Laser Users Guide

v2023Q3.2

Laser Users Guide

1. Laser Safety

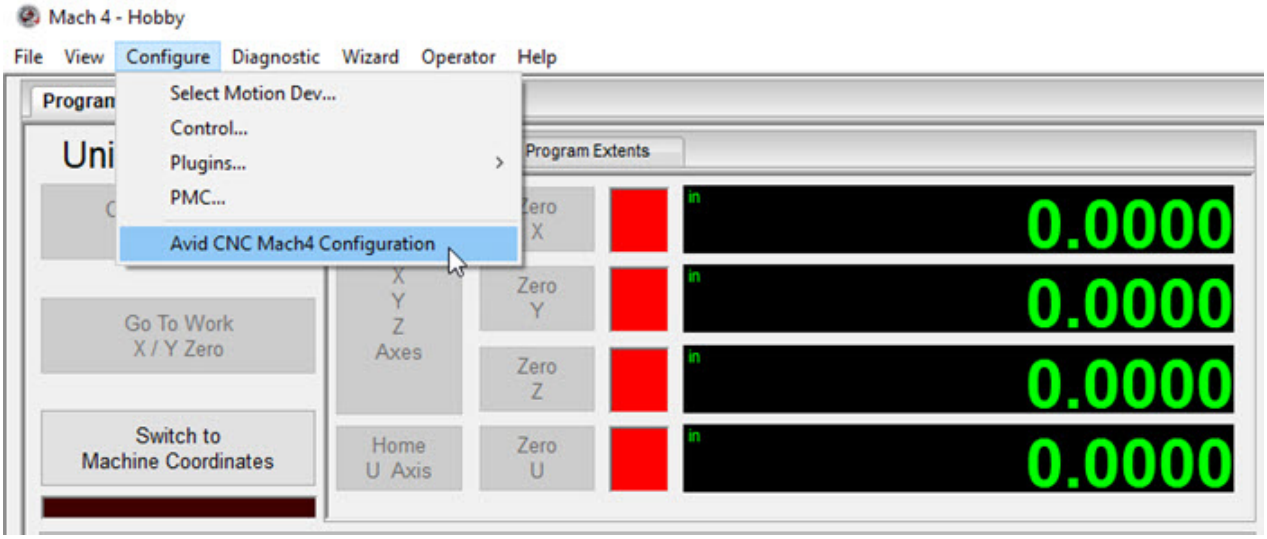
> ⚠ **Safety Measures**
>
> Laser radiation can cause damage to the eyes, skin, and other tissues, and improper usage can also result in fire or dangerous fumes. Follow the guidelines below when using your laser system to maintain a safe environment for yourself and others.

1. **Eye Protection**
   Anyone within viewing range of the laser during operation should wear the provided laser safety glasses or equivalent to protect against diffusely back-scattered laser light. The provided glasses provide a protection level of OD 7+ within the wavelength range of 190 to 540 nm (deep ultraviolet to green) including violet and blue light.
   - Keep the laser mounted on the machine and pointing down. Never place your eyes in the path of the laser beam.
2. **Flammability and Fumes**
   Lasers operate by vaporizing materials with highly focused light that generates intense localized heating. They are capable of igniting flammable materials and creating potentially dangerous fumes. Make sure to adhere to the following guidelines for safe operation of your laser:
   - Never use a laser on materials which can create poisonous gas when cut or engraved. In particular, PVC releases poisonous chlorine gas when heated by a laser. For a list of materials your laser is intended to cut and engrave, reference the laser manufacturer's website: **Opt Lasers**
   - Make sure to operate your laser in a well-ventilated area. A localized exhaust or downdraft system is advisable for materials with particularly noxious fumes.
   - Have an appropriate fire extinguisher immediately accessible in the event of fire.

2. Software Setup

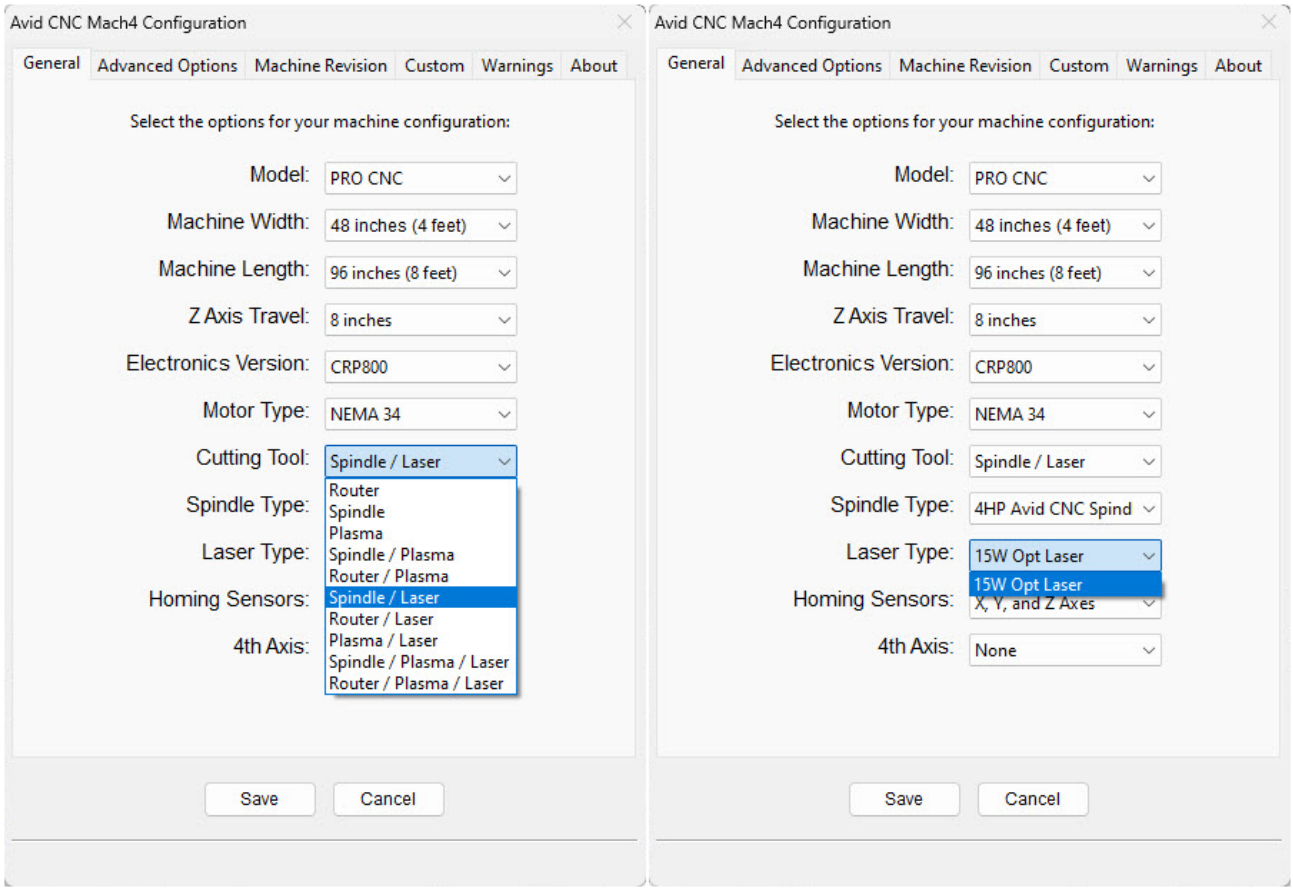
> **Software**
>
> Make sure you are using the latest version of Mach4 available from Avid (2.5.0 or greater). Please Contact Us if you purchased an Avid CNC laser system and have not yet received a software update.

1. Once installed, open the **"Avid CNC Mach4 Configuration"** window located in the Configure menu.



Selecting Configuration Window

2. Under **Cutting Tool**, select the type of cutting tools your machine has. If your machine has a spindle, select "Spindle / Laser". This will expose the **Laser Type** selection box where you can select the model of laser you have equipped on your system.



