the amount of required distance will go up.



The Overscanning setting for a GCode-based laser

Speed, Overscanning Distance, and Job Time

Although you can't control the percentage through LightBurn, Overscanning distance varies based on Speed for DSP lasers as well, and in the same way — as you increase engraving Speed, Overscanning distance increases as well.

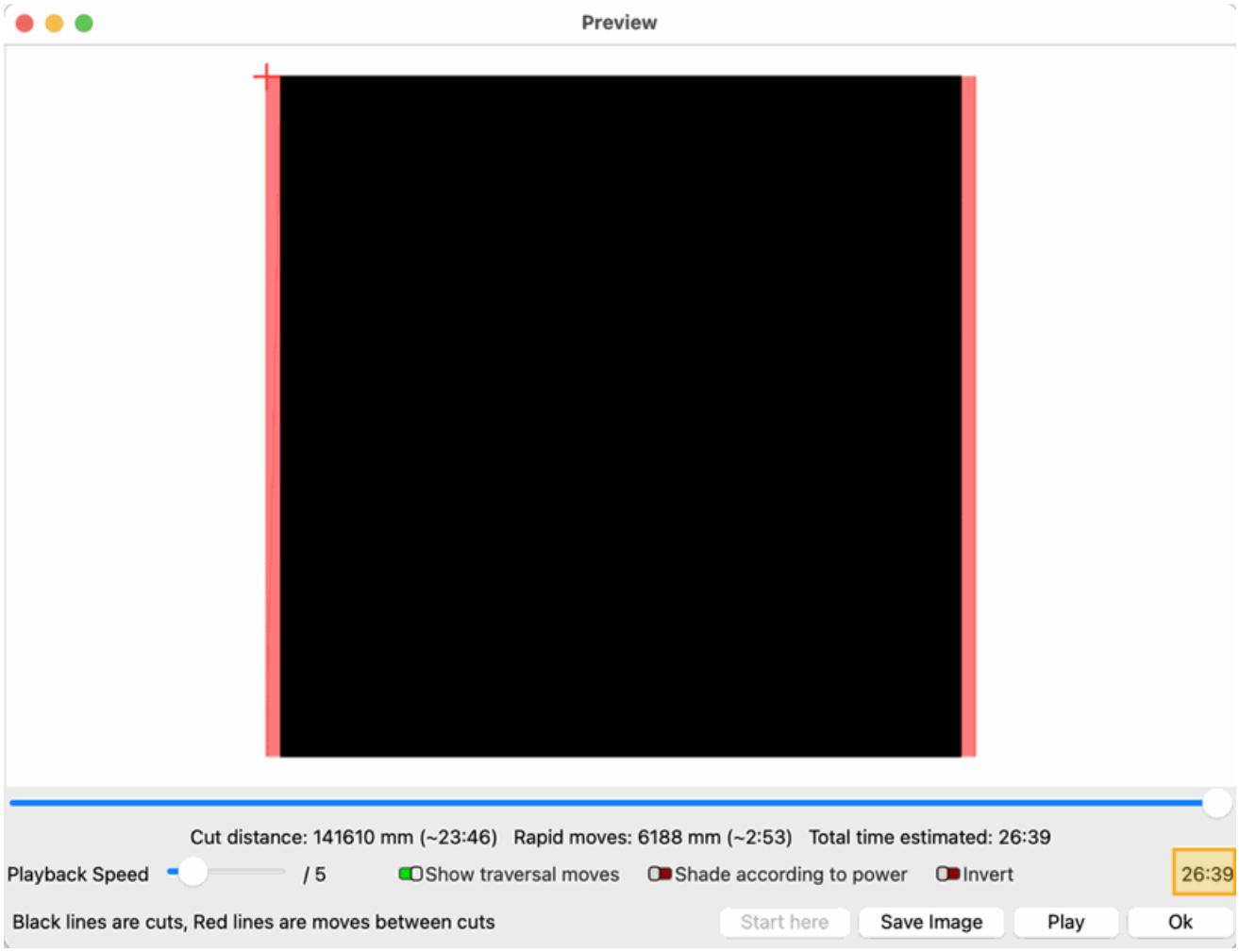
This is, again, due to the physical limits of your laser — while every laser is different, there is a limit to how quickly any laser can slow down or speed up.

No matter what that limit is, it will always require more space and time to slow down from or speed up to a relatively high speed than a relatively low one.

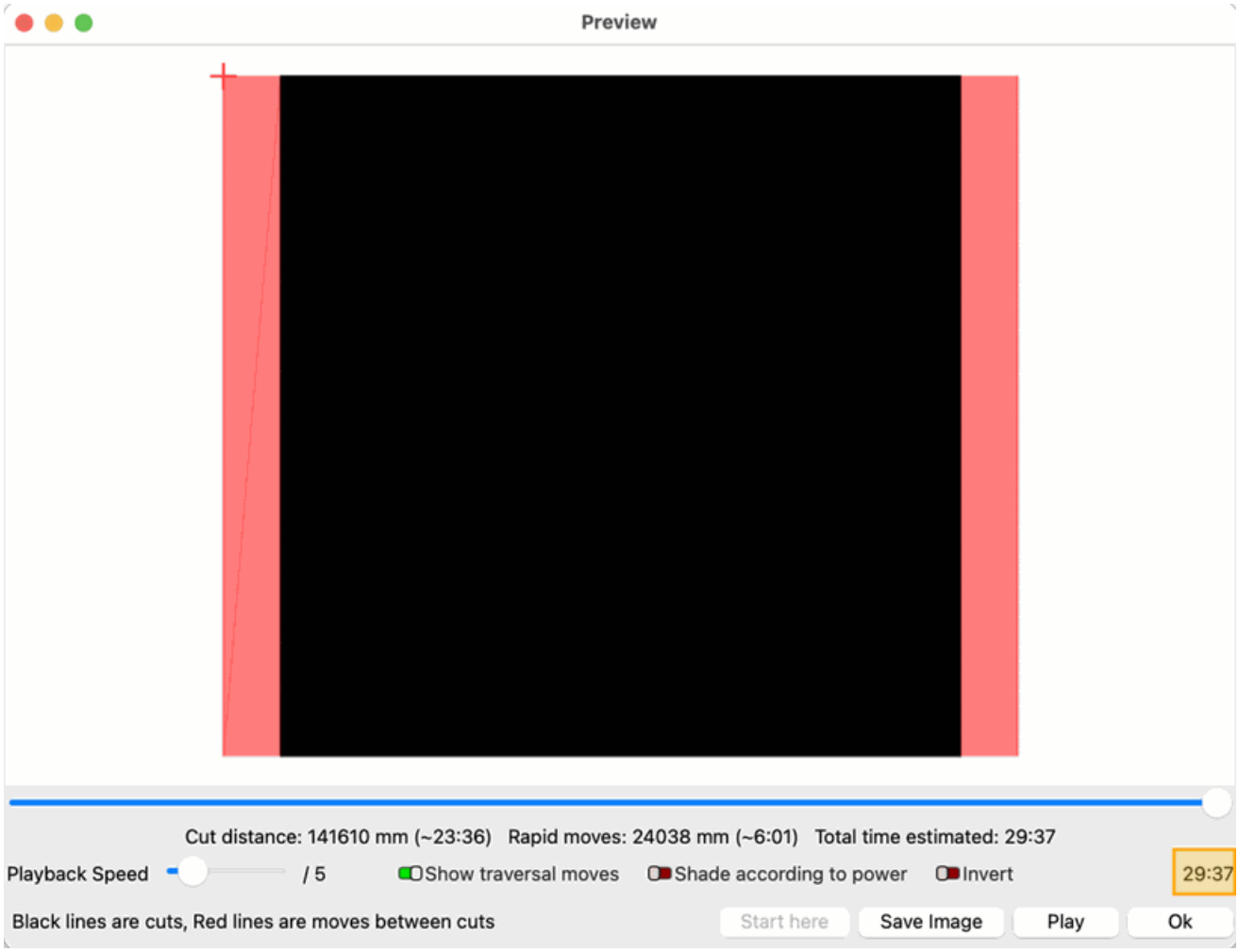
Imagine driving a car and quickly pressing the brakes to bring it to a halt. The car will require a longer distance to stop if you are traveling at 60 miles per hour than if you are only traveling at 10 miles per hour. It will also take more time to halt the faster-traveling car than the slower-traveling one.

Because high Speed settings require more distance and time for acceleration and deceleration, increasing the Speed you set your laser to engrave at will yield diminishing returns in time savings the higher you go, and, in some cases, increasing past a certain point can actually lead to an increase in job time, rather than a reduction.

You can use the Preview window to check your project's estimated run time, and see the effect of a range of Speeds on overall time.



Set to 100 mm/second (6000 mm/minute), this graphic will take 26:39 to engrave.



The same graphic, now set to engrave at 400 mm/second (24,000 mm/minute), will take nearly 3 minutes longer to engrave, due to the high amount of Overscanning required.

Troubleshooting

Error Messages

GCode-based lasers

If you enable Overscanning for a GCode-based laser and there isn't enough room at the edge of the job to accommodate the extra travel distance, you may see LightBurn's **Cut may be out of bounds due to overscan settings** error, or your laser's controller may return an ALARM:2 error.

DSP lasers

If a job on a DSP laser requires Overscanning and there isn't enough room at the edge of the job to accommodate it, your DSP controller may return the **Not Enough Extend Space** error.

Because **Overscanning** requires your laser to travel extra distance outside the exact boundaries of your graphics, if your graphics aren't positioned properly, or you are too close to the edge of your machine when using Current Position or User Origin as your **Start From** mode, you may accidentally command your laser to travel outside of its physical limits, once the Overscanning distance is included.

To correct excessive Overscanning:

- Reduce Speed or Overscanning percentage (if you have a GCode-based laser).
- Move your graphics in toward the center of your Workspace, if you are using Absolute Coordinates positioning.
- If you are using Current Position or User Origin to set a custom Origin, set it closer to the center of your laser's work area.

Overscanning Distance in the Preview Window

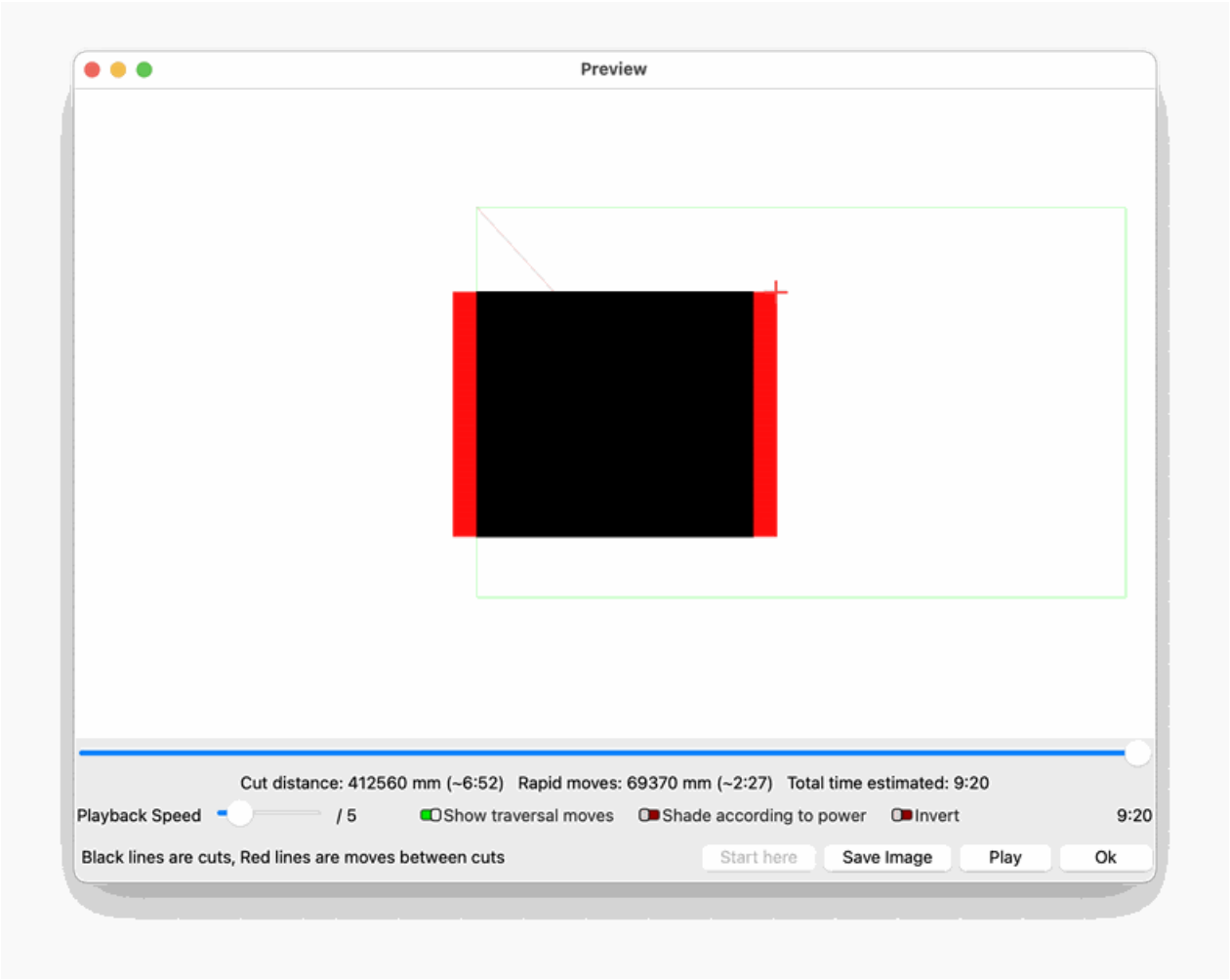
You can check for excessive **Overscanning** using the Preview window.

Make sure **Show traversal moves** is toggled on, and you'll see the additional Overscanning moves indicated as red lines (black lines indicate engraving or cutting moves).

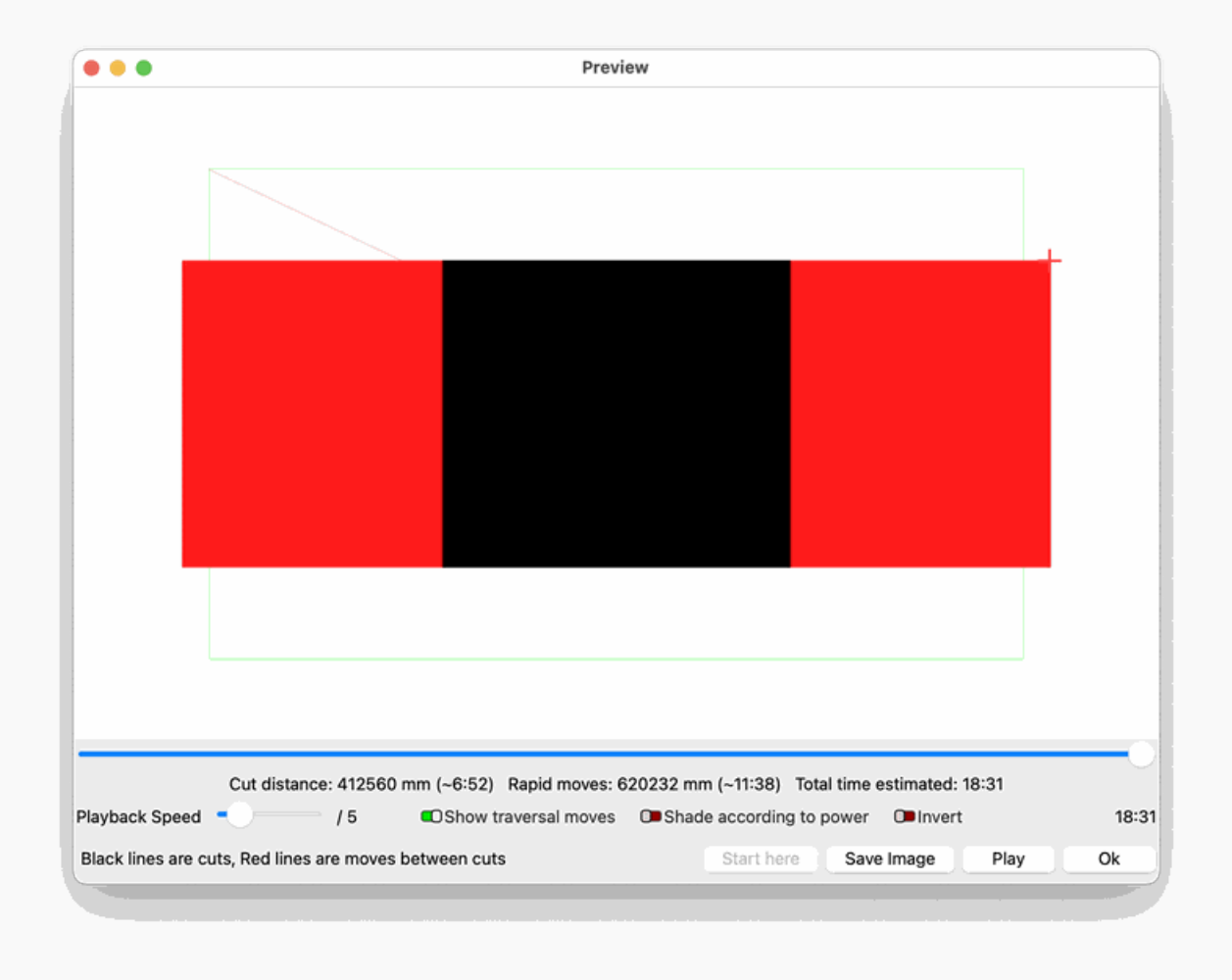
The maximum boundaries of your laser's work area are indicated by a green outline. To see the outline you may need to zoom out by pressing <kbd>-</kbd> on your keyboard, or by using your mouse's scroll wheel (or two-finger dragging if you're using a trackpad).

If the red lines cross the green border on one side, and you've selected Absolute Coordinates as your **Start From** mode, the Overscanning moves will command your laser to travel outside its work area. If you're using Current Position or User Origin, it may still be ok, depending on where you establish your custom origin.

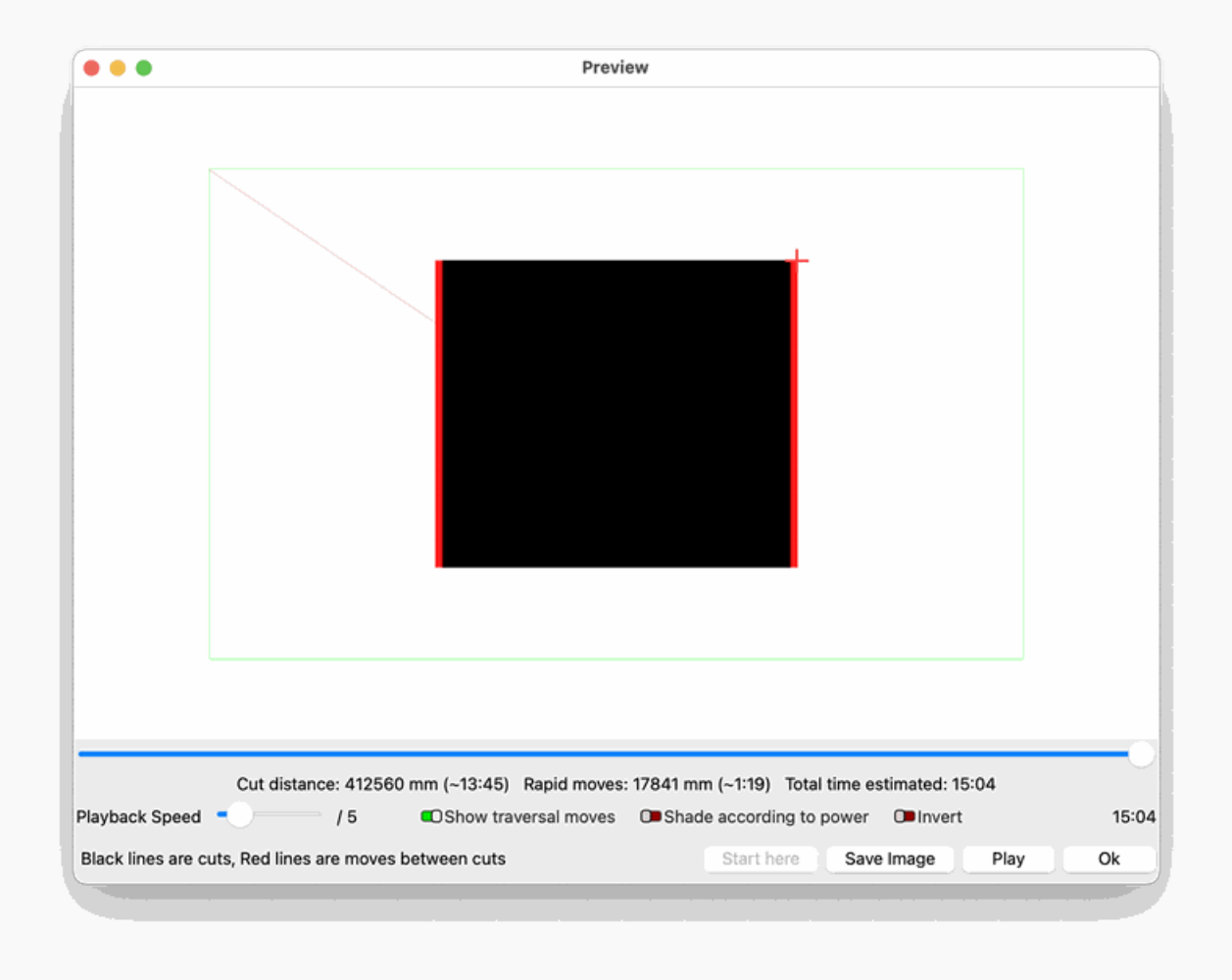
If the red lines cross the green border on two sides, no matter which Start From mode is selected, it means the Overscanning distance is excessive, and will command your laser to travel outside its physical bounds.



Overscanning crossing a single side of the green outline — this will be a problem if the Start From mode is set to Absolute Coordinates, or if the custom User Origin or Current Position is too close to the edge.



Overscanning crossing two sides of the green outline — this will be a problem no matter what Start From mode is selected.



Overscanning crossing neither side. This is a normal amount of Overscanning, and will work well with Absolute Coordinates positioning, but you still need to make sure that any custom User Origin or Current Position isn't too close to the edge of the machine's work area.

Related Topics

- Fill Mode
- Image Mode

Steps per MM

If the output from your laser looks like it's squished or stretched in one or both directions, it may be because your **steps/mm** settings are configured incorrectly.

All laser cutters / engravers must be configured with the correct steps/mm (steps per millimeter) setting to function correctly. If your movement is already close, LightBurn has a built-in Calibrate Axis tool that will help you dial in the correct setting. If you would like to better understand how the **steps/mm** setting works and how to find or calculate the true value, read on.

Background

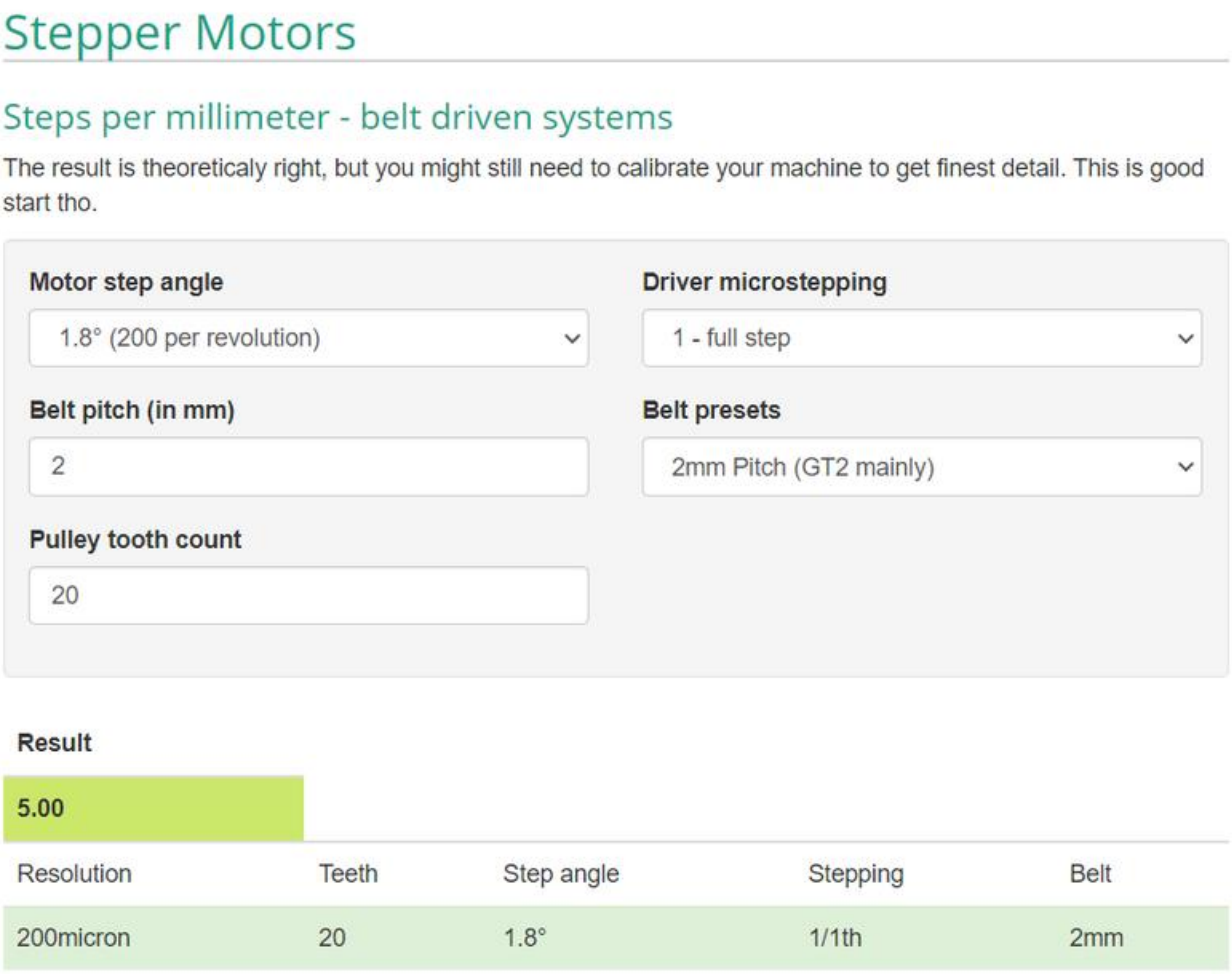
Your laser moves by using stepper motors which, unlike basic motors, don't spin continuously, but in discrete steps, each of which is a specific number of degrees in a full rotation. For example, the most common motor type has a 1.8° step angle, meaning that a full rotation is 200 steps (360° / 1.8° = 200).

In order to tell your machine how to move the correct distance, it has to know how many steps to instruct each motor to move. What that value is, however, depends on not just the motor but the rest of the machine's setup. For small, diode-based lasers, a common setup might look like:

- 1.8° Nema 17 stepper motor
- GT2 belts (2 mm distance between each tooth)
- 20 tooth GT2 pulley

Fortunately, calculating this is made easy by many great online tools, like this Stepper Motor Calculator.

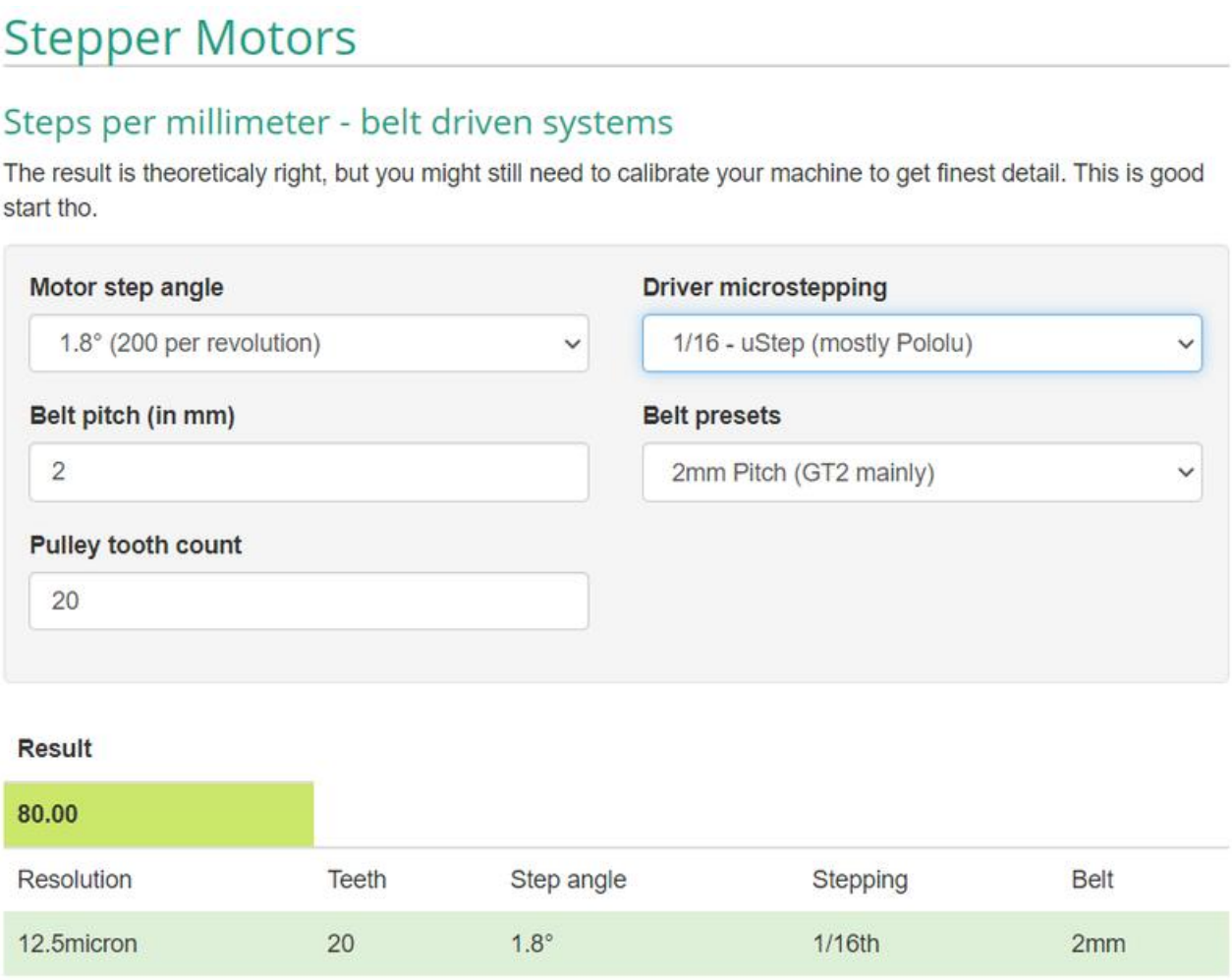
For the above configuration, setup the calculator as shown below:



You would needs 5 steps per mm of movement. This is because a 20 tooth pulley represents 40 mm of movement for one full rotation (20 teeth * 2 mm per tooth) and there are 200 steps per full rotation. So `200 / 40 = 5`.

Microstepping

Very few machines will ever run in "full step" mode as shown in the example above. As you can see at the bottom of the example above, the resolution is 200 micron or 0.2 mm. That means the absolute smallest distance that can be moved is 0.2 mm, which is not nearly good enough. Modern stepper motor drivers, however, can pull off a trick known as microstepping, in which they can move in steps between the natural steps of the motor. A common option is 1/16th microstepping which means that it divides each natural step into 16 sub-steps, multiplying the number of steps required per full rotation from 200 to 3200!



As you can see now, the steps/mm value jumped from 5 to 80 and now has a resolution of 12.5 micron or 0.0125 mm.

What settings does your machine use?

The above is merely explaining the theory, now we need to put it all together.

First off, the best place to find your correct steps/mm setting is generally going to be from the manufacturer of the machine (unless you built it yourself). We highly recommend starting by looking for the correct settings direct from the source.

If that information is unavailable, it can be arrived at with a little work.

Motors

It is highly unlikely the motors you are using are anything other than the 1.8° variety. Sometimes 0.9° are used and those are very often marked as such on the motor body somewhere. You will also likely find a model number marked somewhere on the motor. Find that and look it up online, for example: `17HS19-2004S1`

You will then typically be able to find specs or a datasheet on that specific motor model which will tell you the **step angle**.

Belts

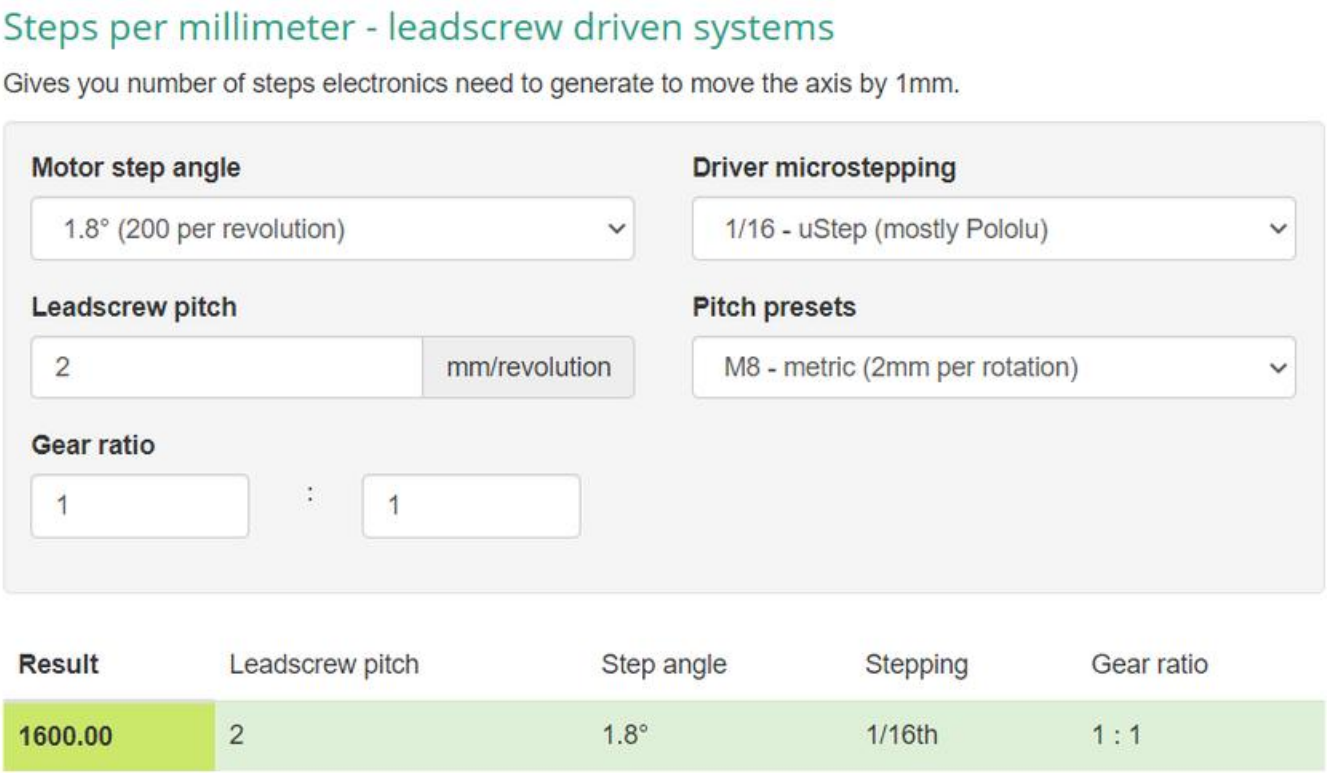
Small hobby diode lasers typically use GT2 (Sometimes called 2GT) belts, which is just the name of a common belt profile that has a 2 mm **pitch**. The pitch of a belt is the distance from the center of one tooth to the center of the next. You can use calipers or an accurate ruler to check if you are unsure. Just note that belts will always use nice round numbers. Commonly you will find 2 mm, 3 mm, or 5 mm pitch. Anything beyond that is unlikely to be used in a laser.

Once you've found this value enter it in the **Belt pitch** field in the stepper motor calculator, or use one of the **Belt presets** from the drop-down.

Screws

Some machines use lead screws instead of belts, such as the 3018 CNC machines which we see converted to laser use very often. Determining the pitch of a screw is very much the same, except that you need to follow a single thread of the screw (sometimes they have multiple threads, unlike a bolt might) for one full rotation (it may be helpful to mark it with a Sharpie) and then measure the distance from start to finish of that rotation.

Just like with the belts, there should be no odd numbers here, as there are very standardized screw pitches. 2 mm, 4 mm, and 5 mm are very common for this application. Just be sure to use the **leadscrew driven systems** version of the calculator, lower on the page:



Motor Driver Microstepping

Finally, you need to determine what microstepping value is configured for your drivers. This is either something that cannot be changed on your particular controller (in which case you need to find out from the manufacturer), or can be changed via DIP switches or jumpers.

Many GCode based devices use what are commonly referred to as "StepStick" style motor drivers - a little 0.6" x 0.8" circuit board with 8 pins on each side that plugs into your controller. On the controller will often be either jumpers or DIP switches to configure 3 different states that control the microstepping.

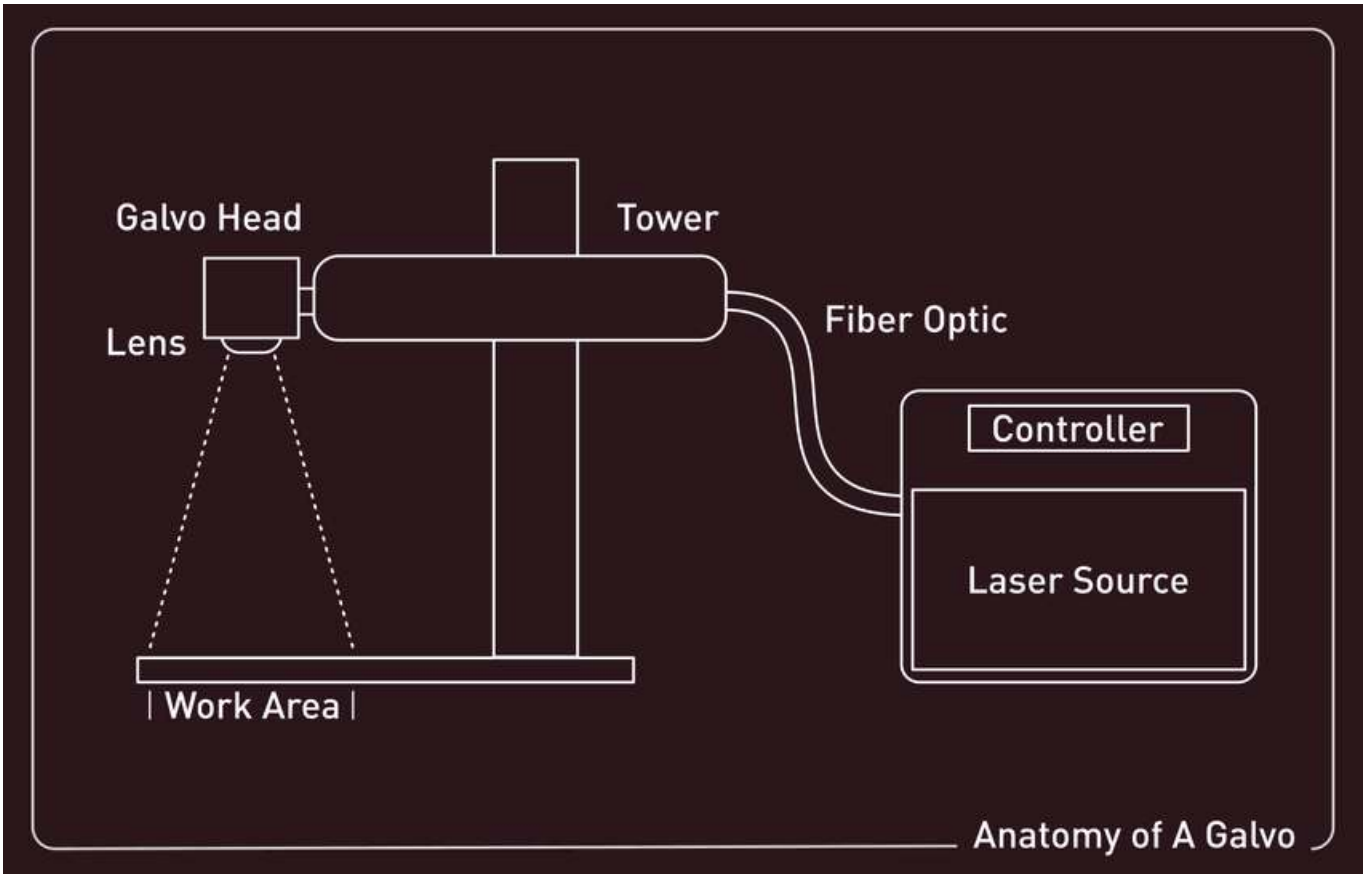
These will typically be labeled MS1, MS2, MS3 or CFG1, CFG2, CFG3. You will need to then determine what driver chip is being used by looking at the black IC on the StepStick board. You may need magnification to read the markings. Once you know the type, you can look up the configuration options online for that specific driver.

For example, the common (though a bit old at this point) DRV8825 from Texas Instruments has the following configuration table:

CFG1	CFG2	CFG3	MicroSteps
O	O	O	1/1
X	O	O	½
O	X	O	¼
X	X	O	⅛
O	O	X	1/16
X	X	X	1/32

`O` means that config pin is left open (no jumper) or off (in the case of a DIP switch). `X` means closed (with jumper) or on. So, for example, with no jumpers on CFG1 and CFG2 but a jumper on CFG3, the driver would be running in 1/16th microstepping mode. In other words, it divides each step into 16 sub-steps, therefore multiplying the total number of steps per revolution by 16.

About Galvos



Sources

Supported Types of Galvo Lasers

The current types and manufacturers of Galvo laser sources supported are as follows:

- Raycus (RAYCUS)
- JPT (JPT)
- IPG (IPG_YLP / IPG_YLPM)
- MAX (MAX / JPT)
- CO_2
- UV

These are devices that may work at some point in the future, have been communicated to our staff that users would like to use them, but we can't 100% confirm behavior just yet.

- QUANTEL_M20HF
- QUANTEL_M20
- MANLIGHT_PL
- VGEN_YPFL

- EO
- QUANTEL_M20EG
- QUANTEL_M30EG
- IPG_YLP_E
- HFB
- Orion Laser

Fiber

Fiber lasers are incredibly common devices, due to their use for long-distance and high bandwidth networking; fiber lasers power the internet.

They make use of an optical pathway, made of silica glass to guide the light, resulting in a narrow, straight beam that's able to carry an incredible amount of energy and data. For galvo actuated fiber lasers, this means that the beam can move extremely quickly, making small adjustments via actuated mirrors, rather than complex gantry systems.

Due to the beam being formed inside of a solid mechanical apparatus, fiber lasers are considered solid state devices, whereas CO_2 laser tubes are generally considered gas state.

Step By Step

1. Light is created in a laser diode, a dedicated source that's "pumped" into the fiber optic cable, resulting in them occasionally being referred to as a pump source.
2. Pump light is guided into the fiber optic cable. While the fiber optic is able to carry the beam, the cladding (or casing) on the fiber is the reason the beam stays inside of the fiber instead of dispersing out of it.
3. Light is amplified inside the laser cavity. As the light passes through this region, the particles rise to a higher energy level, getting caught and excited / relaxed into a singular wavelength of operation. A doped portion of fiber, mixed with certain rare earth elements, makes this possible, reflecting back and forth while doing so inside "Bragg gratings". This is where it becomes a stimulated emission of radiation.
4. Laser light of a particular wavelength is created. The specific element(s) used to dope the fiber determine what wavelength the beam becomes.
5. The beam is released and shaped. The light at this point is collimated (straightened), and to produce a usable shape for engraving, marking, or more, lenses and/or beam expanders are used to adjust the collimation.

Q-Switched vs MOPA

These two terms are often used in conjunction, but what does that even mean? What's being switched? Why is it a MOPA?

Master Oscillator Power Amplifier (MOPA) lasers allow more timing adjustment than Q-Switched lasers. MOPA lasers allow for separate control of pulse duration and pulse frequency to produce highly accurate and controlled engravings or etchings. The general range of MOPAs can be as low as 2ns or as high as 500ns, allowing highly accurate adjustment of the energy dispersed into the material.

Q-Switched lasers are typically not adjustable in this manner, and operate at a fixed duration, typically between 80ns to 140ns.

Laser Timing Adjustment

Galvo lasers sometimes have granular, but adjustable tuning of when the beam is commanded on, or off, as well as how the mirrors are adjusted. Some of this calibration and adjustment is in Lens Correction, or provided by your laser manufacturer, but the fine tuning for corner approach and speed may need adjustment.

Laser on TC (µs)

When attempting to mark, the physical mirrors of the galvo head must reach the desired speed before they begin marking. This fine tuning allows the user to reduce the burn-in potential during that initial acceleration by adjusting the source timing.

Laser Off TC (µs)

Similarly to the above, when the mirrors of the galvo head must slow down as they attempt to end a marking move, there may be a burn-in effect while slowing down. This setting allows the end timing to be adjusted finely.

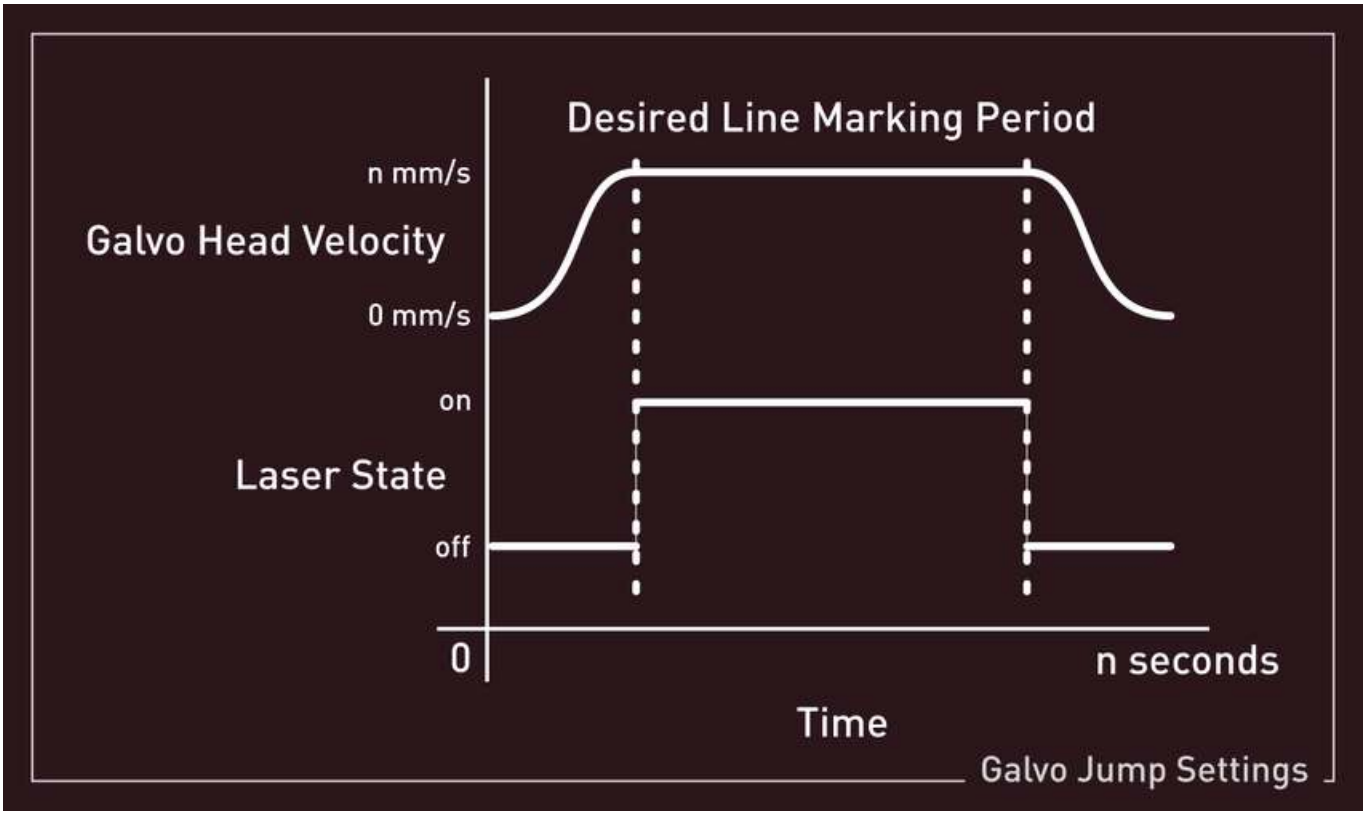
End TC (µs)

This setting adjusts how long the laser waits at the end of a series of vectors. Due to the fact the laser will physically lag behind the commanded state as decided by any software, this delay permits adjustment similarly to the Laser Off TC, except only for the end of polygons.

This setting applies to any marking move where the laser is turned off after execution.

Polygon TC (µs)

This setting adjusts how long the laser dwells at the end of each segment of a polygon, permitting the laser to settle appropriately, compensating for the same lag as outlined in End TC.

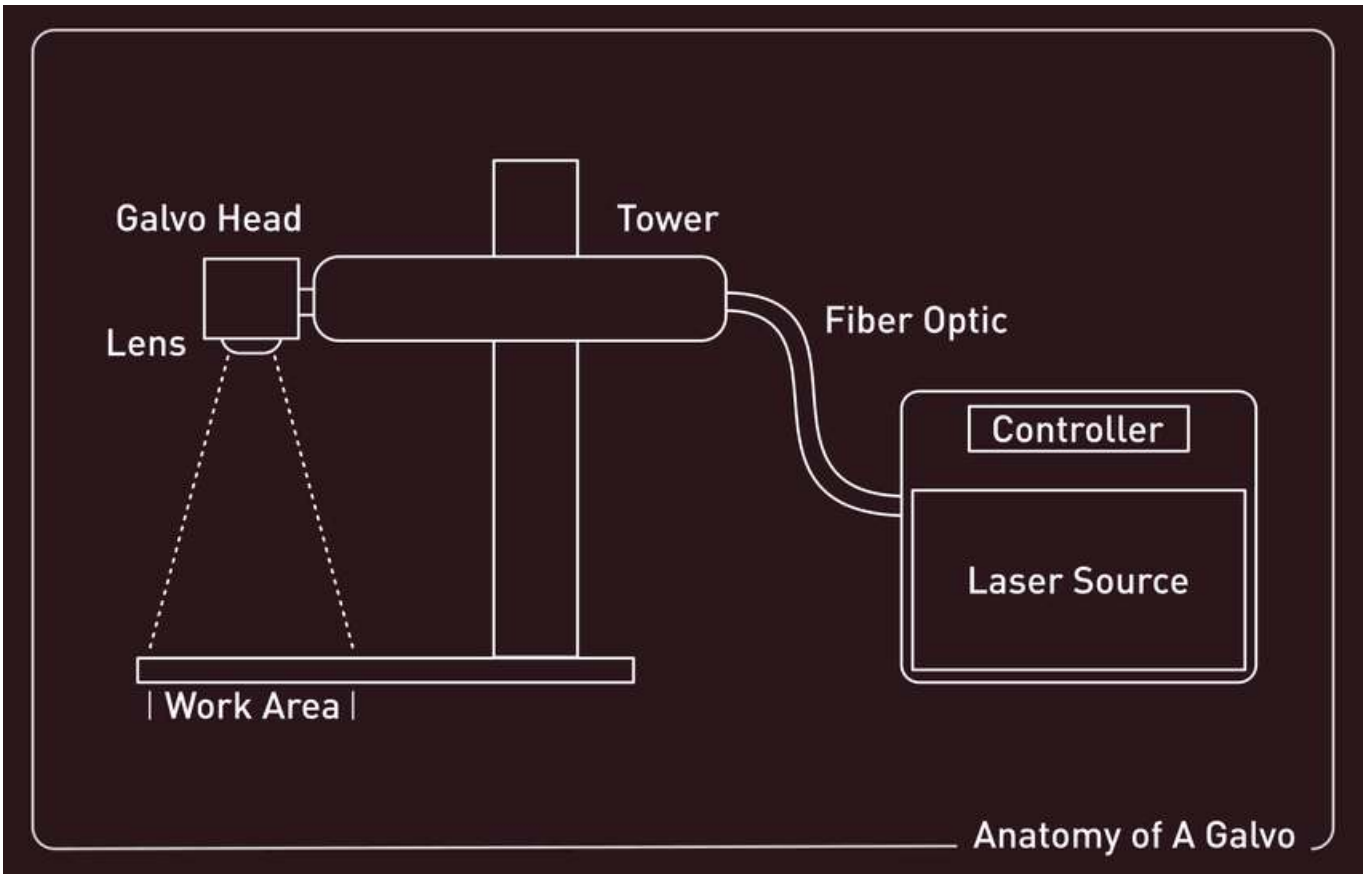

Desired Line Marking Period
n mm/s
Galvo Head Velocity
0 mm/s
on
Laser State
off
0
n seconds
Time
Galvo Jump Settings

Galvo Glossary

Below you'll find definitions of many terms commonly used when working with a Galvo laser.

Galvo

A type of laser apparatus that uses a series of actuated mirrors to reflect the beam of a laser source around at high speeds. This contrasts to gantry style lasers, that move a series of mirrors along a gantry with a lens at the end to focus the beam, or diode lasers that move the laser souce itself.



Source

This is the portion of a Galvo laser responsible for producing the beam itself used for operation. While not a laser tube inherently, laser sources are similar in purpose to the tubes found in CO_2 gantry lasers.

MOPA

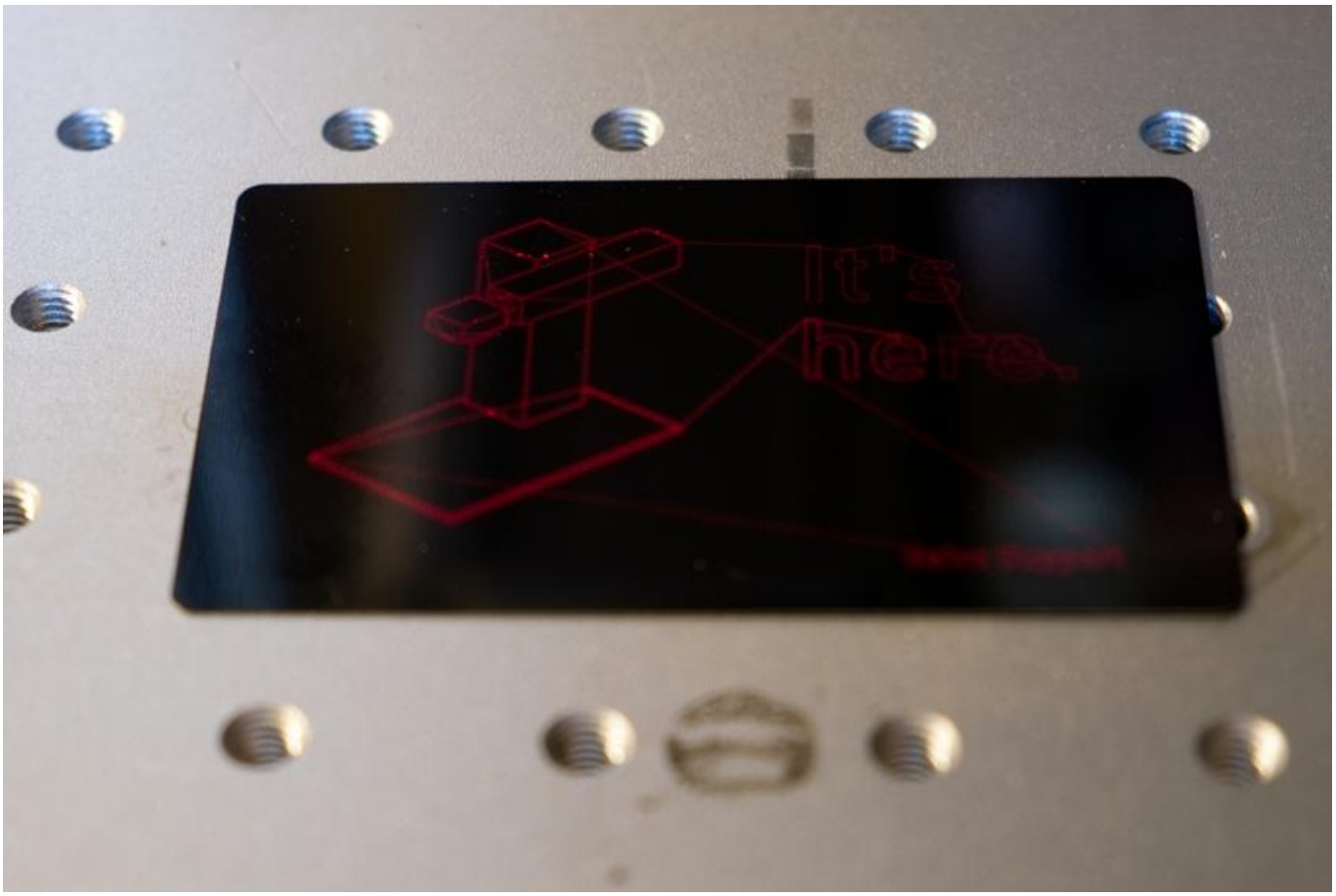
Master Oscillator Power Amplifier (MOPA) lasers are a specific class of fiber laser sources that enable pulse duration to be varied (not just pulse energy and frequency as in other sources). This variability enables use cases such as durable colored marking of stainless steels and titanium by pulsing the beam in such a way it anneals the metal.

Tower

This is the portion of Galvo lasers responsible for moving the galvo head up and down, vertically. The tower connects the head to the fiber cable (that carries the beam from the source), and any other apparatus that needs to move with the lens to focus and effectively work on the job.

Framing

Framing allows you to accurately position work. It's an operation that outlines the job in real life on the object/material, but using a lower power, so it doesn't cut/engrave it. It usually requires your laser to be accurately focused.



Frame/Base

This is the portion of a Galvo laser that holds up the tower and ensures it's rigid and won't tip over. It usually has a work table larger than the tower above it for stability.

Cleanup Pass

A final pass, usually at a tighter line width, higher frequency, and a lower power. Done to to clean up soot and debris from more ablative processing.

Focal Point (Waist)

The thinnest part of the (focused) beam, the point at which it is the most in focus. See the Focusing page for more info.

Focal Length

The distance from the optical centre of the lens to the focal point. See the Focusing page for more info.

Depth of Field (Focal Range)

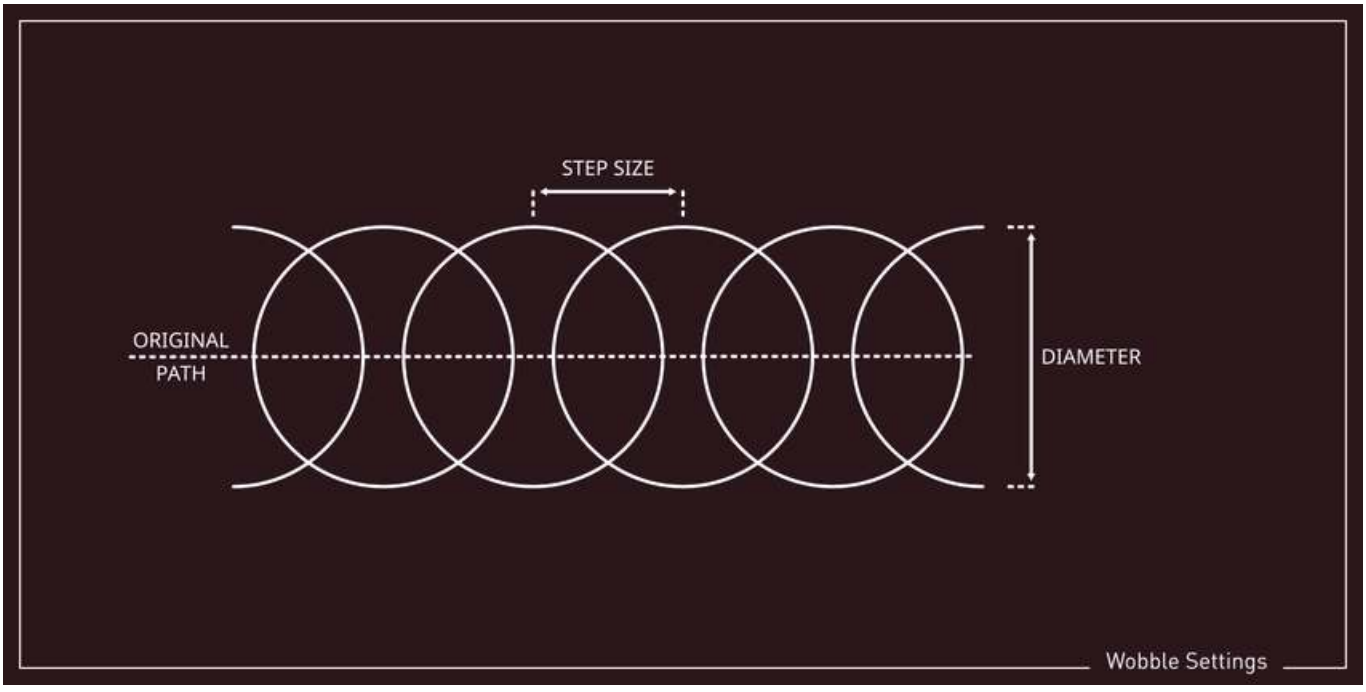
The useable range of the (focused) beam. It appears either side of the focal point, and varies based on various factors including; the focal length of the lense, the material being cut, the power of the laser source, etc. See the Focusing page for more info.

Field Size (Build Volume, Window Size)

These fields describe the maximum area that the lens installed in the Galvo can physically cover when focused. See the Focusing page for more info.

Wobble

Wobble refers to the ability of the beam to rotate about a particular area, imparting more heat to the workpiece and allowing for more material removal.



This feature can be enabled in the Cut Settings Editor on Galvo devices.

LightBurn Cut Settings and EZCAD Hatches

If you're coming from EZCAD2 and having trouble finding hatch settings you're used to, follow this guide to match your usual hatch settings to LightBurn's Fill settings.

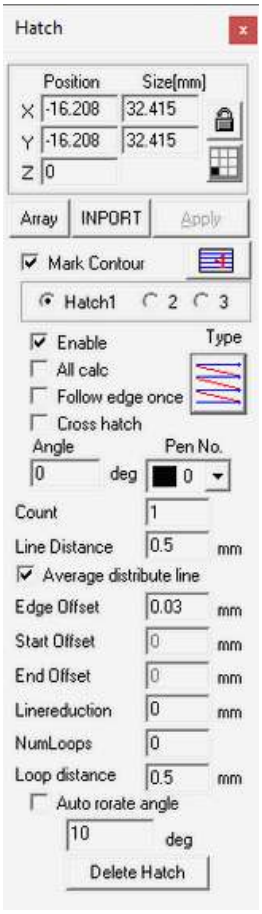
If you're not sure how to adjust cut and engraving settings, see the Cut Settings Editor page.

Mark Contour

EZCAD2's "Mark Contour" option can be replicated with LightBurn's Sub-Layers. By creating a Line Sub-Layer and a Fill Sub-Layer, you can achieve the same effect. The Sub-Layers will be cut in the order they appear in the Cut Settings Editor.

Hatch 1, Hatch 2, Hatch 3

EZCAD2 allows for three different hatches on the same geometry. This is equivalent to LightBurn's Sub-Layers.

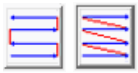


Cross-Hatch

This functionality is the same between the two programs, and is a checkbox in each.

Hatch Patterns and Settings

Unidirectional hatch & Bidirectional hatch



EZCAD2's bidirectional hatch and unidirectional hatch are the same as the default hatch in LightBurn, with the Bi-directional fill toggle either on or off.

Ring-like hatch / Pyramid Hatch



EZCAD2's ring-like hatch is equivalent to LightBurn's Offset Fill.

Snake Hatch

Snake hatch is equivalent to a regular fill in LightBurn with Flood Fill enabled.

Auto Rotate Hatch Angle

LightBurn's Angle Increment setting is equivalent to EZCAD2's "auto rotate hatch angle".

Collections

Lasers are versatile machines, and what you do with them is up to you. No matter your aim, it's likely LightBurn has a tool to help you achieve it, as long you know where to look. We've assembled a range of tools, features, and resources into collections, organized according to the goals they'll help you accomplish.

Click in the navigation to the left to learn more about a collection you're interested in, or scroll down for quick links to resources contained in each collection.

LightBurn Fundamentals

New to lasers or LightBurn? We all started out as beginners at some point, and we know there's a lot to learn — these resources cover first time set up and other fundamental concepts, so you can hit the ground running, and get to making things with your machine.

Learn More

Beginner Pathway

Layer Modes

Images vs. Vectors

Open vs. Closed Shapes

How to Invert a Vector Engraving

Speed vs. Power

Coordinates and Job Origin

Laser Types

Customizing the LightBurn Window

Tips and Tricks

LightBurn is packed with tools that speed up the process of getting from idea to finished product. We've listed some of our most useful learning tools and workflow optimization features below — check them out to become a LightBurn ace.

Learn More

Tooltips and Topic-Aware Help

Quick Tip Videos

Single-Key Align Tool Shortcuts

Arrow Key Nudging and Modifiers

Single-Key Rotation Shortcuts

Tab Key Selection Cycling

Automatic Guidelines

Snapping

Number Pad Jogging

Galvo Framing: Live Adjustments

Equation Support in Number Boxes

Copy and Paste From Other Applications

Drag and Drop From the File Browser

Automatic File Backups

Auto-Start a Job on Send

Pause and Stop From the Keyboard

Reset to Default Layout

Load Preferences Backup

Reset to Default Settings

Setting Up a New Computer

Job Time Optimization

The faster your laser finishes one job, the sooner you can start another. These tools will help you speed up your project runtimes, without any loss in quality of the finished product, so you can spend less time waiting and more time making things.

Learn More

Preview Window Time Estimates

Line Interval

Fill Mode Grouping

Offset Fill Mode

Flood Fill

Fast Whitespace Scan

Optimization Settings

Material Test

Material Library

Overscanning

Material Utilization

No one likes it when unusable scraps pile up in their workshop, so the more you can get out of a sheet of acrylic or piece of wood, the better. These tools will help you lay out and arrange graphics so they're close-fitting, directly next to each other, or perfectly positioned for the material you're working with.

Learn More

Grid Array

Docking

Distribute and Move Together

Align Tools

Using a Camera

True Nest

Quick Nest

SVGNest (External)

Snapping

Move Laser to Selection

Move Selected Objects

Remove Overlapping Lines

Batch Production

Once you've mastered the craft of making one thing really well, you're going to want to make many more of that thing really well. Use these tools to lay out copies of your design, or set your laser up for several runs in succession. If you're a small business owner with orders to fill, these features will help you scale up and expedite your production process.

Learn More

Grid Array

Virtual Array

Variable Text

Bar Codes and QR Codes

Repeat Marking

Print and Cut

Feeder Setup

Rotary Mode

Design Tools

Looking to get creative? LightBurn contains a suite of tools for designing projects from scratch. If you have a background in other design software, many of these tools may be familiar to you — instead of bouncing between programs, you can keep all your laser-design work under one roof.

Learn More

Creation Toolbar

Transform Controls

Shape Properties Window

Art Library

Grouping and Ungrouping

Lock Shapes

Tool Layers

Automatic Guidelines

Circular Array

Copy Along Path

Apply Path to Text

Creative Uses for Advanced Cut Settings

Getting a Design Laser-Ready

So you've imported a design into LightBurn that wasn't made with laser engraving in mind, and it's going to take some work to get it ready to go. Don't worry — the tools below are up to the job and then some.

Learn More

Preview

View Style

Delete Duplicates

Boolean Tools

Trim Shapes

Edit Nodes

Convert to Path

Close Path

Close Selected Paths With Tolerance

Auto-Join Selected Shapes

Optimize Selected Shapes

Resize Slots

Kerf Offset

Measuring Kerf Width

Working With Images

Engraving images is one of the coolest things your laser can do, but it's also one of the hardest processes to get just right. Check out the following pages to learn the basics of image engraving with LightBurn, along with some advanced techniques for when you're a seasoned pro.

Learn More

Image Mode

Images vs. Vectors

5 Steps to Perfect Image Engraving

Dot Width Correction

Adjust Image

Apply Mask to Image

Image Tracing

Convert to Bitmap

LightBurn for Galvo Lasers

If you're new to using LightBurn with your Galvo laser, you might be feeling like there's a lot to learn — and there is! But don't let that worry you, we've got you covered with resources on getting started with LightBurn, learning what you can do with your Galvo laser, and overcoming common stumbling blocks.

> **Note**
>
> If this is your very first time using LightBurn, we recommend taking a look at our beginner pathway for an introduction to LightBurn's interface and features.

Learn More

Galvo Driver Installation

Collections

Galvo Device Profile Setup

Galvo Lens Correction

Changing a Galvo Laser Lens

Timing Settings

IO Port Settings

First Project Video Guide

Galvo Framing

Using a Rotary With a Galvo Laser

Cylinder Correction

3D Sliced Engraving

Glossary of Galvo Terminology

EZCAD Hatch Quick Translation

About Galvos

Galvo Troubleshooting and FAQ

Tips and Tricks

LightBurn is packed with tools that speed up the process of getting from idea to finished product. We've listed some of our most useful learning tools and workflow optimization features below — check them out to become a LightBurn ace.

Tooltips and Topic-Aware Help

To help users learn how to use LightBurn, we've included **Tooltips** and a **Topic-Aware Help** feature in the software.

Tooltips are brief descriptions of the action a tool performs. To reveal a Tooltip, hover your cursor over anything you'd like to learn more about.

When you press `F1` while hovering over a tool, LightBurn's Topic-Aware Help function launches the relevant documentation page in your default web browser.

> **Note**
>
> If you're using a Mac, hold the `Fn` key while pressing `F1`.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=erKGi8LSu_g

Quick Tip Videos

Check out our YouTube channel to see dozens of short videos with tips and tricks for using LightBurn effectively.

We're always producing new videos with time-saving tips, so be sure to subscribe while you're there so you never miss out.



Visit Our Channel

Single-Key Align Tool Shortcuts

You can perform most of the Align commands with single-key shortcuts when the **Edit Window** (Workspace) — the area where you create and edit graphics — is in active focus.

> **What does it mean for a window to be in focus?**
>
> There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.
>
> If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus. Selecting objects in the Workspace will also bring it into focus.

Align Option	Icon	Action	Hotkey
Align Vertical Centers		Aligns selected objects along their vertical centers.	C
Align Horizontal Centers		Aligns selected objects along their horizontal centers.	E
Align Left		Aligns selected objects along their left edges.	L
Align Right		Aligns selected objects along their right edges.	R
Align Bottom		Aligns selected objects along their bottom edges.	B

Align Option	Icon	Action	Hotkey
Align Top		Aligns selected objects along their top edges.	T

Learn More

Arrow Key Nudging Modifiers

You can use your keyboard's arrow keys to move objects in your Workspace by set increments.

These hotkeys only work when you have objects selected and the **Edit Window** (Workspace) — the area where you create and edit graphics — is in focus.

> **What does it mean for a window to be in focus?**
>
> There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.
>
> If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus. Selecting objects in the Workspace will also bring it into focus.

- The `Left`, `Right`, `Up`, and `Down` arrow keys nudge your selection by **5 mm**, by default.
- Hold `Ctrl`/`Cmd` while pressing an arrow key to nudge your selection **1 mm**, by default.
- Hold `Shift` while pressing an arrow key to nudge your selection **20 mm**, by default.
- Hold `Ctrl`/`Cmd`+`Shift` while pressing an arrow key to nudge your selection **0.1mm**, by default. This value is always 1/10th of the `Ctrl`/`Cmd` nudge increment.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=7T5fuJX6fhI

You can adjust the nudge distance of each arrow key and modifier in the **Units and Grids** tab of the Settings window, under Shape Move Increments.



Learn More

Single-Key Rotation Shortcuts

You can rotate objects by set increments using hotkeys.

These hotkeys only work when you have objects selected and the **Edit Window** (Workspace) — the area where you create and edit graphics — is in focus.

> **What does it mean for a window to be in focus?**
>
> There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.

> If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus. Selecting objects in the Workspace will also bring it into focus.

Action	Windows	macOS
Rotate 90° clockwise	`.`	`.`
Rotate 90° counterclockwise	`,`	`,`
Rotate 45° clockwise	`Shift`+`.`	`Shift`+`.`
Rotate 45° counterlockwise	`Shift`+`,`	`Shift`+`,`
Rotate 15° clockwise	`Ctrl`+`.`	`Cmd`+`.`
Rotate 15° counterclockwise	`Ctrl`+`,`	`Cmd`+`,`
Rotate 1.5° clockwise	`Ctrl`+`Shift`+`.`	`Cmd`+`Shift`+`.`
Rotate 1.5° counterclockwise	`Ctrl`+`Shift`+`,`	`Cmd`+`Shift`+`,`

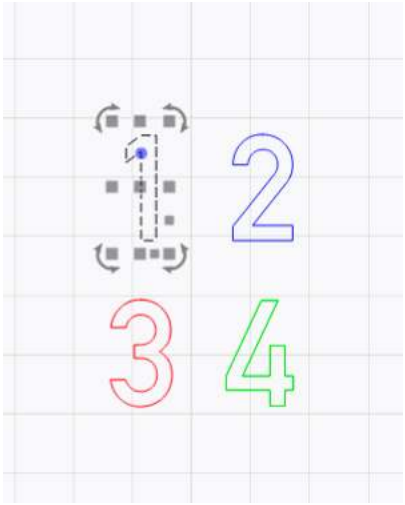
Learn More

Tab Key Selection Cycling

Press `Tab` to cycle your selection sequentially through objects or Grouped objects in your **Workspace**. Press both `Tab` and `Shift` to cycle in reverse order.

Sequence is determined by the order in which the objects were created.

Learn More



Automatic Guidelines

LightBurn features two types of automatic guidelines to help you align objects relative to one another, or along vertical or horizontal planes.

Alignment Guides are temporary visual guides that indicate when an object's edge or center is aligned to an edge or center of another object in your Workspace.

Draggable Guidelines are persistent lines set to the T1 Tool layer — they are perfectly vertical or horizontal by default, and you can snap other objects to them.

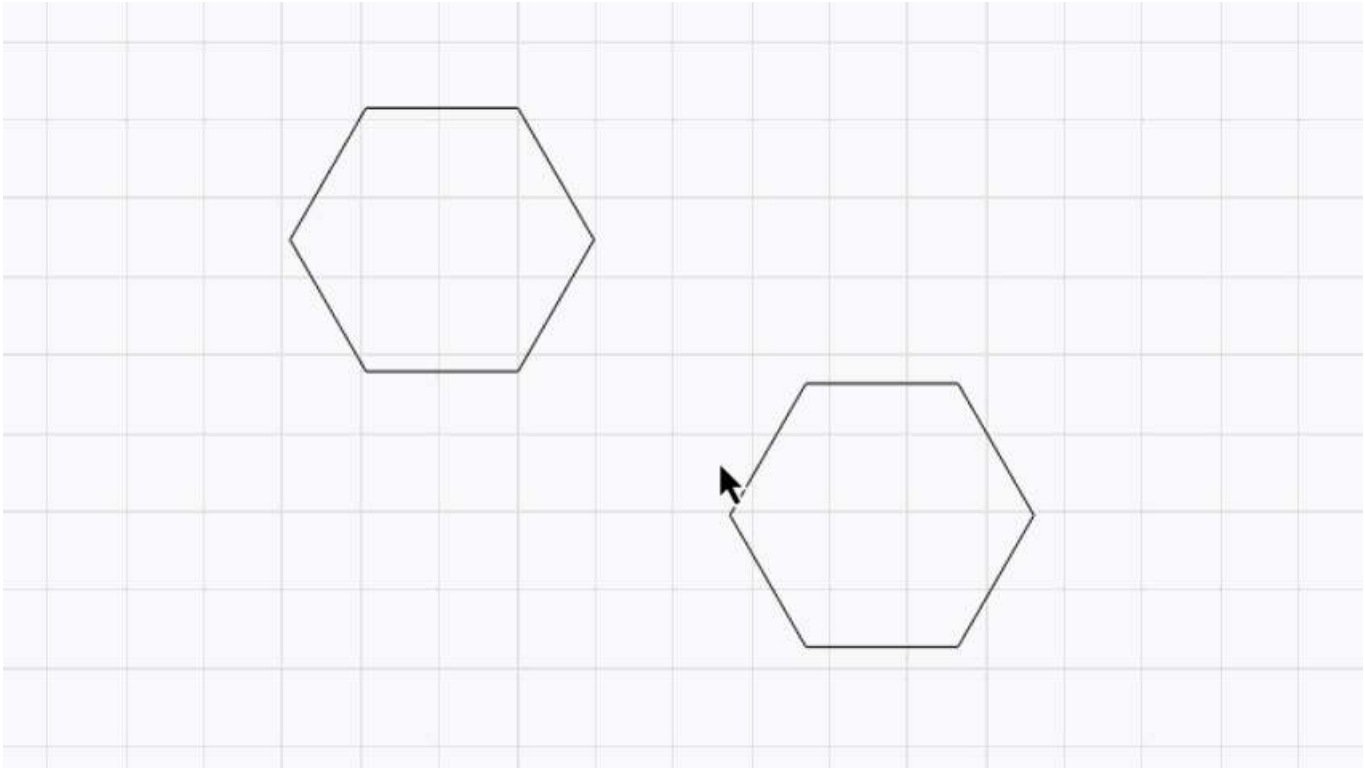
Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=TOF4luuVVTE

Snapping

LightBurn's automatic **Snapping** tools help align objects to other shapes in your Workspace, or ..to the Workspace grid itself — when you move objects close to a valid location, they will snap to that point.

As you hover your cursor over an object, the icon will change to indicate when it is over a node, line, midpoint, center, or intersection. Use these Snapping points when selecting, moving, or creating objects to ensure exact alignment.



Snapping Point	Icon	Description
Node		The cursor is over any node in a vector graphic, or the corner of an image.
Midpoint		The cursor is over the exact halfway point between two nodes.
Center		The cursor is over the center point of an object.
Intersection		The cursor is over the intersection of two lines.

Snapping Point	Icon	Description
Line		The cursor is over a line, but not a node, midpoint, or intersection on that line.

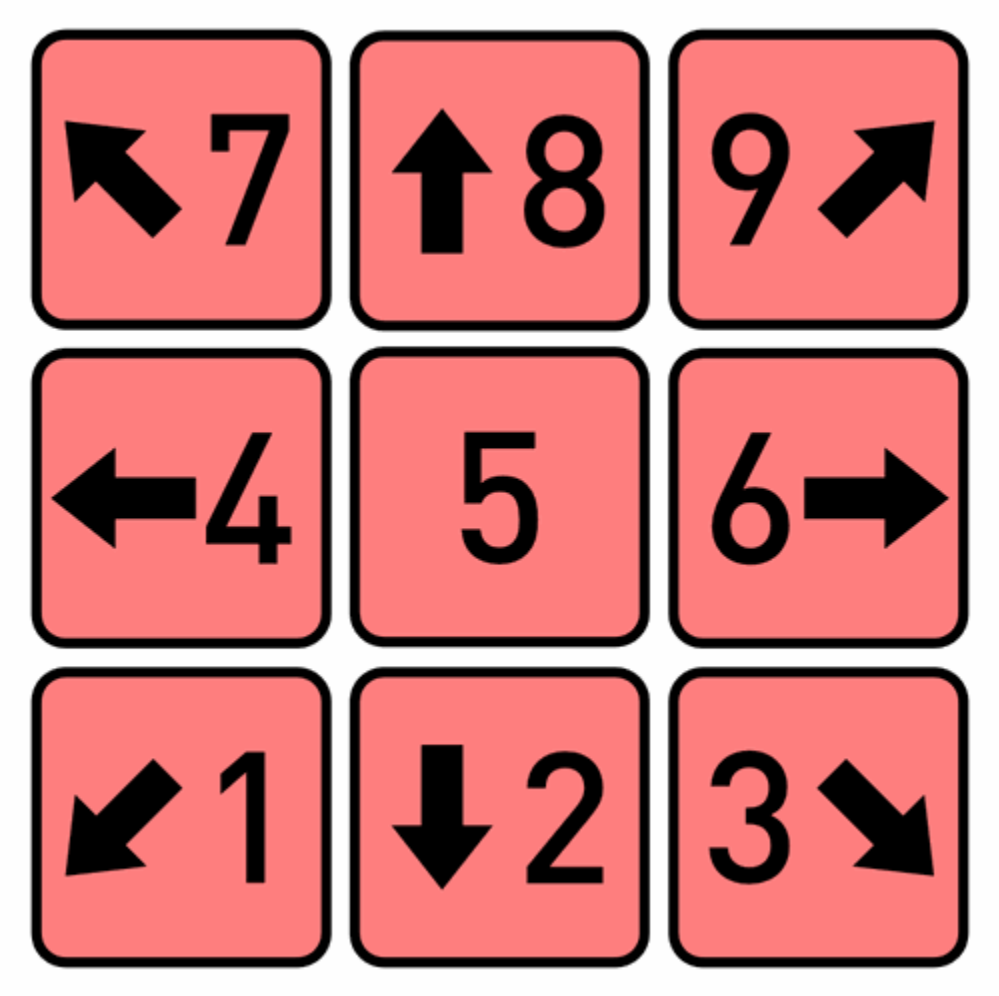
Learn More

Number Pad Jogging

You can use a keypad to Jog your laser. In order to use keypad Jogging, you must enable **Numlock** on your keyboard, and the **Edit Window** (Workspace) — the area where you create and edit graphics — must be in focus.

> **What does it mean for a window to be in focus?**
>
> There are several Windows and Toolbars in LightBurn that accept user input — when a window is clicked on, it is in focus and accepting input.
>
> If a window other than the Edit Window is in focus, click anywhere in the Workspace to bring it into focus. Selecting objects in the Workspace will also bring it into focus.

Each press will move your laser by amount entered in the **Distance** field, at the rate set in the **Speed** field. If Continuous Jog is enabled, the laser will Jog until you release the key.



Learn More

Galvo Framing Mode Nudging, Rotating, and Rescaling

If you're using a Galvo laser, you can use your keyboard to nudge, rotate, and rescale graphics while framing.

Key(s)	Result
`Left` / `Right` / `Up` / `Down`	Move graphics by 5 mm
`Shift` + arrow keys	Move graphics by 20 mm
`Ctrl` + arrow keys	Move graphics by .5 mm
`Page Up`	Scale graphics up
`Page Down`	Scale graphics down
`.` (period)	Rotate graphics 15° clockwise
`,` (comma)	Rotate graphics 15° counter-clockwise

Key(s)	Result
`Shift`+`.` (period)	Rotate graphics 1.5° clockwise
`Shift`+`,` (comma)	Rotate graphics 1.5° counter-clockwise
`Ctrl`+`.` (period)	Rotate graphics 45° clockwise
`Ctrl`+`,` (comma)	Rotate graphics 45° counter-clockwise

If your keyboard doesn't have `Page Up` or `Page Down`, try using `Fn`+`Up` or `Fn`+`Down`.

Learn More

Equation Support and Automatic Unit Conversion

Each of the fields in the **Numeric Edits Toolbar** accepts equations — enter the appropriate symbols for addition (`+`), subtraction (`-`), multiplication (`*`), or division (`/`) and press `Enter` to perform a calculation.

The **XPos**, **YPos**, **Width**, and **Height** fields accept units of measurement as well.

If you are working in millimeters, but you want to create a shape that is 5 inches wide, you can enter 5in or 5" into the width field and LightBurn will convert it to millimeters for you.

This works in the opposite direction too — if you're working in inches, you can enter mm or cm and LightBurn will convert the values to inches.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=tk9BUgZFWWo

Copy and Paste From Other Applications

You can copy text, vector graphics, and images from other design software, and paste them directly into your LightBurn Workspace.

Drag and Drop From the File Browser

Drag and drop graphics saved in supported file formats directly from your system's file browser into your LightBurn Workspace.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=oQQK_ES4Xgo

Automatic File Backups

LightBurn automatically saves backups of your projects as you work. Auto-saves are stored in the same location as you saved the original file, with `_backup` appended to their name. If you've never named a file, the auto-save will be in your computer's Documents folder.

You can adjust the frequency of auto-saves in your Settings.

Learn More

Auto-Start a Job on Send

The **Send** button, if your laser supports it, transfers the current job to the laser as a named file, so you can select and run it from the laser's controller.

When you use **Start**, the controller begins running the job as soon as sufficient data has reached the controller. Using Send instead of Start can sometimes help prevent file corruption during transfer.

Hold `Shift` while clicking Send to have the laser immediately start the job after the entire project has been sent.

> **Note**
>
> The Send option is only available for DSP lasers.

Learn More

Pause and Stop From the Keyboard

While your laser is running, you can use your keyboard's `Pause` key to Pause the laser and `Ctrl` +`Break` to Stop it. `Break` is typically on the same key as `Pause`, but more compact keyboards may not have this key.

Learn More

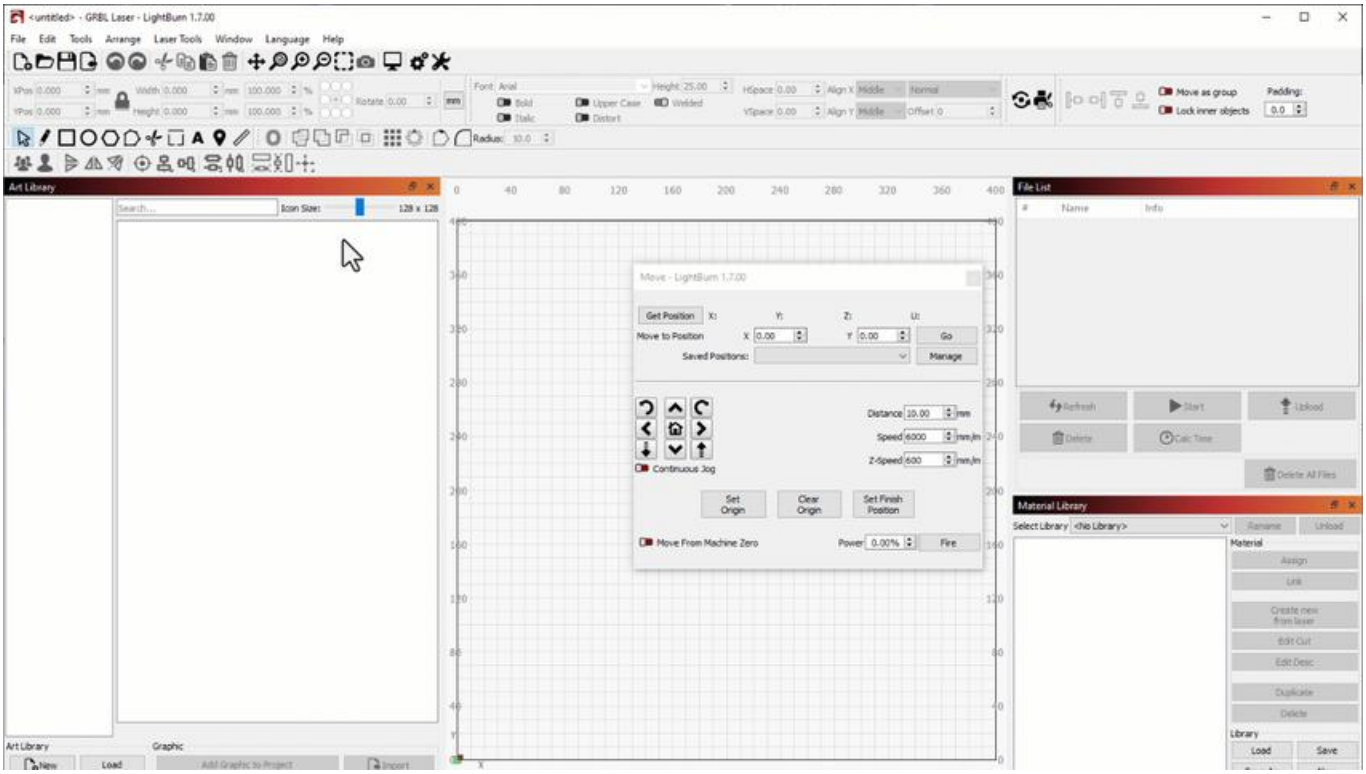
Reset to Default Layout

Reset to Default Layout restores the layout of the full LightBurn window to the same format it was in when you first installed and opened LightBurn. If you've made lots of changes to your layout that you want to reverse all at once, or you've lost sight of the Laser Window, Cuts / Layers Window, Color Palette, or any other default windows or toolbars, use this option to get them back.

Resetting the layout doesn't reset anything else — your preferences, devices, and libraries are all preserved.

Go to Window → **Reset to Default Layout** to re-open and re-dock all default windows in their original order, while closing any that are not visible by default.

Holding `Shift` while clicking to open LightBurn will also reset the layout.

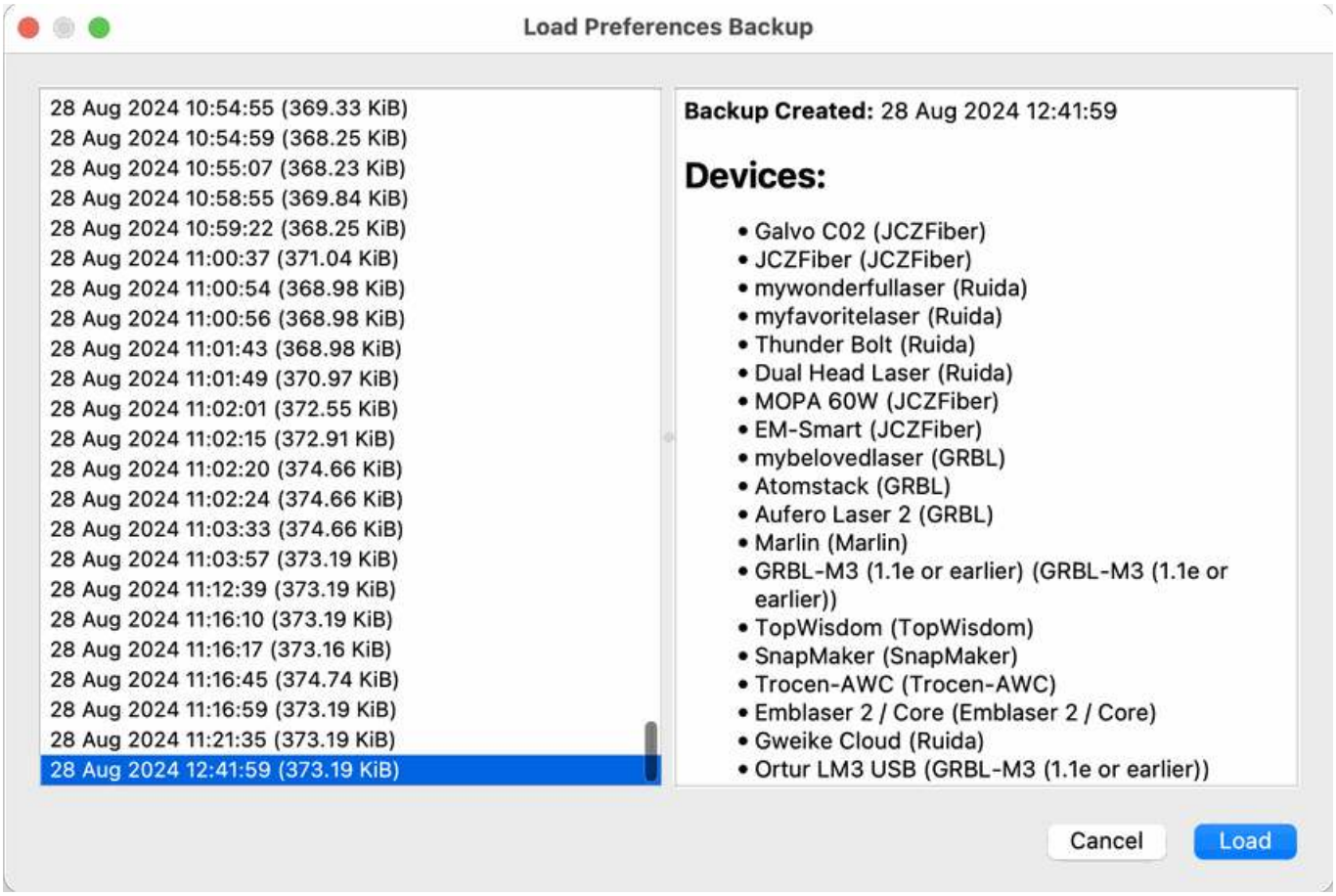


Learn More

Load Preferences Backup

The **Load Prefs Backup** command allows you to import preferences from a timestamped list of automatic backups. Click a backup in the list to see the list of Devices available in that backup, on the right side of the window. Use the list of devices and the timestamp showing when the backup was made to find and select the backup you want to use.

Click **Load** to restore the settings from the selected backup, or **Cancel** to close the window and maintain your current settings.



Learn More

Reset to Default Settings

Follow these steps to restore LightBurn to the same settings as when you first installed — be aware that you will also lose any device profiles you've created in this process.

> **Warning**
>
> While this process can be helpful in cases where you have a great many incorrect settings and want to start over, we recommend trying Load Prefs Backup or Reset to Default Layout before a full reset.
>
> We also recommend making a User Bundle that can be reimported into LightBurn if needed.

1. Open LightBurn.
2. Go to File → **Preferences** → Open Prefs Folder. You should see a pair of files called `prefs.ini` and `prefs.old` along with a folder called `backup`, along with a few other files.
3. Close LightBurn. If you leave LightBurn open, the settings you're trying to delete will automatically be restored when you eventually do close the program.
4. Delete all the files and folders in the prefs folder. If you leave the `backup` folder, LightBurn will automatically restore the settings you're trying to erase from the backups.

5. Re-open LightBurn. All settings should be reset to default, and all devices will be removed.

That’s it. When you re-run you’ll be asked to set up your laser again, and everything will be as new.

Setting Up a New Computer

LightBurn has tools to help you move your settings and devices from an existing installation to a new computer without any hassle.

The migration process starts with exporting a User Bundle from the existing installation and importing it into the new installation.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=S0QrokNepCQ

Learn More

Choosing a Device on Launch

To open LightBurn with a specific device active, launch LightBurn from the command line, using the `--devname` argument to specify a device.

For instance, using PowerShell in Windows, running `.\LightBurn.exe -- devname SmallDiode` would open LightBurn with the "SmallDiode" device selected.

Job Optimization

The faster your laser finishes one job, the sooner you can start another. These tools will help you speed up your project runtimes, without any loss in quality of the finished product, so you can spend less time waiting and more time making things.

Preview Window Time Estimates

The **Preview** window simulates movements and lasering operations before sending a project to the machine. You can use it to see the order in which cuts will be completed, and check that your laser will do what you expect, but that's not all.

