



# SheetCam Software Setup Guide

*v2023Q1.1*

# SheetCam Software Setup & Users Guide

This guide provides step-by-step instructions for installing SheetCam and setting up the Avid CNC Mach4 post processor and tool tables for a variety of Hypertherm torches. If you have already installed and licensed SheetCam, skip to **Section 3**.

The Pro-Tip video below also shows the SheetCam installation process.

## SheetCam Post and Tools Update (September 2021)

For Avid CNC SheetCam Post and Tools v1.7 and newer, refer to the written instructions below for the most current information.

## 1. Install SheetCam



The screenshot shows a web browser window with the URL [sheetcam.com/downloads](https://www.sheetcam.com/downloads). The page features the SheetCam logo and a navigation menu with links for Home, Features, Downloads, Plugins, Support, Purchase, Contact, Distributors, Scanything, and Forum. The main content area is titled "Downloads" and is divided into three sections:

- SheetCam TNG Stable version**: A note states that as SheetCam develops, more features are added, but this can introduce bugs. This version is not the most recent but the functions it does have are thoroughly tested. A note in red text says: "Note this download is limited to approximately 180 lines of generated code. You need to purchase a license to remove this limit." Instructions mention that if installing on a computer without Internet access, the installer may ask to install a Microsoft update: [ycredist\\_x86\\_vc2009.exe](#). Windows 2000 users may need to install [Windows 2000 update rollup 1](#) first. Links for "Download SheetCam TNG V6.0.30" and "Show recent changes" are provided.
- SheetCam TNG Development version**: A note states this is the latest version of SheetCam, with the newest features but may not be as reliable as the stable version. If bugs are found, they should be reported. A note in red text says: "Note this download is limited to approximately 180 lines of generated code. You need to purchase a license to remove this limit." Instructions mention that if installing on a computer without Internet access, the installer may ask to install a Microsoft update: [ycredist\\_x86\\_vc2009.exe](#). Windows 2000 users may need to install [Windows 2000 update rollup 1](#) first. Links for "Download SheetCam TNG Development V6.1.74" and "Show recent changes" are provided.
- SheetCam TNG for Linux**: Instructions state to download the file, right-click on it, select 'properties', go to the permissions tab, and make sure 'execute' is turned on. Then double-click on the file to install SheetCam. If that does not work, the user can simply unzip the file. The data folder in the zip is a tarball containing everything needed to run SheetCam. Execute `run-sheetcam` to run SheetCam. Links for "Download SheetCam setup V6.1.57 32 bit version" and "Download SheetCam setup V6.1.57 64 bit version" are provided.

- Navigate to <https://www.sheetcam.com/downloads>



Home Features Downloads Plugins Support

## SheetCam TNG Stable version

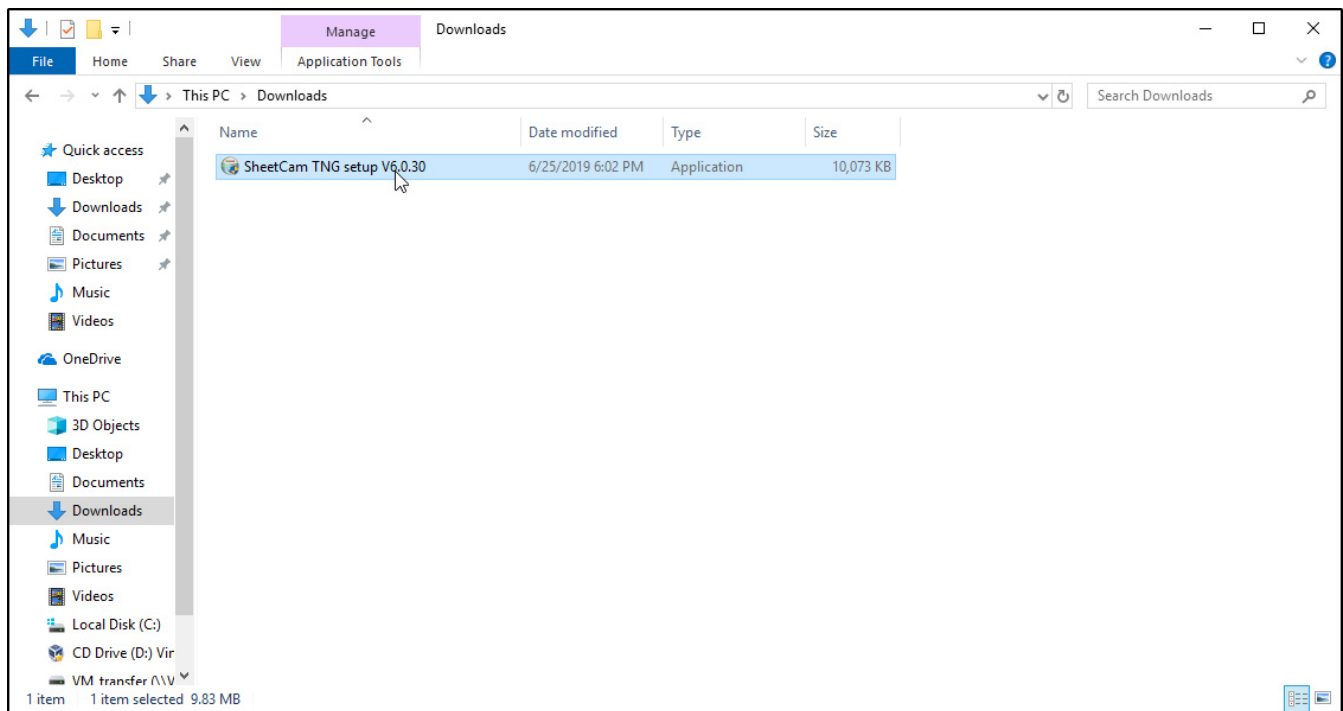
As SheetCam develops, more features are added. However in some cases this can be tested. **Note this download is limited to approximately 180 lines of generated code.**