Selecting Cutting Tool & Laser Type

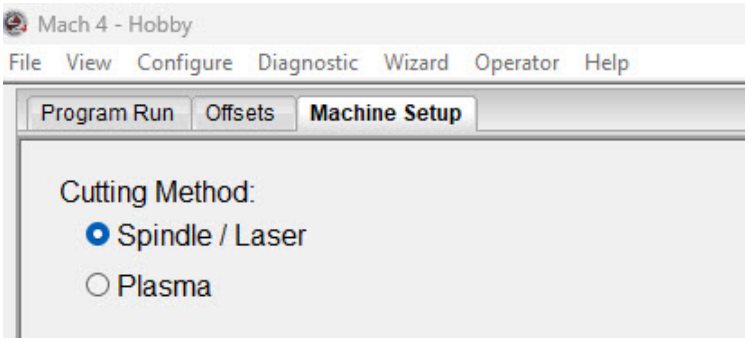
3. After saving your selections, the Laser tab should now be available in the main File Ops and Jogging tab group.

Laser Tabs Visible

Accessing the Laser Features

For Spindle/Laser and Router/Laser machines:
In these configurations, the Laser controls will always be available.

For Plasma/Laser, Spindle/Plasma/Laser, and Router/Plasma/Laser machines:

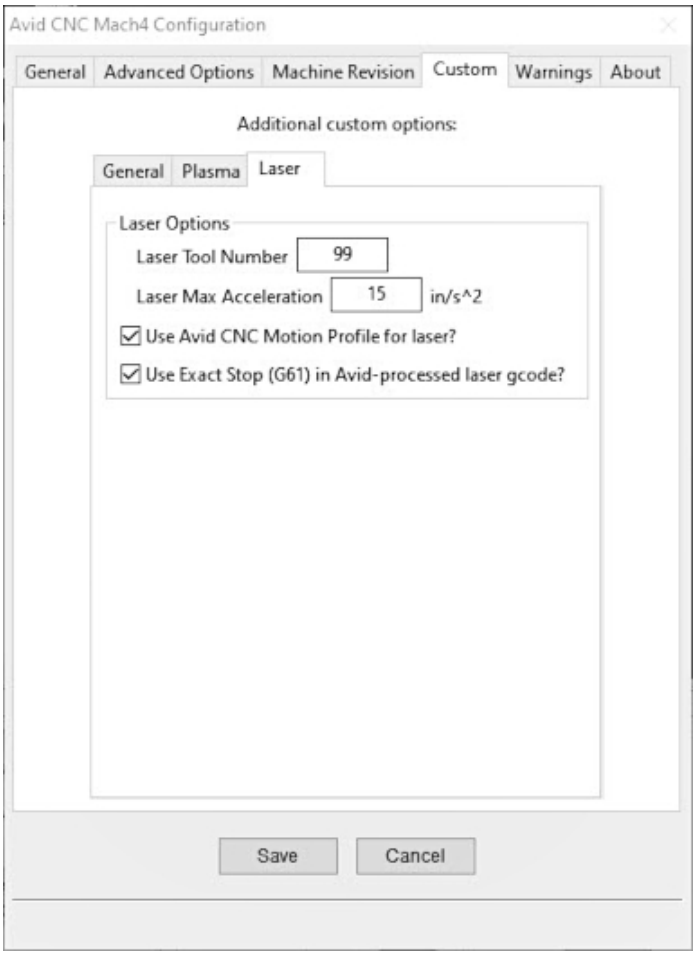


Choosing Cutting Method

In Plasma-enabled configurations the Laser controls are only available when a Laser cutting method is chosen on the **Machine Setup** tab. It is not possible to mix Plasma and Laser cutting without switching cutting methods due to PWM output limitations.

Additional Laser Configuration Settings

There are additional settings on the **Custom** tab to further customize the behavior of the laser system.



Customizing Laser Settings

1. **Laser Tool Number** - This tool number is used by Mach4 to determine when to deploy the laser from g-code tool changes. Deploying the laser will automatically change the current tool to this number. The laser tool number used in Mach4 and in CAM laser toolpaths should always match.
2. **Laser Max Acceleration** - If the Avid CNC Motion Profile is in use for laser, this acceleration and corresponding CV (constant velocity) parameters will be applied to X and Y motion whenever the laser is deployed. It is lower than the default value for routing to reduce vibration and improve the quality of laser engraving. For smoother motion, decrease the acceleration. For faster motion, increase the acceleration. This acceleration will also affect jogging motion while the laser is deployed.
3. **Use Avid CNC Motion Profile for laser** - When checked, the Laser Max Acceleration and corresponding CV parameters will be applied whenever the laser is deployed.
4. **Use Exact Stop (G61) in Avid-processed laser g-code** - When checked, vector g-code processed through the screen button will have a G61 applied to the laser toolpath(s). This will put Mach4 into Exact Stop mode for sharper corners. The downside is slower motion due to the greater acceleration required. This is recommended for detailed engraving but may not be desired for larger or less intricate vectors. For more information see **Vector G-Code Processing**.

3. Arming, Disarming, and Test Firing

The laser is equipped with a number of safeguards (both hardware and software) to prevent it from firing unexpectedly.

3.1 Hardware Arming and Disarming

The following instructions describe the sequence required to arm the laser for operation. The arming sequence is followed by a short diagram that can be used as a guide when enabling the laser for use.

Always wear protective eyewear before initiating the arming sequence.

1. Turn on the laser control box. The power for the laser controller is supplied from the AC outlet to the laser control box.

Laser Control Box - Powered On

2. Turn the key on the laser control module to enable power to the laser. One light should illuminate on the laser control module once the system has initialized.

Laser Control Module - Disabled

Laser Control Module - Enabled

3. Press the red button on the laser control module to arm the laser. Two lights will be illuminated on the laser control module when the system is armed, and the cooling fan on the laser will turn on.

Laser Control Module - Armed

4. To disarm the laser, press (but do not hold) the red button again.

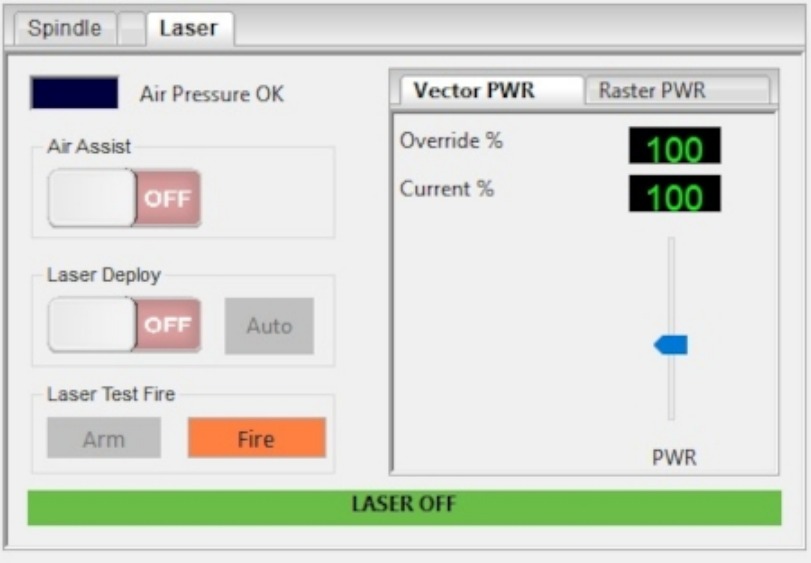
3.2 Initial Test Fire

After executing the steps to arm the laser hardware, the laser can be test fired. There are two methods for this, a hardware test fire which can be executed independently from Mach 4, and a software test fire using Mach 4 to check for proper software configuration.

In either case, make sure the laser is positioned over a sacrificial surface. If the laser nozzle is sufficiently high above the material (more than 2" / 50mm) it should be out of focus enough to avoid burning the material.

1. Hardware Test Fire
    a. To do a hardware test fire, with the system armed, press and hold the red button. This will flash the laser at a low intensity.
    b. Release the red button to stop the hardware test fire.