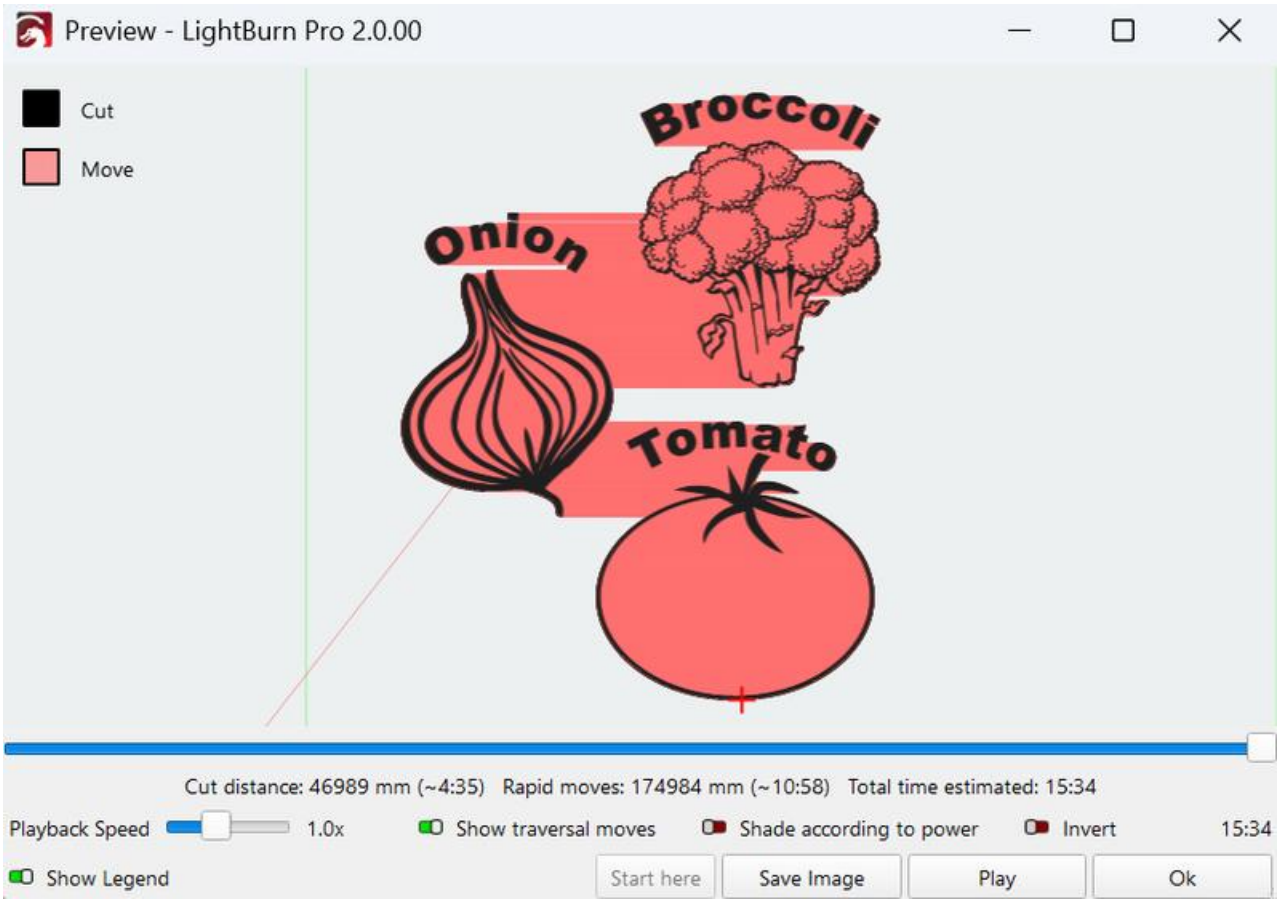
One of the most useful features in the Preview window is the ability to calculate and show the **Total estimated time** it will take your laser to complete your project. As you make changes to optimize your project's runtime, check back in the Preview window to see which settings result in the fastest times.

Learn More

Video: Getting Accurate Time Estimates

The **Additional Settings** tab of **Device Settings** controls the settings used in Preview for simulating a preview and estimating time. Changing these settings will not change anything about how the job actually runs, but can make time estimates more accurate.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=s1jCqB233N0



Line Interval

Line Interval is the distance between scan lines when raster engraving, making it one of the key settings determining the quality of engravings on Image and Fill Layers. It is mathematically tied to the **Lines per Inch** value.

Lower Line Interval values lead to a greater density of lines. The ideal Line Interval will produce lines that just touch, without overlapping.

Increasing Line Interval to the maximum amount that doesn't leave gaps between lines will lead to reduced runtimes for your projects, with no loss in quality, but you may also need to increase Power to achieve dark engravings.

Use the Interval Test or Material Test to find the optimal Line Interval for your laser and material.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=kpMunzp2NM8

Fill Mode Ordering

The Fill Mode order-by options control which shapes are engraved at the same time by the laser. If you run your laser fast, or your laser accelerates slowly, it is often most efficient to scan things all at once, so the laser spends most of its time moving at the speed you've chosen, and less time changing direction.

If you are engraving slowly, your laser accelerates fast, or the design contains a lot of blank space, it can be more efficient to fill clusters of close shapes, or to fill the shapes one by one.

Learn More

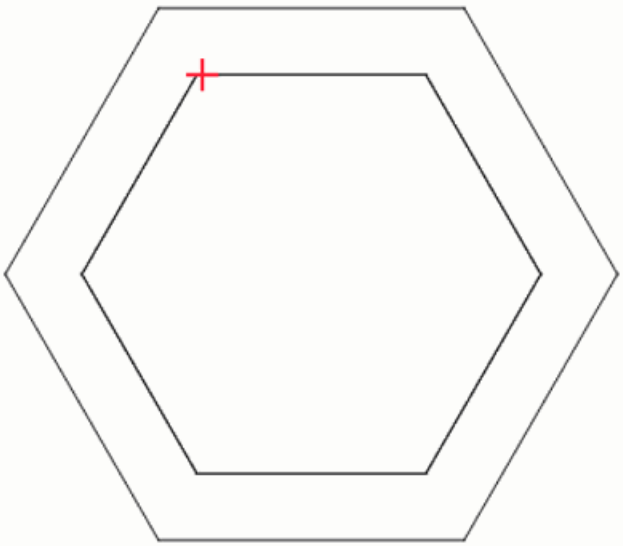
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=ZRGSIm5y8aU

Offset Fill

Offset Fill Mode is a special type of Fill operaton that tells your laser to etch concentric lines that follow the contour of the shape.

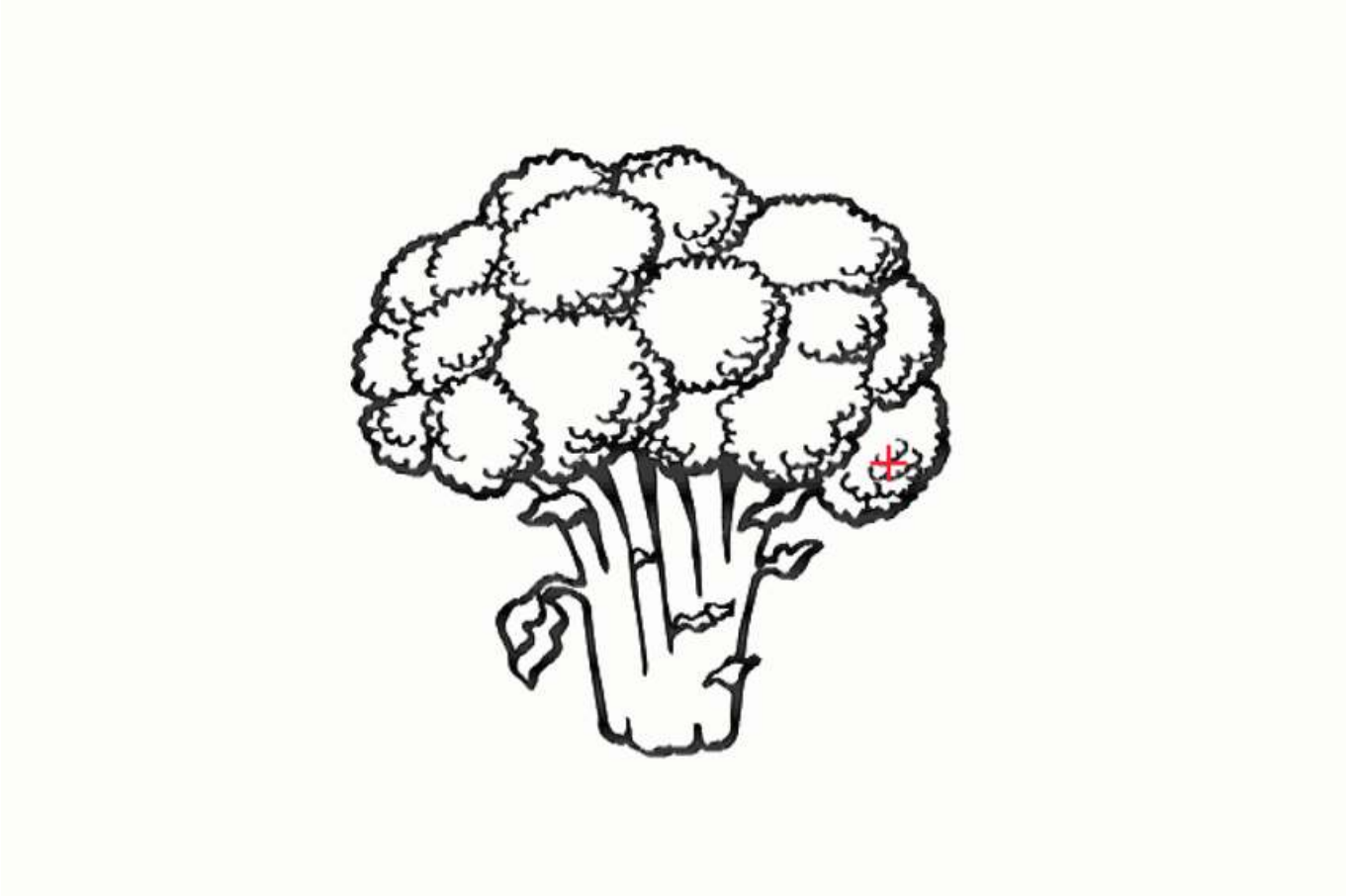
Offset Fill is intended for designs with a lot of empty space within graphics, to cut down on the amount of travel time necessary to fill large, hollow shapes. If that description fits your design, Offset Fill will most likely be much faster than Fill Mode. If that doesn't apply to your graphics, it's usually best to use Fill instead of Offset Fill. Offset Fill is generally not a good choice for complex shapes.

Learn More



Flood Fill

Flood Fill calculates an engraving path that attempts to reduce or eliminate travel moves across blank space. It's useful for engraving designs with lots of empty space within graphics, where traveling across the empty portions would otherwise consume a great deal of time.



Flood Fill is an especially useful option for machines that accelerate quickly, and for Galvo lasers in particular.

Learn More

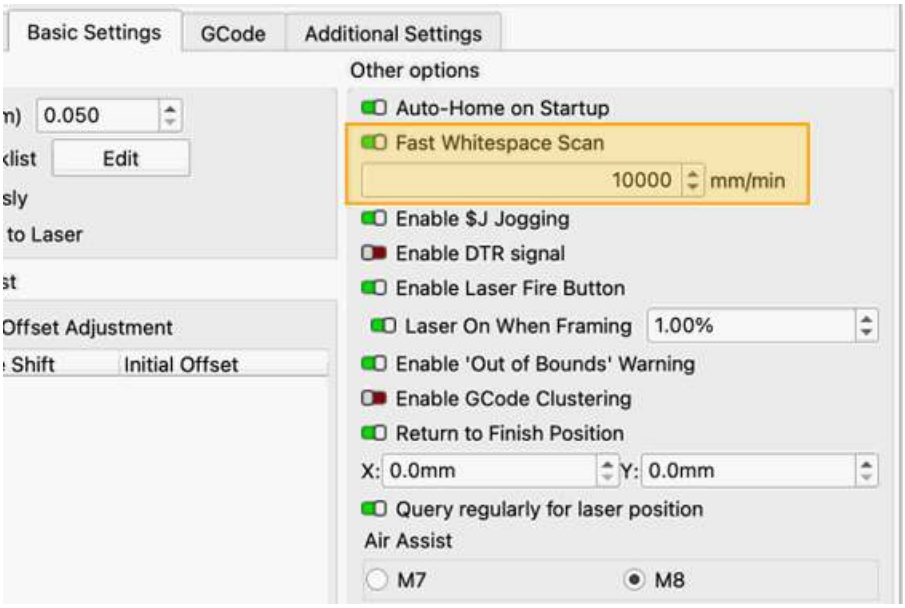
Fast Whitespace Scan

> **Note**
>
> This setting is only available for GCode-based lasers.

This setting boosts your laser's travel speed through blank areas of an engraving, if the speed you set here is faster than your laser's engraving speed.

This can save a lot of time if you're engraving a design slowly to get a good burn, particularly if there are large blank areas. This setting applies to both images and filled vector engravings.

Learn More



Optimization Settings

LightBurn's **Cut Planner** controls the order in which your laser cuts or engraves graphics in your project. The **Cut Optimization Settings** window is where you adjust the variables by which the Cut Planner decides on cut order and pathing.

Understanding these settings will help you optimize for both quality and speed. One of the most powerful options is the Remove Overlapping Lines setting, which tells your laser to cut lines that are shared by two graphics just once.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=tHTnb6MURdU

Material Test

LightBurn has a built-in test pattern generator to help you find the best settings for your laser and any material you're working with.

To open the **Material Test Generator**, go to the Laser Tools Menu and select **Material Test**.

It will take you a little extra time up front, but identifying the ideal combinations of **Speed**, **Power**, and other settings to produce high quality engravings as quickly as possible will help you maximize efficiency down the line.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=pTx0RG_4Wbs

Material Library

LightBurn's **Material Library** provides a place to store, organize, and quickly re-apply preset Cut Settings for different operations and materials.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=jt8M2cJlkH4

Overscanning

When your laser engraves graphics set to Fill or Image Mode, the line-by-line, back-and-forth movement across the graphics is sometimes called scanning.

Overscanning refers to additional moves added to the beginning and end of each line to give the laser space to speed up before firing, and slow down afterward, which in turn prevents the edges of engraved areas from burning darker than the centers.

To optimize job time it's important to understand the relationship between job **Speed** and Overscanning amount. Because high Speed settings require more distance and time for acceleration and deceleration, increasing Speed does not always lead to a much faster overall job time — in some cases, increasing past a certain point can actually lead to an increase in job time, rather than a reduction.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=oNW8lTxuul4

Material Utilization

No one likes it when unusable scraps pile up in their workshop, so the more you can get out of a sheet of acrylic or piece of wood, the better. These tools will help you lay out and arrange graphics so they're close-fitting, directly next to each other, or perfectly positioned for the material you're working with.

Grid Array

The **Grid Array** tool creates copies of an object (or objects) in regularly spaced rows and columns, and includes options to adjust spacing, mirror the shapes, randomize orientation, and more.

It's a powerful tool for duplicating graphics and creating patterns, but that's not all it can do — you can use the **X/Y Spacing**, **X/Y Shift**, and **Mirror** options individually or in combination with one another to efficiently lay out objects for maximum material utilization.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=WhjJvWE7T1k

Docking

The **Docking** tool moves selected objects left, right, up, or down, until they meet the edge of the Workspace or another object. Docking can bring objects together or move them in unison, making sure they maintain their relative position to one another.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=gmIcv56YXB4

Distribute and Move Together

The **Distribute** tools move selected objects to evenly space them either vertically or horizontally, with options to set the same distance between the centers, or edges of each object.

The **Move Together** tools move selected objects so that they are evenly distributed, and their edges are abutting along horizontal or vertical planes. a **Learn More**

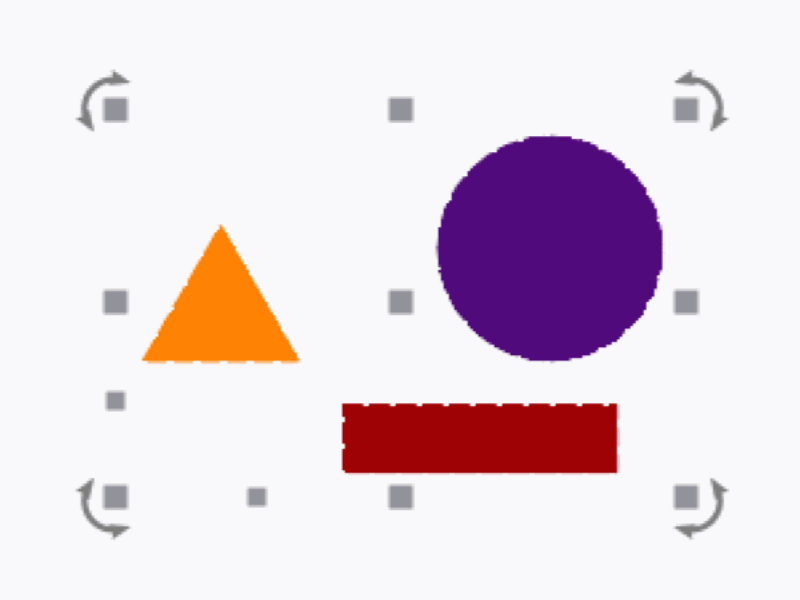
Align Tools

The **Align** tools reposition selected objects to align them along their:

- Top, bottom, left, or right edges
- Vertical or horizontal centers
- Center points

By default, the Align tools move objects in your selection to align them to an edge or center of the last object you added to your selection.

Learn More



▶ Watch the video on YouTube: https://www.youtube.com/watch?v=jbSoR-FcHSY

Using a Camera

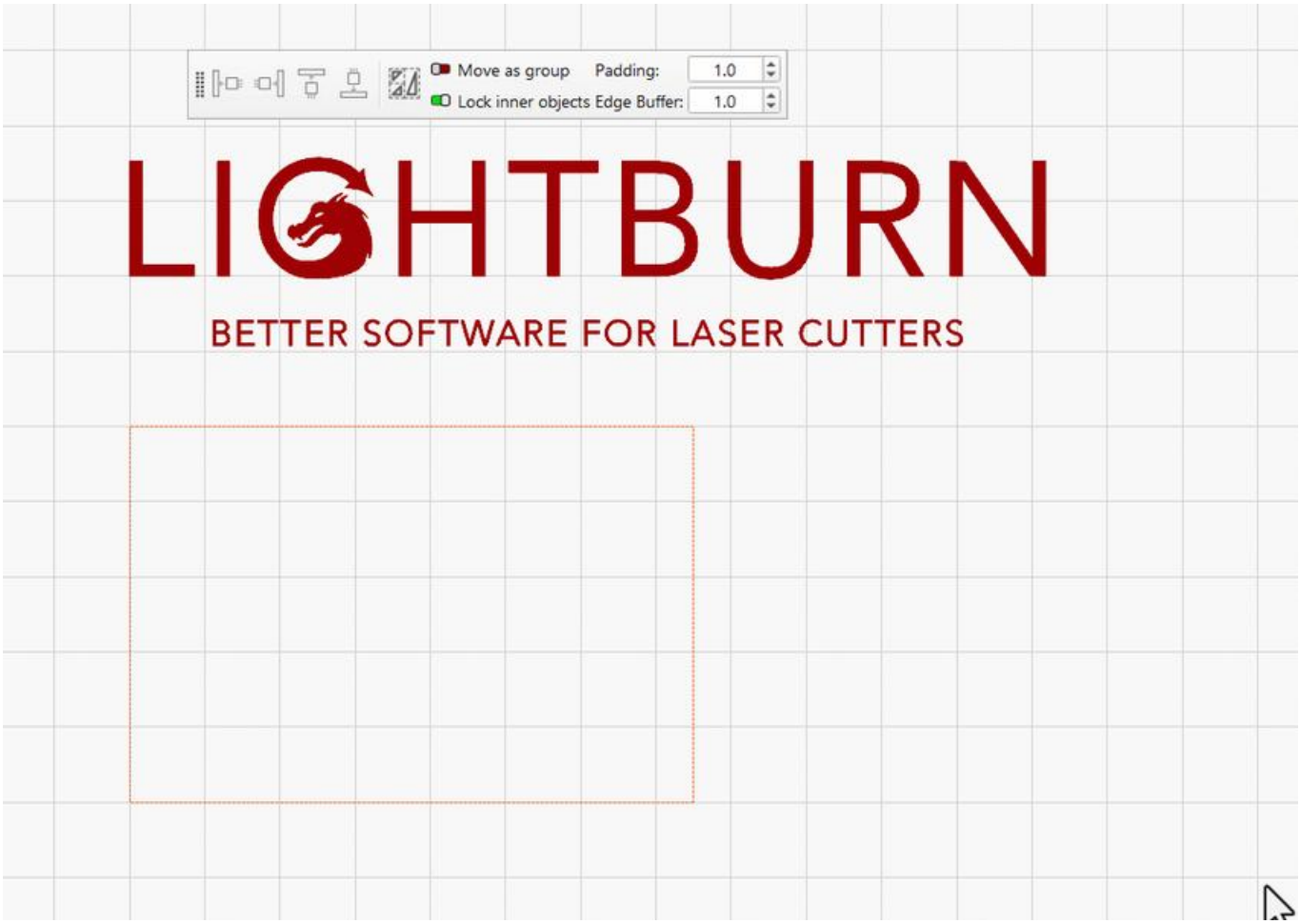
With a properly calibrated and aligned camera, you'll be able to position graphics directly over objects shown in your Workspace Overlay, and they'll be cut or engraved on the objects, using Absolute Coords positioning.

Cameras are especially helpful for getting the most of out of oddly shaped or previously cut materials.

Learn More

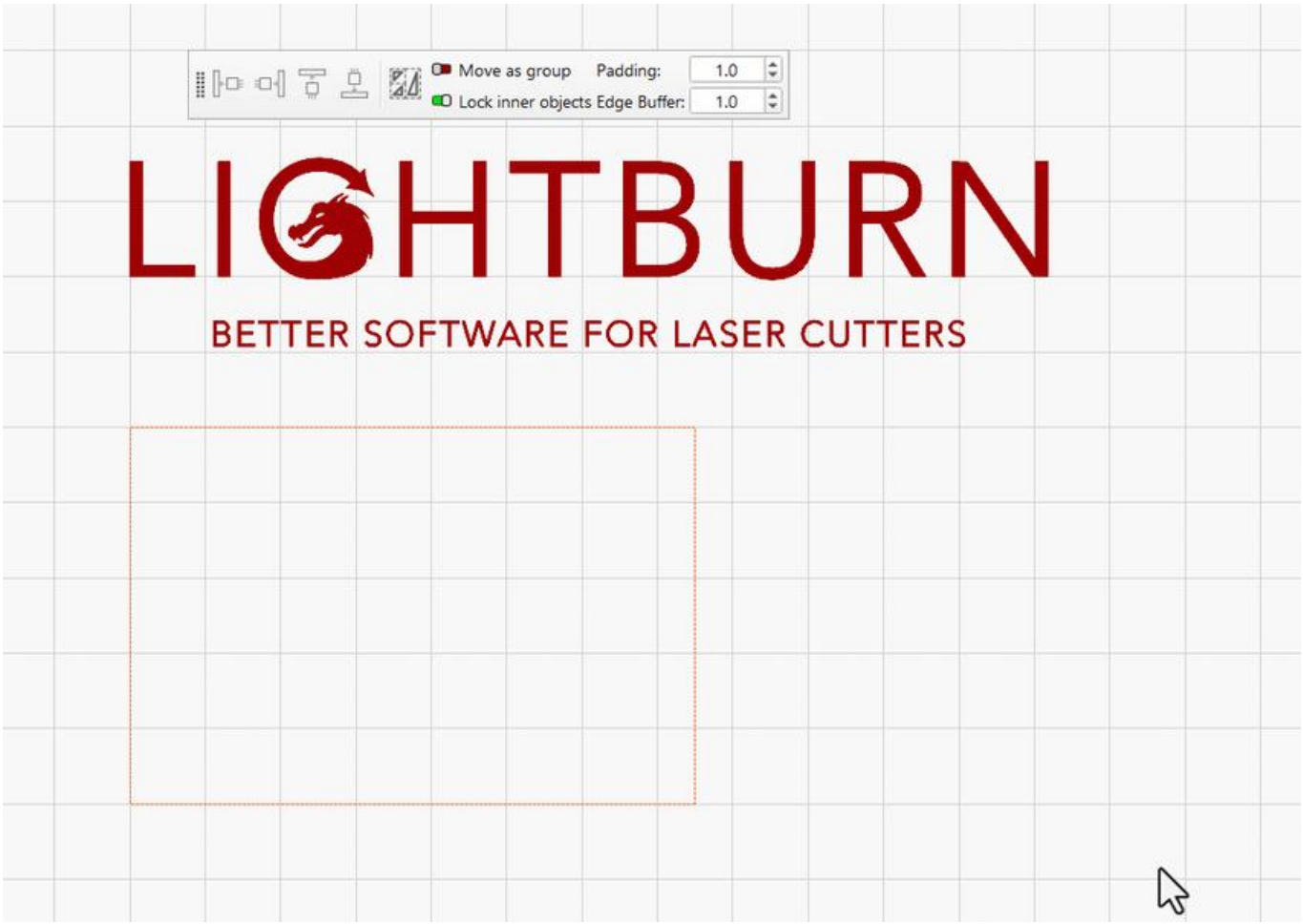
True Nest

Nesting allows you to fit more parts in less material by packing your shapes efficiently into the space availble. LightBurn 2.2 introduces a new feature, **True Nest**, which offers built-in nesting using the true shape of the objects, including multi-sheet nesting for working with more parts than you can fit on a single sheet og material. True Nest is only available for LightBurn Pro.



Quick Nest

Quick Nest offers quick, simple nesting without leaving LightBurn. Quick Nest uses bounding box nesting, which calculates nesting based on the smallest rectangle each shape fits into. Quick Nest requires LightBurn 2.1 or newer. You don't need LightBurn Pro to use Quick Nest.

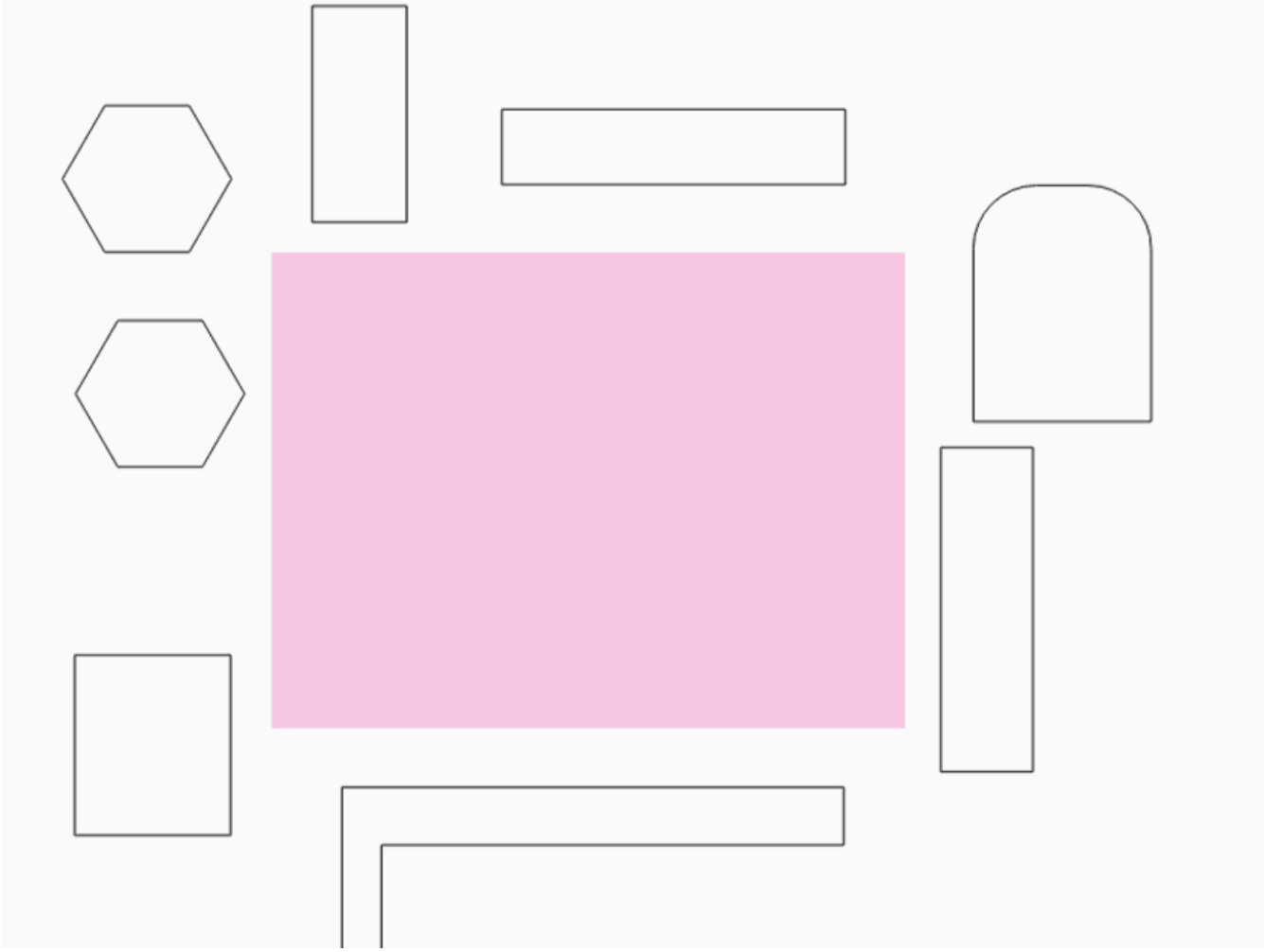


Learn More

SVGNest (External)

If you need denser nesting than **Quick Nest** offers and aren't using LightBurn Pro, you can use **SVGNest** to automatically create and export an SVG of selected graphics, upload it to SVGnest.com, and re-import it after it nests the graphics contained in the file.

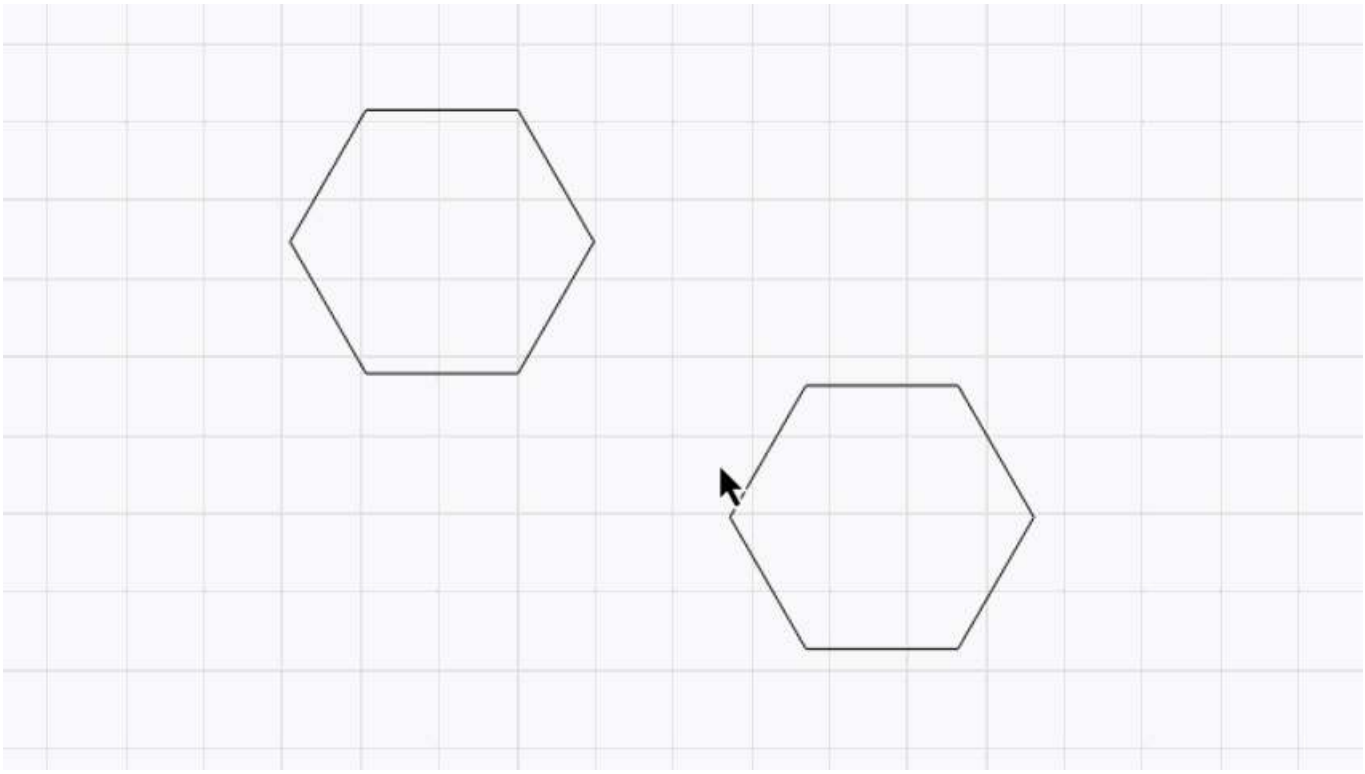
Learn More



Snapping

LightBurn's automatic **Snapping** tools help align objects to other shapes in your Workspace, or ..to the Workspace grid itself — when you move objects close to a valid location, they will snap to that point.

As you hover your cursor over an object, the icon will change to indicate when it is over a node, line, midpoint, center, or intersection. Use these Snapping points when selecting, moving, or creating objects to ensure exact alignment.



Snapping Point	Icon	Description
Node		The cursor is over any node in a vector graphic, or the corner of an image.
Midpoint		The cursor is over the exact halfway point between two nodes.
Center		The cursor is over the center point of an object.
Intersection		The cursor is over the intersection of two lines.
Line		The cursor is over a line, but not a node, midpoint, or intersection on that line.

Learn More

Move Laser to Selection

The **Move Laser to Selection** tools move your laser in its physical work area to a location that corresponds to a position on your current selection in your LightBurn Workspace.

Use these tools to align your laser and physical objects to the positions of graphics in your LightBurn grid.

> **Warning**
>
> In order to use **Move Laser to Selection**, LightBurn must be connected to your laser, and it must be capable of reliably homing and reporting its position accurately. If the position reported by the laser is inaccurate, using this feature may cause your laser to attempt to travel outside of its physical bounds.

Learn More

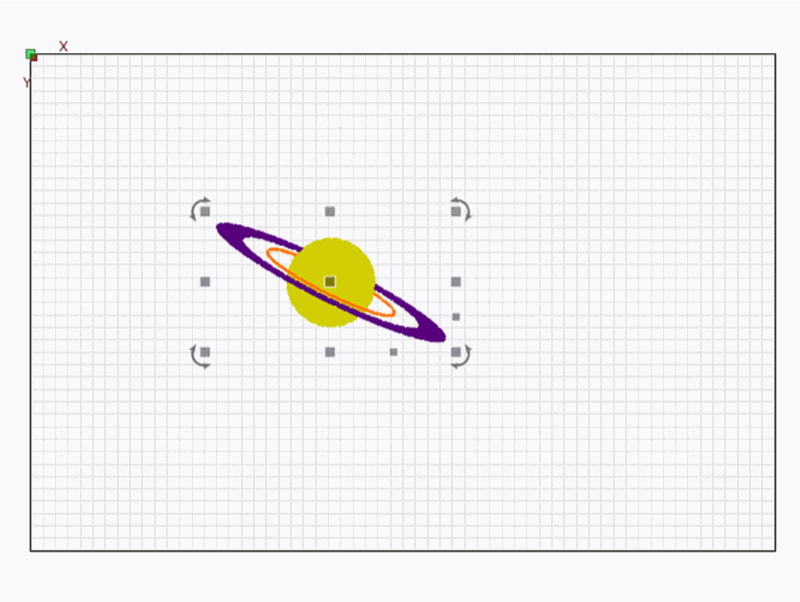


Move Selected Objects

The **Move Selected Objects** tools move objects to a corner, midpoint, or center of your Workspace. If you have a Gantry style laser, it can also move objects to the last reported position of your laser head.

Use these tools to rearrange objects in your Workspace, or to align graphics in LightBurn to the location of material placed in your laser's bed, when using Absolute Coordinates positioning.

Learn More



Remove Overlapping Lines

The **Remove Overlapping Lines** setting of the Optimization Settings window tells your laser to cut lines that are shared by two graphics just once.

The **Distance** field to the right of the switch controls how close two lines need to be in order for the Cut Planner to remove one of them when sending the job to your laser.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=tHTnb6MURdU

Tool Layers

There are two colors/layers at the end of the palette labeled T1 and T2. These refer to Tool layers — special layers that have no cut parameters and will never be output to the laser.

You can use them to describe the boundaries of your material in your LightBurn Workspace, and position graphics within to maximize usage.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=FgCRrQC16xs

Batch Production

Once you've mastered the craft of making one thing really well, you're going to want to make many more of that thing really well. Use these tools to lay out copies of your design, or set your laser up for several runs in succession. If you're a small business owner with orders to fill, these features will help you scale up and expedite your production process.

Grid Array

The **Grid Array** tool creates copies of an object (or objects) in regularly spaced rows and columns, and includes options to adjust spacing, mirror the shapes, randomize orientation, and more.

It's one of the fastest tools you can use to duplicate your designs for batch production.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=WhjJvWE7T1k

Virtual Array

When using the **Grid Array** tool, you can even toggle on an option to create a **Virtual Array**, which allows you to continue modifying your original design, and automatically apply the changes to virtual copies.

You can even adjust your original Grid Array parameters, or add new objects to the array.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=2YeA8xxZ0Kk

Variable Text

You can use **Variable Text** to have LightBurn automatically substitute specially formatted expressions with alternate data when you output a project to your laser. LightBurn can substitute those special expressions for serial numbers, date/time stamps, or even your project's cut settings.

But one of the most powerful options is Merge/CSV formatting, which tells LightBurn to swap in entries from a spreadhsheet, making it extremely handy when you're batch-producing personalized products.

Learn More

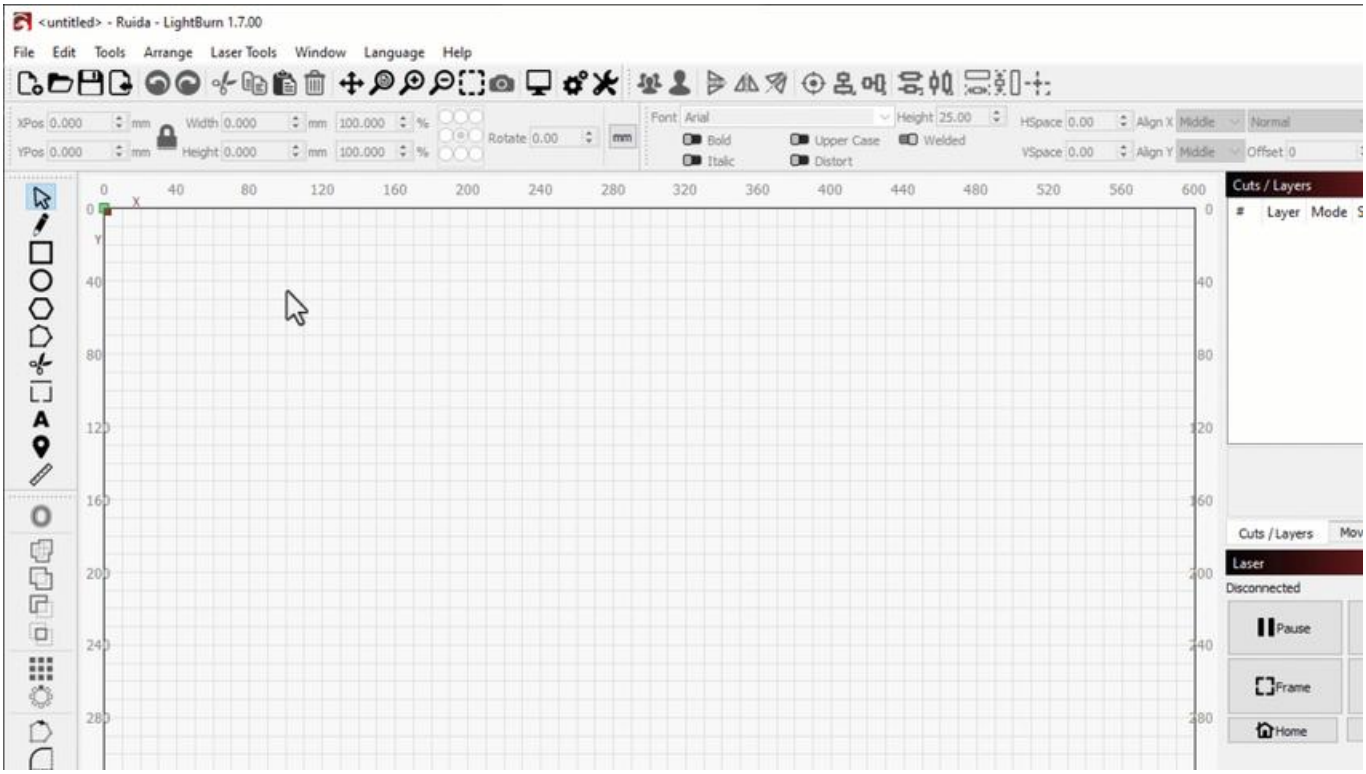
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=5KONOAPuXV8

Barcodes and QR Codes

Create Bar Code generates a Bar Code which stores data that's visible when the Code is scanned. This tool offers a wide range of common Bar Code formats, including QR Codes.

You can even use Bar Codes in combination with Variable Text, so each successive code you engrave stores different data from the last.

Learn More



Repeat Marking

> **Note**
>
> This feature is available for Galvo lasers only.

The **Repeat Marking** tool enables the use of a **Rotating** or **Linear** axis table with Galvo lasers, to move parts past a laser while running repetitions of identical jobs.

By specifying an **Increment** of movement as well as a desired number of repetitions, or **Count**, you can command your laser to mark a design, move an external axis by the specified increment, then run the project again, and repeat that process until it has completed the set number repetitions.

Learn More

Print and Cut

Print and Cut aligns a current project to existing **Target Positions** — also called registration marks — in a previously output design.

Print and Cut works by measuring the difference in location, orientation, and scale between two Target Positions in your laser's physical work area and the positions of two corresponding graphics in your LightBurn Workspace.

You can use Print and Cut to align laser output to registration marks that were specially added to a file in order to use them as Target Positions later. In this case, the original file is intended to be printed by one device, and cut out (or around) by a laser later on.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=ddRW12VKIwY

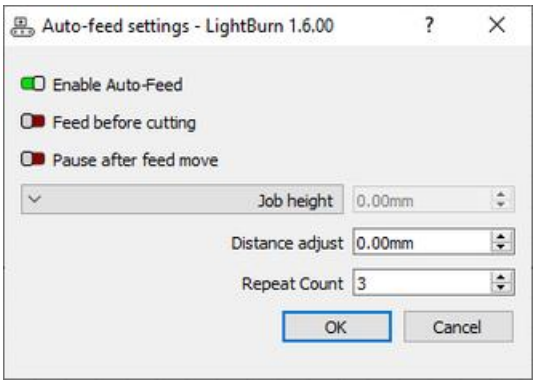
Feeder Setup

Auto-feeders are conveyers that feed materials such as fabric or leather into a laser in stepped increments. The laser runs repetitions of the same engraving or cutting job between each increment.

Go to Laser Tools → **Feeder Setup** (`Ctrl`/`Cmd`+`Shift`+`F`) to open the **Auto-feed settings** window, where you can set the parameters by which your auto-feeder will feed material into your laser, and the laser will run repeat jobs.

> **Note**
>
> Feeder Setup is only available for DSP lasers with Ruida or Trocen controllers.

Learn More



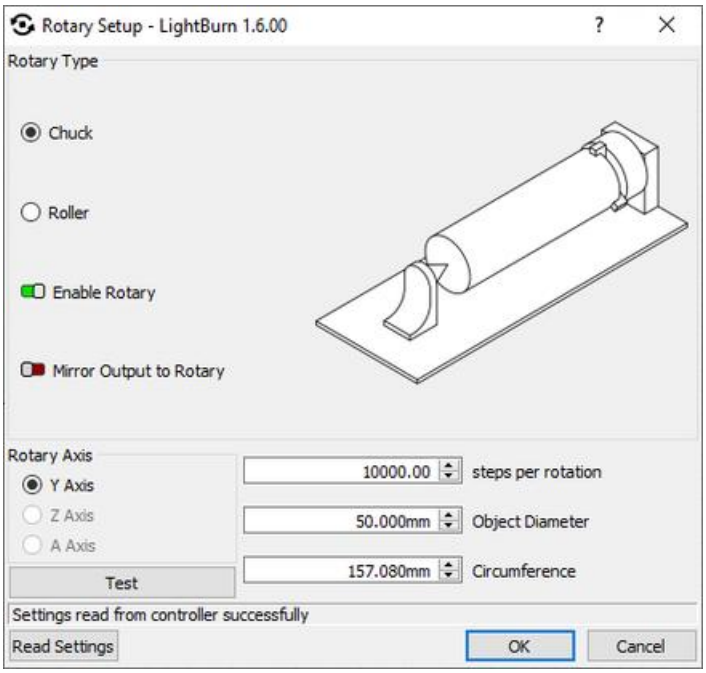
Rotary Mode

Rotaries are used to turn cylindrical objects so that their surfaces can be marked by a laser. In order to use your laser with a rotary, you must first establish several important parameters in the **Rotary Setup** window that control the rotary's motion.

> **Info**
>
> If you're working with a tapered object, use the largest end of the taper for the object diameter.

To open Rotary Setup, go to Laser Tools → **Rotary Setup** (`Ctrl`/`Cmd`+`Shift`+`R`), or click the rotary icon in the Modes Toolbar.

Learn More



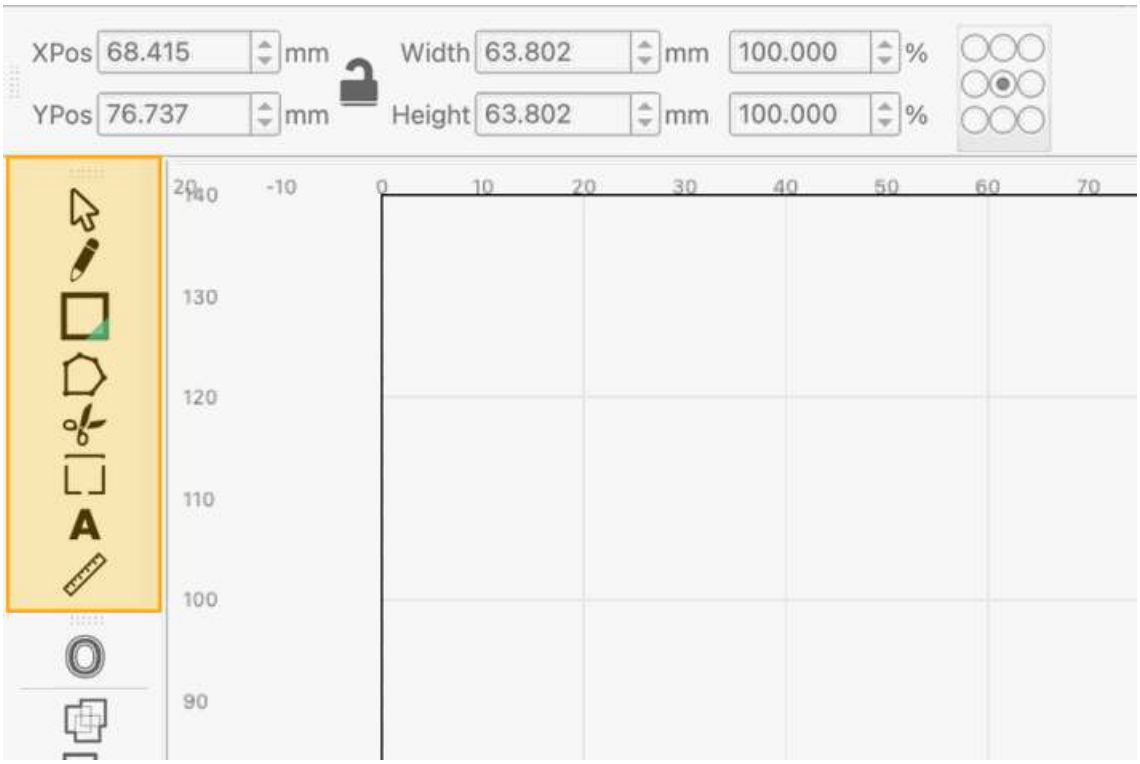
Design Tools

Looking to get creative? LightBurn contains a suite of tools for designing projects from scratch. If you have a background in other design software, many of these tools may be familiar to you — instead of bouncing between programs, you can keep all your laser-design work under one roof.

Creation Toolbar

The **Creation Toolbar** contains all of LightBurn's most essential design tools, including tools to Draw Lines, create Primary Shapes, Create and Edit Text, Edit Nodes, and more.

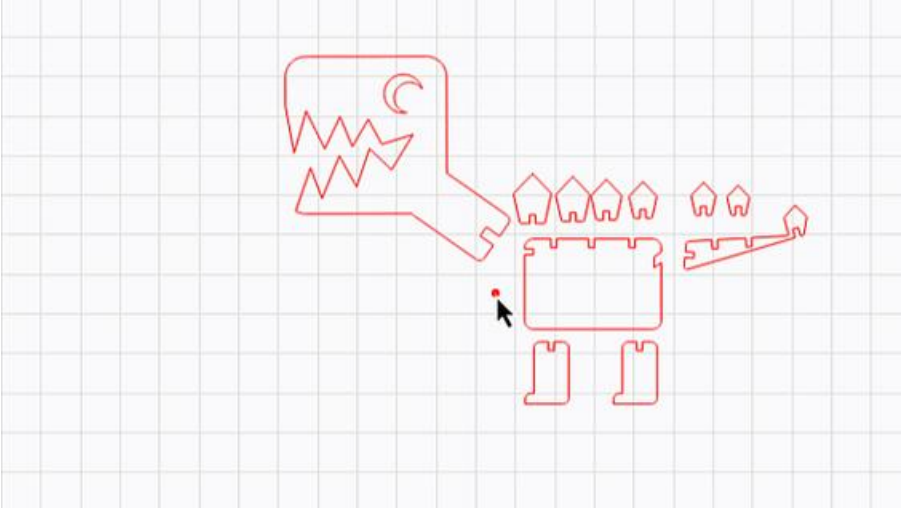
Learn More



Transform Controls

Transform Controls are tools for directly editing selections in your Workspace by clicking and dragging control handles that alter their position, dimensions, orientation, or shape.

Learn More



Shape Properties Window

The **Shape Properties Window** shows editable properties of selected shapes. Those properties vary depending on the object, but include **Power Scale** and **Cut Order Priority** for all objects, as well as **Width** and **Height** for all vectors, number of **Sides** for polygons, **Corner Radius** for rectangles, and **Max Width** for text.

Learn More

Art Library

An **Art Library** provides a way to store artwork so you can quickly access and reuse it in future projects.

You can load multiple Art Libraries at the same time, and separate Libraries are a great way to split artwork up by theme (holidays, birthdays, animals), or purpose (tests, tools, templates, boxes).

Learn More

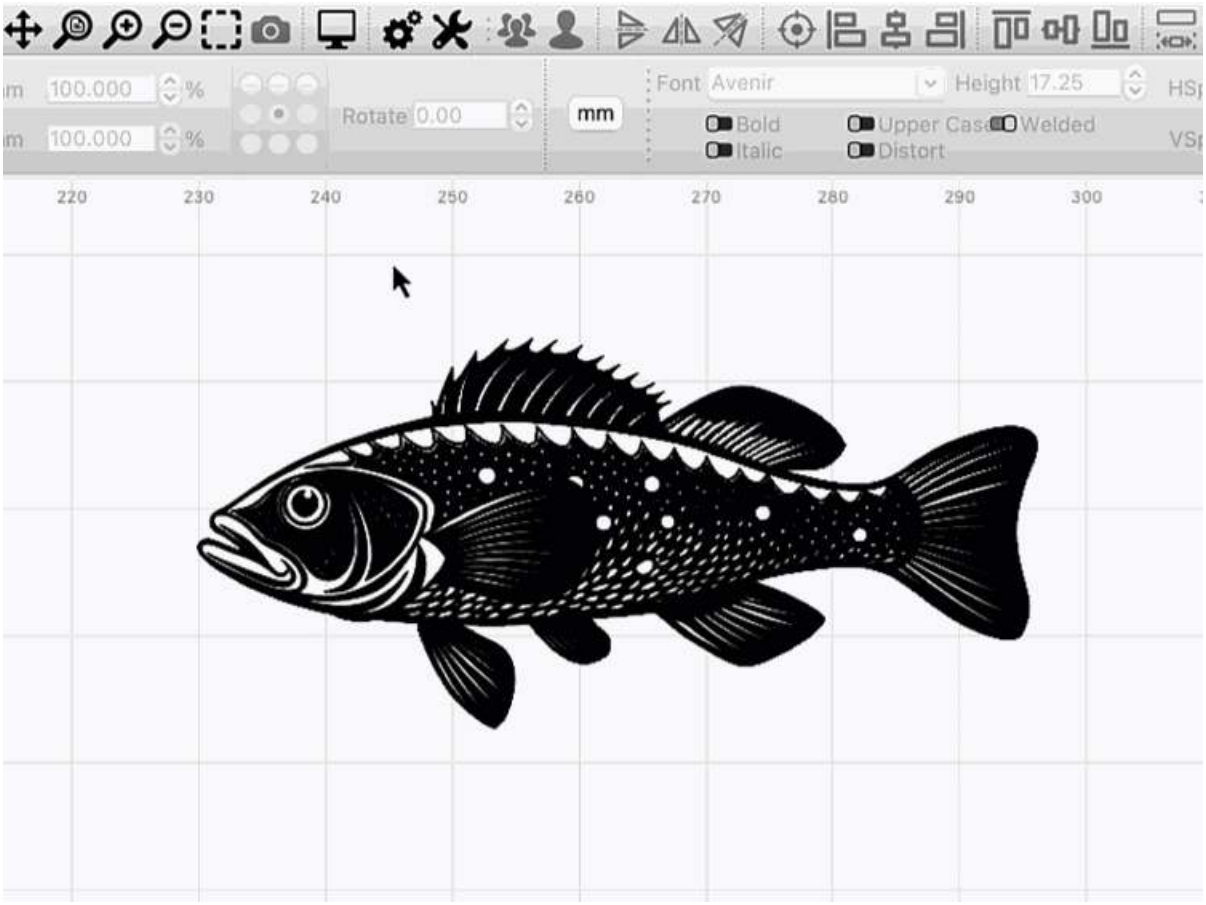
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=30hYVXQ-4CA

Grouping and Ungrouping

The **Group** tool combines collections of objects into single units, and the **Ungroup** tool splits Groups back into the individual objects that make them up.

Normally, all objects are independent of one another, and can be manipulated individually. But it's sometimes helpful to treat a collection of things as a single unit, to make sure they keep their relative position and size when you're moving them around.

Learn More



Lock Shapes

The **Lock Shapes** tool locks selected objects in your design, preventing any form of editing (including moving, resizing, deleting, and more).

Use Lock Shapes to prevent unintentional design edits, secure graphics used for alignment, preserve templates, and restrict editing in collaborative projects.

Learn More



Tool Layers

There are two colors/layers at the end of the palette labeled **T1** and **T2**. These refer to **Tool** layers — special layers that have no cut parameters and will never be output to the laser.

Use Tool layers to create non-output shapes in your designs. For example you can use them to:

- Describe the boundary of your material
- Create guidelines for aligning shapes
- Apply a Path to Text without the path being output
- Apply a Mask to an Image

Learn More

Automatic Guidelines

LightBurn features two types of automatic guidelines to help you align objects relative to one another, or along vertical or horizontal planes.

Alignment Guides are temporary visual guides that indicate when an object's edge or center is aligned to an edge or center of another object in your Workspace.

Draggable Guidelines are persistent lines set to the T1 Tool layer — they are perfectly vertical or horizontal by default, and you can snap other objects to them.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=TOF4luuVVTE

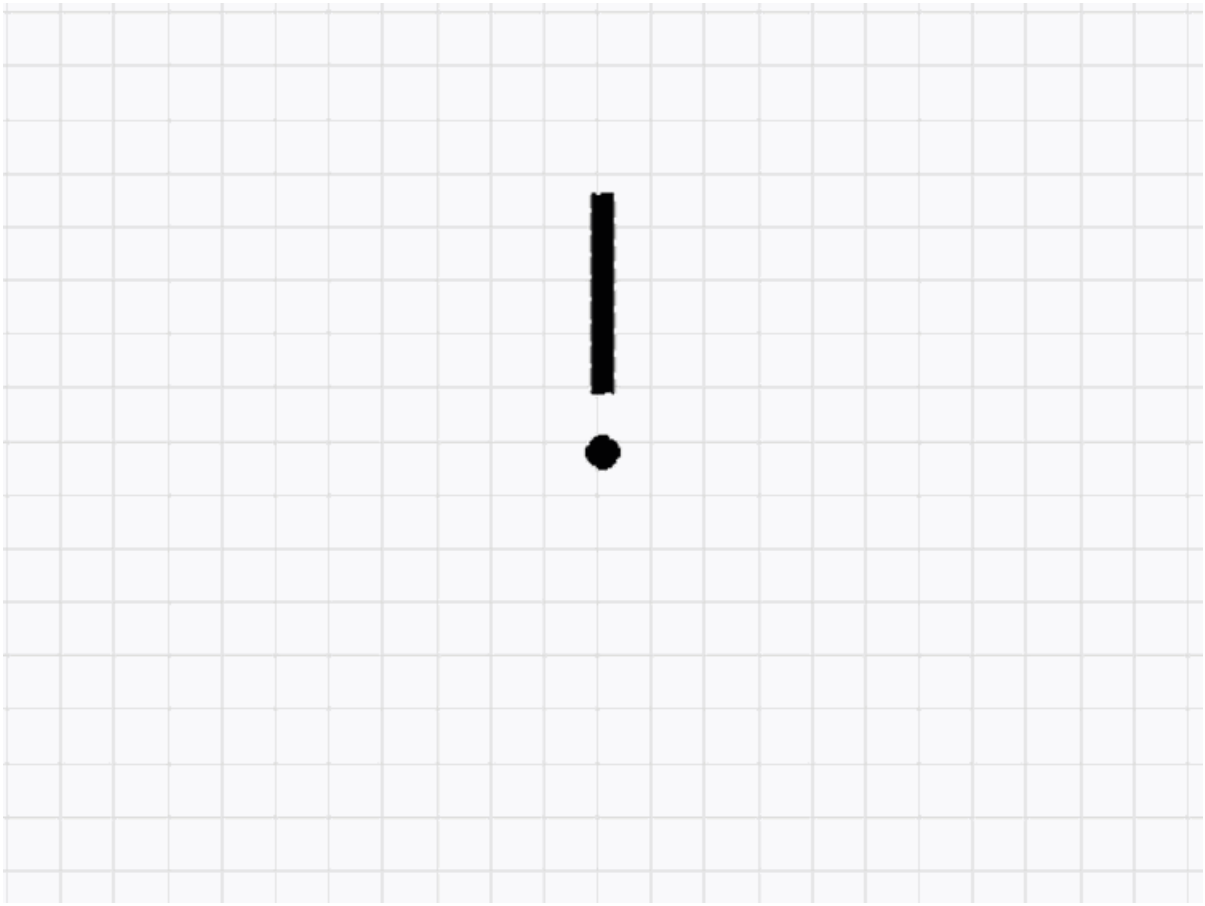
Circular Array

Circular Array arranges multiple copies of an object — or multiple objects — in a radial pattern around either a point in space, or a central object.

Use the Circular Array Tool to:

- Create organic radial patterns such as mandalas and flowers by duplicating simple shapes
- Create even spacing around a circle, such as numerals and markings on a clock face
- Make geometric radiating designs such as starburts or cogs
- Create decorative round frames or borders
- Distribute mounting holes in a circular formation

Learn More

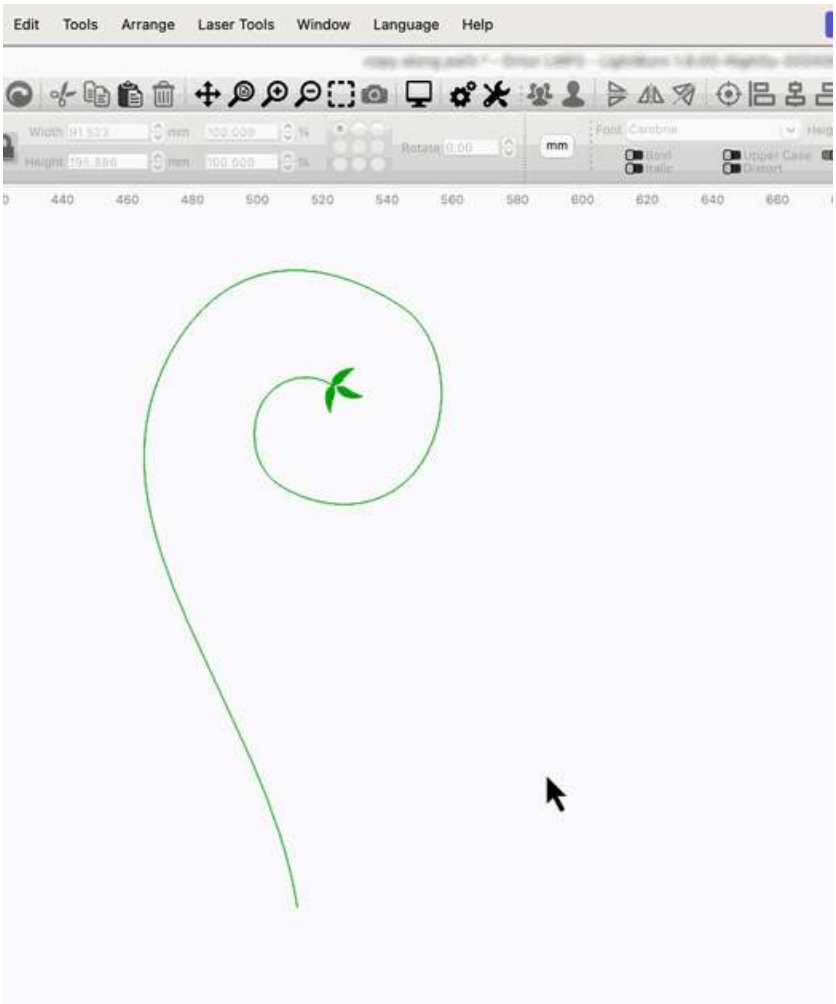


Copy Along Path

Use **Copy Along Path** to create repeated designs that follow a line, curve, or perimeter of a shape, for example:

- Decorative borders along, inside, or outside edges of a shape.
- Repetitive designs that flow along a line such as ropes, vines, and chains.
- Precisely-spaced patterns that follow a vector path, for example measurements on a curved ruler.

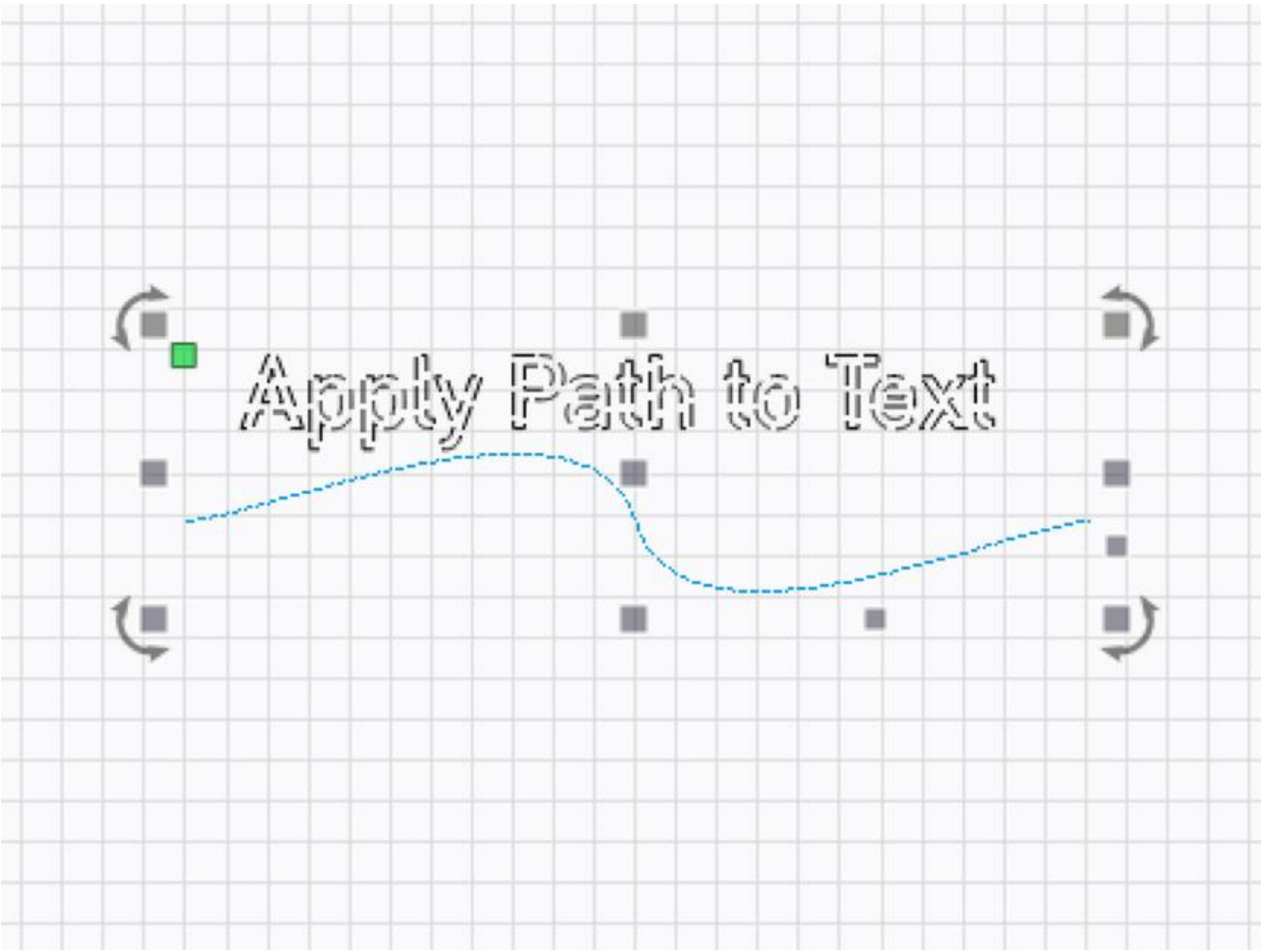
Learn More



Apply Path to Text

Apply Path to Text attaches text to a vector object, reshaping it to follow the contour of the chosen path. This tool makes it possible to use existing or custom-created paths to apply more complex curves to text than can be achieved using the text Bend tool.

Learn More



Creative Uses for Advanced Cut Settings

You can use settings to achieve some neat and speedy effects. We explore some of them in this video!

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=KrAQY9wFv2c

Getting a Design Laser-Ready

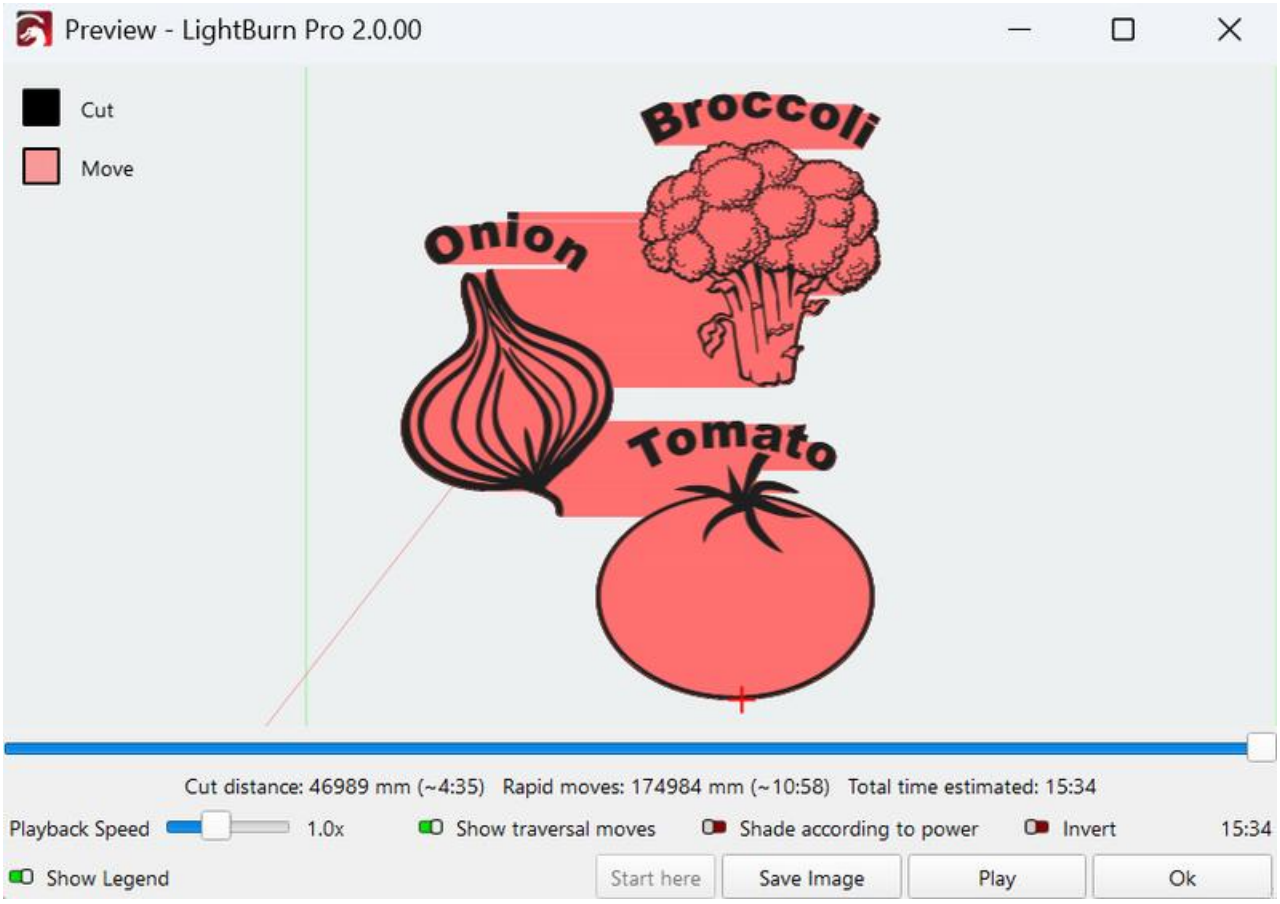
So you've imported a design into LightBurn that wasn't made with laser engraving in mind, and it's going to take some work to get it ready to go. Don't worry — the tools below are up to the job and then some.

Preview

The **Preview** window simulates movements and lasering operations before sending a project to the machine. Because it can be difficult even for experienced users to keep track of which layers are set to which mode, and whether all of their cut settings are in order, we strongly recommend previewing every project, before sending it to your laser.

If what you see here doesn't align with your expectations, you've got some editing to do. Make sure all of your shapes are set to the correct layers, and those layers are set to the correct modes. If you're working with layers set to Fill Mode, make sure they're overlapping only as you intend.

Learn More



View Style

Changing the **View Style** alters the appearance of vector graphics in your Workspace, which can be useful for spotting potential design issues and visualizing the final output from a project.

These options only change the rendering of the objects in your Workspace and do not affect the final work produced by the laser.

Learn More

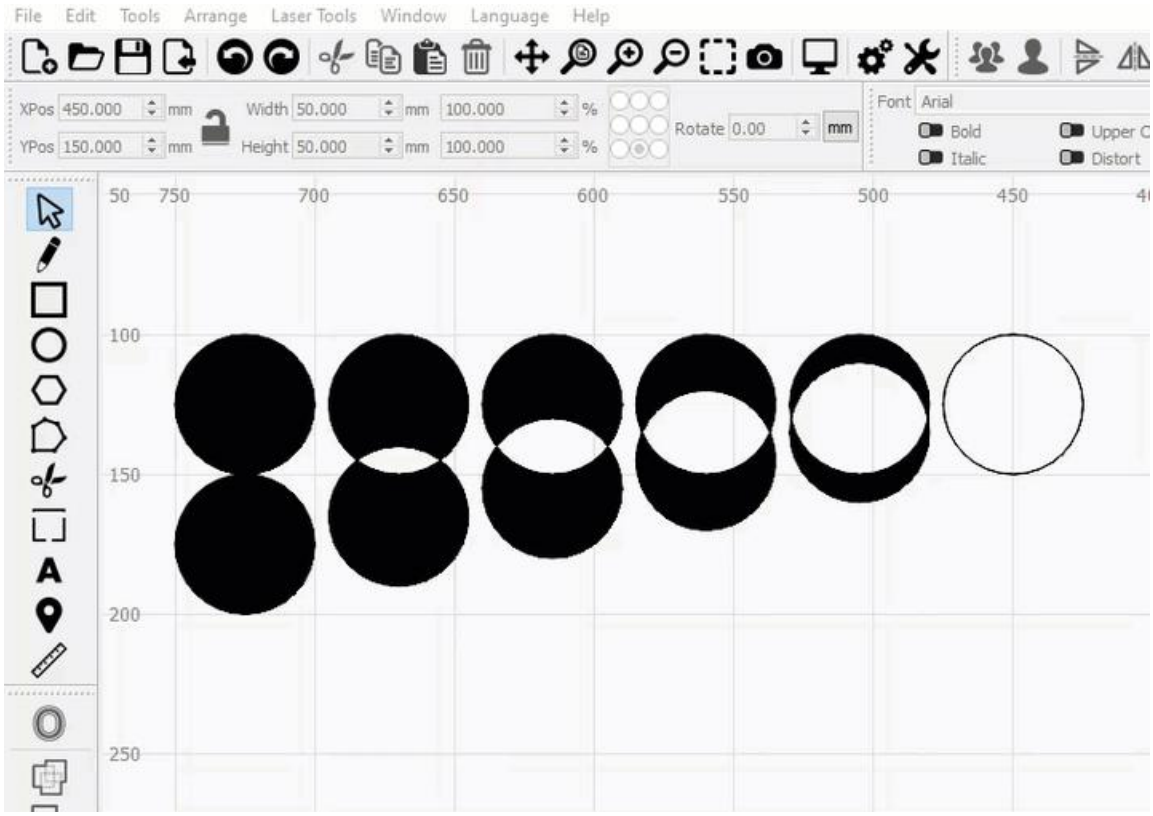


Delete Duplicates

Delete Duplicates finds and removes duplicated objects in your project. Objects don't need to be selected in order to be identified and deleted, but must be Ungrouped and identical in size, shape, and position.

To use Delete Duplicates, go to Edit → **Delete Duplicates** or press `Alt`/`Option`+`D`.

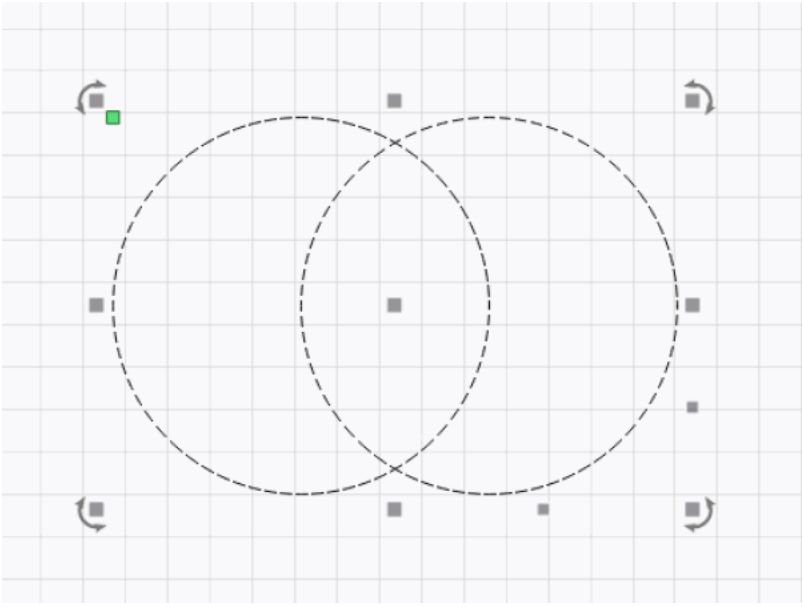
Learn More



Boolean Tools

Use the **Boolean** tools to combine existing shapes (or Grouped shapes) into new or more complex shapes. There are three Boolean options, along with one very similar option, **Weld**.

Learn More



Trim Shapes

Use the **Trim Shapes** tool to cut line segments back to the next intersection with another line.

If a shape does not intersect with any other lines, Trim Shapes deletes the entire shape.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=DGkAJtQxhKM

Edit Nodes

Edit Nodes allows you to make precise, granular edits to vector paths, by controlling the placement and behavior of **nodes** (points in space, sometimes called anchor points) which are linked by straight **lines** or **curves**. The curved lines are described by the length and direction of **handles** (tangents) which appear on one or both nodes at either end of a curve.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=eFmLKBr5Xxg

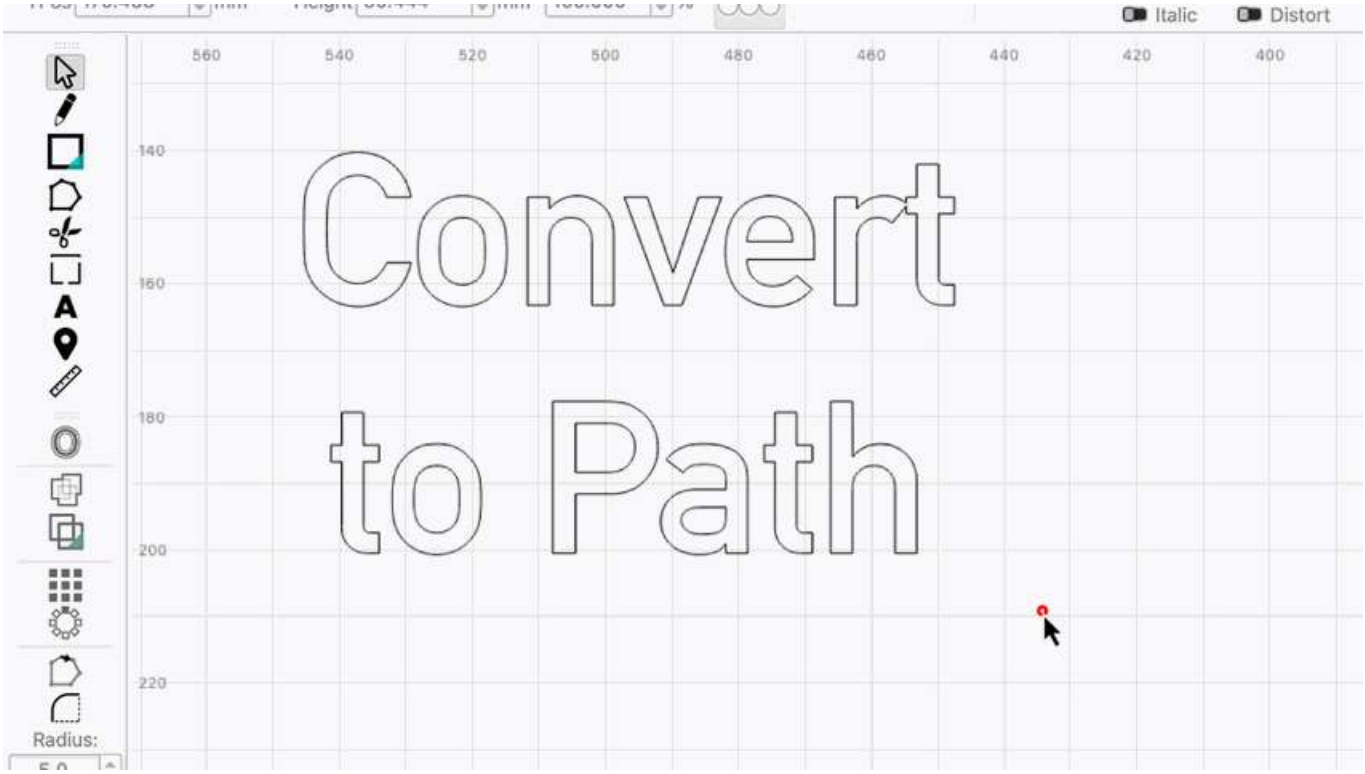
Convert to Path

Convert to Path transforms built-in shapes created in LightBurn into paths made up of lines, curves, and nodes. These paths can be edited using the Edit Nodes tool.

The following built-in shapes must be converted to paths in order to edit their lines, curves, and nodes:

- Primary Shapes
- Text Objects
- Bar Codes

Learn More

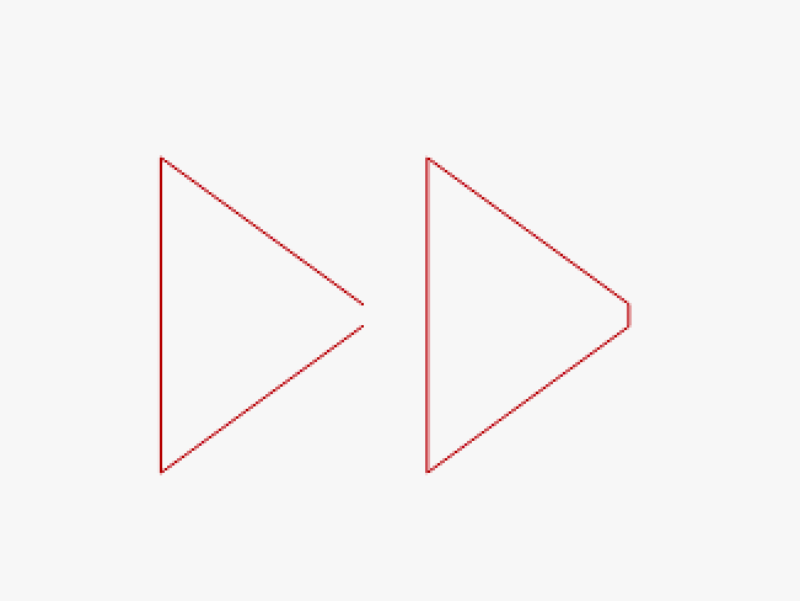


Close Paths and Auto-Join Shapes

Several functions in LightBurn, including Fill Mode, require shapes that are closed loops, with start and end points that are the same. The following tools help to close shapes that are currently open.

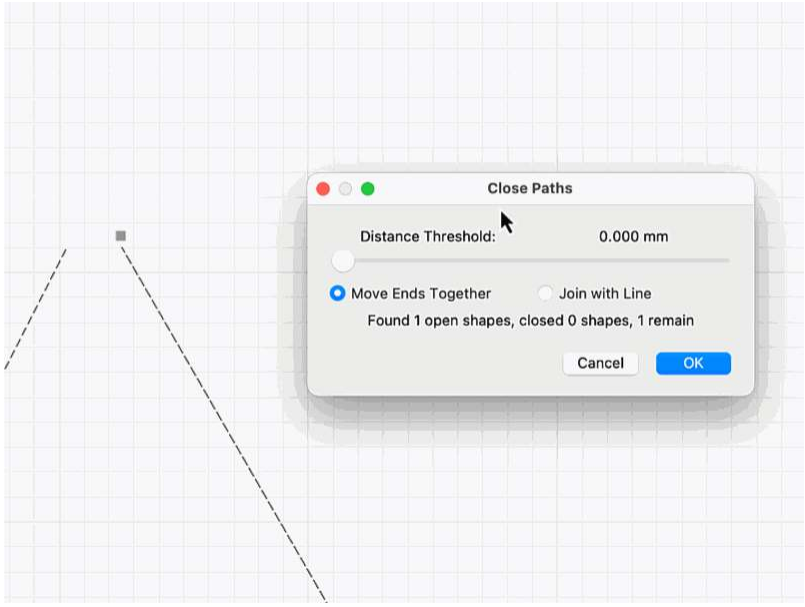
Close Path connects individual shapes with one start and one end point by creating a new line between the points.

Learn More



Close selected paths with tolerance works similarly to Close Path to close shapes with start and end points that are not the same, while providing greater control over how far apart they must be in order to connect them, and how they will be connected.

Learn More

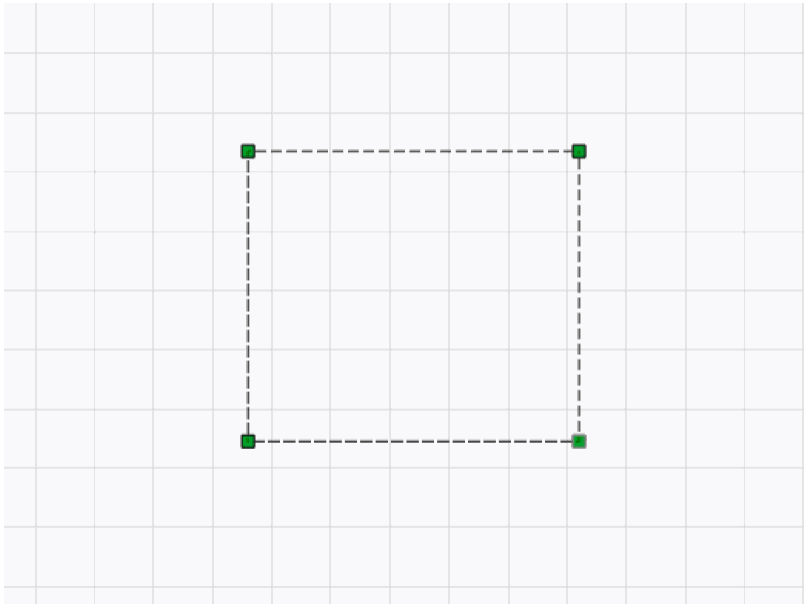


Auto-join selected shapes connects line or curve segments whose start and end nodes are directly on top of (or very near to) one another, but not technically connected. When the segments are connected, they are combined into a single path (a sequence of lines, curves, and nodes).

Auto-Join is most often helpful in cases where you've Imported graphics containing open shapes that you would like to use with a function that requires closed shapes.

To connect start and end nodes using Auto-Join, the nodes must be within .05 mm of one another.

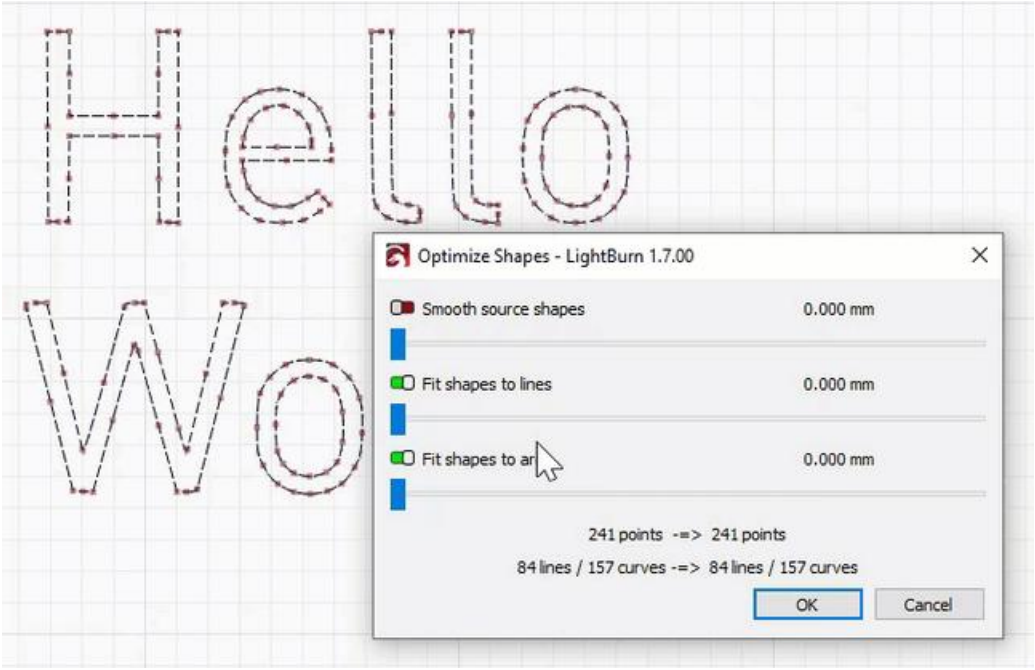
Learn More



Optimize Selected Shapes

The **Optimize Shapes** tool simplifies and smooths selected shapes by reducing the node-count or by fitting them to arcs or lines. This is often useful when importing art created in other software, or cleaning up traced artwork.

Learn More



Resize Slots

Use **Resize Slots** to adjust the dimensions of slots or tabs in selected objects.

You can use Resize Slots to adjust a jointed design that was intended for a material of a different thickness, or to adjust slots or tabs back to their original size after you rescaled an entire design.

This tool works on two main forms of geometry:

- Slots, like those found in 3D puzzles or partitions in trays, somewhat similar to a traditional halved joint or dado.
- Tabs, like a tenon or the fingers in a box joint, usually found in boxes and drawers.

Learn More

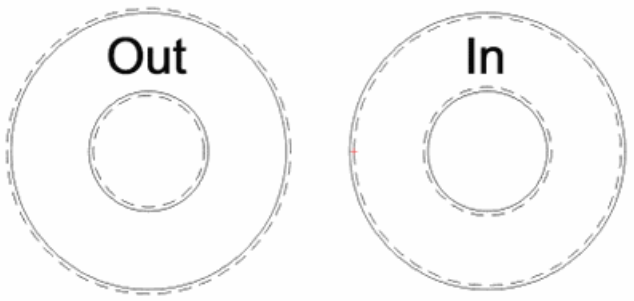
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=0Yep7sYi9Rs

Kerf Offset / Measuring Kerf Width

Kerf refers to the thickness or width of a cut made by a cutting tool. The kerf of a laser is much narrower than that of a saw blade, but the extra material removed can interfere with close-fitting parts such as tabbed boxes or inlays.

Kerf Offset compensates for the extra material removed by offsetting the path of the laser to the outside of closed shapes (for positive values) or the inside (for negative values). This doesn't modify the underlying shape, so you can use your design on lasers with different kerfs by adjusting the Kerf Offset value.

Learn More



Kerf Offset Guide

Follow this guide to learn how to calculate **Kerf Offset** accurately and easily, while using minimal material.

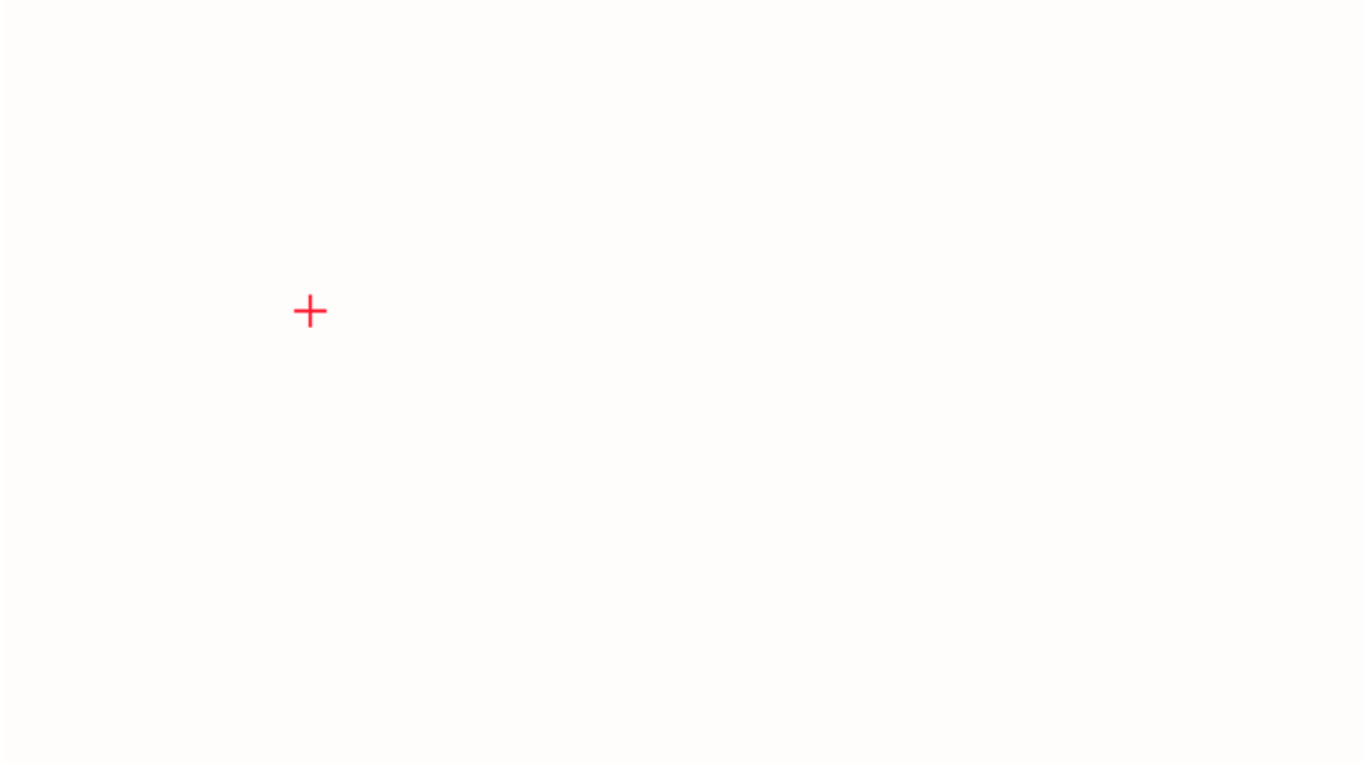
Working With Images

Engraving images is one of the coolest things your laser can do, but it's also one of the hardest processes to get just right. Check out the following pages to learn the basics of image engraving with LightBurn, along with some advanced techniques for when you're a seasoned pro.

Image Mode

Image Mode is a special engraving mode for images, which works similarly to Fill Mode, but allows you to select an additional image processing mode that affects the appearance of the engraving.

Speed, Power, and Line Interval are all crucial settings for achieving great image engravings, but the appropriate settings are heavily dependent on your laser and material.