If you are installing on a computer without Internet access, the installer may ask you to install [Windows 2000 update rollup 1](#) first.

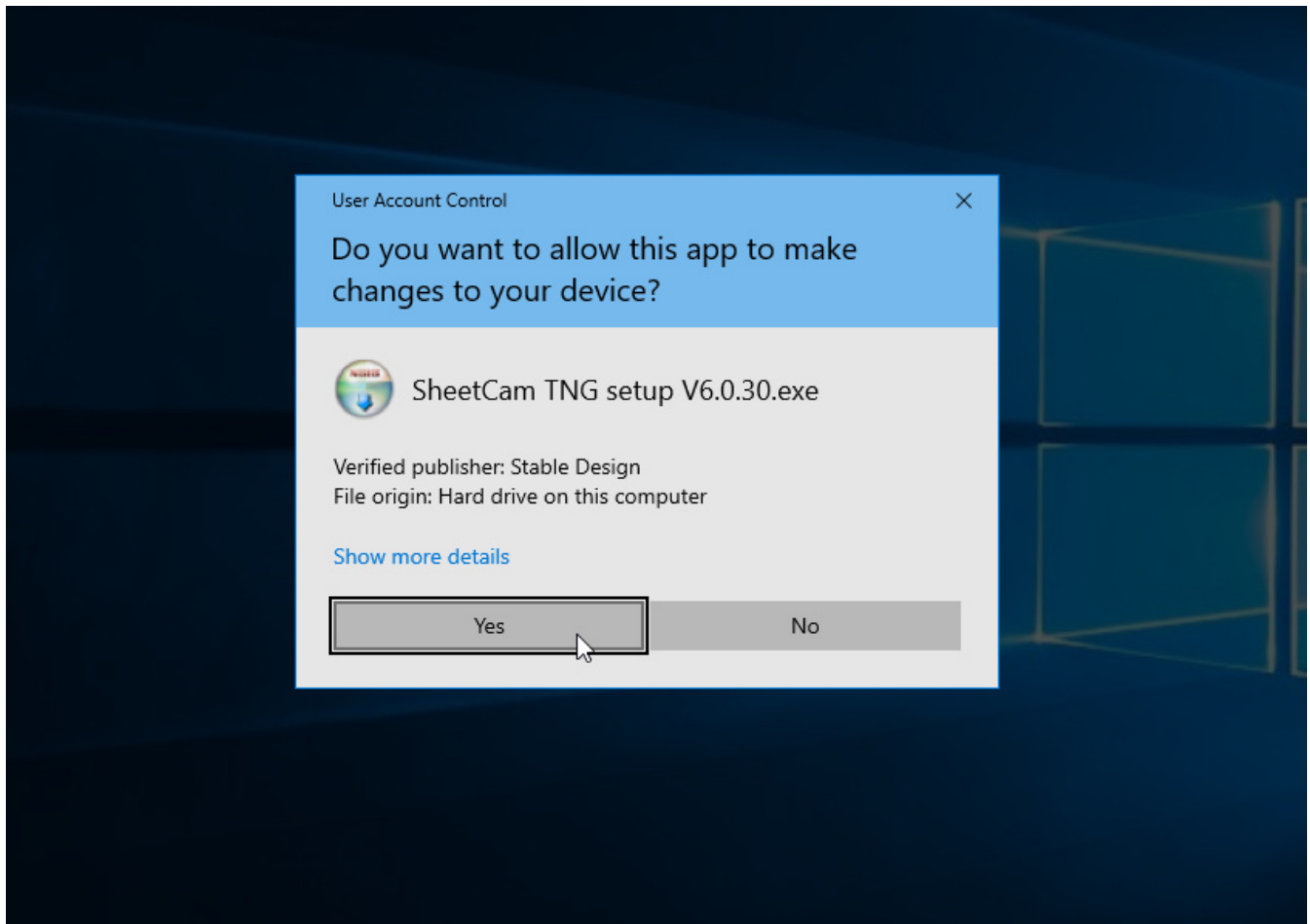
[Download SheetCam TNG V6.0.30](#)