2. Software Test Fire



Laser Tab Detail

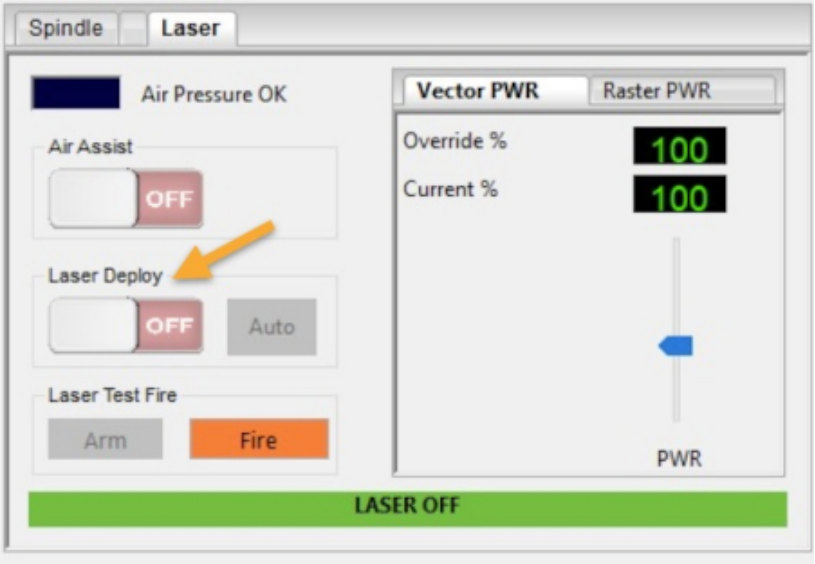
a. From the Laser Cutting Tool tab in Mach4, click the **Arm** button in the Laser Test Fire box. You should see a note in the History bar that states "Laser Test Armed".

b. Click the **Fire** button to fire the laser. The laser will fire for as long as the button is clicked.

c. When finished, click the **Arm** button again to disarm the laser in software. **Note:** The laser hardware will still be armed until the red button on the laser control module is pressed.

4. Deployment and XY Offset

The laser system is designed to work alongside your cutting tool. The laser is deployed via a pneumatic actuator that extends the laser head below your primary cutting tool. This allows you to use the laser without removing tooling from your spindle. When you are finished with the laser, it should be retracted to keep it safely above your primary cutting tool.



Laser Tab - Retracted

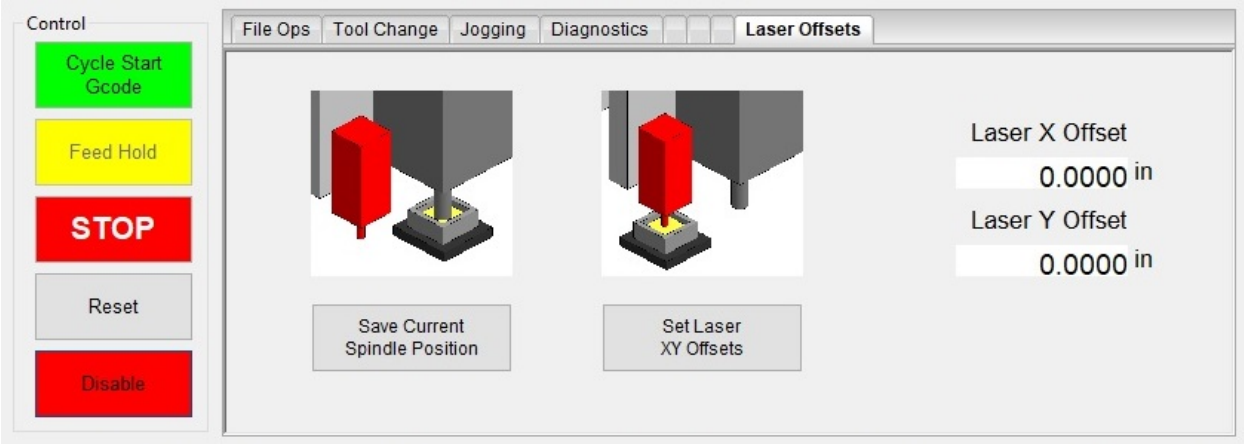
When the laser is deployed, the software will apply an XY offset that aligns the laser head to the origin of your workpiece, rather than your primary cutting tool. When the laser is retracted, the XY offset is removed, shifting the coordinate system back to correspond to your primary tool. The calibration of this XY offset is detailed below.

5. Calibration of Laser XY Offset

Before using the laser for cutting or engraving, it is important to calibrate the XY offset between the laser head and the spindle. The calibration procedure uses the Avid Auto Z Touch Plate to reference both the spindle and the laser, and then calculates the XY offset between them. This process is detailed in the steps below.

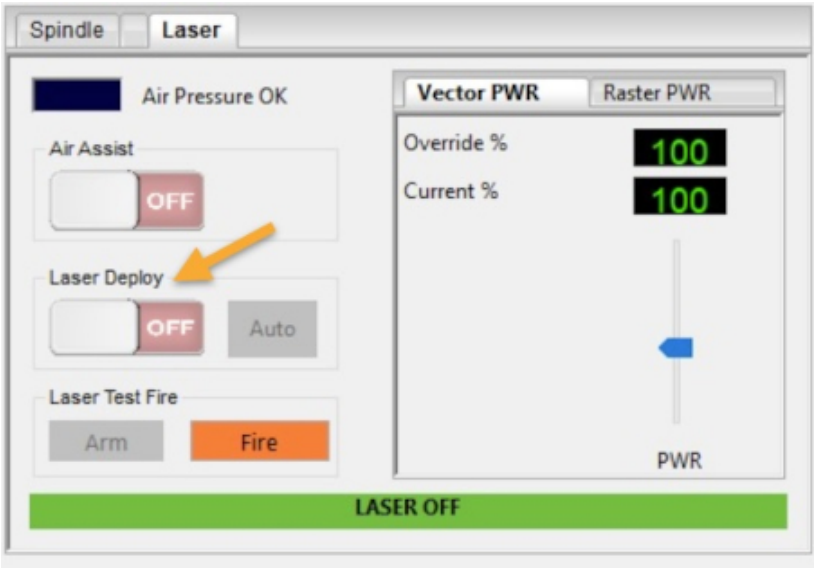
1. Use the touch plate to locate the material corner with the spindle

    a. Select the Laser Offsets tab.



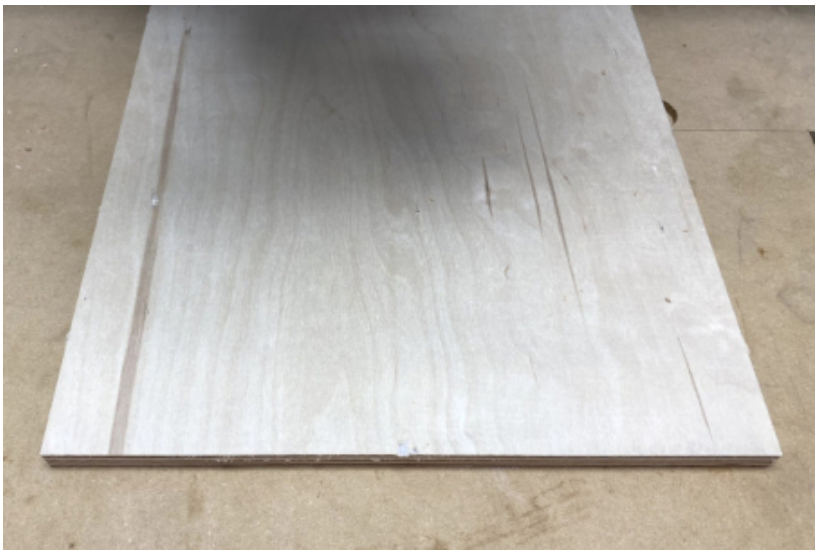
Laser Offsets Tab Detail

    b. Make sure the laser is retracted (Laser Deploy button is **OFF**).



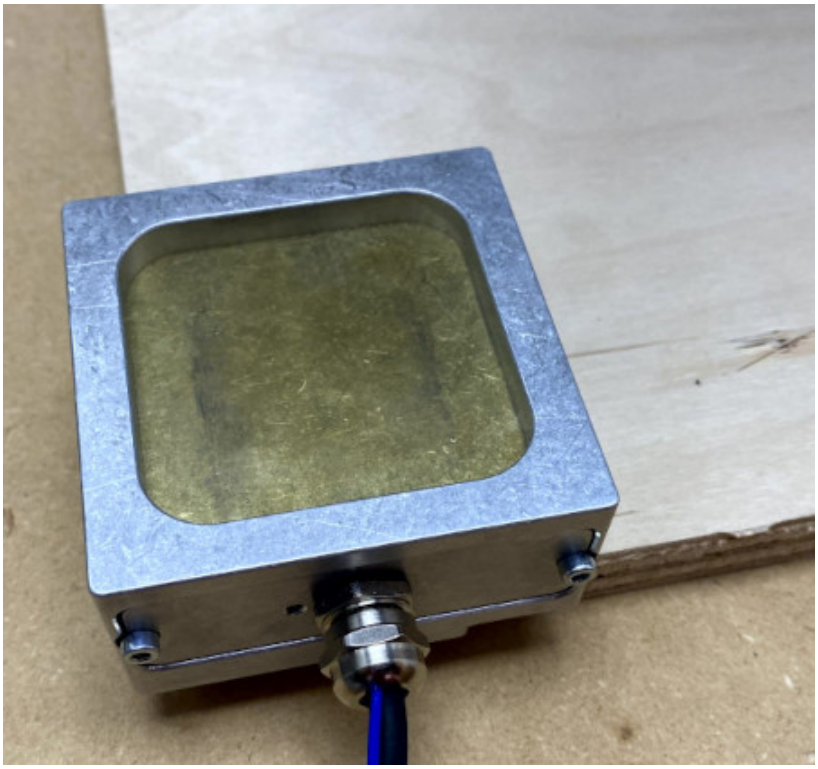
Laser Deploy - Off

c. Mount a rectangular piece of material to your machine so it will not move. Clamps or double stick tape are fine for this. The size and shape of the material are not important - this is simply to provide a fixed reference point for calibration. The only positional requirement is that the touch plate must be reachable by both the spindle and the laser.