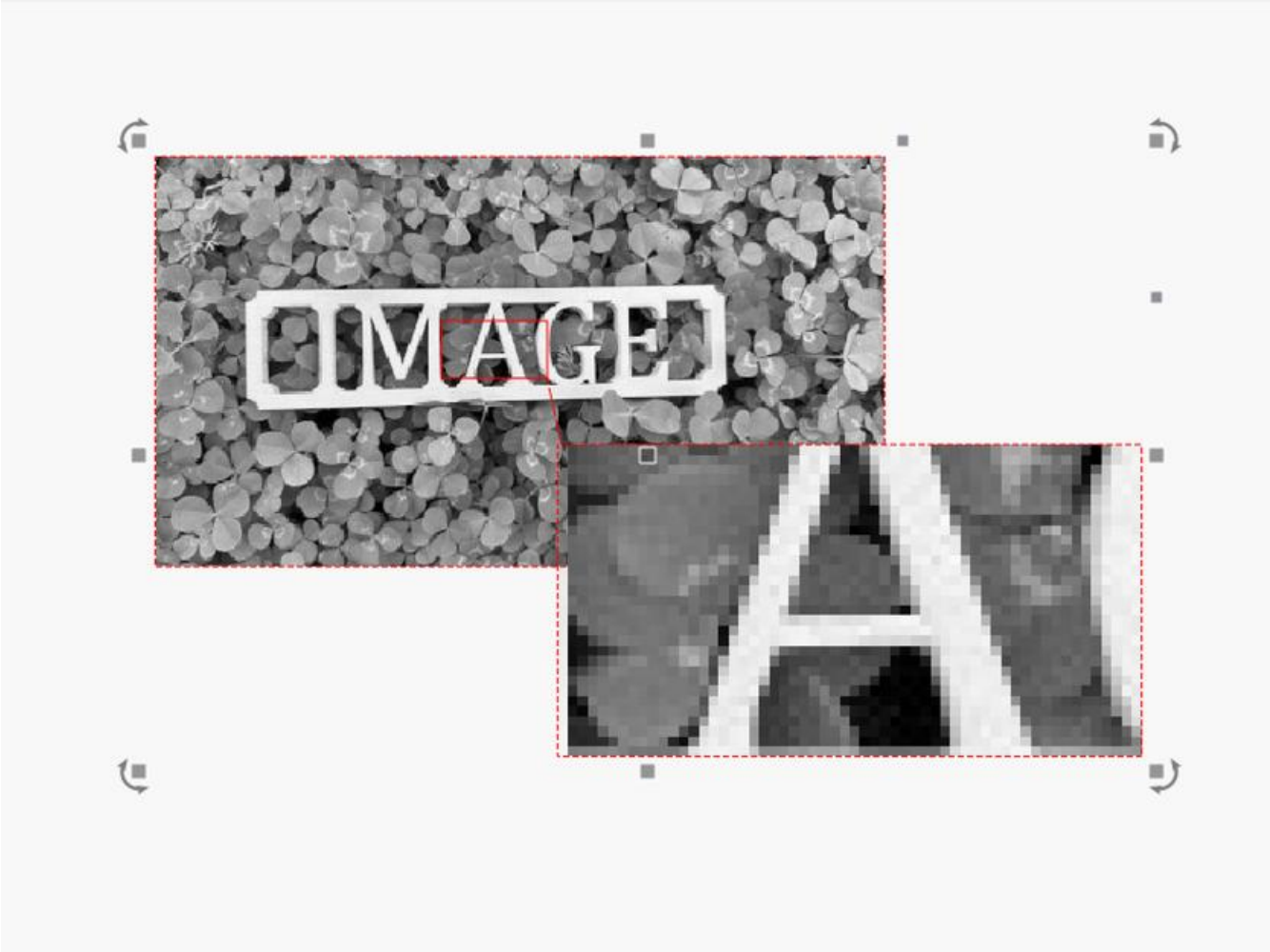


Learn More

Images vs. Vectors

Understanding the difference between **images** and **vectors** is essential for creating and editing custom projects. It influences the tools and methods used in LightBurn, as well as the settings, speed, and quality of jobs.

Learn More

Images	Vectors
	
Images are made up of pixels — tiny blocks of color in a grid. Sometimes called bitmaps or rasters, Images include digital photographs, flatbed scans, digital drawings, and computer-generated renders.	**Vectors** a always be

Five Steps to Perfect Image Engraving

Getting an image to engrave well is a task that often stumps even experienced laser-folk. Often images appear too dark, too light, or "muddy" and lacking detail.

There are many settings that influence the quality of an engraved image, and each one needs to be refined for the overall image to engrave well. In this article we briefly explain the purpose of these settings and, more importantly, how to get them perfect.

Learn More

Line Interval

Line Interval is the distance between scan lines when raster engraving, making it one of the key settings determining the quality of engravings on Image and Fill Layers. It is mathematically tied to the **Lines per Inch** value.

Lower Line Interval values lead to a greater density of lines. The ideal Line Interval will produce lines that just touch, without overlapping.

Increasing Line Interval to the maximum amount that doesn't leave gaps between lines will lead to reduced runtimes for your projects, with no loss in quality, but you may also need to increase Power to achieve dark engravings.

Use the Interval Test or Material Test to find the optimal Line Interval for your laser and material.

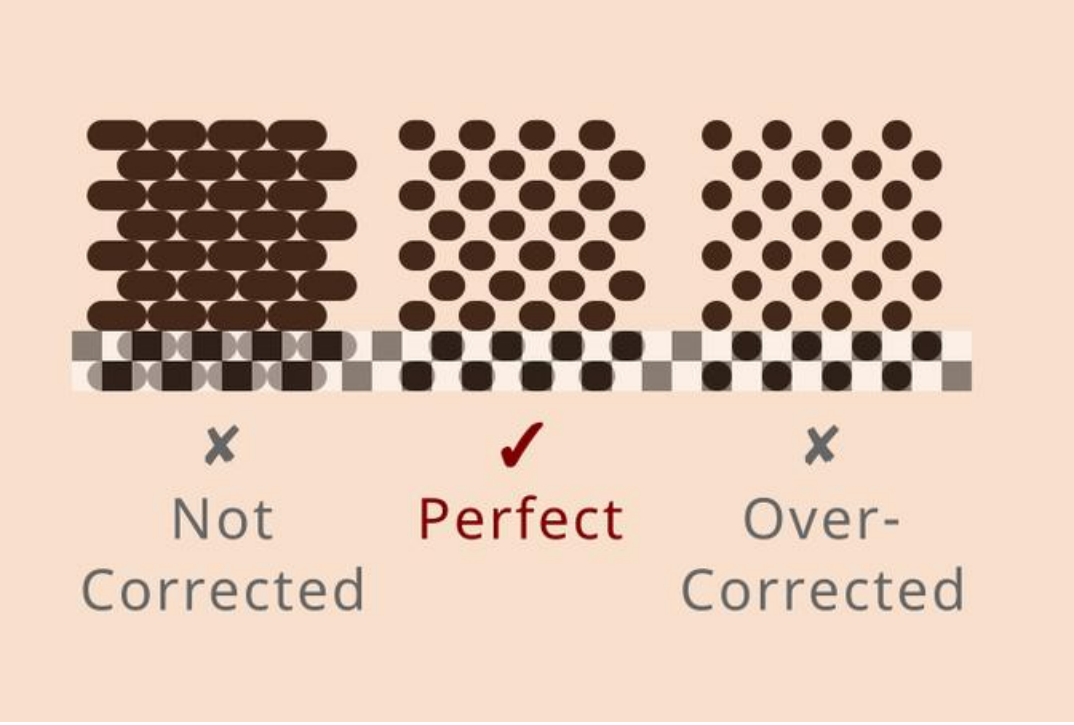
Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=kpMunzp2NM8

Dot Width Correction

Dot Width Correction compensates for the thickness of the laser's beam by shortening the length of engraved scan lines. The ideal Dot Width Correction value is about half the total thickness of the laser's beam. Due to varying affects of power output on different materials, this ideal value can vary depending on the material you're working with.

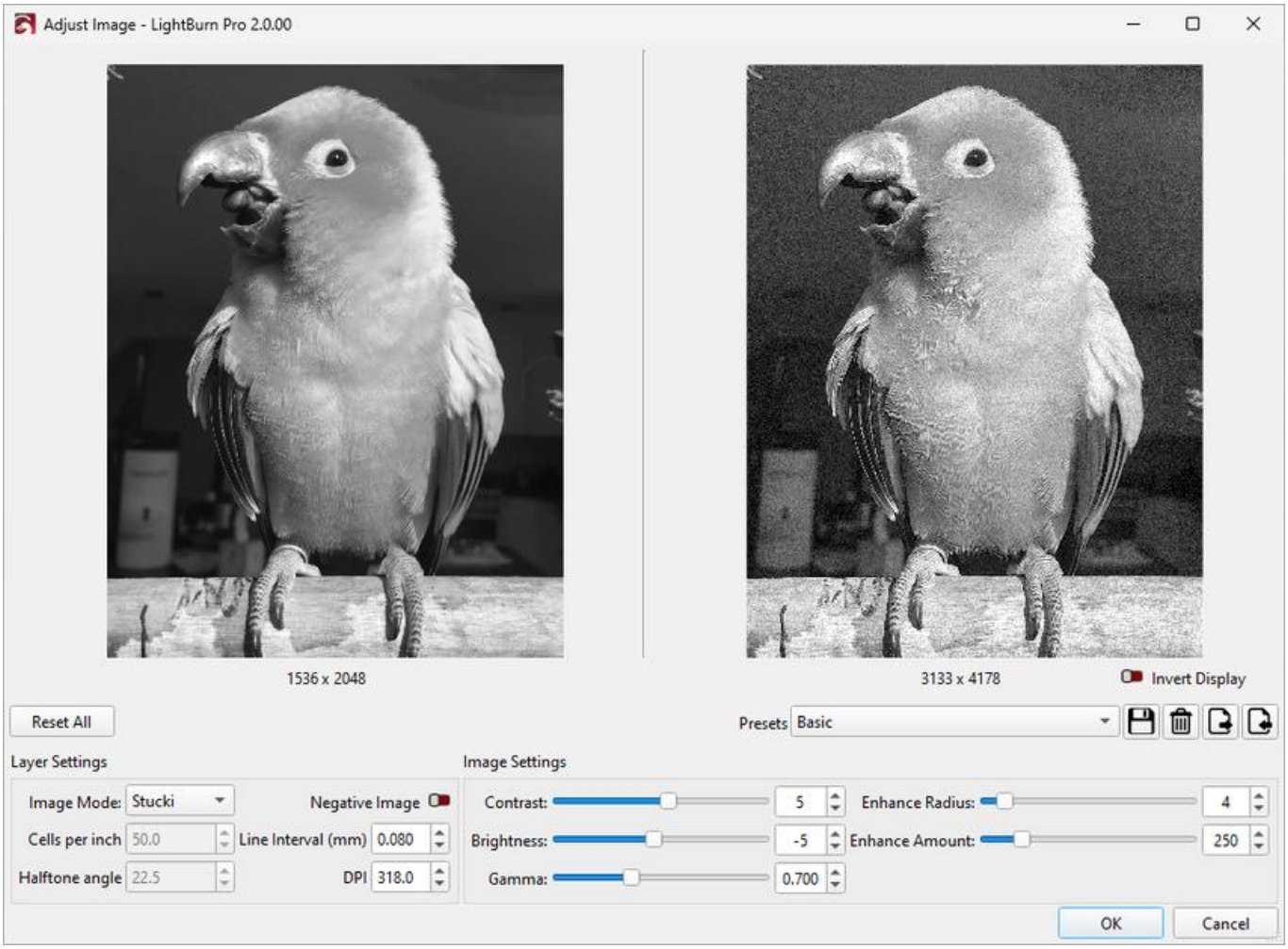
Learn More



Adjust Image

The **Adjust Image** tool streamlines the process of tweaking Image Settings such as **Contrast**, **Brightness**, and **Gamma** while also changing Layer Settings, allowing you to adjust all of these properties from a single window, and see an instant preview of resulting output.

Learn More



Apply Mask to Image

Use **Apply Mask to Image** to hide portions of images that you do not want to engrave, or to divide them into more complex engraving patterns.

By default, image masking doesn't delete the data of the removed part of the image, it just hides it from view and doesn't output those parts to the machine.

Learn More

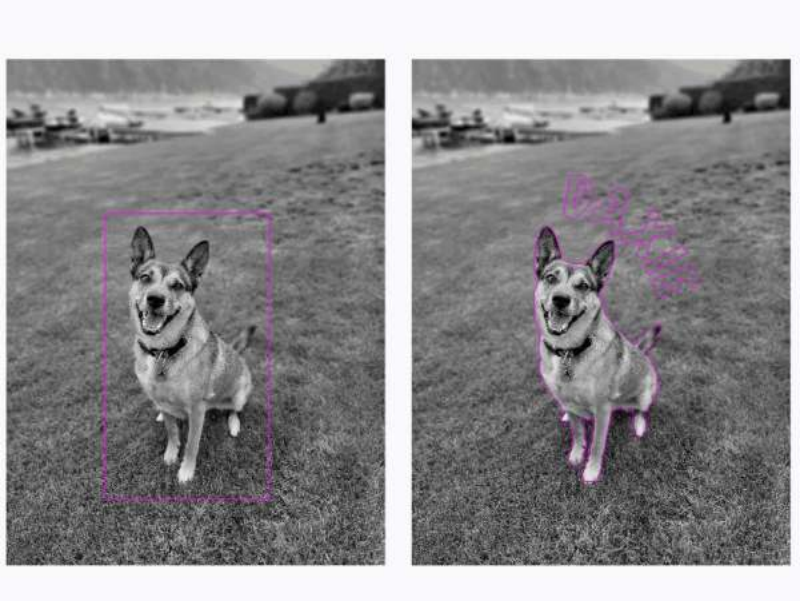


Image Tracing

Use the Trace Image tool to convert imported images into vector graphics.

This is useful if you want to edit an image that has high contrast or well-defined edges, including logos, silhouettes, cartoons, sketches, and handwriting. You can trace photographs as well, but results will vary based on the composition of the photograph.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=OXNKFEl6pjA

Convert to Bitmap

Convert to Bitmap turns selected vector graphics into bitmap images. Once a vector graphic has been converted to a bitmap image, it is set to **Image Mode** and can be engraved using a dithered pattern.

Learn More


Vector
Bitmap

LightBurn for Galvos

If you're new to using LightBurn with your Galvo laser, you might be feeling like there's a lot to learn — and there is! But don't let that worry you, we've got you covered with resources on getting started with LightBurn, learning what you can do with your Galvo laser, and overcoming common stumbling blocks.

> **Note**
>
> If this is your very first time using LightBurn, we recommend taking a look at our beginner pathway for an introduction to LightBurn's interface and features.

Setting Up Your Galvo Laser

These resources will help you connect your Galvo laser to LightBurn, create a device profile, and adjust settings as needed.

Galvo Driver Installation

If you're using a Galvo laser with Windows, you'll need to make sure the correct driver is installed.

If you have an EZCAD-based laser, the easiest way is to check the **Install EzCad2** or **Install EzCad3 Driver** box in the LightBurn installer. If you've already installed LightBurn without installing the driver or if you need to update the driver to support an EZCad3 laser, you can re-run the installer with the driver checked to install or update it.

> **EZCad2 and EZCad3 Driver Support**
>
> LightBurn 2.1 installed a combined EZCad 2 and 3 driver. LightBurn 2.2 has separate drivers for EZCad2 and EZCad 3. If you're using older drivers, please update to the new one using the LightBurn installer.

If your laser uses a **BSL** control board, you may need to install a different driver by installing your laser's stock software, or searching the files provided by your machine's manufacturer for an executable file named `CypressDriverInstaller.exe` or similar. Both the stock software and driver are usually provided with your laser on a USB drive.

> **Not sure if your laser has an EZCAD2, EZCAD3, or BSL board?**
>
> If your laser uses EZCAD2 as its stock software, it has an EZCAD2 control board.
>
> If your laser uses EZCAD3 as its stock software, it has an EZCAD3 control board.
>
> If your laser uses SeaCAD, SimpleCAD, or BSLCAD as its stock software, it most likely has a BSL control board. There are many other variants — consult your laser's manufacturer for further assistance identifying your type of laser.

> If your laser uses different software, you may need to investigate further. Some lasers use a rebranded version of one of the programs listed above, such as the GWeike G2 Pro, which uses GLaser and is a BSL machine. You can see our compatibility matrix for more information, or contact support for help.

Learn More

Galvo Device Profile Setup

When adding a Galvo device, LightBurn will ask you to import the machine-specific configuration file from your EZCAD2 or SEACAD installation drive or an existing installation. If you don't have this file, you can also manually configure your laser with settings provided by the manufacturer, or use LightBurn's built-in Lens Calibration tool.

Learn More

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=YbzSO20bSuA

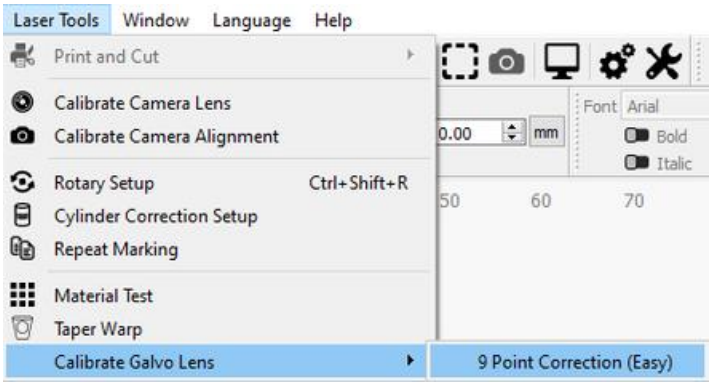
Galvo Lens Correction

All Galvo lenses have distortion which must be corrected in order for output from your laser to have an appearance that's true to the design you see in your LightBurn Workspace.

When working with a new laser, you can usually import lens correction values provided by your machine's manufacturer when you first set up your laser with LightBurn.

If your manufacturer didn't provide the necessary file, or you purchase a new lens for a laser you already own, however, you may need to use LightBurn's **Galvo Lens Calibration** wizard to find the appropriate corrections.

To open the Galvo Lens Calibration wizard, go to Laser Tools → **Calibrate Galvo Lens** → **9 Point Correction (Easy)**.



Learn More

Changing a Lens

To ensure optimal performance following a physical lens swap on a Galvo laser, there are several changes and settings you'll need to confirm. Some manufacturers provide the calibration information for your new lens, and we strongly recommend you check their website before proceeding to perform the calibration manually.

Learn More

Timing Settings

Timing settings correct for the physical limitations of your Galvo laser's lenses. Usually these values are provided by your laser's manufacturer, but you may also need to adjust them manually for perfect results.

Jump Settings control how the laser moves between cuts. Each jump causes the mirror to shake, with higher speeds and longer distances increasing the shaking. If you have wobbly lines at the start of a marking move, you may need to increase the delay and/or reduce the speed of jumps.

Delay settings control the delay between starting (or stopping) mirror movement and firing the laser. **Laser On/Off TC** and **End TC** control the delays for the start and end points of a path. **Polygon TC** controls the delays for other corners within a path.

Learn More

Check out this video guide to timing settings made by our friends at the Laser Everything YouTube channel.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=gFvbrNnvijo

IO Port Settings

IO (input/output) Ports control external switches attached to your laser's control board. They can trigger the beginning of a marking operation, activate your laser's red dot for framing, monitor a safety switch to ensure it's engaged before laser fires, and more.

Learn More

First Project Video Guide

There's no better way to learn how to use your laser than to get started with a beginner project. Follow our step-by-step video guide to complete your first project with your Galvo laser.

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=XYmMc6lE518

Galvo Framing

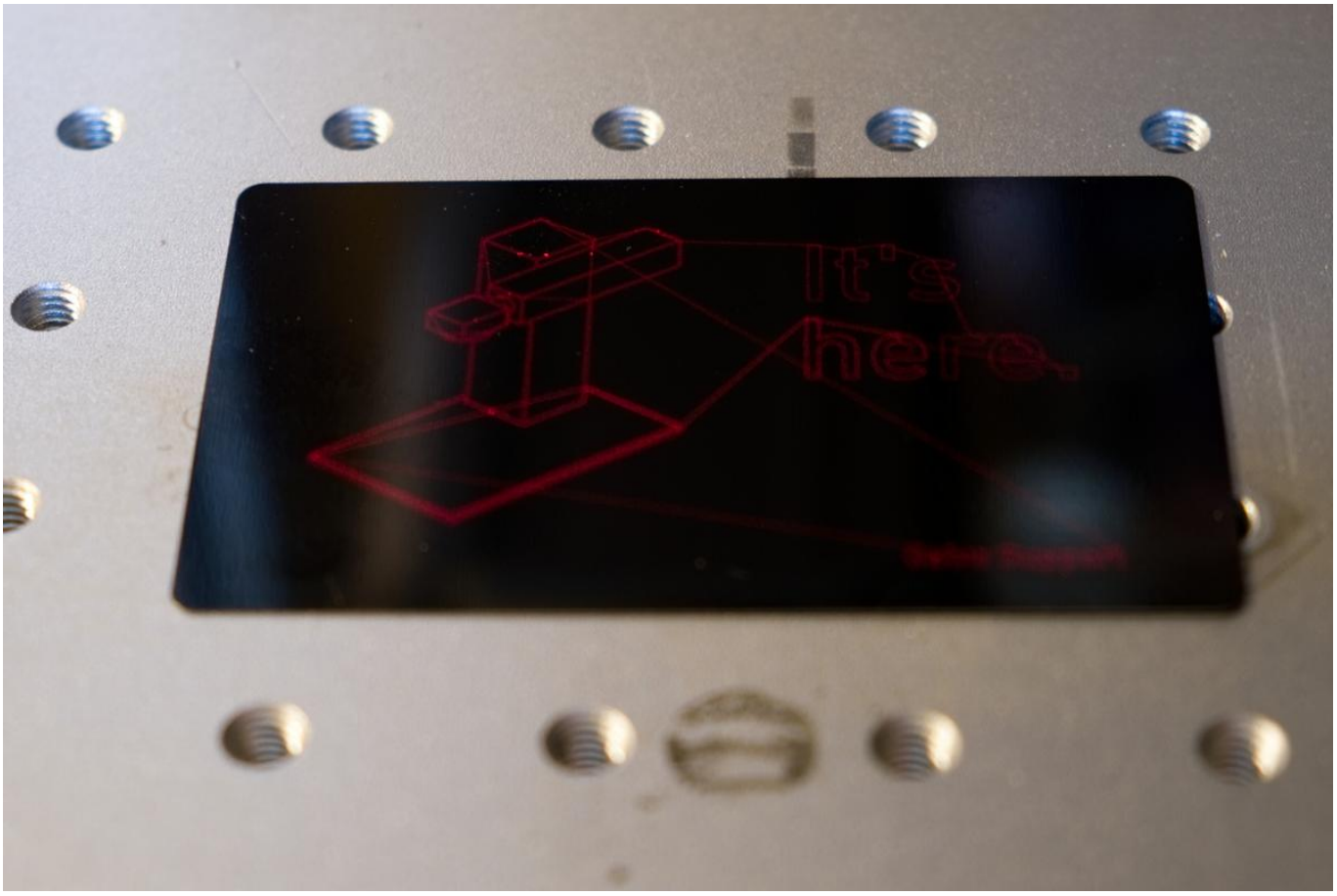
Framing is a function of Galvo lasers that outlines projects in your laser's work area using a red dot light, allowing you to see where your design will output on a physical object.

Click the **Frame** button in the Laser Window or press `F1` to open the **Live Framing** window.

> **Note**
>
> You may need to hold the `Fn` key while pressing `F1` to open the Live Framing window.

By default, the Live Framing window will also open when you press the **Start** button in the Laser Window, but you can disable **Require framing before start** in your Device Settings.

Learn More



Using a Rotary With a Galvo Laser

Rotaries are used to turn cylindrical objects so that their surfaces can be marked by a laser. In order to use your laser with a rotary, you must first establish several important parameters in the **Rotary Setup** window that control the rotary's motion.

> **Info**
>
> If you're working with a tapered object, use the largest end of the taper for the object diameter.

To open Rotary Setup, go to Laser Tools → **Rotary Setup** (`Ctrl`/`Cmd`+`Shift`+`R`), or click the rotary icon in the Modes Toolbar.

Learn More

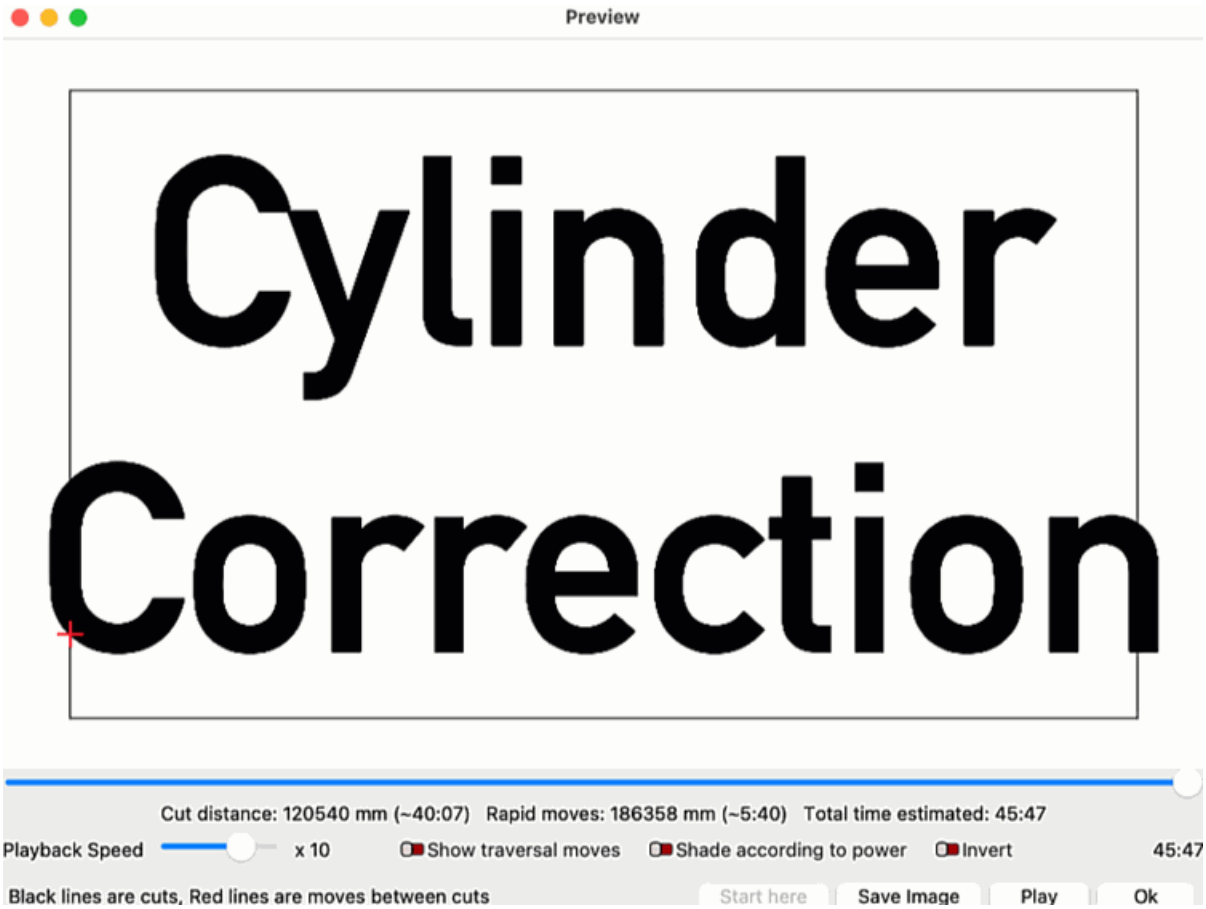
▶ Watch the video on YouTube: https://www.youtube.com/watch?v=St8PWtp17K0

Cylinder Correction

Available for Galvo lasers only, Cylinder Correction is a special mode that warps the output of your design to correct for distortion caused by engraving cylindrical objects, without using a rotary. EZCAD users may be familiar with this tool by the name ProjectMark.

Open the **Cylinder Correction Setup** window by going to Laser Tools → **Cylinder Correction Setup**, or clicking the Cylinder Correction button in the Modes Toolbar.

Learn More



3D Sliced Engraving

LightBurn's 3D Sliced Image Mode enables Galvo users to create engravings with both visual and physical depth. This 2.5D engraving mode does not require a live Z Axis and can produce highly detailed engravings, but does not offer precise control over the engraving's depth.

Learn More



Terminology and Hardware Information

In many fundamental ways, Galvo lasers work differently from other types of lasers LightBurn supports. Below you'll find information on common terminology, as well as the nuts and bolts of how a Galvo laser works.

Glossary of Galvo Terminology

Find definitions for the common terms you'll come across when you read about working with your Galvo laser.

Learn More

EZCAD Hatch Quick Translation

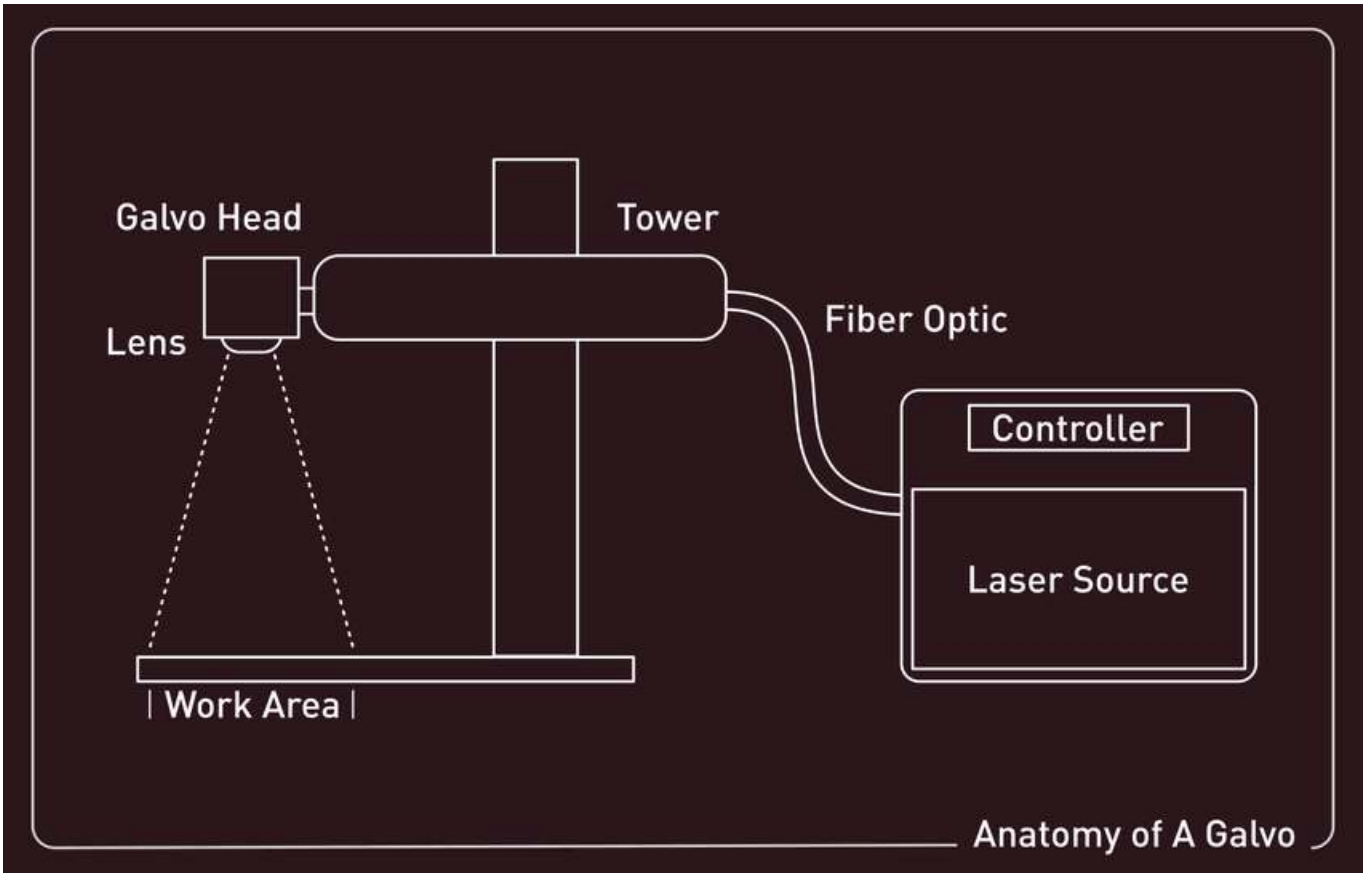
If you're new to LightBurn but not to Galvo lasers, there's a good chance you've previously worked in EZCAD. This page explains how the hatch-related terms you're familiar with from EZCAD translate over to LightBurn.

Learn More

Anatomy of a Galvo

If you've ever wondered how a Galvo laser works, check out this page.

Learn More



Troubleshooting and Frequently Asked Questions

Find answers to common problems and frequently asked questions related to using a Galvo laser with LightBurn.

Learn More

Learn

Learn

LightBurn Learn: Volume 1

Welcome to LightBurn Learn! We'll be featuring creators, LightBurn staff, projects, and other content to inspire your projects and help you do more with LightBurn. With schools starting back up (at least in much of the northern hemisphere) in August and September, we'll be starting off with a release focused on schools and education. Some of the content here is specific to formal schools, but much of it is also relevant to less formal learning spaces, such as makerspaces.

If you're not a teacher, not to worry - we have some general interest content here for you, and future releases will focus on other topics.

Meet the Team Member

For our education-oriented first release, we'll be starting off with our documentation lead Jackie Gerstein, a former high school technology teacher.

Read More





Maker Spotlight

Evan Brand is the Events Coordinator for Hive76, and he joined us to talk about his projects and his involvement with his makerspace.

Read More

Purchasing, Privacy, and Licensing for Schools

We know that schools, especially public schools, often have specific requirements for purchasing software. These vary from school to school, but we've got information for you on how our license works, what data we collect and how we use it, and how to purchase LightBurn. We also have commonly requested forms such as our W-9 available for you to download.

If you haven't been able to purchase LightBurn in the past because you work for an institution that requires the use of purchase orders, I've got some good news - we can accept purchase orders now, with payment by debit/credit card or ACH transfer. You can find more information here.

One thing to note - while we want to support educators of all sorts, and will happily provide information and help you to the best of our ability, as a small company selling software for $99 or $199, we're not equipped to handle custom legal agreements for individual purchases.

Licensing and Purchasing

Frequently asked questions about educational licensing, commonly required documents, and information on our purchasing process.

Read More

LightBurn EULA

A copy of LightBurn's End User License Agreement, available for your convenience.

Read More

Educational and Volume Licensing

Information on volume licensing, along with our quote calculator.

Read More

Laser Safety in the Classroom

Laser cutters are wonderful tools, and with proper supervision and precautions, they're well suited to classrooms. As with any tool, there are some hazards to using lasers, and it's important to know how to manage them. The main risks of a laser are:

- Eye and skin damage (including potentially blindness).
- Hazardous fumes
- Fires

A classroom or other shared workshop setting doesn't add new types of hazards, but it can make them a little more challenging to manage. Working alone in my office, I can run an unenclosed fiber laser because I know I'll keep my safety glasses on - and since I only need a single pair, I have good ones that fit tightly to my face. In a room of 20+ teenagers (who, as every teacher knows, are renowned for their judgment and consistency in following directions), setting up safer systems that rely less on individual making good decisions is always going to be a good choice.

Always use your laser cutter according to manufacturer directions. Include your school district's facilities manager in your planning, and if you work at an institution with a laser safety officer, consult with them as well. In a school setting, the use of laser cutters should always be supervised by a teacher with the appropriate training and certifications, as is the case with all tools.

More about Classroom Safety

Fire Safety

Fires can happen fast, and lasers should never be run without supervision. Click here to see the aftermath of a laser fire and read some fire safety tips.

Read More

Safety Disclaimer

Safety is serious business. Our legal disclaimer regarding safety.

Read More

Classroom Workflow

Getting an entire group of students using the laser in a short time is rough, especially when there's just one teacher! The first time I had to do it, it was a rough week, but I learned a lot and worked out how to get things running more smoothly.

Checklist

There are a lot of little details to remember when laser cutting and even experienced adults forget steps from time to time. Not that I'd ever accidentally start a job with the wrong Start From mode...

LightBurn's built-in Job Checklist can help, but it only comes up when you're getting ready to send the file to the laser.

Back when I was teaching, one of the things that helped me getting entire classes through a project on the laser cutter, CNC router, or other tools was building checklists into their assignment sheets.The checklists helped prevent common mistakes, and provided students with information they needed right when they could act on it, which cut down on repeated questions. My checklists evolved over time, especially as students discovered new and interesting ways to make mistakes. I usually inserted a checkpoint or two where students needed my initials to progress further. On the laser, that might be before sending a job to the laser control computer and before actually running the job.

While everybody's classroom workflow is going to be a little different, I'm sharing a checklist based on how I used to set them up in my own assignment sheets.

Get the Checklist

Template Files

LightBurn's Template Files are an easy way to provide a LightBurn file as a starting point for a project. Template files can't be accidentally overwritten with an ordinary save, and they can contain artwork and pre-configured layers to give all your students a consistent starting point on a project.

Read More

Shared Settings

If you're using more than one computer, sharing LightBurn settings among multiple computers can make things a lot easier for you. Our Shared Settings guide helps with initial setup and ongoing updates to your settings.

Read More

Projects and Other Resources

- Our Laser Engraved Pencils project walks you through creating custom engraved pencils. You'll start by creating a fixture to hold the pencils, then use our Template File to create and repeat your design before engraving it on the pencils. You can use this project as written, or take the methods we used — creating a fixture, using Grid Arrays, and using Variable Text — to customize any number of other things.
- Our Image Engraving on Coasters project is a good starting point to get students working with the laser. The example coasters were made in slate, but you can use whatever material works best for you and fits your budget. This is a good opportunity to learn about the Current Position **Start From** mode and get started working with images.

From our Forum

If you haven't already visited our forum, it's worth checking out. At our forum, you'll find LightBurn and MillMage users and staff sharing their projects and troubleshooting when things go wrong. It's also a great place to get updates about what's coming up in LightBurn and MillMage.

- For those of you with an Epilog laser, LightBurn's 2.2 Early Access build now supports Epilog lasers as a Labs feature.

  Read more on our forum
- LesSables on our forum has an ongoing thread with a variety of MillMage projects, including a watch mechanism clock, a decorative lamp, and a home automation panel.

  Read more on our forum

- This is an older post, but a few years ago we had a good conversation on the forum with a teacher using LightBurn and a Thunder Laser in his classroom.

  Read more on our forum

Maker Spotlight 1

For our first Maker Spotlight, we're happy to introduce Evan Brand. You may have seen him around at a variety of maker events; he's hard to miss in his denim jacket with a light-up or spinning back patch.



Could you tell me a little about yourself? Who are you, and what do you do?

My name's Evan. I'm the events coordinator for Hive 76, a community-run makerspace here in Philadelphia. I help out with the space for both on-site and off-site events, and events that we partner with to be able to do different activities and try to help build some good community relations with other spaces both here in Philadelphia and out in the wider world.

How did you get involved with Hive 76?

Actually it was a bit of happenstance. I ended up during the pandemic finding a video talking about makerspaces from the YouTube channel Strange Parts and that led me down the rabbit hole of looking into them in my loca area which led me to Hive 76.

So you've been involved there for about five or six years, right?

Yeah, and one of those situations where we had enough ideas about classes that I'd want to either teach or have someone else teach or events that we should go to, the previous events organizer for the space just kind of turned to me one day and was like "I've been planning to retire for a year or two now. Do you want to just do this?"

Had you had any related experience before you got started as the events coordinator here?

I've had a little bit of communication experience when it came to topics more from a business side. I'd had a degree in accounting and got my MBA during the pandemic. So I had a reasonable amount of ability to take a topic and pitch it, but not necessarily in the STEM fields. I only got into this about seven years ago, a little bit before the pandemic when I got into some 3D printing.

You mentioned you started with 3D printing, which ties in nicely with the next thing I wanted to ask you - what got you into making, and what types of projects have you done?

So I did kind of start with that 3D printing, which I'd started doing some D&D miniatures and terrain with, but the more I got into it the more I kind of was fascinated by the technology, which led to researching the community, which led to looking at all of these cool projects. At a certain point, if you're that kind of person who gets inspired to do things that kind of spark just lights up and for me it was just like "I wonder if I could do _____" and I've been running with it ever since.

That's really cool. What's your favorite tool that isn't a laser cutter, CNC router, 3D printer, or other type of digital fabrication tool?

By functionality, it's a hot glue gun. The number of times in which I've solved a problem just by having one on hand. For anyone watching this, the biggest hack that you'll ever need to know is that hot glue comes in every color under the sun. Get some black hot glue or color match it to your project if you don't want somethign showing. You'd be surprised about how many problems having a few different colored sticks of hot glue can solve.



Nice. That's a good one, and I wasn't expecting it. We make heavy use of hot glue in combat robotics too, they're great for securing connectors and protecting electronics.

Yeah, the amount of times that I've kept something from short circuiting by just having someone hold the two wires apart while one of you shoot the hot glue in and hope... it's like "Hey, as long as this stays on one this cools, we're good."

Do you have anything that you think really should have had a bit more recognition than it got?

So for me, there are two projects in particular.

I like doing logistics and experiential stuff. So, the biggest one that I've gotten pointed at and someone says "You're the guy from this", was I run an ARG (augmented reality game) at PAX Unplugged and a few other events where I ahve a full walking fantasy shop, including shelves and a desk and a questboard on the back that I have some custom quests to send people around the event that I partnered up with exhibitors for. I custom laser etched some coins on the space's fiber laser and then would hand those out as rewards to then let people buy stuff from the shop. So that's probably the most photogenic one that I've seen people spot me with, especially because I've had a few people think I was actually a booth until I started moving. It's always a great reaction.



The second is actually a group community project that I started up this past year called The Repair Zone. I'd been running cosplay repair and other repair stuff at various events, but the last three years at the convention Open Sauce I'd run a mobile repair zone. The first year was just me and one person who asked to help. Second year there were a bunch of different exhibitors who did it without communicating with each other and I kind of organized this on the discord and created a ticketing system. Going into year three of doing this I'd reached out to the organization to actually have us do this as a partnership with the event so that it would be a bit more official and we'd have a bit more communication back and forth. So we had our own 12 person team going the entire time. 750 repairs I believe over the course of three days.



Outside of projects, a cool thing that I've made - I love playing those augmented reality game situations where I can bring these resources in and help other people who make cool projects. I'm pretty good at that pre-planning admin stuff and very proud about how the team and I were able to get things together this year. It's a great time.

Having benefited from your repair zone at Open Sauce, I very much appreciate that project. And the booth is very cool.

The best time to get a chance to talk to someone amazing is while they're word vomiting while you're helping them fix up a project. It's the thing they care about most in the world and you're there with the one resistor they needed to keep it up and running. I have some good stories with it.



Do you have one repair that you'd love to share right now?

There's a wide variety of repairs that have been fun for me. We were able to fix two people's wheelchairs at the event. We actually did data recovery for some creators who otherwise would have lost their footage from the event. The Animatronic Awesome was one that I did find quite entertaining that we ended up facing up. But other than that just fun stuff like we had our friend Sasha live 3D modeling and printing stuff out on some Prusa Core Ones. So during the event we were actually able to look at a problem, live model a fix for it, and have it to them within an hour, which is incredible. I think next year we do need some laser cutters, but that's already in the books.

Yeah, laser cutters are great for that if you don't need full 3D or if you can slot things together. I love how fast they can be.

It's amazing.

What are some things you've used a laser cutter for? I know you've mentioned the coins already.

The coins, I've done a lot of fun signage stuff, pulled in some woodworking techniques. You can actually do a resin inlay if you laser deep enough into a thing. Makes it easier to do cool signs. I've done some maps for some tabletop games and it's just a useful tool, especially once you get used to it. I've used it for the coins and being able to just throw an image in the osftware and just hit go is

really useful. Especially since my favorite thing for using a laser cutter in 3D printers is to turn something around in an an unrealistic amount of time. A joke is funny, and actualizing a joke in less than a day and getting something to someone an hour after a conversation happens - that look of amazement and a little bit of fear in someone's eyes of "How did you do that?". It's great.

It really is. I've had those moments where somebody says "It'd be really cool to have this" and then you hand it to them and they're just shocked.

My business cards for Hive have the full poster for Hive76 on one side and my info on the other side. I have people ask me all the time "I could just take a photo, do you need this back?" and I say no, that's just the card. And then I get the baffled look of "This is a metal business card?!". I make them myself at home. They're cheaper than printed cards.

I love that speed. 3D printing does some really cool stuff, but sometimes having something right away makes such a difference.

Have you played around with CNC routers at all?

Yes. I need to do more.

We have a large one at Hive76 and it is lovely and it scares me. It's one of those things where it uses VCarve and a few other softwares that need to be kind of jumbled otgether to run the thing, and has a multi-step process to get it up and going just because it is a really good machine, but it's on the more industrial side than the prosumer side. So it's the kind of thing that I have a respectful appreciation for. I've seen it do amazing things, and I've worked with people to do amazing things on them, but it's the kind of thing where it's not like the 3D printer where you walk away and it's fine. It's the kind of thing where you sit in the corner, and you make sure that it's well enclosed and no one is walking around it when you have a router plunging through wood or aluminum.

Yeah, they're cool, though, and they let you work with other materials, like metal. Something you might be interested in - we actually have new software that came out recently for CNC routers. So since you already use LightBurn, MillMage uses that same interface. And one of my favorite things is that if your router has a compatible controller, you can actually directly control it from inside of MillMage instead of going from program to program. So you can iterate fast.

That genuinely would be amazing. And this is something I've been trying to update our machiens for because having everything in an ecosystem is lovely, especially the fact that LightBurn is machine agnostic so it's not like I have to get one laser and then if I switch brands everything is ruined. So being able to have one thing you train people on that follows the same design language instead of each tool having its own traing would make things infinitely easier to get people up and going with services.

Yeah, that was part of what we were aiming for. And being able to iterate without having to go through the export and import step over and over makes things so much easier.

What are some of your favorite skills or techniques that you've picked up along the way?

I would say a few of the fun ones I'd mentioned before, like that wood inlay stuff where it's either different pieces of wood or doing a resin pour to not have to carefully pour on top of and sand away a laser project. I'm also personally a huge fan of adding LEDs to things. Anything can be fun, but if you make it light up or move, the most basic thing becomes infinitely more interesting to folks. So

using stuff like WLED, or even just embedding some LEDs with coin cells and on/off switches and different stuff. I've been able to do fun things like spinning back jacket patches for a denim jacket or actual exclamation points and question marks that light up like a video game that I've used for a lop party that I was helping some friends run. The new patch on the back now actually spins.

Yeah, I remember your jacket. It spins now?

Yeah, and I made the back of the jacket modular. So I can swap out the patch, which means that this project jacket can do the one thing that everyone who's ever made a wearable wishes they could do, which is throw it in the washing machine. So that was the goal - if I'm bringing this to enough things, I want to actually enjoy wearing it to all the things and then spat it out if I have new ideas instead of having to have 10 different jackets and get patches for all of them because you have 10 different ideas.



I will say this, and this goes to anyone who's working on projects too. Look up to see if there's a standard for a lot of things. Even if you didn't think there, someone may have made a standard that's already existent. If not, make one yourself. Doesn't matter what it is, even if it's arbitrary. It's just making sure you can have something that you can consistently go back to and design around. It will make your life so much easier to just be able to import this design file on top of your project and it already has the mounting holes attached or whatever way you'd want to go about things.

At Hive76, how do you get people started using the laser or other equipment tha trequires training? What sort of pitfalls have you found? And do you have any advice for other makerspaces?

This goes in a few directions. Weirdly, the lasers are one of the easiest things to train people on. They deserve and require respect, more so than the 3D printers do. So it's largely making sure people are aware that you can't just walk away from them while they're in u se, but they're fully enclosed machines and we have the software preloaded on the computer next to them. So instead of

having to be a whole class, it's usually a 15-20 minute one-on-one convo with someone who's already been trained and then running some demo project to make sure it sticks in their mind, and people can always ask if they have questions. They're relatively simple for starting out.

It gets tricker with things in the woodshop, whether it be saws or especially the CNC machine when it has that multi-step process and actual risk to the individual. Making sure that it's someone who's trained up properly, not just so that we can sign a check mark, but making sure they know what they're doing safely. That takes expertise and it takes a dedicated class which does slow down training of individuals occasionally. Especially for the CNC, we have one person who is a trained operator and uses them for their day job and he's the one that teaches our classes. So it is a pain point in if this person is busy or had already taught a class recently. They usually do it bi-monthly, so if a person misses that it becomes a whole ordeal of whether you take a day out of our trainer's life to teach this one person, or do they have to wait a month or two until the next class comes up?

So my advice would be that if there are tools that are within a reasonable spec to have trusted individuals go through a more intensive training so you can have several individuals who have enough expertise to teach others how to use it. Because there are certain tools that can be used but require a higher level of competency to actually train someone. and as lovely and hard working as any individual is, burnout exists, and it needs to be respected for long-term health of the space.

And second, if anything relies on one individual, it's fragile even if they're a superstar. So having other people who understand how to do the things means that if that person leaves, whether by their own choice or because of life circumstances, then that tool is still usable by the community instead of it just becoming a dead weight.

That's great advice. I've often heard that referred to as the bus factor - how many people can get hit by a bus before your organization has a crisis.

Yeah, pretty much. I also think setting up expectations for community spaces is key. If you're a place which is charging hundreds of dollars for membership and is poised to be an incubator for small businesses, then there's a higher standard of tool uptime and training availability than if you're charging $50-100 in a community space that's more targeted towards the amateur hobbyist and is known to be a space run by the community. So having those expectations set and understood before people join makes it a lot easier to talk to folkds and explain why something may be delayed or certain classes might not happen every other week to get people started with the equipment.

That's a really good point about the expectations. What are your favorite things to show people how to do or get them started with?

It's fun doing a quick demo project and occasionally showing 3D printers off. More than them being the best at anything, the speed with which you can show a thing starting up and then start asking questions that help get the gears turning like "What kinnd of projects are you thinking of?". The best way to get people involved with making things is chatting about ideas and showing them immediately solutions for how they could do the thing. Because a lot of times, the thing they thought is impossible is not the most difficult thing in the world. This is the biggest benefit of a makerspace - the community who can point out a think and say something like "I tried that three years ago. Go to this website, you'll find the information you need."



I love that about makerspaces, and for me it's always been more about the community than the actual tools. You can potentially access the tools elsewhere or buy your own, but without the community to help the tools are a lot less use. My makerspace back in New Jersey has had times where they didn't even have a physical space but kept going because they had community.

It's a third space and it's that brain trust of people who've walked in your shoes, and even if they haven't done exactly what you want to do, they can probably point you in a direction that saves you a few hours because they've made similar mistakes to what you're about to do. Not saying mistakes are bad, they're great for learning. We're all going to make mistakes and it lets you become the person you need to be. But occasionally if you can cut down on the unecessary learning lessons it makes it a lot better.

Definitely. And speaking of mistakes, this is actually a question I was thinking about earlier today. What's your favorite project that went horribly wrong? Or at least something that went wrong but ultimately was fun or a good learning experience.

The more funny one was that walking shop costume that I talked about earlier and a few situations in which I went to pick something up while wearing it. As I stood back up my knees gave me just the whisper of "You get the one time to stand back up" and realizing I need to redesign the weight distribution on this thing, otherwise I'm going to hurt myself.

The less fun one that taught me more was years and year back, actually pre-pandemic. I 3D printed a project on a CR10 mini and was clearing off the bed. I was being stupid and doing it with a razor, and my hand slipped and I had to get a few stitches. I'll describe the issue as Tarantinoing a part of my room, but I nicked a nerve a little. Eventually the feeling came back, but it ws a bit of a process of healing. Luckily I didn't need physical therapy or anything, but that kind of stupid decision of I was much younger and was impatient in a way that most makers are when you're getting started. And that's what I'd consider the capital lesson that a lot of makers run into when strating out. You have a close call or something goes wrong because of a rushing decision that makes you stop and forever think "Wait a minute, slow down. How should I handle this properly?". And from now on I always use plastic razors to clear things, and I take my time to think through problems a bit more than I did previously. So it's never a full solution, but I feel like most makers reach that lesson at some point. And hopefully it's not a permanent issue that you get from it, but a moment you take away that you can think back on that reminds you to consider things a little more carefully.

Those are both really great answers, thank you.

What advice do you have for other makers?

For fabrication, or for just being part of the community?

Both, actually.

In fabrication - learn from every discipline. The amount of skills that I've gained in prototyping from talking to cosplayers and the amount of stuff I've adapted from woodworkers to use in 3D printed. there's a lot of skills and mental practices that are considered comon in one that are just anathema to others. And being able to put all these pieces together means that you're not the expert of any one thing, but you're better than that. You're an amateur in many things and can often find solutions that would otherwise require being at the top of a class to even consider, just by grafting a different community's problem solving onto it.

And as for the community aspect of things, I would say chat with your local makers, talk about projects, and know that most folks at a makerspace are there because they care about community and are interested in others and what they're doing. So it's not a burden to talk about your project or idea, and often that's what people are coming in for. And the more you talk about the stuff that you're doing, the more people can talk about their own experiences that will relate and often can give you ideas for future projects or ways to go about something that you're working on you otherwise wouldn't. Also, you make friends that way.

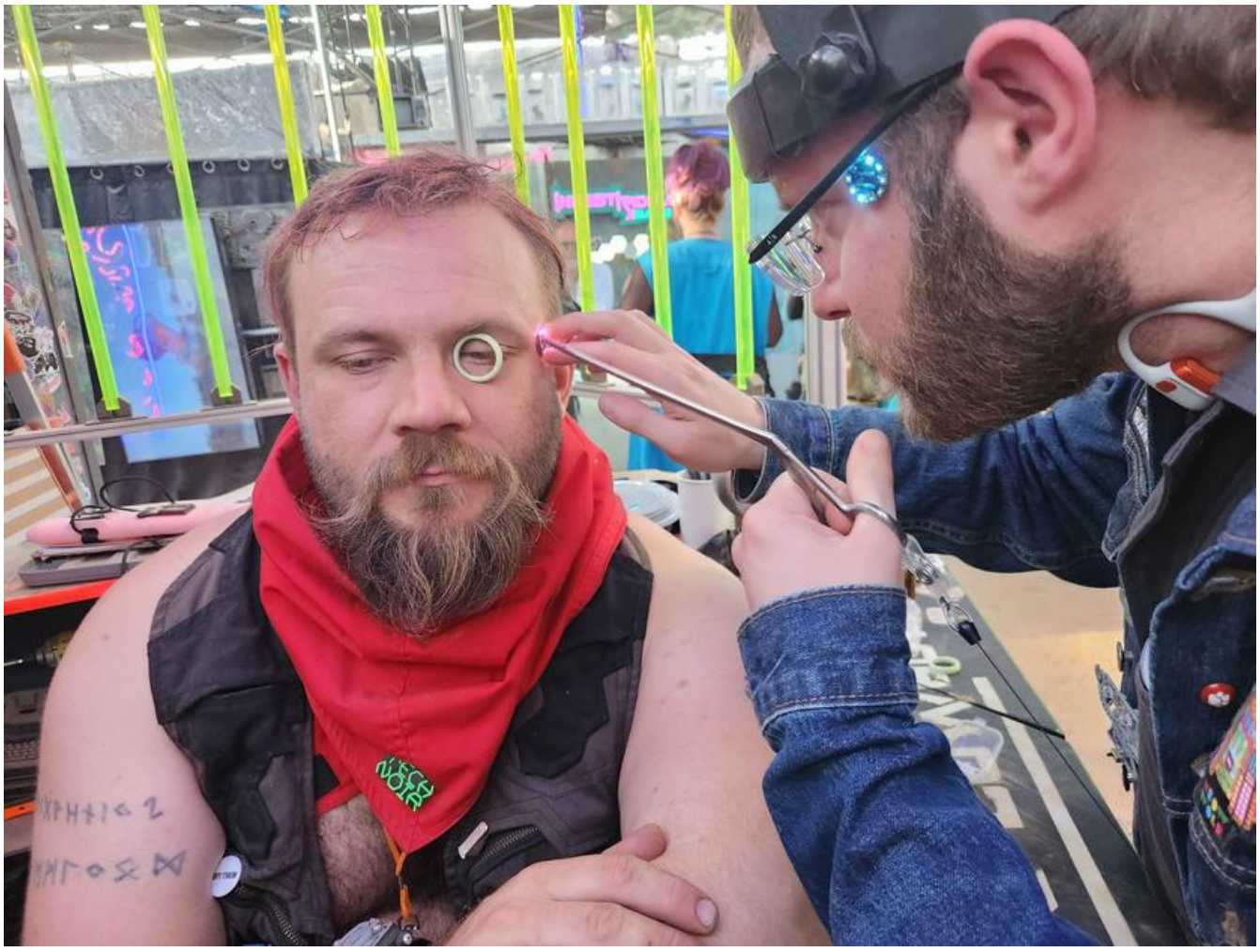
And the secret about makers is that they go everywhere. So it doesn't matter once you join a community like this. It's niche, but also distributed enough that you'll run into people wherever you go. I quite literally ran into people in both Seoul and Tokyo from the maker comunities that I'd known because I was there for different maker events and happened to run into people I knew from California in both locations. Which, again, how many communities can you go to a random even tand see someone that you met six months ago in a state you're not even living in?

I had similar experiences. I even moved to another continent and when I got here, I joined my local makerspace and it turns out that one of the guys here, we have mutual friends through a project he worked on.

This is both a useful tip and a warning for anyone who does join the community. Everyone knows each other and that both means that it's excellent that if you're trying to get in contact with someone or work on a cool project, there's a non-zero chance you known someone who may be connected to them. But also, a warning to treat everyone with respect both because you never know who someone is or becomes. And oftentimes it's that one hobbyist with a weird idea that either starts a company or has been in the community 30 years and knows everyone. And you never want to be that guy that was being an ass to someone and it turns out, there goes your entire community.

So you've already touched on this a bit, but what communities do you usually hang out in?

These days, it's largely the maker community for running makerspaces and various other community organization meetups. A lot of community organizing here in Philly both for the makerspace and helping a few other community orgs with logistics because at a certain point once you do one you kind of have the baseline for doing a bunch of them. I've done a bit of stuff like live action roleplay and gaming things with friends, which has led into the costuming and props works that I've made, lots of tabletop RPG things. Dungeons of Dragons, Call of Cthulhu, that kind of stuff.



And travel. Before I got into making as a community thing, travelling used to be my big hobby. I'd find a country, plan a week in which I'd hit up everything within a 40 mile radius of where I'd be landing, and head back. That kind of thing. So I do love doing that and it's been nice since combining the traveling with maker community stuff in places I've been needing to check out anyway.

That's wonderful. What's coming up next for you? Do you have any plans for projects or things that you're going to be working on in the next few months?

Yeah, a couple of them. I'm currently putting together an exhibitor table for 3DPrintopia, previously East Coast Rep Rap festival. It's the east coast's community 3D printing convention. So Hive76 is going to be tabling now that it's come to Philadelphia hopefully permanently, and so just getting the logistics for that set up. I learned when I joined Hive that some of the members were part of the reprap movement from the beginning and helped out with some of the heat bed stuff. So our logo was actually on some old heat beds and boards that were used for printers back in the Mendel days. So there's getting stuff together for that, and getting ready for the PAX Unplugged ARG because we're starting to reach the point in which people are planning for that event.

The biggest lesson of doing any logistics that I've learned is to think about the time frame that you need for something. Triple it, and you're about how long before the event you should actually try to reach out to folks before the stressful part of planning starts and then know that I'll probably get a response at least a week before.

You mentioned that 3DPrintopia is coming to your city. Does Hive76 have anything special planned to coincide with that?

So in this case, we're being more of a community resource, mentioning a few items around the city since this is the first year that it's here. We're going to be putting on a bit of a 3D printing museum as part of our exhibit. Since we've been part of the community for the last 15 years, we have prints and boards and machines that go back that long, some of which have been in storage for a while. So we can bring those out ot actually show a l ittle bit of a history of printing from the maker side of things.

And then if I can finish getting it programmed in time, I did build one of the walking coffee tables. If you've seen that project from a while back, that's a strandbeest coffee table and it would be bringing snacks around the event if I have time to finish it. It'll be really cool if we get that working in time.

That's really cool. What's your moonshot project? If you had all the time you needed and someone else paying for it, what would you create?

So, I've already talked about the repair zone stuff. And there's been an idea I've had for a couple of years now of wanting to do a maker zone, which would be a company-agnostic makerspace for conventions and events where we could partner with different orgs to be able to demo and run classes on different tools where it's both a place where someone at the event can use a tool if they know how to use it to help maybe repair a thing or make an addition that they've been meaning to, using something they don't have at home. And then for the people who aren't used to these tools, we could help get them up to speed, inspired, and knowing that these thigns exist. Because the amount of places that I've brought a 3D printer and people have nver seen one, when I know a lot of people in the maker community will consider that one of the most commonly owned items. Being able to bring some of these things out gets more community recognition while also being able to sustain it without being tied to one company.

I feel like that would be a lovely thing to be able to bring around different places and just have something that can inspire and create. Having something that someone does is infinitely more useful to get them to wrap their head around something than just showing off an item. So even if it is jus

having them assemble a key fidget, which we've done at a local Comic Con for the makerspace, just something that gets them hands-on with the different tools, gets ideas going, and reframing it from a thing that exists to a thing that could exist for them, and getting them into that mindset that they can do this - it's not just something other people do.

I love that. So, if people want to find out more about what you're working on and more about Hive76, where can they do that?

You can find me as Memento Mechanica on Instagram and a few other spots around the internet. That's where I tend to post some of hte projects I work on as well as different event work that I do. You can find Hive76 online at hive76.org as well as in northeast Philadelphia if you happen to be in the area. We do open houses every Sunday from 2 to 5. I'm also around at various maker events both for Hive76 and myself. If there's an event going on that has somethign cool to do with abrication and you see someone with a glowing, spinning denim jacket, there's a good chance that's me.

One last question for you. What do you wish I'd asked so you could answer it?

So, you did touch on this a bit, but I do love always asking how do you connect with your community? Not just how have you joined this community or what do you think of it, but how do you actually connect with them? And to answer it in my case, I'd say communication, trying to find points of interest where I can work with others and build projects and idea witht hem. But I do think the more fun part of that is asking that to others because there's a fun interaction that happens when you ask someone about how they connect with their community or asking someone what is something that they do that doesn't often come up. That leads to a little spark starting up int heir head where they start thinking about the communities that they're part of, the things they care about, and the people who they can connect with that may otherwise just go unconsidered due to it washing over in the day-to-day. So it's always one of my questions.

Everyone has something amazing and interesting about themselves. A lot of times it just doesn't come up in conversation, whether it just be circumstance or that no one's asked the right question that they feel comfortable to bring it out.

Staff Spotlight 1

Who are you and what do you do at LightBurn?

I'm Jackie Gerstein. I've been writing documentation and other content to help our users learn to use LightBurn since 2022. Before that, I was a high school technology teacher for 15 years. I taught CAD, digital fabrication, robotics, programming, woodshop, and other technology courses, and mentored FIRST Robotics Competition teams, in addition to creating and running my school's makerspace.

How did you end up at LightBurn?

I'd been teaching digital fabrication classes for years before I started at LightBurn. I'd been using a laser cutter in my classroom since 2015, and a CNC router since 2016. Trying to train an entire classroom of teenagers to use these tools and get everybody through a basic project on each machine was an interesting challenge, but creating templates for students to use and standardizing as much as possible helped a lot. LightBurn's Template Files would have been nice to have back then!

I was looking for jobs where I could still use some of those educational and technology skills when I met somebody working for LightBurn volunteering at a robotics event and eventually started discussing documentation and how to improve remote learning.

There may also have been a 2 AM email manifesto on how to improve the documentation involved.

What got you into making? What kinds of projects have you done?

My earliest start was through sewing and other crafting. My mom was a quilter and a computer nerd, and she introduced me to both sewing and computers when I was still very young.

I started with sewing, paper crafting, and helping my mom upgrade and fix computers, even if at the start my 'help' was handing her screwdrivers and holding a flashlight. My first CAD experiences were in 1991, using a program called Electric Quilt to design my own quilt blocks.

My interests have shifted around over the years, but a few of the things that really stand out and which I always seem to find my way back to are robotics (especially combat robotics), drawing, sewing, Power Racing, and all the things that make them possible.



Some of my earliest memories around making involve sewing and going with my mom to meetups for online quilting and other crafting communities. Even though I was a bit young to participate much, I'm glad to have been exposed to the idea of these communities so early.

How long have you been making things? What's the first thing you remember making?

I genuinely don't remember a time when I wasn't making things. I did a lot of crafting and learned to sew before I started kindergarten, using an old treadle sewing machine that had been electrified and ran very slowly.

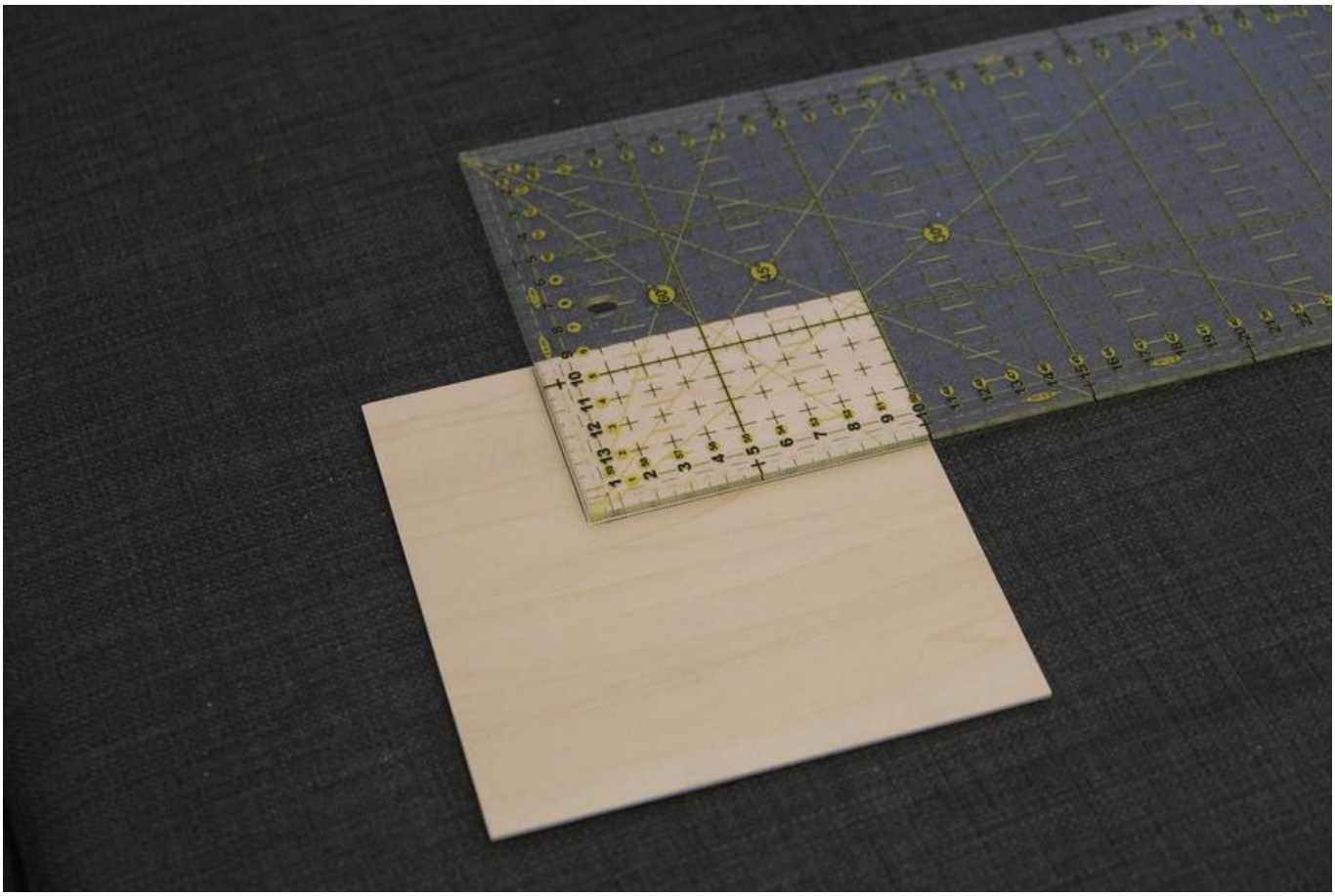
The first thing that really sticks out as a project of my own that didn't have directions or adult involvement actually got me in a bit of trouble. Home Alone had just come out, and my cousin and I were up late in our grandparents' house and started wondering what would happen if someone broke into the house. Naturally, our solution was to try to make traps and weapons to defend the house with. Certainly not my most impressive or practical project (to be fair, we were only 6), but it was certainly fun.

What's your favorite tool that isn't a laser, 3D printer, or CNC router? Or the one you think might surprise people?

> **Tired of rulers slipping around?**

> A little trick that might be useful to you if you do have a ruler that slips around easily when you try to use it as a straight-edge: Cut out some small pieces of a fine grit sandpaper and stick them to the bottom of the ruler with double-stick tape. It'll keep the ruler from sliding as easily, but isn't rough enough to damage most surfaces.

A quilting ruler. It doesn't replace a pair of calipers and isn't my only ruler, but it's such a great tool that I think most people who don't sew just don't know about. It's a clear, rectangular acrylic ruler marked in a grid instead of just a straight line, usually with some angles and arcs marked as well. Being able to quickly measure the width and height of an object, mark right angles, or cut a straight edge with a ruler makes projects so much faster and easier. As a bonus, the newer ones often have a non-slip texture on the bottom so it's easier to keep them from slipping around when you're cutting or marking lines.

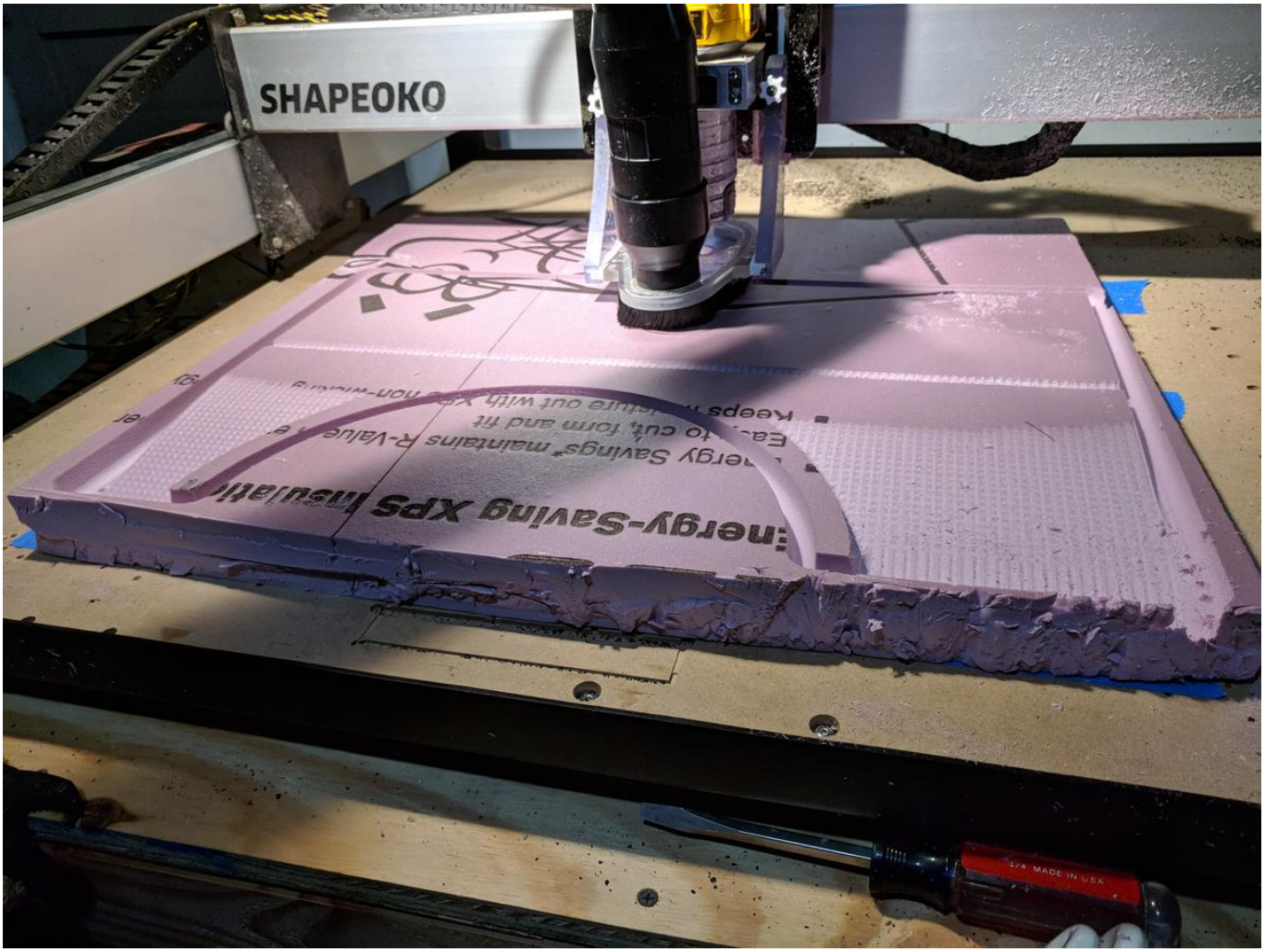


Do you have any project you're known for? What have you made that you think deserves a bit more recognition than it got? What are some of the projects you've done that really excited you and why?

I think that being on TV counts here, even if I'm not particularly well known for it - I've been on BattleBots with HiJinx!



If I had to pick a project that I'd love to see get out there a little more, it would be the Big Orange Truck I made for the Power Racing Series. This is a race series that started out with a bunch of adults (chronologically, at least) racing Power Wheels cars around a parking lot in Chicago, and still doesn't take itself too seriously. Each kart needs to be based on a children's ride on toy, or at least look like something that could have been one. Since a 1980s Toyota Hilux didn't seem to exist in this format (I can't imagine why), I opened up my CAD software and got to work. I modeled up something that looked reasonably similar, carved it out of pink insulation foam on my CNC router, reinforced the surface with fiberglass, and painted it orange to match the Tacoma we'd be driving it to races in.



Carving one of the side panels.



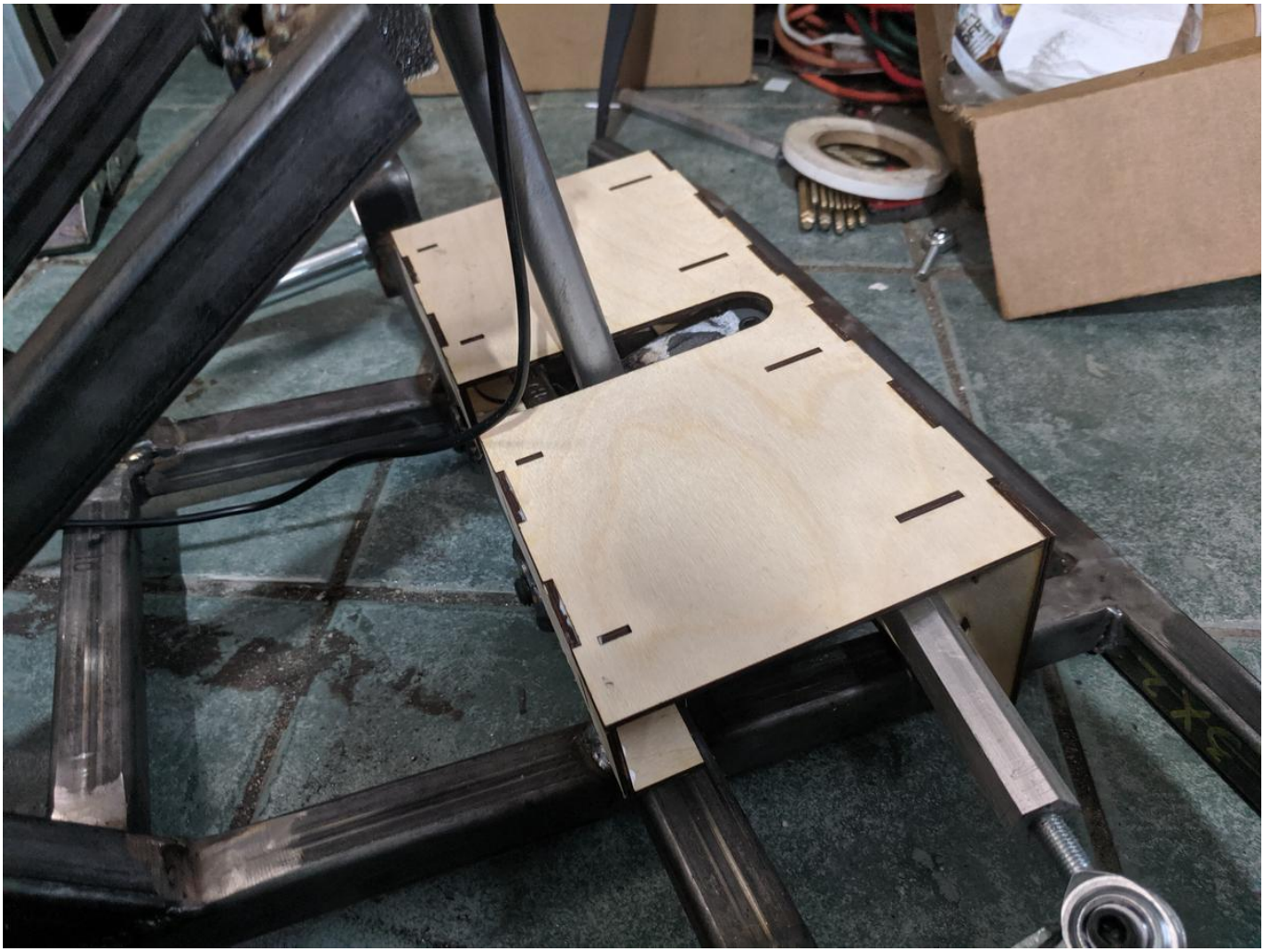
Getting started on the fiberglass. This was the worst part of the build.



The entire body before painting and mounting it to the frame.



After a few races.



I used my laser cutter to make fixtures for welding, and some parts for the finished build.

❮ ❯

What are some of your favorite skills/techniques that you've picked up along the way?

Not a specific technique, but definitely a skill - I can pick up new skills quickly. I'll never be the world expert at any one thing, but I can do a lot of things well enough to get by, and those varied skills reinforce each other to be useful at the weirdest times.

What's your favorite project that went horribly wrong?

I mentioned the Power Racing Series earlier - my wife taught me to weld while we were working on a build together. I am, at best, a pretty mediocre welder (although I've gotten pretty good with an angle grinder because of that, which was useful at BattleBots). Ultimately, only one of my welds actually made it onto the finished kart. I was pretty proud that one of my welds was good enough to make it into the finished kart.



I don't have any pictures of the weld that tried to kill me, but I learned a lot and made much better welds here.

Turns out it wasn't quite good enough, as I discovered coming around a curve, going full speed downhill towards a wall, when the steering column came loose in my hands. Luckily, the column was still connected through a universal joint, even if it wasn't anchored to anything, and I managed to steer enough to avoid smashing face-first into the wall.

Since then, I try to make sure my testing matches the intended environment. If I want a kart to survive steering as I'm bouncing around on a rough course, I need to test it in similar situations. If a combat robot can't survive falling to the floor or being driven by a 5 year old, it's probably not going to hold up well in an actual fight.

Are you involved in any makerspaces or other similar groups?

Yes! I joined FUBAR Labs in NJ back in 2010, and it absolutely changed my life to have a community of makers to share projects with and generally get excited about learning and doing new things.

Through FUBAR, I got involved with the Power Racing Series, and helped start GSCRL, a combat robotics league in NJ.

Last fall, I moved to Portugal, and before I'd even finished moving in, I'd started getting involved with LCD Porto.

If you haven't already been to a makerspace or something similar, I'd strongly encourage checking to see if there's one near you. Even if you don't need the tools they might have, having community that shares your interests, or at least gets excited about them with you, is so important.

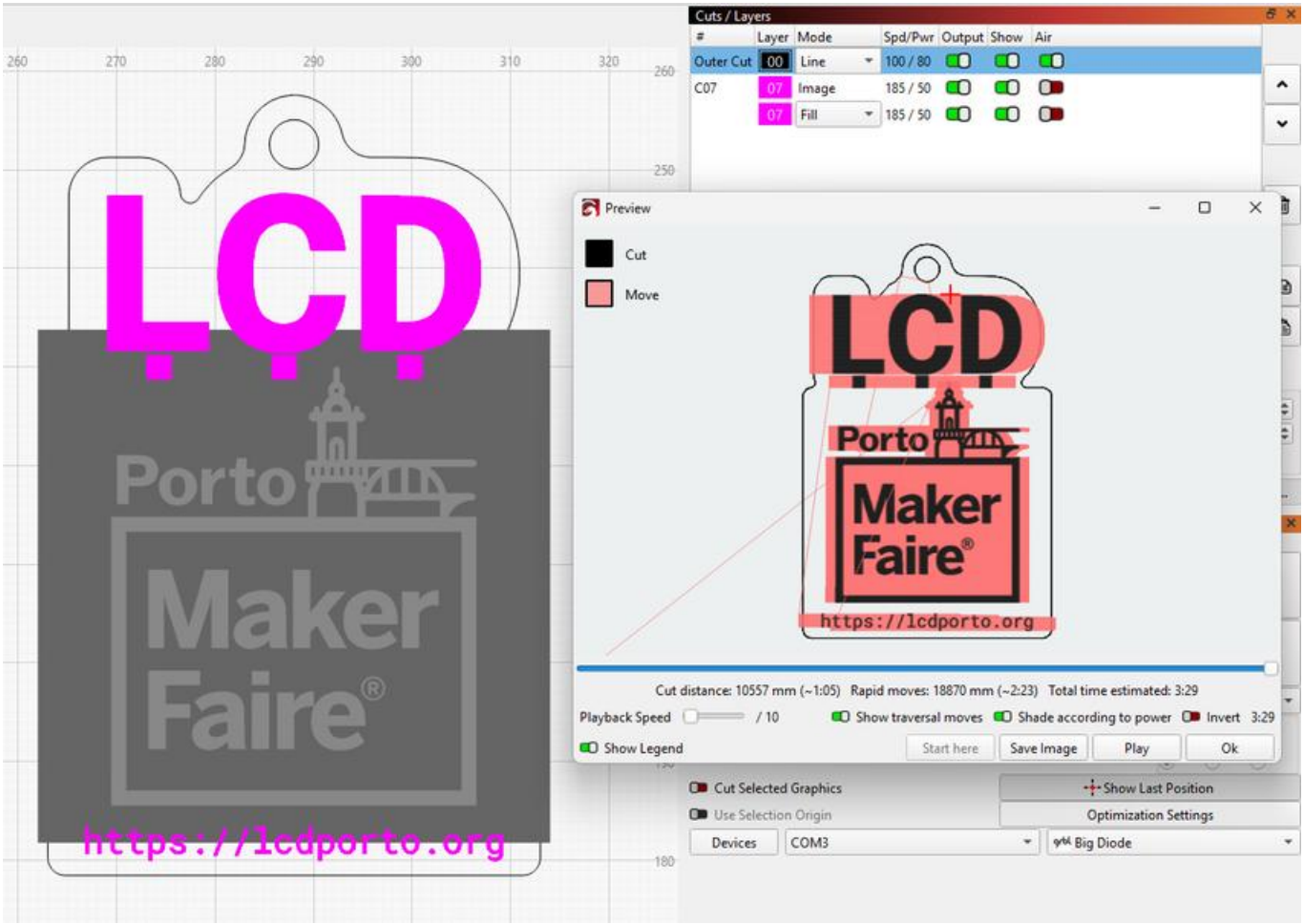
What communities do you usually hang out in? How did you find them?

On the maker side, currently mostly LCD, Liga Rec, and GSCRL. I've also been starting to get involved with the 3D printing community again. Beyond that, I've got some local groups and general social groups.

What's a LightBurn/MillMage tip you'd like to share? Do you have any surprising ways you use it?

I think the Threshold image mode is a really useful option that a lot of people miss. I see a lot of people tracing black and white images just to engrave them, when doing a Threshold image engraving would preserve more detail and be easier. The screenshot here shows an example of how we used Threshold to engrave the Porto Maker Faire logo on a keychain. This one required some adjustments to the Threshold settings because of low contrast, but it worked well and gave us more detail than tracing would have.

I use LightBurn a lot for laying out parts even if I won't be using LightBurn or MillMage to cut them, and for cleaning up and fixing vector files.



What's next for you? What will you be focusing on in the next few months?

I'm going to be working on some more upgrades to my current combat robot, Wobbegong, and getting ready for the next event in November. I've got some workshops coming up at my local makerspace, and I want to play around with 3D engravings on both my router and my new galvo laser. I'm also working with a few other people at my makerspace to help a local theater group automate treadmills to use as part of a set.

What's your moon-shot project? If you had the time to devote to it and someone else covered the budget, what would you create?

Do I really have to pick just one?

On a personal level, I'd like to create really nice custom furniture perfectly suited to my home and the things I want to store. And if we're really ignoring cost and time - a heavyweight combat robot of my own. They're fun, but expensive and time-consuming.

On a broader level, I'd want to really put a lot of time into building up stronger outreach, networking, and resources for combat robotics and makerspaces. I'd love to be able to host workshops and build sessions with materials provided, and helping other people with their ideas contributes to my own projects as well.

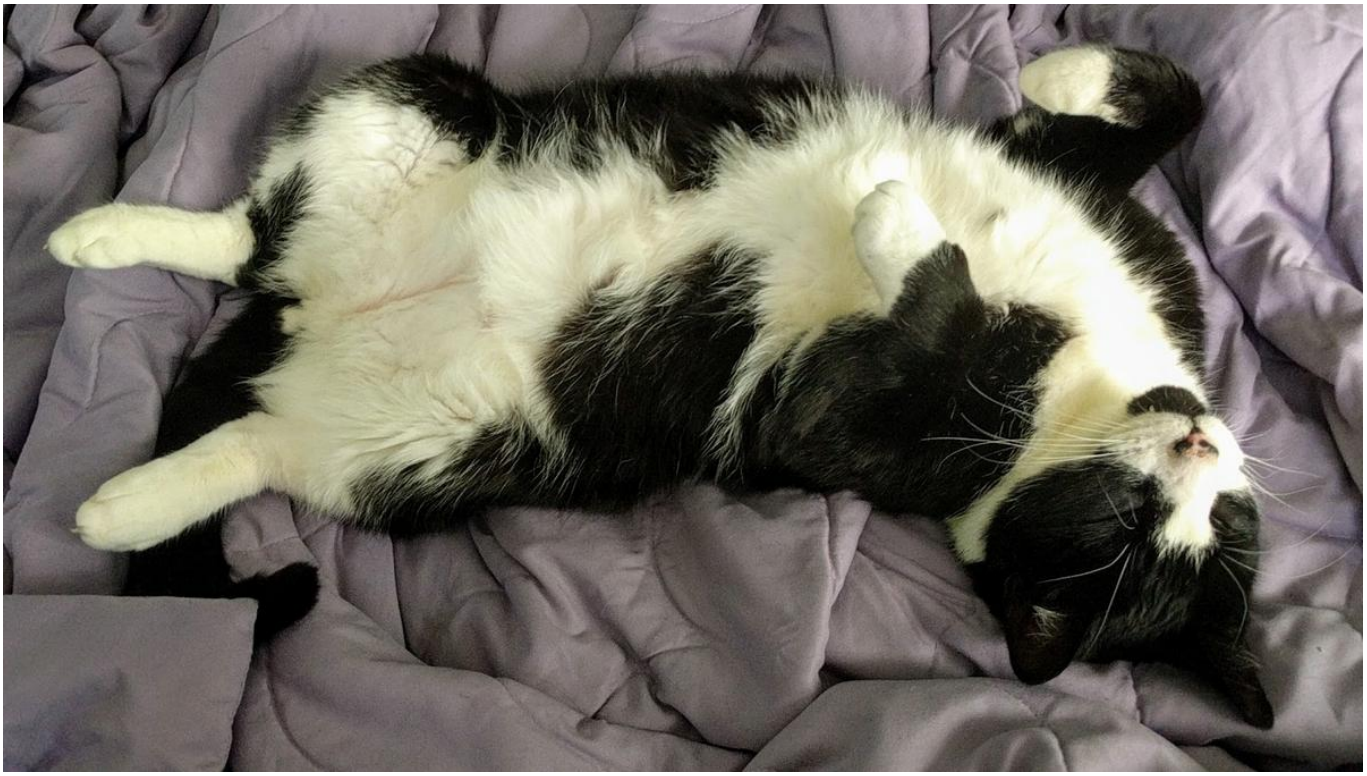
Where can people find you

For now, the best place to find me is the LightBurn Forum. If I go back to having more public social media or maintaining my website again, I'll update here.

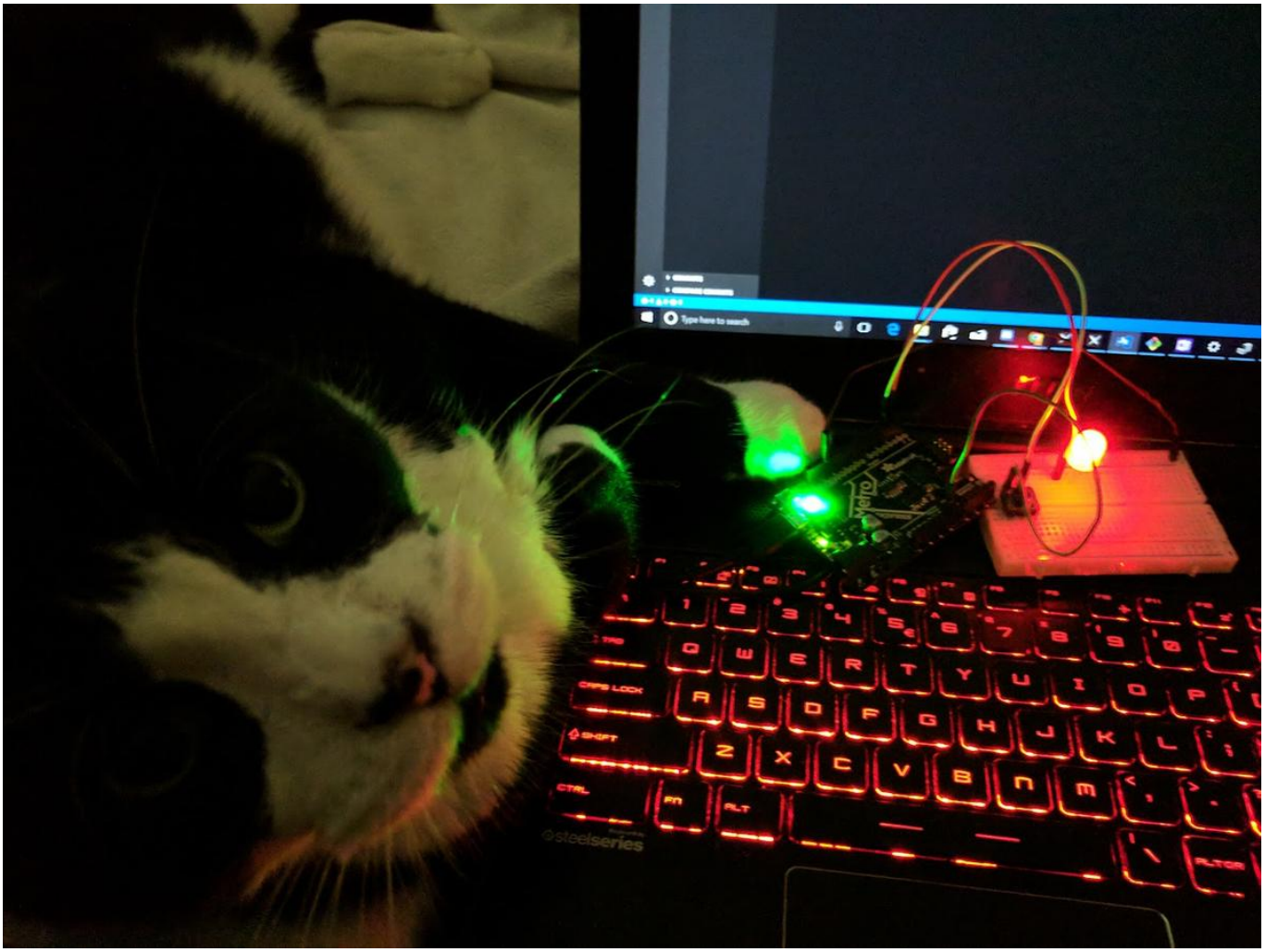
What do you wish I'd asked you so you could answer it?

Show me your pets, please.

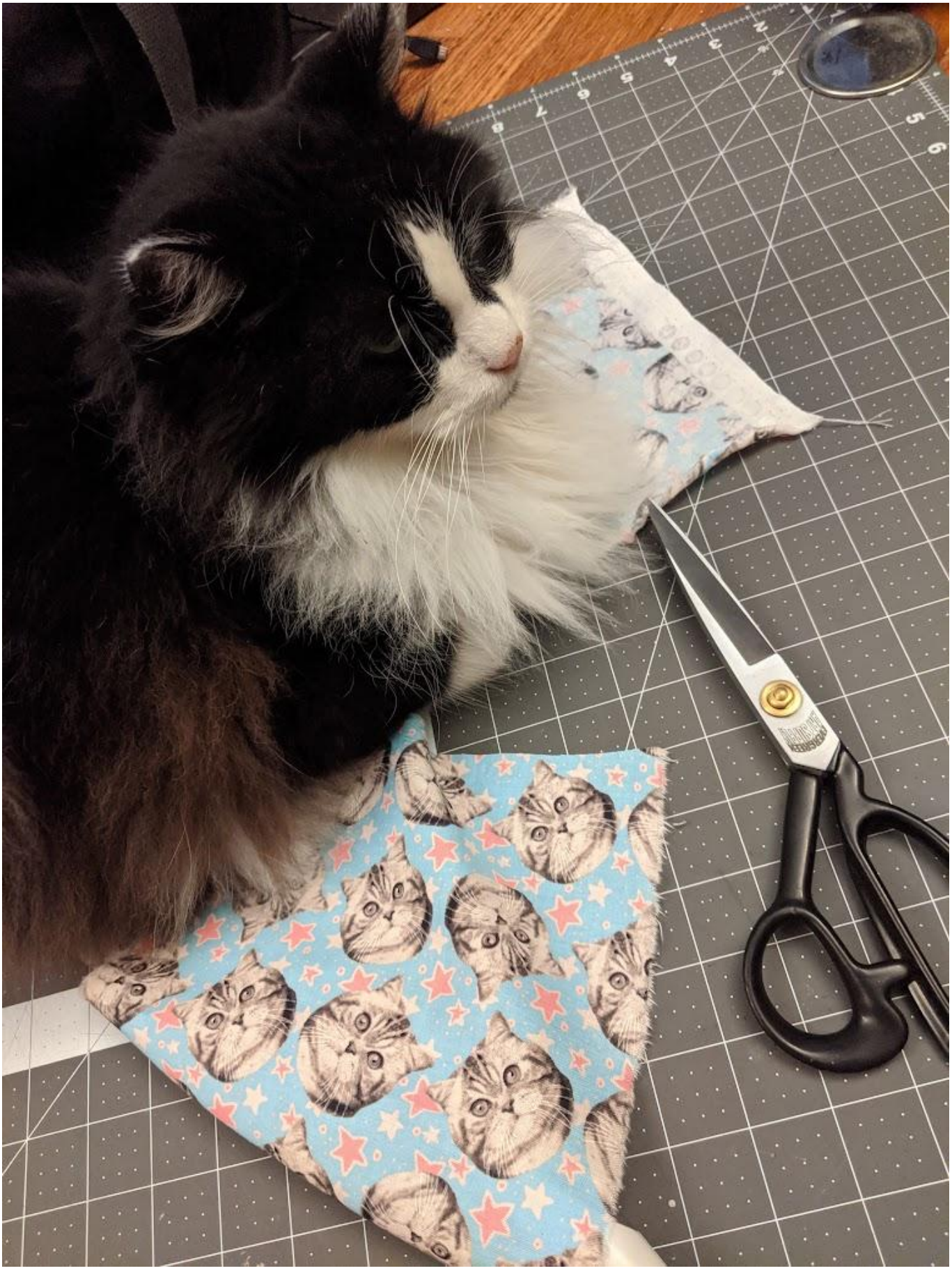
I don't have any pets at the moment, but that doesn't mean I won't take this opportunity to pull out some old photos.



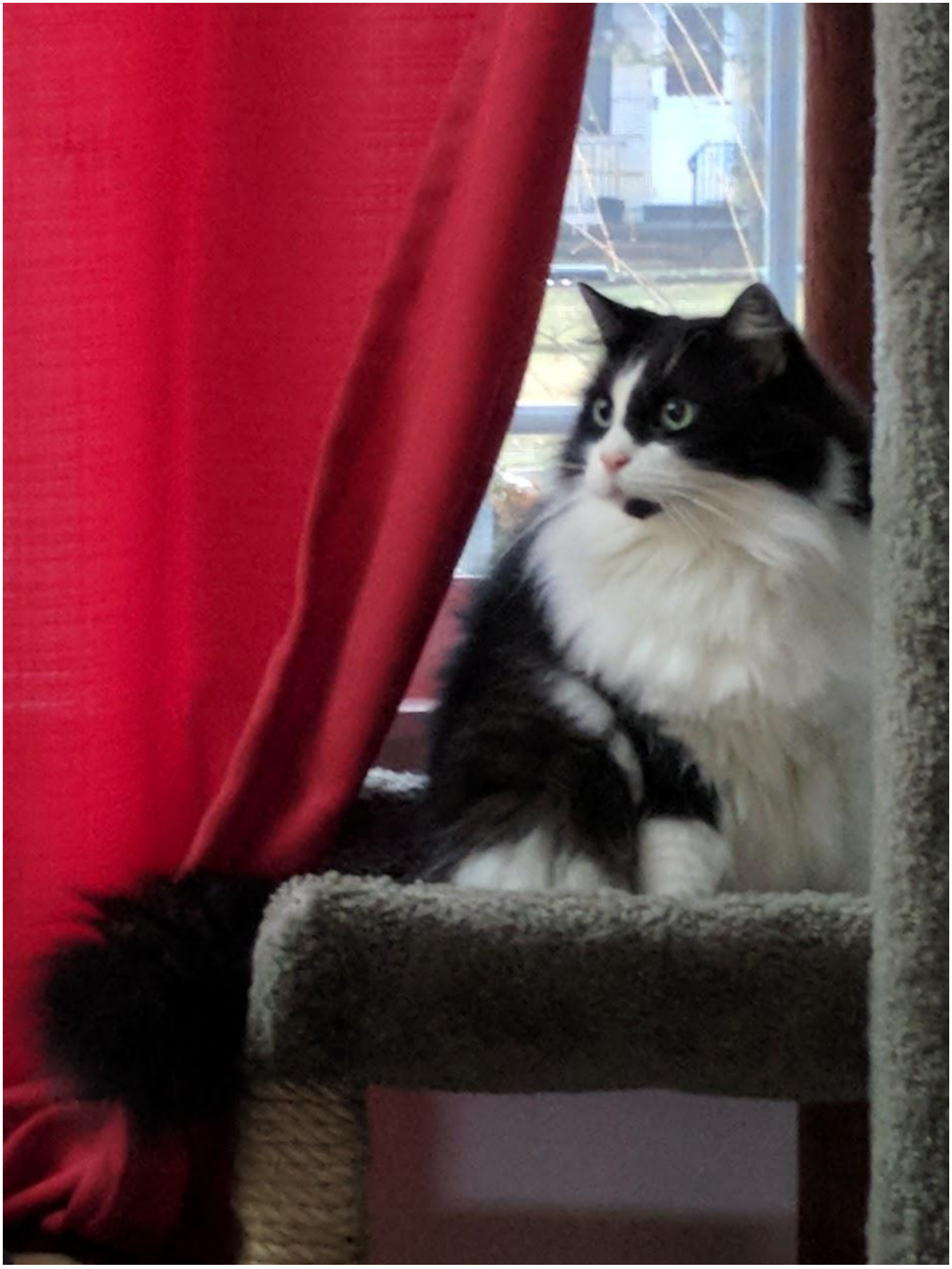
A genuine request for belly rubs.



Helping prep for an Arduino workshop. Or at least being cute and distracting.



"Why do you bother with these other cats when you already have the prettiest, floofiest boy right here?"



I wasn't joking about the floof



Taking a break from stealing vegetables to enjoy the sunshine on a warm day.

❮ ❯

Classroom Safety

Laser cutters are wonderful tools, and with proper supervision and precautions, they're well suited to classrooms. As with any tool, there are some hazards to using lasers, and it's important to know how to manage them. The main risks of a laser are:

- Eye and skin damage (including potentially blindness).
- Hazardous fumes
- Fires

A classroom or other shared workshop setting doesn't add new types of hazards, but it can make them a little more challenging to manage. Working alone in my office, I can run an unenclosed fiber laser because I know I'll keep my safety glasses on - and since I only need a single pair, I have good ones that fit tightly to my face. In a room of 20+ teenagers (who, as every teacher knows, are renowned for their judgment and consistency in following directions), setting up safer systems that rely less on individual making good decisions is always going to be a good choice.

Always use your laser cutter according to manufacturer directions. Include your school district's facilities manager in your planning, and if you work at an institution with a laser safety officer, consult with them as well. In a school setting, the use of laser cutters should always be supervised by a teacher with the appropriate training and certifications, as is the case with all tools.

Eye and Skin Damage

Lasers can cause eye and skin damage. The best way to avoid this is to use a properly enclosed laser cutter that's categorized as Class 1, which includes most commercially sold CO_2 lasers and some enclosed diode lasers. Class 4 lasers, such as open-framed diode laser engravers, require significant safety controls to be used safely.

Laser Classification

> **What About Class 2 and 3?**
>
> Because of the power output needed to engrave and cut, laser cutters are generally going to be either Class 1 (for enclosed and interlocked systems) or Class 4 (for the bare laser and for cutters without the built-in protections of laser cutters categorized as Class 1). If you're curious to know more about other classes, we have links below on the laser hazard classification system.

When it comes to laser safety, lasers are categorized into **Class 1** through **Class 4** by the level and type of hazard they present.

Class 1 lasers are not considered hazardous and don't require additional safety controls. These are either very low-power lasers, or systems that are fully enclosed with safety interlocks that prevent exposure to the laser beam. Enclosed CO_2 laser cutters are typically Class 1 systems because of their interlocked enclosures.

Class 4 lasers are the most dangerous and require strict safety controls to prevent harm. These are high-powered lasers that can cause eye and skin damage from a direct beam, and possibly even from a reflection. These lasers require significant safety precautions. Lasers with an output above 500 mW (0.5W) are automatically Class 4, although they can be part of a Class 1 system as described above. Open-framed diode laser cutters, and even some enclosed ones, are Class 4 lasers, as are the actual lasers in Class 1 systems.

Resources on laser hazard classification

Some useful sources of information.

- ANSI Blog: Z136.1
- Laser Institute of America: Laser Hazard Classifications
- University of Wisconsin-Madison Laser Classification Requirements

Classroom Considerations

In a classroom or other shared workshop space, it's particularly important to use Class 1 laser cutters. The more people you have present in the room, the harder it is to maintain appropriate safety controls for a Class 4 laser. Class 4 lasers in shared spaces should be managed very carefully and in consultation with a laser safety expert.

Laser Safety Glasses

While properly rated laser safety glassses and goggles are available and highly effective as part of your safety controls for a Class 4 laser, they're not an ideal solution for classrooms.

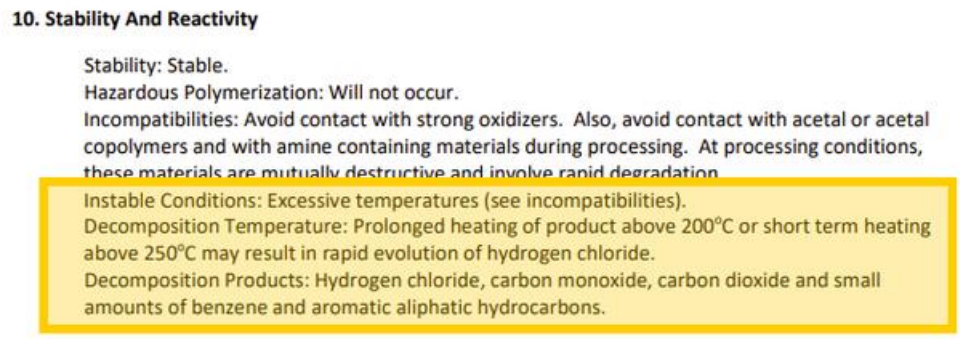
1. The glasses that come with inexpensive diode laser cutters are sometimes tinted glasses rated for impact resistance, not properly rated for laser usage. Properly rated, trustworthy laser safety glasses are expensive and a full class set is probably going to be more expensive than the laser itself.
2. If you've ever taught in a shop setting that requires wearing safety glasses, you know how hard it is to keep students from taking them off. It gets even harder when the glasses are extremely dark and make it hard to see.

Hazardous Fumes

When you use a laser cutter, you're burning material away, which produces unpleasant and potentially harmful fumes. The fumes vary based on the materials you're using. Some materials produce unpleasant smells or even toxic and corrosive fumes.

- Be careful with your material choices. Lasers are versatile and can cut a wide range of materials, but not all of them are safe to cut. We've listed a few common examples below, but can't list every hazardous material. If you're unsure whether it's safe to burn a material, a good starting point is to look for an SDS (or your country's local equivalent) and look for firefighting precautions

and decomposition products/conditions.



10. Stability And Reactivity

Stability: Stable.
Hazardous Polymerization: Will not occur.
Incompatibilities: Avoid contact with strong oxidizers. Also, avoid contact with acetal or acetal copolymers and with amine containing materials during processing. At processing conditions, these materials are mutually destructive and involve rapid degradation.
Instable Conditions: Excessive temperatures (see incompatibilities).
Decomposition Temperature: Prolonged heating of product above 200°C or short term heating above 250°C may result in rapid evolution of hydrogen chloride.
Decomposition Products: Hydrogen chloride, carbon monoxide, carbon dioxide and small amounts of benzene and aromatic aliphatic hydrocarbons.

    - PVC is a common material that you should never laser — burning PVC or significantly heating it releases hydrogen chloride gas, which is corrosive both to your laser and your lungs. Never laser PVC or any other plastic that contains chlorine.
    - Plywood can be made with a variety of glues, some of which release noxious fumes when burned, so the safest choice is to use plywood specifically intended for lasering.
- It's important to vent the fumes outside or use a properly rated and maintained filtration system to clean the air if venting outside isn't an option.

Classroom Considerations

- In a classroom or other shared space, it's easy to mix up laserable and non-laserable materials, especially in scrap piles. Particularly for scraps or unlabeled materials, make sure that you know exactly what a material is before lasering.
- It's critical to make sure everyone with access to the laser understands what can and can't be cut, or checks with someone who does.

> **What not to do**
>
> Some of my recommendations come from lessons I've learned the hard way. Years ago, I told a student to test a complex design in cardboard before cutting it out of wood, expecting him to use a piece of cardboard from the supply I kept by the laser. Instead, he used a greasy pizza box left over from the robotics meeting the night before.
>
> After putting out that fire, I'd never been so awake in first period before. That's also when I instituted my rule that all students had to show me their material before cutting.
>
> Speaking of which...

Fires

Fires are the most dramatic and visible hazard of working with lasers, but they're also quite manageable as long as you're prepared and always supervise your laser. It doesn't matter if you've run the same job 100 times without an issue, there's always a first time.



Never run your laser unsupervised!

To reduce your risk:

- Have spray bottles of water and appropriate fire extinguishers on hand near the laser, and make sure anyone using the laser is trained to use them.
- Be careful with materials that may have voids (hidden air pockets), because the air pockets can flare up into a fire when you hit them with the laser.
- Never run the laser unattended.

Classroom Considerations

- Check with whoever is in charge of your facilities on purchasing and maintaining fire extinguishers.
- Make sure your students are trained on fire safety and know what to do, but always supervise the laser along with the students while it's running. Don't count on students being able and ready to respond to a fire.

Conclusion

While laser cutters do come with potential risks like any tool, those risks can be easily managed by choosing an appropriate machine, avoiding unsafe materials, supervising your laser properly, and being prepared to extinguish a potential fire.

If you've already got a laser, why not try making a project. And if not, consider taking a look at our Hardware Compatibility Matrix for LightBurn-compatible lasers.

Projects

We've been working on projects to share with our users. Each of these projects might introduce you to features you might not be familiar with. You can use these projects just as they are, or as a launching point for your own creations.

Laser Cut Dinosaur



Make a 3D puzzle in LightBurn, and learn to adjust tabs and slots automatically for different materials and correct your laser's kerf.

Go to Laser Cut Dinosaur

Graham Cracker House



Celebrate the holidays in style with LightBurn by making laser-cut graham cracker houses.

Go to Graham Cracker House

Image Engraving on Coasters



Make your own custom coasters in LightBurn to protect your favorite table.

Go to Image Engraving on Coasters

Laser Cut Fridge Magnets



Afraid to commit your stickers to something impermanent? We walk you through turning them into fridge magnets using LightBurn.

Go to Laser Cut Fridge Magnets

Laser Etched Clock



Create an analog clock out of common materials using LightBurn.

Go to Laser Etched Clock

Laser Engraved Pencils



Use a custom jig to engrave pencils quickly and easily in LightBurn.

Go to Laser Engraved Pencils



Graham Cracker Houses

To celebrate the holiday season, we designed and made these small, simple, and, in our not-so-humbug opinion, utterly delightful graham cracker houses. We used graham crackers instead of gingerbread because they're relatively flat, uniform, and available in any grocery store. Making gingerbread just takes too much time we'd rather spend lasering. While this project might seem like pure holiday frivolity, stay frosty and you'll also learn a thing or two along the way.

What You'll Learn

In this guide, we'll cover several design and layout techniques, including:

- Using Tool Layers to represent material and serve as guidelines in your LightBurn workspace
- Using snapping behaviors for alignment and positioning
- Patterning with the Grid Array tool
- Modifying designs with Offset, Cut Shapes, Weld, and Edit Nodes
- Using the Offset tool and Cut Selected Graphics to create a jig for your project
- Aligning your output to the jig using User Origin or Current Position

Materials

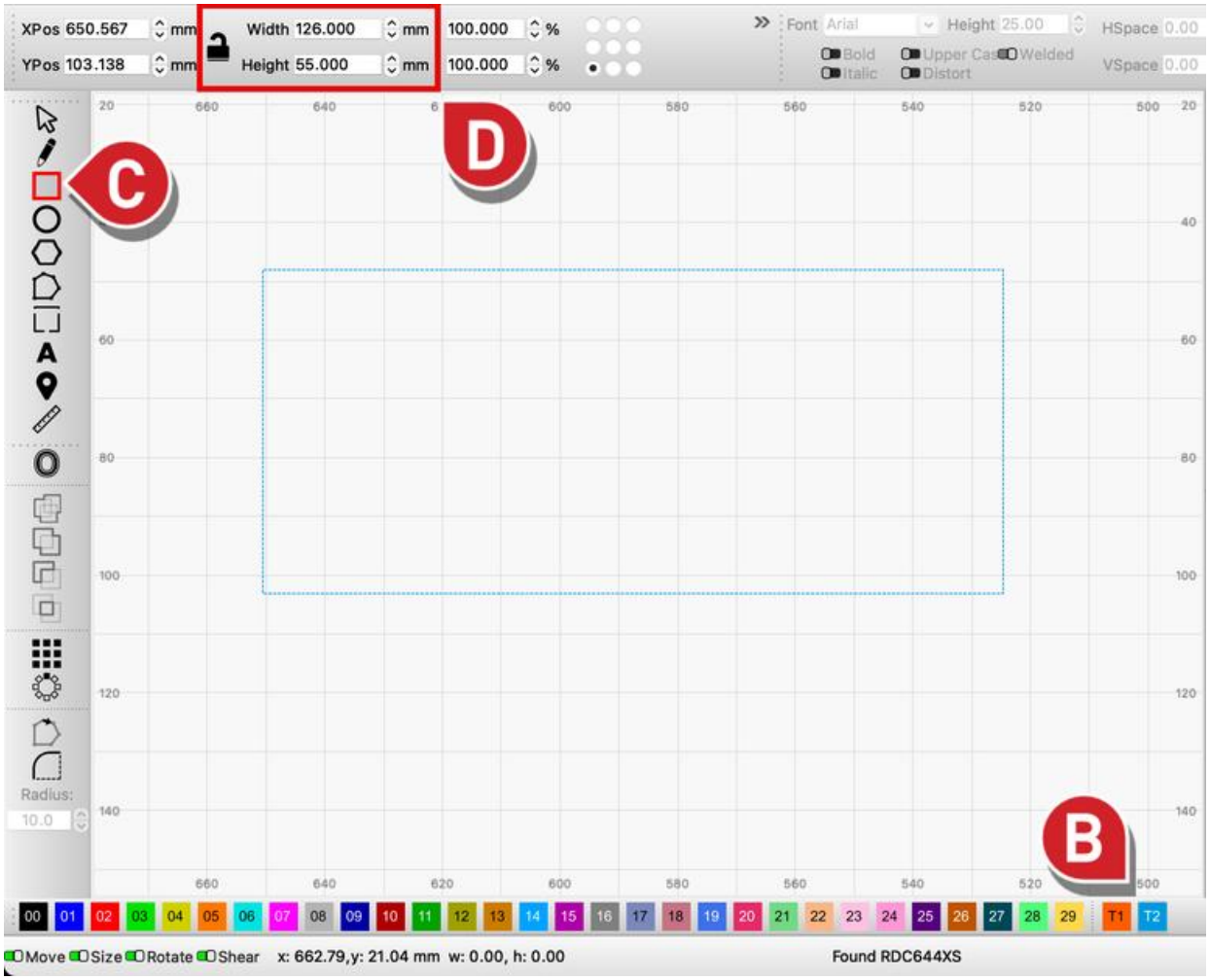
- **Graham crackers** of any size
- **Cardboard** to create a jig (optional)
- A **laser** capable of cutting through your graham crackers
- **LightBurn**, of course!

Steps

1. Make a Graham Cracker Template

We'll start off by making a template to represent a single graham cracker in our LightBurn workspace.

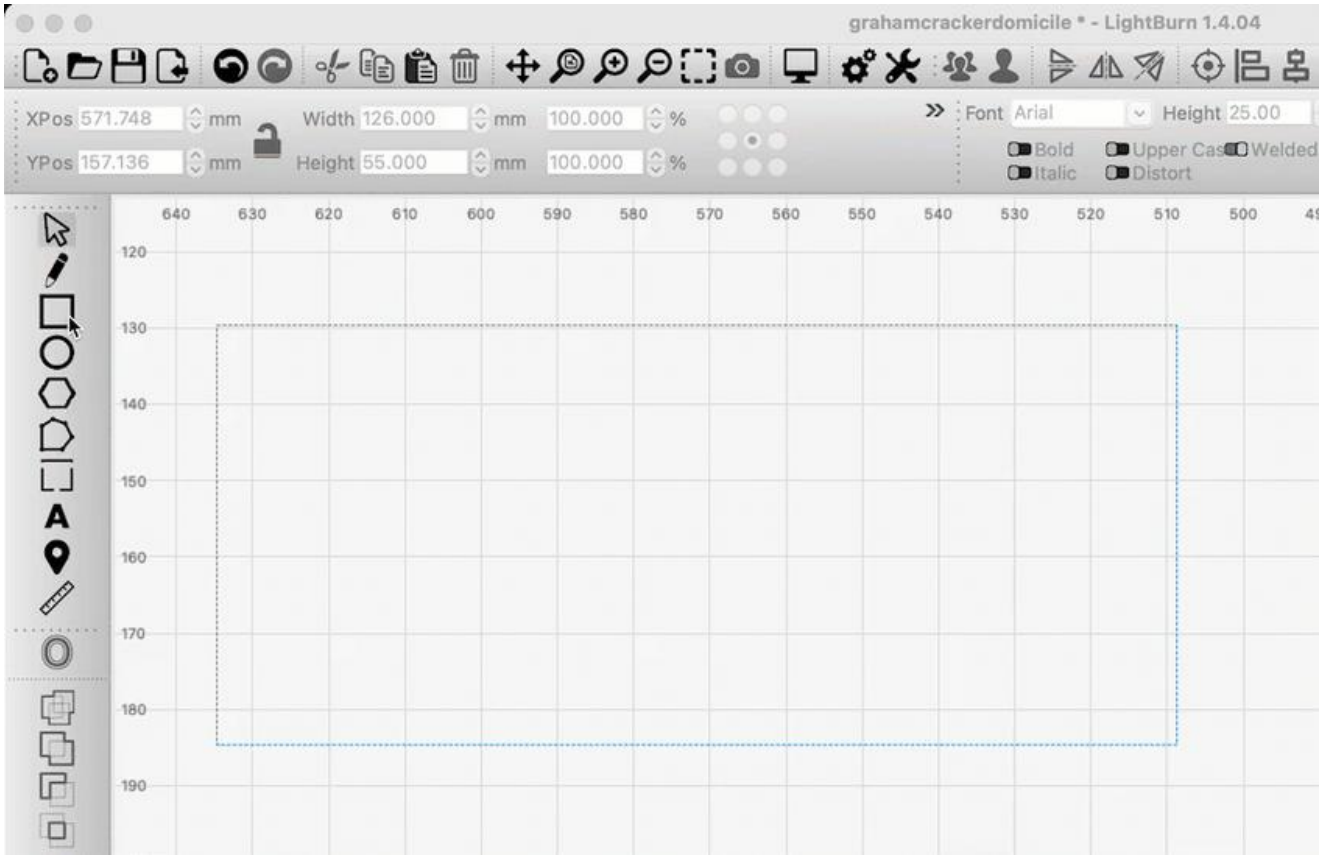
1. Measure one of the graham crackers you'll be using. You'll need to know its length, width, and thickness. We'll be using 126 x 55 x 5.5 mm crackers, but any size cracker will work with some modifications. A bit of rounding is ok for this project.
2. In LightBurn, select a Tool layer.
3. Create a rectangle using the Rectangle tool (`Ctrl`+`R` / `Cmd`+`R`).
4. Adjust its dimensions in the Numeric Edits Toolbar, matching the length and width to the measurements you took of the graham cracker. Make sure the lock icon is unlocked so you can adjust each dimension independently.



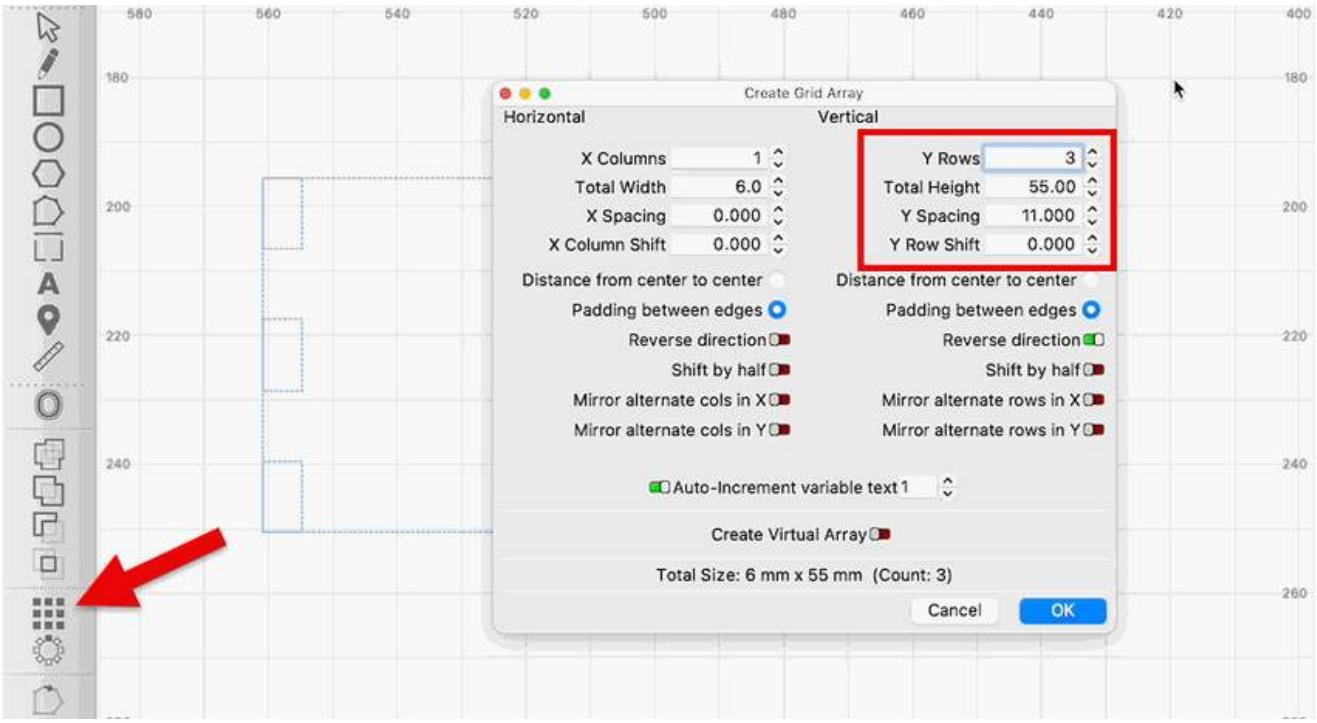
2. Add Guidelines for Tabs

Now we'll add some additional objects set to a Tool layer to serve as guidelines for snapping and alignment.

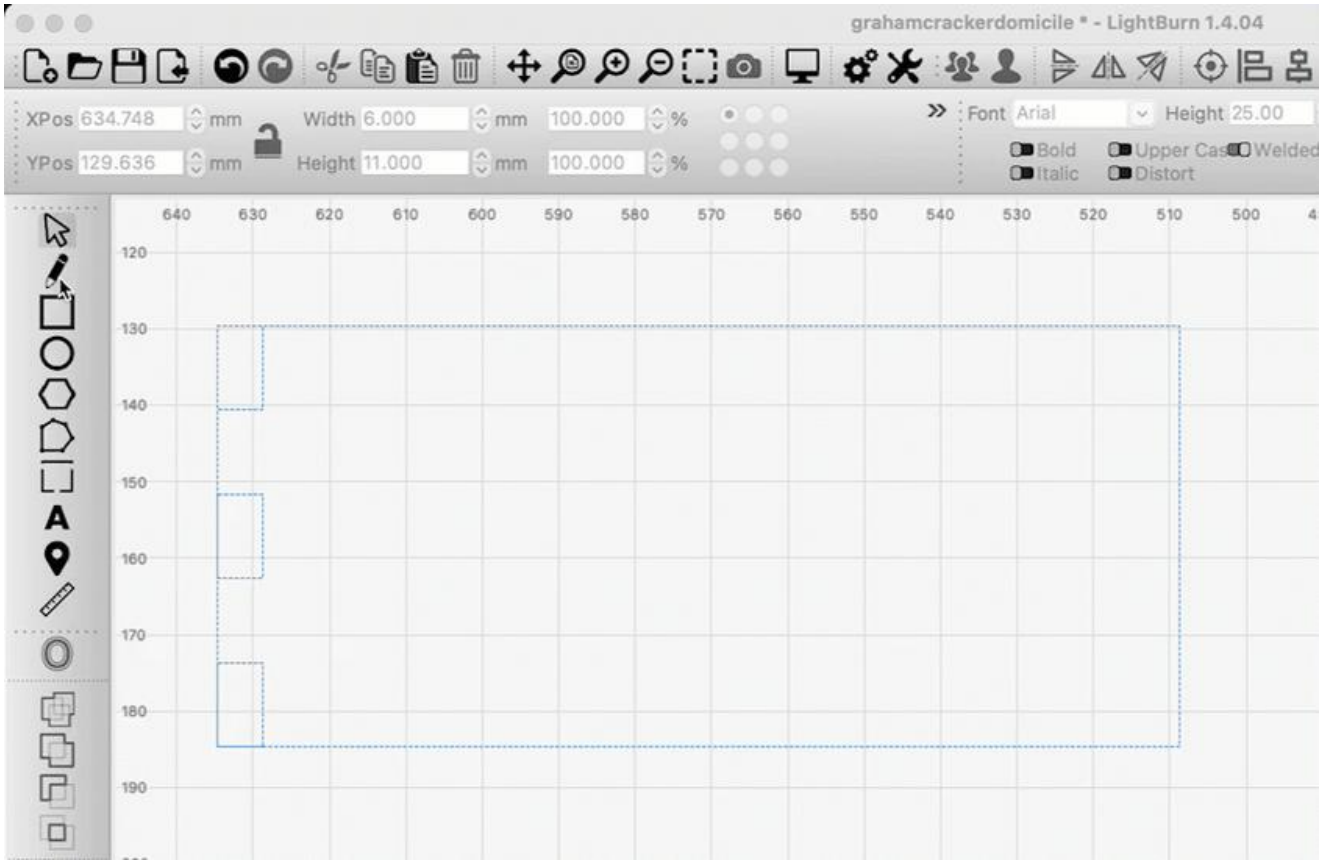
1. Create a new, smaller rectangle, starting from the top left corner of the large rectangle. As you start the rectangle, make sure you see the crosshair icon indicating that the start will be snapped to the corner of the large rectangle.
2. Then, divide the large rectangle's height by 5, and set the small rectangle's height to that value (11 mm in our case). Set the width to a number that's a bit larger than the thickness of the graham cracker (6 mm in our case). Before you adjust height and width, make sure the 9 dot control button is set to the top left corner — that will anchor the shape at that point.



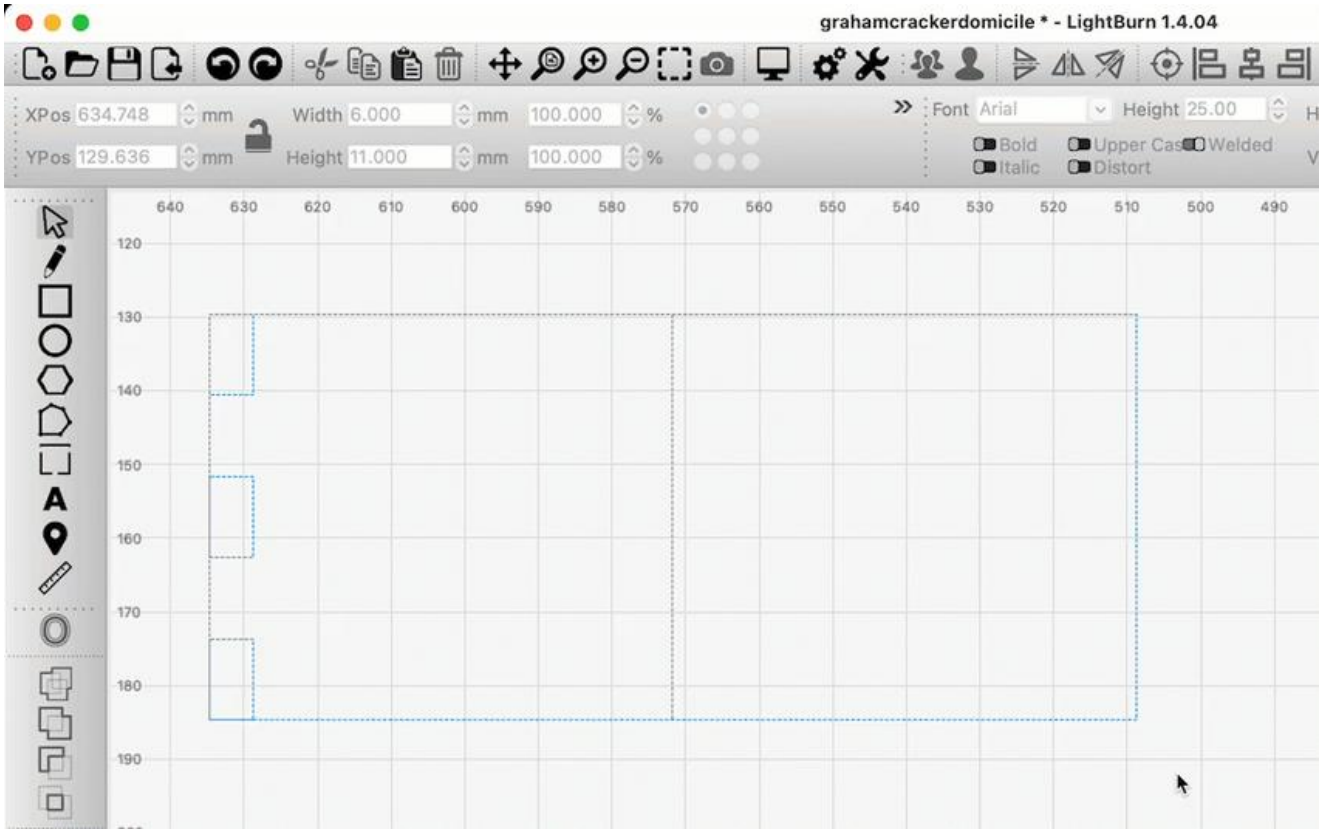
3. Select the small rectangle, then click the Grid Array tool. Create 3 **Y Rows**, with **Y Spacing** set to match the height of the small rectangle, and **Padding between edges** selected. You'll also need to enable **Reverse Direction**.



4. Select the Draw Lines tool (`Ctrl`+`L` / `Cmd`+`L`) and hover over the top center of the large rectangle, moving your cursor over the line until you see the midpoint snap icon appear. When you see the icon, click and extend the line down to the bottom, making sure you see the midpoint snap icon again at the bottom before clicking to release. Press the ESC key to finish drawing that line.



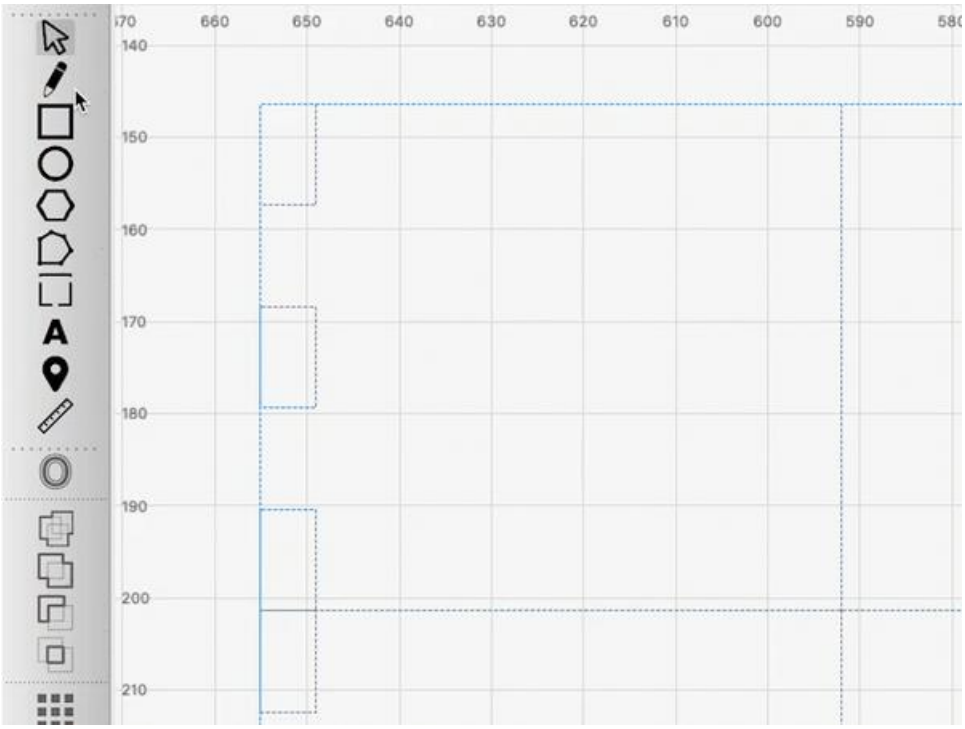
5. Select everything we've made so far, and **Duplicate** it (`Ctrl`+`D` / `Cmd`+`D`). Select the duplicate from its top center (click where you see the midpoint snap icon) and drag and snap it to the bottom center of the original. We'll come back to this piece in a bit.



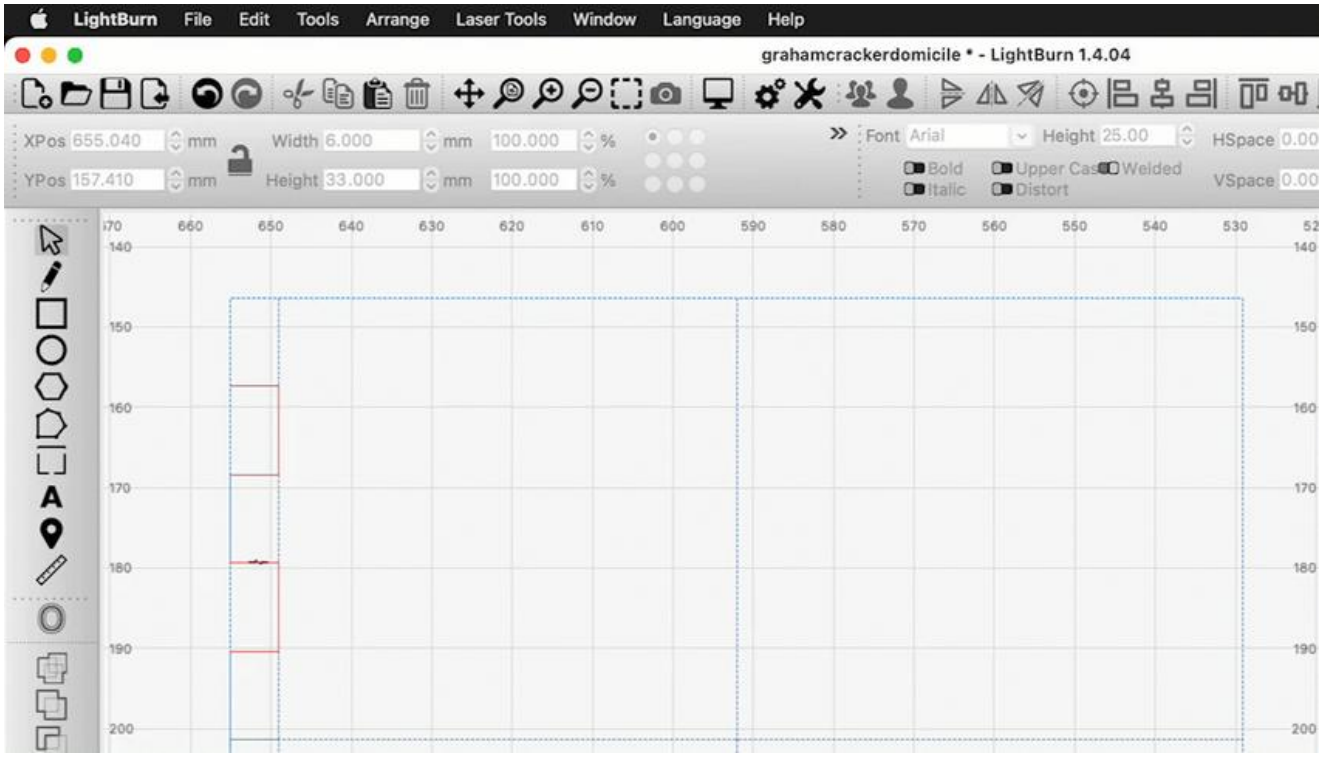
3. Front and Back Walls

So far, all of the objects we've made are set to Tool layers, meant to serve as guidelines. Now, we're going to be creating some actual cut lines we'll be sending to the laser.

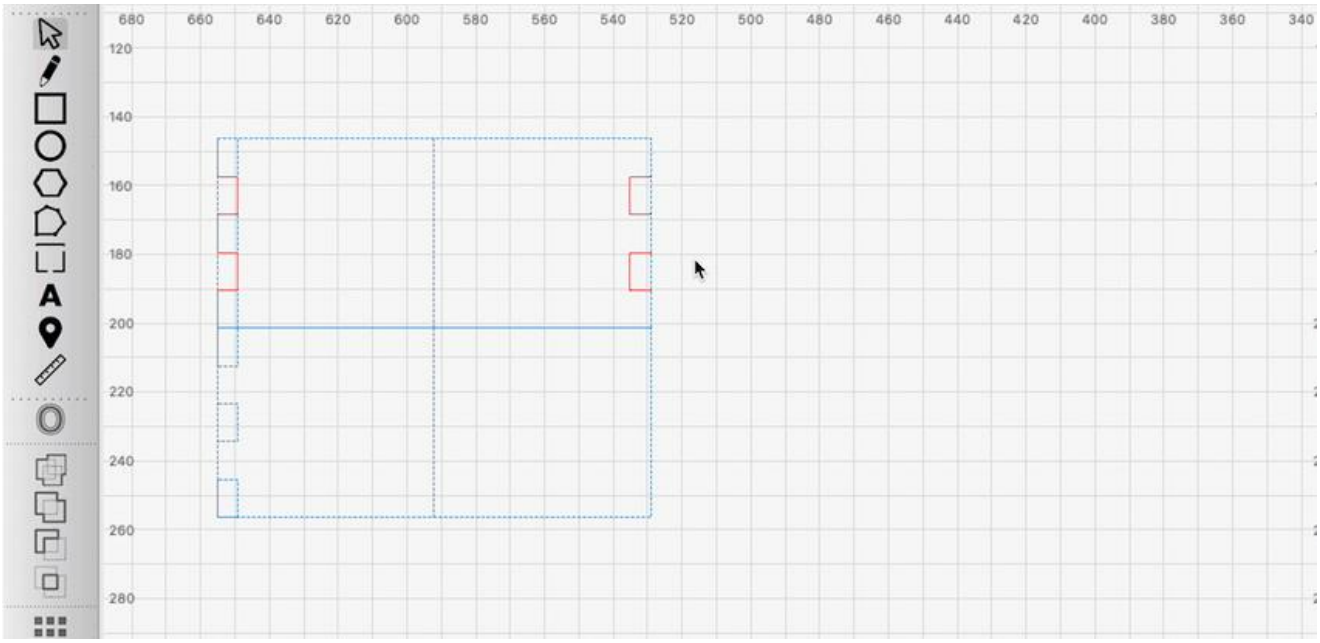
1. Change the selected layer to one you like to use for cutting, and enable the Draw Lines tool.
2. Utilize snapping to start a line at the bottom left corner of the topmost small rectangle — you'll see a crosshair icon when your cursor is over a corner node that you can snap to. Create a line going from that corner to the right corner (again utilizing snapping), then down to the top right of the next rectangle, and finally over to the top left.
3. You should have a red, reversed “C” shape between two of the small rectangles. Notice we're creating only the lines we need to cut away a bit of the graham cracker and create tabs. The outer Tool layer shows us where the graham cracker will be, and where we don't need to cut.
4. Repeat the above process, going from the bottom of the middle rectangle, down to the bottommost rectangle.



5. Select both reversed “C” shapes, then select the line we created earlier down the middle of the large rectangle. You can hold Shift while clicking on additional graphics to add them to your selection without losing your current selection. Order of selection matters for this next step, so make sure you follow that exact order.
6. Go to **Arrange** → Mirror Across Line (`Ctrl`+`Shift`+`M` / `Cmd`+`Shift`+`M`). Duplicate “C” shapes will appear on the opposite side of the large rectangle.



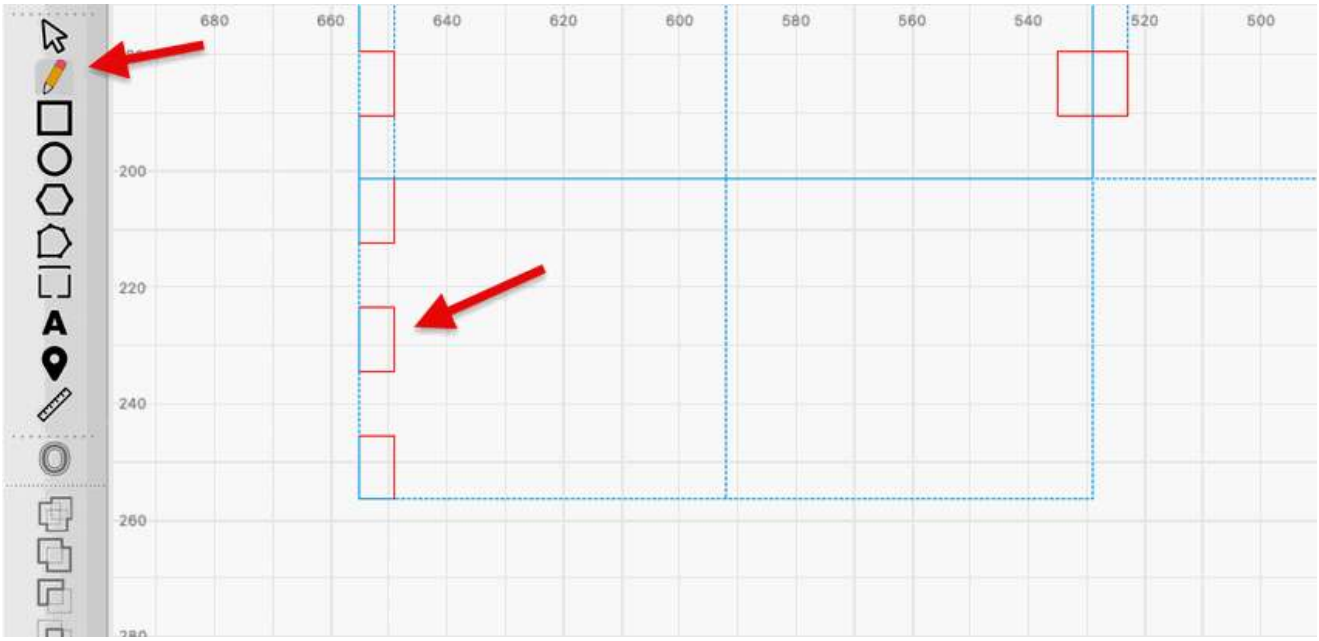
7. Duplicate everything and Group it (`Ctrl`+`G` / `Cmd`+`G`), then set it off to the side. You can snap the center right side of the duplicate to the center left side of the original. We'll use these templates to make the front and back walls of our house, and return to the front wall a bit later to add some decorations.



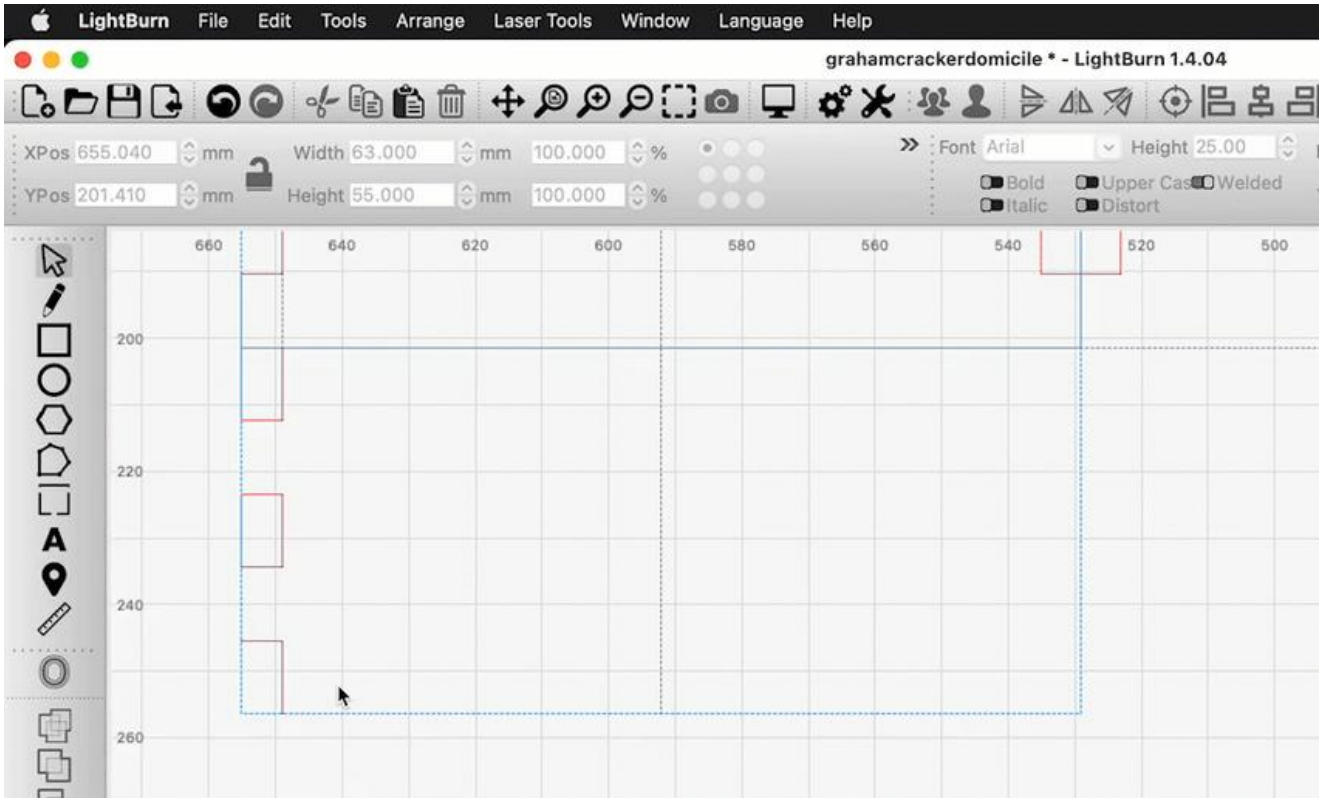
4. Side Walls

Head down to the spare copy of the large rectangle and three small rectangles we made in Step 1E. We'll use this template to create both side walls.

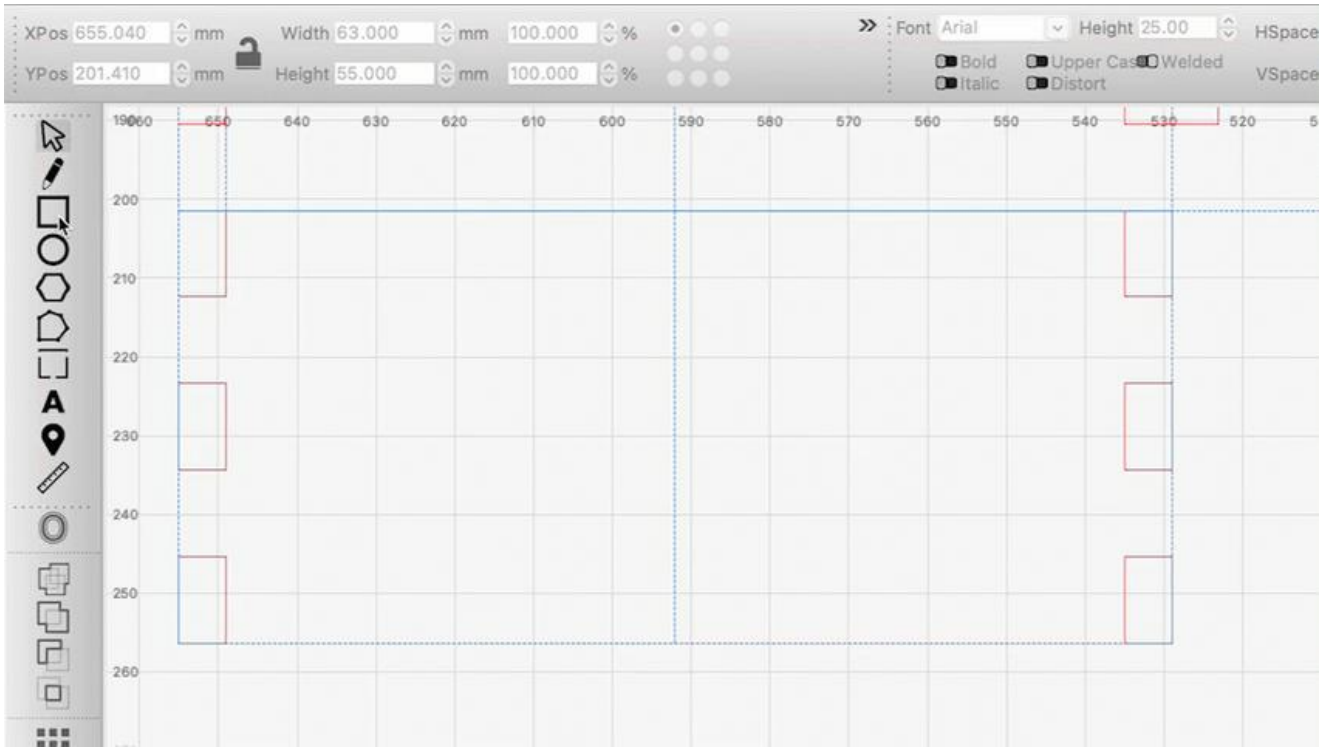
1. Select the cut layer and **Draw Lines** tool again. Create lines along the small rectangles that are the inverse of the tabs we created on the front and back walls — two reversed “L” shapes and one reversed “C” shape.



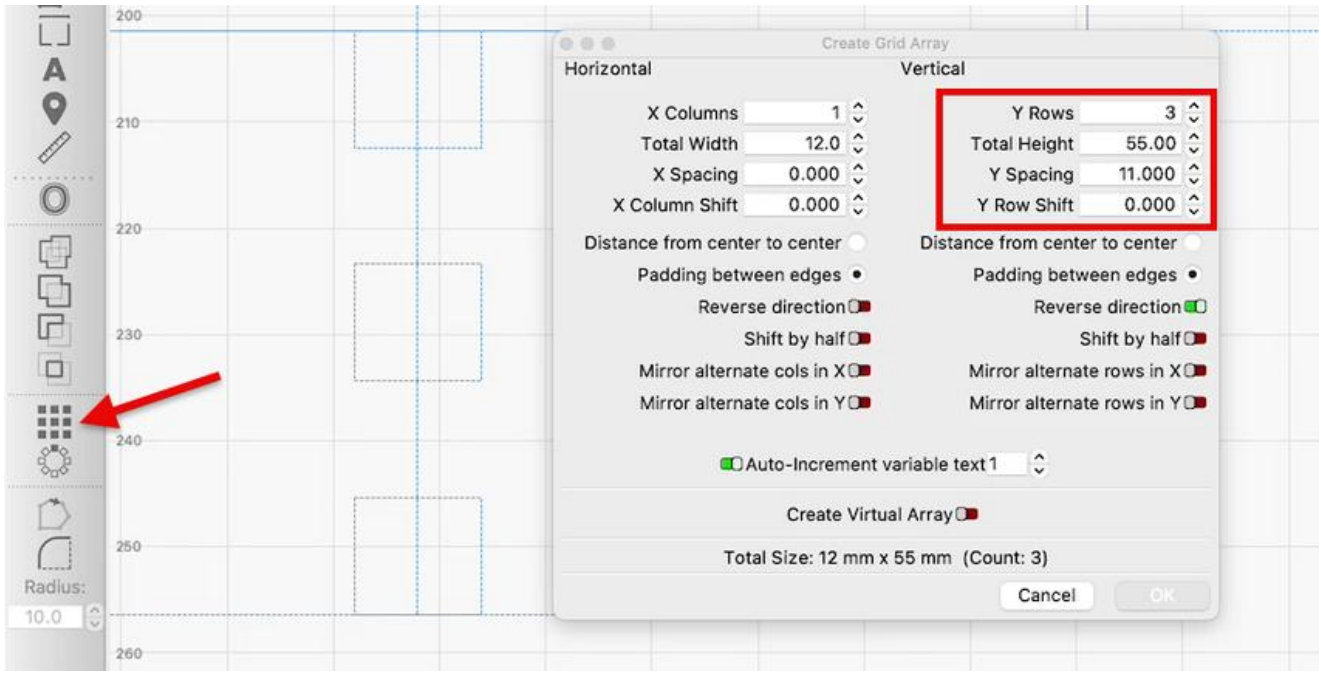
2. Select the three new red lines, then the line down the center of the larger rectangle, and go to **Arrange > Mirror Across Line**.



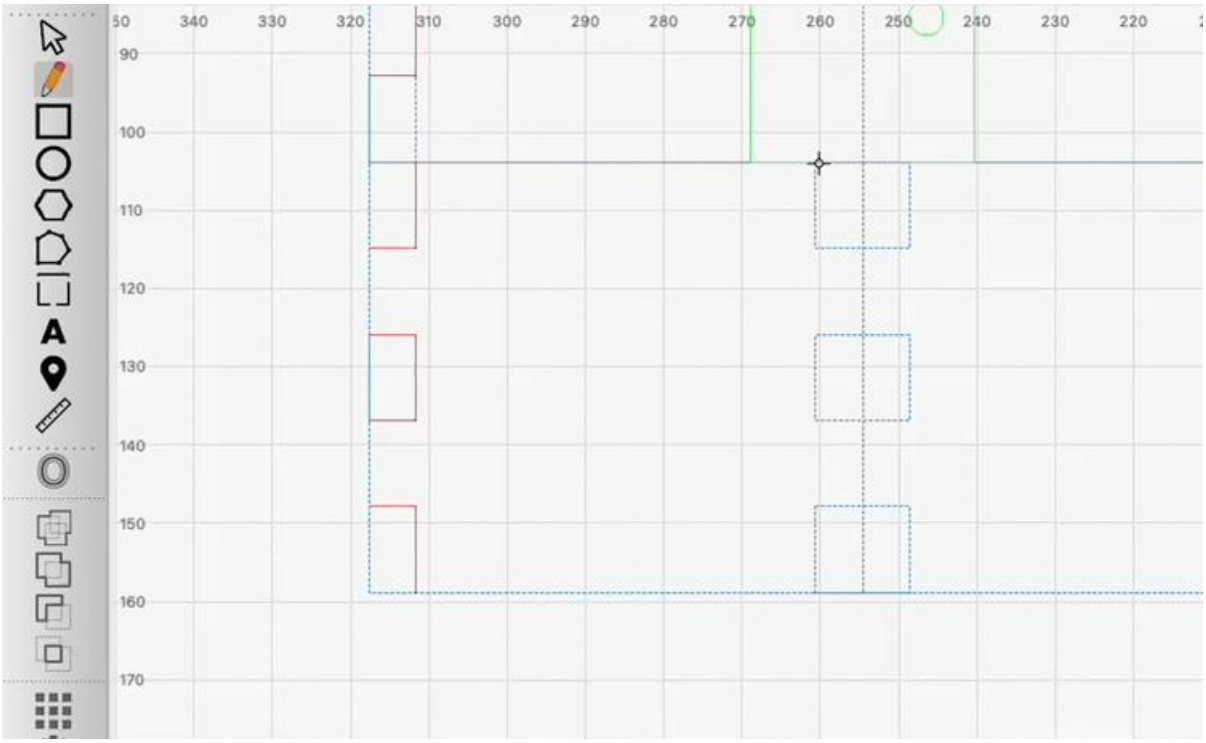
3. We want to divide this rectangle in two to create two sidewalls, so we're also going to add a cutline down the middle to divide the graham cracker, and add tabs in the process.
4. Select a **Tool** layer and create a new rectangle that is twice the width of the small rectangles on the left side, but exactly the same height. Use snapping to align its top center to the top center of the large rectangle.



5. Select the new rectangle, then the Grid Array tool. As before, create 3 **Y Rows**, set **Y Spacing** to match the height of the rectangle, and select **Padding between edges**.



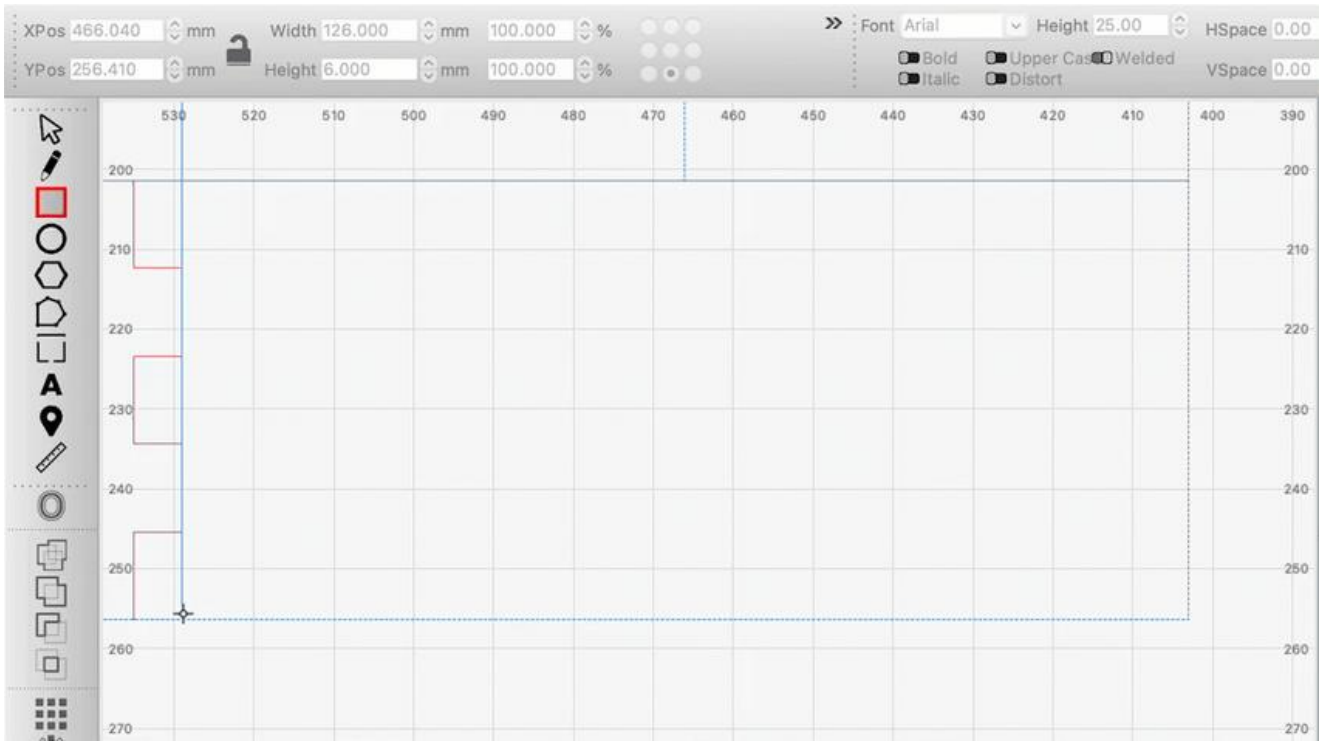
6. Use the **Draw Lines** tool to draw a line following the outline of the new rectangles down the center, starting from the top left and going down, then over to the center, then down again, then back over to a side, down, and so on. Then, reverse the process from the bottom up. You can skip the middle lines connecting the rectangles on the way back up. We've got our sidewalls.



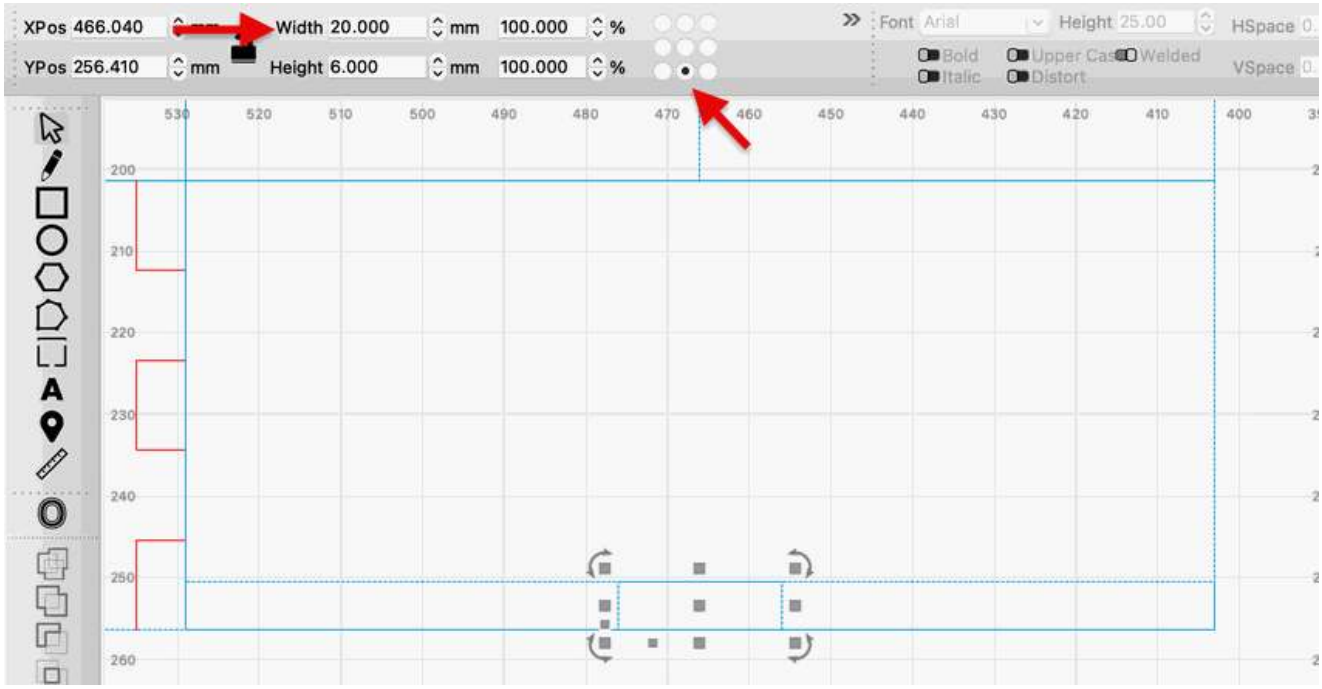
5. Roof Supports

Next up, we'll make two walls to support the roof, starting from our basic graham cracker sized template.

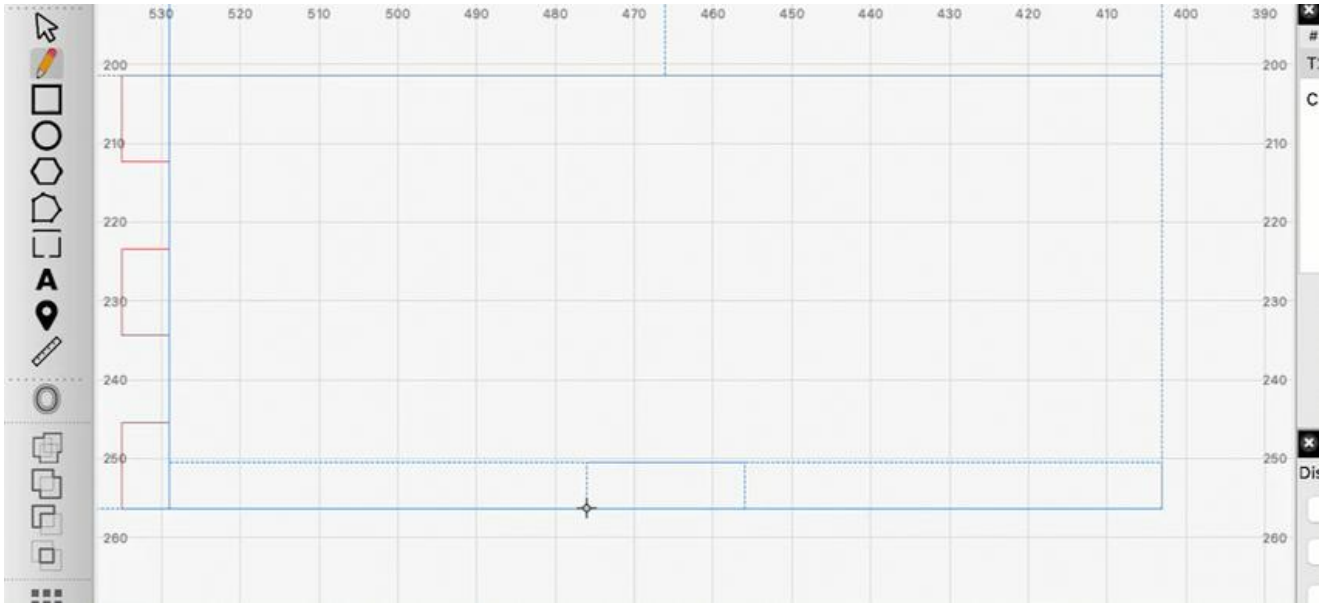
1. With the Tool layer selected, and starting from the bottom left corner, create a new rectangle extending all the way to the other side, making sure to use snapping on both sides. Adjust its height to be a bit more than the thickness of the graham cracker.



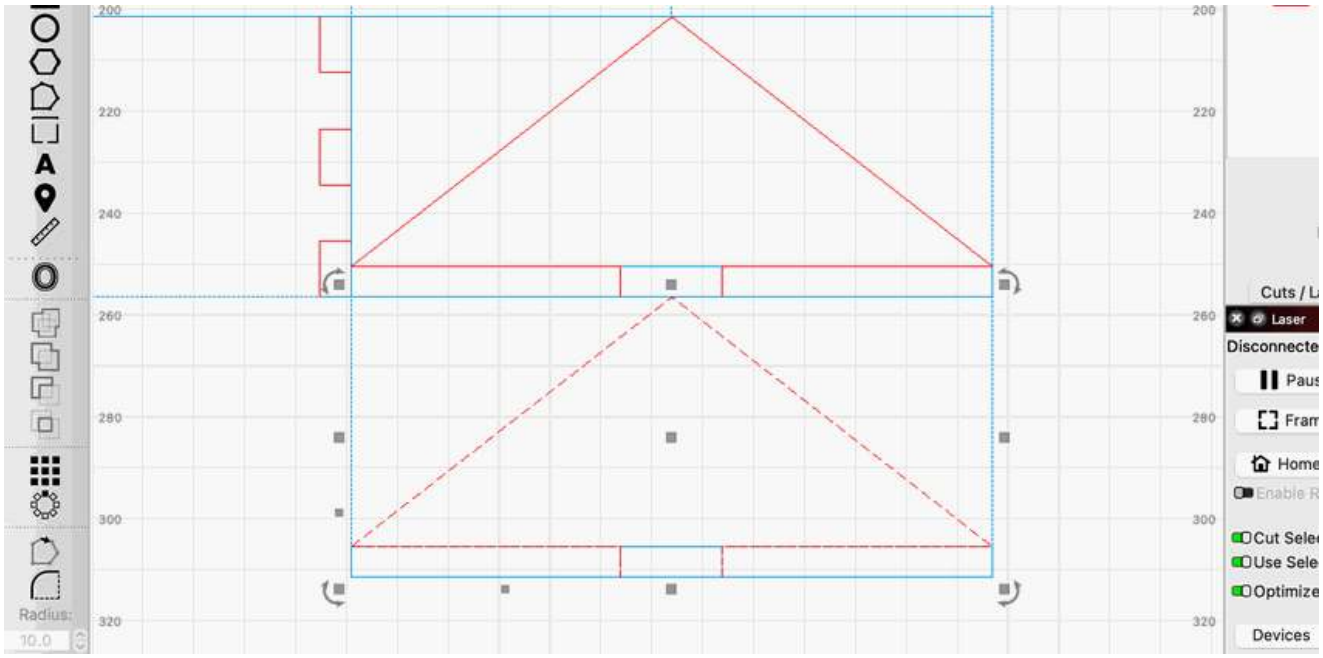
2. **Duplicate** the new rectangle and, with the 9 dot control set to the middle center, set the duplicate's width to 20 mm.



3. Select the cut layer and Draw Lines tool again and, starting from the bottom left of the smallest rectangle, work your way up, then over, then make a slanted line across to the top center of the large rectangle, and then the reverse until you get back to the bottom right of the smallest rectangle.



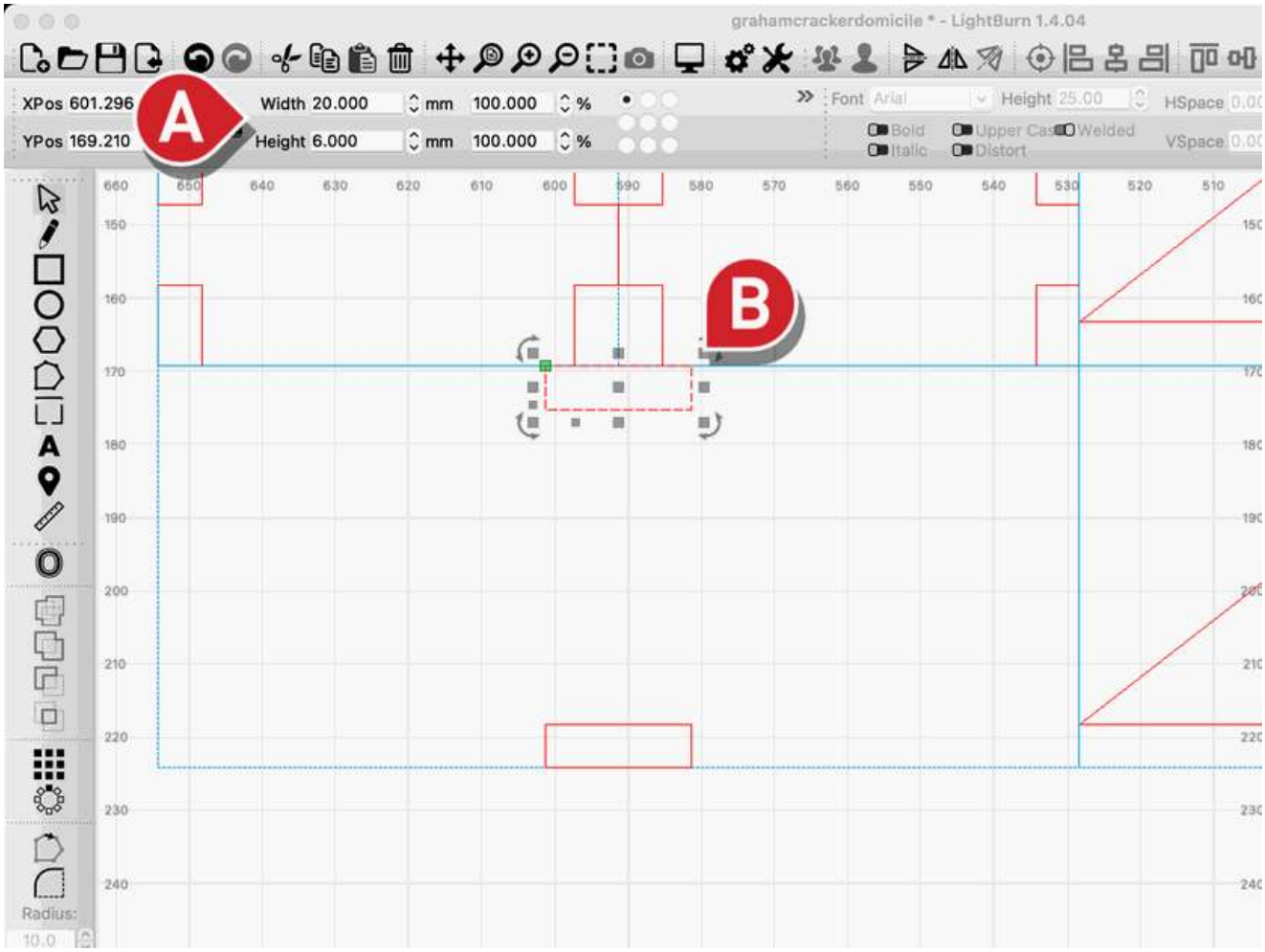
4. To finish the roof supports, **Duplicate** that entire template and snap it to the bottom of the original.



6. Interior Ceiling

Start with another of the graham cracker sized rectangles. This one will serve as the inner ceiling of the house, which will hold the pieces that hold the roof.

1. With a cut layer selected, create a small rectangle and set its height to the thickness of your graham cracker and width to whatever value you set in Step 5B — that was 20 mm in our case. **Duplicate** the small rectangle once you've adjusted its dimensions.
2. Snap the top center of one duplicate to the top center of the large rectangle, and the bottom center of the other to the bottom center of the large rectangle.



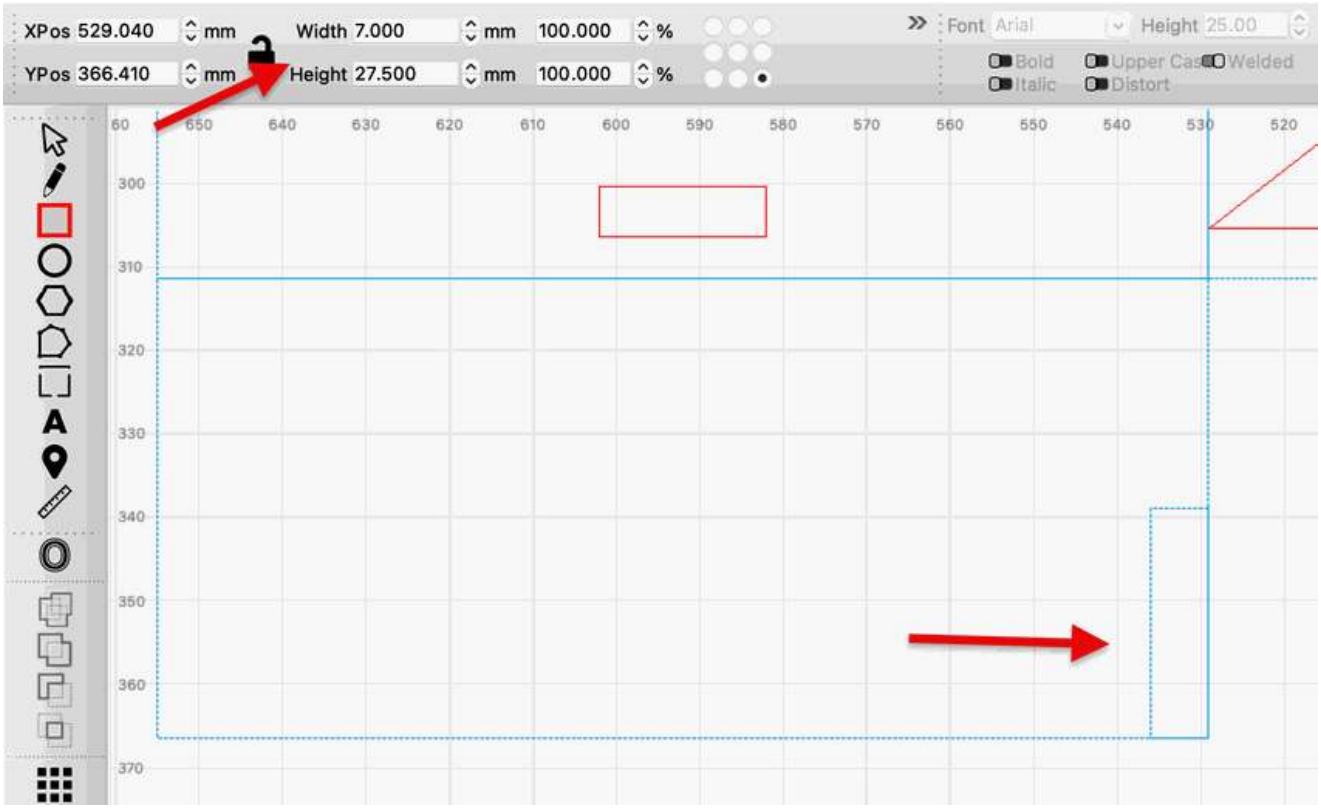
3. Use the arrow keys to nudge the rectangles in a bit toward the center. By default each nudge with an arrow key moves an object 5 mm — one nudge down for the top one and one nudge up for the bottom one will do the trick.



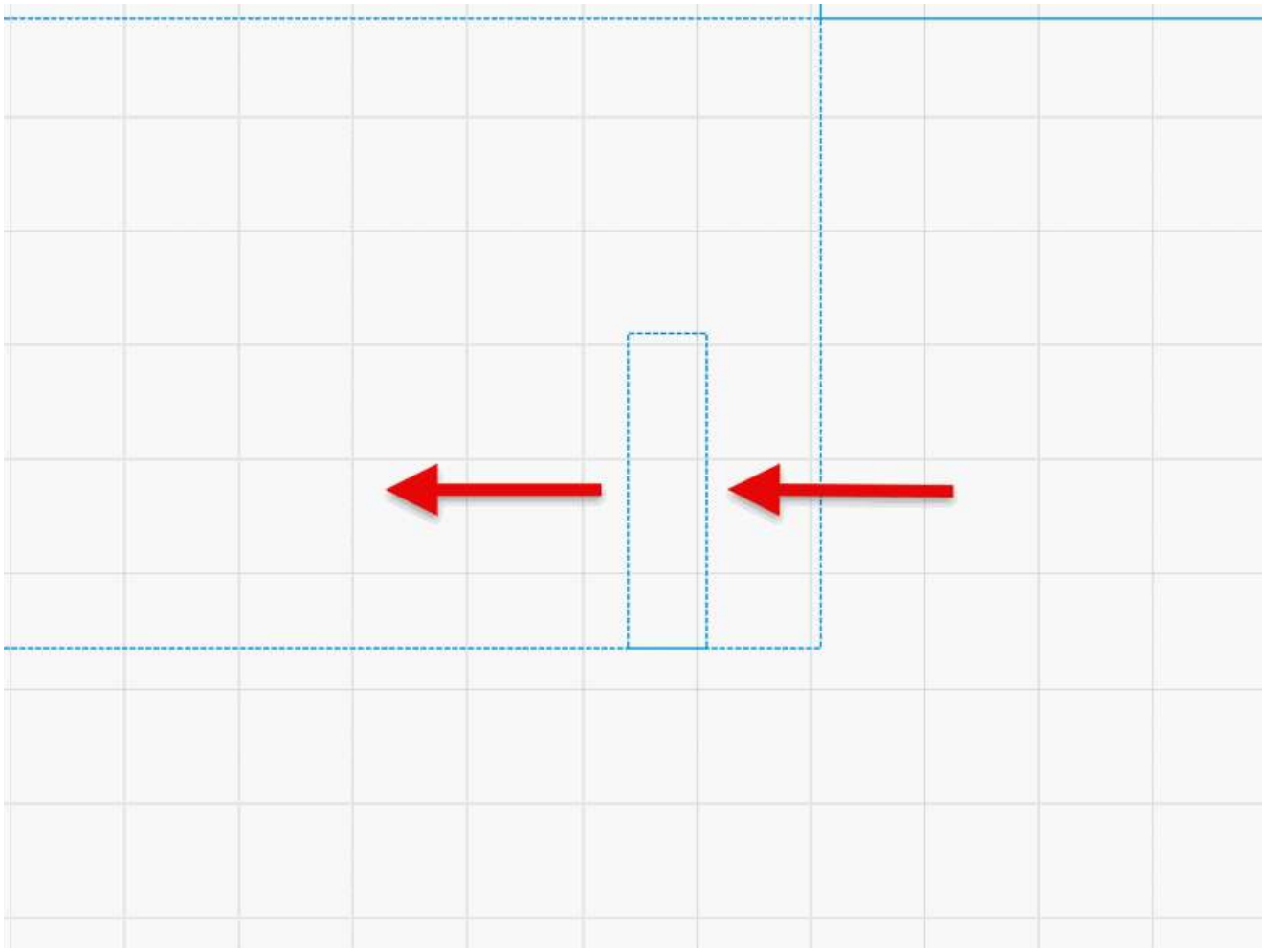
7. Roof

And now, the roof. Start again with a basic graham cracker template.

1. Create a new rectangle, starting from the bottom right corner of the large rectangle. Set its height to half the height of the large rectangle (27.5 mm in our case), and width to a bit more than the thickness of the graham cracker, plus another mm for good measure (7 mm in our case). These slots need a bit of wiggle room for the roof to rest gently on the roof supports.



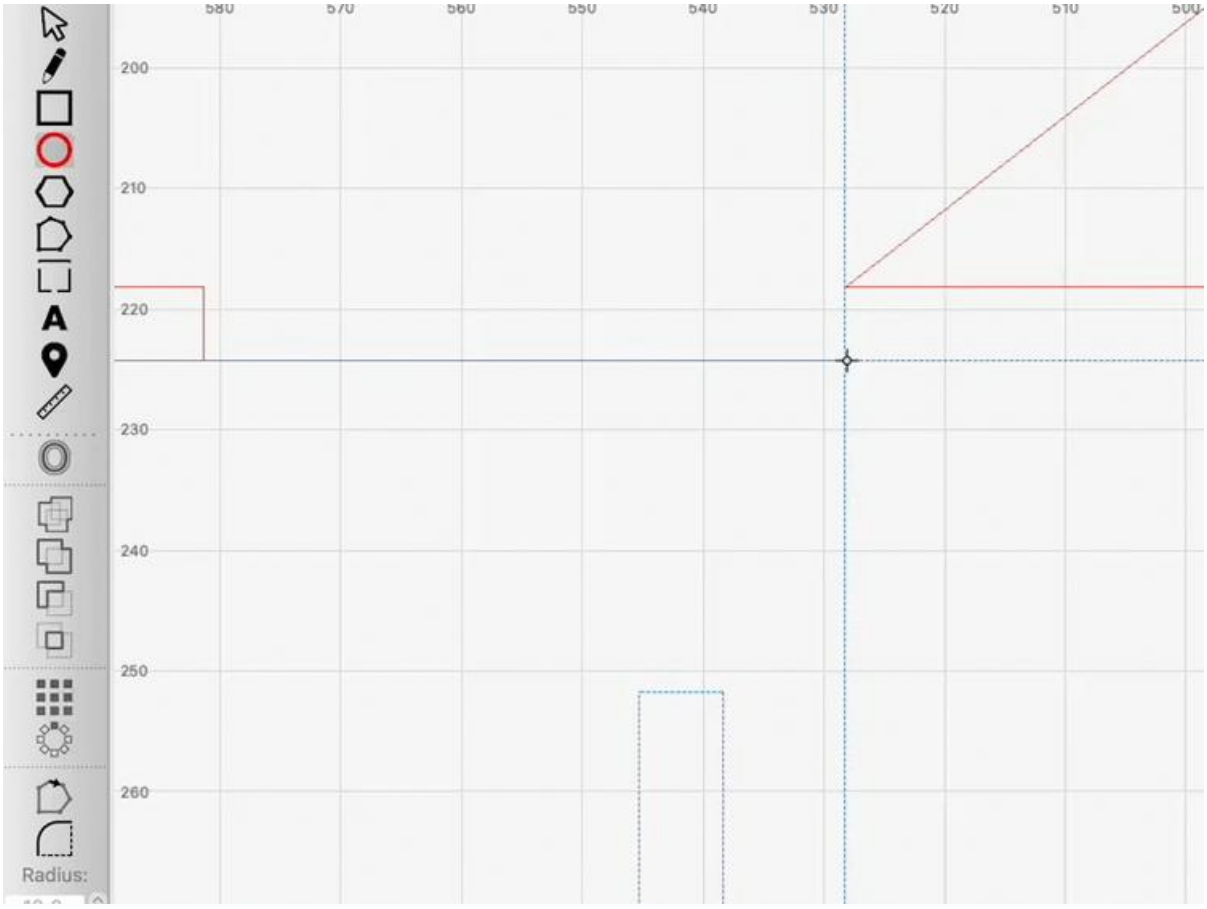
2. Nudge the rectangle twice to the left (10 mm).



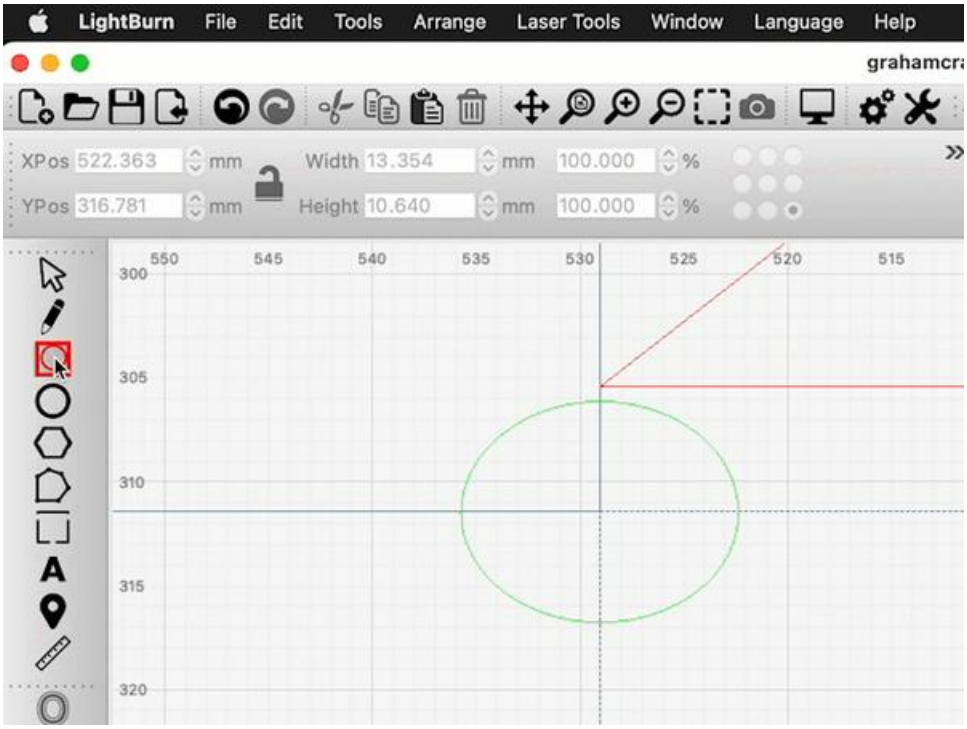
8. Decorate the Roof

Before we add cut lines and duplicate the above roof piece, it's time to start adding decorations.

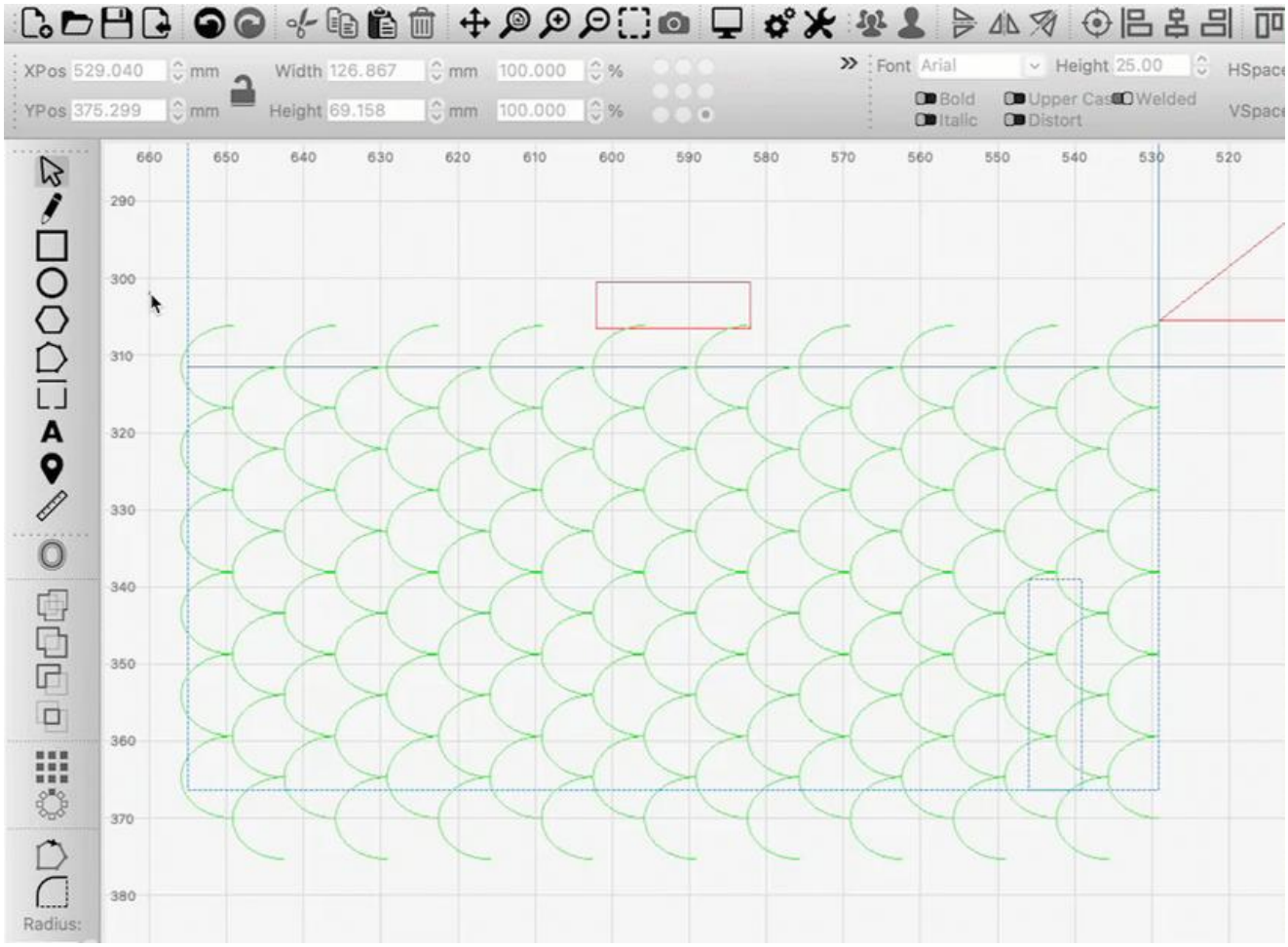
1. Select the Ellipse (`Ctrl`+`E` / `Cmd`+`E`) tool and any layer you like to use for etching (make sure to set the layer to Line mode). Then hover over the top left corner of the roof piece, hold the CTRL / ⌘ key, and click and drag to start an oval centered on that point.



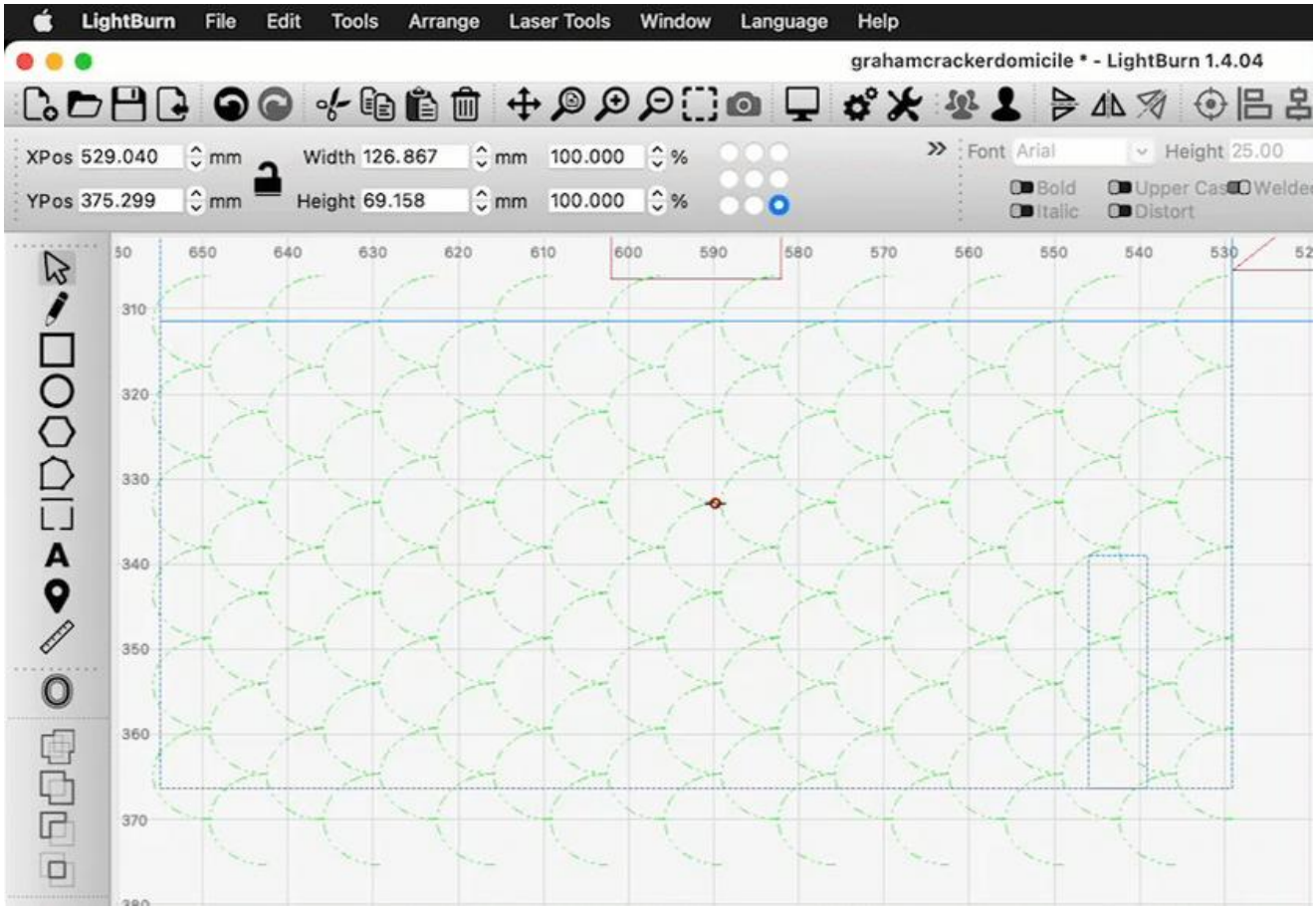
2. Create a rectangle to use as a cutting shape, starting from the oval's top center and extending to its bottom center, and wide enough that it contains the entire half of the oval to the right. Select the oval, then the line, and go to **Tools** → Cut Shapes (`Alt`+`C` / `Option`+`C`). The oval will be cut in half — select and delete the half on the right. The rectangle we used as a cutting shape will disappear.