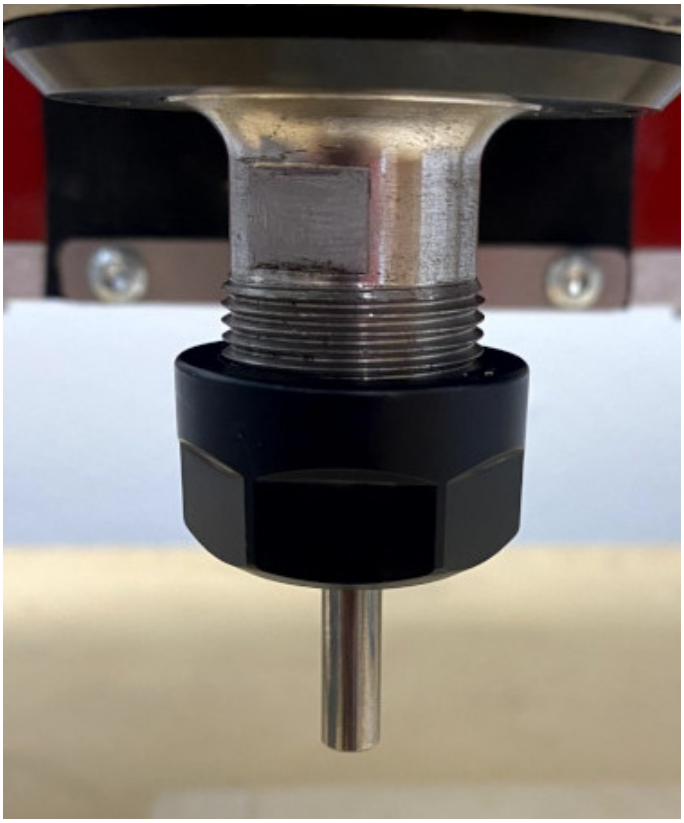
Test Material Mounted

d. Place the touch plate on a corner of the material, and ensure the plate is referenced securely against the corner. The touch plate will be referenced by both the spindle and the laser, and should not move between calibration steps.



Touch Plate

e. Load a tool into the spindle if one is not loaded. Because this is for calibration, using a dowel pin or an upside-down tool is recommended to eliminate error from rotation of the flutes on the tool.



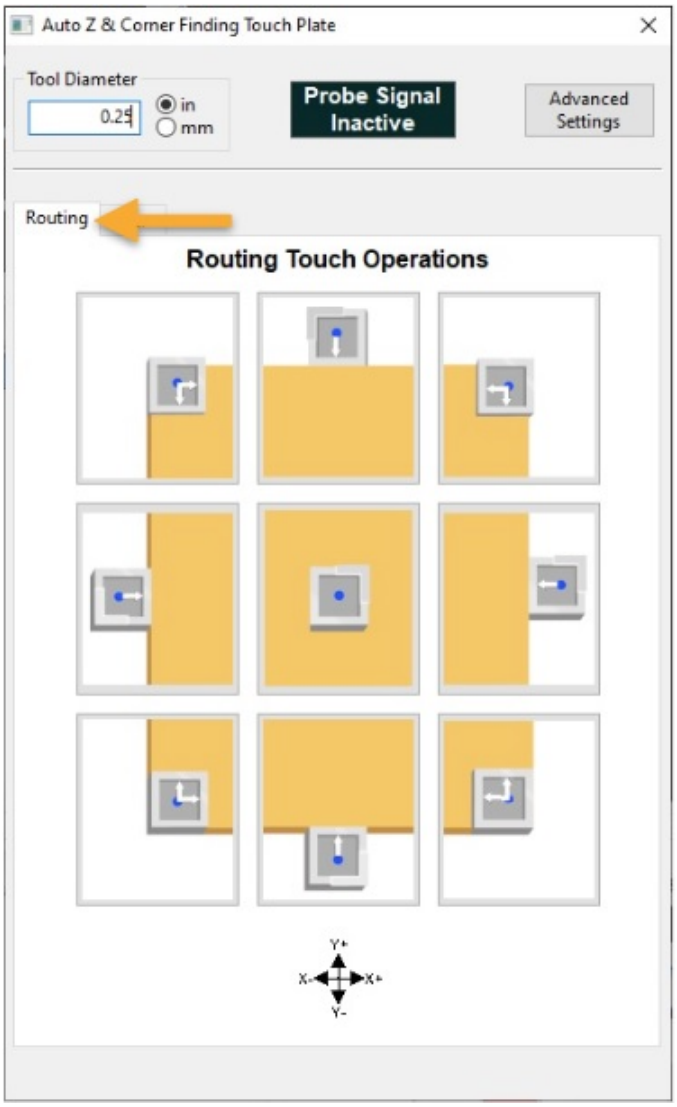
Tool Loaded

f. Jog the machine so that the spindle is roughly centered over the touch plate. Bring it down so the tool is within an inch (25mm) or so of the top of the touch plate and attach the magnet to the tool.

g. Open the Auto Z Touch Plate menu.

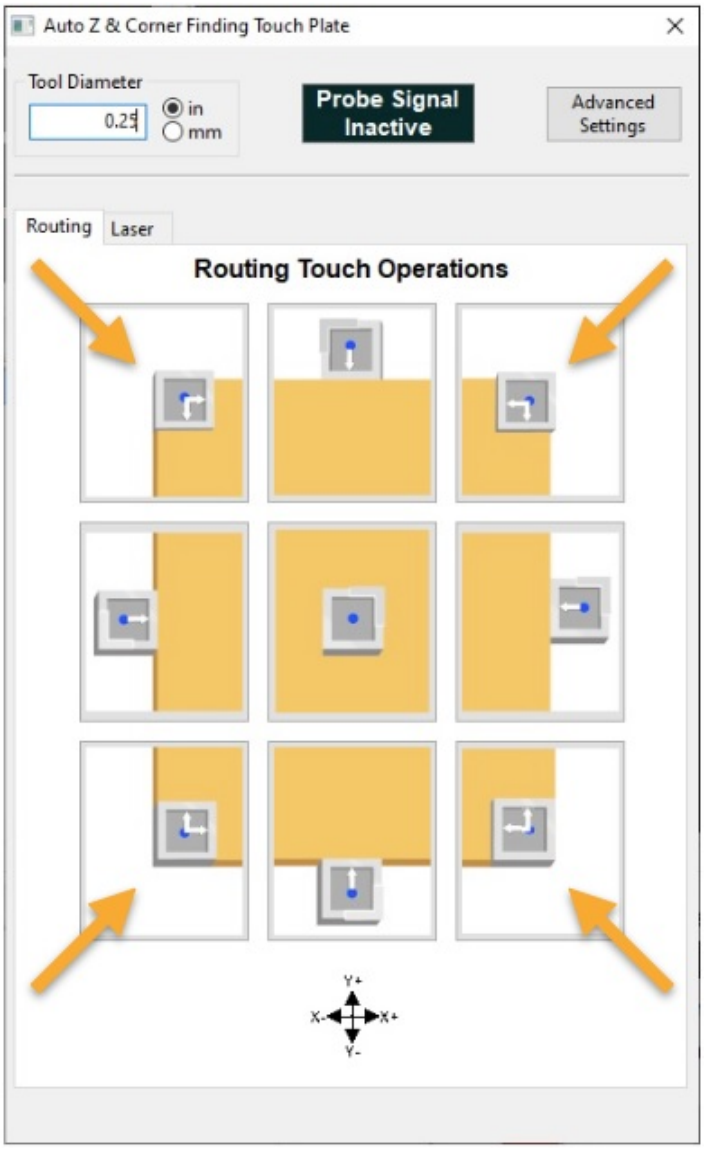
Auto Z Touch Plate Button

h. Select the Routing tab and enter the diameter of the tool installed in your spindle.