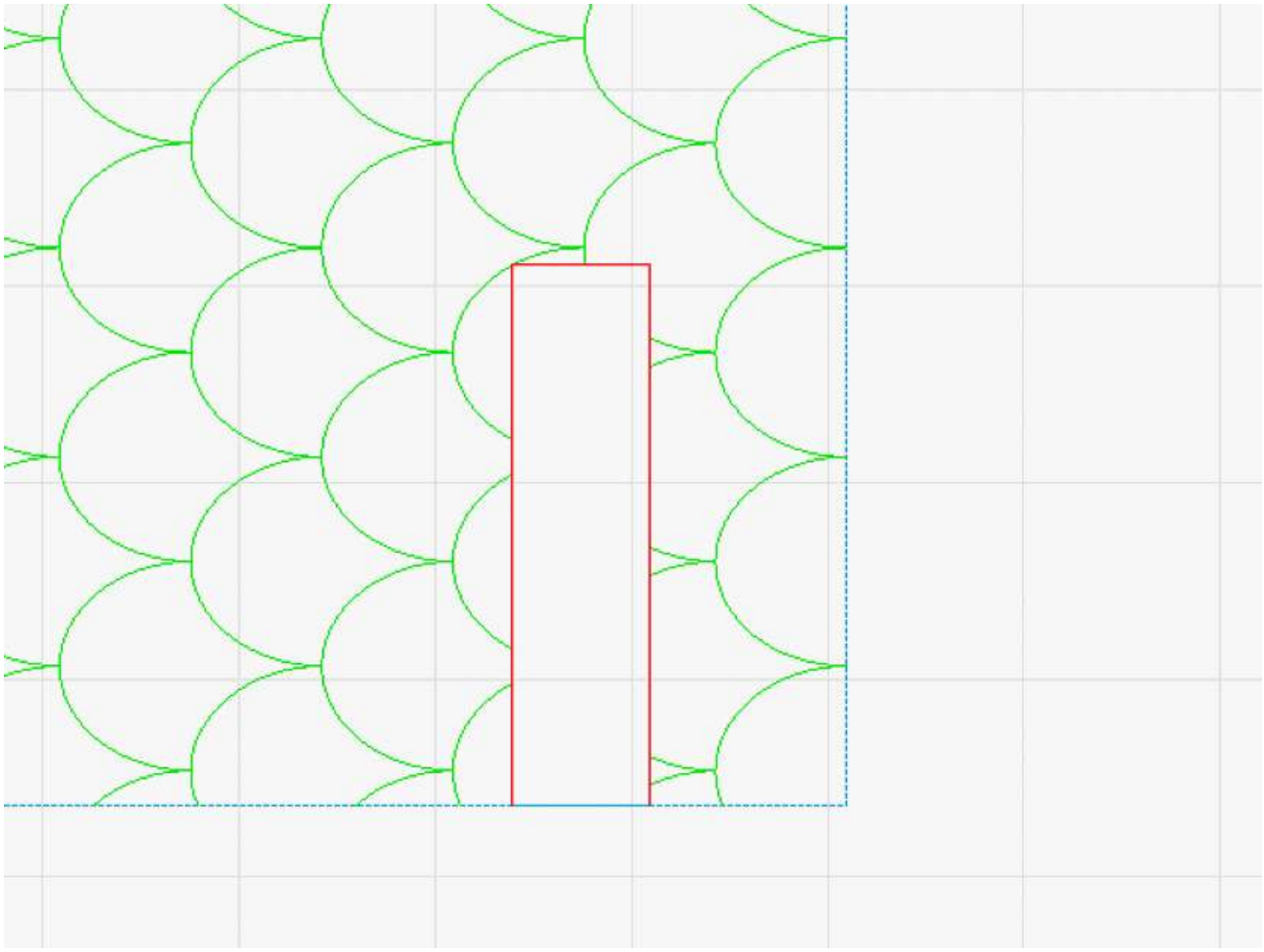
3. Select the remaining half and then click the Grid Array tool. Create enough **X Columns** to duplicate the half-oval all the way to the other end of the large rectangle, and enough **Y Rows** to duplicate them all the way to the bottom. Select **Shift by half** in the X dimension.
4. Select and Group the resulting array. To do that it's helpful to Drag Select the entire template from the right — grabbing from the right selects only those graphics entirely contained in the box selection. Then hold `Ctrl` / `Cmd` while clicking on both the large rectangle and smaller rectangle to deselect them. Finally, with only the arrayed graphics selected, press the **Group** button.



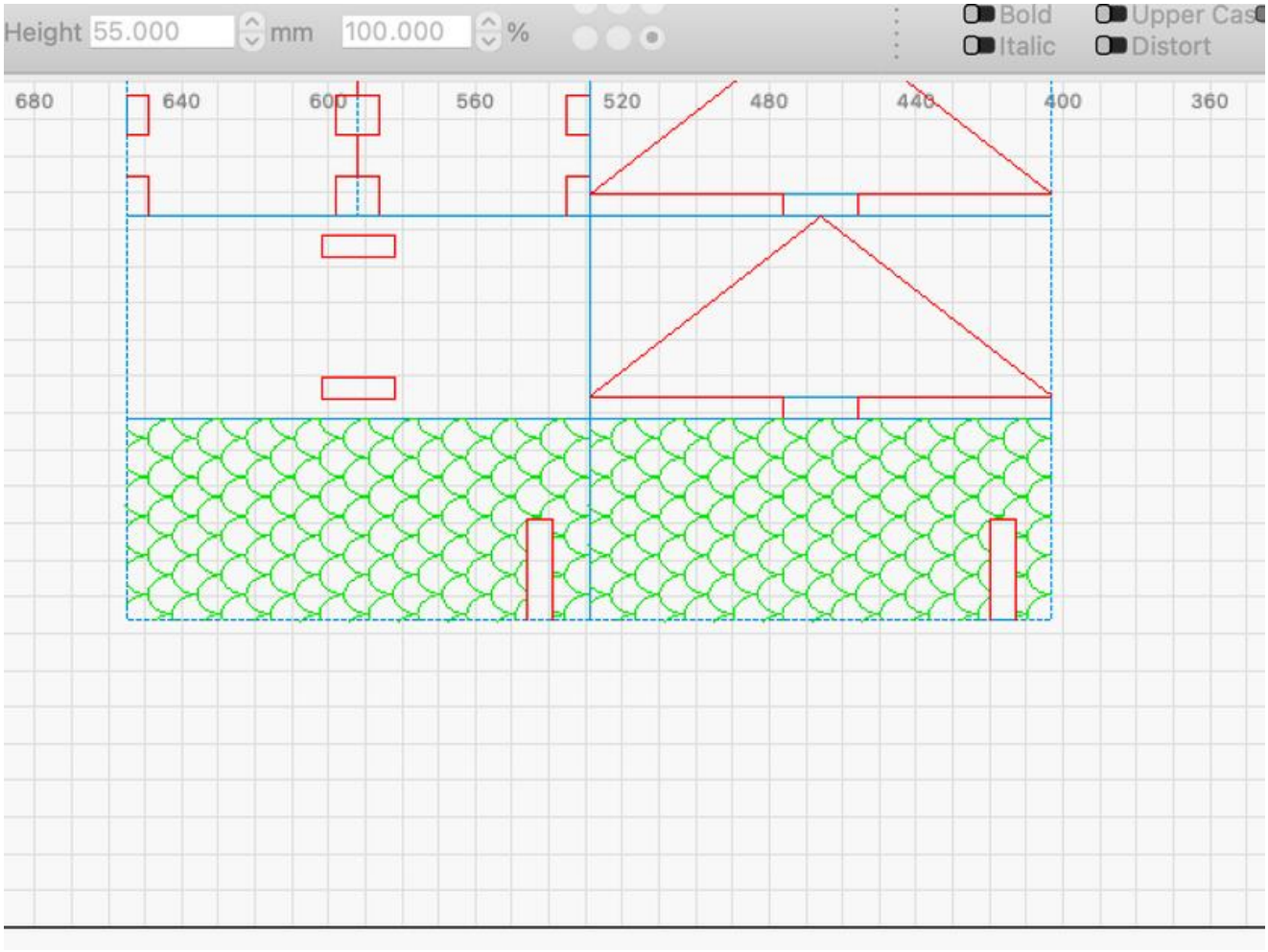
5. Select both rectangles and **Duplicate** them. We're going to use them as cutting shapes, but we'll still want the originals around afterward.



6. Select the arrayed tile graphics and then the large rectangle, and go to **Tools** → Cut Shapes. Delete the remainder that's outside the large rectangle. Re-select the tile array and then the small rectangle, and use the **Cut Shapes** tool again. Delete the remainder inside the small rectangle.



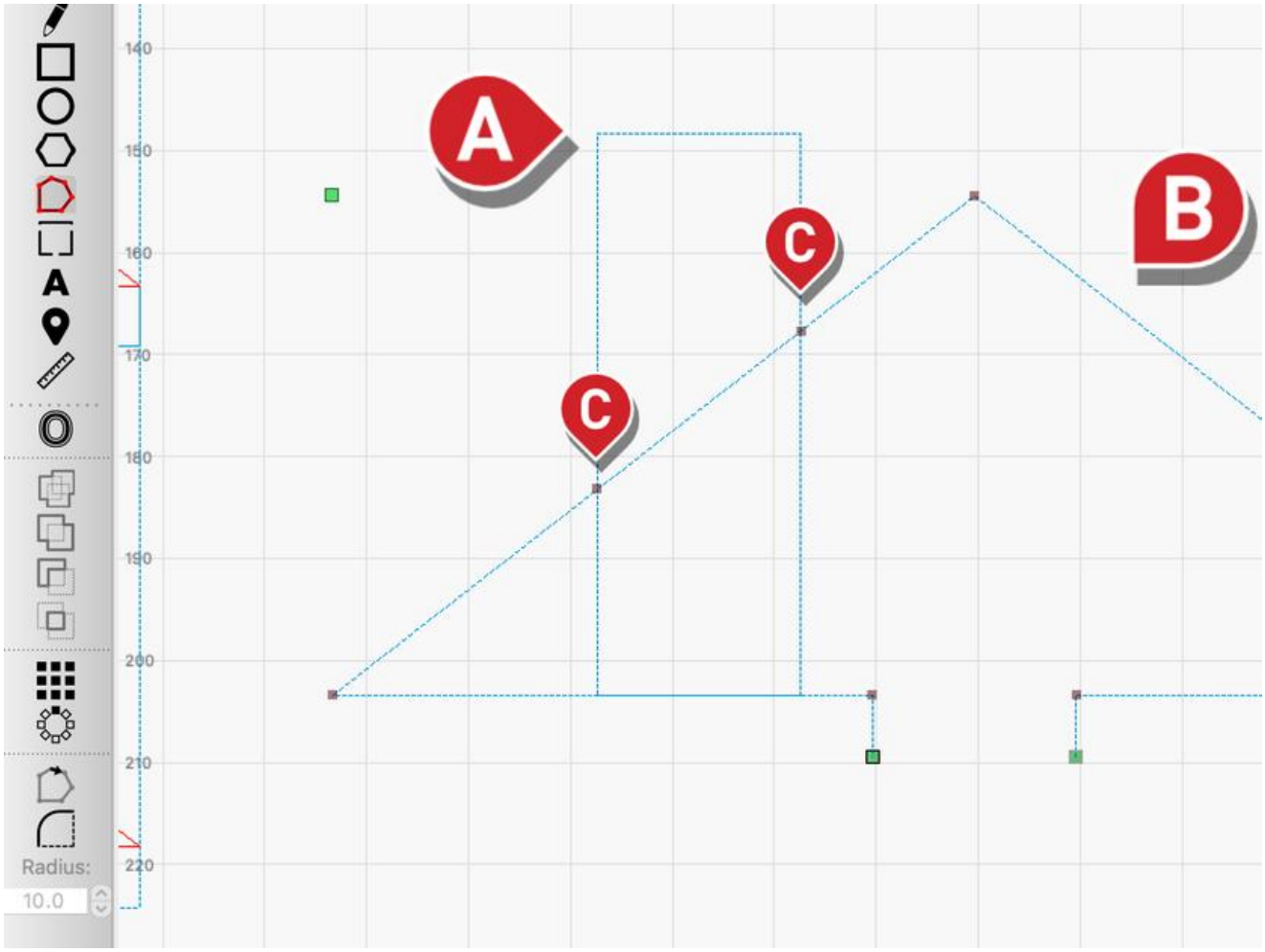
7. Now select the cut layer and draw lines around 3 sides of the smaller rectangle — every side but the one that's overlapping with the outer edge of the large rectangle.
8. Select everything, **Duplicate** it, and snap it to the side of the original. These are our two roof panels.



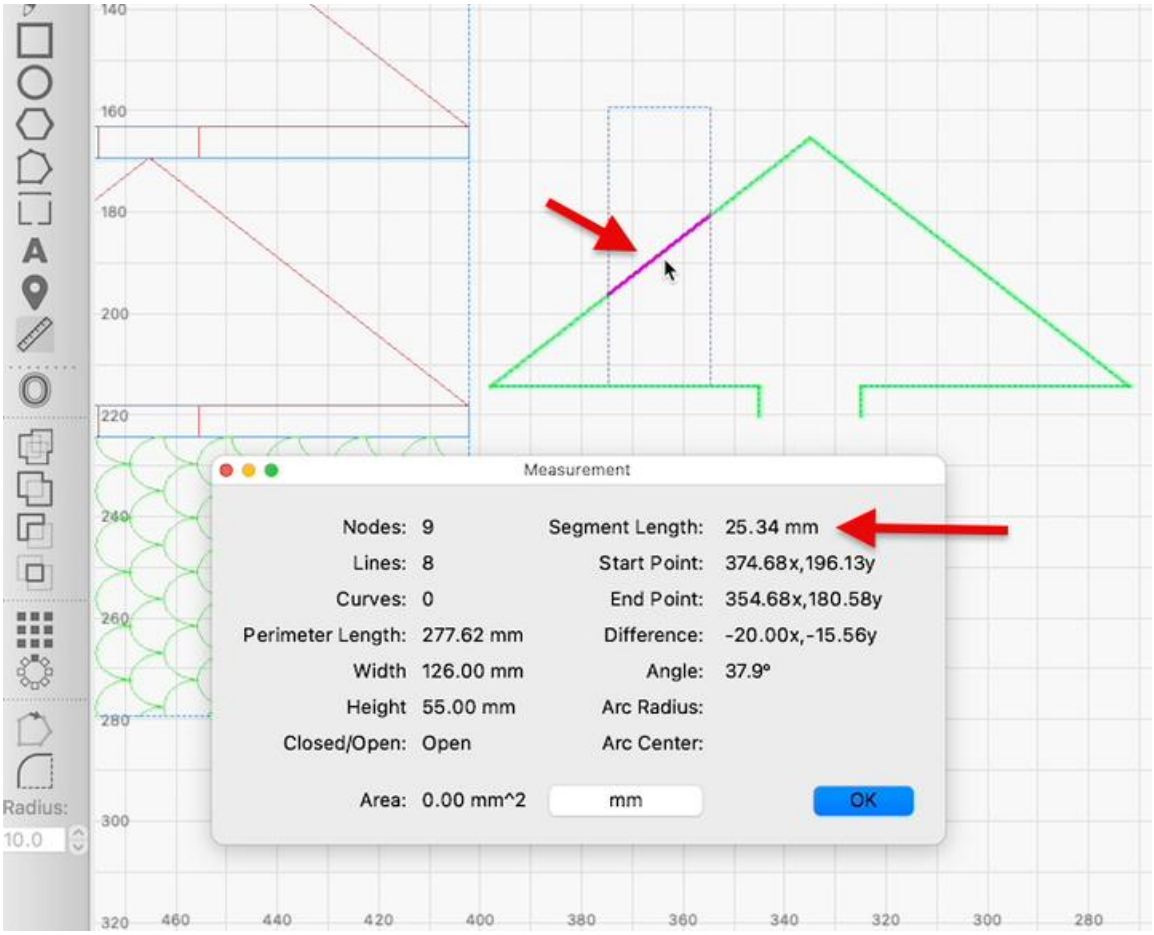
9. Create a Chimney Slot

There's one more thing we need to make before we're done with the roof — a hole for a chimney on one of the panels. To make the hole the right size, first we'll need to decide how wide our chimney will be (we decided on 20 mm), and then overlay our roof support to the chimney and measure the length of the slanted line across it.

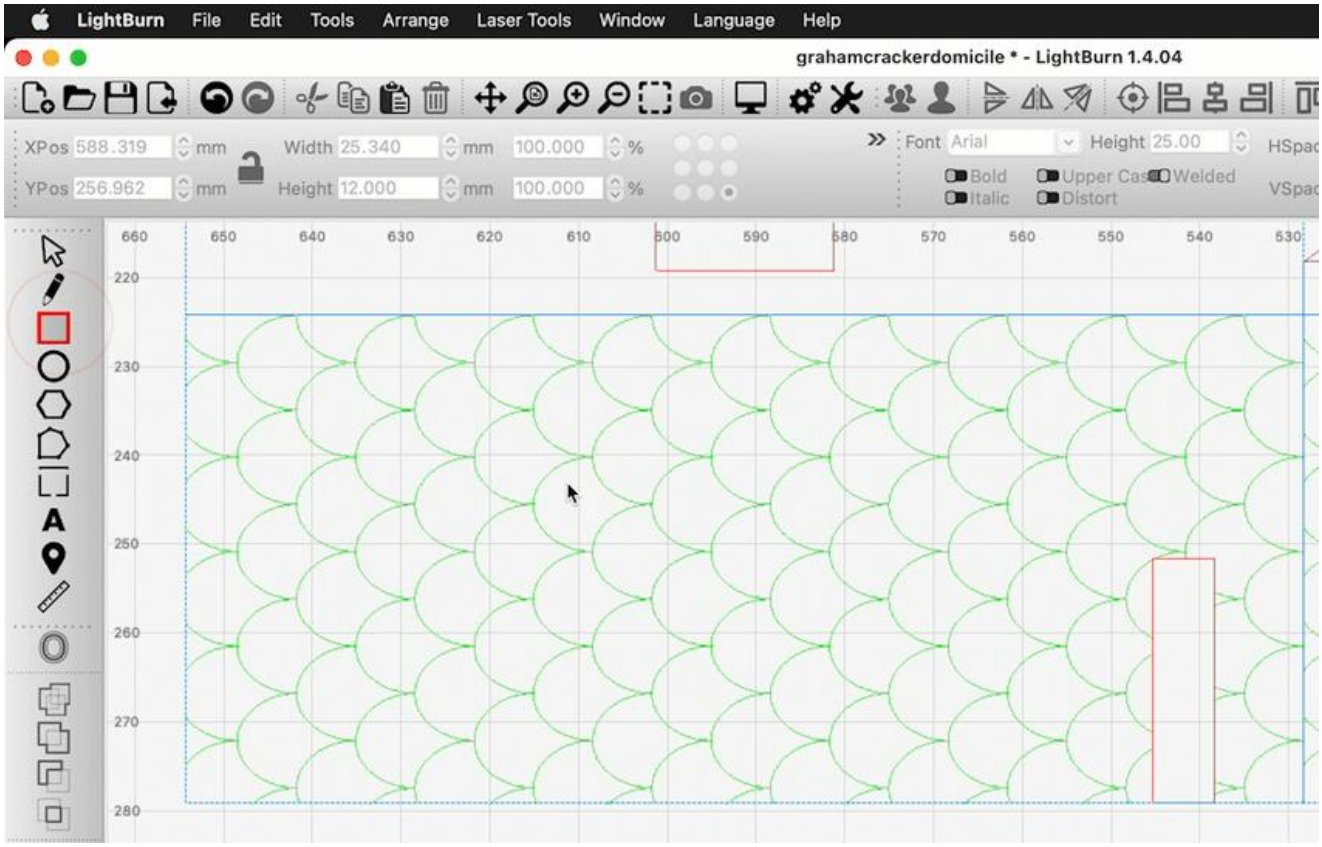
1. Create a 20 mm wide and 55 mm long rectangle (it doesn't need to be quite this long, but this value will ensure the roof support will fit entirely).
2. Then create a **Duplicate** of your roof support and lay it across the rectangle.
3. Next, select the Edit Nodes tool (++ctrl+`++ / ++cmd+`++). Select the roof support, hover over the roofline at both points it intersects with the rectangle, and press **I** to insert nodes at both intersections.



4. Once you've added the nodes, use the Measure tool (`Alt`+`M` / `Option`+`M`) to hover over the line between the rectangle, and record the **Segment Length**. Write the measurement down somewhere and delete these shapes.



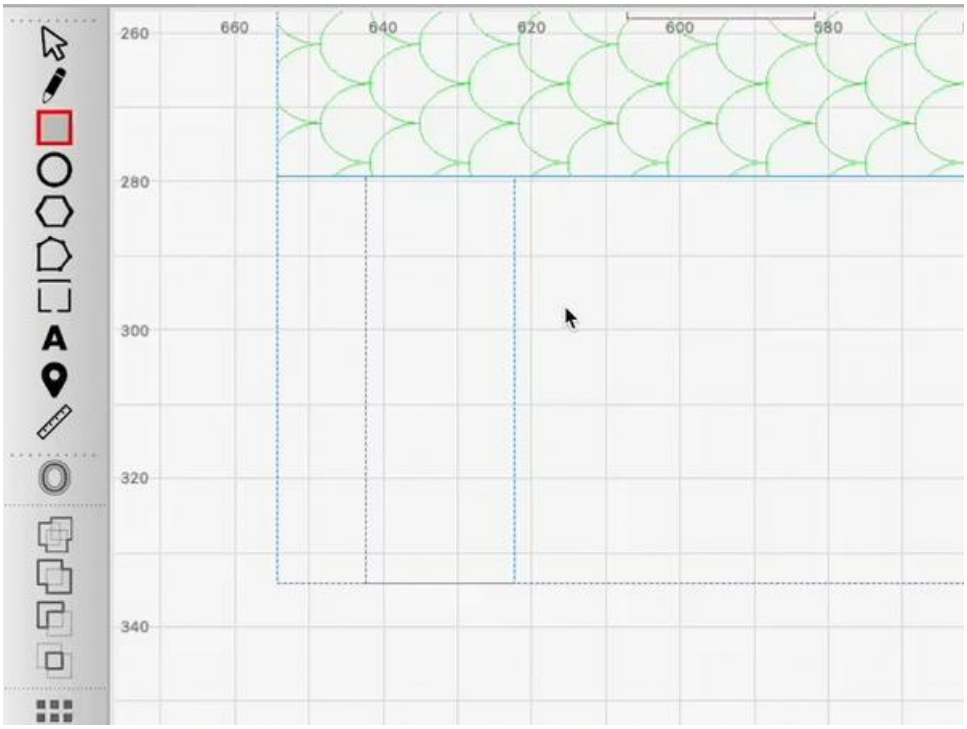
5. On one of the roof panels, create another rectangle, on a cut layer, with its height set to twice the thickness of the graham cracker (12 mm in our case), and width to the length you just recorded (25.34 mm in our case). Position it roughly in the center of the panel, and **Duplicate** it.
6. Select the tile array, then the duplicate of the chimney slot rectangle, and use **Cut Shapes** again. Delete what's left inside the rectangle.



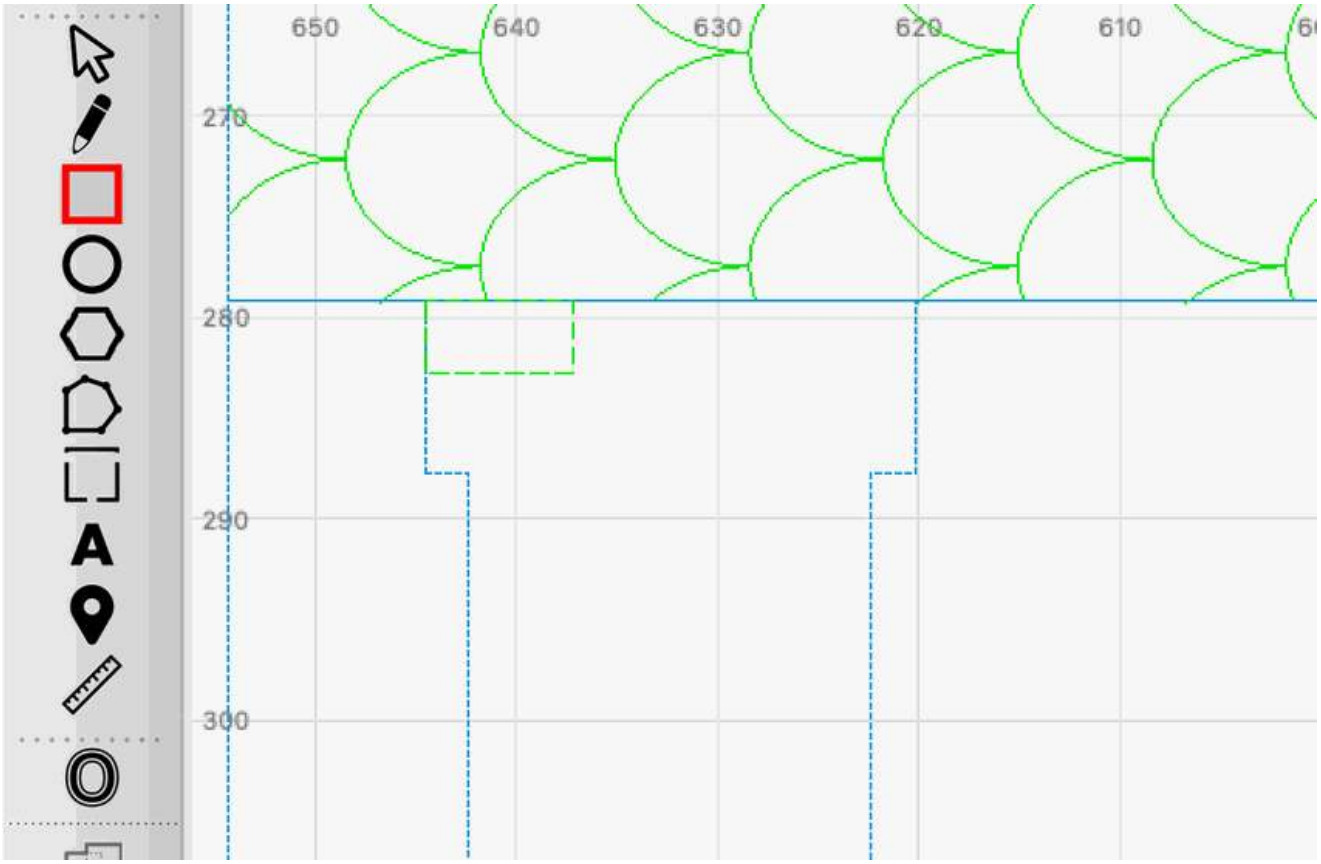
10. Chimney

Making the chimney will require another graham cracker, so make or **Duplicate** another graham cracker sized rectangle.

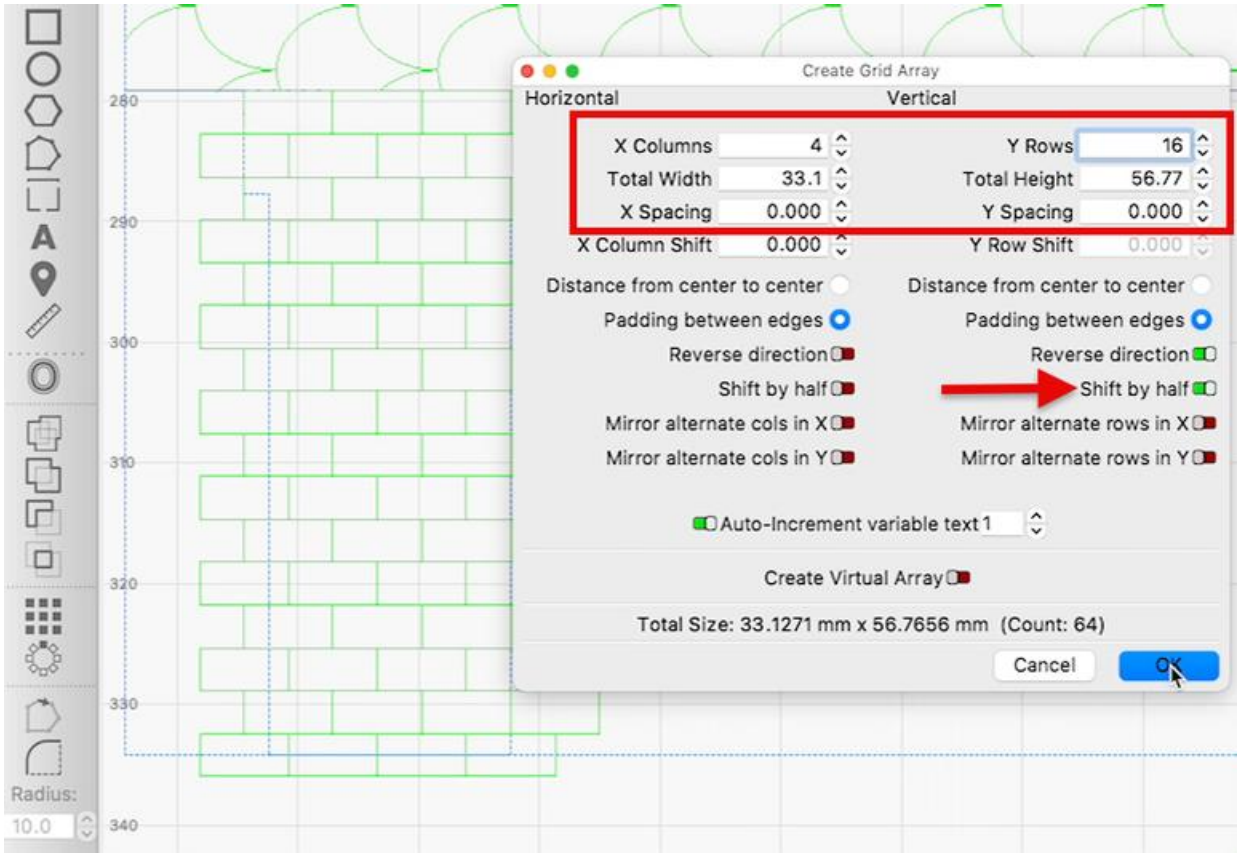
1. Then, with a Tool layer selected, make two rectangles within it. The first should be the width you decided on earlier, and the same height as the full, large rectangle. The second should be wider than the first, and quite a bit shorter — think of the rim around the top of a chimney. Snap the second to the top center of the first.
2. Select both and Weld (`Ctrl`+`W` / `Cmd`+`W`) them together.



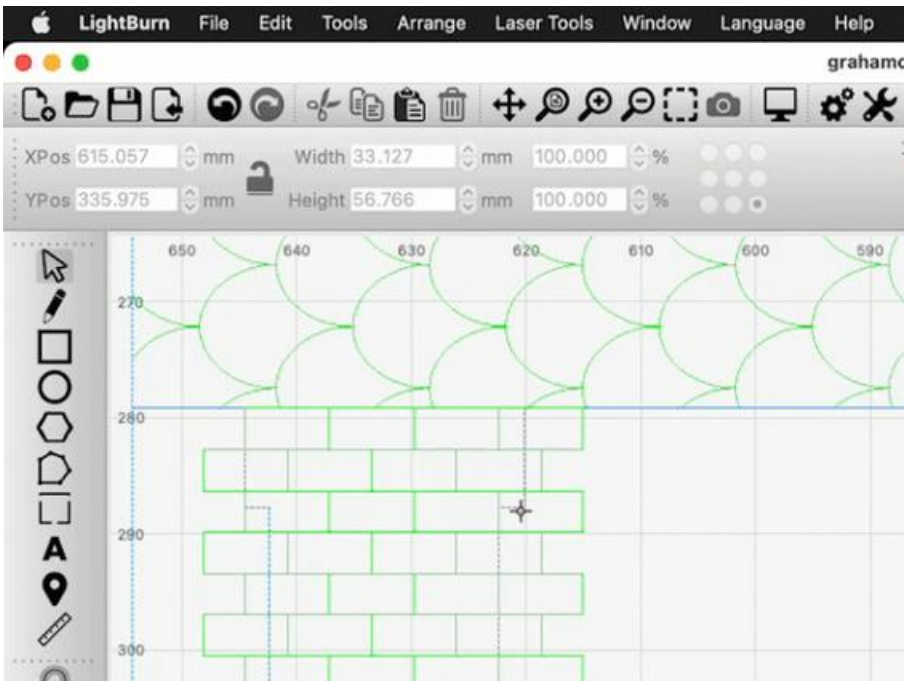
3. Create another small rectangle, set to the etch layer, roughly the size of a “brick” that's proportional to the chimney we've made.



4. Select it, then use the Grid Array tool to array it in sufficient columns and rows that it covers the entire chimney. Enable **Shift by half** in the Y dimension.

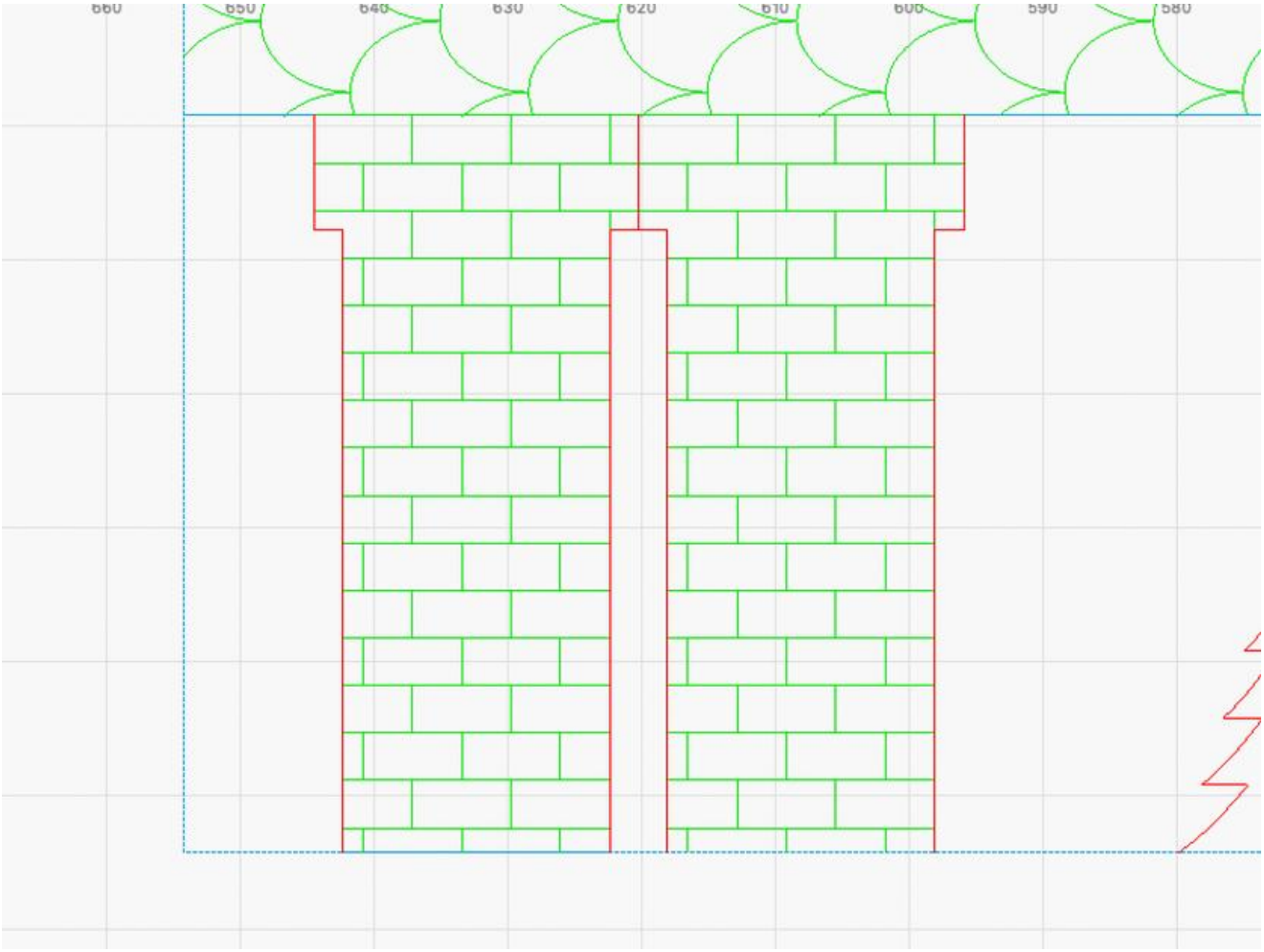


5. Select and group the arrayed bricks, then **Duplicate** the chimney outline. Finally, select the arrayed bricks, then the duplicate chimney, and use Cut Shapes. Delete everything outside the chimney.



6. Draw cut lines on all sides of the chimney except for the two sides that are along the outer edge of the large rectangle.

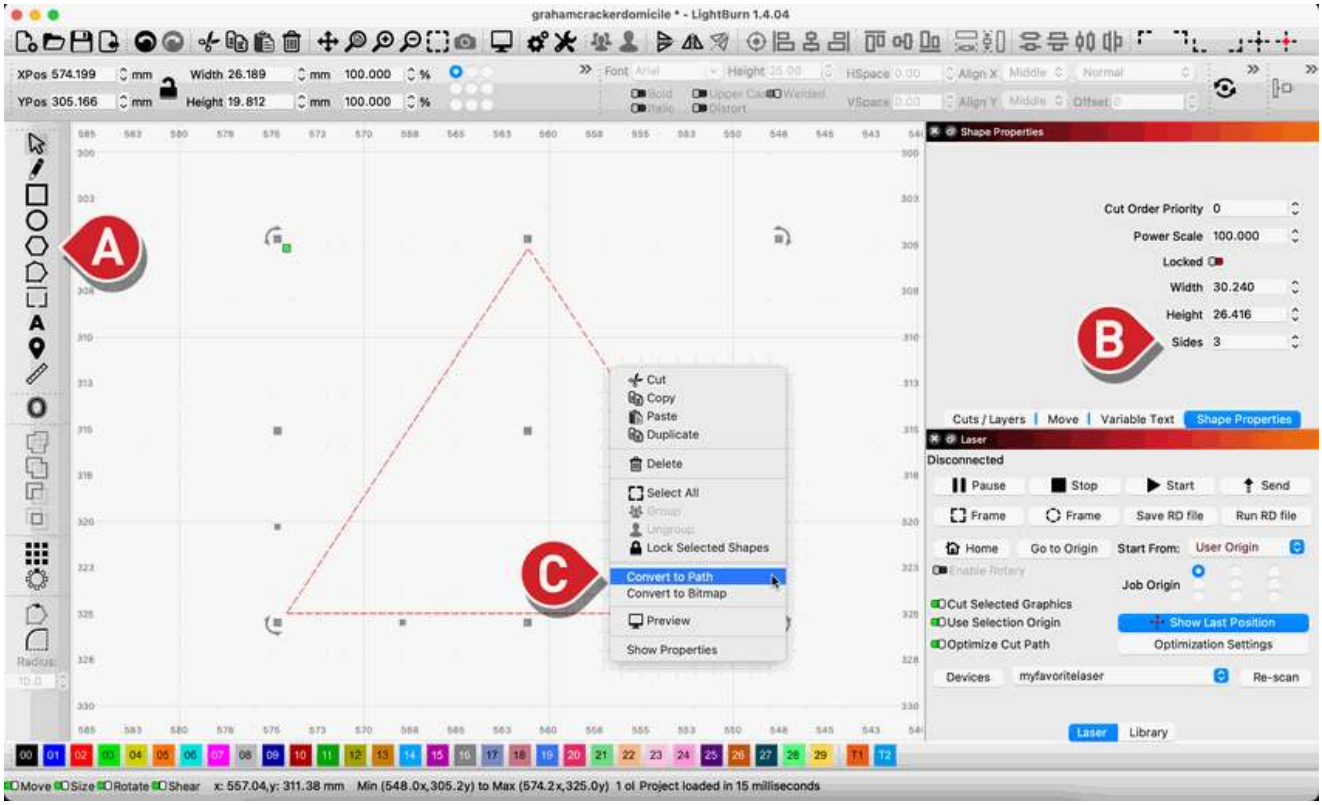
7. Select everything you just made, chimney and bricks, **Duplicate** them together, and move the duplicate to the side of the original.



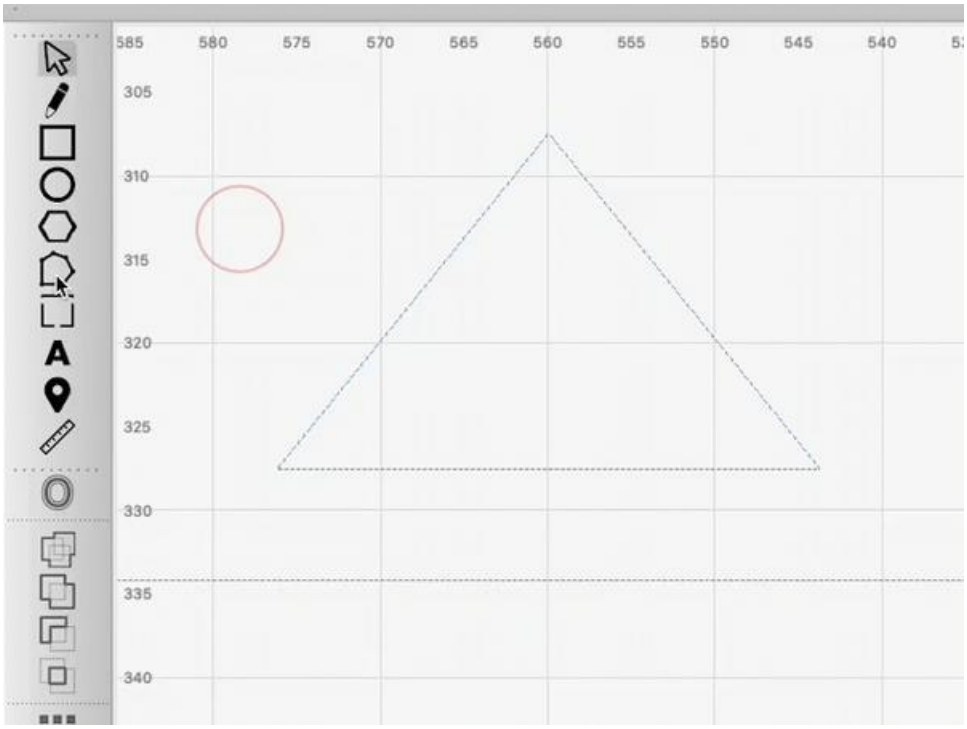
11. Christmas Tree

We've got some space left on this cracker, and we're going to use it to make a Christmas tree.

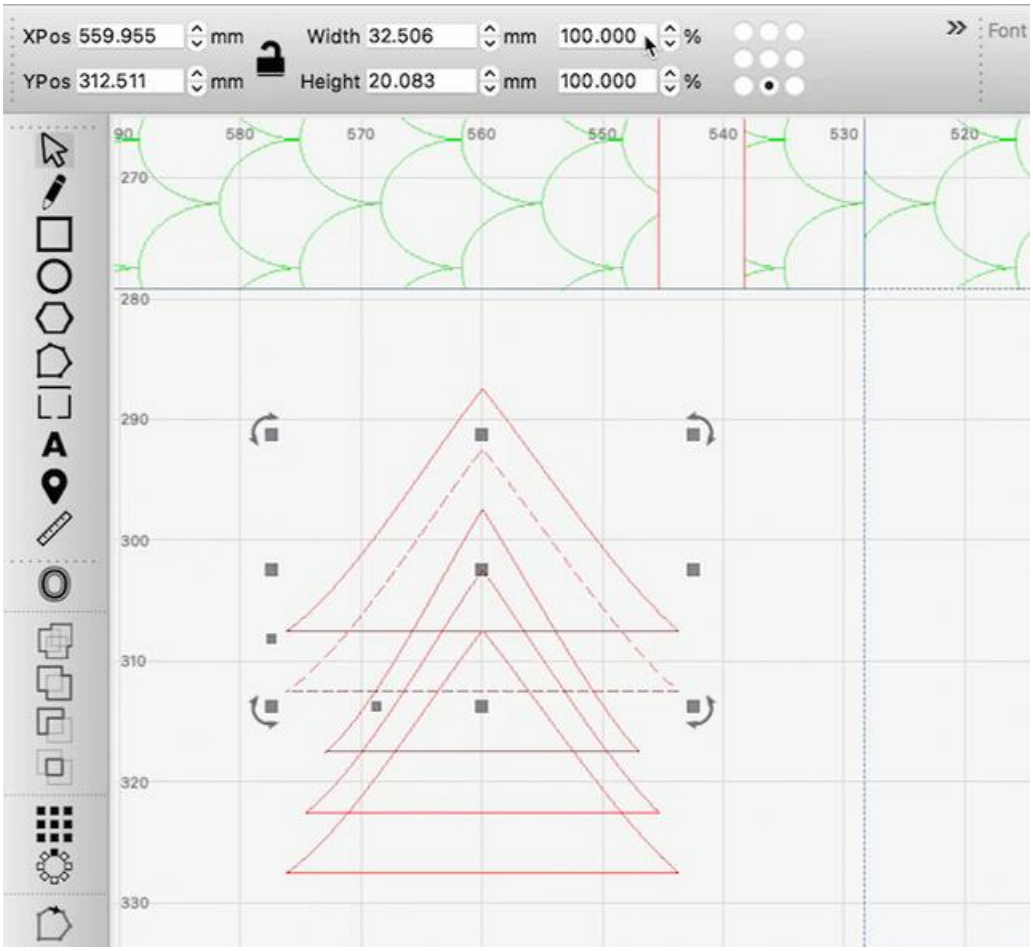
1. Select the Polygon tool then click and drag to create a six sided polygon.
2. Then, with the polygon still selected, go to the Shape Properties window and change the number of Sides to 3 to turn it into a triangle.
3. We're going to use the Edit Nodes tool on the triangle, so first we need to right-click it and select Convert to Path (`Ctrl`+`Shift`+`C` / `Cmd`+`Shift`+`C`).



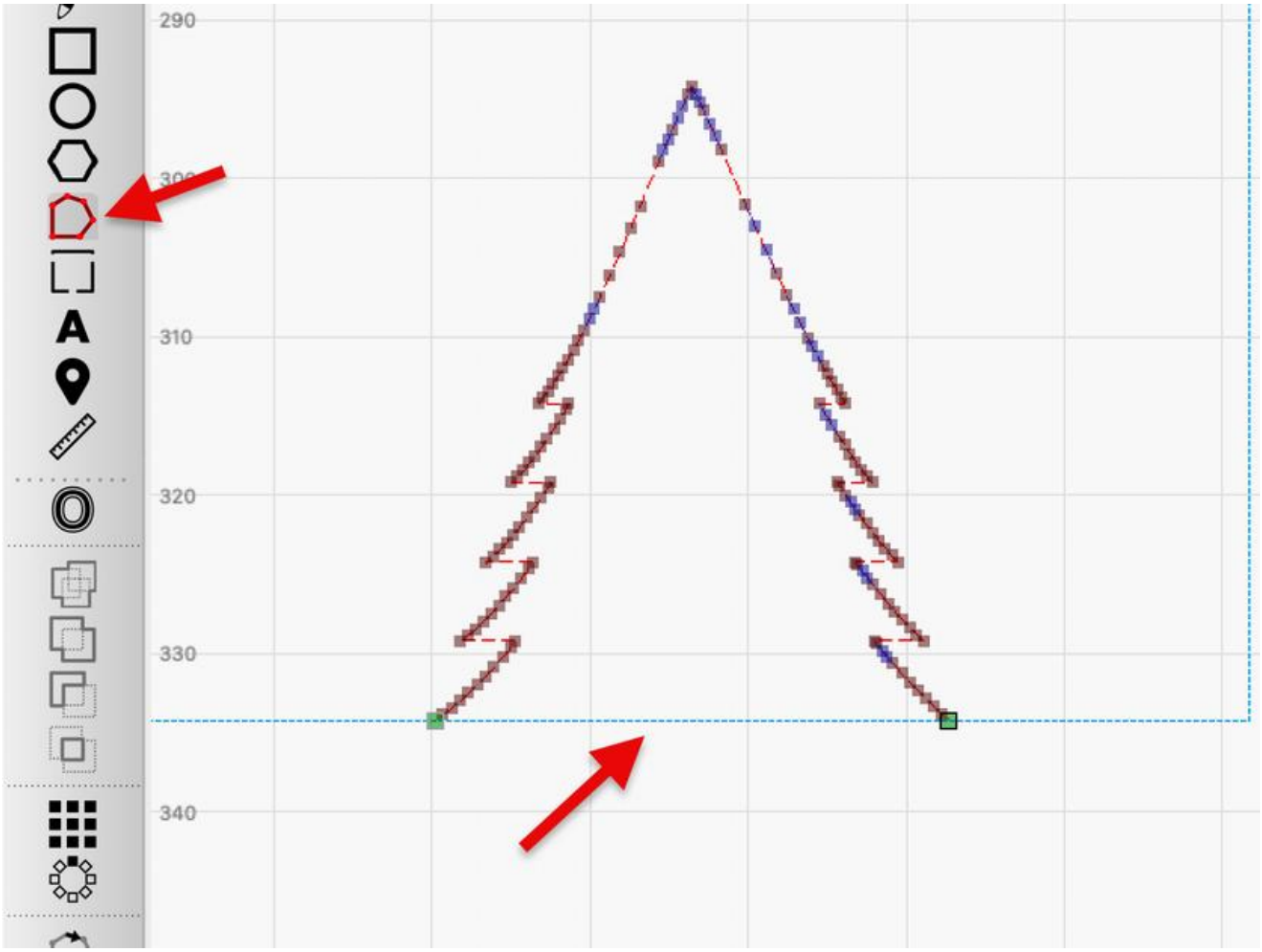
4. Then, select the **Edit Nodes** tool and the triangle. Hover over the lines on each side and press S to turn them into curves.
5. You'll see four handles appear on either side, pull the bottom one on each side down so the lines curve in a bit. They don't need to be perfectly symmetrical on each side.



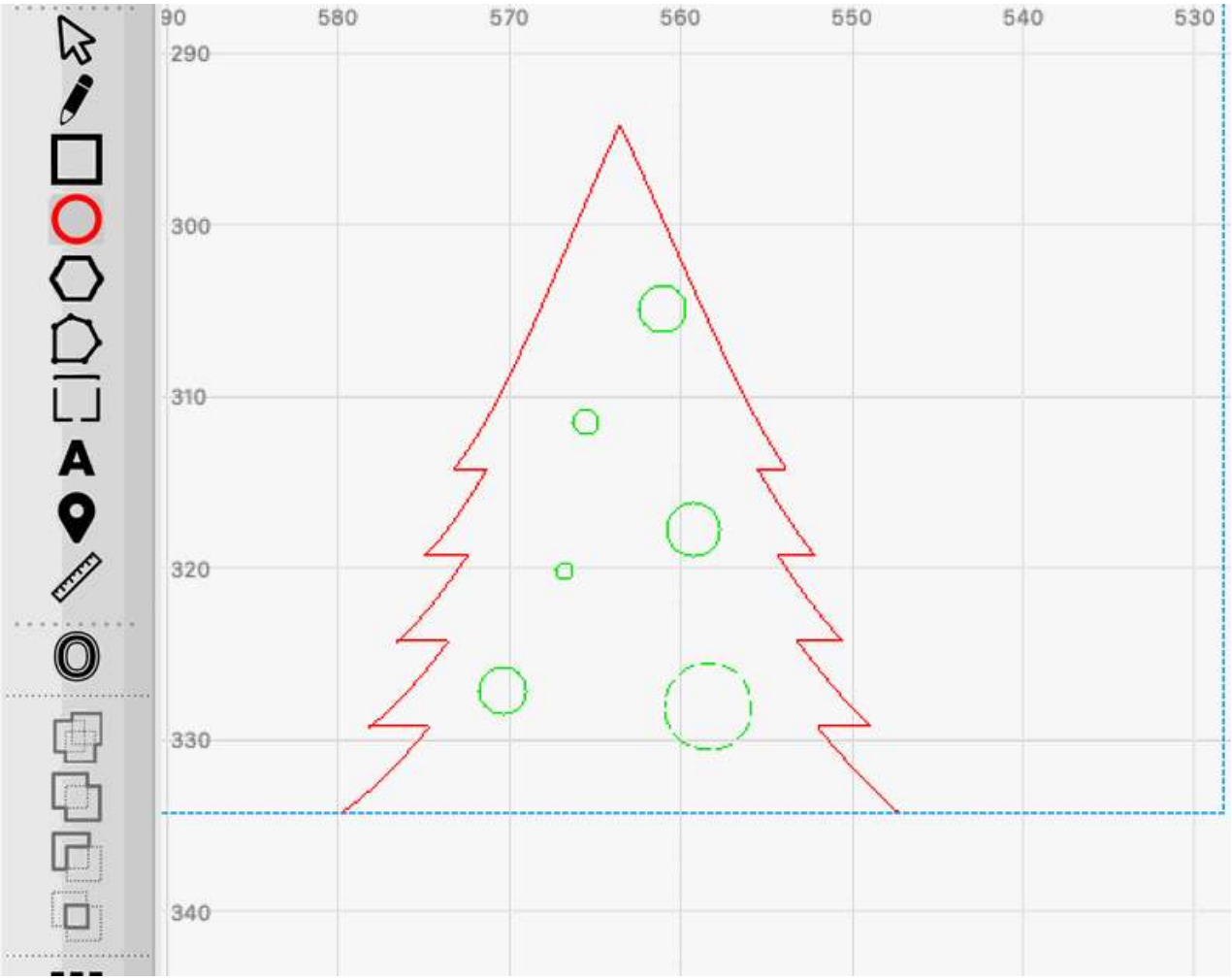
6. **Duplicate** the triangle, then nudge it up one step, then **Duplicate** the duplicate and nudge it up, and so on, until you have 5 triangles.
7. Select the second triangle from the bottom, and, in the Numeric Edits toolbar, shrink its width by 90%. Then select the next one up and shrink it by 80%, and so on until you've shrunk the last triangle.



8. Select all of the triangles and Weld them together. Resize the finished tree using the size handles, according to your preference.



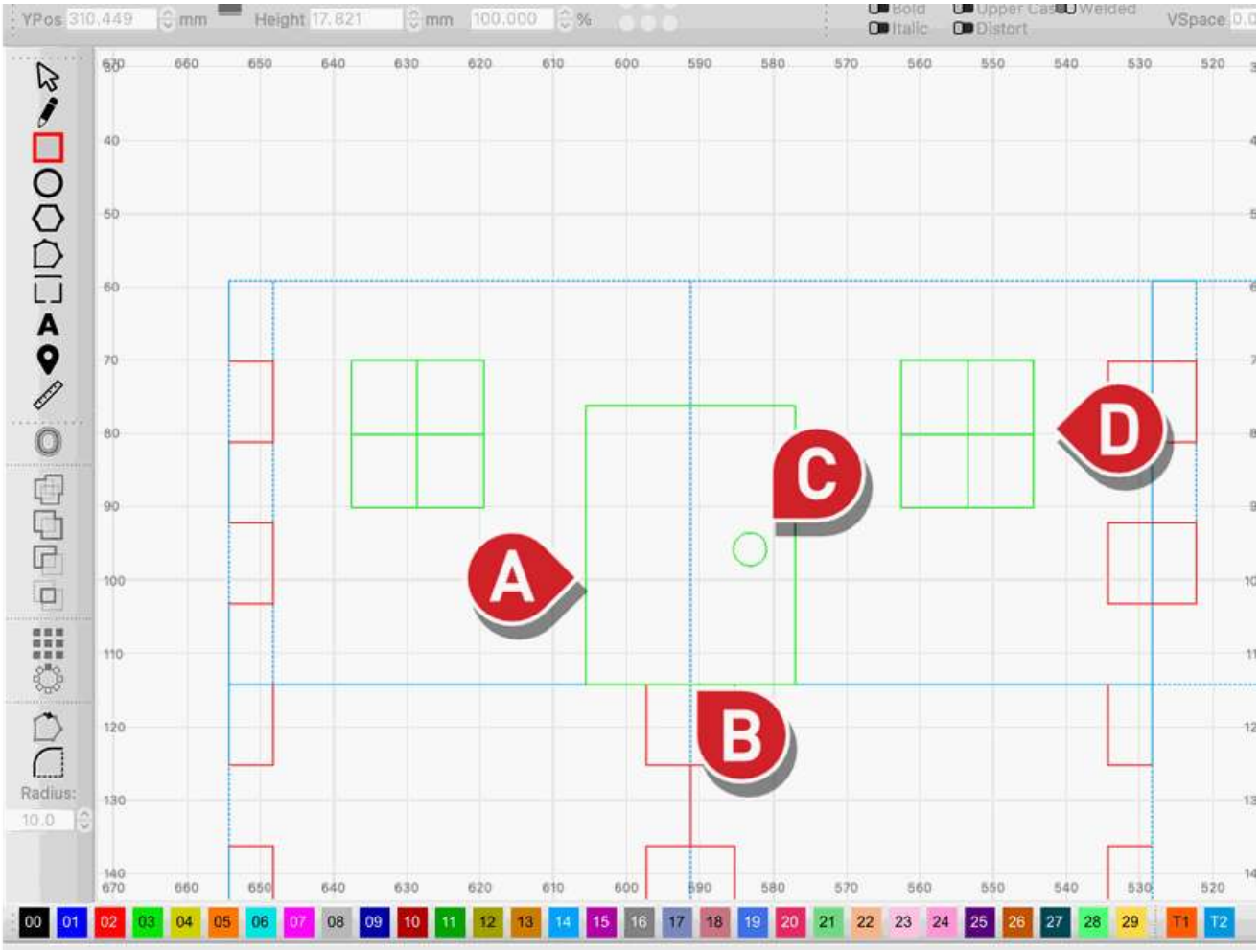
9. Snap the bottom of the tree to the bottom of the large rectangle and then select the **Edit Nodes** tool and hover over its bottom line and press D to delete that line. Since it's along the edge of the graham cracker, we don't need to cut that.
10. The last thing our tree needs is ornaments. Inside it, add a few circles set to an etch layer. Hold Shift while dragging to make perfect circles.



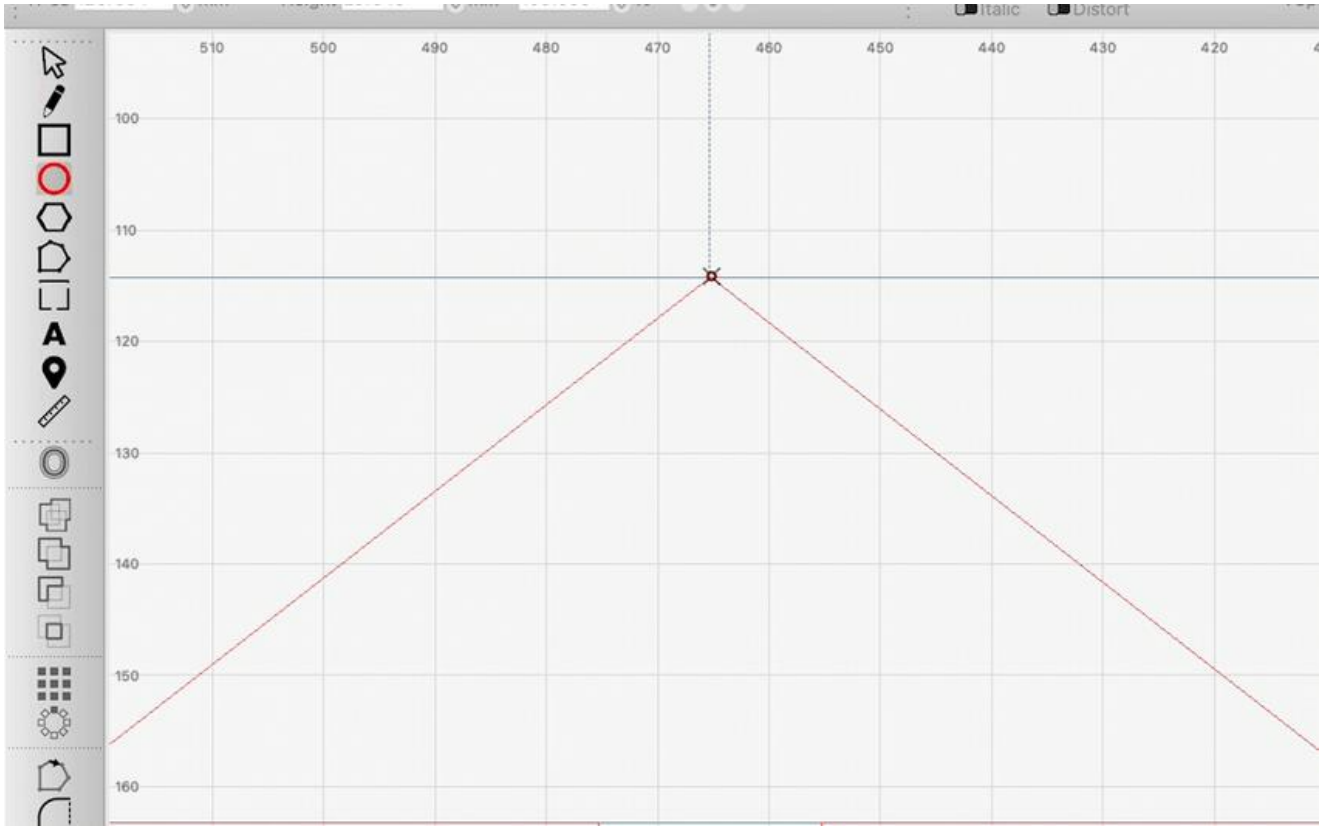
12. Add a Door and Windows

Now, we're going to add a few additional features — our graham cracker residents need a way in and out of their house, and a nice view of the neighborhood.

1. Head back to the front wall, select the etch layer, and create a new rectangle near the center of the large rectangle. It's meant to be a door — as this part is purely decorative, its size is up to you.
2. Positioning is up to personal preference as well, but you can use snapping to select the new rectangle from its bottom midpoint, and then drag it to snap to the bottom midpoint of the large rectangle, centering it in the whole design.
3. Use the Ellipse tool to create a small circle near the middle and off to the center of the new rectangle. We now have a door and door knob!
4. Create another rectangle to etch a window with. To make window panes, use the **Draw Lines** tool and snapping to draw a line from top-center to bottom-center, and right-center to left-center. **Group** and **Duplicate** the window and position one copy on each side of the door.



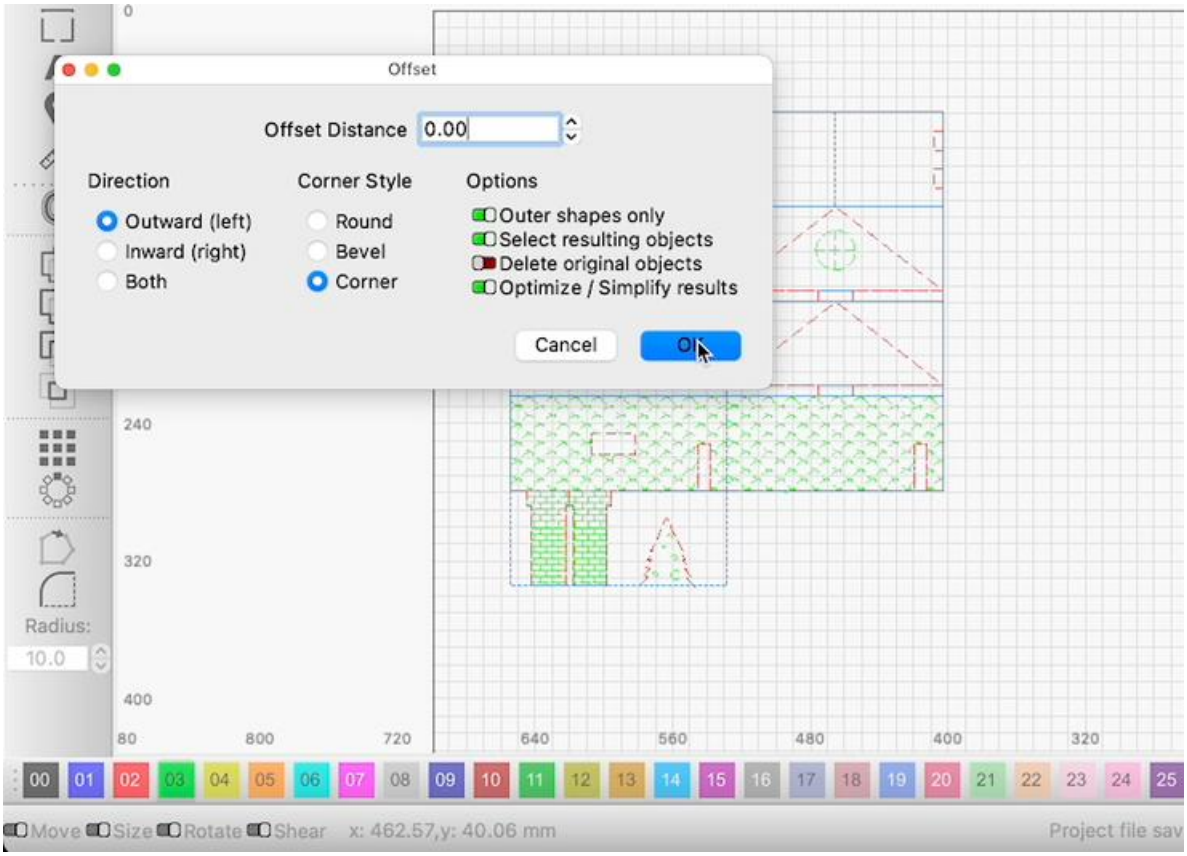
5. Now head over to one of the roof supports, and use the **Ellipse** tool while holding the Shift key to create a circle shaped window. To keep it centered, start it from the point at the top of the roofline while holding the `Ctrl`/`Cmd` key, then nudge it downward. Use the **Draw Lines** tool again to create panes down and across its center.



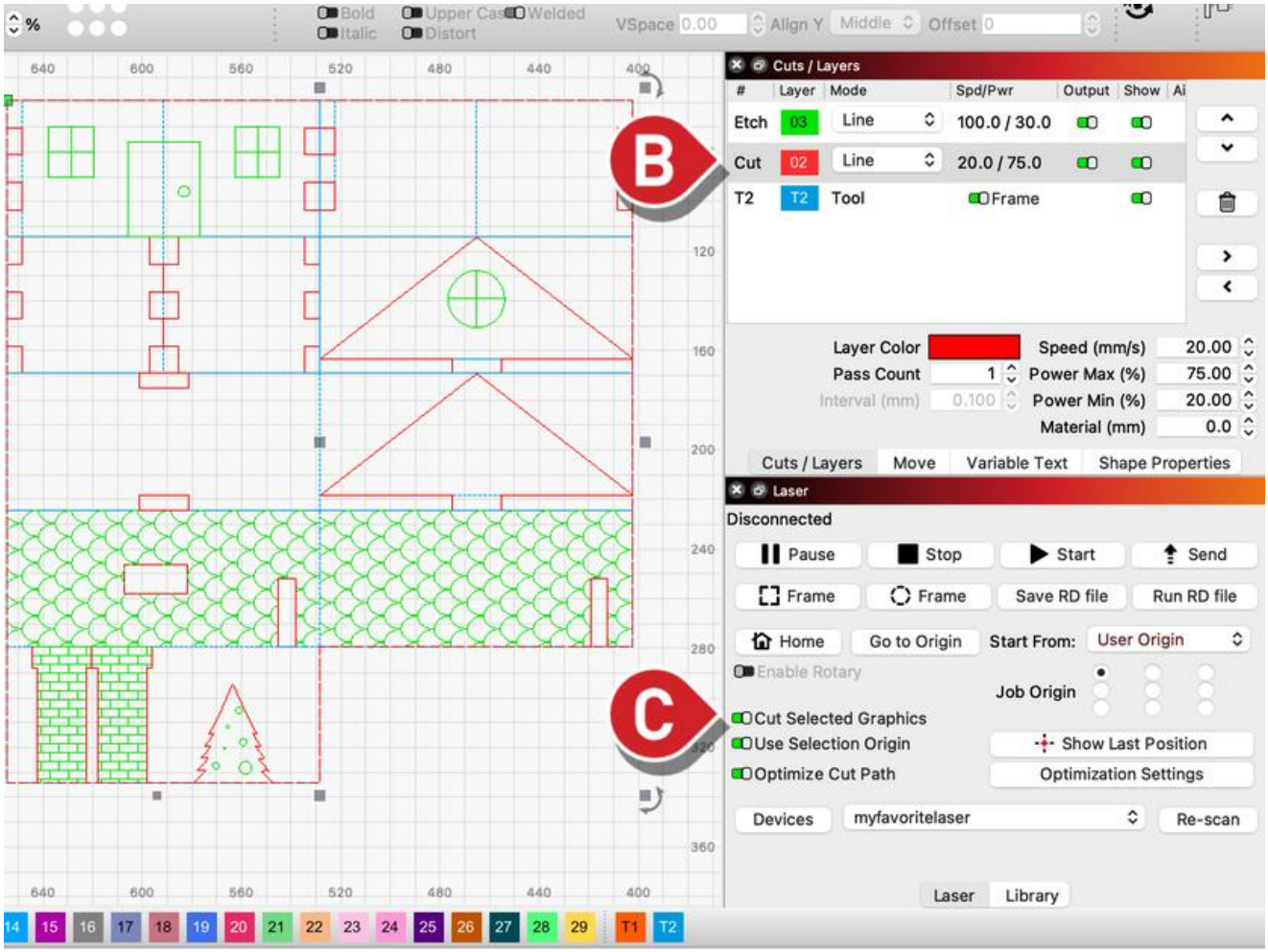
13. Make a Cardboard Jig

We're going to make a quick jig out of cardboard that we'll align our output to. This step is optional, but strongly recommended — it will make squaring up your graham crackers and aligning your laser to them much easier.

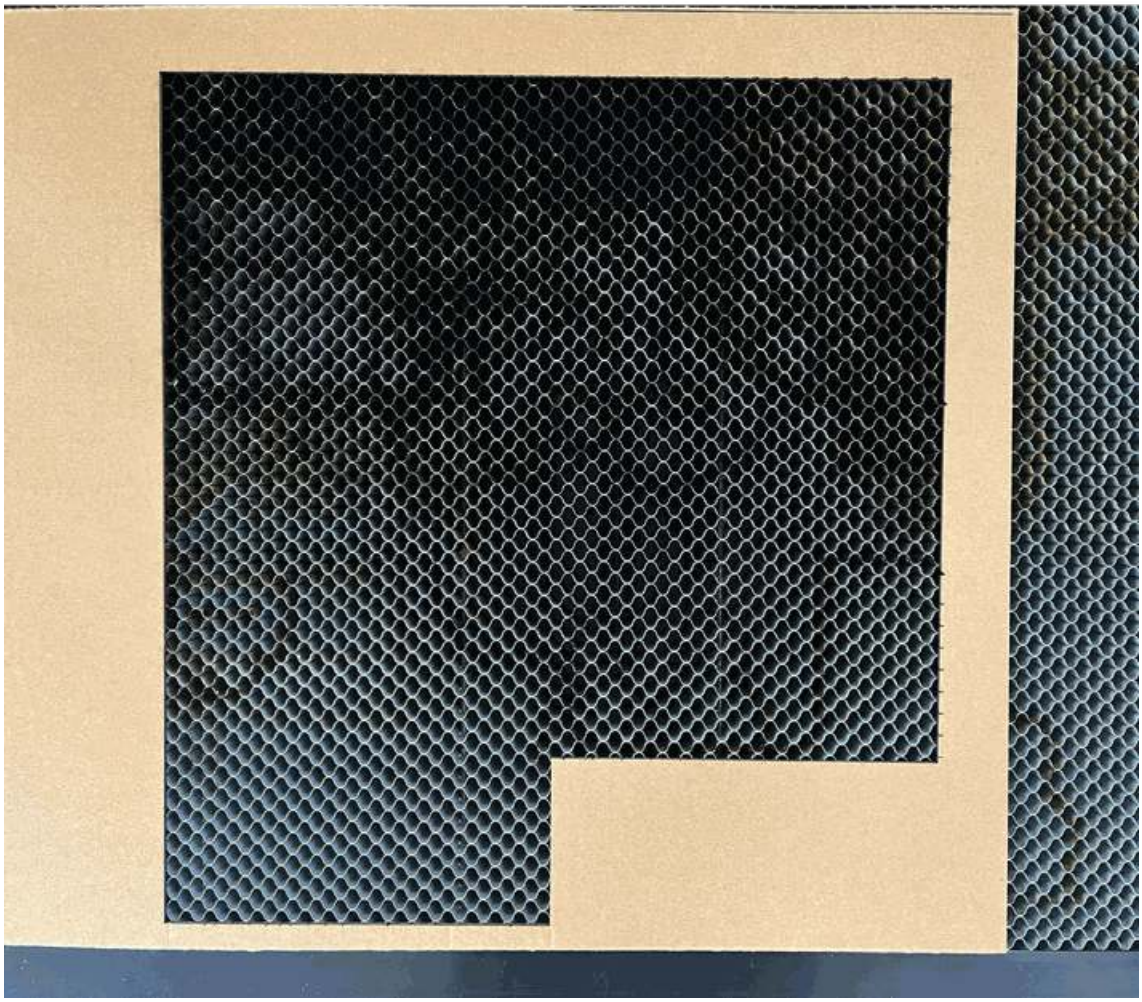
1. Select all of the templates and then the Offset tool (`Alt`+`O`/`Option`+`O`). Set the **Offset Distance** to 0, and make sure **Outer shapes only** is toggled on.



2. Place the resulting offset on a cut layer, with a speed and power setting sufficient for your laser to cut cardboard (or any other material you choose to use for your jig).
3. Enable Cut Selected Graphics and select the offset only.



4. Use whatever Start From mode you prefer to align this part of the job. Use Preview (`Alt`+`P`/`Option`+`P`) to make sure you have only the offset selected, and Frame (`Alt`+`R`/`Option`+`R`) to double check your positioning. Run the job, then remove the cut out material, and leave the rest in your laser to serve as a jig.

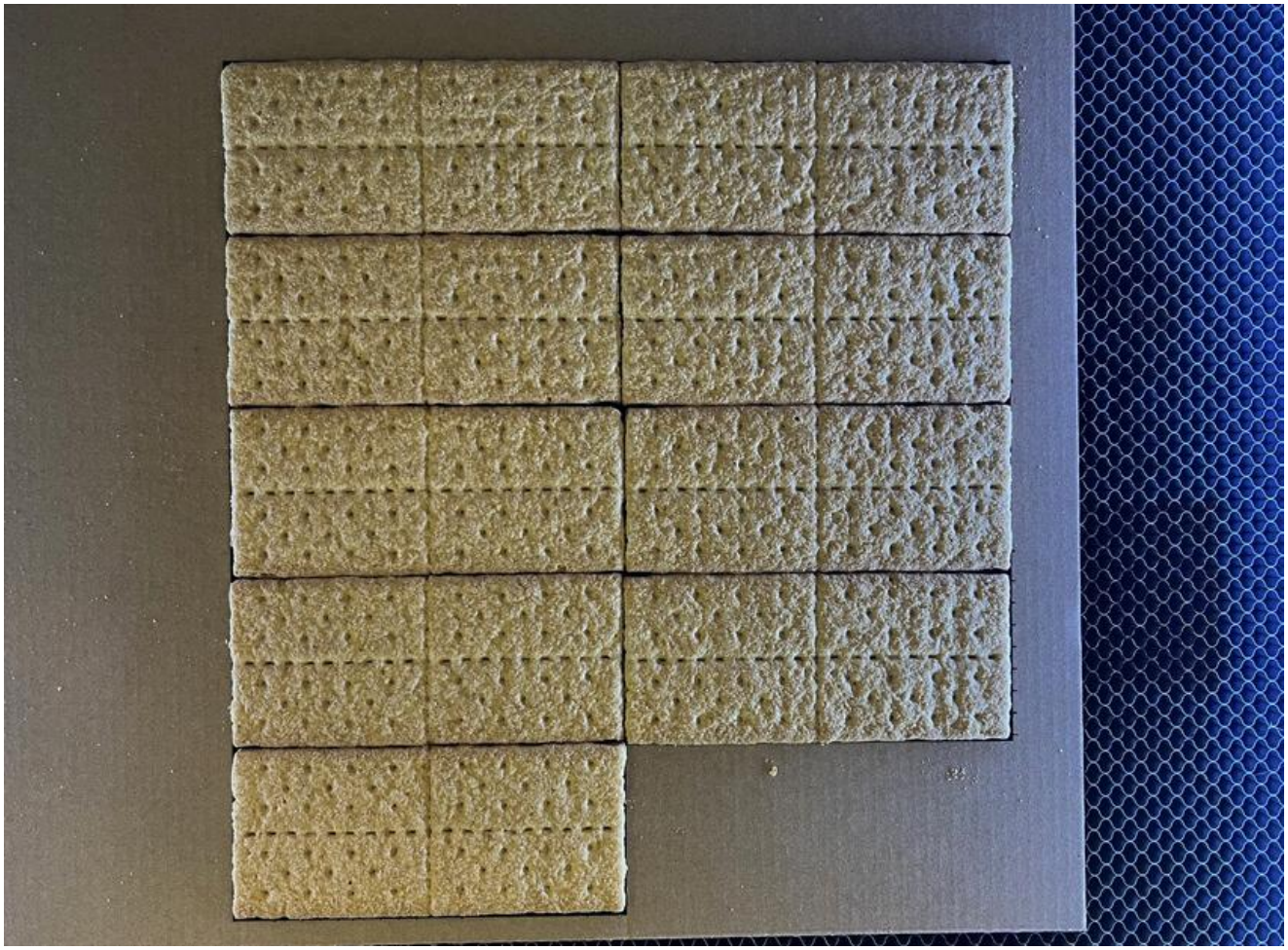


5. Back in the LightBurn workspace, put the offset onto a Tool layer.

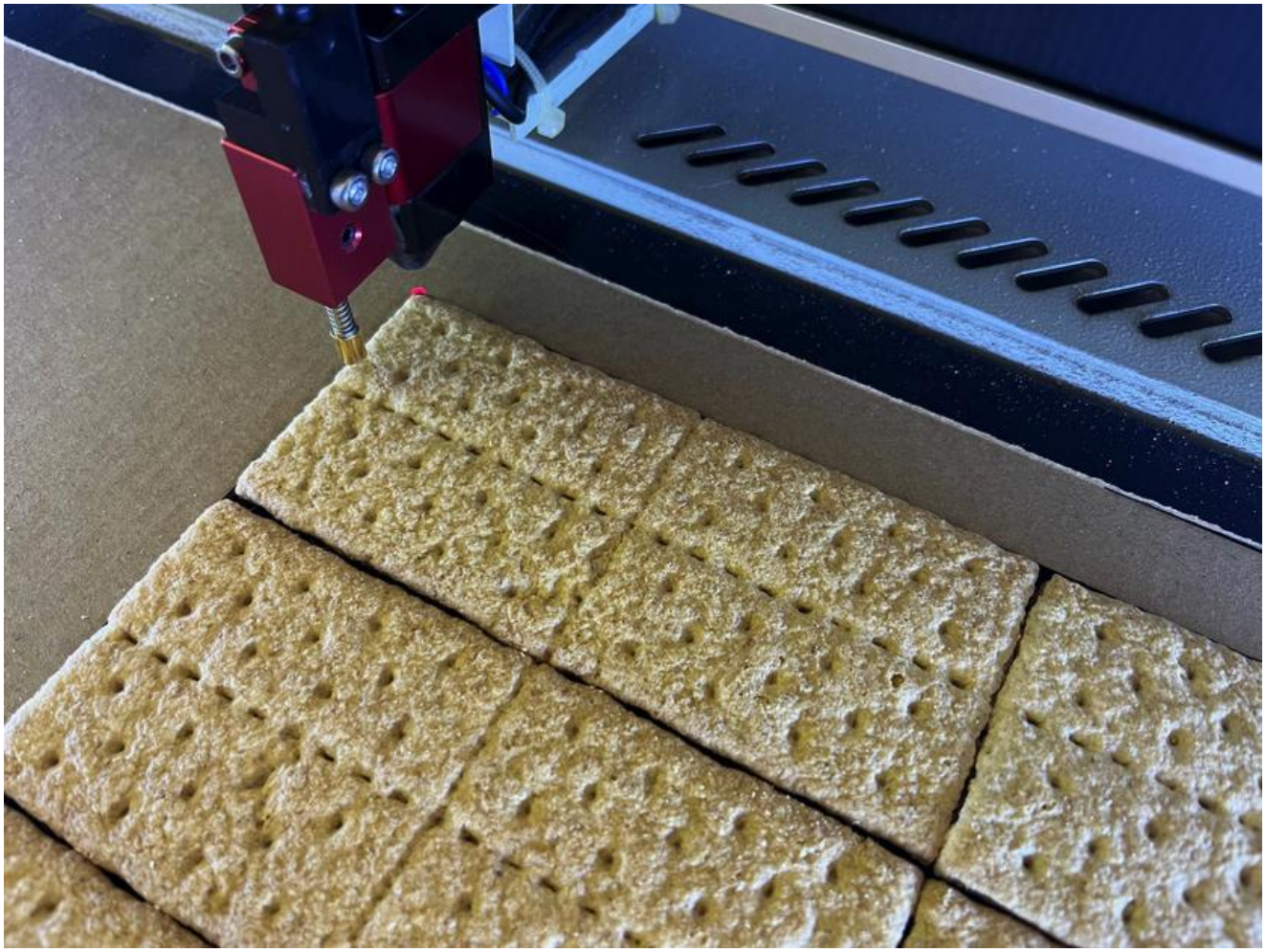
14. It's Time to Laser the Graham Crackers!

The big moment is here.

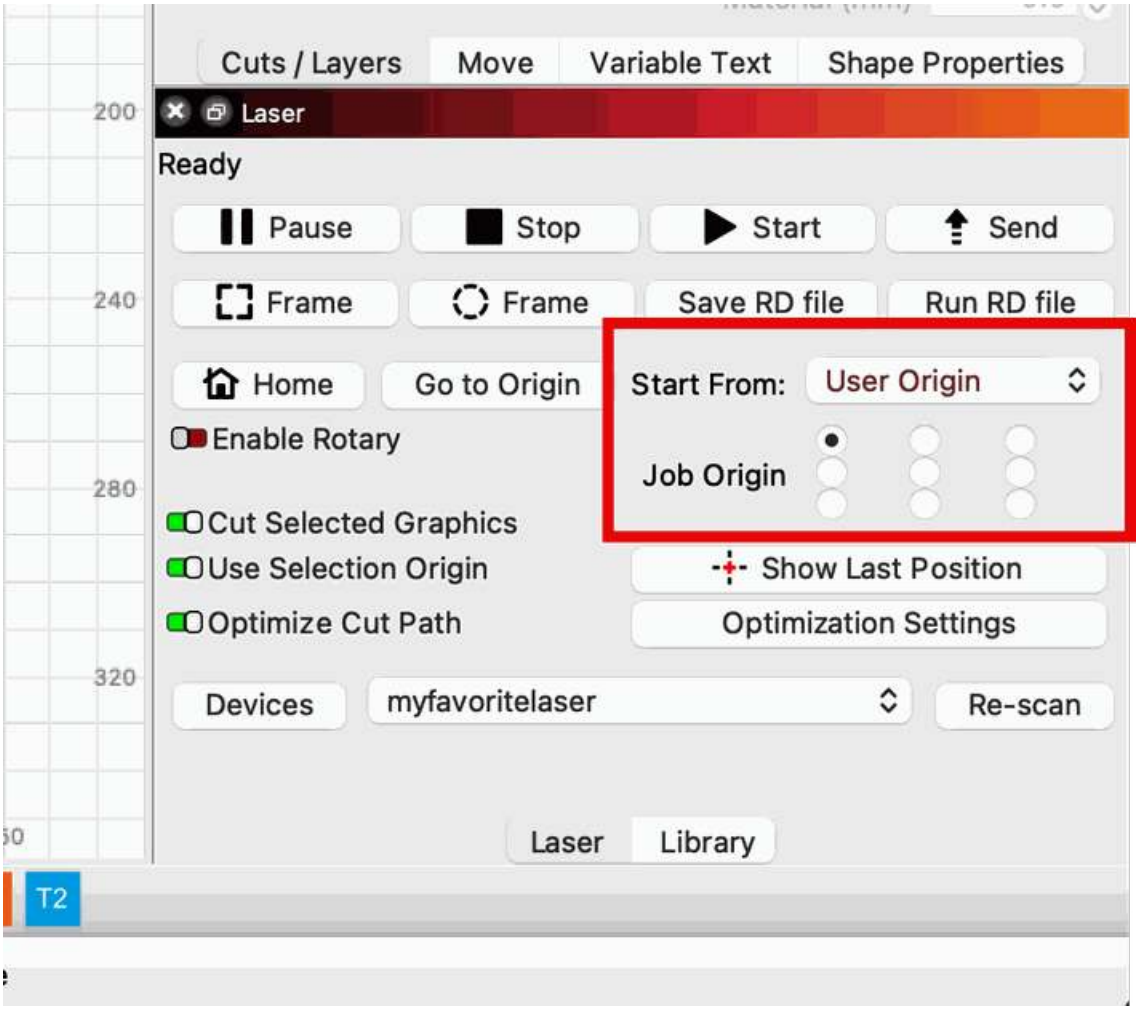
1. Make sure your cut layer is now set to an appropriate power and speed to cut through a graham cracker, and the etch layer is set to an appropriate power and speed to mark it, but not cut through. If you're not sure what to set, you can dedicate a spare graham cracker or two to run a Material Test on.
2. Square the jig up in your laser's workspace, then place 9 graham crackers inside it. Try to choose the most uniform graham crackers you can find, and eat the broken and crumbled ones.



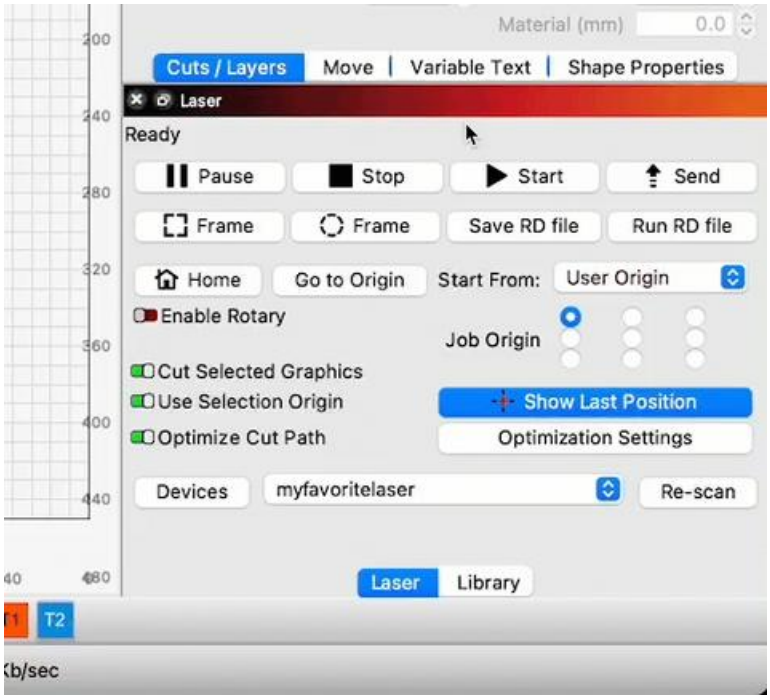
3. Align your laser to an inner corner of the jig: top left, top right, or bottom left. If you're using a diode laser, the **Fire button** can assist with alignment.



4. Set the **Job Origin** location to the same corner you've aligned your laser to on the jig, and set the **Start From** mode to **User Origin** or **Current Position**. If you select **User Origin**, make sure to set the origin in the **Move** window (if you're using a GCode-based laser) or through the control panel (if you're using a DSP laser).



5. Select everything in your workspace and **Frame** the project to make sure you've got your **Start From** mode and **Job Origin** set correctly



6. This is it — run the job!
7. Remove he pieces from your laser and assemble the house. It will stay together without any help, but you can add frosting or melted chocolate along the joints to make it sturdier, and candy or any other decorations to make it merrier.

Once you've decided to decorate your house further or admire it as-is, it's time to sit back, enjoy the holiday season, and show all your friends and loved ones what you made. After that, think of s you can apply the above tools and principles to creating something a bit more permanent, and get to making it!





Image Engraving on Coasters

Coasters are a must have item in any wood-respecting household. If you or anyone you know has a beloved end or coffee table they want to protect from scratches and condensation, but is tired of boring, store-bought coasters, this project is for you.

What You'll Learn

In this guide, we'll show you how to etch an image of your choosing on a slate coaster. We'll cover how to:

- Import an image into LightBurn
- Find and apply appropriate settings to etch your material
- Adjust the image to get it laser-ready
- Use tool layers to reate a template to represent your coaster, and use it to Mask an image
- Select a Start From mode in LightBurn, and position your coaster in your laser's workspace

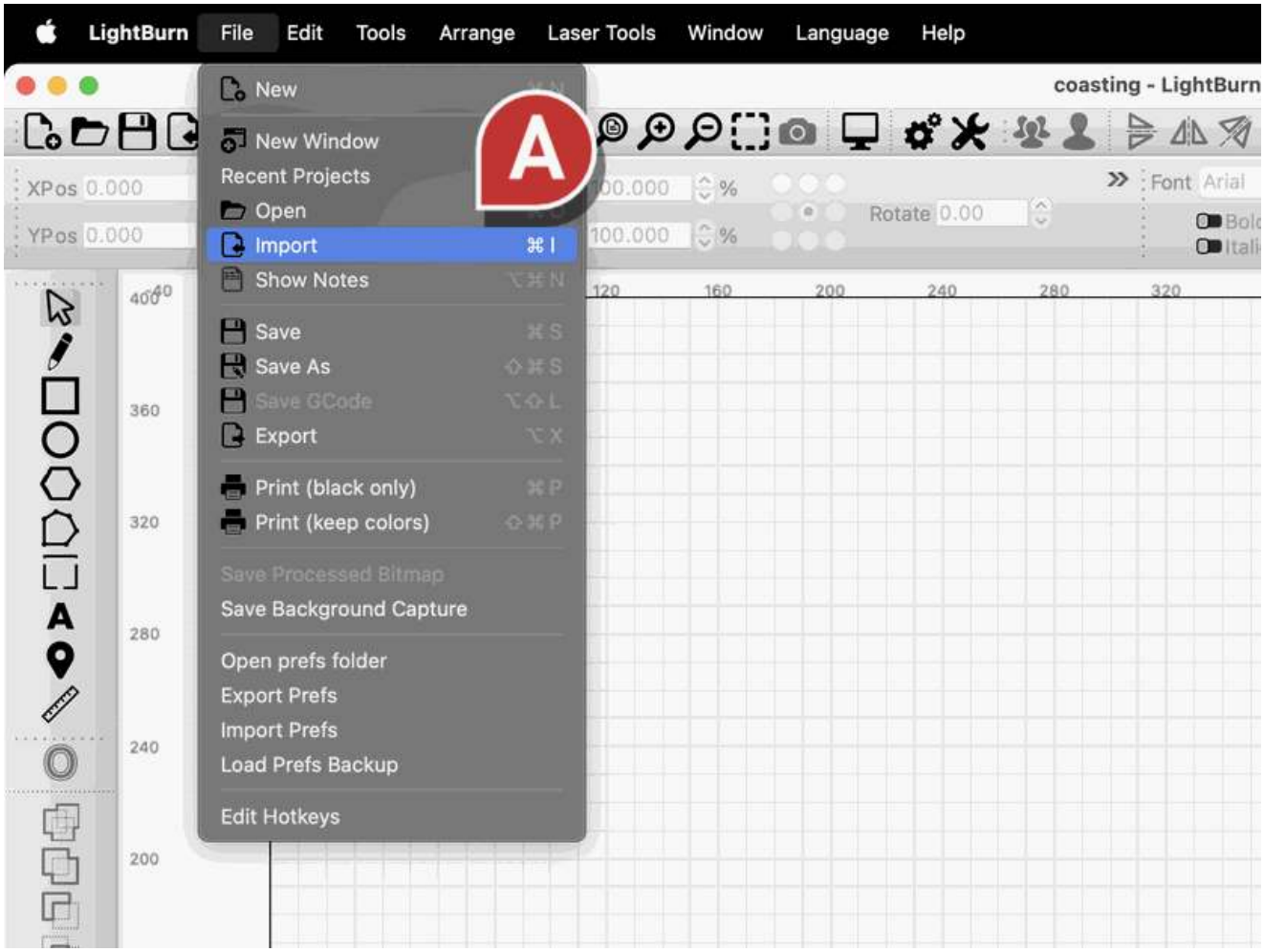
Materials

- An image you want to immortalize in coaster form. You can select any image you like, but some will work better than others - if you try to engrave a photo of your pet bunny in a snowstorm, you may have a hard time. Images with sharp contours and contrast will work best.
- Premade coasters of any material. We used slate, but you can use wood, tile, or anything else you'd like. Slate makes a great beginner material, while wood can sometimes be a bit more challenging.
- A laser capable of etching the material you choose.