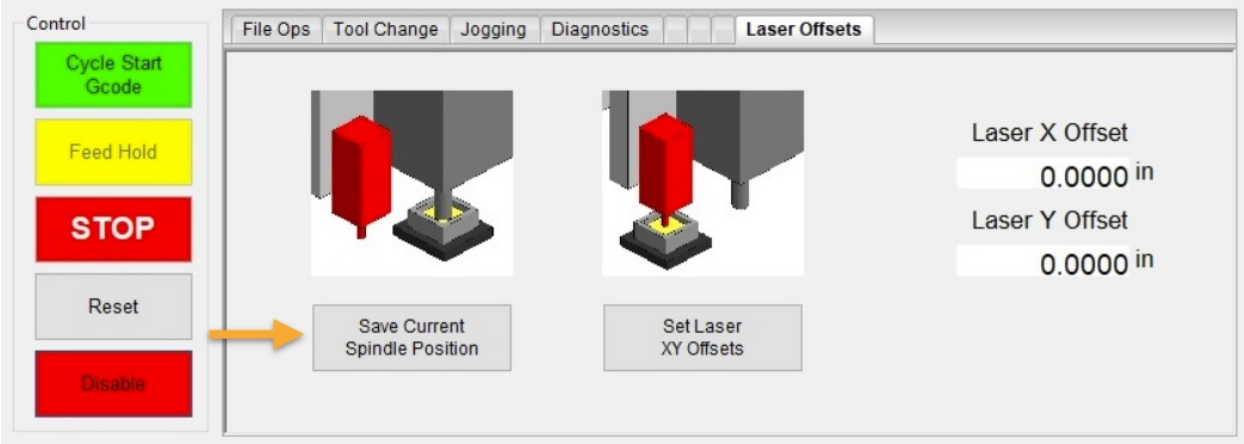
Touch Plate - Tool Diameter

i. Execute the appropriate corner touch plate routine for your material to locate the workpiece corner in X, Y and Z.



Touch Plate - Choose Material Corner

j. Once the spindle retracts, without jogging the machine or moving the touch plate, click the **Save Current Spindle Position** button.



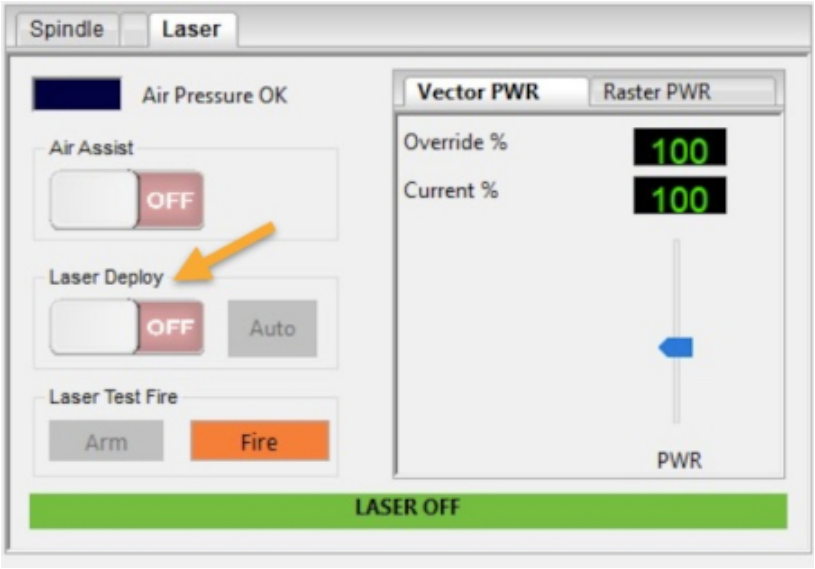
Laser Offsets Tab - Save Current Position

k. Hit **"Continue"** to store the spindle position values.

2. Use the touch plate to locate the material corner with the laser.

   a. Keep the touch plate in the same location on the material.

   b. Deploy the laser. The Z-axis will automatically raise up to its highest position before the actuator deploys the laser.

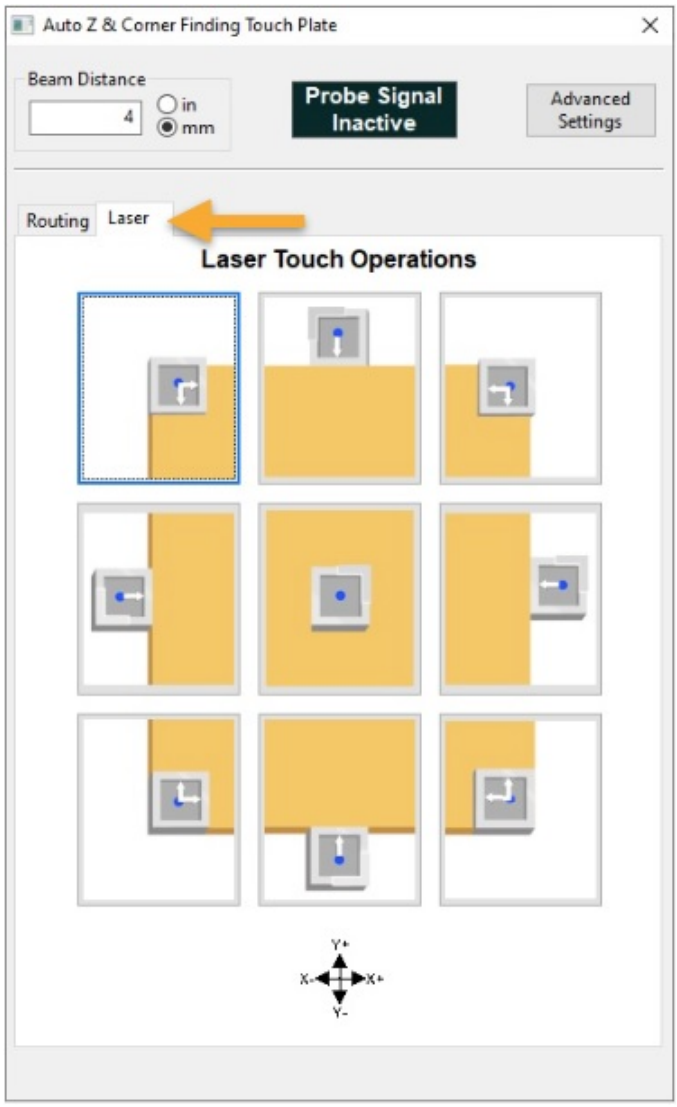


Laser Tab - Laser Deploy Button

   c. Jog the machine so that the laser nozzle is roughly centered over the touch plate. Bring it down within an inch (25mm) or so of the top of the touch plate and attach the magnet to the screw on the flange at the front of the laser.