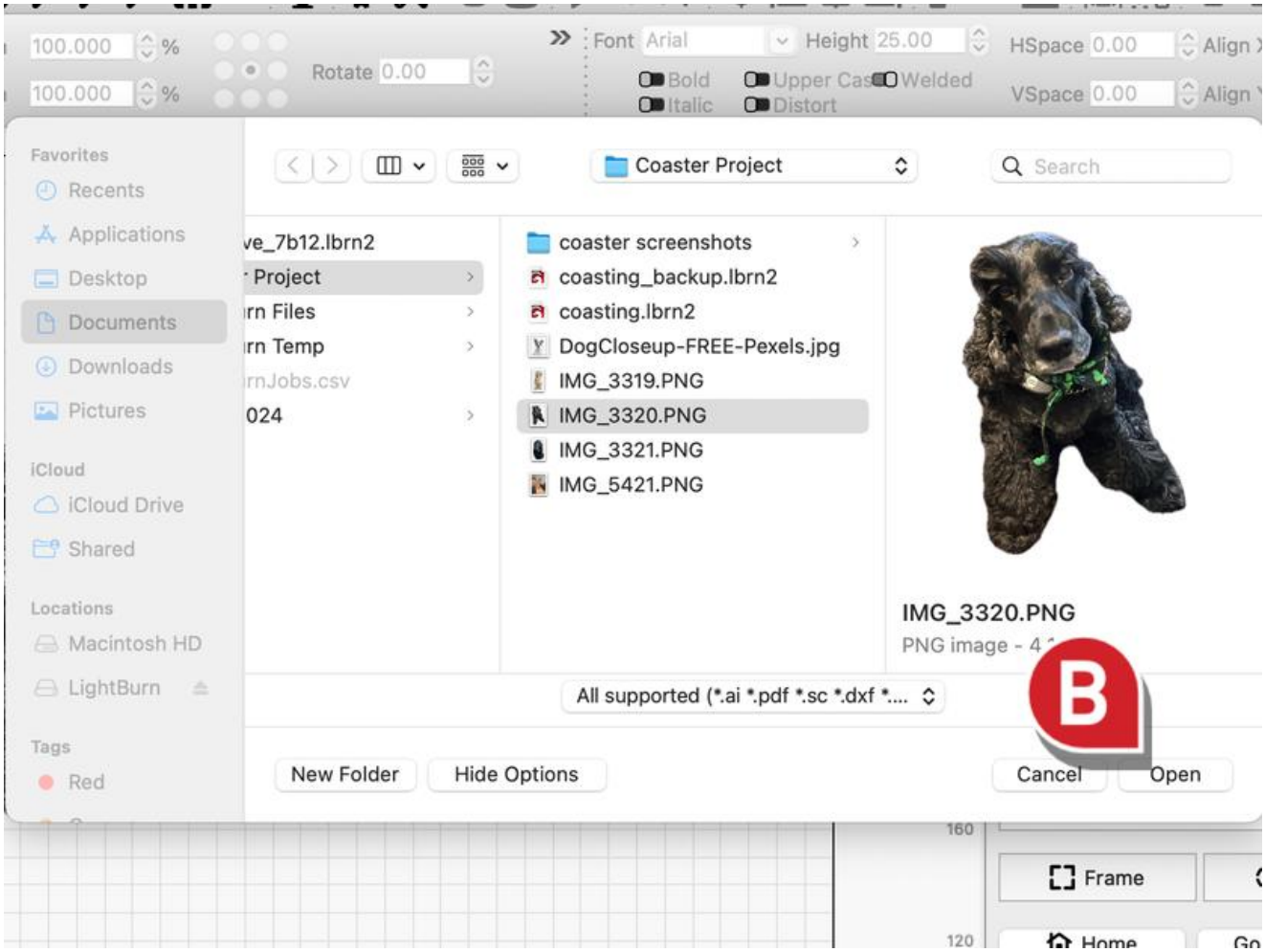
Steps

1. Import Your Image Into LightBurn

1. Go to the File menu and select **Import**, or press `Ctrl`+`I` for Windows, or `Cmd`+`I` for Mac.



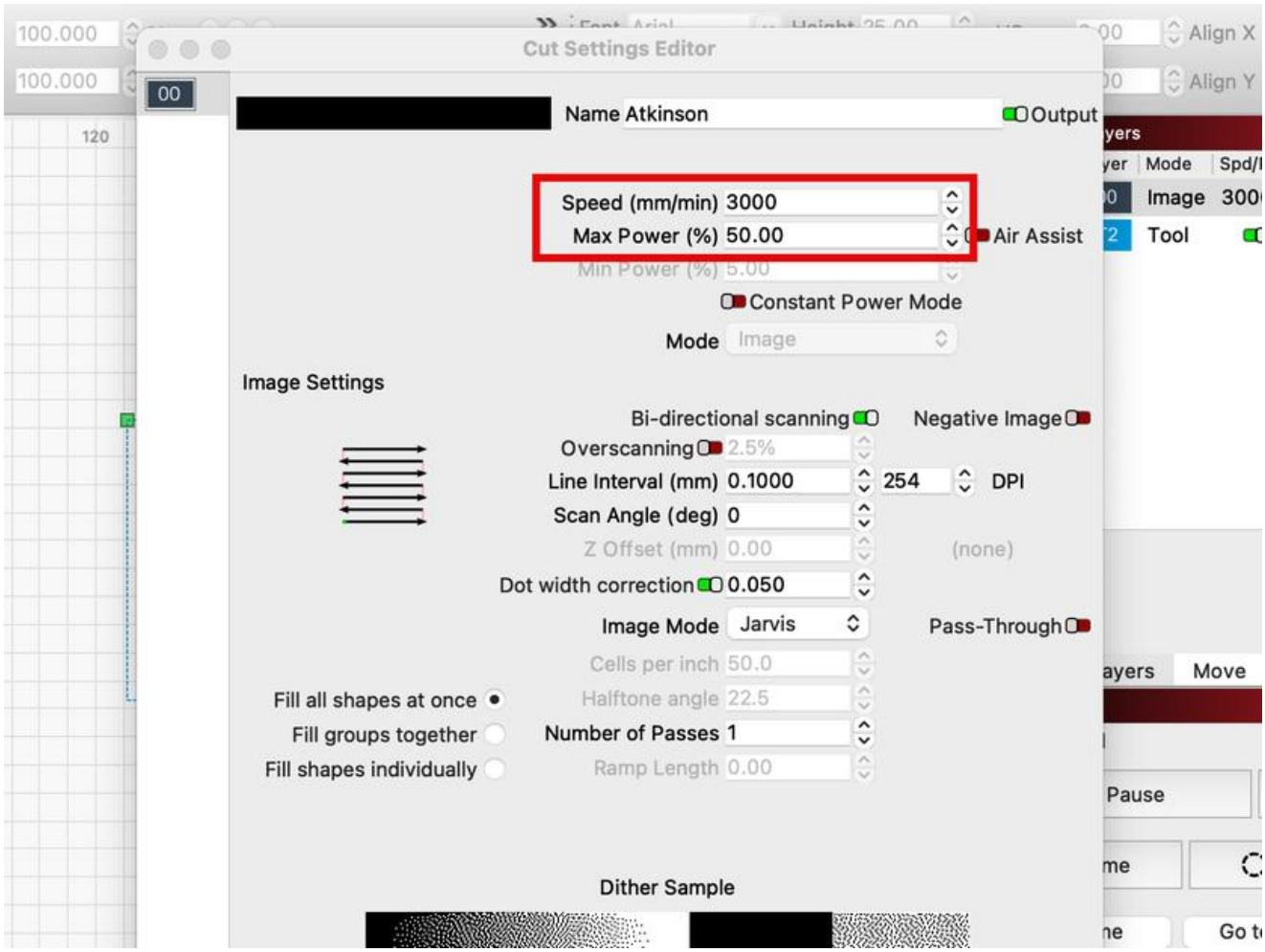
2. Navigate to the location your image is saved to on your computer and select it.



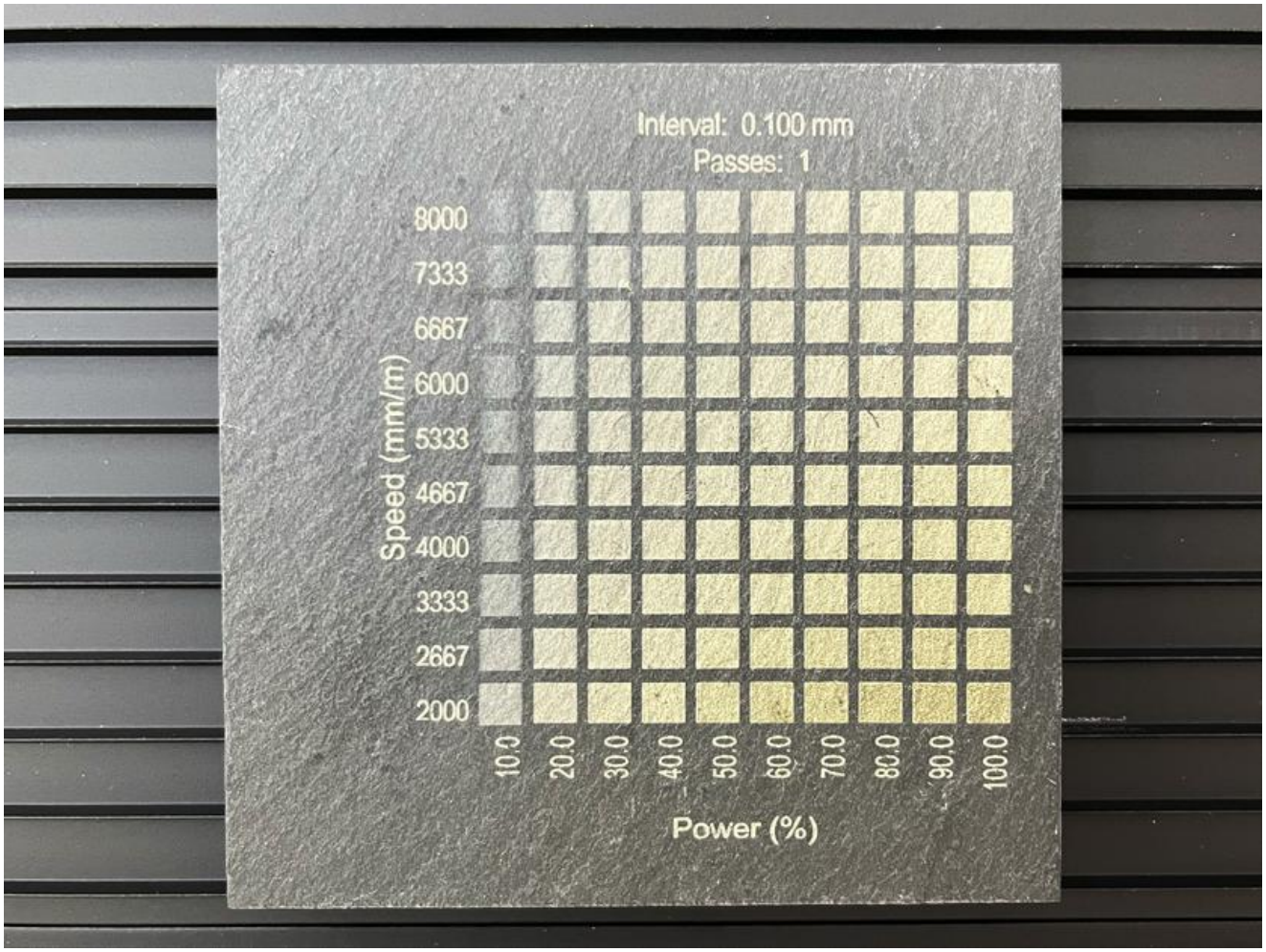
Note: I removed the background from this photo using my iPhone's Lift feature, but you can also use a free tool like Pixelcut or delphitools to do the same.

2. Apply Layer Settings

1. Double-click the Image layer in the Cuts/Layers window to open the **Cut Settings Editor** and apply **Speed** and **Power** settings.



2. If you're unsure what settings to apply, try running a Material Test. A test grid ranging from 10% to 100% power, and 2000 mm/minute to 8000 mm/minute is a good place to start. Watch our full video guide on setting up a **Material Test** here. You may also enjoy our guide to better image engravings.

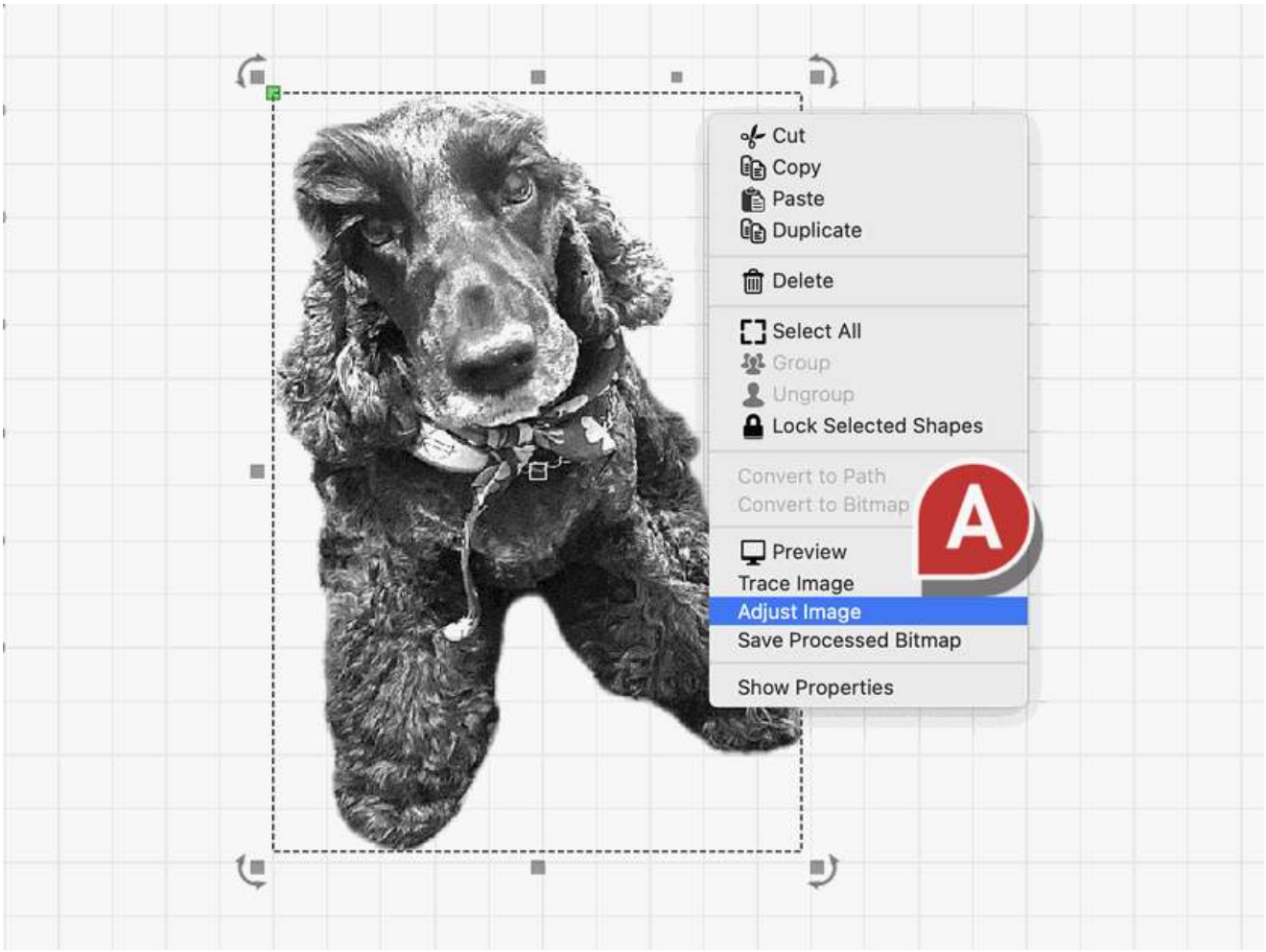


3. You may also be able to find some suggested settings on your laser manufacturer's website.

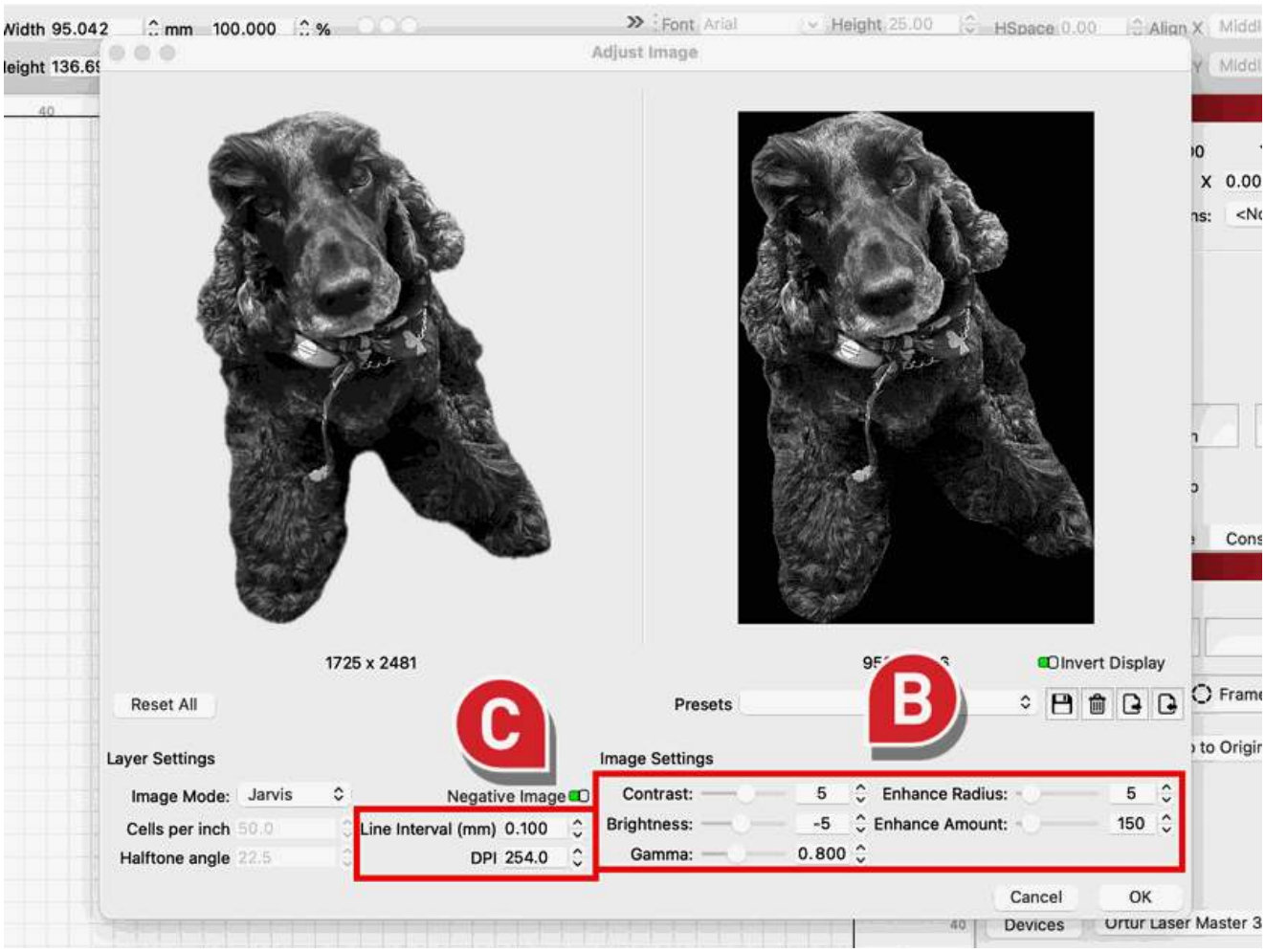
There is a lot of variability between lasers, and no one-size-fits-all settings, so be prepared to test and experiment with your settings.

3. Adjust Image

1. Right-click the image and select Adjust Image.



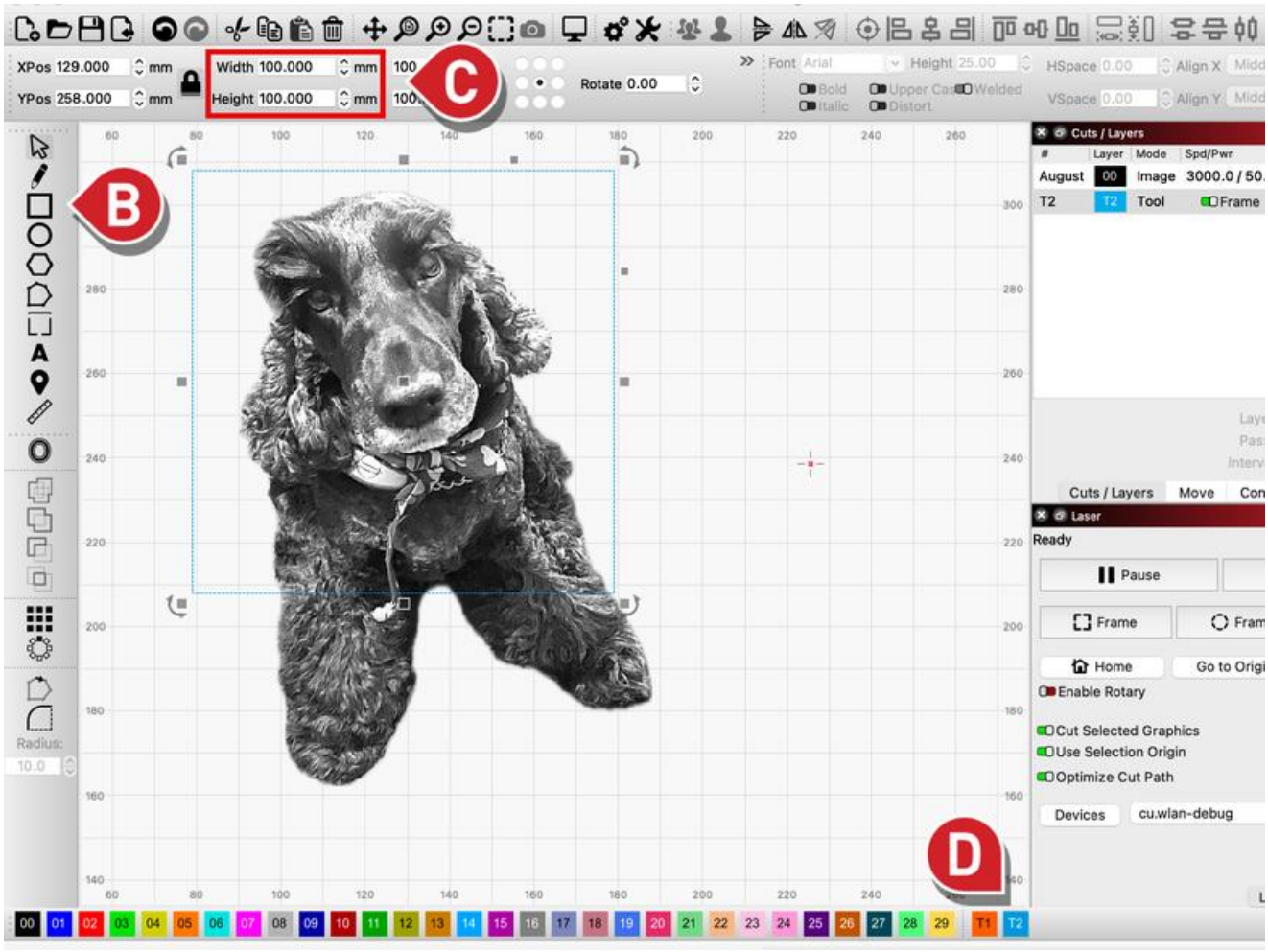
2. Apply some standard starting point settings:
    - **Stucki** or **Jarvis** Image Mode
    - DPI: 254
    - **Gamma**: 0.800
    - **Contrast**: +5
    - **Brightness**: -5
    - **Enhance Radius**: 5
    - **Enhance Amount**: 150
    - **Gamma** adjusts mid-tone shading. Lower values make the mid-tones brighter, and higher values make them darker. **Contrast** makes bright shades brighter, and dark shades darker. **Brightness** makes all shades brighter or darker. Together, **Enhance Radius** and **Enhance Amount** increase contrast along edges.
3. When engraving on a dark surface like a slate coaster, you'll want to enable Negative Image.



As with the **Layer** settings, be prepared to test these settings, and adjust according to the results you see.

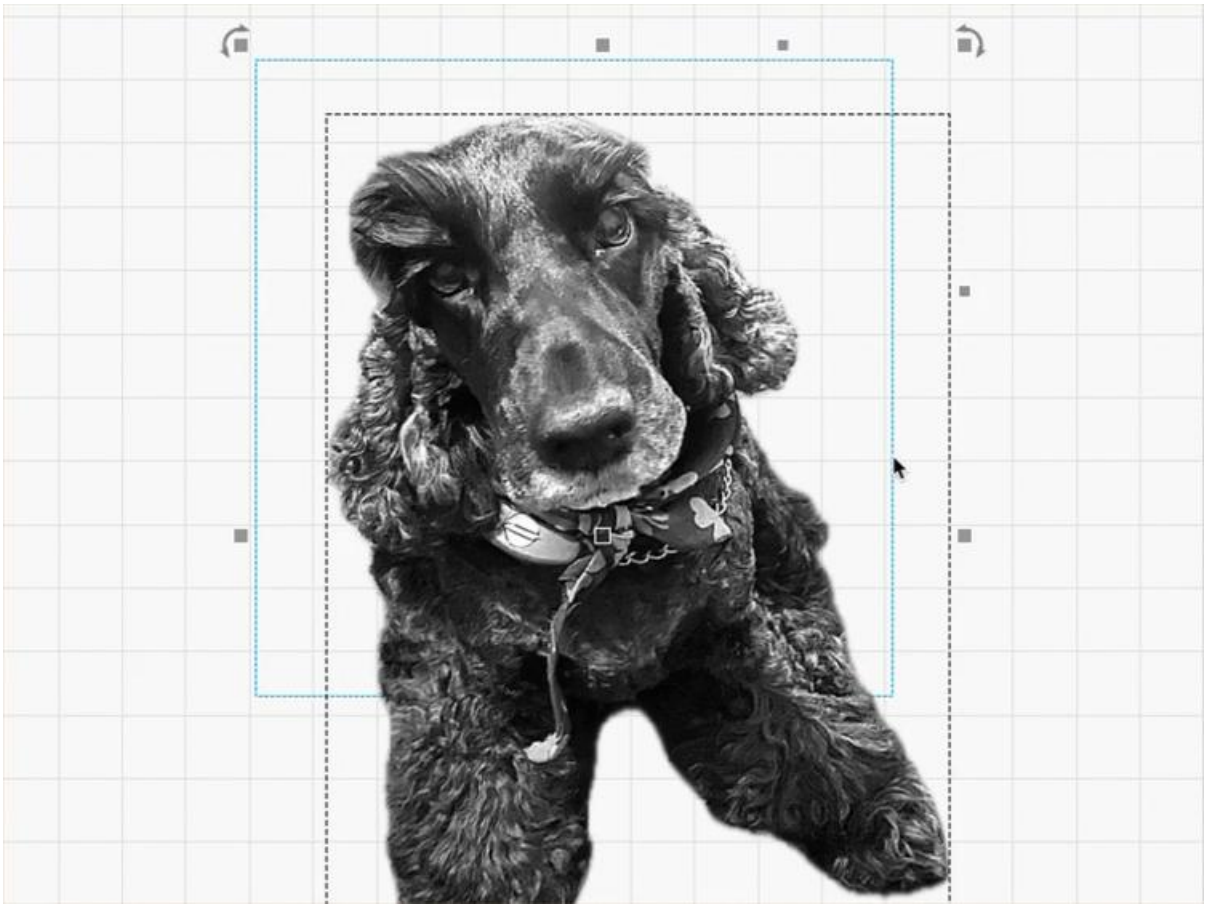
4. Create a Template of Your Coaster

1. Measure your coaster.
2. Create a shape in LightBurn using the Rectangle tool, for square coasters, or Ellipse tool, for circular coasters. Holding the `Shift` key while creating the shape will keep it perfectly square or circular.
3. In the Numeric Edits toolbar, set the shape's dimensions to match the measurement you took of your coaster.
4. Set it to a Tool layer.
5. Resize the image to fit to the shape - it's ok to leave some of the image outside the shape, we'll take care of that in the next step.

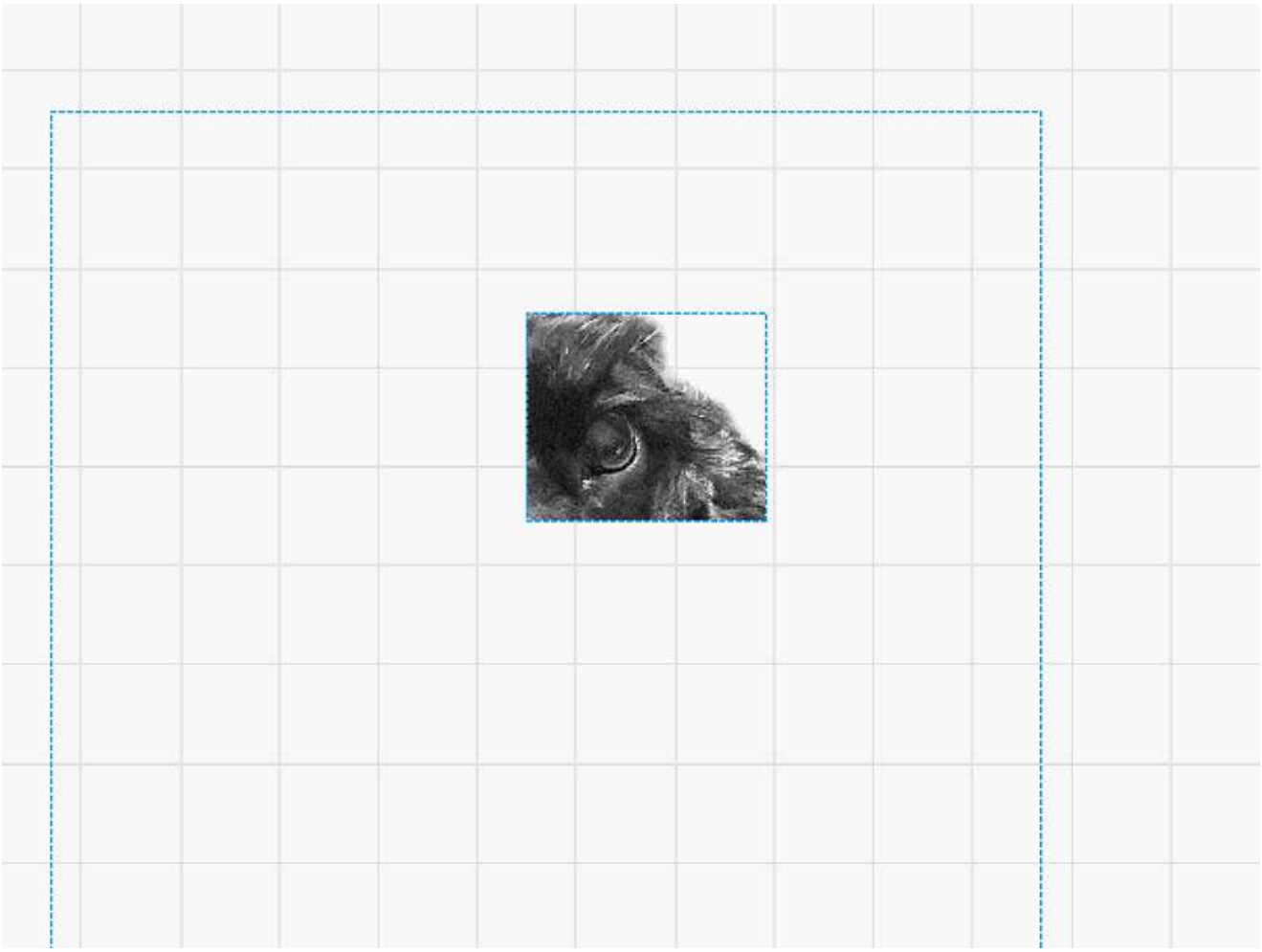


5. Apply Mask to Image

1. Select your image and the Tool layer, then right click and select Apply Mask to Image, or go to **Tools → Apply Mask to Image**.



2. You can move the **Tool** layer around after applying the mask, if you'd like to adjust the area of the image that is masked. To remove a mask, right-click and select **Remove Mask from Image**.



3. Masking can also be a great help in testing settings. You can mask off a small area of your image and etch just that section to keep runtimes low as you test different settings.

6. Select a Start From Mode and Job Origin

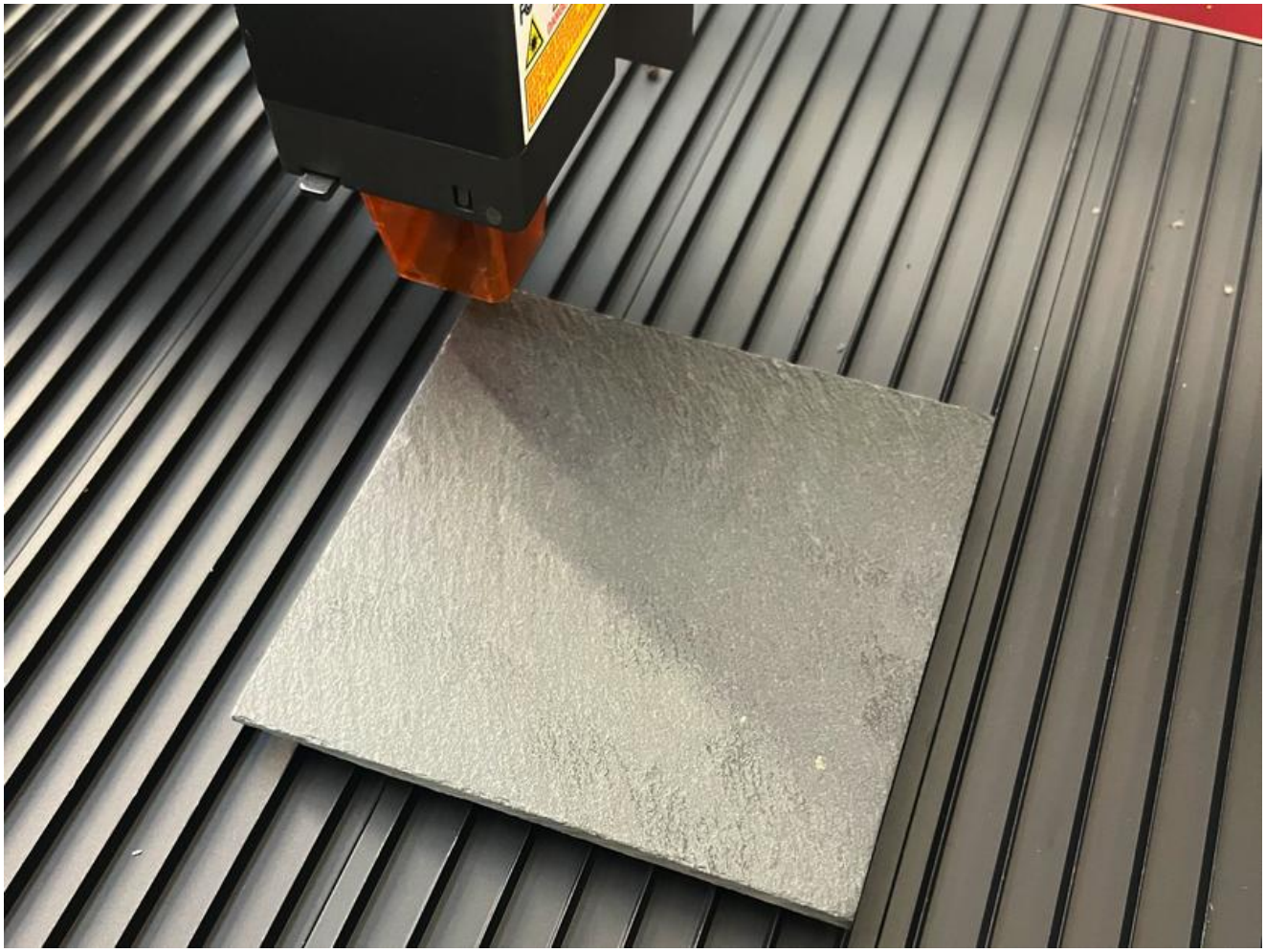
1. Select Current Position as the **Start From** mode.
2. Set **Job Origin** to a corner if you're engraving on a square coaster. If your coaster is circular, select the center.

For more information on the different **Start From** mode options, watch our full video guide here.

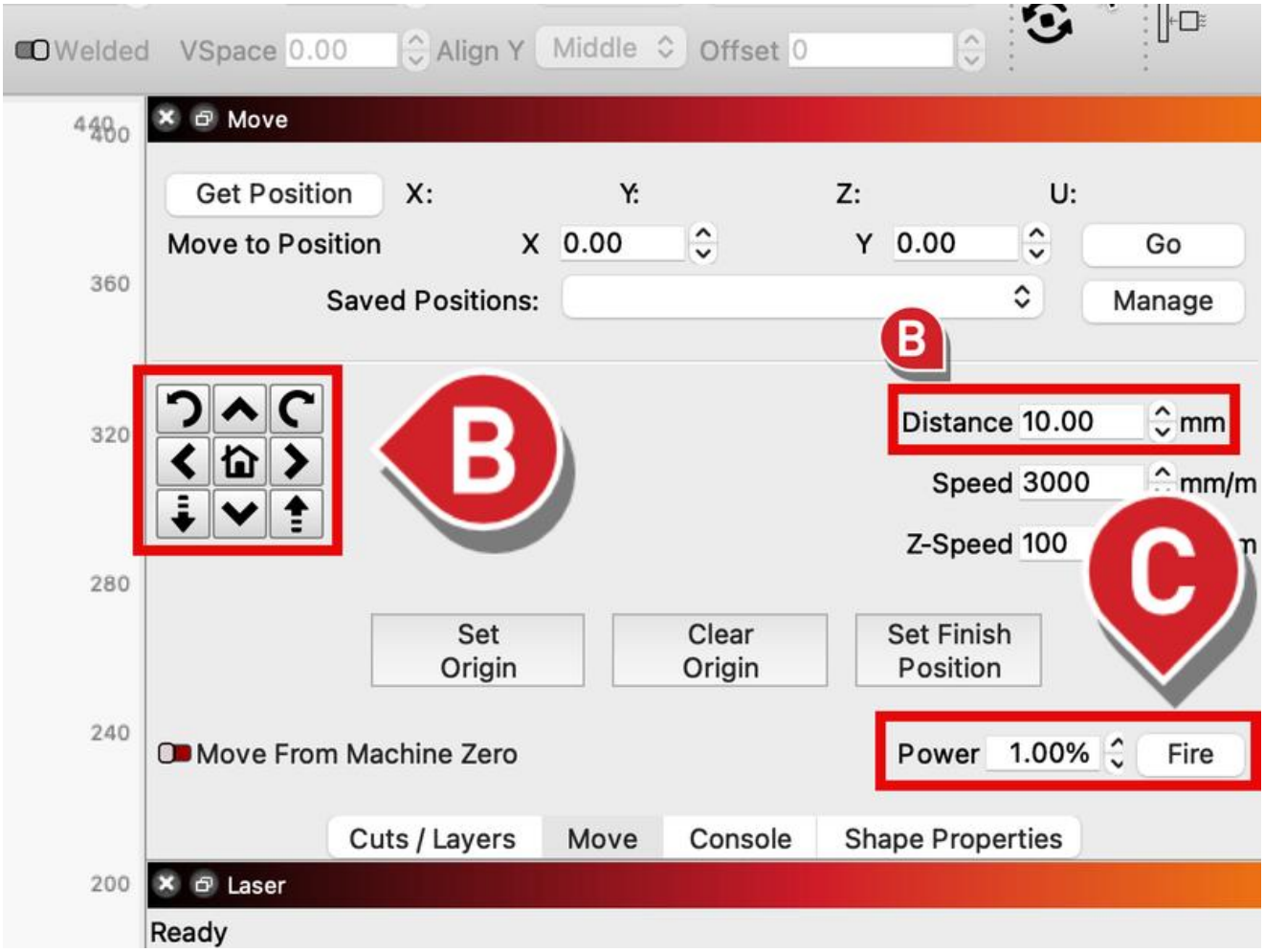
```
![](./img/startfrom-1.png){ class="spaced-image"}
```

7. Position the Coaster and Run the Job

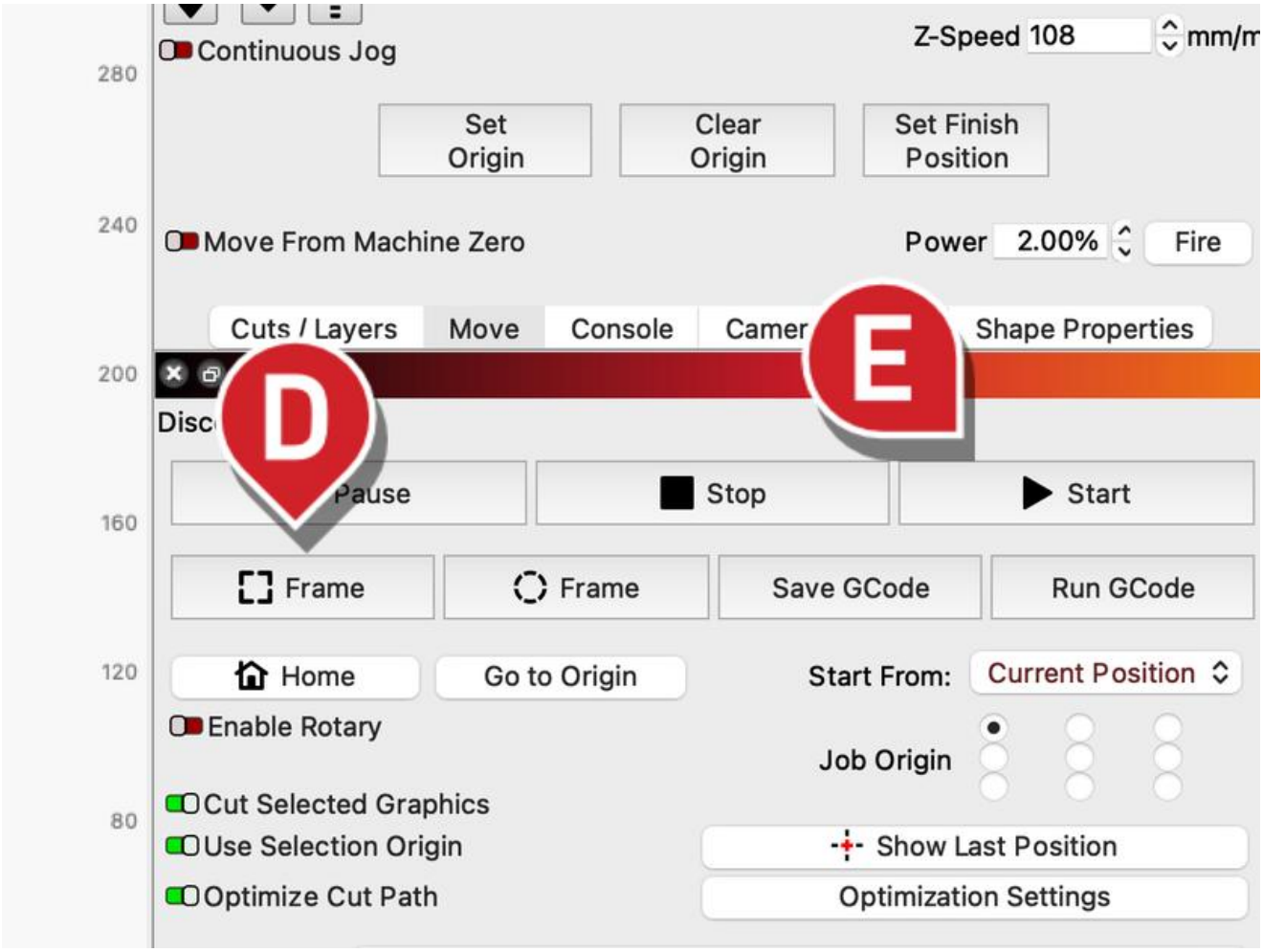
1. Place the coaster near the center of your laser's workspace. It doesn't have to be exact, but try to provide roughly equal space from each side of the coaster to the edge of your workspace.



2. Using the arrow keys in the Move window, **Jog** your laser to the same location on the coaster as you selected for the **Job Origin**. To refine positioning, you can lower the increment in the **Distance** field.
    - For our project, we selected top left for **Job Origin**, so we jogged the laser up and over to the top left corner of the coaster.
    - If you're using a circular coaster, you may want to pre-mark the center of the coaster with a ruler and pencil, and align to that mark. Alternatively, you can use the Center Finder tool.
3. If your laser does not have a red dot pointer, you can enable and use the Fire button to aid in alignment. When you **Fire** the laser, it should pulse directly over the position on your coaster that corresponds to the **Job Origin** you selected. Make sure you are wearing safety glasses before ever firing the laser.



4. Frame your project to verify alignment.
5. Press Start to run the job!


Continuous Jog
Z-Speed 108
mm/m
280
Set Origin
Clear Origin
Set Finish Position
240
Move From Machine Zero
Power 2.00%
Fire
Cuts / Layers
Move
Console
Camer
Shape Properties
200
D
E
Disc
Pause
Stop
Start
160
Frame
Frame
Save GCode
Run GCode
120
Home
Go to Origin
Start From:
Current Position
Enable Rotary
Job Origin
Cut Selected Graphics
80
Use Selection Origin
Show Last Position
Optimize Cut Path
Optimization Settings

Other Ideas



That's it, you've got a coaster! No one wants just one coaster, so you'll surely want to make more. Here are some ideas for your next coaster project:

- Use the Grid Array tool to lay out duplicates of the same coaster you just made, and run a full batch at once. You can even use Variable Text to customize each one with someone's name.
- Create a series of coasters with different images on each.
- Etch something other than an image - favorite graphics, logos, or inspirational quotes can all make excellent coasters.
- Instead of using premade coasters, you can also cut your own - just turn the Tool layer you used as a coaster template into a Line layer instead, and give it appropriate settings to cut through a material of your choosing.



Laser Cut Dinosaur

Laser cut models are among the easiest and most enjoyable beginner laser projects. Ranging in complexity from simple to intricate, their parts join together by use of well-placed slots and tabs, and they make great puzzles, party favors, or decorations. There's no shortage of designs available on the internet, but what to do when you download a design with slots that don't match the material you're working with, or you want to adjust the scale of a model? And how to ensure a snug fit between parts?

By using Resize Slots and applying a Kerf offset, you'll be able to adapt any model to your material, scale, and laser. In this guide, we'll walk you through modifying a rudimentary project of our own design. To follow along, you can download our project here, or use any alternate design you'd like - it'll just need to have slots or tabs.

Download Project File

Resize Slots

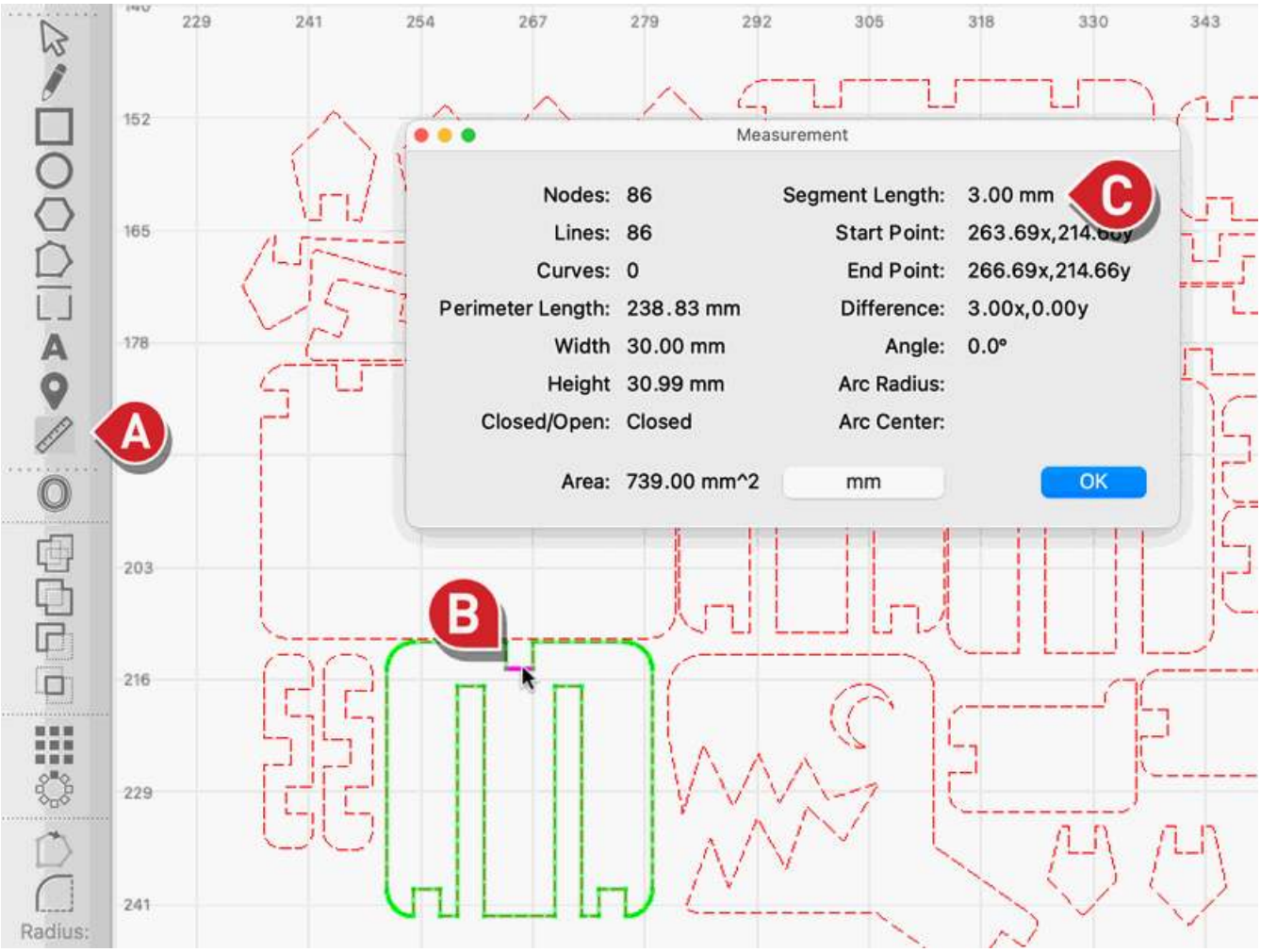
In order for a model to fit together properly, its slots or tabs must be sized correctly for the thickness of the material being cut. If you want to cut a model out of a material of a different thickness than it was originally designed for, or if you've adjusted the scale of the entire design, you can use LightBurn's Slot & Tab Resizer to adjust the size of the slots without having to alter each one individually.

1. Measure Existing Slots

Before using **Resize Slots**, you'll need to know the size of the original slots or tabs in the design we want to adjust.

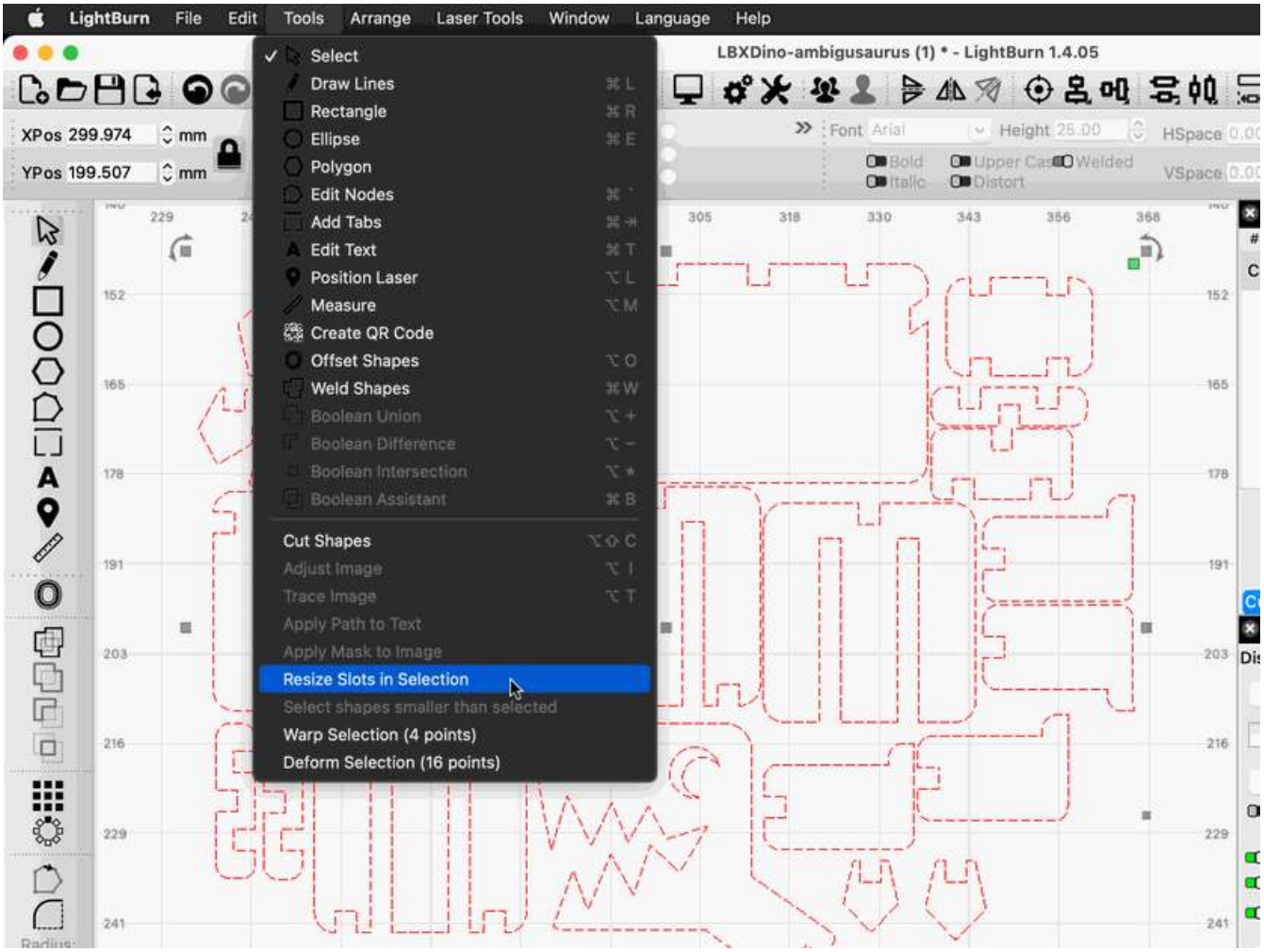
1. Click the View Shape Measurements button in the left toolbar.

2. Hover over the segment of the slot or tab you want to adjust. We'll be adjusting slot width, so that's the segment we'll hover over.
3. The length of the line is displayed as the **Segment Length**. Our design has a starting slot width of 3 mm, because we designed it to be cut out of 3 mm thick material.



2. Prepare The File

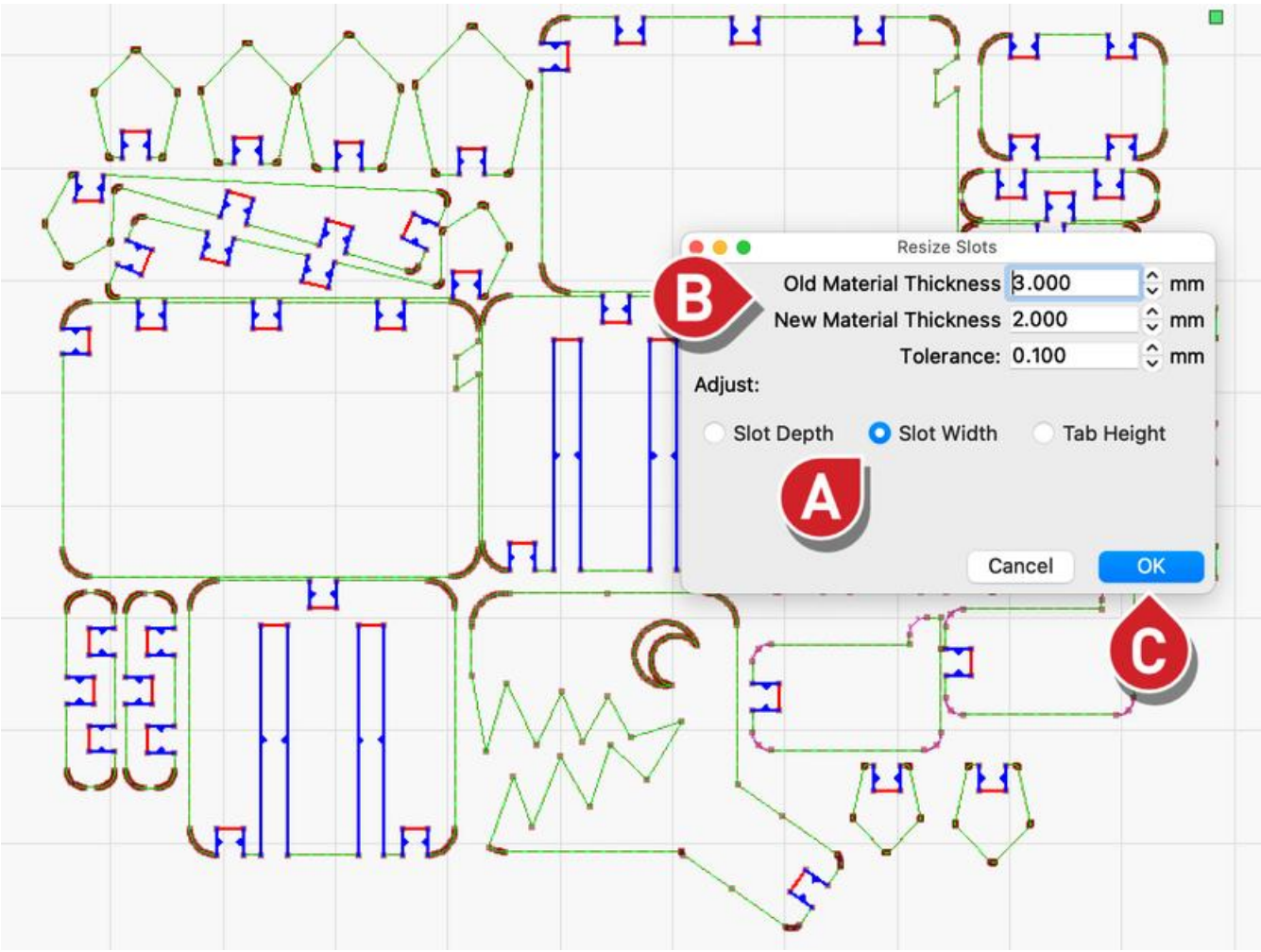
1. Before opening the Resize Slots tool, it's important to make sure none of the graphics are grouped, or the resizer won't be able to identify the slots or tabs. You can ungroup graphics by selecting them and clicking the Ungroup button or pressing ++ctrl+u+ / `Cmd`+`U`.
2. Select the parts that need to be adjusted. Pressing `Ctrl`+`A` / `Cmd`+`A` will select all graphics in your project.
3. Go to **Tools → Resize Slots in Selection** to open the Resize Slots tool.



3. Resize Slots

1. Select the **Adjust: Slot Width** option
2. Set **Old Material Thickness** to the width you measured in the previous step (3 mm in our case), and **New Material Thickness** to the thickness of the new material you'll be cutting (2 mm in our case).
3. Now that we've set the proper values, click **OK** to apply the changes.

The sample file only needs adjustment for the **Slot Width**, but the same process can be used to adjust **Slot Depth** and **Tab Height** as well. Our Slot & Tab Resizer documentation has more detailed information on this process, including troubleshooting tips.



Kerf Offset

To ensure a snug fit between parts, you'll also need to set an appropriate **Kerf Offset**. Kerf is the thickness of the material burned away as your laser cuts, and is determined by the width of the laser's beam and the type of material you're cutting. This thickness needs to be accounted for when cutting slotted or tabbed parts, or the laser will burn away too much material, and the slots or tabs will end up bigger or smaller than they ought to be. Once you've determined the kerf, you will use the Kerf Offset setting to account for the kerf without modifying the underlying shape.

There are a number of different tests to determine your laser's kerf for a given material, and to calculate the necessary **Kerf Offset**. In this guide, we'll be using a test file with a built in Vernier scale designed by Billie Ruben, a LightBurn staff member. The file contains detailed instructions for use, and you can also use our guide. Both are linked below.

Download Kerf Test File

Kerf Test Guide

1. Cut the Test File



1. Open the test file and set up the **Cut** and **Score** layers with your usual settings - if you're unsure of what settings to use, you can use LightBurn's Material Test feature and some additional spare material to find out. Make sure the **Kerf Offset** is set to 0 in the Cut Settings Editor.
2. Load your material into the laser and cut the test file. You may want to place an additional sheet of material underneath to make removal easier and prevent small parts from falling through if cutting on a honeycomb bed.
3. Remove the test part from the laser, keeping the pieces in place during removal by applying a piece of masking tape over the entire test part.
4. Place the part on a smooth, stable surface. Discard the piece marked **B**, then slide the remaining pieces all the way to the right. If the parts scatter, use the horizontal line as a guide to place them back in the correct orientation.

2. Use the Vernier Scale



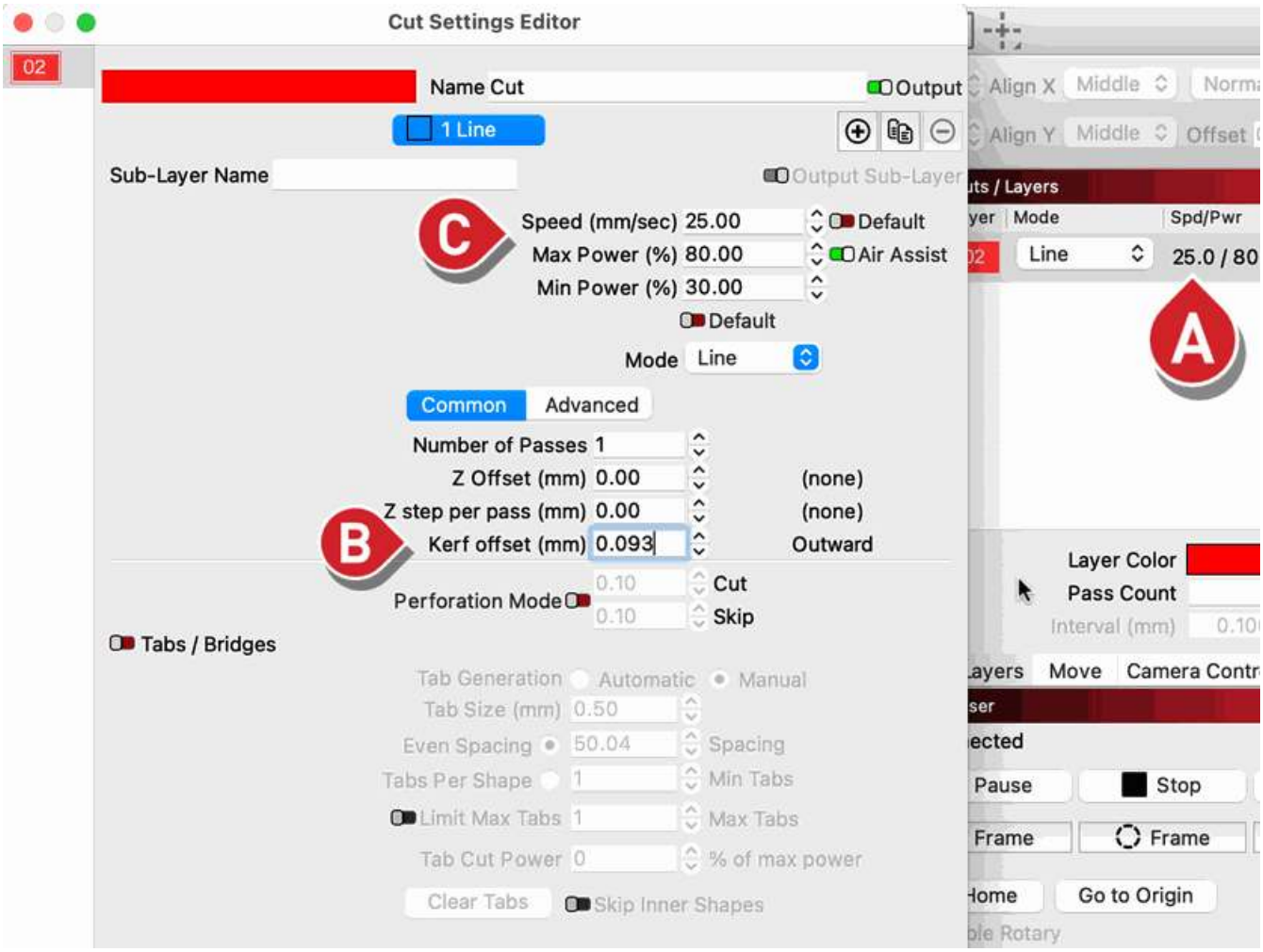
1. Find and record the largest number the line marked **D** passes. In this example, it's **3**, meaning that the measurement is at least 3 mm, but less than 4 mm.
2. To determine the value to the right of the decimal place, find which of the numbered lines to the left of **E** matches up closest to the lines etched on the bottom frame of the test piece. In this example, it's **7**, meaning that the measurement is **3.7 mm**.
3. To get the width of a single kerf, divide the measurement by 20. In our example, the width of the kerf is **0.185 mm**.
4. Next, we'll divide the total kerf by 2 - since half of the kerf will be within the material we're cutting away, we only need to worry about offsetting the half that's cutting into the material we want to keep. That gives us a final Kerf offset value of .0925.

3. Set Kerf Offset

1. Open the Cut Settings Editor by double-clicking the **Line** layer you've got your graphics set to in the Cuts/Layers window.
2. About halfway down the window is Kerf offset. Enter the value you calculated in Step 2. Make sure the value is positive for an **outward** adjustment.
   - Positive values will adjust cuts **outward**, or away from the shapes being cut.
   - Negative values will adjust cuts **inward**, or into the shapes being cut.
3. Make sure the speed and power settings are the same as those you used when you cut the test file in Step 1.

Now the cut path will be offset to account for the laser's kerf without changing the underlying shape, and the slots will be accurately sized to the thickness of the material we're cutting. We're ready to cut our model!

> **Why not just offset the shape?**
>
> The **Kerf Offset** will change based on the material and layer you're using. If you offset the shape itself, the adjustment won't necessarily be right for next time. Using Kerf Offset Adjustment will let you adjust for your kerf on each material without having to keep track of and undo previous changes.



Other Ideas

Using Resize Slots and Kerf Offset Adjustment is great for multi-part designs that need to fit together.

Check out our list of external resources for some great tools to create multi-part designs, such as Boxes.py and the Jigsaw Puzzle Generator we used in our Pet Puzzle Postcard project.



Laser Cut Fridge Magnets

If you’re anything like me, you’ve got a few stickers lying around that you don’t know what to do with. It’s not that I don’t like them — quite the opposite, I’m afraid to commit them to anything impermanent. Finally, it hit me — you know what’s not going anywhere? Magnetism or refrigeration. One’s a law of nature, the other of food safety. Thus was a project idea born - turning stickers into fridge magnets by using a laser to cut out form-fitting backing pieces for the stickers, and then attaching magnets to them.

What You'll Learn

Below, we'll describe two different methods for cutting the backing pieces. In the first method, we'll:

- Check and adjust our Camera Alignment using the **Shift** fields in the Camera Control window
- Use the Trace feature to produce vector graphics from a camera capture of our stickers, pre-stuck to black acrylic in our workspace
- Edit the resulting vectors so we're left with just outlines of our stickers
- Use Absolute Coordinates positioning to cut those outlines precisely around the stickers

In the second, we'll:

- Take a photo of our stickers, laid out on black acrylic, with a digital camera or smartphone
- Import the photo into LightBurn
- **Trace** the imported photo to create outlines of the stickers
- Edit the resulting vectors so we're left with just outlines of our stickers
- Precisely rescale the outlines to size them correctly

Materials

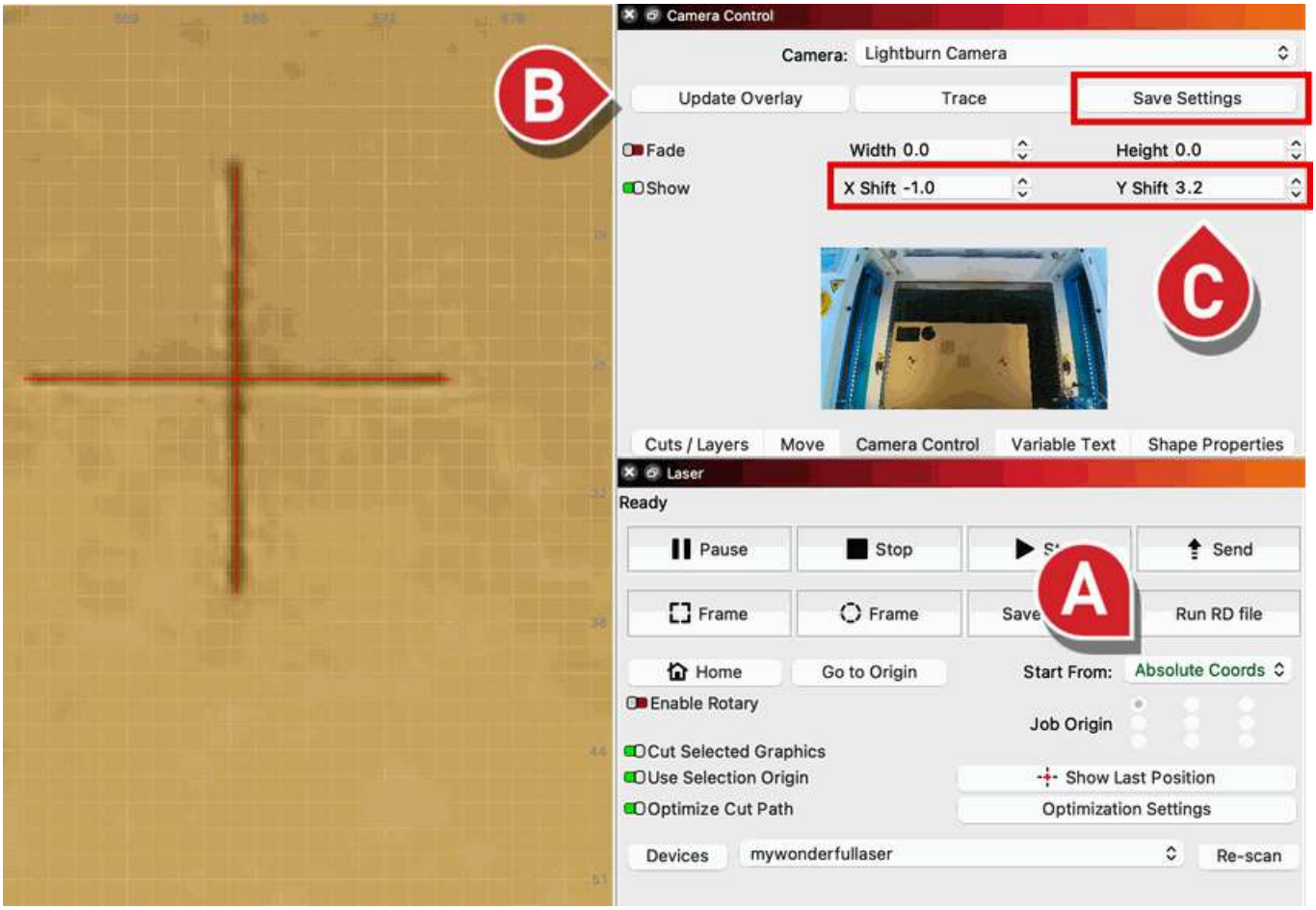
- A USB camera, mounted above your laser and calibrated and aligned in LightBurn **OR** any digital camera you can use to take a photo and transfer to your computer. A smartphone will work.
- To calibrate **Camera Alignment** and follow the first method, you will need a laser that has limit switches and can home automatically.
- 1/16" black acrylic. Very thin acrylic is great for this project because it's easy to cut and lightweight. You can choose any color you like, but black is strongly recommended because it will make it easier to get a clean **Trace**.
- One or more of your favorite stickers.
- Magnets with adhesive backing.

Method One — Stick & Cut

This method makes use of LightBurn's ability to produce an overlay from a USB camera mounted above your laser's workspace. Your stickers will be attached to the backing acrylic, and the laser will cut around them, so you'll need to have high confidence in the accuracy of your overlay's alignment, or the stickers may get cut into. If you're not prepared to risk your stickers, skip ahead to method two.

1. Verify and Adjust Camera Alignment

Before beginning this project, you must **Calibrate** your Camera Lens and Camera Alignment in LightBurn. After the initial setup, you should check on the alignment before proceeding with the project.



1. To test **Camera Alignment**, use the Draw Lines tool to create a crosshair mark set to a Line layer in your LightBurn workspace. Set the Start From mode to **Absolute Coords**, and mark the crosshair onto piece of material placed in your laser. Cardboard works great for this step — you want a strong contrast.
2. Then, click **Update Overlay** in the Camera window, and see how far off the marked crosshair is from the crosshair design in LightBurn. If they're perfect, you're all set!
3. If they're not perfect but relatively close, adjust **X Shift** and **Y Shift** until they overlap perfectly, click **Save Settings**, then re-test.
4. If they're very far apart, re-run **Camera Calibration** and **Alignment**, then re-test.

If you're having trouble getting a consistent alignment, here are some tips:

- Make sure your camera is securely mounted, with no wiggle on its mounting point.
- If it's mounted to a lid, make sure the lid is returning to the exact same position each time you open it.
- If your laser does not have an adjustable height bed, make sure the material you use in the **Camera Alignment** step is the same height as the acrylic you use for the project. An alignment is only valid for a constant distance between the camera and a given surface height.
- Even subtle changes in the camera’s position relative to your laser’s workspace or material surface can make an important difference in alignment.

2. Stick the Stickers, and Load the Acrylic

Next we'll adhere the stickers to the acrylic. There's no turning back after this point - your stickers are committed now!

1. Stick your stickers to your sheet of black acrylic.
2. Place the acrylic in your workspace.
3. Position the acrylic so that the stickers are as close to the center of your workspace as possible - the accuracy of the alignment will be better toward the center in most cases, and can drift a bit toward the edges.

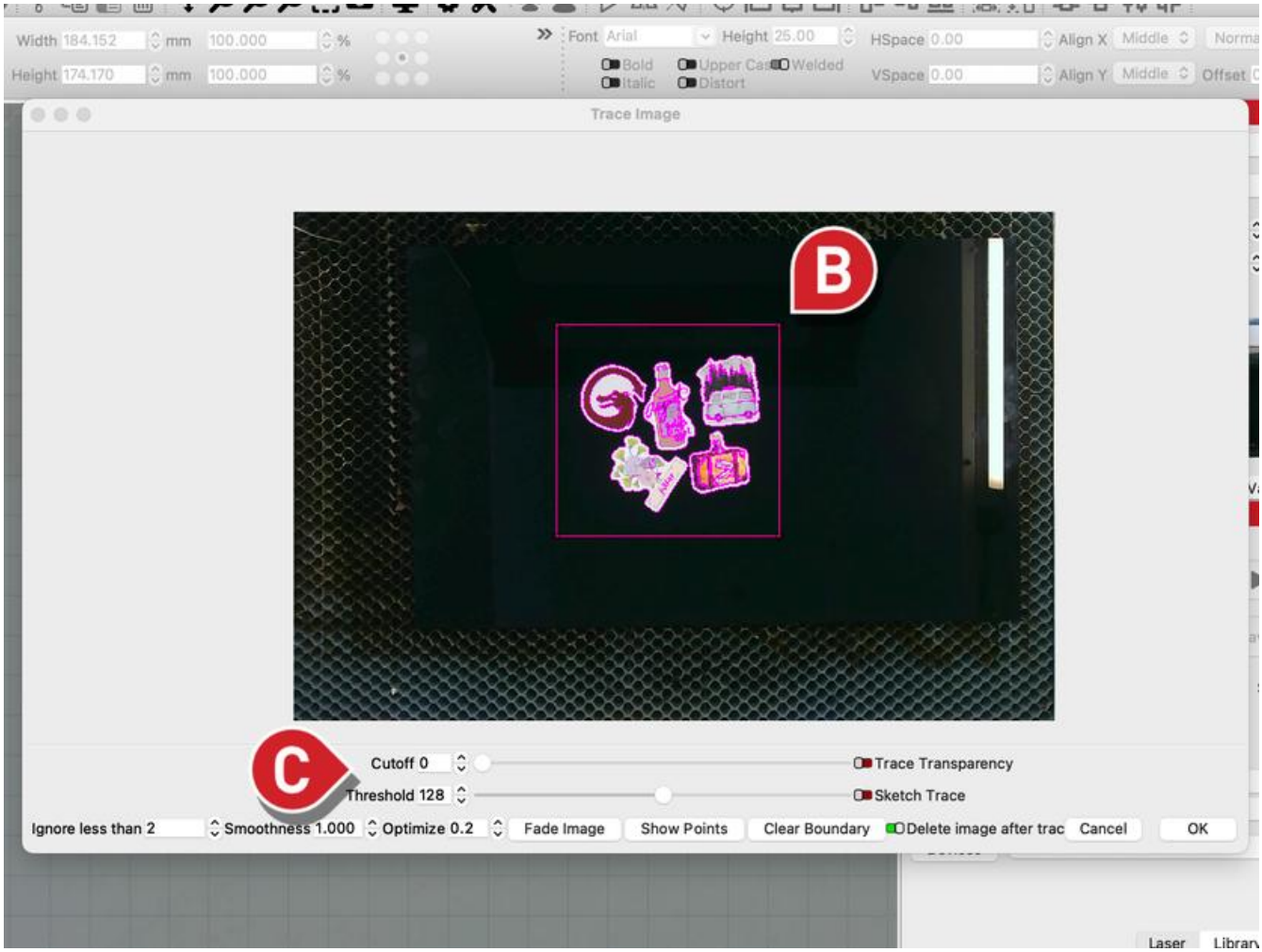


3. Trace Image

Next, we'll use **Image Tracing** to make outlines around the stickers.

1. Click **Trace** in the **Camera Control** window
2. In the Trace window, box-select only the portion of the capture where your stickers are.
3. With the black background, you should have no trouble getting outlines of each of the stickers, but, if necessary, you can adjust the **Cutoff** and **Threshold** sliders until you get complete outlines for each sticker.

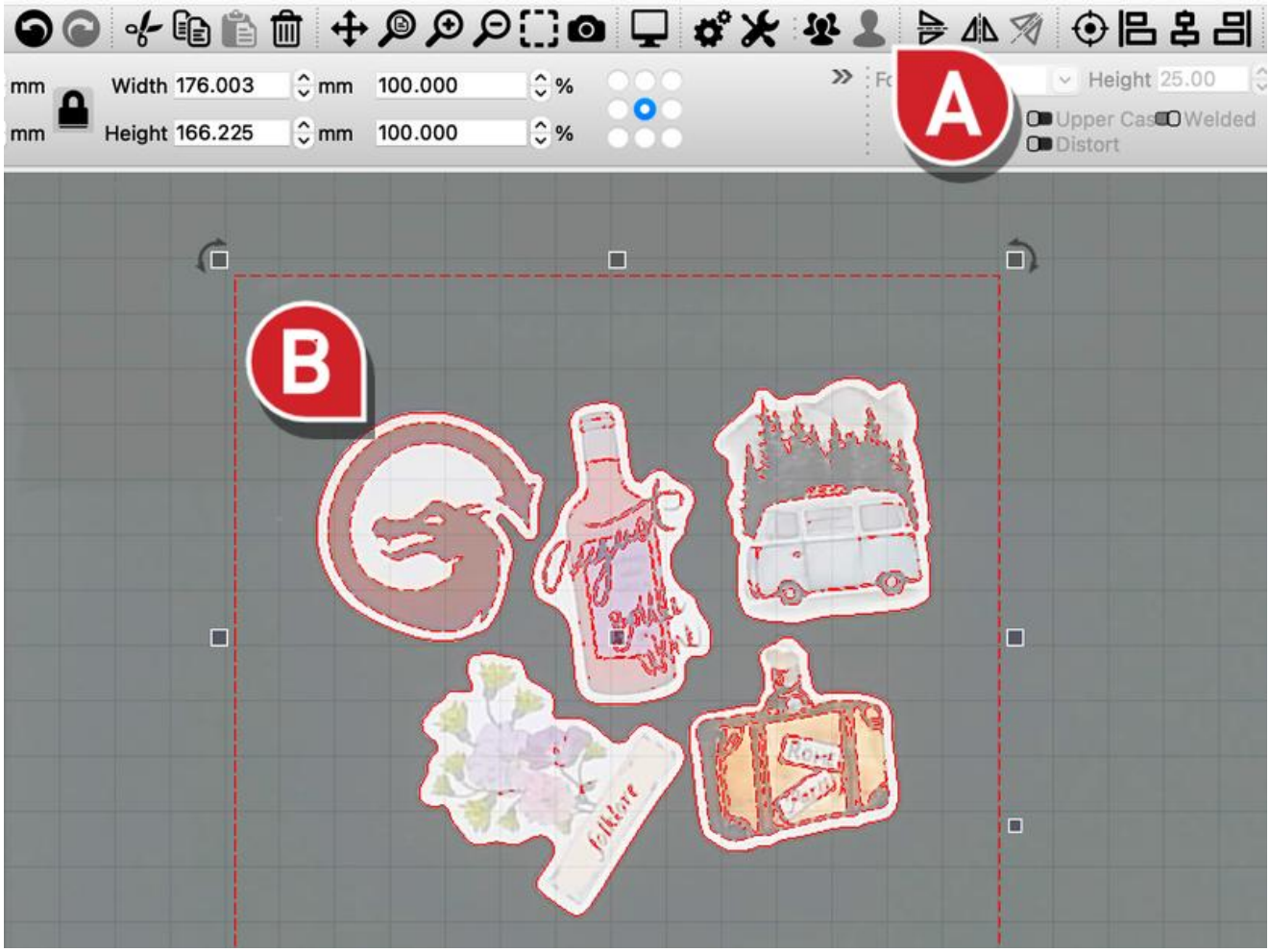
4. The Cutoff slider controls the lower end of the range of brightness values that LightBurn will outline with vectors, and the Threshold slider sets the upper end.



4. Edit Vectors

The resulting vectors will automatically be selected and grouped. In addition to the outlines, they'll likely include some additional graphics you don't want.

1. Ungroup (`Ctrl`+`U` / `Cmd`+`U`) the vectors, but keep all of them selected.



2. Then, Hold `Ctrl` or `Cmd` while clicking on each of the outlines of the stickers to Deselect them.
3. When all of the outlines are deselected, **Delete** the remaining selection.



5. Cut Around the Stickers

This is the big moment, it's time to cut out the stickers. Reminder: make sure you're confident in the accuracy of your overlay alignment before running the job. If you're not, head back to Step 1.

1. Assign the outlines to a layer set to Line mode. Apply appropriate settings for your laser to cut through 1/16" acrylic. If you're unsure on settings, run a Material Test.
2. When you updated the overlay earlier, Absolute Coords was automatically selected as the **Start From** mode. Verify that it is still selected, and then run the job.
3. Remove the sheet of acrylic from your workspace, leaving only the acrylic-backed stickers.
4. After pausing briefly to admire your handiwork, remove those too.



Method Two — Cut & Stick

This method doesn't put your stickers in any danger of being accidentally lasered, but the stickers may not end up as perfectly placed on the backing pieces as they are in the method one, and it requires a bit of math.

1. Take a Photo

Using a digital camera or smartphone, we'll start by taking a photo of our stickers and a reference object, laid out on black acrylic.

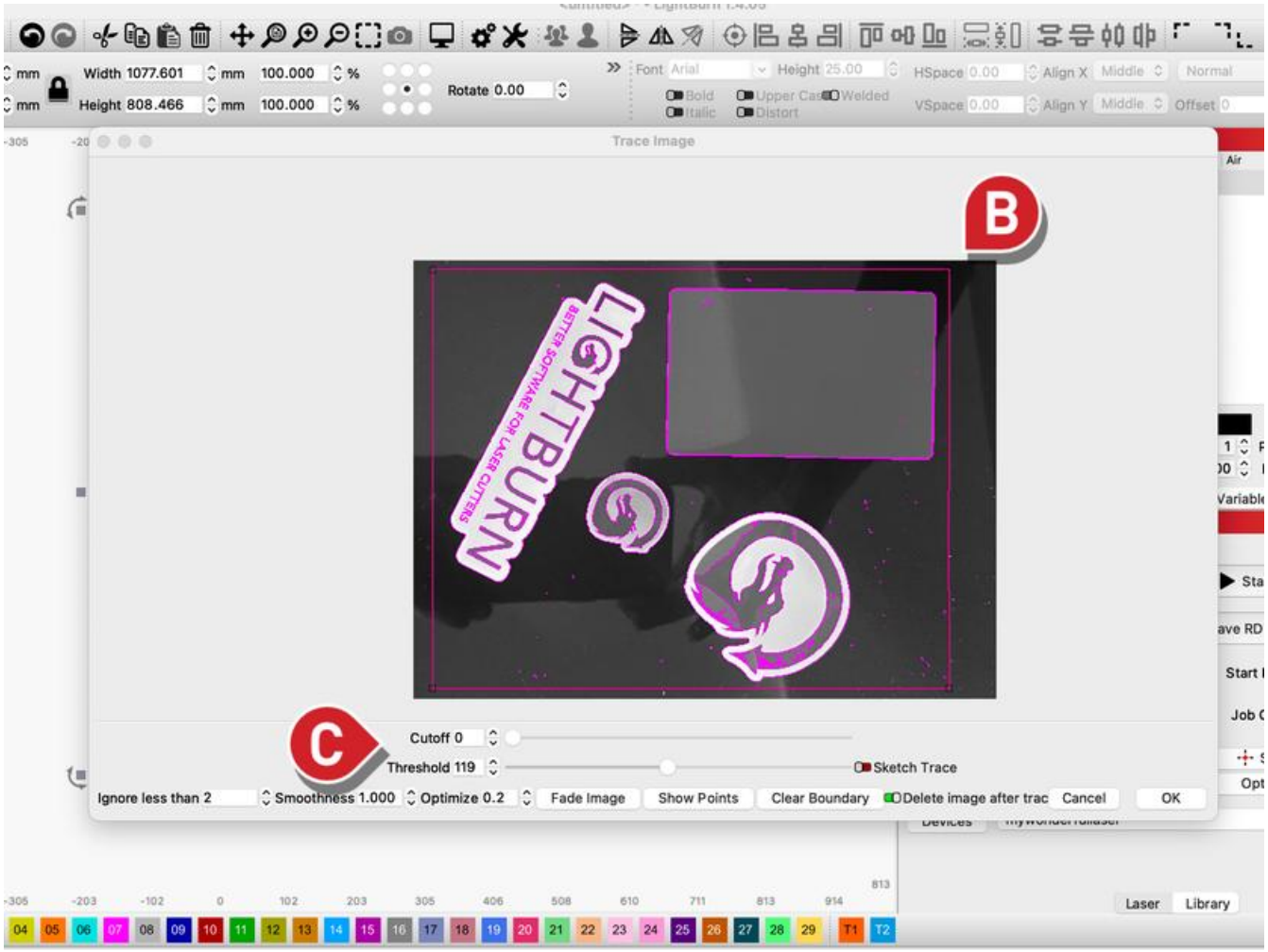
1. Place your stickers and a flat rectangular or square reference object (we used an aluminum business card blank) on a sheet of black acrylic.
2. Take a photo of the stickers and the object. Try to keep the reference object roughly horizontal/vertical in the camera’s field off view. To avoid perspective distortions, keep the stickers centered and the camera angle as top down as possible.
3. Send the photo to your computer by SD card, Airdrop, email, or a cloud storage service.



2. Import and Trace

Next we'll **Import** the photo into LightBurn and **Trace** it.

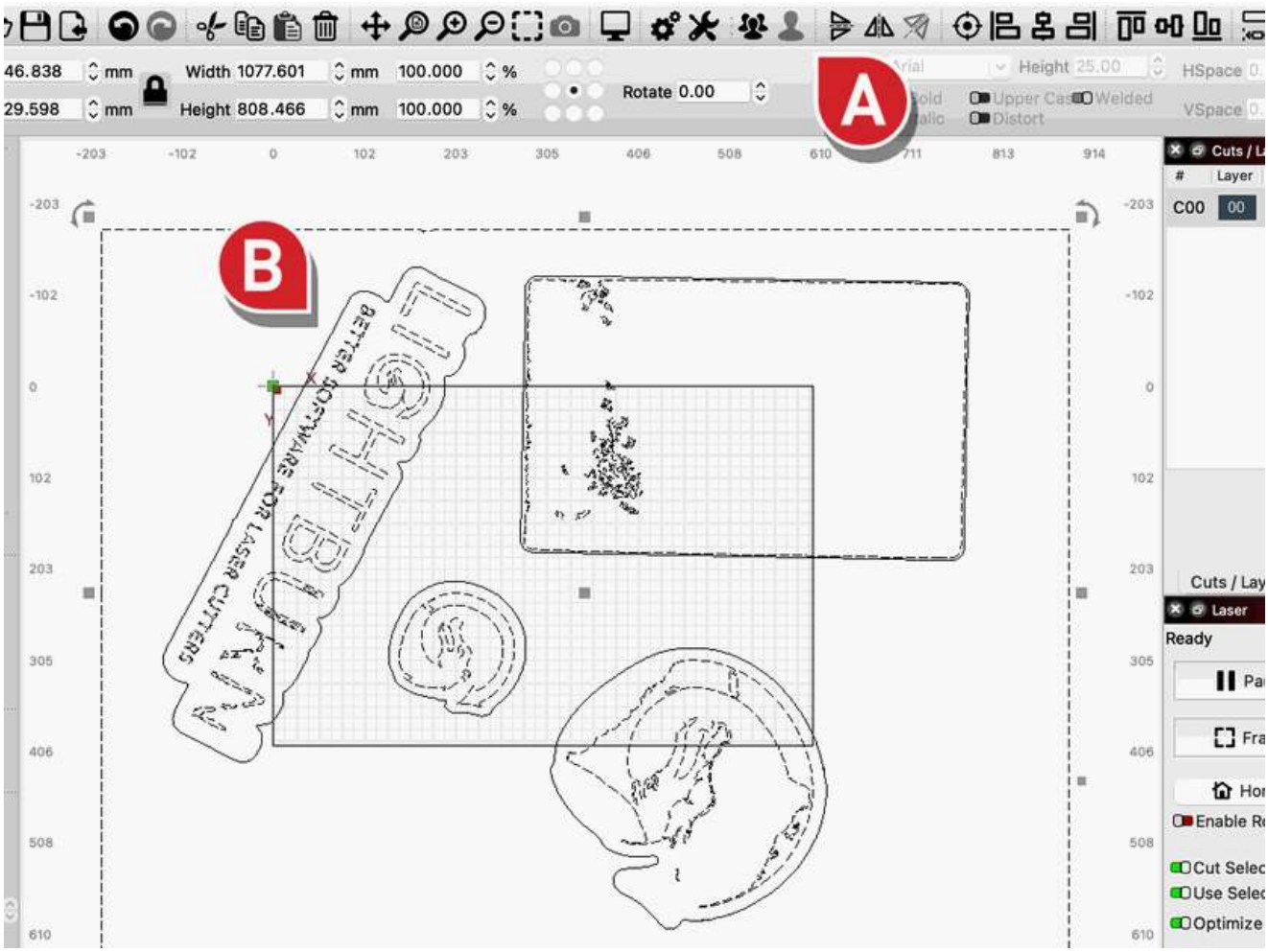
1. Import (`Ctrl`+`I` / `Cmd`+`I`) the photo into LightBurn.
2. Right-click the image and select Trace Image. In the Trace window, box-select only the portion of the capture where your stickers and object are.
3. With the black background, you should have no trouble getting outlines of each of the stickers and object, but, if necessary, you can adjust the **Cutoff** and **Threshold** sliders.
4. The Cutoff slider controls the lower end of the range of brightness values that LightBurn will outline with vectors, and the Threshold slider sets the upper end.



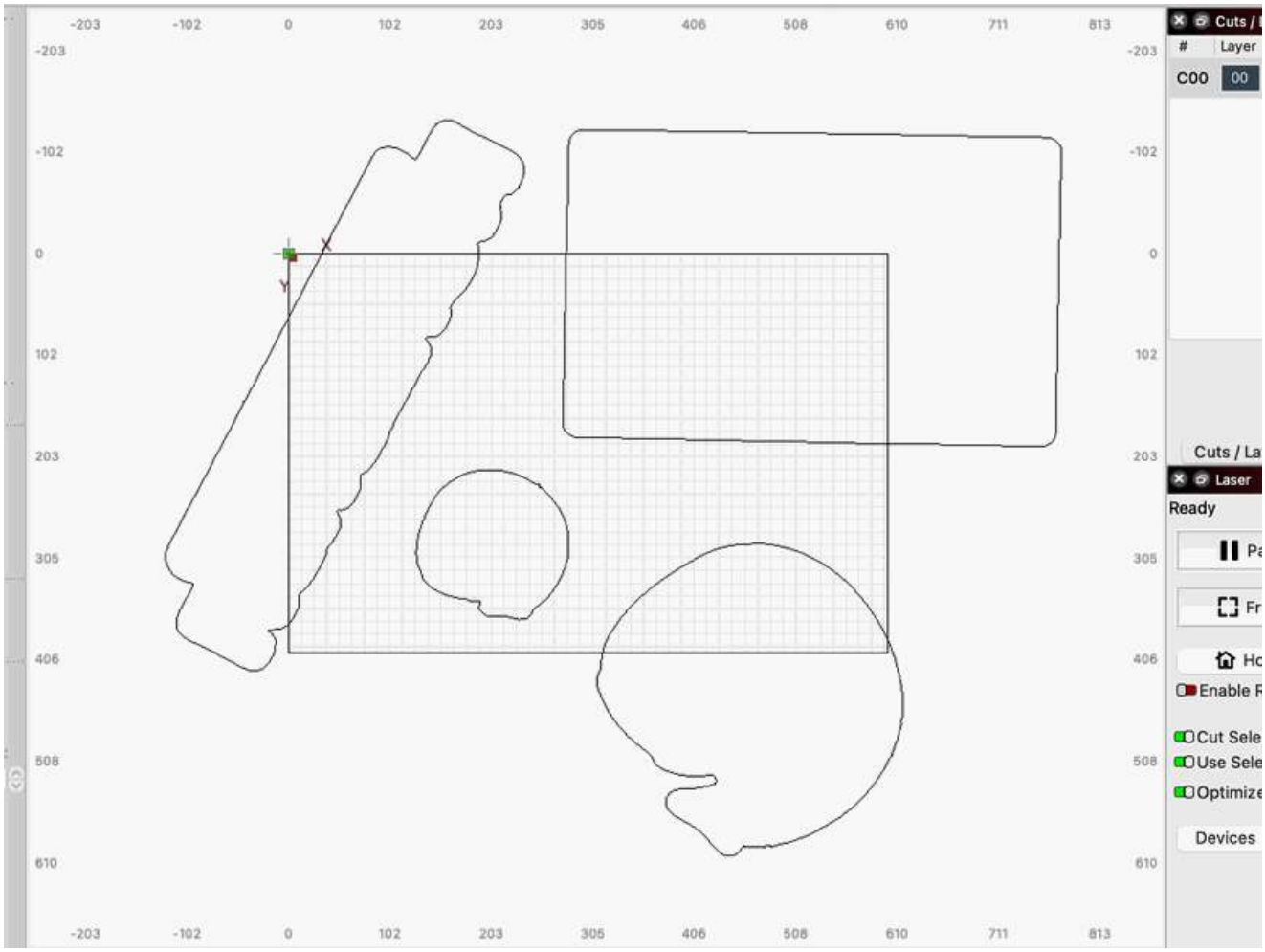
3. Edit Vectors

The resulting vectors will automatically be selected and grouped. In addition to the outlines, they'll likely include some additional graphics you don't want.

1. **Ungroup** (`Ctrl`+`U` / `Cmd`+`U`) the vectors, but keep all of them selected.
2. Then, Hold `Ctrl` or `Cmd` while clicking on each of the outlines of the stickers to **Deselect** them.



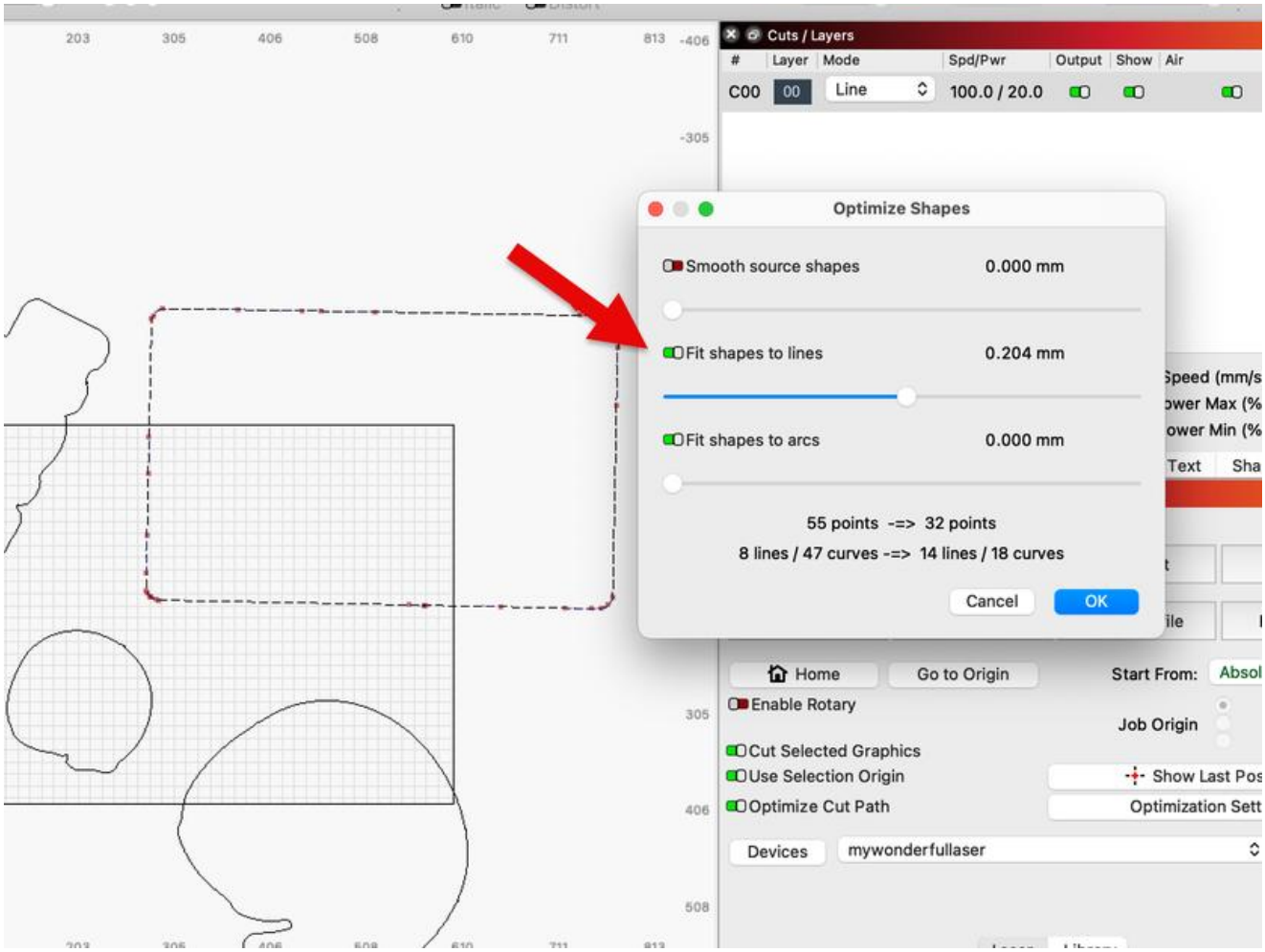
3. When all of the outlines are deselected, **Delete** the remaining selection.



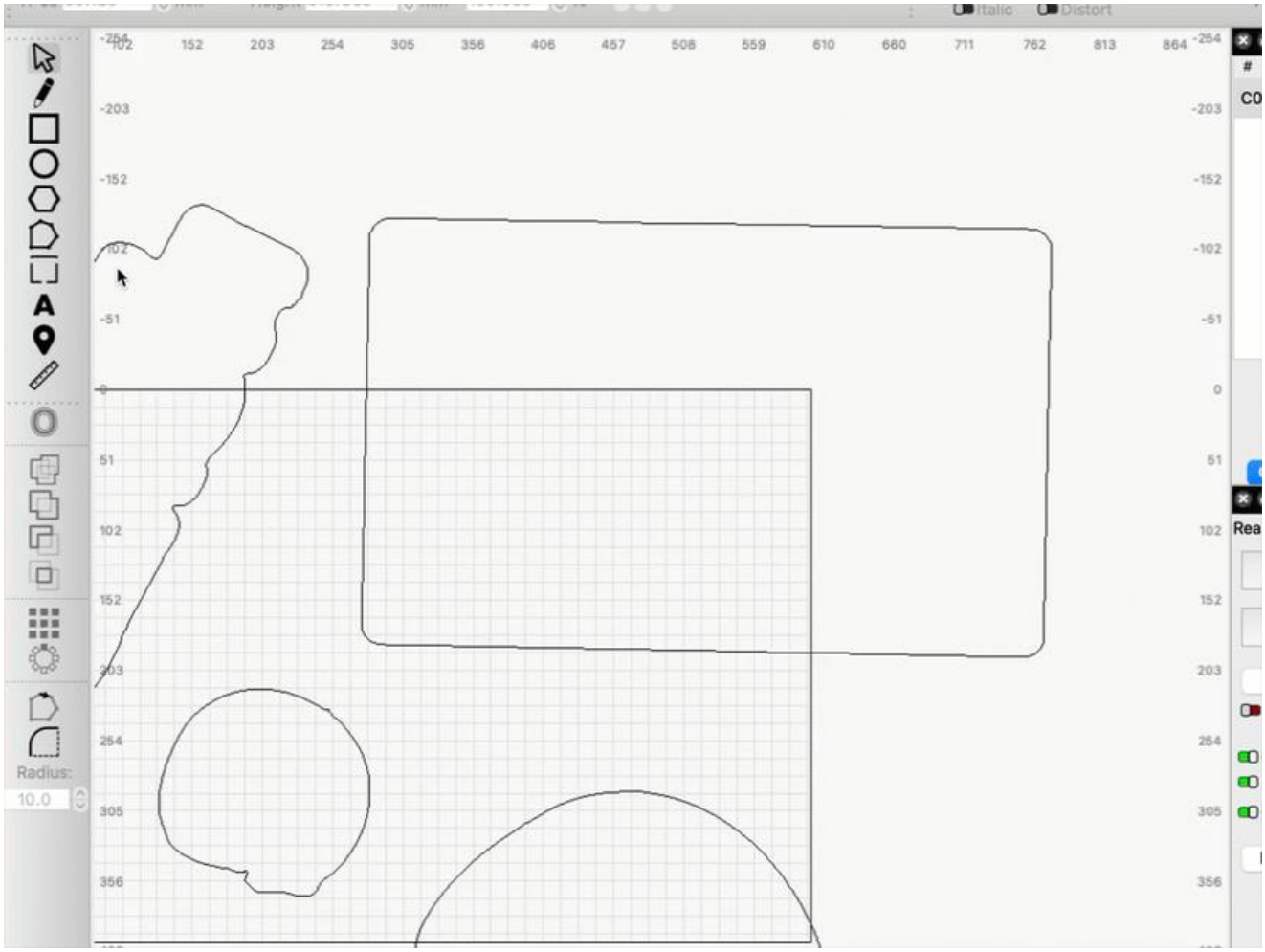
4. Rescale Outlines

Here comes the math part — don't worry, it's simple and will be over quickly.

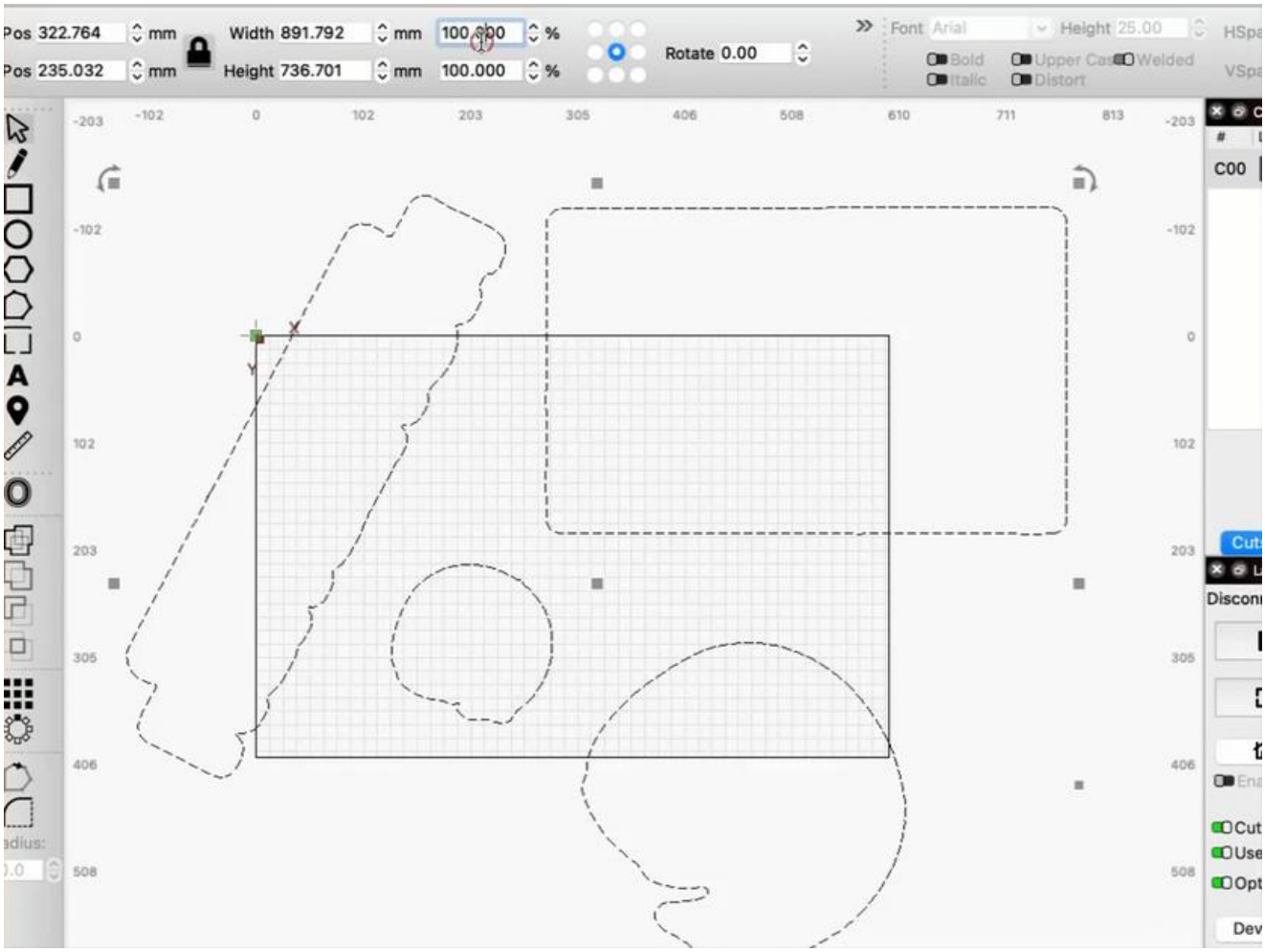
1. To get the outline of the reference object perfectly vertical or horizontal, first Select it and go to **Edit** → Optimize Selected Shapes and use the **Fit shapes to lines** slider to fit it to lines. About halfway along the slider will do.



2. Then, select the Edit Nodes tool, hover over one if its edges, and press A to align it to the nearest vertical or horizontal angle.



3. **Select** the outline of the reference object you traced, and read its **Width** or **Height** measurement in the Numeric Edits toolbar.
4. Divide the known Width or Height of the actual object by the Width or Height of the outline you read in the Numeric Edits toolbar. In our case, the known Width of our cards was 86 mm, and the Width in LightBurn was 494.115 mm, for a result of ~0.174. We'll use that value to rescale our graphics.
5. Select all the rest of the graphics and enter the value, expressed as a percentage, in the **Numeric Edits** toolbar. For our result of .174, that meant entering 17.4% in the Numeric Edits toolbar. All of the outlines will now be scaled to the actual size of your stickers.



6. **Select** and **Delete** the outline of the reference object.

Note: it can be hard to get the stickers lined up perfectly to these backing pieces, so you can elect to rescale the outlines to be just a touch larger than real size, or add an Offset after rescaling them, to provide a bit of an extra border.

5. Cut the Sticker Backing Pieces

1. Set the outlines to a Line layer. Apply appropriate settings for your laser to cut through 1/16" acrylic. If you're unsure of what settings to use, run a Material Test.
2. Use your preferred Start From mode to position your job and cut the outlines.
3. Remove the backing pieces from the laser and very carefully stick the stickers onto them — it can be tricky to get them lined up perfectly this way, so go very slowly.



Both Methods

Stick on the Magnets

We're almost done, this part is easy!

1. Peel one side of the adhesive and apply it to a magnet.
2. Then peel the other side of the adhesive and attach the magnet to an acrylic backing. Repeat for all magnets and backings.
3. If your stickers are relatively large, you may want to strategically place a few magnets around the edges.
4. Put your new fridge magnets on your fridge!


ROME
Paris
folklore
August
SIPPED AWAY LIKE A
Bottle OF Wine
LIGHTBURN
BETTER SOFTWARE FOR LASER CUTTERS



Laser Engraved Pencils

Custom engraved pencils can be a quick and easy way to make useful gifts, or alternatively, might help keep your pencils from vanishing. We've made these pencils as giveaways at events, and I've made class sets of them for teachers. If you want to make pencils like these, you can follow along with the project as it's written, or you can use the methods described here to branch off into customizing other items.

The trickiest part of engraving a pencil is that the flat space you have to work with on a typical pencil is only about 3 millimeters tall, so you need to be really precise in how you place your artwork onto the pencil. On a larger object, you can use the Camera Overlay to place your design on the material, or you can jog to the corner or center and use Current Position, like we did in the Image Engraving on Coasters project. But for this project, where being off by a millimeter just won't work, we'll take a look at how to use a template with a fixture to hold the pencils exactly where we want them.

What's a fixture? Isn't this a jig?

Short version: What you'll make in the first part of this project is often called a jig, and people will know what you mean if you call it that. Technically, though, it's a fixture.

Pedantic version: Jigs and fixtures aren't quite the same thing. A jig is used to hold a part in place and guide a tool. A fixture just holds the part in place. So this is a fixture, but regardless of what term you use, creating something to hold the items you want to engrave will make your life a lot easier.

What You'll Learn

- How to select a Start From mode in LightBurn and use a fixture to position things in your laser's workspace the same way every time.
- How to use tool layers to place your design in the right place relative to a template.
- How to use a Grid Array to repeat artwork.
- Optionally, how to use Variable Text to engrave something different on each pencil.

Materials

How much material do I need for the fixture?

The biggest single piece is 170 millimeters x 187.6 millimeters (just under 6 ¾ inches x 7 ⅜ inches), and the full design as provided is 170 millimeters x 226.6 millimeters (just under 6 ¾ inches by 9 inches). You can rearrange the rails to fit differently on your material with the baseplate or cut them from a separate piece of scrap material if you don't have enough space on a single piece.

This version of the fixture is designed to hold 12 pencils, since they're typically sold in multiples of 12, and uses 12.7 millimeter (½ inch) spacing so you can enter the Grid Array spacing easily regardless of your units. If you want a more compact version that fits comfortably on a single 200 millimeter square sheet of material, reach out and we can create alternate versions.

- Pencils. Our fixture is designed for standard hexagonal pencils, although you could modify it for other pencil styles. You could try this with round pencils, but the curve of the surface will make it hard to get a consistent engraving.
- Material to make the pencil fixture. Our example uses 3 millimeter plywood, and plywood sold as ⅛" is usually close enough to work fine without adjustments. If your material is thinner or thicker, you'll need to adjust the slot width. Don't worry, LightBurn's got the Resize Slots tool to make it quick and easy.
- Our pencil engraving template. The downloadable file is a LightBurn Template File. When you open the file, it creates a new project ready for you to save without accidentally overwriting the original template.
- A laser. This project is intended for use with a laser that can home and use **Absolute Coords**, although I'll describe a workaround in case your laser doesn't have this.

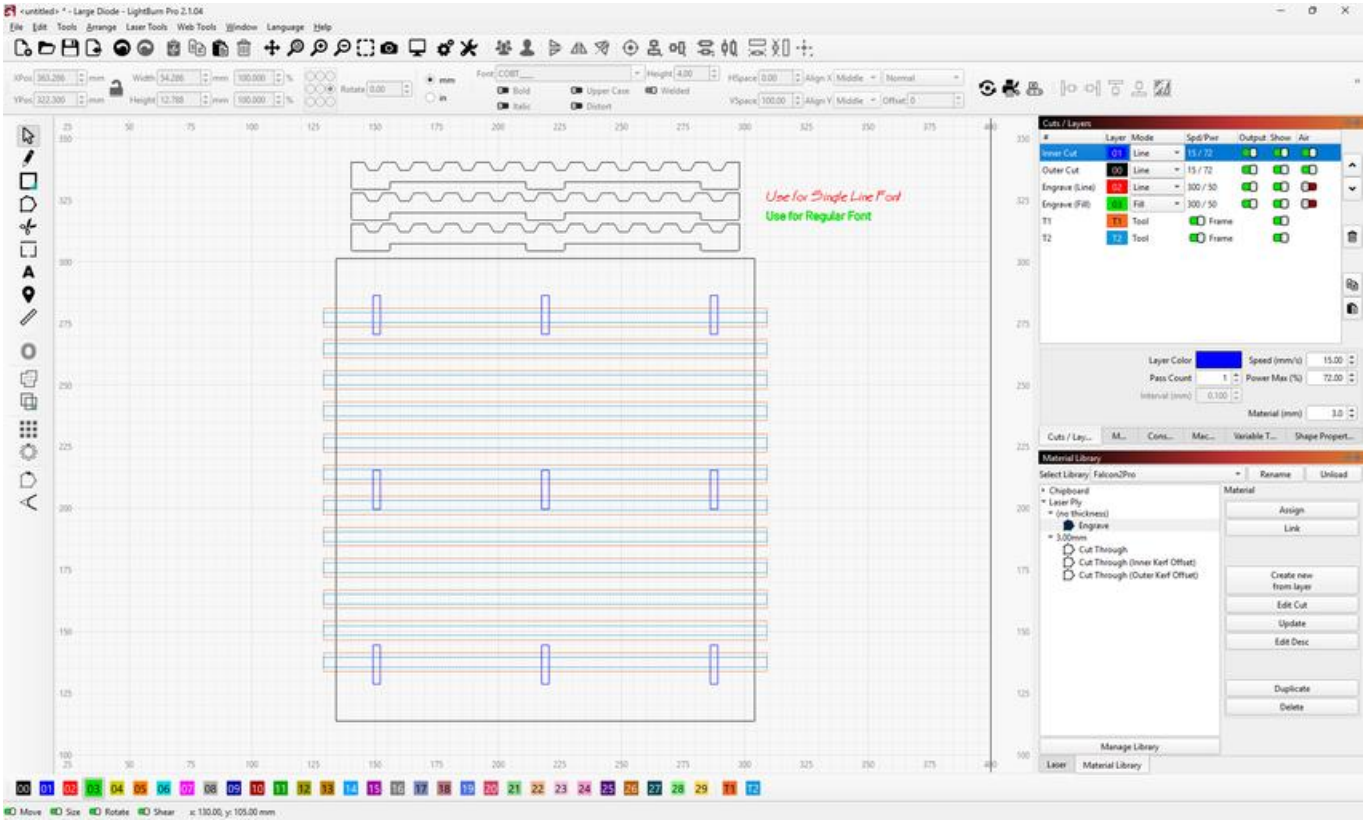
Steps

Part 1: Make the Pencil Fixture

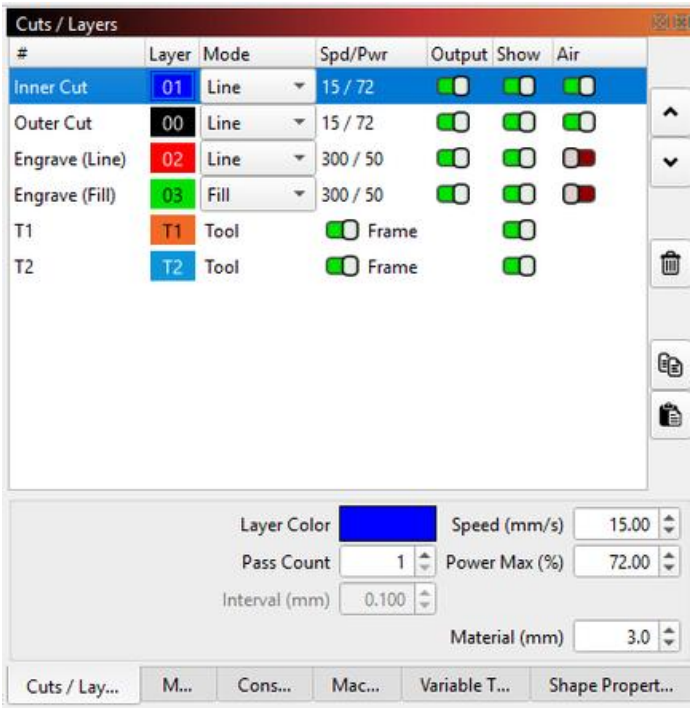
We'll start off by making a fixture that holds pencils in place while you engrave them. Our template holds 12 pencils at a time, but we'll put out additional versions for people with larger machines.

1. Open the Template File

Open the pencil fixture template file you downloaded and take a look around. The pencil placeholders are grouped together, the three rails are grouped together, and the baseplate and cutouts are grouped together. When you move the template around the LightBurn workspace, make sure you select all the items you want to move so they stay together.



- The cut lines for the fixture are on **Layer 00** ■ and **Layer 01** ■, and set to Line Mode. Layer 01 is only used for the inner cutouts where the rails will fit into the base plate.
- The orange and blue rectangles on the Tool layers are placeholders for the pencils. The orange rectangles on **T1** ■ represent the full width of the pencil, and the smaller blue rectangles on **T2** ■ show the engravable area on a typical pencil.
- **Layer 02** ■ and **Layer 03** ■ aren't visible when you first open the file, but Layer 02 is set up to engrave in Line Mode, and Layer 02 is set up to engrave in Fill Mode. When you add your text to one of these layers, the layer will show up with the name and settings I used.



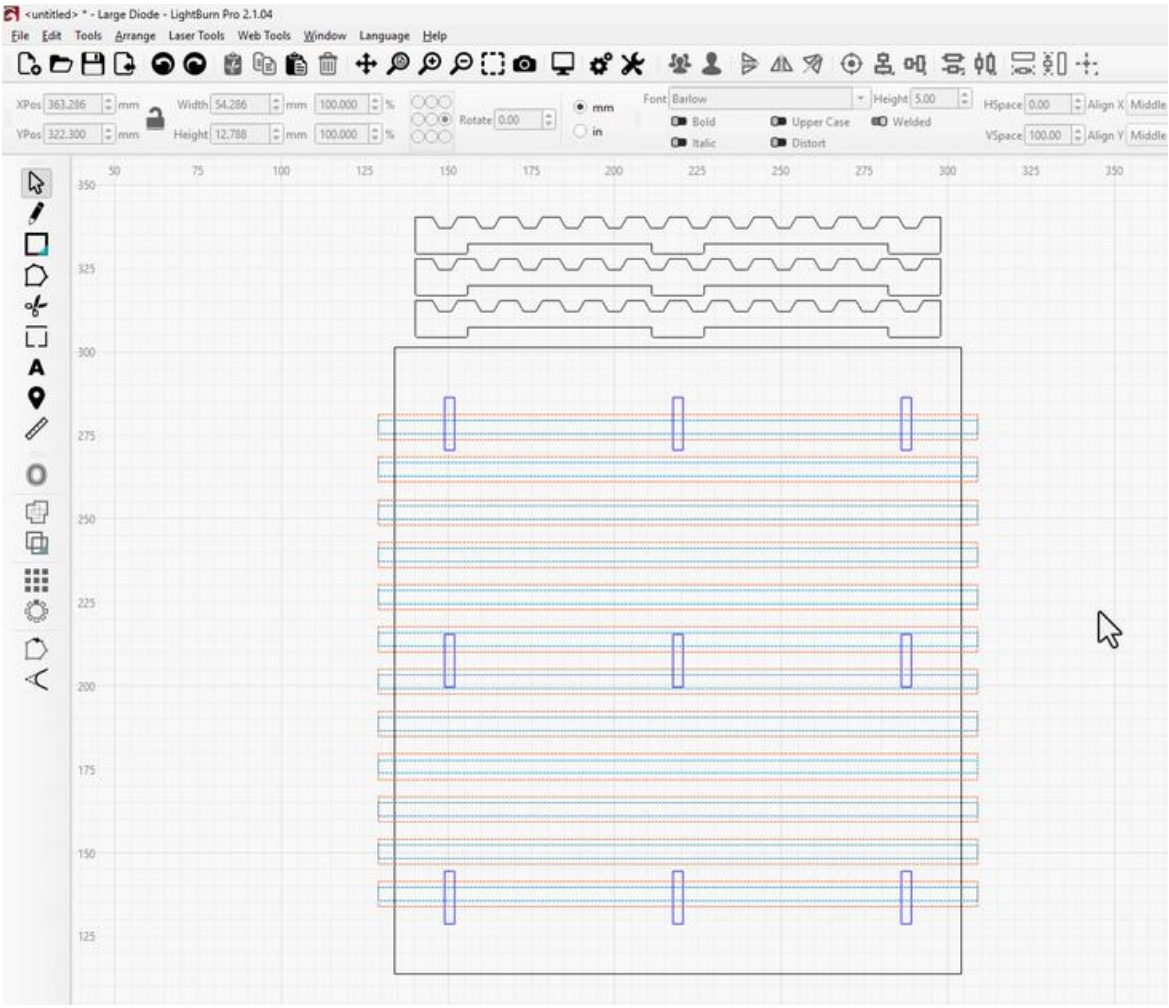
2. Adjust the Slot Width

If your material is 3 millimeters thick, you can skip this step. If you're using material sold as ⅛" thick, it's likely close enough to work without changes.

The three rails that will hold the pencils are meant to slot into the rectangles that will be cut into the baseplate. Whenever you make parts that are meant to slot into each other like this, you need to account for the thickness of the material.

If your material is not 3 millimeters thick, you'll need to make the slots narrower or wider to match. This is easy to do using LightBurn's Resize Slots tool. You'll need to Ungroup the shapes involved before you can use the Slot and Tab Resizer.

1. Click on a blue rectangle to select it. Because it's grouped with other shapes, you'll see a lot of shapes selected, with the dot-dot-dash pattern that LightBurn uses to show that the selection is grouped.
2. **Ungroup** using `Ctrl`+`U` / `Cmd`+`U` or by right-clicking and choosing Ungroup. You'll need to do this multiple times, until the blue rectangles are ungrouped.
3. Make sure that all the blue rectangles are selected, if they aren't already. You can shift-click on Layer 01 in the **Cuts / Layers Window** or drag a box around the blue rectangles to select them. It's okay if you have more than just the blue rectangles selected.
4. Go to **Tools → Resize Slots in Selection**.
    1. Set **Old Material Thickness** to **3 mm**.
    2. Set **New Material Thickness** to the thickness of your actual material.
    3. In the **Adjust** section, select **Slot Width**. You should see arrows showing the way your rectangles will be adjusted.
    4. Click **OK** to accept the changes.



If the **Resize Slots** window has a message warning you that "Your selection contains the following unsupported items: Grouped Shapes", you'll need to go back and ungroup your objects before you can use this tool.

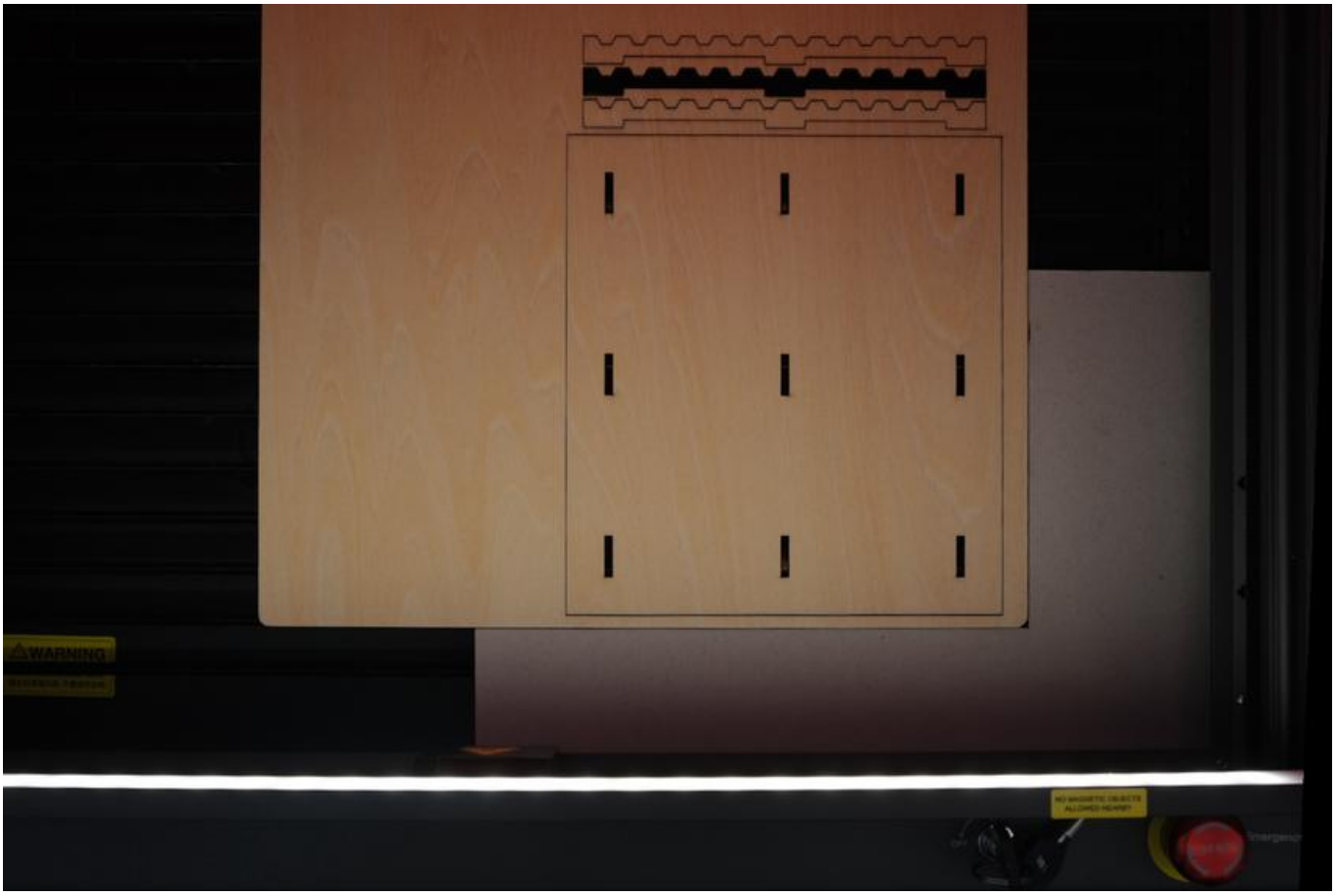
> **What About the Kerf?**
>
> If you want to really make this fit together nicely, check out the Kerf Offset setting, described in ourKerf Offset Test guide and our Laser Cut Dinosaur project. This will let you account for the thickness of the laser beam and the extra material that will be removed.
>
> For this project, though, making the slot width match the thickness of the material will give a close enough fit on most materials and lasers. If you want to really dial in your settings, or if you're having trouble getting your pieces to fit together, come back and take a look at the links here.

3. Cut the Fixture

It's time to start cutting! I'm assuming this isn't your first time using your laser, so I won't be going into a lot of detail here.

- Set your cut layers (**Layer 01** and **Layer 02**) to appropriate settings for your laser and the material you're using.

- Load the material into your laser and focus the laser
- Send the job to the laser and run it however you usually would.



4. Assemble the Fixture

Insert the rails into the baseplate, using a bit of glue to hold them securely in place. You don't need a ton of strength here, so just about any glue (within reason) will work. PVA glue (like Elmer's) and superglue will both work fine for plywood and similar materials.

5. Test the Fixture

Try loading up some pencils and make sure everything fits.

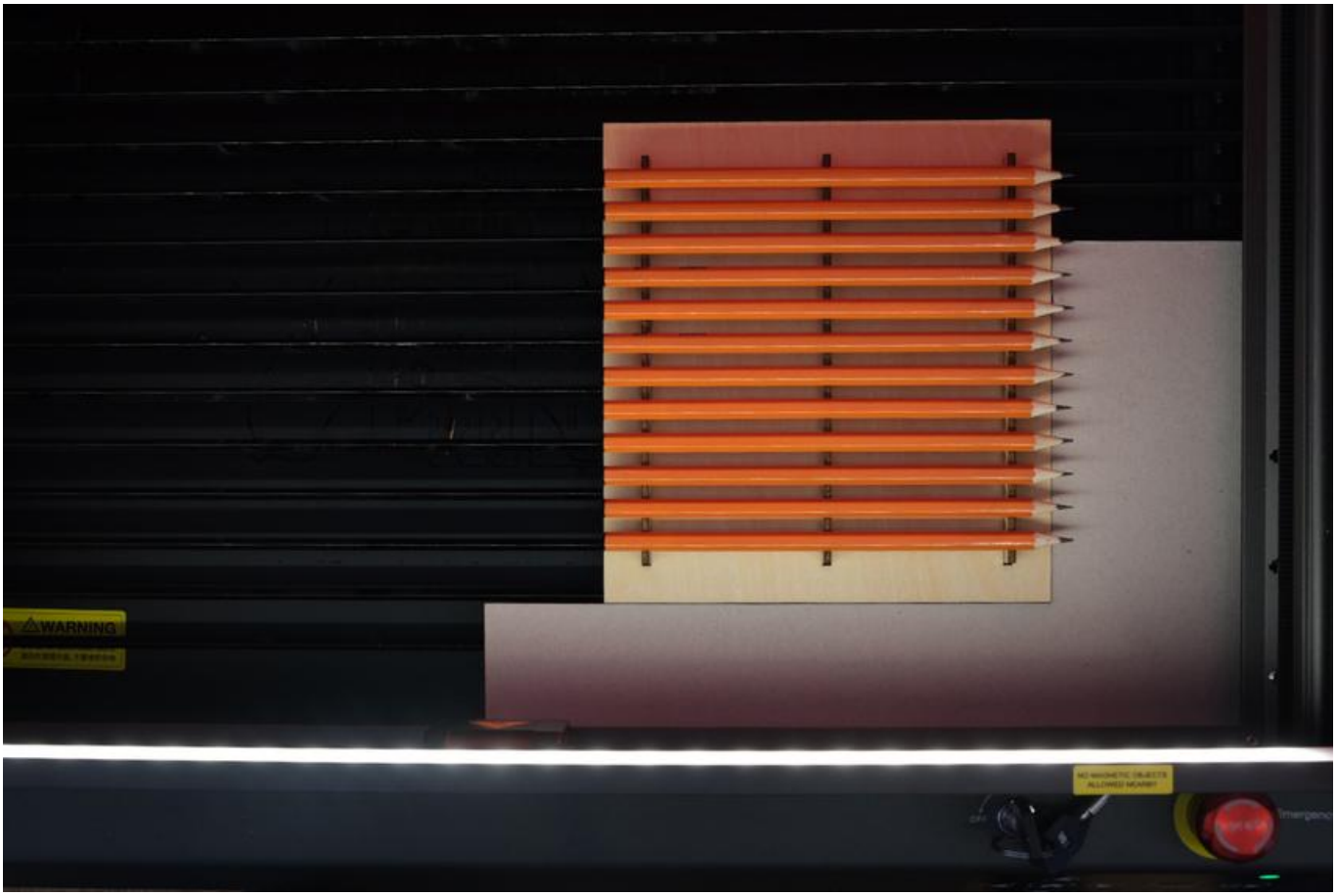
Ready? Let's go to Part 2!

Part 2: Engrave Your Pencils

Now that you've got your fixture, let's engrave some pencils!

1. Place the Pencil Fixture in Your Laser

Place the physical fixture in a corner of your laser and load pencils into it. You can use any corner that's convenient, as long as it matches up with a corner of your workspace in LightBurn. We're going to use a corner because it's an easy way to match the physical location of your fixture with its positioning in LightBurn.

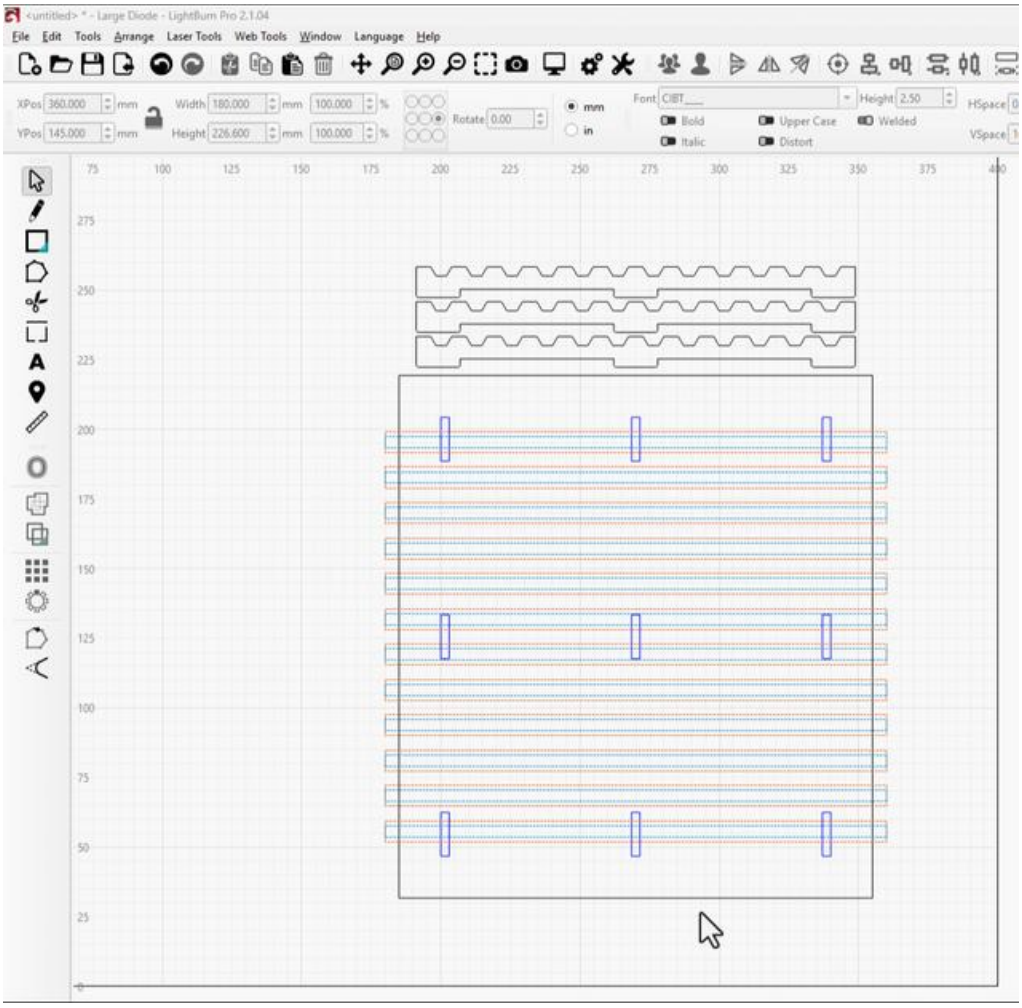


The edges of my laser's bed don't match up with the workspace

For some lasers, there isn't a convenient edge that lines up with the total travel area of the laser. The laser I'm using for this project, for example, has a bed that's significantly larger than the area the laser can travel, and if I put the fixture in a corner, the laser won't be able to reach where I want my pencils engraved. I created a spacer that aligns my material with the corner of my laser's bed to fix this.

2. Place the Template in the Matching Corner in LightBurn

Move the template into the same corner of the workspace you placed the fixture in. You can use Snap to Grid or the Numeric Edits Toolbar to move it.

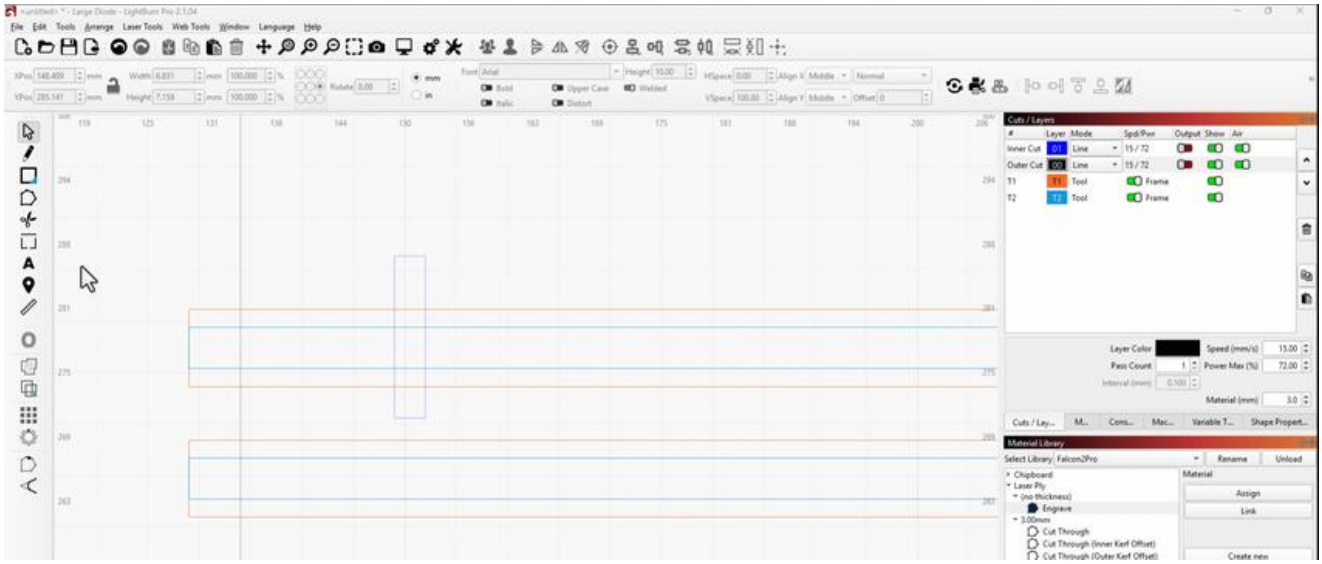


4. Add Your Design

I'll be using text in my example, but you can use anything you want - just keep in mind that really complex designs won't necessarily show up well when they're this small.

> **Have you tried single line fonts?**
>
> If you haven't already tried using single line fonts, this would be a great chance to try them! When you use a single line font with Line Mode, you can get crisper results very quickly. If you'd rather stick with standard fonts, that's fine too! You can use Fill Mode to fill in your text.
>
> To use single line fonts, you'll need to download some SHX fonts and set your SHX font directory.
>
> **Looking for single line fonts? Find them in the "Zip Files" section**

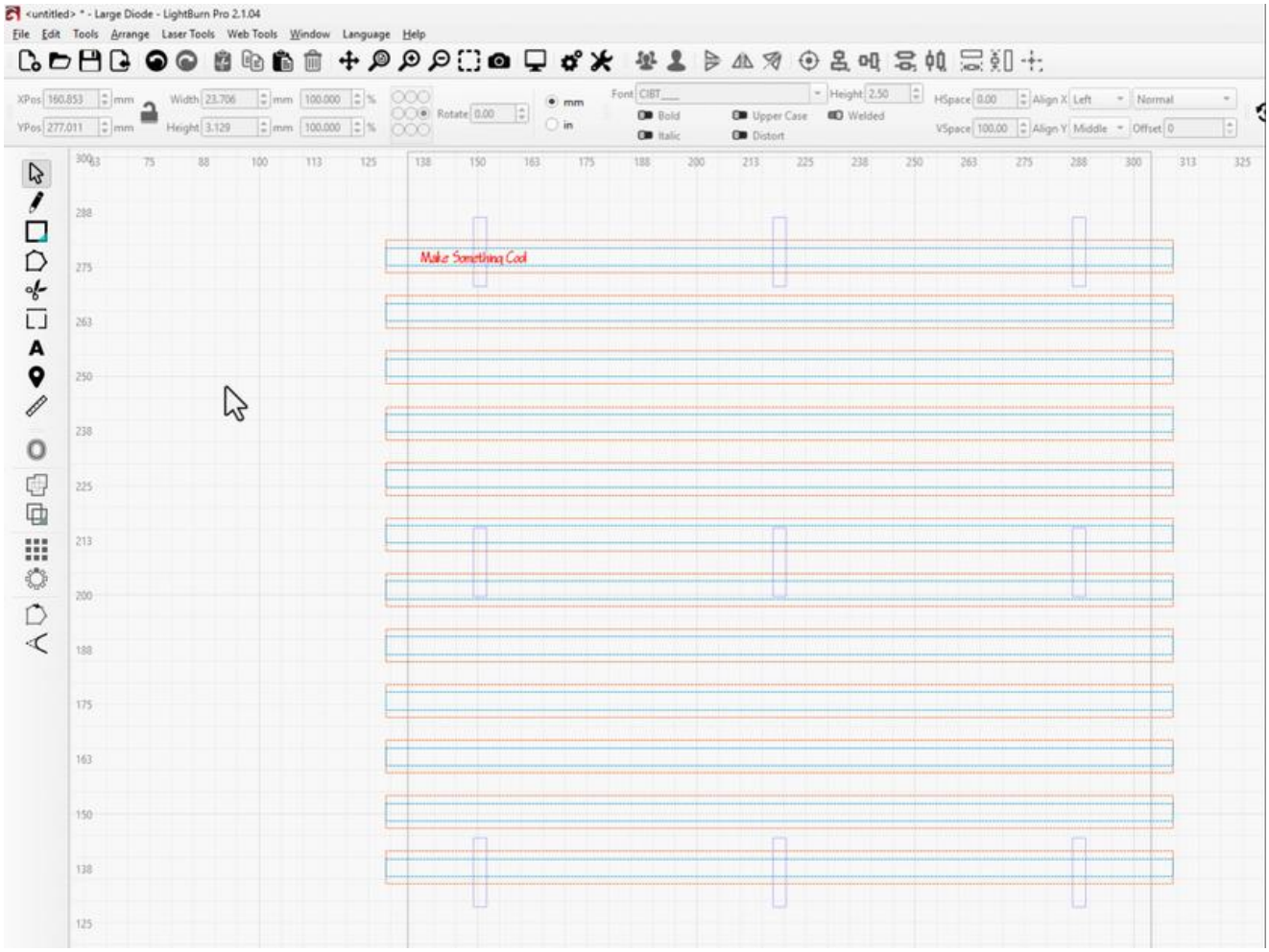
1. Open the Text Tool by clicking on it in the sidebar or using `Ctrl`+`T` / `Cmd`+`T`. Click to start typing. Choose a font you like, and then set the size of the text with the Height setting. A height of about 2.5mm (a little less than ⅛ inch) is generally good. Write your message, press `Esc` to go back to the **Select** tool, and then move your text to place it in the pencil placeholder wherever you want to engrave your design.



2. With your text selected, move it to a new layer by clicking on an unused layer number in the Color Palette at the bottom of the window. If you're working with our template, Layers 02 and 03 are already set up for engraving. Set the layer/layer mode according to the style of engraving you want:
   - If you're using a single line font, select **Layer 02** or set the layer you chose to Line Mode. Line Mode is mostly used for cutting, but by increasing the speed and reducing the power, you'll get a crisp, clean engraving that's perfect for something this small.
   - If you're using a regular font or other artwork, select **Layer 03** or set the layer you chose to Fill Mode to do a raster engraving.
3. Let's be honest - I don't want to write the text 12 times, and you probably don't either. We could just use Duplicate and move the copies, but let's do something a little cooler with a Grid Array.

> **Using variable text**
>
> If you've got a list of different things you want to engrave on the pencils, such as a class list of names (or a list of some of your favorite BattleBots), you can use Variable Text in Merge/CSV mode to run through a list of names. If you want to give it a try, we have an alternate version of this step here that uses Variable Text.

   1. With your text selected, click the Grid Array button in the sidebar or go to **Arrange → Grid Array**.
   2. Adjust your settings as follows. You can use the mouse wheel to zoom and pan while the Array window is open, if you need to see more of the workspace:
      - Y Rows: 12
      - Y Spacing: 12.7 mm (0.5 inches)
      - Turn on **Distance from Center to Center** as the distance type
      - Enable **Reverse Direction** if needed
   3. Click "OK" to accept settings.



4. If you want to edit the text on individual pencils, you can ungroup the array object (`Ctrl`+`U` / `Cmd`+`U`) and edit individual text objects. You may want to change the **Align X** setting to either **Left** or **Right** so they stay aligned with each other on the pencils.

5. Adjust Cut Settings

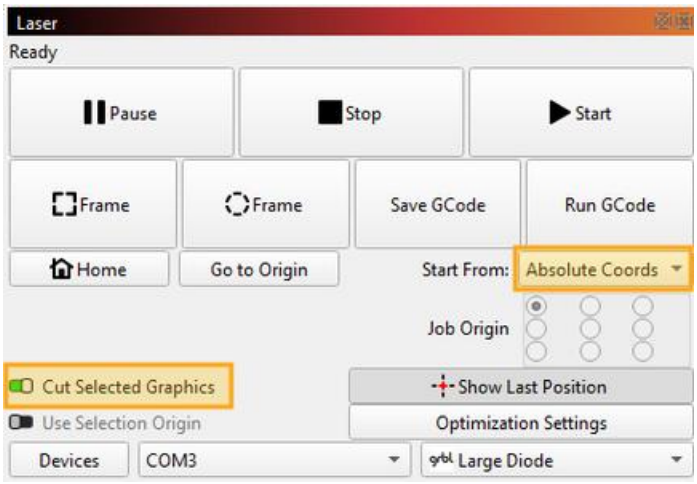
1. Adjust the cut settings for your engraving layer — particularly speed and power — to something suitable for engraving with your laser. Typically, I've done well with settings similar to what I use for engraving on wood. You'll probably want to set a few pencils aside for testing these settings before you do the finished pencils.
2. In the Cuts / Layers Window, you may want to turn off the **Output** toggle for the layers you aren't engraving. They'll display slightly dimmed, and won't run when you send your design to the laser.

6. Get Ready to Engrave

1. Focus your laser to the top surface of the pencils. If you need a larger flat surface to focus to, lay a thin sheet of material such as a business card across the top of the pencils.



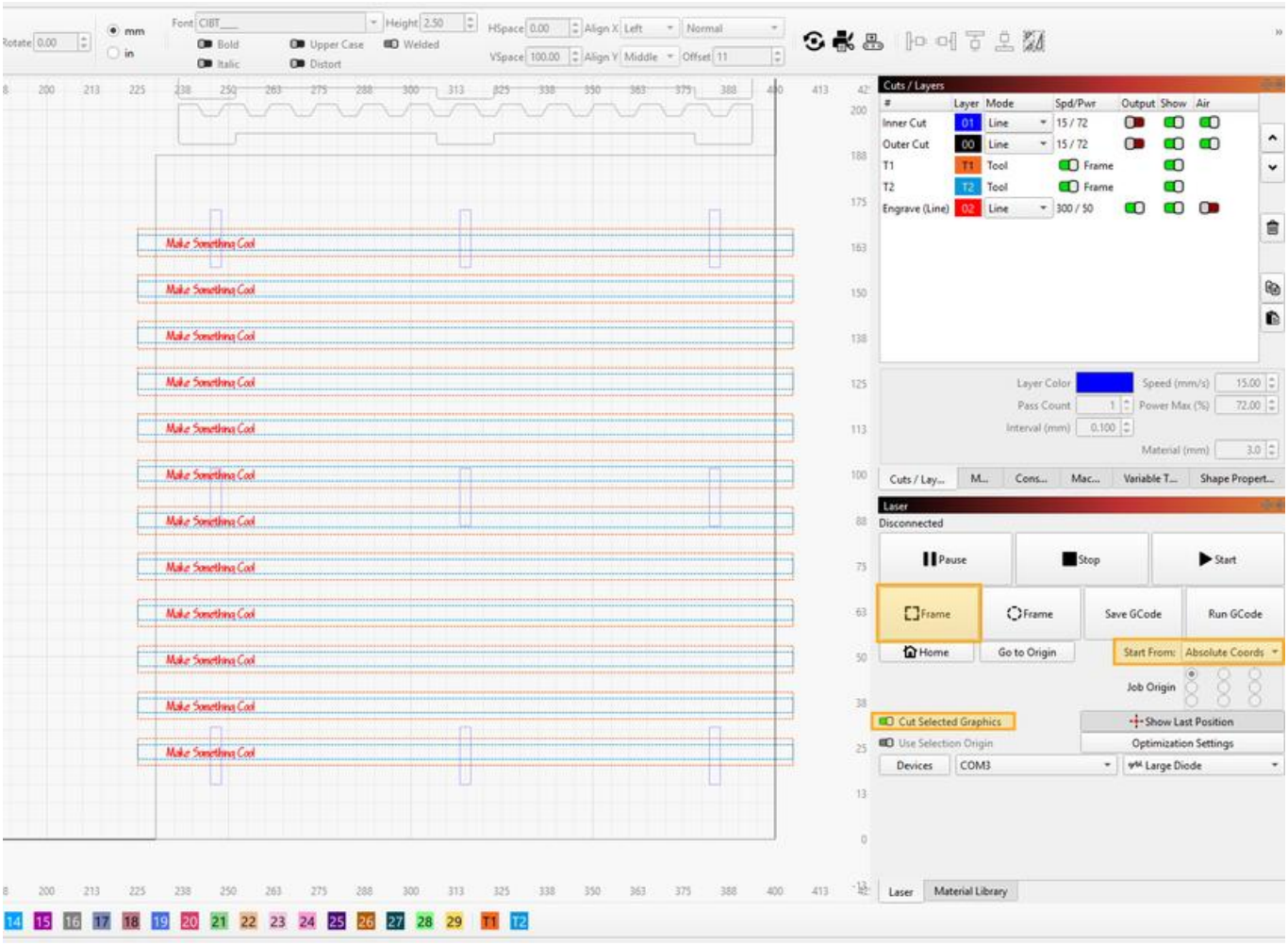
2. Go to the Laser Window to adjust settings.
    - Set the **Start From** mode to **Absolute Coords**. Other Start From modes have their uses, but this is the mode you want if you want to set the engraving location based on where your artwork is placed in LightBurn.
    - Enable Cut Selected Graphics. This isn't strictly necessary, but it does make things a lot easier - only the pencils you select will be engraved. The rest of the directions will assume this setting is enabled.



3. Home your laser if needed.

> **What if my laser can't home?**
>
> It's not ideal, but if your laser doesn't have homing switches, you can move the laser to its origin by hand before turning it on. Framing will be extra important for you to make sure the laser's positioning is right.

4. Select the artwork for the pencils you want to engrave, and click Frame to show where the engraving will be located. If the placement is wrong, you can adjust the position of the pencils, the fixture, or the template to match. When I engraved my pencils, I had to shift the artwork up by about half a millimeter.
5. When you've confirmed the position is correct, run the job! You can make more at any time just by loading up the template, updating the artwork to whatever you want to engrave, and putting the fixture back in the same corner.



Success!

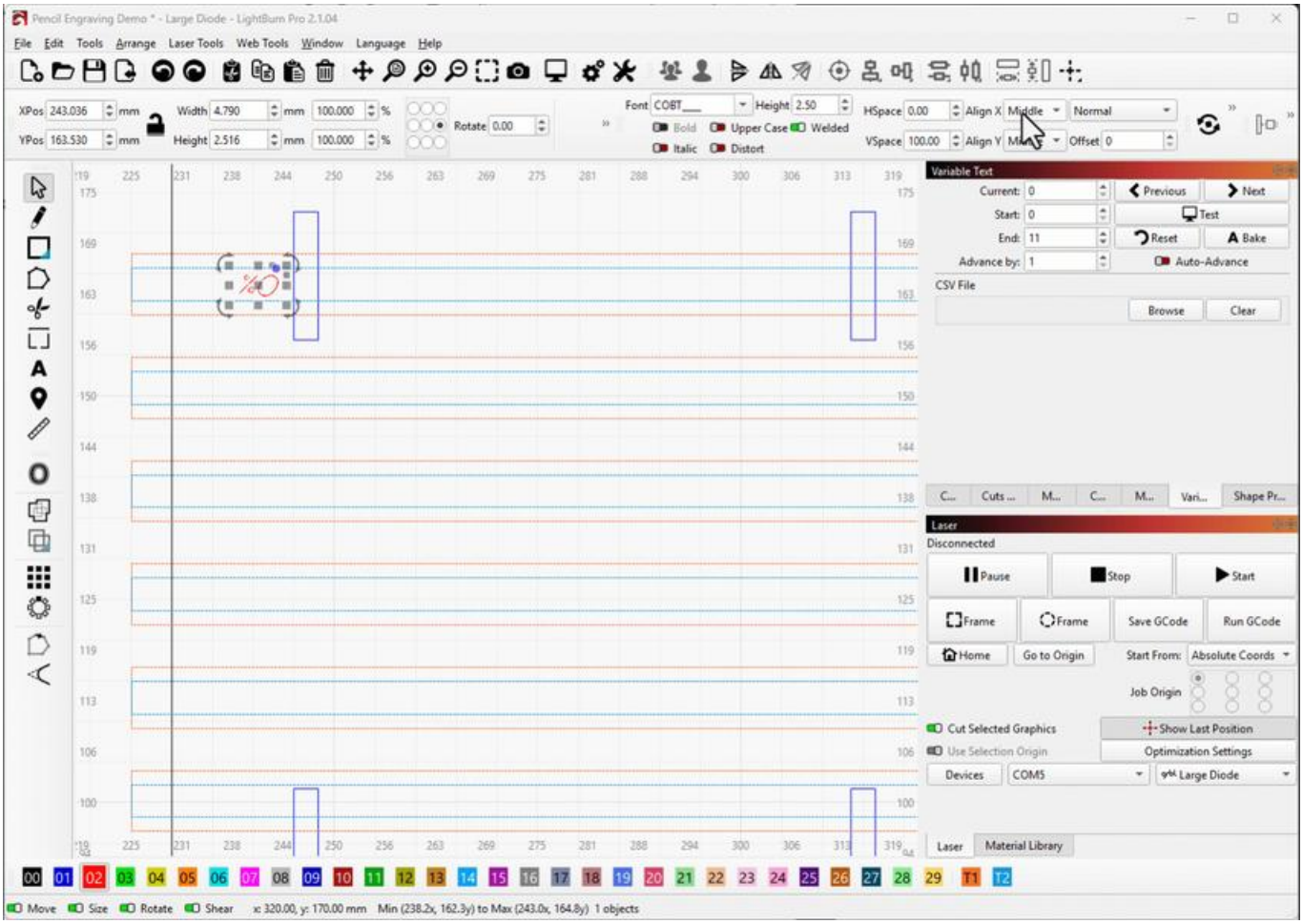
You've got custom engraved pencils!

> **Finishing tip**
>
> Depending on whether the pencils are painted and what paint is used, you may find that the engraving isn't quite as dark or consistently colored as you'd like. The pencils I used for this project didn't engrave as nicely as some of the other pencils I used.
>
> If this happens to you, there's an easy fix! I grabbed a bit of acrylic paint, brushed it over the engraved area, and wiped it off the surface of the pencils. The wet paint wipes off the glossy surface quickly and easily, but stays behind in the engraving.



Using Variable Text

With LightBurn's Variable Text feature, you can type out a list in a separate file and automatically insert the contents into your LightBurn project. That's not all Variable Text can do, but it's the most relevant use for it here.



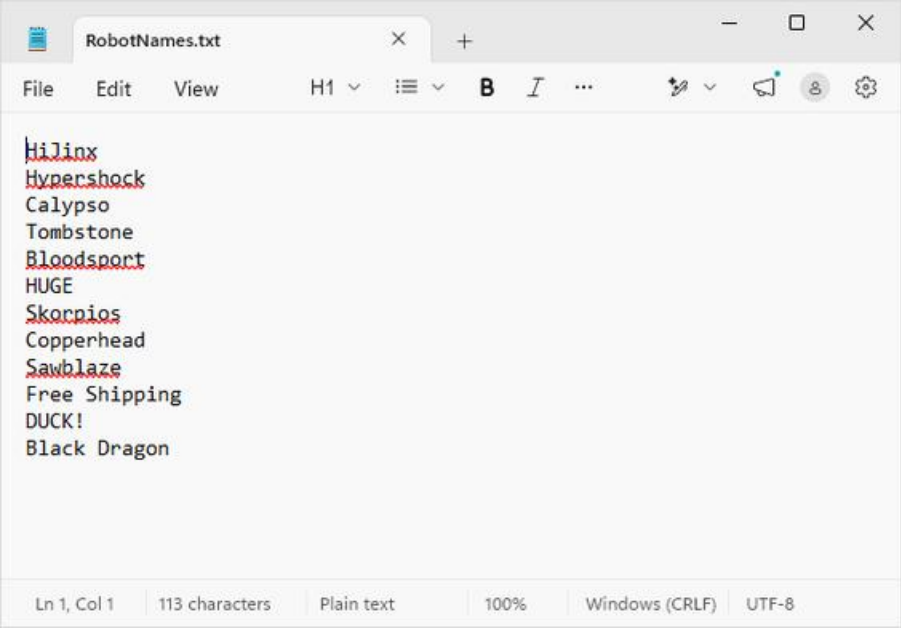
This is going to be pretty similar to making the Grid Array in Step 4: Add Your Design, but with just a few extra steps.

1. Prepare the List

LightBurn's Variable Text tool can load CSV or text files to use with Merge/CSV mode. If that's something you're already comfortable, great! Make yourself a CSV file or a plain text file with the text to insert for each pencil on a separate line and skip ahead to Adjusting the Text Settings.

If I lost you for a moment there, don't worry. Here's the process:

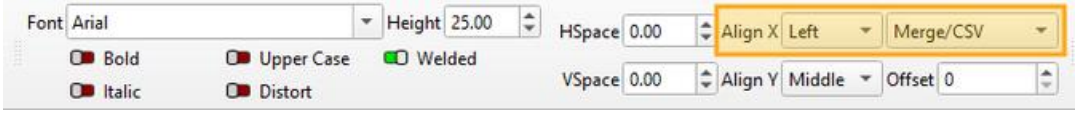
1. Open a text editor. For Windows, you can use **Notepad** and for macOS you can use **TextEdit**.
2. Type in the text you want for the first pencil, press `Enter` / `Enter` to start a new line, and continue until you have your full list.
3. Save the file as a text file (`.txt` extension) somewhere you can find it.



2. Adjust the Text Settings

Now that you've got a list, let's prepare the text in LightBurn.

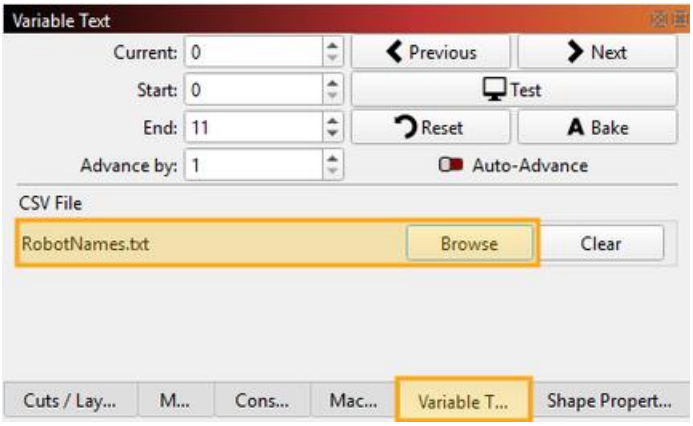
- With your text selected, find the dropdown in the Text Options Toolbar for the text mode and change it to **Merge/CSV**. By default, it's set to **Normal**.
- You'll probably want to change **Align X**. The default is **Middle**, but you can change it to **Left** or **Right** so that as the text changes, it stays aligned at the left or right sides of the text boxes.



3. Load the List

Open the **Variable Text** window. It's not shown by default, so you'll likely have to go to **Window → Variable Text** to open it.

Click the **Browse** button, find the list you made earlier, and click **Open** to load it. You'll see the file name under **CSV File** like in the screenshot below.



4. Variable Text Formatting

Variable Text uses special formatting so LightBurn knows what text you want inserted into your design. The formatting for the **Merge/CSV** mode is pretty simple.

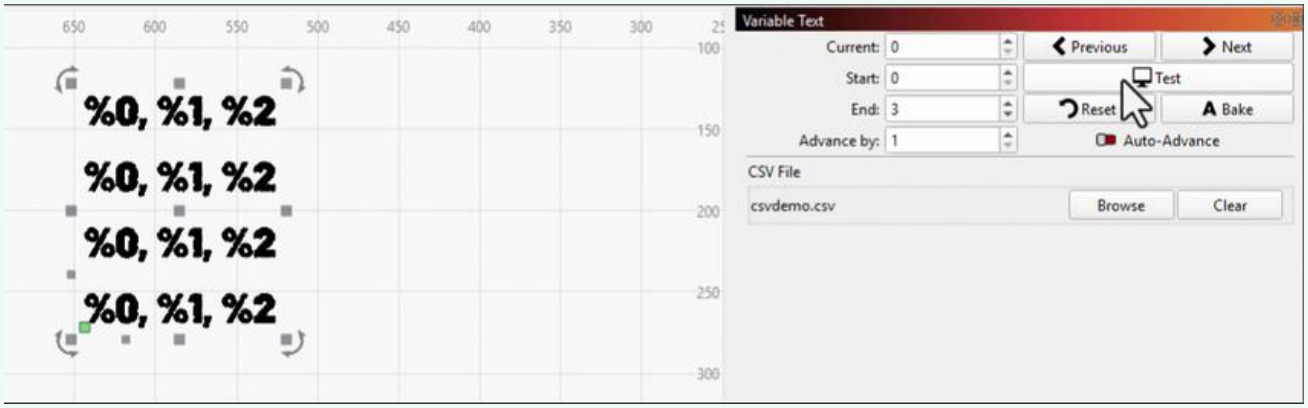
With your text file loaded and the **Variable Text** mode set to **Merge/CSV**, **%0** will automatically be replaced with each list item in order. You can include **%0** as part of a line of text, and only **%0** will be replaced, like in the example below. Clicking the **Test** button in the Variable Text window will temporarily show your text with the Variable Text inserted. You can also see the Variable Text in Preview, or freeze it with the Bake button.

More With Merge/CSV

I've simplified a bit here to keep things easier for people who are new to using Variable Text, but there's a little more to Merge/CSV than just %0. Keep reading if you want to know more.

CSV stands for Comma-Separated Values, and it's a format for creating tables/spreadsheets in plain text. You can put more than one item of text on each line, with commas in between, and insert them using **%0**, **%1**, **%2**, and so on. The first item in the row is always **%0**, not **%1**. You can see an example below, along with the text that was loaded as a CSV file.

```
apple,banana,carrot
aardvark,badger,cat
amphitheater,bridge,causeway
archery,boxing,climbing
```



5. Create the Grid Array

Just like in the version without **Variable Text**, we'll use a Grid Array to repeat the text — but now each item in your grid array will show different text when you engrave it!

1. With your text selected, click the Grid Array button in the sidebar or go to **Arrange → Grid Array**.
2. Adjust your settings as follows. You can use the mouse wheel to zoom and pan while the Array window is open, if you need to see more of the workspace:
   - Y Rows: 12
   - Y Spacing: 12.7 mm (0.5 inches)

- Turn on **Distance from Center to Center** as the distance type
- Enable **Reverse Direction** if needed
- Make sure **Auto-Increment Variable Text** is enabled and set to **1**.

3. Click "OK" to accept settings.

6. Test the Variable Text

Click the **Test** button in the **Variable Text** window to make sure the text displays as expected.



If it doesn't display correctly, check the following:

- Make sure your text mode is set to **Merge/CSV**. See directions.
- Make sure you loaded your text file. See directions.
- Make sure you have **Auto-Increment Variable Text** enabled. See Directions

If it's still not working, email support or visit our forum.

7. Continue With the Rest of the Project

You can continue on with Adjusting Cut Settings.

Other Things to Try

- Try Using Variable Text to create a set of pencils from a list of names or messages.
- Our template is designed for pencils, but there's no reason you couldn't design your own fixture for other objects you engrave repeatedly. For something flat like a dogtag, you can engrave a pocket in a sheet of material without bothering with the interlocking pieces to hold the pencils like we have here.



Laser Etched Clock

Clocks and the ability to track time are among the most important inventions in the history of mankind. Not only do they help us keep on schedule, for nearly 400 years they have also been crucial to navigation, from ship chronometers to GPS. While we have plenty of digital clocks around us nowadays, it doesn't mean that we can't still appreciate the aesthetics of analog clocks for decoration.

In this guide, we will explain how to create a good looking analog clock out of easy to find materials using LightBurn and even a low power laser.

What You'll Learn

This project will teach you how to:

- Use Tool layers as guidelines
- Precisely align graphics using the Alignment Tools
- Transform specially formatted characters into clock numbers using Variable Text
- Evenly space the clock numbers using Circular Array
- Position output to your laser using Current Position as your **Start From** mode

Materials

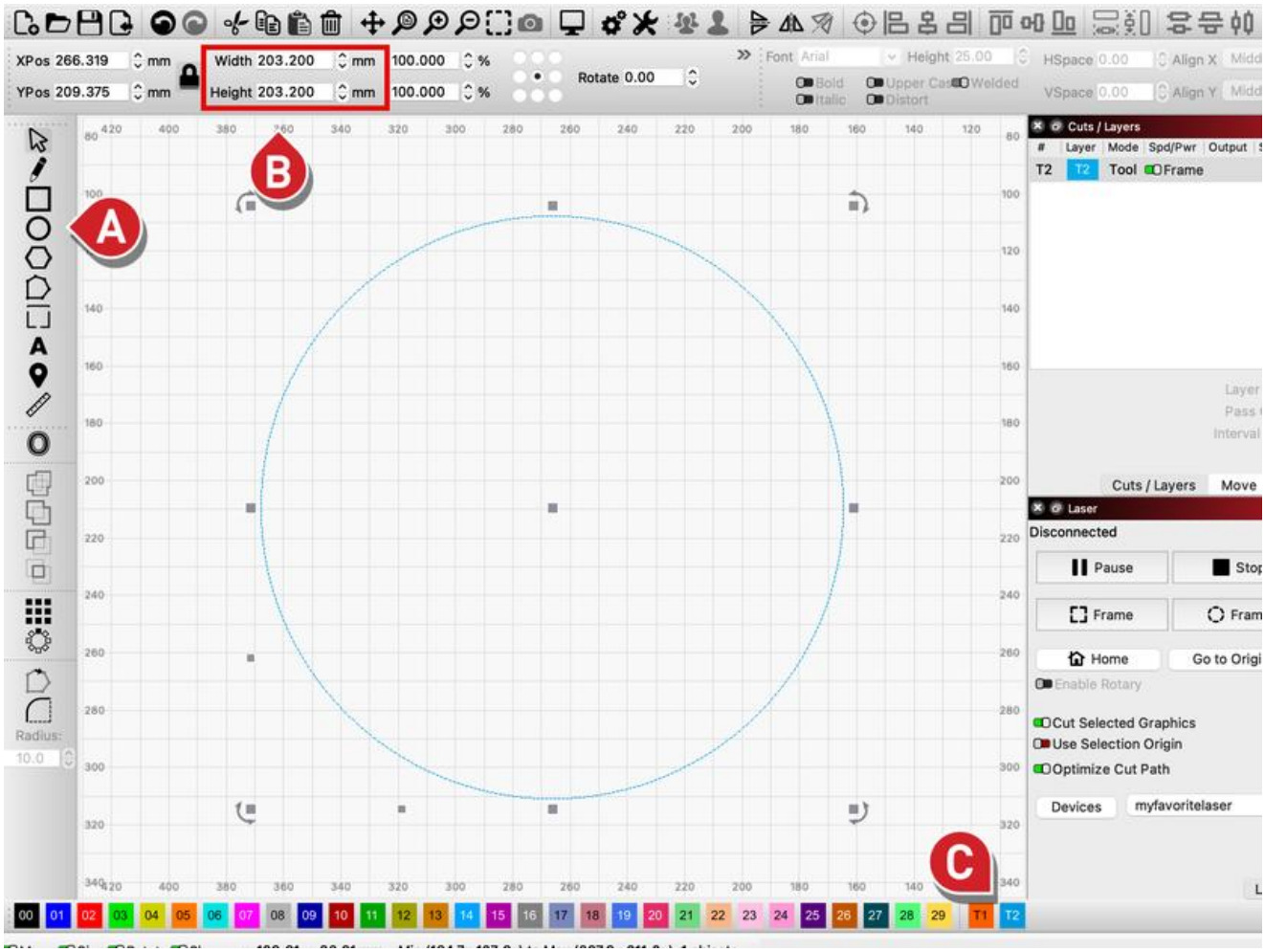
- A chosen material to engrave a clock face onto.
  - We aren't going to cut this clock out on our laser, instead we are going to find a good item to modify and turn into our clock. Anything that can fit in your laser's workspace will do.
  - We selected a set of nice looking mini cutting boards that worked perfectly, but you can select a material of any size or shape that you think will make a nice clock — it doesn't have to be round.
- A battery powered analog clock module.
  - These are relatively inexpensive and are available at craft stores or can be ordered online.
- A power drill and drill bit bit that is the same size as the post of your clock module.

Steps

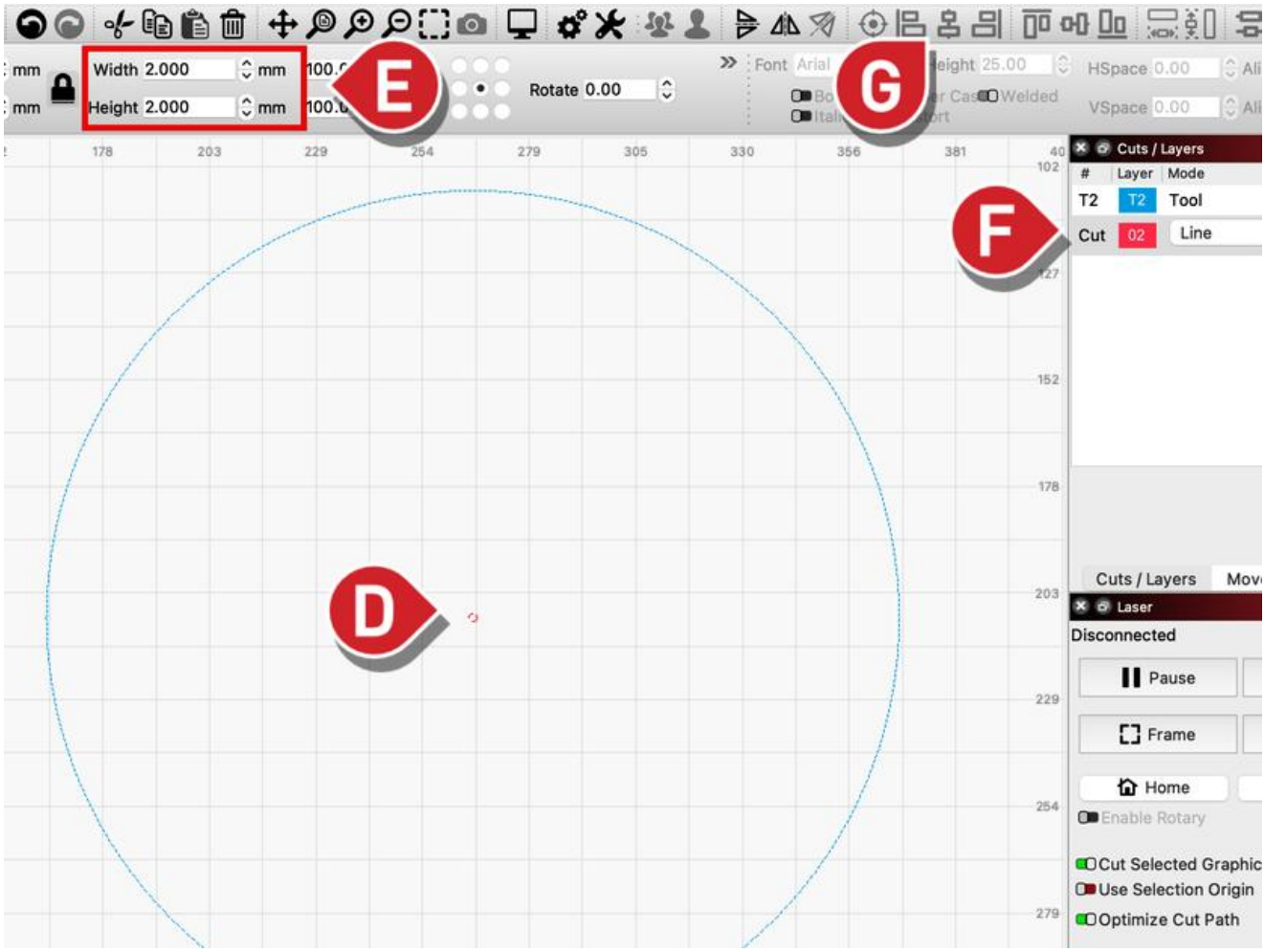
1. Outline the Clock Face

Before opening LightBurn, measure the width of the object you have chosen to use for your clock. Be sure to measure at the space where you want your design to be centered so you get the proper width.

1. In LightBurn, use the Ellipse tool to create a circle. Holding the Shift key while dragging will make it a perfect circle.
2. In the Numeric Edits Toolbar, set its **Width** and **Height** to match the material you selected to make your clock out of. If you made a perfect circle in Step A, you only need to adjust one dimension, as long as the lock icon in the toolbar is locked.
3. Set it to a Tool layer. Tool layers are never output to your laser, so this will just be used for visual reference and alignment.



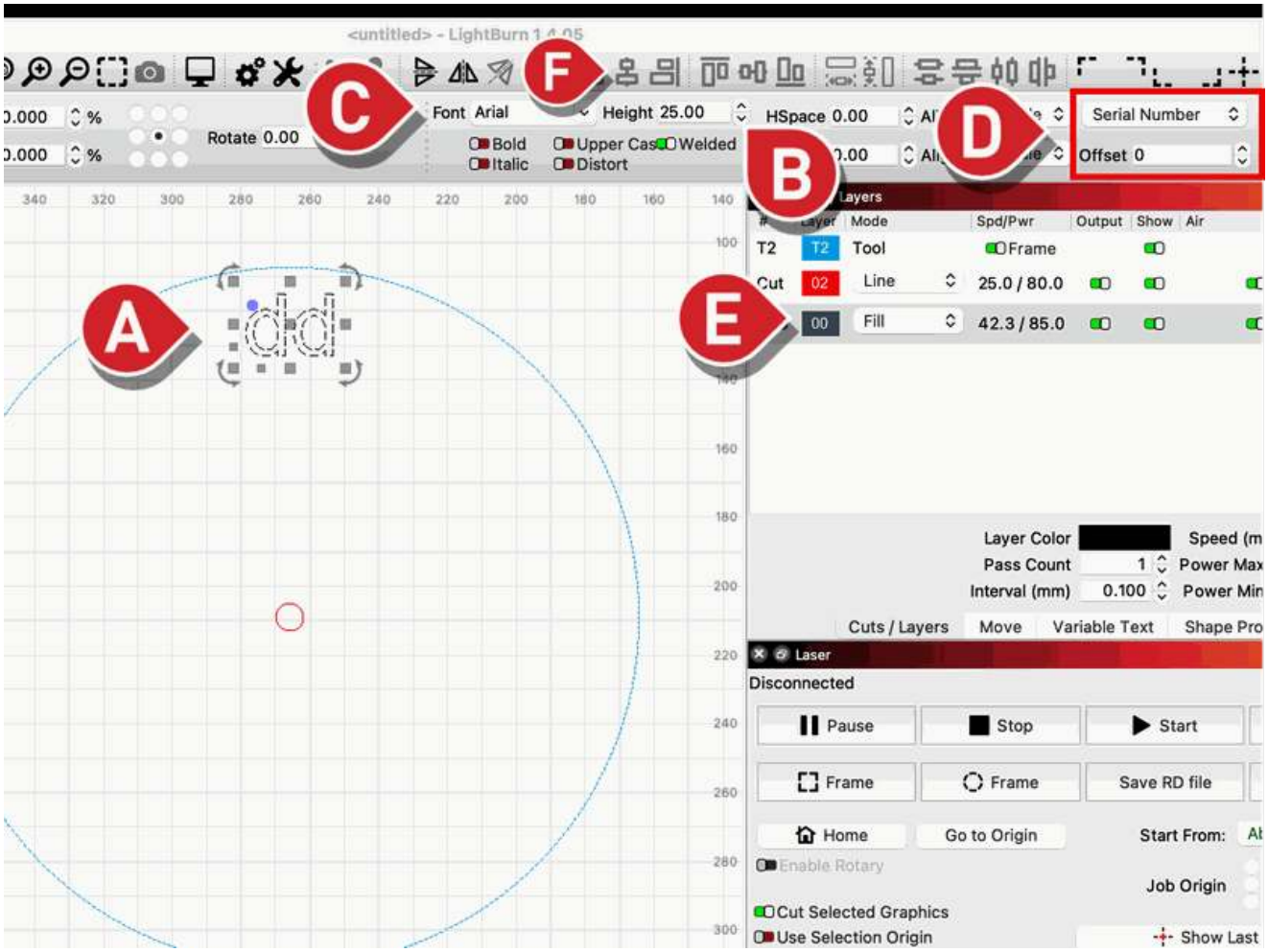
4. Create another circle.
5. Set its **Width** and **Height** to 2mm or .08". This small circle will make it easy for you to align the point of your drill bit later.
6. Set it to a Line layer.
7. Select both circles and use the Align Centers tool to make sure the small circle is perfectly centered in the large circle.



2. Set Up the First Clock Number

Now we'll position and size the first clock number for our clock face, while using Variable Text formatting that will allow us to quickly create the rest of the numbers.

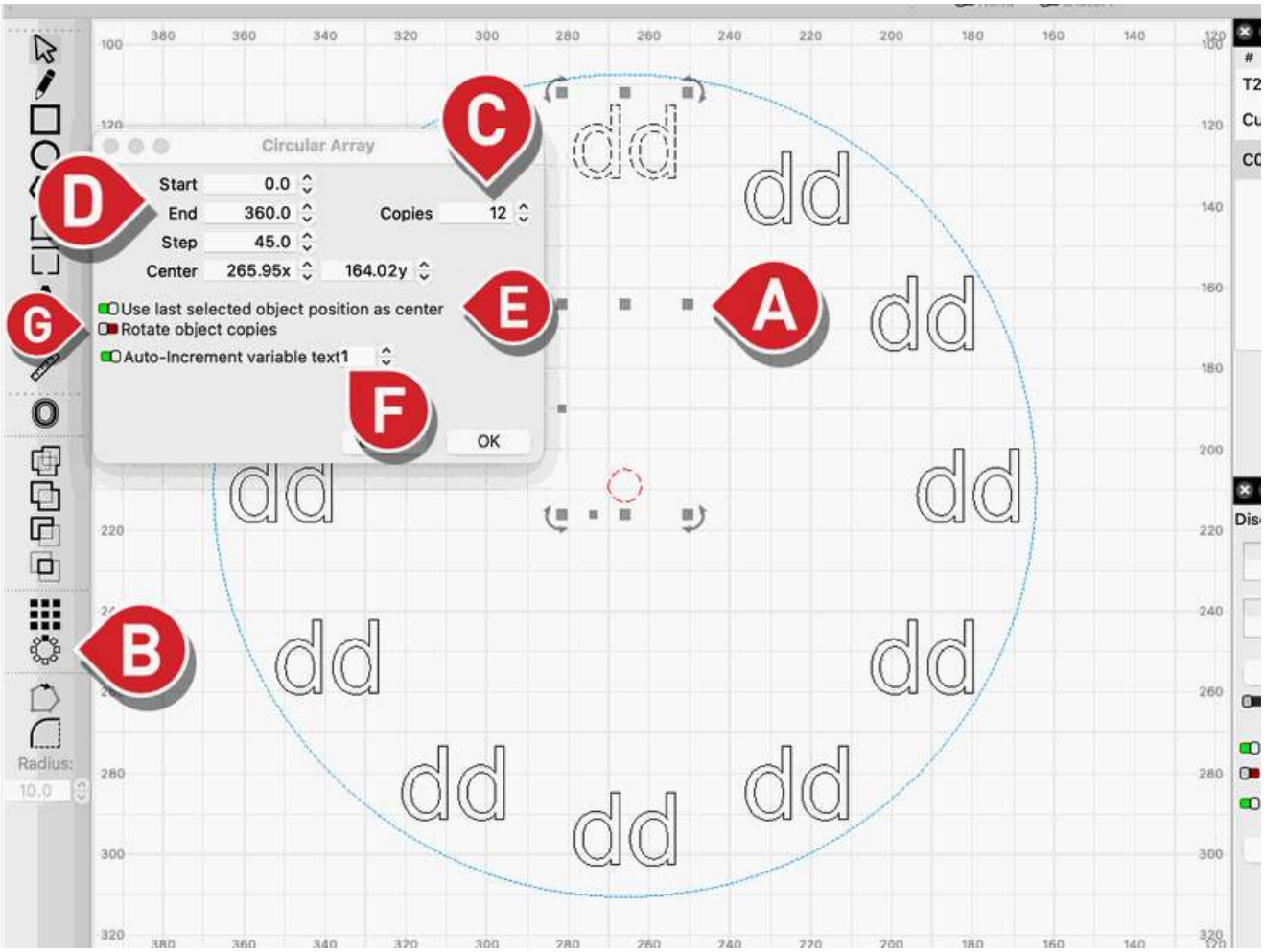
1. Use the Create Text tool and enter the letters "dd" inside the larger circle. Place it near the top center, and set in from the outer circle according to your visual preference.
2. In the Text Options Toolbar, set the **Height** of the letters. They can be as large or small as you like, as long as you're able to fit 12 numbers around the clock face.
3. Choose whichever **Font** you'd like to use.
4. Set the text type to Serial Number and make sure the **Offset** is set to 0. With this formatting, the "dd" text will be swapped for a number when we Preview or send the job to the laser.
5. Set the text to a Fill layer.
6. Select the text, then both circles, and use Align Vertical Centers to align the text's center with the circles'.



3. Array the Variable Text Around the Clock Face

Using the Circular Array tool, we'll create the rest of the numbers for the clock face.

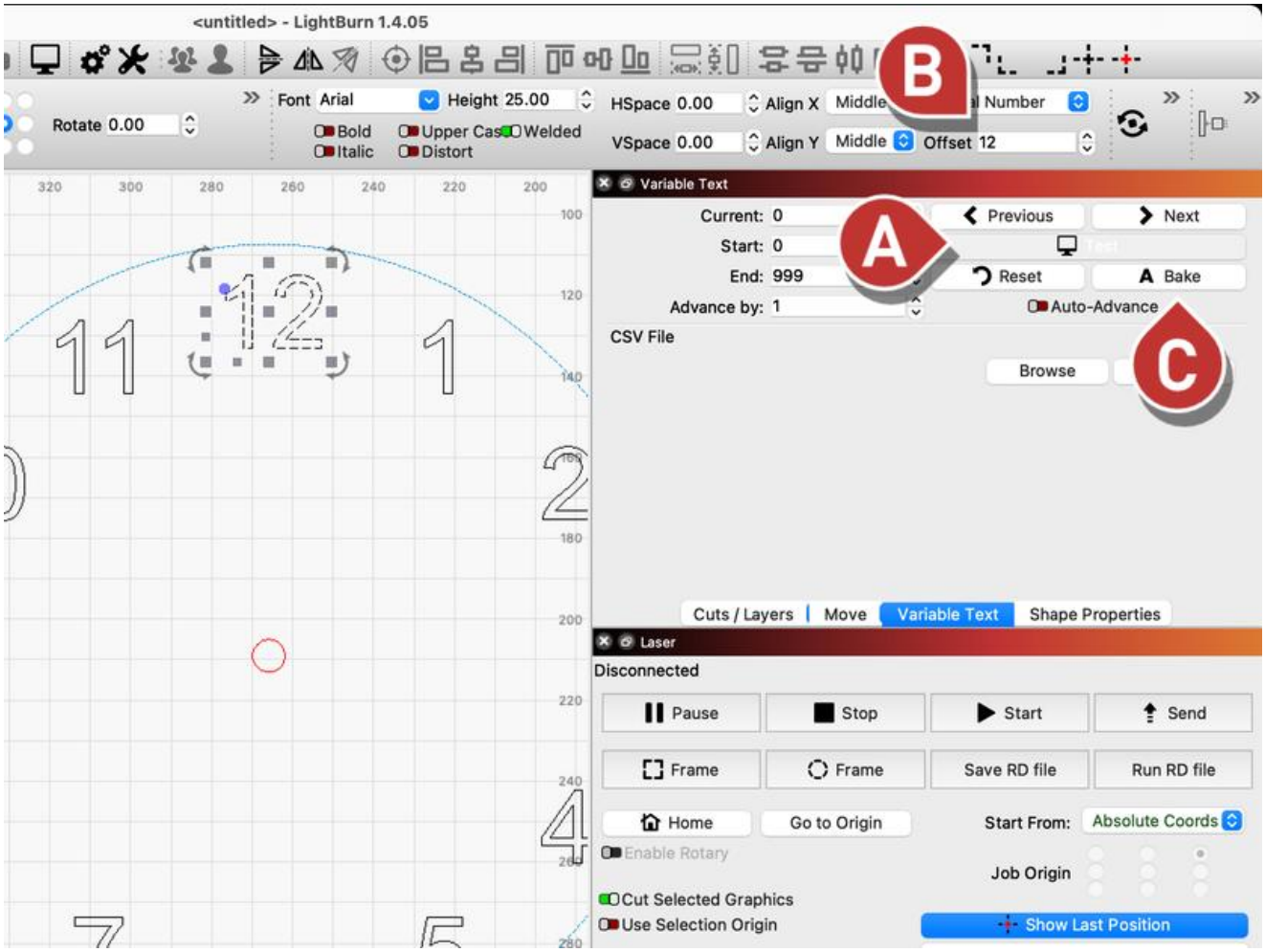
1. Select the text and then either of the circles.
2. Click the Circular Array Tool.
3. In the **Circular Array** window, set **Copies** to **12**.
4. Set the **Start** to **0**, and **End** to **360**.
5. Enable **Use last selected object position as center**.
6. Make sure **Auto-increment variable text** is enabled and set to **1**.
7. Disable **Rotate Object Copies** to keep the numbers rightside up all the way around the clock, or enable it to have them turn upside down.
8. Click **OK** to make the array.



4. Test and Adjust the Variable Text

To finish off the clock face design, we need to make one small tweak to the Variable Text.

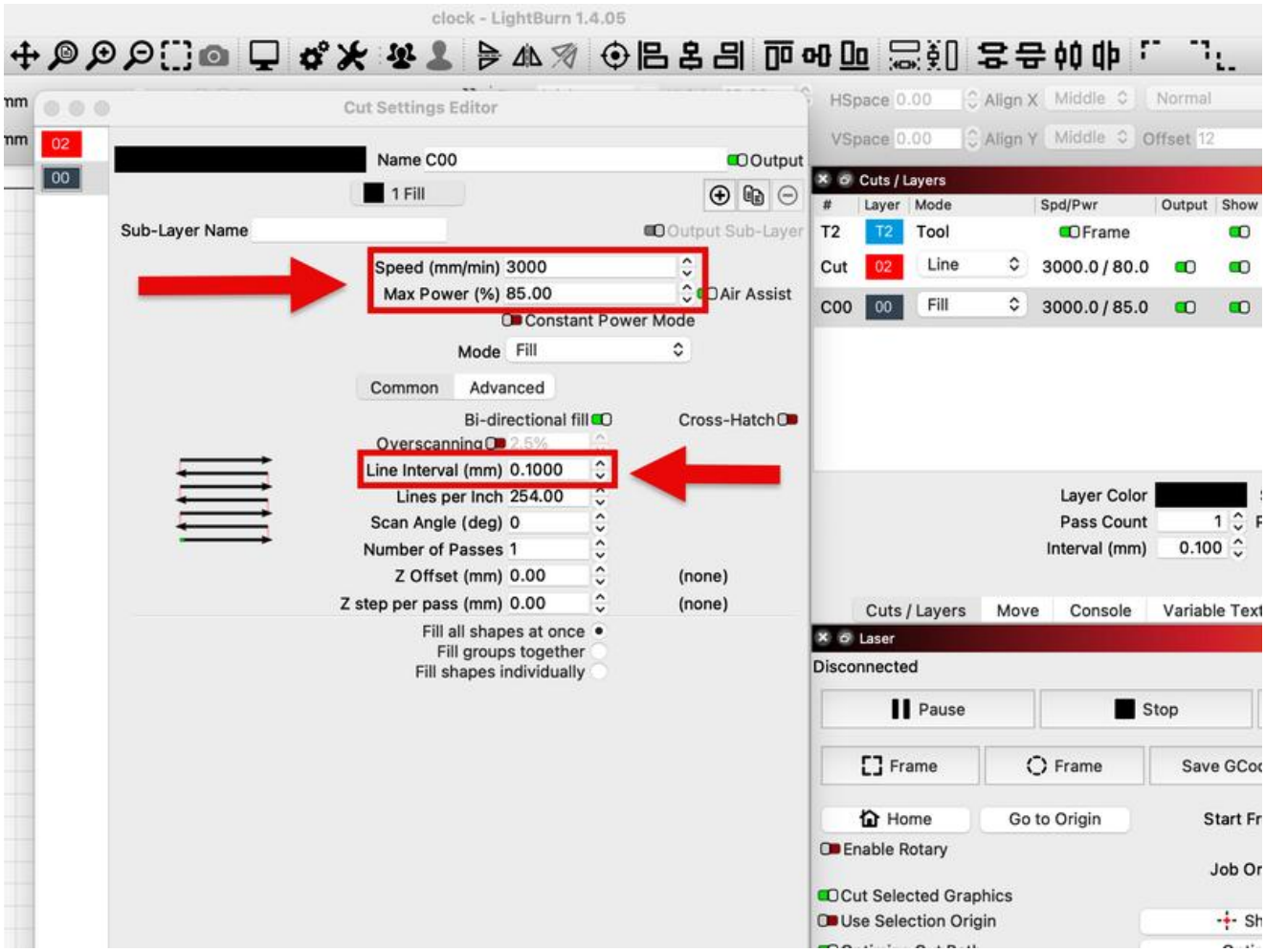
1. In the **Variable Text** window, click **Test**. You should see the numbers 0-11, starting from the top center and advancing clockwise around the circle.
2. Of course, we don't want 0 on our clock, so select the top center text, head up to the **Text Options Toolbar**, and adjust its **Offset** to **12**.
3. Click Test again to make sure it now reads 12. Once everything is correct, you can click **Bake** to permanently transform the Variable Text into numbers in your workspace. If you don't click Bake, the Variable Text will still output as numbers when you Preview or Start the job.