Laser Above Touch Plate

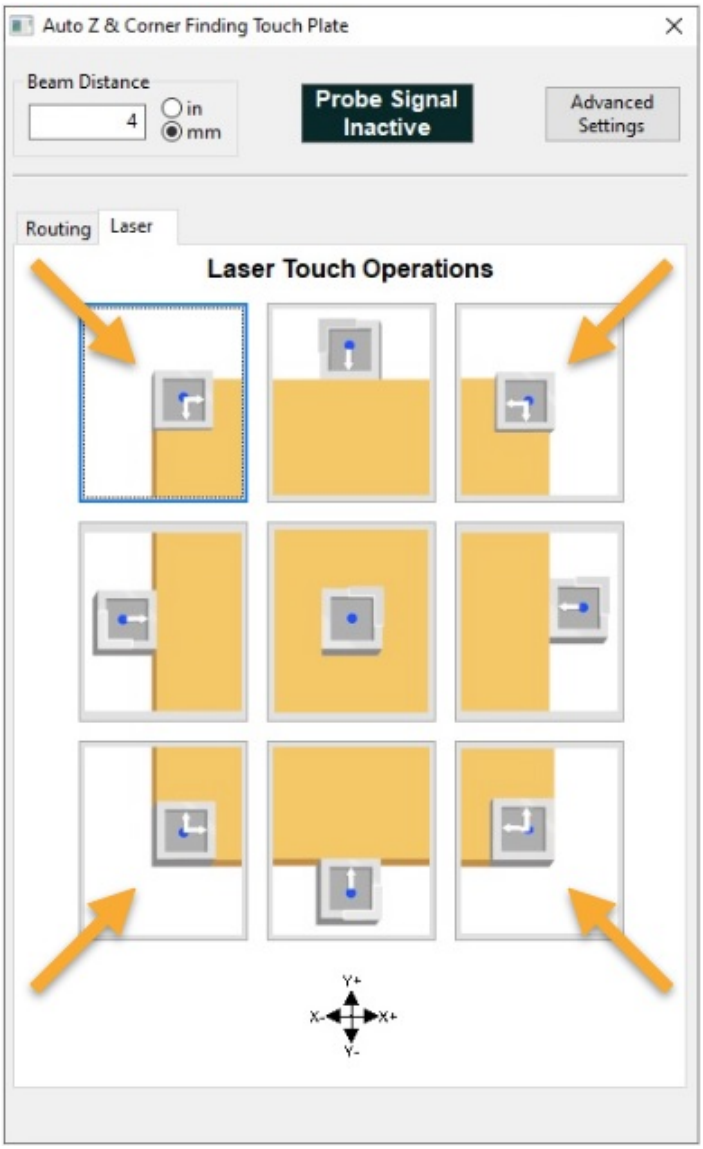
d. Open the Auto Z Touch Plate menu and select the Laser tab.



Touch Plate - Laser Tab

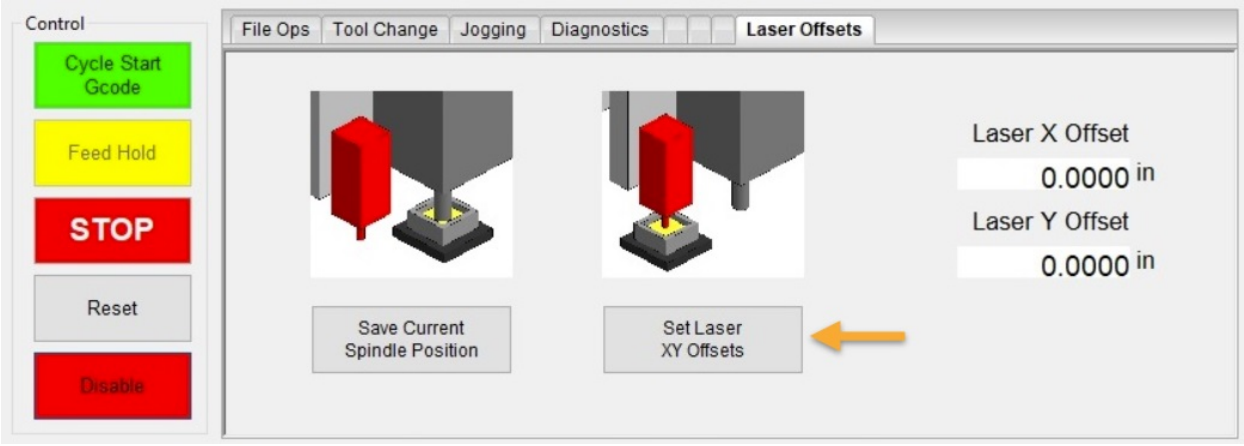
e. Unlike with the spindle, the diameter of the laser nozzle has been set by default (it can be edited from an advanced menu). Note the **"Beam Distance"** setting, which can be edited, but has also been preset to a standard focal height of 4mm. This offset normally doesn't need to be changed, but can be tweaked for special situations or fine tuning of beam width.

f. Run the same corner touch plate routine that you previously ran to locate the workpiece corner.



Touch Plate - Choose Material Corner

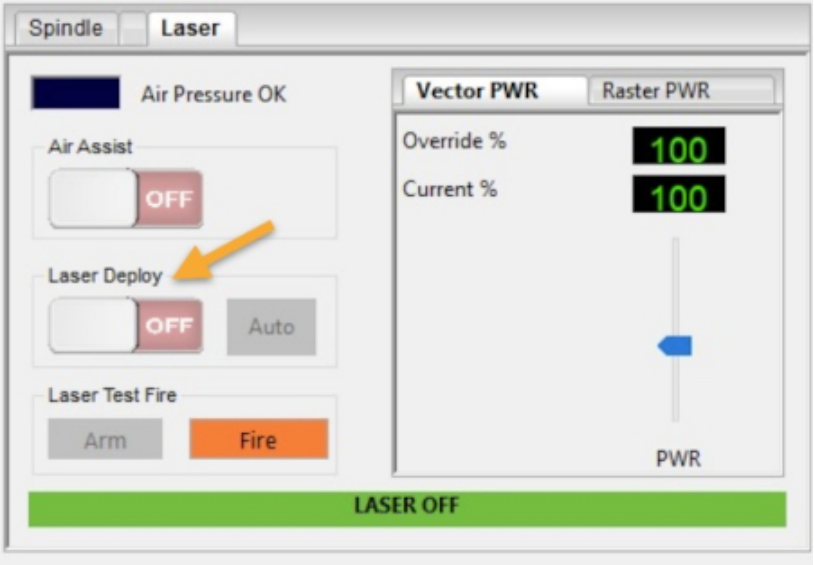
g. When the Z-axis retracts, without jogging the machine, click the **Set Laser XY Offsets** button.



Laser Offsets Tab - Set Laser Offsets

h. Hit **"Continue"** to store the values.

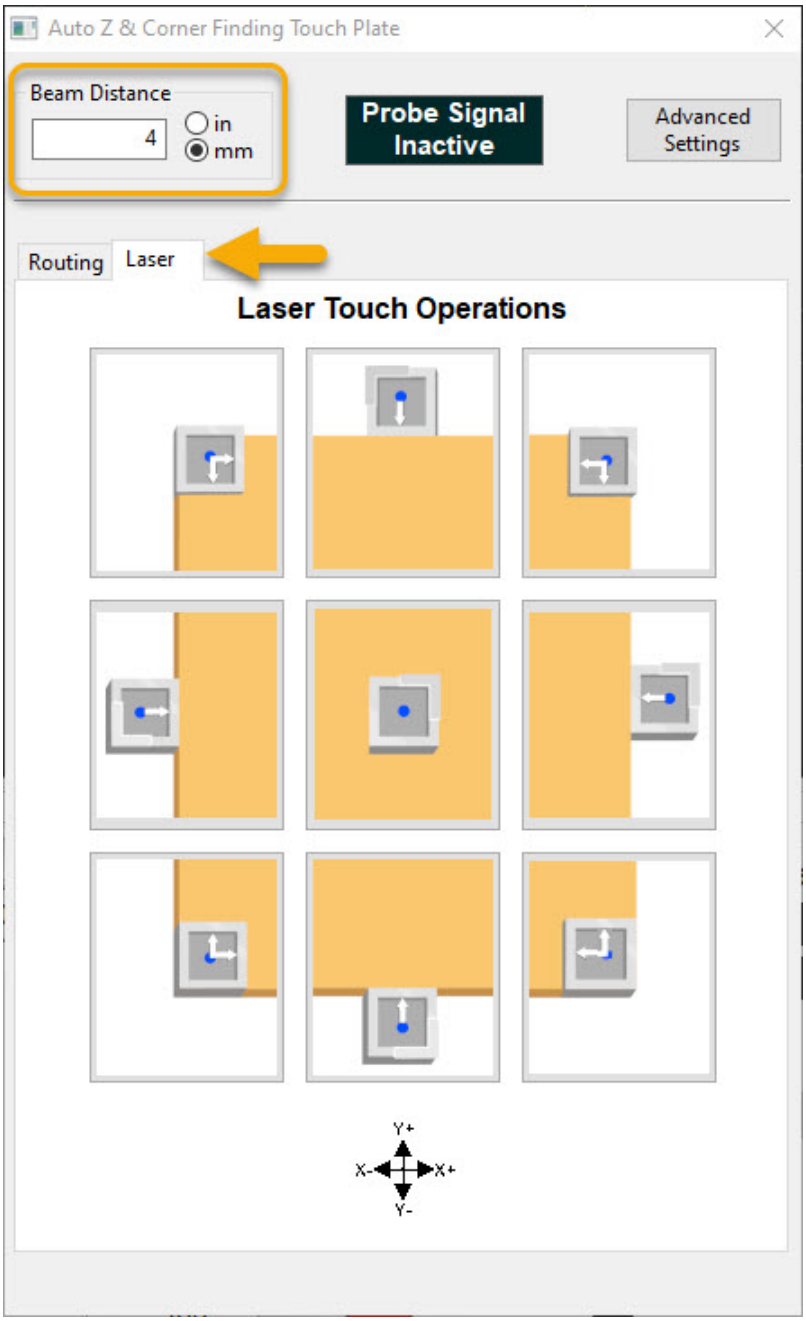
i. Retract the laser.