5. Assign Layer Settings

Before running the job, we need to apply appropriate settings to our **Layers**.

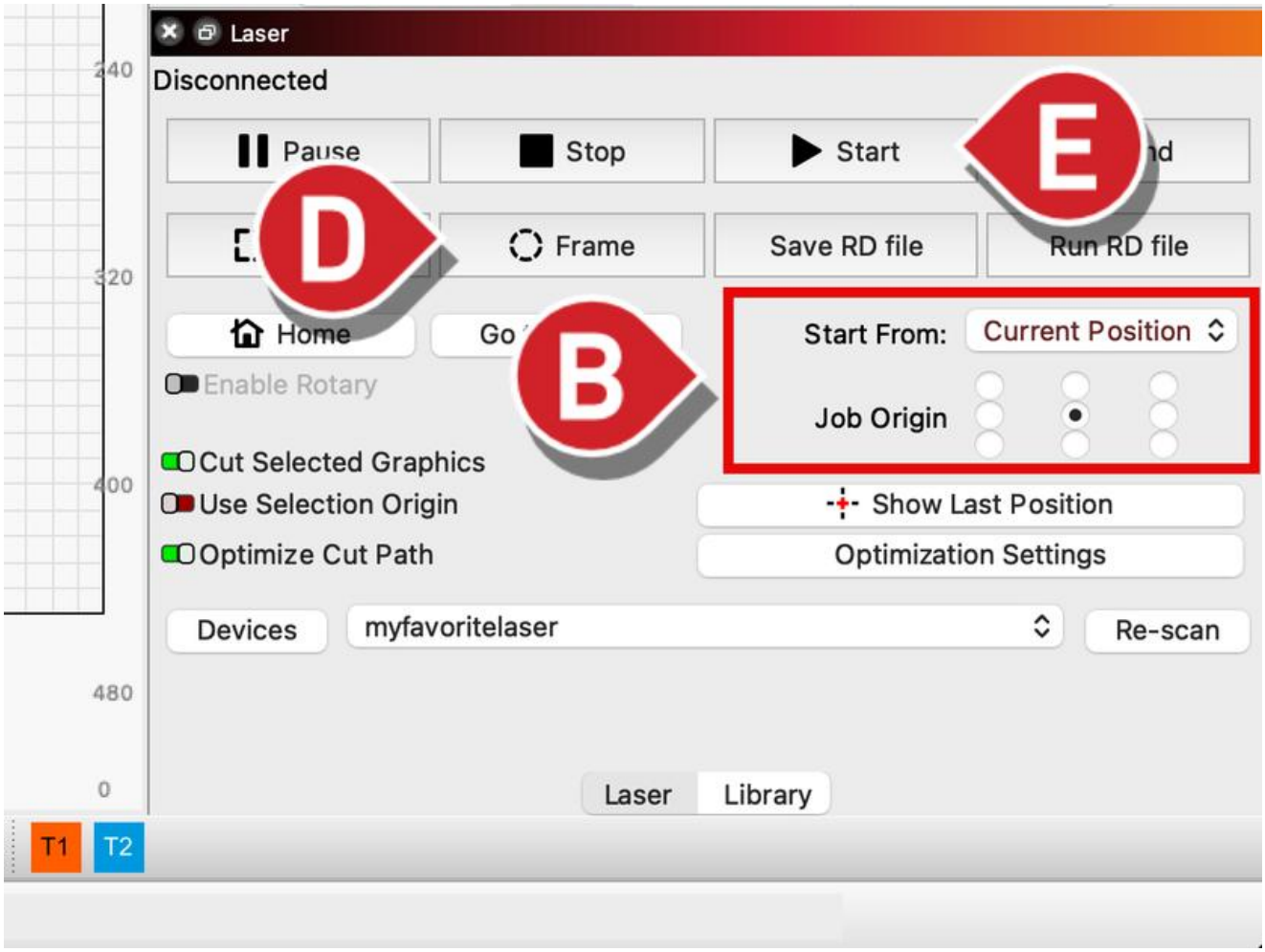
1. Set the Fill layer to a speed, power, and line interval that produces a good etch on your material. If you're unsure what settings to use, you can run a Material Test on some spare material to find optimal settings.
2. Set the Line layer to an appropriate power to etch the material just enough that you can use the mark as a guideline for drilling a hole.



6. Position the Job

Next, we'll position our material, and select an appropriate **Start From** mode in LightBurn.

1. Jog the laser head to around the middle of your machine using the arrows in the Move window.
2. In the Laser window, set the **Start From** mode to Current Position and the **Job Origin** to the center position.
3. Place your material centered under your laser's nozzle.
4. Run the Rubber Band Framing routine to make sure your job is properly aligned and centered.
5. Start the job!



7. Drilling

Now that we've engraved our material, we still need a hole for the clock module to come through.

1. Grab a hand drill and a drill bit that is the same size as the post of your clock module. A brad-point drill bit is best for this as it is easy to align and will not walk as you attempt to drill your piece.
2. Put your piece on top of a spare piece of wood so you don't drill into your table when you come through your piece.
3. Now line everything up, making sure to hold your drill perfectly vertical, and drill your hole.



8. Mount the Clock Module

To finish the project, we just need to mount the clock module.

1. Place the clock module's post through the hole and attach any nuts that came with it to keep it mounted on your clock face.
2. Now, following the instructions that came with your clock module, add the hands, and your clock is complete.
3. Add a battery, set the time, and enjoy your new clock for years to come!



After you've made your first clock, consider the ways you can modify the process and make another. You could embellish the face by etching an Image or pattern onto it, or cut out custom-designed hands for your clock module. The possibilities are endless — so long as you've got the time!



Purrfect Present: Pet Puzzle Postcards

Birthdays, holidays, illnesses, and engagements - no matter the occasion, people love seeing their pets! And most of us don't mind a little puzzle either...

This quick and personalized gift is the perfect project for a beginner. Design it, cut it, and send it to your loved one. Imagine putting together a bunch of random puzzle pieces and slowly realizing it's your very own furry friend!

What You'll Learn

- How to use a free puzzle generator and import it into LightBurn.
- How to add a border to your puzzle with the Offset tool.
- How to create fancy custom-shaped pieces with the Draw and Node Edit tools.
- How to add your photo and crop it to the size/shape of the puzzle with the Image Masking tool.
- How to set up your job with optimal Cut Settings.

We'll also give you a bunch of tips for how you can take what you've learned and customize it further to suit your own goals!

Materials

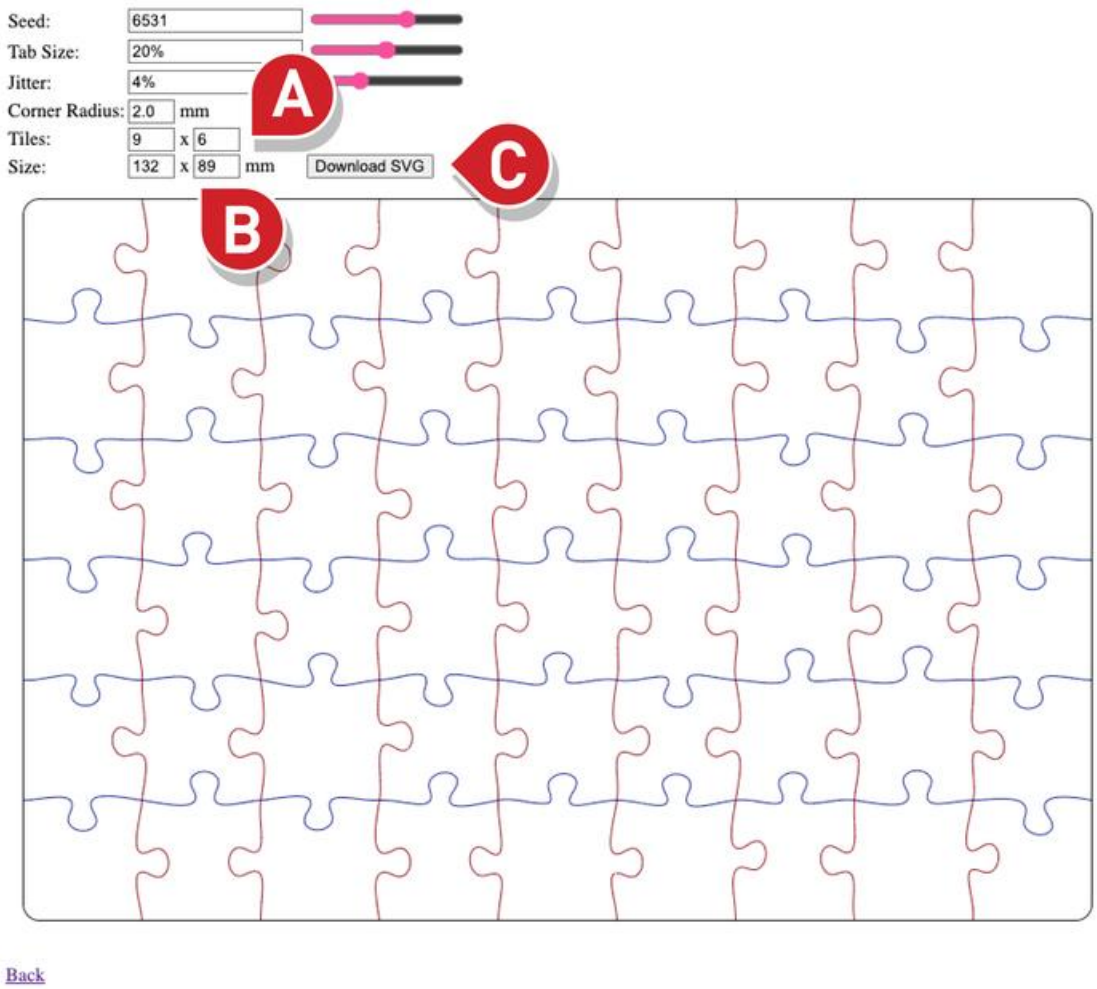
- Your chosen **material**. Roughly 1 - 2mm (≈ ⅛ - ¼ in) thick + corresponding cut and engrave settings: laser plywood is nice!
  - If you don't know your settings for this material yet, you can read more about the image (engrave) and line (cut) settings in our Cut Settings Documentation. You might also like to use LightBurn's Materials Test Generator to automate the settings-finding process.
- **An image** you wish to engrave.
  - You can raid the social media of the pet-parent for some good ones!
- **Dimensions** of your card. If you are sending it in the mail, this will be a bit smaller than your chosen envelope.
  - I'm making my finished size ISO A6, (105 x 148.5mm) which will fit into an ISO C6-style envelope.
- I'm going to create a border for my puzzle of 8mm so the puzzle within will be 89 x 132.5mm

Steps

1. Create The Pieces

Use a free online Jigsaw Puzzle Generator, such as this one by Manual Kasten (Draradech), to create the puzzle pieces.

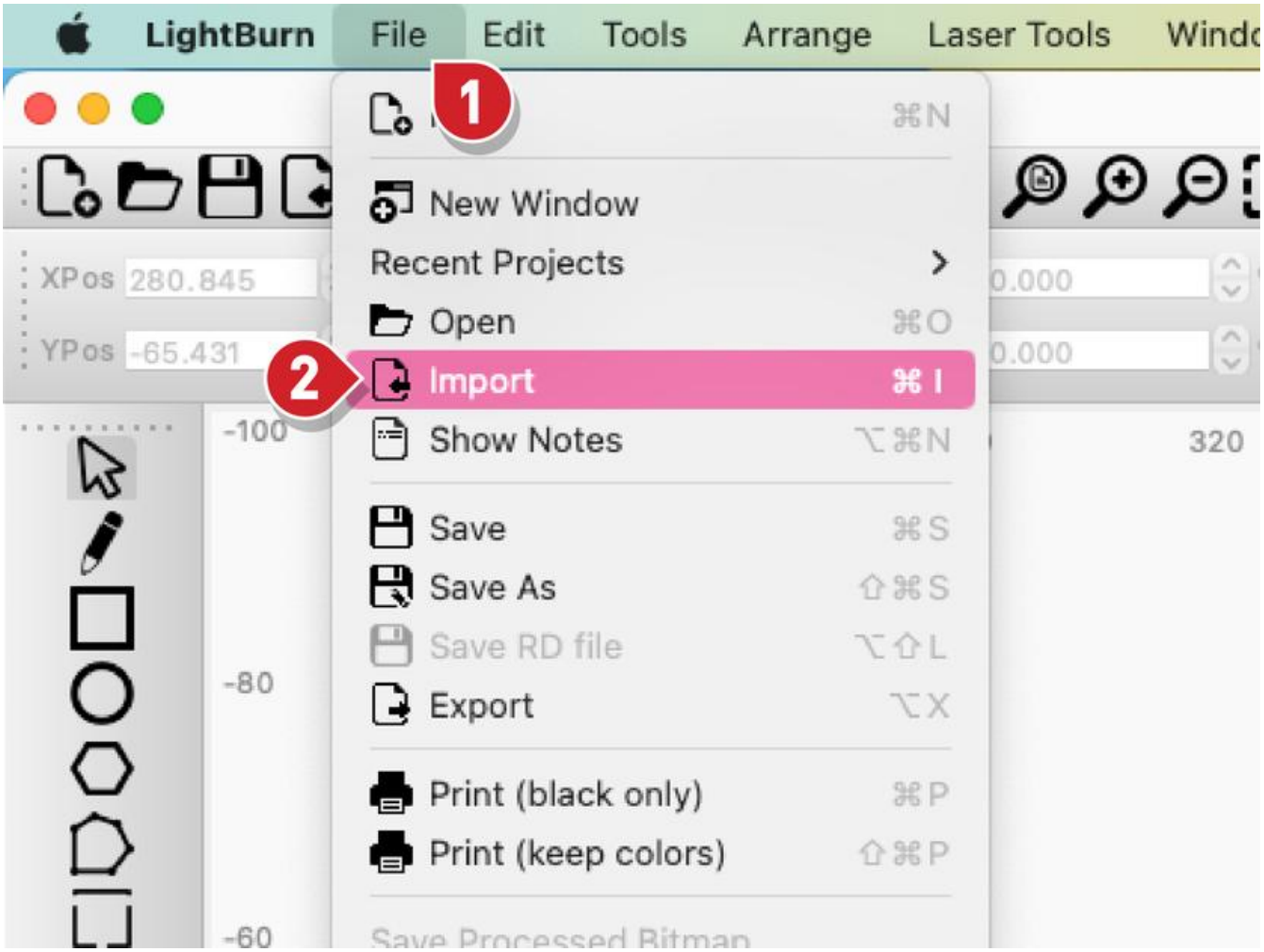
1. Set the **Tiles** to the number of puzzle pieces you want across and down (I used 9 x 6).
2. Set the **Size** to the width and height of your puzzle (I used 132 x 89mm)
3. Click **Download SVG** to save the vector file to your computer.

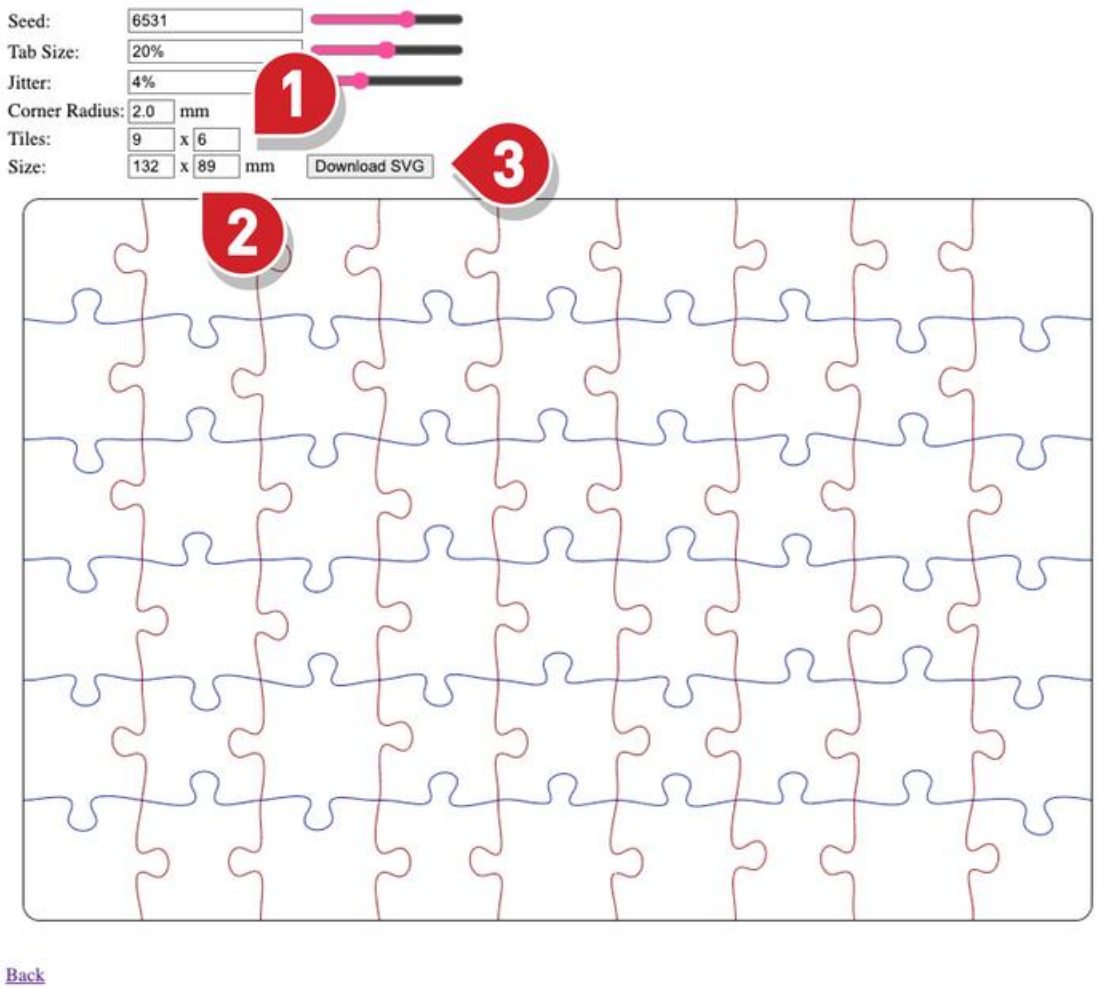


Go To The Puzzle Generator Website

2. Import The Jigsaw Into LightBurn

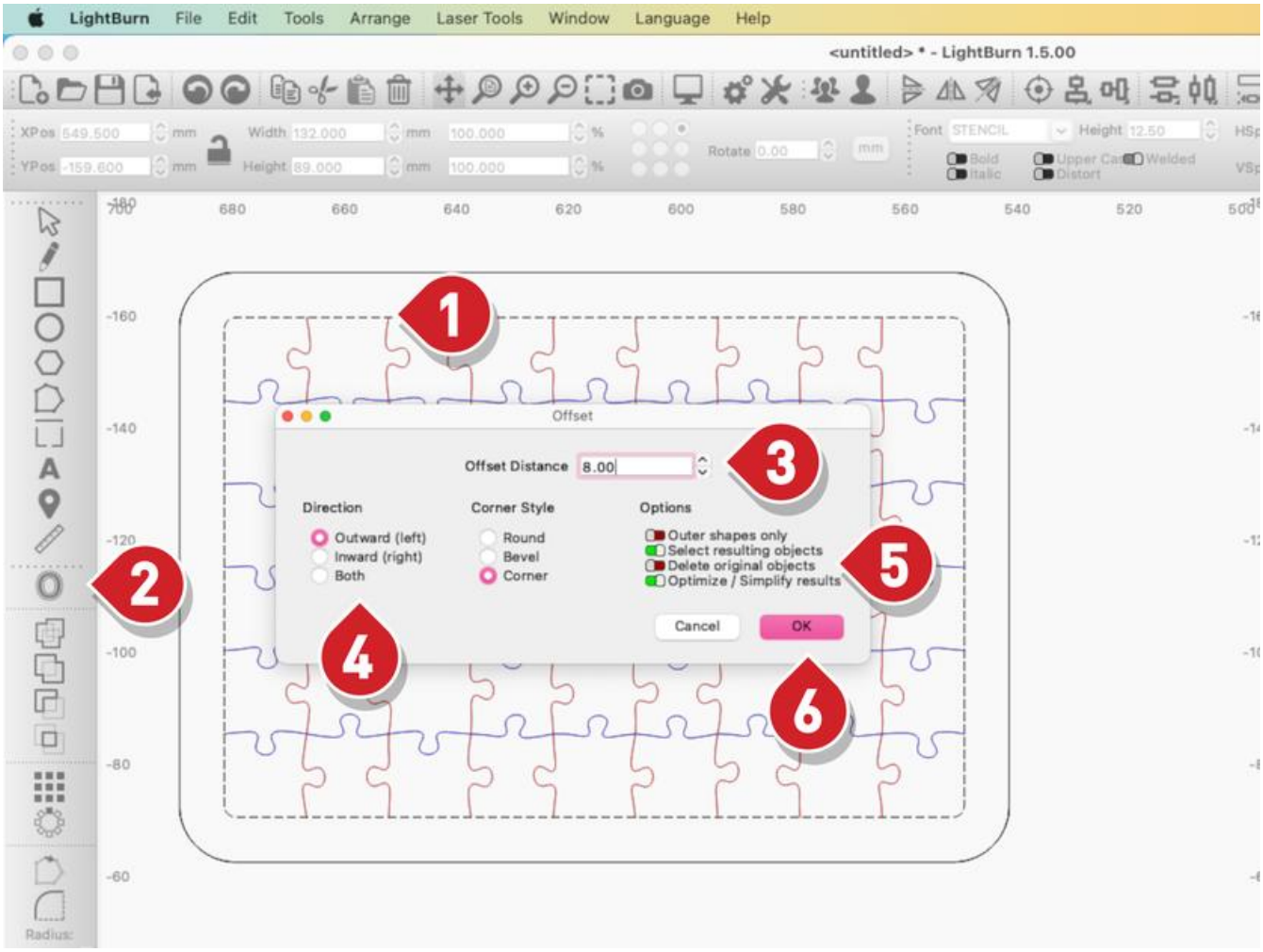
1. Open LightBurn and click **File** → **Import** (or `Ctrl`+`I` / `Cmd`+`I` on your keyboard).
2. Navigate to your jigsaw file, and click **Open**.


LightBurn
File
Edit
Tools
Arrange
Laser Tools
1
⌘N
New Window
Recent Projects
Open ⌘O
2
Import ⌘I
Show Notes ⌥⌘N
Save ⌘S
Save As ⇧⌘S
Save RD file ⌥⇧L
Export ⌥X
Print (black only) ⌘P
Print (keep colors) ⇧⌘P
XPos 280.845
YPos -65.431
-100
-80
-60
320



3. Create A Border Using Offset

1. Click the black outline around the puzzle to select it.
2. Click the Offset Tool from the toolbar on the left.
3. Change the **Offset Distance** to the thickness of your border (in my case, 8mm).
4. Ensure the direction of the offset is set to **Outward**
5. Ensure **Delete original objects** is off.
6. Click **Ok**.

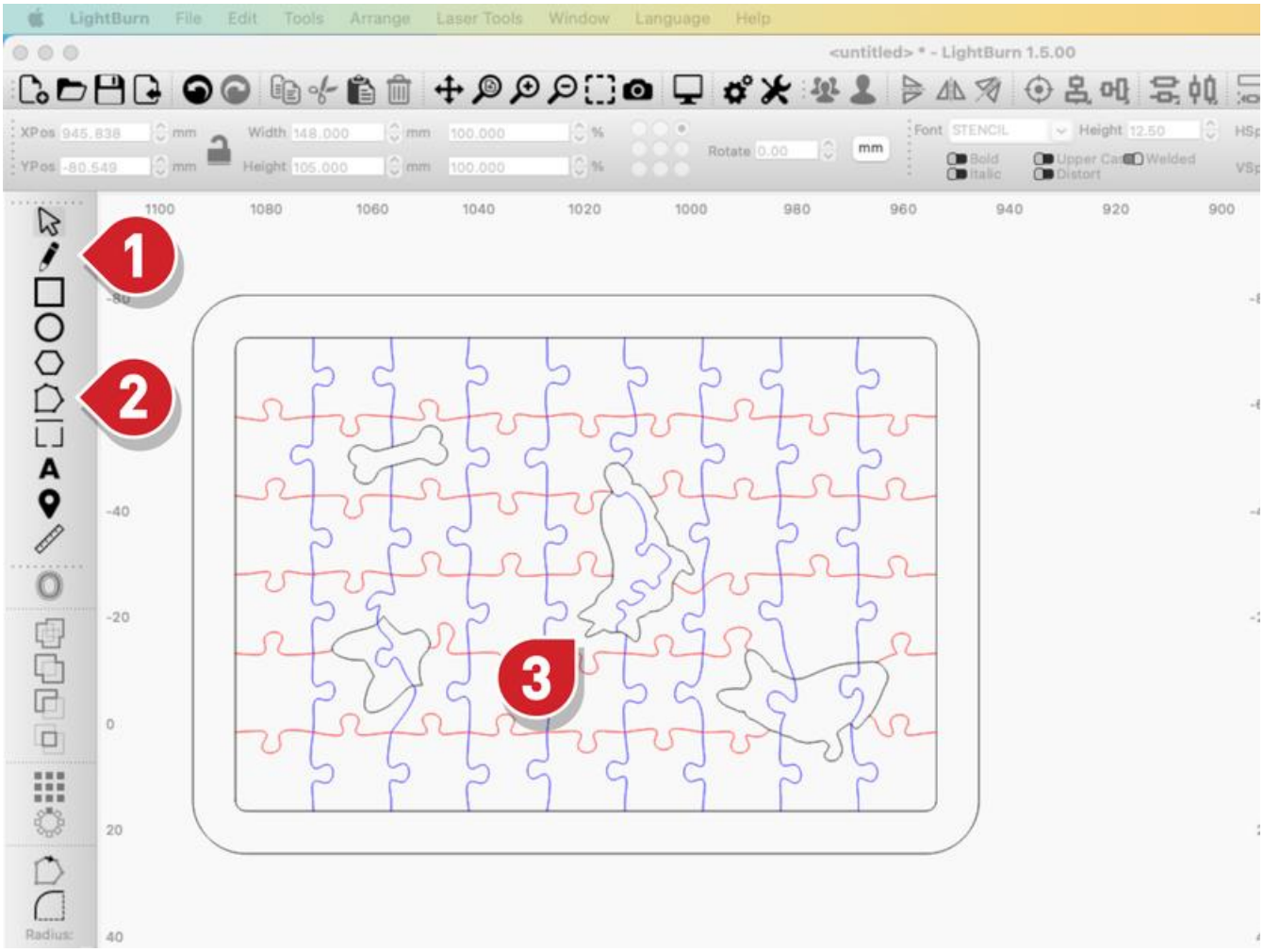


4. Add Fancy-Shaped Pieces

1. You can create new shapes using the Draw Lines tool, or find an existing vector online from somewhere like The Noun Project and import it. You also might want to get LightBurn to trace a pixel-based image for you.
2. You can edit both the new shapes and the puzzle using the Edit Nodes tool.

   > **Tip**
   >
   > You can hover over the node edit tool icon to see a list of hotkeys! The Cut Shapes tool might also be helpful.

3. As you customize the pieces, remember your shapes still need to be able to lock together, so add nubbins, and be sure not to make any parts too small, or they may end up too delicate!



5. Add The Pet Photo

1. Import the image using **File → Import** (or `Ctrl`+`I` / `Cmd`+`I` on your keyboard)., navigate to your pet photo file, and click **Open**.
2. Move and scale it into position using the Transform Controls, making sure your image is at least as big as the outer border of the puzzle.
3. Trim off the excess of the image to the outermost border line using the Image Masking tool. You can toggle the image to not show in the cuts/layers menu, if you're having a hard time selecting the border line.

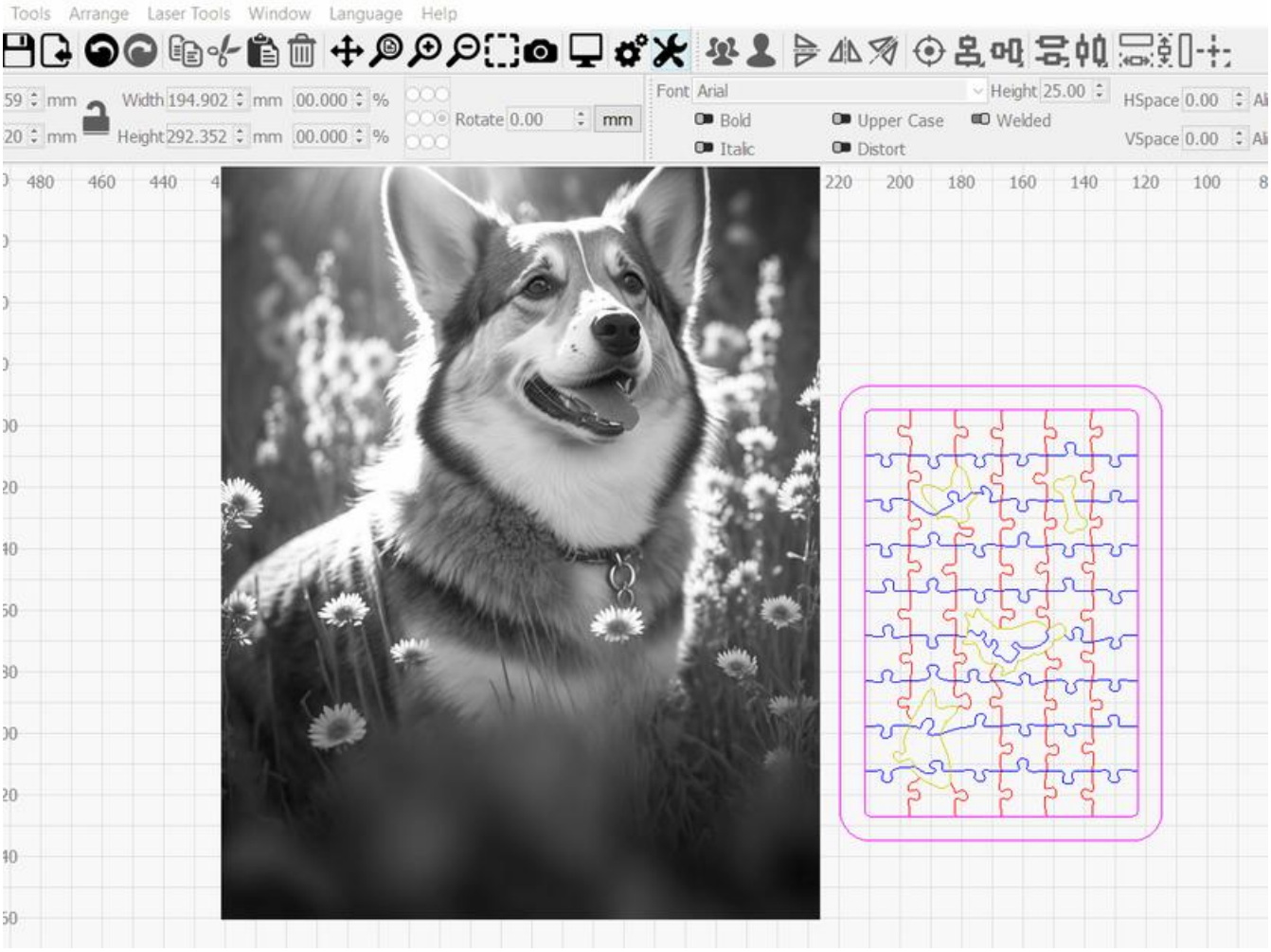
6. Set Up The Cut

1. The order of your cut layers should be as follows:
    1. Engrave Image
    2. Cut Vertical Pieces
    3. Cut Horizontal Pieces
    4. Cut Fancy Shapes
    5. Cut Border

2. Set your cut settings appropriately for your material for each layer. If you haven't found settings for your material, you can use the Material Test Generator.
3. Move your job into cut position as per your normal routine. If this is your first job, you may need to check your coordinates and job origin settings.
4. Preview your job.
5. Start your job.

7. Gift It To Someone You Love

Now that it's done, pop it in an envelope and send it to someone you love!



If you ever get stuck, please reach out to us, we're here to help!

Other Ideas

This is only one way to make a puzzle, you can customize this project to suit your own goals!

- Perhaps your loved one doesn't have a pet, but you could instead make it a picture of their kids, their marriage photos, a great holiday, a new house, a favorite movie poster, anything they love!

- Or you might like to use a printed color photo, in which case, our Print and Cut tool will come in handy when you go to align your puzzle to the image in real life.
- You could add a name, date or title to the puzzle, our Text tool is great for that! You'd then set that text layer to Fill Mode, to color it in.
- You don't have to make a little card, you can make it a full-sized puzzle by changing the size and number of pieces (tiles) in the puzzle generator.

No matter what you do, we hope you enjoy making it!

EDU

Welcome to LightBurn's educational resources. This section is still growing, so check back soon.

Why Use LightBurn

LightBurn works with a wide variety of laser cutters, allowing you and your students to design right in the software, import designs from other software you're already using, prepare designs to cut and engrave, and control supported lasers.

Our software is not a subscription and protects your students' privacy and personally identifiable information by never collecting it in the first place.

- Once you buy a LightBurn license, your license is good forever. Your purchase includes one year of updates[1].
- LightBurn does not require user accounts, and does not collect sensitive user data.
- LightBurn does not require an internet connection.

Licensing

A standard license comes with 3 seats – this means that with one license, you can run LightBurn on up to three computers. If you need more than that, you can purchase additional seats at a reduced price. If a computer is no longer in use, you can deactivate your license for that computer, freeing up the seat it was using for another computer.

Calculate the price with our quote calculator

Classroom-Friendly Licensing Setups

LightBurn offers two advanced licensing setup options intended for classrooms and other shared spaces. By using **System Locked** or **Floating** licenses, you can give access to more students than you have seats without having to use shared computer logins.

	System Locked	Floating
Summary	A **System Locked** license is locked to the computer it's activated on[2]. Each seat is tied to a specific computer, but can be used by anybody who logs into that computer. This does not require a special license type.	With a **Floating** license, instead of purchasing a seat for each computer you want to use LightBurn with, you purchase as many seats as you want to be able to use at the same time and install it on as many computers as you want.
Seats to Purchase	Purchase enough seats for the total number of computers you want to install LightBurn on.	Purchase enough seats for the total number of users that will run LightBurn at the same time.

	System Locked	Floating
Use Case	Ideal when you don't have a reason to install LightBurn on extra computers, or need to run it without internet access.	Ideal when you have significantly more computers that might be used with LightBurn than will be used at any particular time, for instance with multiple computer labs or student-issued computers.
Setup	Can be set up by anybody with admin rights on the computer, following our directions.	Generally set up by your IT department, folowing our directions.
Internet Connectivity	No internet connection required[3].	Internet connection required so your computers can check licenses in and out from our license server.
Number of Seats	No minimum number of seats, add as many as suits your needs.	Minimum of 10 seats. When you use our Quote Calculator, calculating for 10+ seats will generate links for purchasing either a regular or floating license.

Privacy, Security, and Accessibility

Privacy, security, and accessiblity are always important topics, and privacy is particularly important when working with minors. You can find answers to frequently asked questions from our educational customers below.

How is student data protected?

LightBurn does not manage or collect sensitive data from end users. See below for more information on what data LightBurn does collect for purposes of purchasing and managing software licenses.

Is LightBurn a cloud service? Does it require an internet connection?

LightBurn is a software package that runs entirely on the local computer. An internet connection is not required, except to activate a free trial.

Does LightBurn utilize an internet connection in any way?

If available, LightBurn utilizes an internet connection for three purposes:

- License activation and deactivation. If this is not an option, you can complete an offline activation instead.
- Checking for updates to the software and built-in links. This check takes place once a day when the software is opened, and accesses LightBurn's servers as well as a publicly accessible GitHub repository.

EDU

- If you choose to use the Web Integration for Cuttle, an internet connection is used to connect to Cuttle. LightBurn does not receive or send any additional information through the web integration.

LightBurn does not connect to the internet for any other purposes.

How does LightBurn use, collect, and share personal information?

At present, we do not collect or process personal information considered to be "Sensitive Information", "Sensitive Personal Information", or "special category data" in connection with our services.

For customers who license our software or make other purchases, we collect the following information:

- Name and contact information, including company affiliation.
- Order information.
- Payment information, including credit card account or bank account and routing information.

We use this information for the following purposes:

- To process payments and generate license keys.
- Service analysis, security, and administration.

Information is collected directly from the individual, and may be shared with our payment processor, our license key processor, and our sales tax remittance processor. It may also be shared with the following:

- **Processors/service providers:** We share your personal information with third parties that provide services to us. We engage these kinds of third parties with contracts that require them to use your personal information only for the purpose of delivering the services for which we have engaged the third party and as required by law. These kinds of third parties provide website hosting, business, professional, administrative, or technical support functions for us, such as payment processing, billing, data storage, quality assurance, and marketing.
- **Legal compliance recipients:** We disclose personal information to the courts, the government, law enforcement agencies, litigants, and similar recipients when required by law.
- **Successors:** We may disclose personal information associated with a part of our business to a buyer, potential buyer, or other successor to our business.

Where do you store data?

Your personal information will be stored and processed in the United States, where our servers are located. If we transfer personal information to the United States from another jurisdiction, we will do so in a way that complies with applicable legal requirements.

Does LightBurn back up the software configuration?

Yes, LightBurn's preferences are automatically backed up regularly. All backups are saved to the local computer. Read more in our documentation.

Does LightBurn provide regular alerts for software updates?

Yes. If an internet connection is available, LightBurn checks for updates on a daily basis and will notify users of an update when available.

Does LightBurn require any third party agreements or use third party software or components?

Lightburn utilizies the following third party software/libraries:

- **Potrace** image trace library licensed from IcosaSoft Software Inc
- **OpenCV** Copyright © 2026, Intel Corporation Download License
- **zstd** Copyright © 2016-2026, Yann Collet, Facebook, Inc. Download License
- **Sparrow** Copyright © 2025 Jeroen Gardeyn, KU Leuven Download License
- **jagua-rs** Copyright © 2025 Jeroen Gardeyn and contributors Download License

Frequently Requested Documents

These are some of the most requested documents from our educational customers. If you need additional documents, please contact support.

HECVAT

VPAT

W-9

Form 889 (by request)

Purchasing LightBurn

Purchase Orders

The best way to purchase LightBurn is directly through our online store. Our automated process is quick and easy to use at any time.

However, we recognize that some schools and other institutions require payment by purchase order. We are able to accept purchase orders, with some restrictions:

- We are unable to accept payment by check. Purchase orders can be paid by debit/credit card or ACH transfer.

- We are unable to accept Net30 or other similar payment terms. Your license key will be provided after we receive payment.

To pay by purchase order:

1. Set up an account in our online store. If the person handling the purchase is not the one making the account, please make sure they have access to the email you use for the account.
2. Contact support to let us know you want to pay by purchase order. Please include what you want to purchase, your tax-exempt status (and documentation of tax-exemption if applicable), and the email used for your account. If you're not sure exactly what you need to purchase, our support team can help you.

Please understand that at the present time, purchase orders are not automated. If you need your license key ASAP, purchasing directly from our online store is the fastest option.

Tax-Exempt Status

To request tax-exempt status:

- Create an account on our web store. Use the email you'll use for the actual purchase.
- Email support from the email you signed up with, requesting tax-exemption. Include documentation to show your tax-exempt status.
- Once given tax-exempt status, you'll be able to purchase on our website without being charged tax.

---

1. Your LightBurn license key will keep working after your update period expires, but only for versions of LightBurn that were released before the end of your update period. If you want to update to newer versions, you don't need to buy a whole new license – you just need to purchase another year of updates.
2. You can still deactivate the seat if you're no longer using this computer with LightBurn, freeing up the seat for use with another computer.
3. Without an internet connection, you will not have the built-in Cuttle integration, automatic checks for software updates, and one click access to the help pages by pressing F1 on the LightBurn UI.

What's New

What's New

▶ Watch the video on YouTube: https://www.youtube.com/watch?v=QxftdP83dww

LightBurn 2.2 is out now! We've got a lot to say about all the new features and improved hardware support in 2.2, so be sure to read to the bottom for full details.

Download LightBurn 2.2 Add a year of updates to your license

First, here are the highlights:

- New design, layout, and workflow efficiency tools: **True Nest** (Pro-only), **Center Line Trace**, and **Multi Page Projects**
- **Camera Print and Cut (Labs)**, a new option that uses a head-mounted camera to automatically identify target markers and align laser output
- We now support the **ACCELaser HD1 Series**, **Bambu Lab's new R1** laser, and a wide range of **Epilog Laser** hardware. We've improved or added support for several **xTool** machines. Epilog and xTool support are both "Labs" features
- **My Laser Tools**, an online project generator and design platform, is now available in the Web Tools Menu
- **REST API (Labs)** added, providing a way to communicate and integrate between LightBurn and other applications

True Nest (Pro-Only)

It's here — a full nesting solution, built into LightBurn. In 2.1 we added **Quick Nest**, a blazing-fast option that nests objects based on their bounding boxes.

Now we've upped our game with **True Nest**, which nests objects based on their actual contours. You can choose from four **Optimization** levels that determine how long it spends generating the best layout, but all of them are shockingly fast — especially compared to doing it by hand.

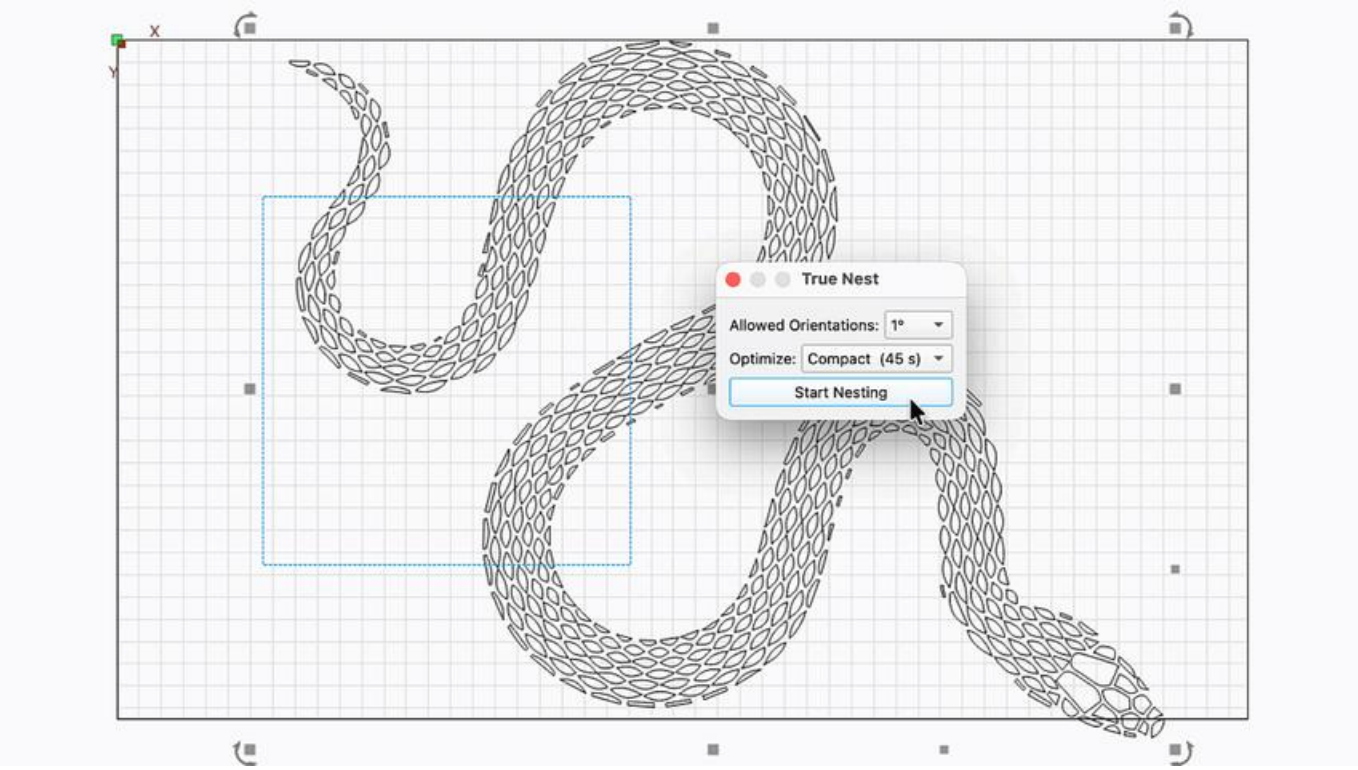
An important note: True Nest is the first LightBurn design/layout feature that is limited to Pro licenses only. We made that choice because premium features like True Nest are really hard to write, and the only way we can develop features like this and still keep the lights on is to charge a bit more. We also recognize that not all users need it, so rather than raise prices across the board, we chose to make it Pro-only as the fairest option.

If you're a Core user, upgrade to Pro if you think True Nest will help you work faster and save money on material. It shouldn't take more than a few sheets-worth of savings to pay for itself.

Not sure yet? Head to **Help → License Management** and click the **Trial Pro** button for two weeks of Pro access.

Learn More Upgrade to Pro

What's New



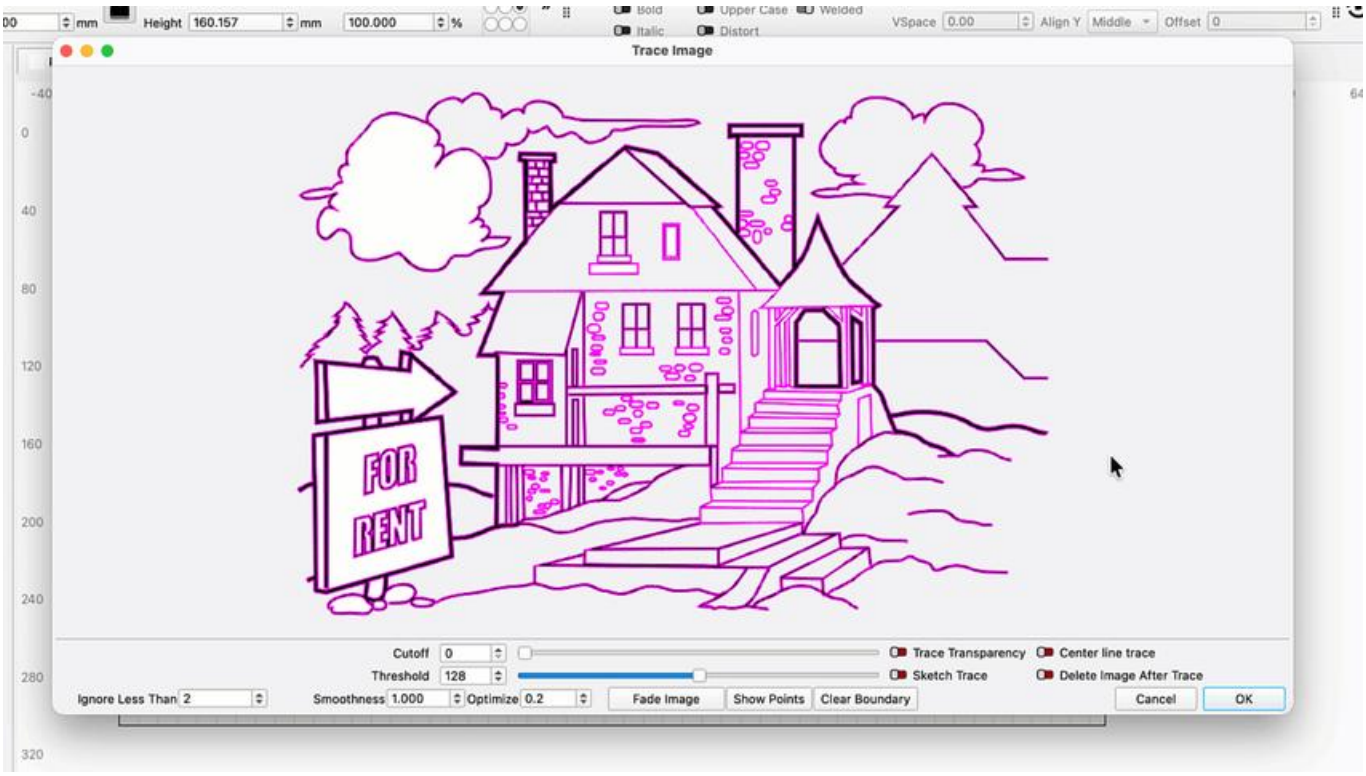
Center Line Trace

We've added a new option to the **Trace Image** tool — **Center Line Trace**. Unlike a standard trace, this option produces lines running along the medial axis of shapes identified in your image. It's a great option for tracing logos, sketches, and text.

While "medial axis" is sort of a fancy way of saying "center line", it also carries an important distinction — this is cold, hard math, and the result might not be quite what you think it should be by eyeball. Especially if you feed in a low resolution image, you'll get a result with lots of offshoots where the pixel-to-path equation says the medial axis is. Stick to high-res images with clear outlines for best results.

Learn More

What's New



Camera Print and Cut (Labs)

Camera Print and Cut uses a head-mounted camera to identify registration marks on your material, then aligns your output to cut around pre-printed graphics.

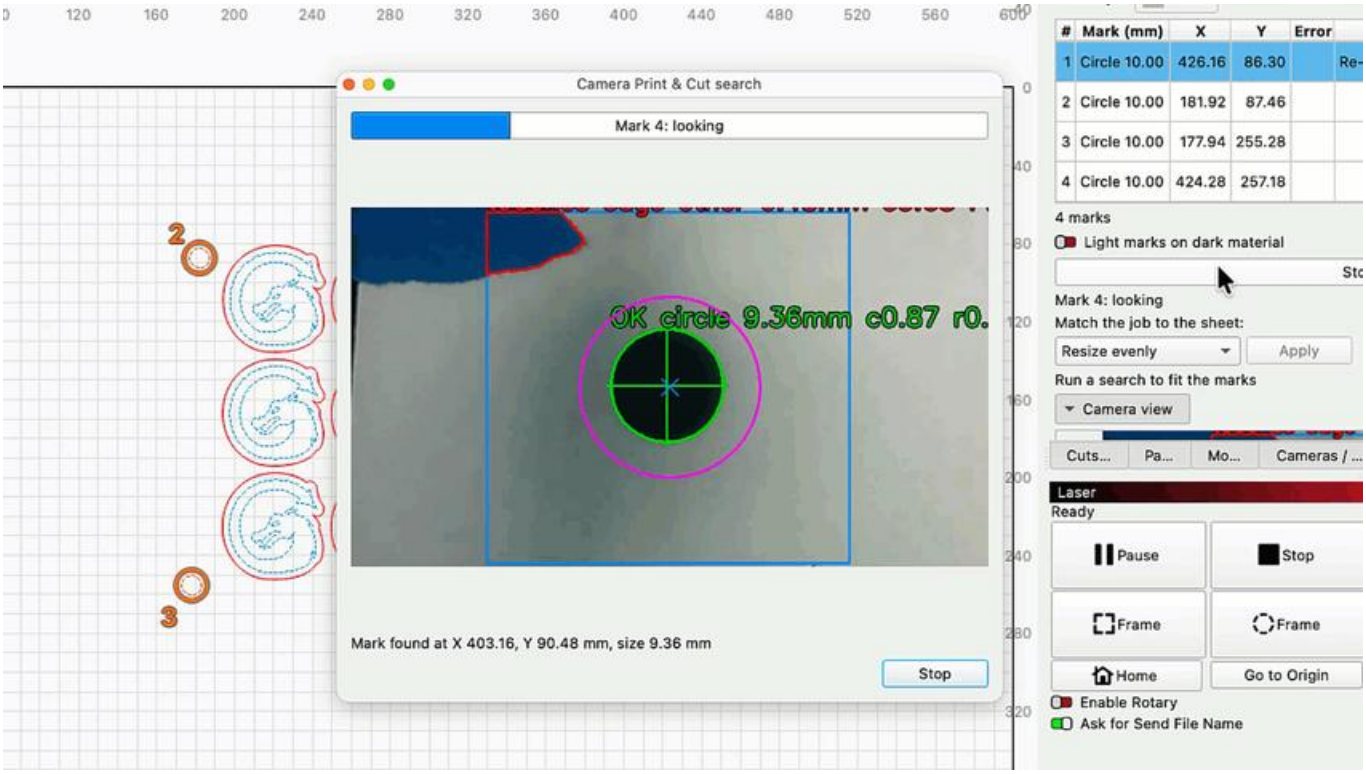
Manually jog your laser over the first one and hit go. LightBurn will automatically jog the laser to identify the rest of your targets, then position and orient output to match.

If your laser didn't come with a head-mounted camera, don't be discouraged — with zip ties and an intrepid spirit you can strap an endoscope onto just about any laser on the market and make use of this tool.

Our initial implementation is extremely accurate, but we're marking this as a "Labs" feature nonetheless. It was a late arrival to 2.2 (we couldn't wait, and we didn't want you to wait), so there is some potential for bugs or unexpected results. As with all Labs features, use with care — and cheap materials at first.

Learn More

What's New



Multi Page Projects

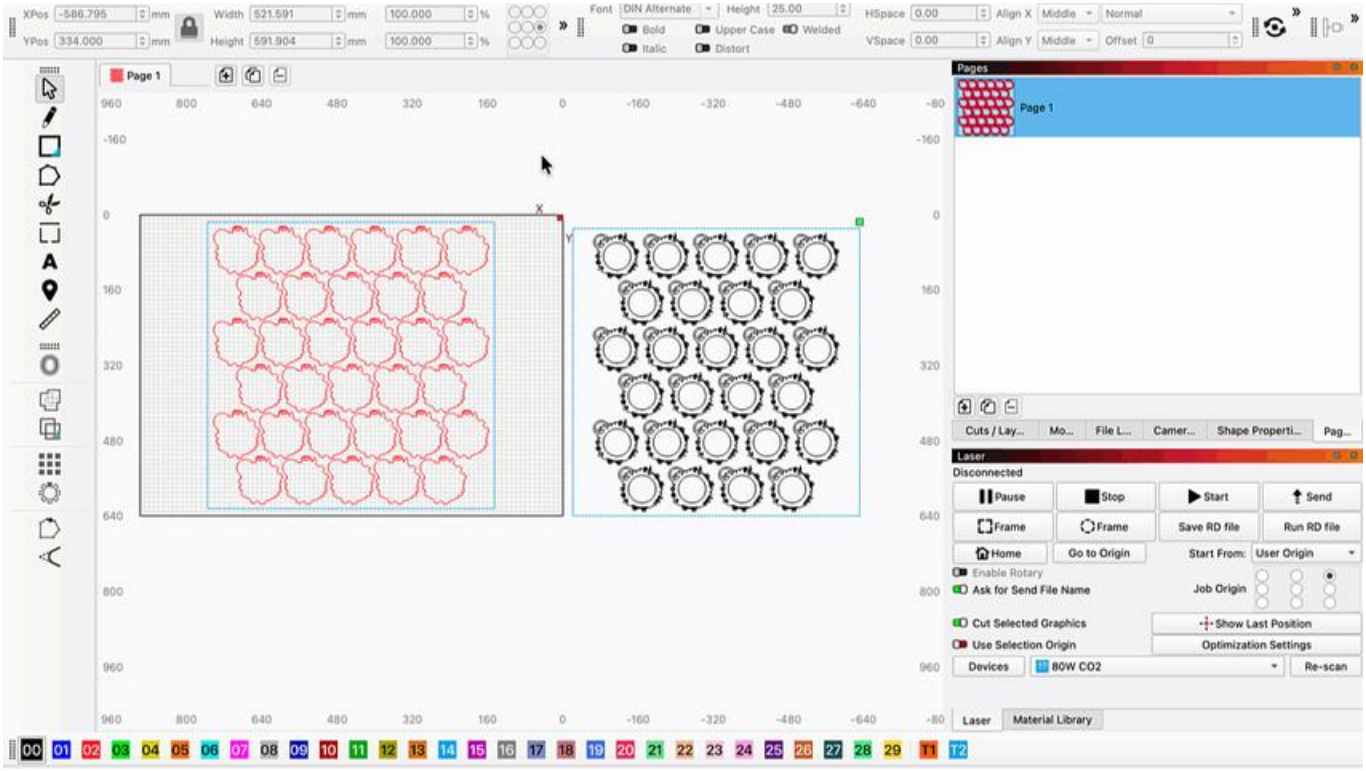
We see you out there with your graphics all spread around the margins of your Workspace. Please stop doing that, it makes us nervous. We've seen the devastation when a file like that gets lost or corrupted.

With **Multi Page Projects**, you can get more organized. It's still not a great idea to load up a project with tons of unrelated graphics in different tabs, but Pages let you sort related graphics into separate spaces.

You can **Add**, **Remove**, **Copy**, and **Rename** Pages from the interface at the top of **Workspace**, or in the new **Pages Window**.

Learn More

What's New



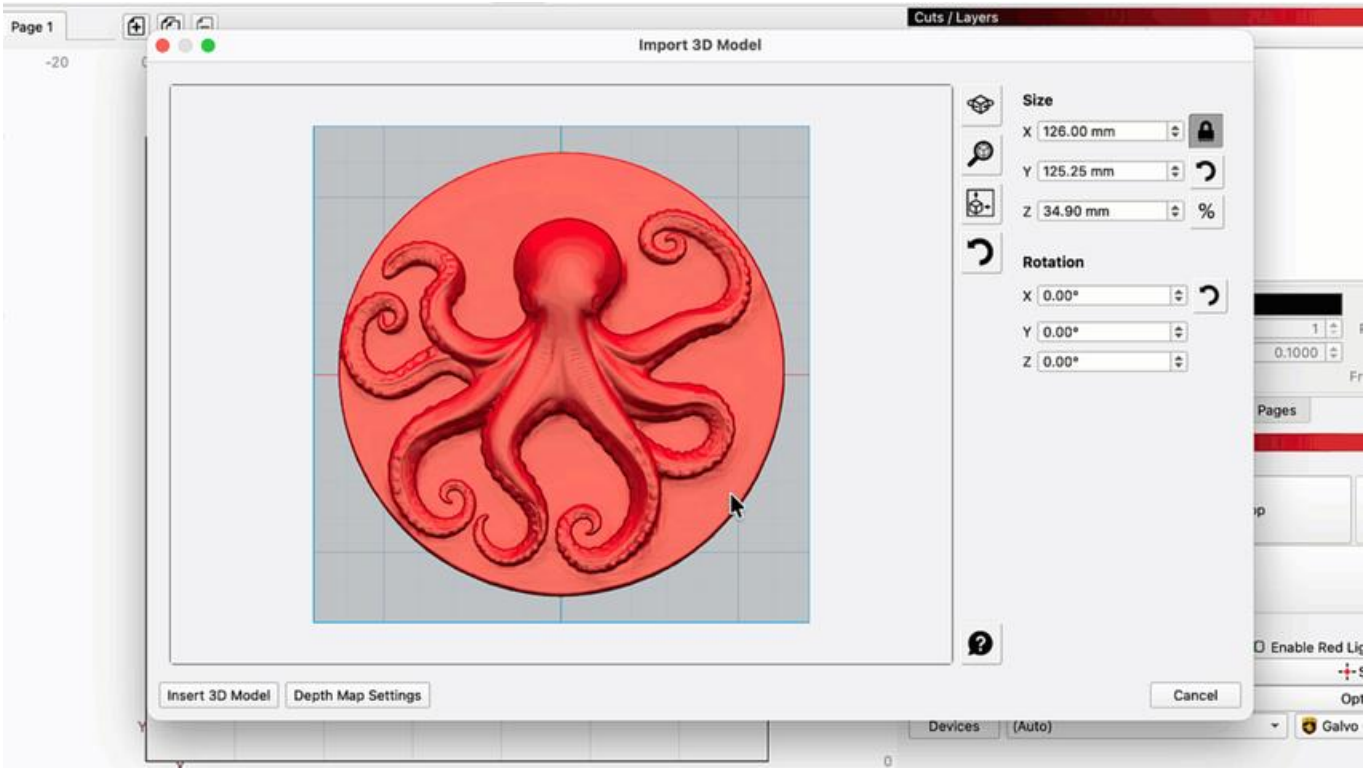
3D Model Import

We've added a new way to source depth maps for your 3D engravings — you can now import 3D models in .stl, .3mf, .amf, .obj, .off, and .ply formats.

Adjust the model's **Orientation**, **Scale**, **Bit Depth**, **Resolution**, and other settings, then add it to your **Workspace**. It will appear in red while it remains an adjustable 3D model. Right-click and select **Convert to Bitmap** to make it a true grayscale image (note that you'll lose the ability to adjust the model's properties after converting).

Learn More

What's New



REST API (Labs)

We've added a REST API to LightBurn, providing a way to communicate between LightBurn and other applications. The API is intended to allow users and developers to create integrations and improve interoperability with other software for advanced workflows, automated processes, status monitoring, and data exchange between LightBurn and third-party applications. The API is intended for use on a single computer or within your local network.

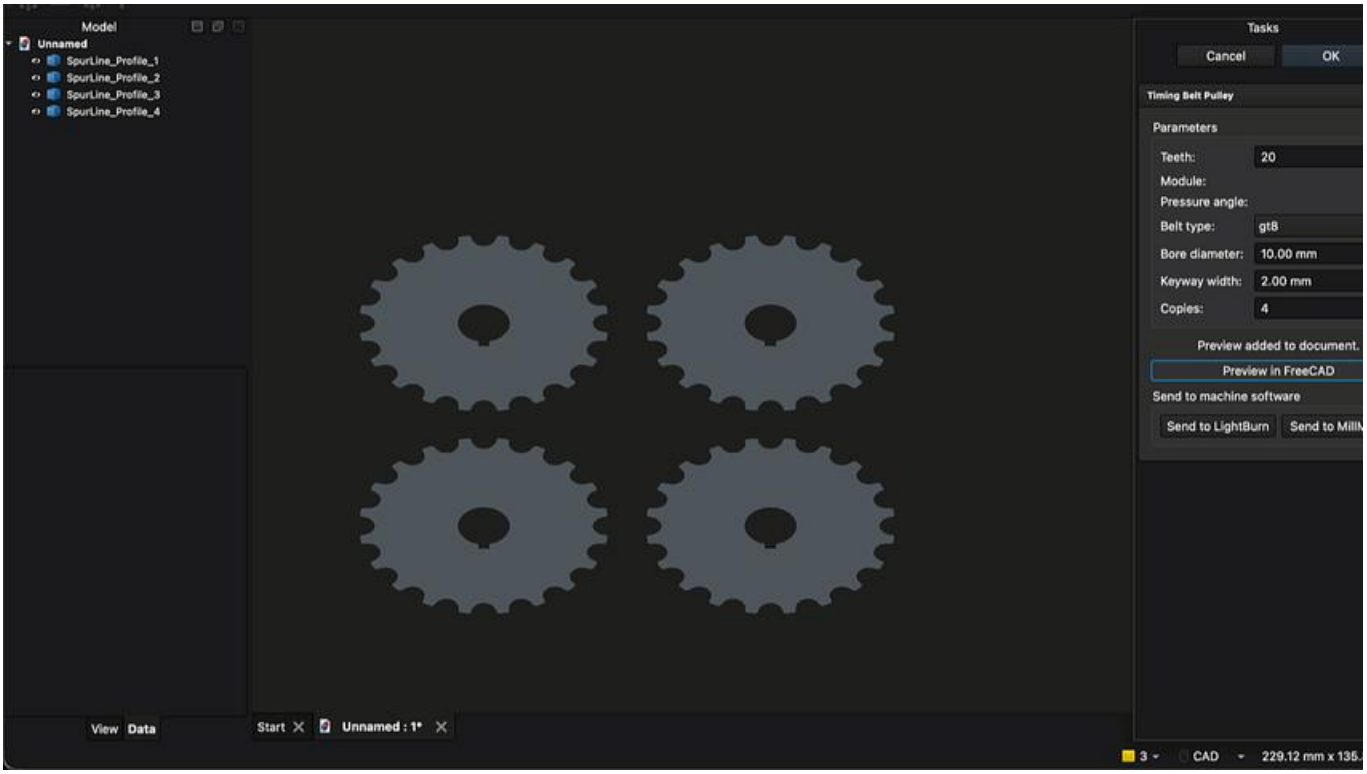
Public documentation for the LightBurn REST API is available now on GitHub.

More detailed documentation is coming soon.

As a reminder, LightBurn does not endorse, recommend, support, or encourage unattended use of lasers or other machinery. LightBurn's API is not a replacement for direct operator supervision.

Learn More

What's New

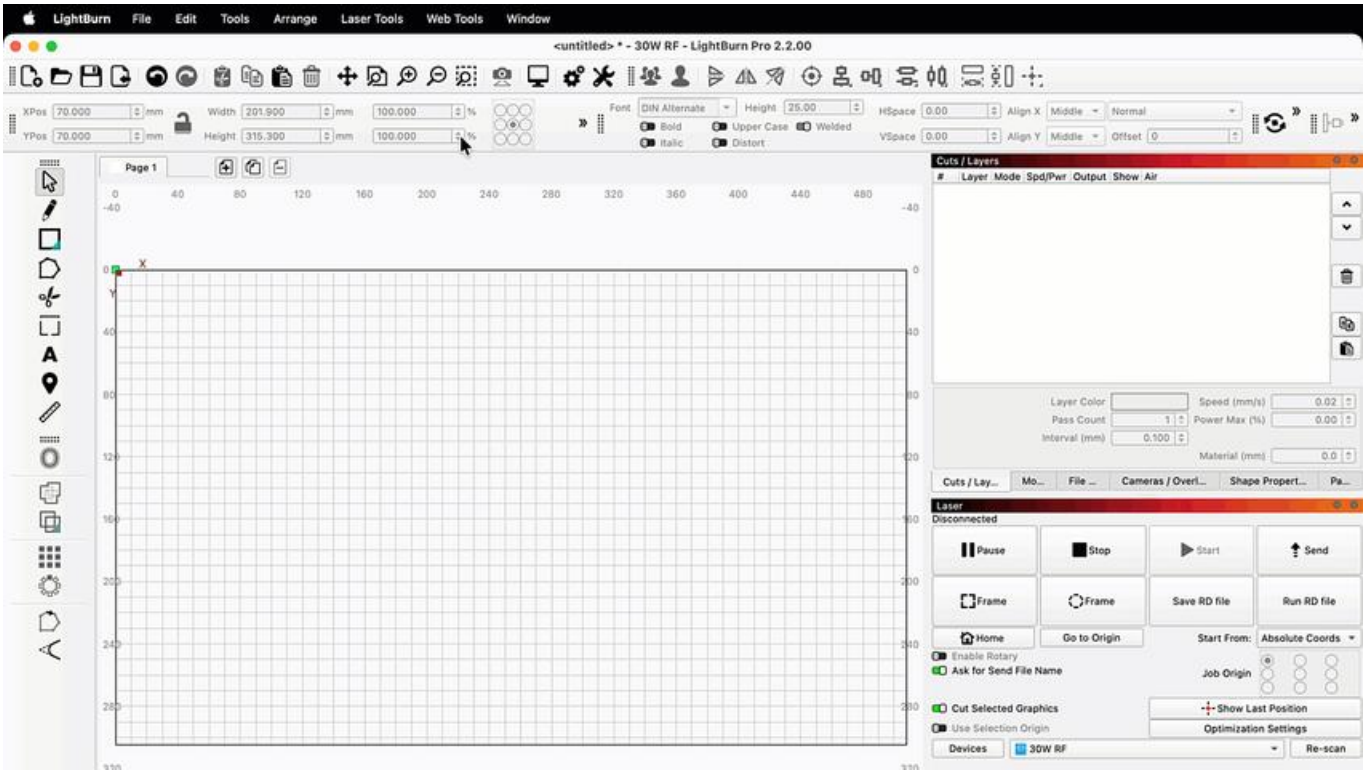


User Tools

You can now add custom entries to the **Web Tools Menu**. Click **Manage User Tools** then **Add**, and enter a **Name** and **URL**. You can also specify a **Category** to keep things organized.

User Tools work the same as built-in Web Tools — use your chosen generator in the built-in browser, then send your result directly into LightBurn by using the download button.

What's New



Learn More

ACCELaser Support

The first LightBurn-supported flying galvo lasers are here: the ACCELaser HD1 series.

The HD1 series introduces game-changing technology, combining both gantry and galvo motion systems into a single machine and leading to much faster processing times without sacrificing cutting ability or working area.

It features dual diode and fiber laser sources, so you can select the right option to mark or cut a range of materials, from wood, to slate, to metal.

For positioning, it offers a dual camera system — one global overhead camera, and another mounted to the laser head.

Finally, the HD1 series supports LiDAR ranging for curved surface and multi-height engraving.

We've worked closely with the ACCELaser team from the start to make sure we could provide full support for this exciting new technology.

Learn More

Bambu R1 Support

We've added initial support for the new Bambu R1 CO2 laser. Our work on this is not yet done, and we're coordinating closely with Bambu to expand our support.

What's New

Here's where we're at: in 2.2, LightBurn has the ability to create and export G-code compatible with the R1. You'll need to use a special device configuration file provided by Bambu to do so.

In 2.3 (coming in the not-too-distant future), we're planning for more support: direct connection, all the R1's special features, and the complete start-to-finish LightBurn workflow.

Epilog Support (Labs)

We're thrilled to announce that we've added support for Epilog lasers in LightBurn 2.2.

Epilog has long been a leading manufacturer in the laser industry, with a reputation for quality, high-performance machines. We've worked closely with their team to implement LightBurn support for their full range of hardware.

Note that our support for Epilog lasers is a "Labs" feature in 2.2. That means we've tested to the point that we're confident it's ready for public use, but we also know it needs a little more testing before we call it 100% complete.

Please use with caution, and be sure to report any issues you encounter to us at Support@LightBurnSoftware.com.

Learn More

Improved xTool Support (Labs)

Our support for xTool lasers has improved across the board in 2.2, with better support for all machines that previously worked with LightBurn, and added support for a handful of new machines.

There's so much to share that we'll have to express ourselves in bullet points:

- We've reworked and expanded our native xTool protocol to additional devices, resulting in a much more stable experience across all supported machines
- We've added WiFi/network support for F1, F2, and P2 series machines
- Multi-source jobs are now supported for the F1 and F1 Ultra
- We now support auto-focus for P2 and P2S lasers with IR rangefinders

Note that our support for additional xTool machines is also a "Labs" feature — a lot of these changes are hot off our devs' keyboards, and in need of a bit more testing. Please report any issues to Support@LightBurnSoftware.com or on our forum.

Learn More

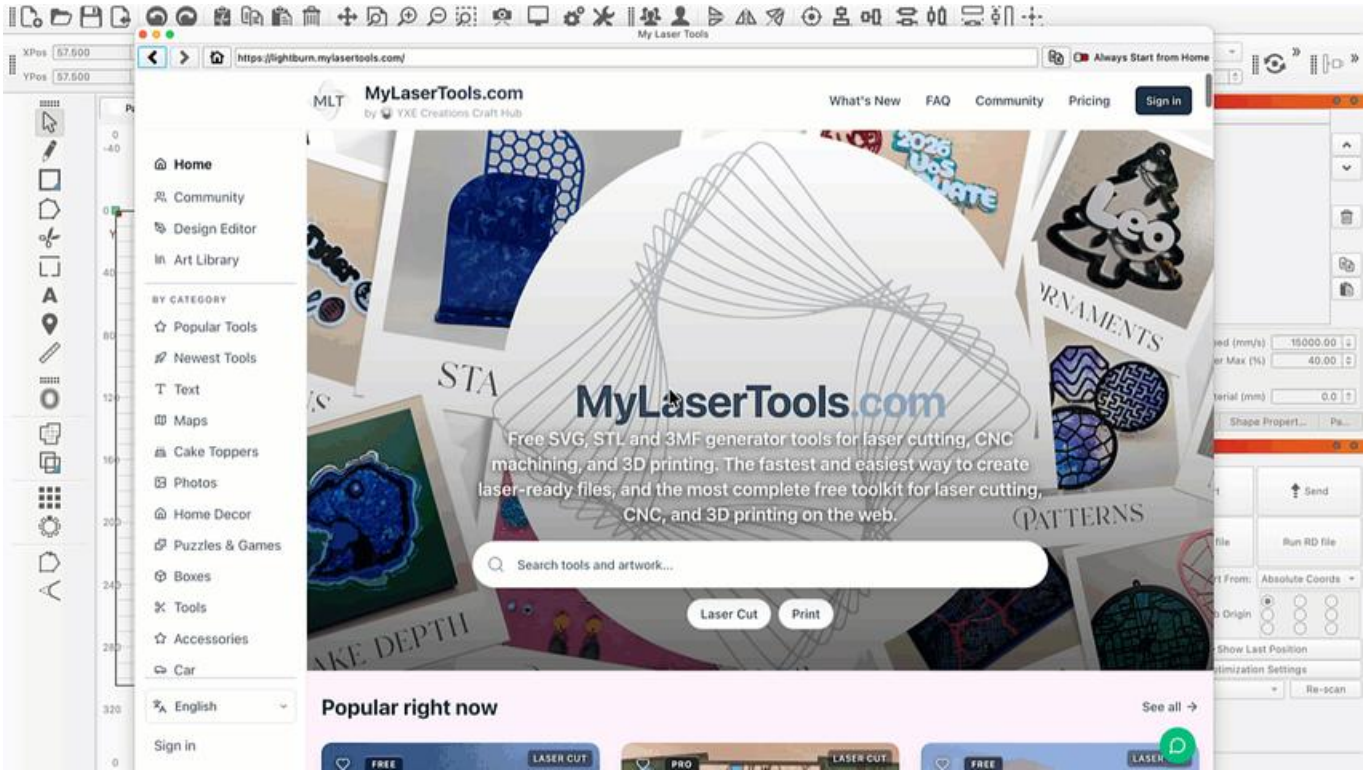
My Laser Tools Integration

MyLaserTools.com is now available in our **Web Tools Menu**. My Laser Tools provides a set of design tools for laser cutting, engraving, and beyond. Most are free to use, ranging from connected text and boxes to keychains and signs, alongside art assets and a premium designer for creating custom map art.

What's New

As with previous integrations, My Laser Tools offers both free and paid tiers. As a reminder, this isn't a financial arrangement for us, and we don't take a cut of any subscription payments.

Our goal with all integrations is to help makers find useful tools and give visibility to the tool-makers who provide them. As the options grow, you'll find more overlap among our integrations — that's by design. We're committed to giving our users choices, and not locking anyone into a specific machine ecosystem or any single design platform we partner with.



ARM Builds for macOS

We now have macOS builds natively compatible with ARM-based CPUs (sometimes referred to as "Apple silicon" or "M series" chips).

That means you won't have to wait a minute or two for LightBurn to open on your Mac the first time you run 2.2!

Apple has announced that with the release of macOS 28 (anticipated fall 2027), they will no longer support apps that are not specifically built for ARM. Any user that has a Mac with an "M series" chip and updates to macOS 28 will be **required to update to LightBurn 2.2.00 (or later)**. Previous versions of LightBurn will not run on macOS 28. This will also affect any user who purchases a new Mac after the release of macOS 28.

In-App Bug Reports

We thought you all might need a new way to tell us what we're doing wrong, so you can now submit bug reports directly through the software. Go to **Help → Report a Bug...** then fill out the form to let us know what happened. You can also choose to send a screenshot and additional data.

What's New

Please limit usage to bug reports only. If you have a feature request, click **Submit Feature Request** a little higher up in the same menu.

Other Notable Additions

- Find My Laser can now discover network connections to supported lasers, including Epilog, Creality, and xTool devices
- Duet/RRF HTTP flavor added to the Custom G-code Editor
- Custom hotkey support added to the Macros Window

Changelog

Click to Read our Full Changelog

2.2.01

Changes Since 2.2.00

New Features

- EZCad3 Laser Ready input in Device Settings, which holds marking until the laser reports it is ready, as EZCad3 does

Existing Feature Updates

- Help → Report a Problem offers Suggest a Feature, Troubleshooting and Report a Bug, and bug reports attach the open project
- Accelaser network discovery and galvo framing
- Acclplanner v0.31.2
- EZCad3 laser status checks follow the Disable Laser State Check and Check Laser State When Marking settings from the imported config
- EZCad3 UV and YAG cut settings use pulse width and frequency in place of power, as EZCad3 does
- EZCad3 laser power follows the Power Map from the imported config
- EZCad3 red pointer uses the laser serial command on UV, YAG, CO2, SPI and some IPG fiber sources
- EZCad3 config imports convert inch configs to mm, read the stop marking input, and bring in internal galvo correction only when EZCad3 uses it

User Interface Updates

- Head camera alignment verification zooms in on the area the camera sees
- Continuous Jog toggle in the camera alignment wizard, on devices that support it
- EZCad3 Laser Type list offers UV/YAG for UV devices, and drops None and UV
- EZCad3 FPK settings are labelled as pulse widths in Âµs

What's New

Performance Improvements

- Camera calibration preview image processing runs on the render thread

Bug Fixes

- LightBurn could fail to start on Windows with Smart App Control enabled
- Quitting could crash or hang
- Importing a JPEG with a CMYK color profile could crash
- xTool, Creality Falcon and Duet devices could crash if LightBurn closed during an upload
- The Extensions tab in Galvo Device Settings misbehaved
- Camera preview could freeze after a reconnect
- Camera alignment wizard crashed when importing an Image-mode Material Library entry
- Camera alignment patterns could show bi-directional scanning offsets
- Camera alignment wizard hid Power in favor of Q-Pulse on sources that support both
- Head camera manual alignment kept points clicked before a jog
- xTool S1 reported an open lid as a "cover open" console error in place of a "Lid opened" message
- EZCad3 UV and YAG machines showed a false Temperature alarm and could not start jobs
- EZCad3 UV and YAG machines got fiber power commands with no pulse width, frequency or laser enable, and could not mark
- EZCad3 SPI and some IPG fiber sources could report the wrong laser alarm
- EZCad3 SPI laser sources got no frequency command
- EZCad3 CO2 and YAG sources with a maximum frequency of 20 kHz or less used the wrong time base
- EZCad3 lasers with a maximum frequency of exactly 1000 kHz used the wrong pulse clock
- EZCad3 PWM On Delay values were always sent as the longest possible delay
- EZCad3 Laser On Lag Time never reached the controller
- EZCad3 machines with 18 or 20 bit galvo scanners got the wrong scanner range
- EZCad3 configs with an SPI or QCW laser source could not be imported
- EZCad3 fiber config imports left Q-Pulse Width off even when EZCad3 had pulse width enabled
- EZCad3 config imports put the laser ready input port into the Ready Light, so an unrelated output switched on and off every job
- EZCad3 config imports reversed the start marking and start framing input polarity, so a foot pedal started jobs on release in place of press
- EZCad3 config imports lost a mirrored galvo axis
- EZCad3 start framing polarity and hardware fly settings from imported configs had no effect

What's New

2.2.00

New Features

- Added True Nest
- Center Line Trace.
- REST API for local and network inter-application communication.
- Target buffer override for G-code devices whose firmware doesn't report the buffer size or reports it wrong.
- 3D model import.
- Depth map creation from imported 3D models.
- Extra controls for Custom G-code devices, similar to GRBL.
- FluidNC flavor filtering to clear [MSG] spam from the Console.
- 3D model import added. Your Benchy finally has somewhere to go
- Creality Falcon HTTP flavor for Custom G-code.
- Duet/RRF HTTP flavor for Custom G-code.
- Accelaser hybrid gantry + galvo device support.
- LiDAR surface scanning with height-compensated engraving.
- Rotary setup on Emblaser Pro.
- Accelaser LiDAR ranging dialog improvements.
- REST API added. LightBurn itself still does not rest
- Default G-code file extensions for Custom G-code flavors.
- Revert to Manual Alignment button on the Overhead Auto Alignment page.
- WeCreat HTTP flavor for Custom G-code, for sending job files to WeCreat lasers.
- Device Classification (Model and Source) in Device Settings, allowing full offline design for adaptive protocols.
- TopWisdom rotary support.
- Galvo Move panel as a dockable widget.
- PCB file import (Gerber and drill files).
- Epilog laser support.
- Calibration Target Collector.
- xTool F1, F2, and P2 series support over network (F1, F1 Lite, F1 Ultra, F1 Ultra 2, F2, F2 Ultra, F2 Ultra Single, F2 Ultra UV, P2, P2S).
- Camera and Depth Overlay Registration dock widget.
- Report a Bug in the Help menu.
- REST POST endpoints and Project write capability for writing settings to a project.
- REST API reports and edits dual-source, galvo and MOPA laser settings.
- Split Marking on EzCad3 devices, tiling along one or two page axes as a grid while any other axis holds a fixed position.

What's New

- External Axis Setup on EzCad3 lists every configured axis with a tiling or fixed position role, and refuses two rows that drive the same motor.
- EzCad3 split marking starts tiling from the worktable origin set up in EzCad3, so the first tile lands under the galvo head instead of at the home switch.
- Table shear correction for split marking cancels the seam drift of a worktable whose rails are not square to the galvo field, imported from the EzCad3 configuration and editable in Split Marking Setup and the runner window.
- External Axes Settings tab in Device Settings for EzCad3, with one control group per axis covering motor selection, step scaling, speeds, homing, position readout, jog, Home and Stop, and on the rails a move to the worktable origin.
- xTool rotary support for D1, F1, F2, P2, S1 over network in Absolute and Current Position modes.
- Rotary Park Position to 'park' the head at a specific position before switching into rotary mode (initially for xTool).
- REST endpoints to fetch the existing camera overlay image and its metadata (see https://github.com/LightBurnSoftware/LightBurn-REST/pull/2 for API changes).
- REST API changes for galvo vs gantry cut settings endpoints; clients must update, see https://github.com/LightBurnSoftware/LightBurn-REST/pull/3.
- xTool cloud credential updates to support new firmware.
- xTool and Creality Falcon network devices now appear in Find My Laser - Search Network.
- Galvo Move window and Z axis settings for BSL galvo lasers, with configurable homing that stops the axis.
- Print and Cut using a head-mounted camera.
- xTool workspace size read during network device connection.
- LBP 0.8 Labs Integration.
- Custom web tool support.
- BSL door protect support.
- Print and Cut with a head-mounted camera. The camera is mounted on the laser head, not yours
- Extensions tab on the galvo settings page.
- Vector troubleshooter.

Bug Fixes

- Typos in Quick Nest messages.
- Some Custom G-code values were unit-converted when they shouldn't have been.
- Launching and immediately closing could crash the web tool loader.
- The on-screen keyboard didn't appear when touching a number field on a tablet (Windows only).
- The 3D import dialog viewport didn't render.
- Min tabs was not always respected.

What's New

- Equations in number entry fields were parsed wrongly when using a comma vs a period.
- The dogbone tool could fail in some cases; it now uses double-precision math.
- Text could change position after copy and paste because its new width wasn't handled.
- The edit window rebuilt its render buffer constantly, including on every repaint.
- The trim tool didn't work in some cases.
- Select all shapes for an operation accidentally included shapes from other pages.
- Toggle Panel visibility didn't toggle the Pages panel.
- 3D depth maps weren't imported with an image cut setting.
- Memory leak in the cut settings editor.
- The Device Settings dialog cut off Custom G-code options.
- Framing could use the wrong origin offset.
- Custom G-code didn't use the custom header when framing.
- Reset to Default Layout didn't work.
- Network permissions were missing on macOS 27.
- Re-exported images were sometimes rotated 180 degrees.
- The Device Settings dialog could cut off or squash widgets; its minimum size and stretch policy are now set correctly.
- Newer Qt was no longer resolving host names in some cases.
- Opening a LightBurn or MillMage file from the other app showed an incorrect warning and didn't preserve pages the way importing does.
- SVG import was missing content from files using SVG 2.0 features and sized text wrongly; skew and SVG 2.0 / CSS unit types (%, em, ex, ch, q...) are now supported.
- LightBurn no longer Panic(s) at the Disco
- Number entry fields could change value randomly.
- Pause/resume sent two pause commands while streaming G-code.
- Shape Properties panel fields didn't support flexible number input, unit conversion, and formulas.
- The position readout didn't refresh after a discrete jog or work-offset change.
- 3D model rotation was wrong after switching between mouse rotation and the rotation fields.
- Tab order of controls in the 3D import dialog was wrong.
- Connection buffers could block in rare circumstances on flaky grblHAL controllers.
- PDF files with object entries split across lines weren't handled.
- PWM frequency raised to 4000 kHz. Dogs within a three-mile radius have been notified
- Print and Cut label icons didn't follow the theme colors.
- Some checkable menu options didn't show check mark icons.
- Thunder Titan controllers now get separate laser enable flags.
- The PWM frequency limit is raised to 4000 kHz.
- Units typed in upper case (MM or IN) didn't convert.

What's New

- Preview could crash after importing or opening a MillMage file.
- EzCad3 fiber marking was slow when Always Open MO was disabled. Every jump closed and reopened the master oscillator and waited the Open MO Delay, which added several milliseconds per jump. The oscillator is now opened once per job regardless of the Always Open MO setting.
- EzCad3 devices created with Find My Laser were saved with gantry simulation parameters. The protocol now supplies galvo defaults, and the Read Sim Params button is available for EzCad3 devices.
- Shape Alt-drag didn't snap properly to vertical positions.
- Sparrow and jagua-rs license details were missing.
- The REST API status didn't report job progress correctly.
- The 'Enable Laser 2 Controls' checkbox didn't look disabled when it was disabled.
- Swap X/Y Axes had no effect on Custom G-code devices.
- Z axis homing settings in Device Settings misbehaved: Enable Homing was bound to the limit switch flag instead of the homing flag, turning homing on or off didn't take effect until the dialog was reopened, and the homing direction, homing switch level, limit switch level and jog direction radio buttons could show neither option selected.
- Stepper speed and homing timeout spin boxes in Device Settings and External Axis Setup clamped imported values, silently halving a fast Z axis and capping its homing timeout.
- Framing or sending a design with split marking enabled from the main window no longer reports "No shapes in bounds".
- The frame preview for a split marking or rotary job moved the external axis using the page axis number instead of the motor port.
- EzCad3 external axis commands enabled the controller limit switch mode whenever homing was enabled, so an axis with homing but no limit switches homed away from its switch and ignored the Homing Direction setting.
- EzCad3 external axis homing used a fixed 30 second timeout instead of the go-home timeout imported from Motors.ini, aborting slow homing moves partway through.
- The steps distance of a split marking or rotary axis was not saved on EzCad3 devices and reverted to one unit each time the dialog was reopened.
- Closing or switching away from an EzCad3 device while an external axis was moving or homing blocked for up to the move timeout before the connection shut down.
- Connecting to an EzCad3 machine with Find My Laser and no config files froze.
- EzCad3 galvos didn't return to the red dot rest position after a job was stopped, skipped the rest position jump when the cached position already matched, and went to field center instead of the configured rest position at the end of a job.
- Frequency control was missing for EzCad3 devices.
- Camera overlay was never updated for negative-W alignment matrices.
- UI font scaling could enlarge the UI on low-resolution screens and crop some UI elements.
- Invalid environment variables carried over from older LightBurn versions could cause UI scaling issues.

What's New

- The Galvo Move window appeared on gantry, JCZ and BSL devices, and once shown was remembered for that device. Each Move window now decides from the device profile whether it applies, the galvo one defaults to hidden and refuses to show elsewhere, and profiles that had stored it as visible correct themselves.
- The Jog Laser Down hotkey did nothing due to a typo.
- Preview frame rate issue in the new auto camera calibration wizard.
- Editing Machine Settings could crash when the device returned errors.
- Deleting a page or undoing a page delete didn't update the list of active layers.
- 16-bit greyscale and ARGB32 images couldn't be adjusted in the image adjust dialog.
- Epilog engrave speeds just under 1% showed ?? on the machine display and ran too fast.
- Epilog CO2 fill and image layers ignored the power setting and engraved at full power.
- Docked panels overlapped and clipped their contents at large font and icon sizes.
- MOPA QPulse and Frequency support were confusing for Thunder devices; those settings are now consolidated.
- Custom G-code didn't output rotary settings in headers.
- Changing the active page refreshed twice.
- Galvo Rotary Marking and Split Marking advanced the serial number when no job was completed.
- REST API jog settings could not be changed, and invalid jog, home and override values were silently accepted.
- REST API cut settings repeated the base layer in the sub-layer list and left out sub-layer settings.
- Overcut cut short and ended off the outline on curves.
- Camera UI and thread safety improvements.
- The start point editor could show the arrow in the wrong direction on closed shapes.
- Camera settings didn't persist in preferences.
- Epilog device settings didn't match the actual device capabilities; they're now locked to match.
- Camera auto modes overrode saved manual settings.
- Depth maps from imported 3D models were invisible when the view mode was set to filled.
- Depth maps lost their image type when moved to a new layer.
- REST API omitted the first sub-layer's name and wrongly reported the Show checkbox status.
- The Window menu's Move entry showed unchecked when the Move window was docked behind another tab, and could be selected on galvo devices without a Move window, where nothing appeared.
- EzCad3 Repeat Marking only turned the rotary a fraction of a degree per step when the axis type was set to Rotating.
- Titan controllers showed a current position option they don't support.
- Camera buttons were missing from the Camera / Overlays widget.

What's New

- SVG export wrote varying line weights and incorrect line attributes.
- UI issues in connection request dialogs.
- Under certain conditions the calculation of required resolution for current font and icon size settings was incorrect, and could prompt users to change display settings when unnecessary.
- With "Require framing before start" enabled, the Start Marking foot pedal on BSL lasers started marking immediately instead of opening the framing window first.
- The external Stop Marking input on BSL lasers did not stop jobs started from the main window.
- On EzCad3 lasers, Run Continuously and Repeat in the framing window counted jobs rapidly while only marking occasionally.
- BSL split marking settings weren't saved.
- Epilog speed limits weren't clamped in the Cuts / Layers window.
- Rows in Cuts / Layers didn't resize when their height changed after linking to the material library.

Existing Feature Updates

- Number entry fields accept a comma or dot as the decimal separator, ignore earlier commas and dots, and show a thin space as the thousands separator.
- Settings show 'Cmd/Ctrl-Arrow:' instead of 'Ctrl-Arrow:' to indicate the Mac Command key.
- Path and unusual characters are filtered out of the document title in exported AI files.
- Latest Visual C++ Redistributable, with the unneeded version removed from the installer.
- Break apart and Auto-join now preserve tabs.
- Custom shortcut keys for user-defined macros.
- App version number saved in preferences.
- Split marking for BSL galvos.
- Double-click in node edit converts to a path.
- Warning when importing a galvo or DSP device with a Core license.
- Selected items and progress bars keep part of their original blue color instead of turning white in the light theme and black in the dark theme.
- The New Device Wizard notes that comms settings only need editing when resolving problems.
- Consistent "G-code" in messages and the UI, and "Error:" capitalized in the Console.
- Warning to remove the probe and wires before starting a job after the probe wizard has been run.
- New "Not Responding" comm status, so error messages distinguish between disconnected and not responding.
- Clearer License dialog with links to help pages.
- Faster bitmap loads for large projects.
- ESC key closes the measure tool.
- Guide lines excluded from thumbnail bounds.

What's New

- Ruida UDP file retransmit support.
- Better multi-mesh support for 3D import and depth map conversion.
- New Device Wizard and Device Settings include Z axis height setup for galvo controllers.
- Single Accelaser device with a model selection page.
- Jog widget and camera layout fixes.
- Separate EzCad2 and EzCad3 drivers.
- Galvo Z control is back, and it beat Avengers: Doomsday to release
- Windows ARM EzCad drivers included.
- AI Escape rate decreased 42%
- Updated FTDI drivers and libraries, including the Windows ARM driver.
- Auto calibration pattern collection procedure upgrades.
- Accelaser planner library update and increased wobble precision.
- Variable axis letter names for controllers that support them.
- Frame preview positions every configured axis and shows grid tiles where they will actually be marked.
- The laser status shows Moving while external axes reposition during a galvo job, and Stop during a split marking axis move takes effect immediately.
- Split marking elapsed time includes the homing before a job and the return to the start position after it.
- The Galvo Move window keeps its Z position across a Stop that did not interrupt a move, and no longer waits on a Home request the axis could not accept.
- Split Marking Setup shows the fixed position target for the Z axis in the same Home is Zero convention as the Move window.
- The EzCad3 Z jog stays within the configured Z axis height once the axis is homed.
- Registration / Camera widget layout updates.
- Alignment intro page removed.
- Accelaser planner library loads on demand instead of at startup.
- The Alignment Wizard sets the cut origin to Absolute and restores the previous setting on exit.
- Focus on the calibrate button in the New Camera Wizard.
- Improved camera dialog tooltips.
- Device Settings and External Axis Setup refuse to close on an unusable axis value, such as a minimum speed above the maximum, instead of silently clamping it.
- A new split marking axis row on EzCad3 starts from the imported steps, distance, homing speed and timeout of the chosen motor.
- Preset cameras now also go through all setup steps.
- Alignment Wizard verification update.
- Warning when the wrong AprilTags are detected.
- Option to fill alignment pattern features.

What's New

- Cameras and Registration docks merged into one.
- Accelaser camera presets.
- Focus Test and Interval Test are no longer shown for Accelaser devices.
- Unsent crash reports can be deleted.
- Entering a Z axis height in the New Device Wizard turns on Z axis control.
- The Galvo Move window only appears when Z axis control is enabled for the device.
- Camera Calibration QR code points at the docs.
- Accelaser REST API features for framing and LiDAR scan cancelling.
- Rotate view buttons in camera calibration.
- Snap dragging can take X from one shape and Y from another.

Miscellaneous

- Pro mode trial with a valid Core license.
- Valid URLs can be pasted into the Web Tool browser.
- 3D Modelling temporarily stuck in the 8th Dimension.Â Buckaroo Banzai has been dispatched to retrieve it.
- Legal warning that must be accepted before SVGNest can be used.

User Interface Updates

- Improved visibility of toolbar drag handles and overflow arrows.
- Padding between items in the main toolbar.
- Buttons in the 3D import and depth map settings dialog show more clearly when they have focus.
- Split marking menu text and window titles no longer say one axis or single axis.
- Application font size setting now applies to tooltip text.
- The steps pair is labelled Steps and Distance in Device Settings and External Axis Setup, matching the imported values.
- Cut settings dialog now includes Epilog-specific settings and device limits.
- mDNS search for Epilog devices in the device discovery dialog.
- Minimum screen size check tables updated to account for UI updates and fixes when High DPI Scaling is disabled.
- New startup check and prompt to enable the "High DPI Scaling" setting if a high DPI display is connected.
- Title case is consistent across all right-click menu entries.
- Smoother expand animations in the Camera / Overlays widget.
- 'Paste in Place' in the right-click menu.