Laser Tab - Laser Deploy Button

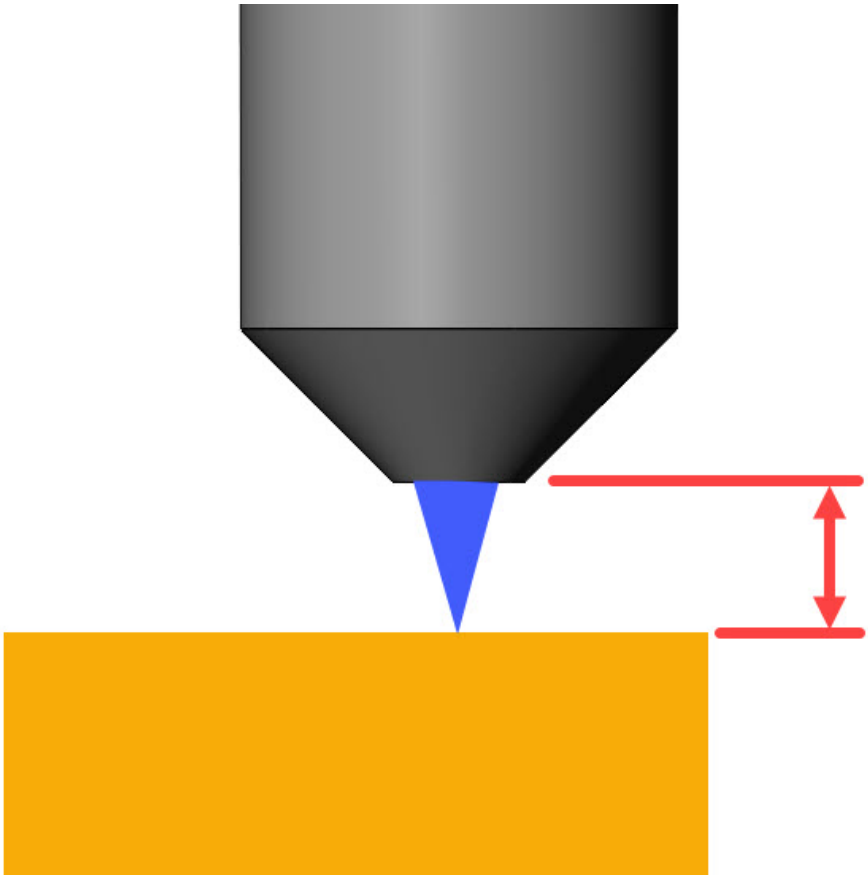
6. Auto Z Touch Plate

The Auto Z Touch Plate can be used with the laser to set work offsets in X, Y, Z. The laser should be deployed and the **Laser** tab should be used.



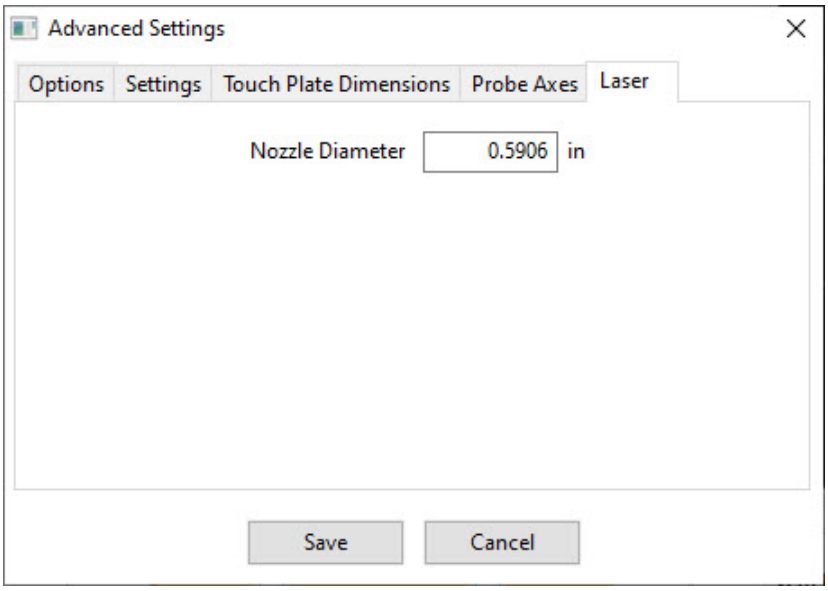
Touch Plate - Laser Tab

When the Auto Z Touch Plate is used to set work offsets with the Laser, the offsets are automatically shifted by the permanent XY calibration values. This ensures that whether the laser or primary routing tool is deployed, the cutting tool will end up in the correct location.



Beam Distance

The **Beam Distance** setting is preset at 4mm (0.1575") and can be modified. It sets a Z offset between the top of the workpiece and the tip of the laser nozzle so that the laser focal point is on the Z work zero plane.

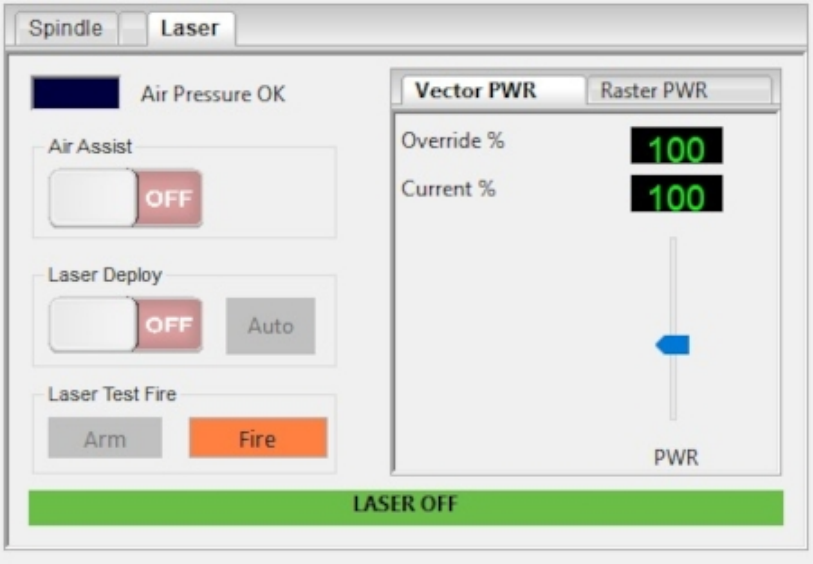


Touch Plate - Advanced Settings

The **Nozzle Diameter** is preset at 15mm (0.5906"). It can be edited in the Advanced Settings menu but this is not necessary in normal circumstances.

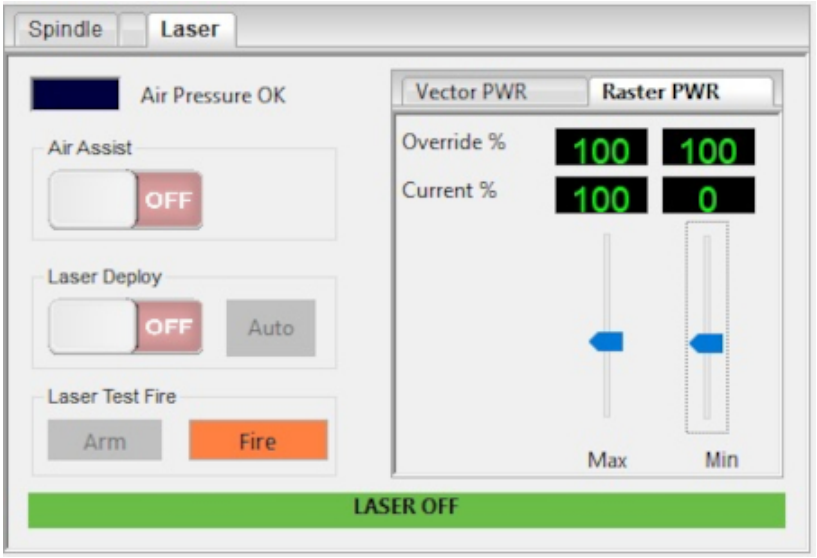
7. Menu Controls

This section details the options and indicators available from the Laser cutting tool tab. This tab is available in the same tab group as the Spindle on/off and speed settings, and provides an interface to important laser functions.



Laser Tab

1. **Air Pressure OK** - This signal is an indicator that sufficient air pressure has been provided to the laser control box to run the laser. If this indicator is not on, certain laser functions will not be available and will return errors.
2. **Air Assist** - This control allows you to turn on the air assist, which is a low pressure stream of air through the laser nozzle to help clear vaporized material and reduce burning or charring of material
3. **Laser Deploy** - This control will raise the z-axis up to its maximum height, then deploy the air cylinder actuator for the laser. It will also apply the XY offset established during the XY offset calibration process. Finally, this setting will temporarily update motion tuning CV (constant velocity) values for smoother motion when in laser mode.
    a. **Auto** - Turning Auto on will disable the manual deploy toggle so that the laser's deploy state will be controlled by tool changes in g-code. With Auto off the laser will NOT deploy during g-code tool changes.
4. **Vector Power Settings** - The settings in the Vector PWR tab apply to vector G-Code (generated from CAM software packages such as Vectric).
    a. **Override %** - This will modify the laser power in the G-Code, up to but not exceeding 100% power. For example, an override of 200% on a G-code power setting of 20% will result in 40% power being sent to the laser. Likewise, an override of 50% on a G-Code power setting of 20% will result in 10% power being sent to the laser. Because the actual laser value cannot be greater than 100%, an override of 200% on a G-Code power setting of 80% will max out at 100% laser power.
    b. **Current %** - This displays the actual commanded power level (0-100%) being sent to the laser. This is the combined value of the commanded power setting in G-code after being modified by the override %.

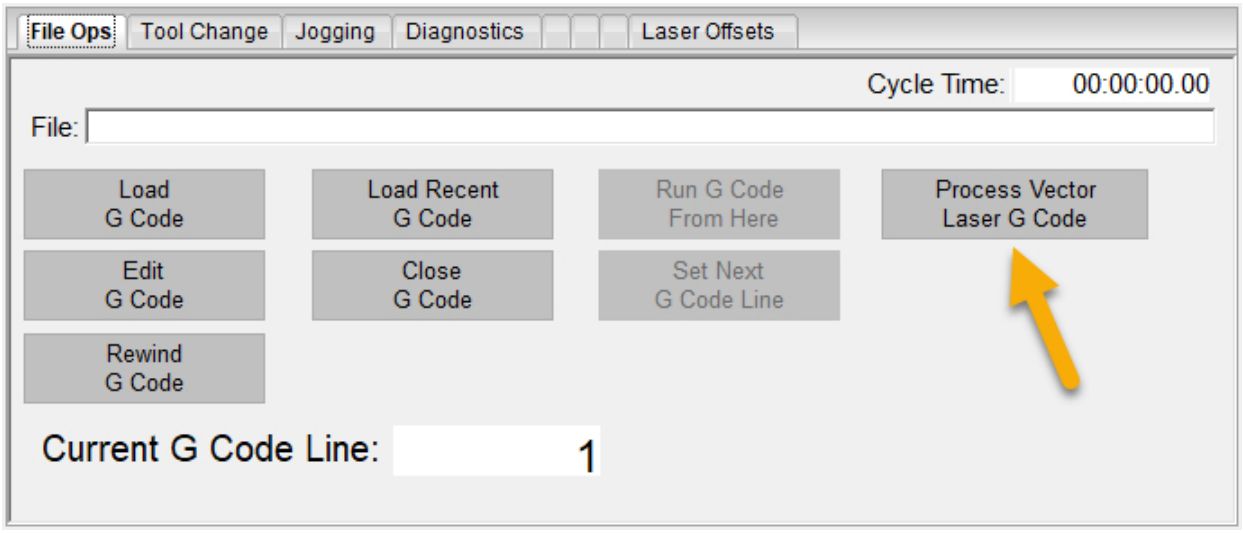


Laser Tab - Raster PWR

5. **Raster Power Settings** - The settings in the Raster PWR tab apply to G-Code generated from the raster image workflow provided by Warp9.
    a. **Override %, Min** - This will modify the laser power in the G-Code, and will scale the minimum (lightest possible) values in the G-Code.
    b. **Override %, Max** - This will modify the laser power in the G-Code, and will scale the maximum (darkest possible) values in the G-Code.
    c. **Current % Min** - This shows the lowest possible laser value for the raster after modification by the Override % Min.
    d. **Current % Min** - This shows the highest possible laser value for the raster after modification by the Override % Max.

For information on how to proceed with running cutting and engraving files, see **Machine Operation & Projects**

8. Vector G-code Processing



File Ops Tab Detail

The **Process Vector Laser G Code** button converts generic Mach4 routing G-code into laser-ready vector code. To use this function:

1. Use the normal **Load G Code** button to load the g-code file that you intend to process.
2. Once the file has loaded, click the **Process Vector Laser G Code** button.
3. The processor will quickly scan the target file for basic criteria. If there are no issues it will ask to continue.
4. While processing the file a progress bar will appear. Note that on very large files the processing may take a couple moments, but on most files it will complete quickly.
5. If the processing is successful, the new file will be loaded automatically. The new file and a log file will be saved in the C:\Mach4Hobby\GcodeFiles\AvidCNC folder.

File Requirements

In order to be processed successfully the original g-code needs to meet these criteria:

- Valid Mach4 g-code. When loading your original g-code ensure that there are no error messages. Processing invalid g-code for Vector Laser use may have unintended results.
- File size <10 MB.
- Must contain "T<lasertool>", with the tool number matching the Laser Tool Number configured in Mach4.
- Must contain "M6" to change tools to the laser tool.
- Must contain "S#", like S100, in the laser toolpath section. The number will be converted to a PWM percentage and must be 0-100.
- Must not use subprograms (M98) in the laser toolpath section.

The processor will handle all forms of acceptable Mach4-ready g-code. It is insensitive to case, spaces, and leading 0's. For example:
`G1x002 y4 F00200` is equivalent to `G1 X2 Y4 F200`

File Processing

The g-code processor identifies sections within a g-code file that use the laser tool, and prepares the code within that section for laser use. The laser section is defined by tool changes to/from the configured "laser" tool number.

The g-code processor does not modify the original file. It creates a new file, saves it in the C:\Mach4Hobby\GCodeFiles\AvidCNC folder, and loads the new code. There is also a log file saved for each processed file.

For example, if the Laser tool number is 99 then an example simplified file would look like below. The section identified by "**" would be converted for laser use. Other sections would be untouched.

```
T1 M6 (tool change to Tool 1)
S16000 M3
G0 X0 Y0
G1 Z-0.5 F100
G1 X1 Y2
G0 Z1
M5
T99 M6 (tool change to Tool 99)
** S95 M3       **
** G0 X5 Y5     **
** G1 Z0 F100   **
** G1 X6 Y7     **
** G0 Z1        **
** M5           **
T2 M6 (tool change to Tool 2)
S12000 M3
M5
M30
```

Within the laser section the changes that the processor will make are:

- Add preamble lines.

```
G49 (cancel tool height)
M2004 (Enable the Vector Laser)
M64 P8 (Turn on Air Assist)
M67 E1 Q0
G4 P2
G61*
```

***Note:** only if the option "Use Exact Stop (G61) in Avid-processed laser gcode?" is selected. See **Additional Configuration** section.

- Remove all S# commands and use the # to set laser power.
- Remove g-code that is not relevant to laser.
  - M3, M5, G43, H#, M7, M8
- Set laser power to 0 before all rapid moves and Z-only moves.
  - This prevents unwanted burns throughout the program.
- Set laser power to the "RPM" setting for that section before sections of feed (G1, G2, G3) moves.

The converted example laser section would be below. Added lines are identified by "**".

```
** G49 (cancel tool height)         **
** M2004 (Enable the Vector Laser)  **
** M64 P8 (Turn on Air Assist)      **
** M67 E1 Q0                        **
** G4 P2                            **
G61
** M67 E1 Q0                        **
** G4 P2                            **
G0 X5 Y5
** M67 E1 Q0                        **
** G4 P2                            **
G1 Z0 F100
** M67 E1 Q95                       **
** G4 P2                            **
G1 X6 Y7
** M67 E1 Q0                        **
** G4 P2                            **
G0 Z1
```