[Show recent changes](#)

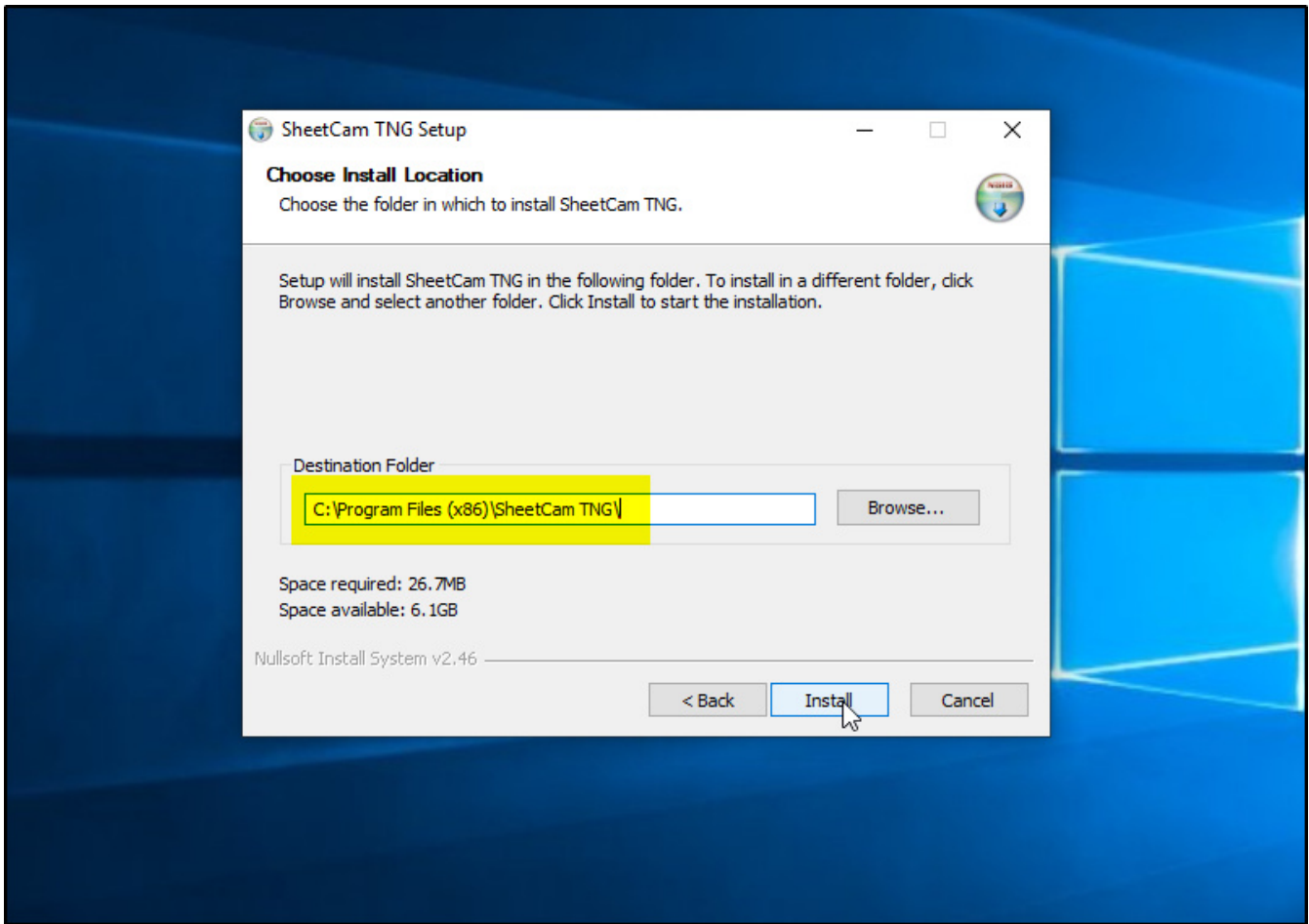
- It is recommended to download "SheetCam TNG Stable version".



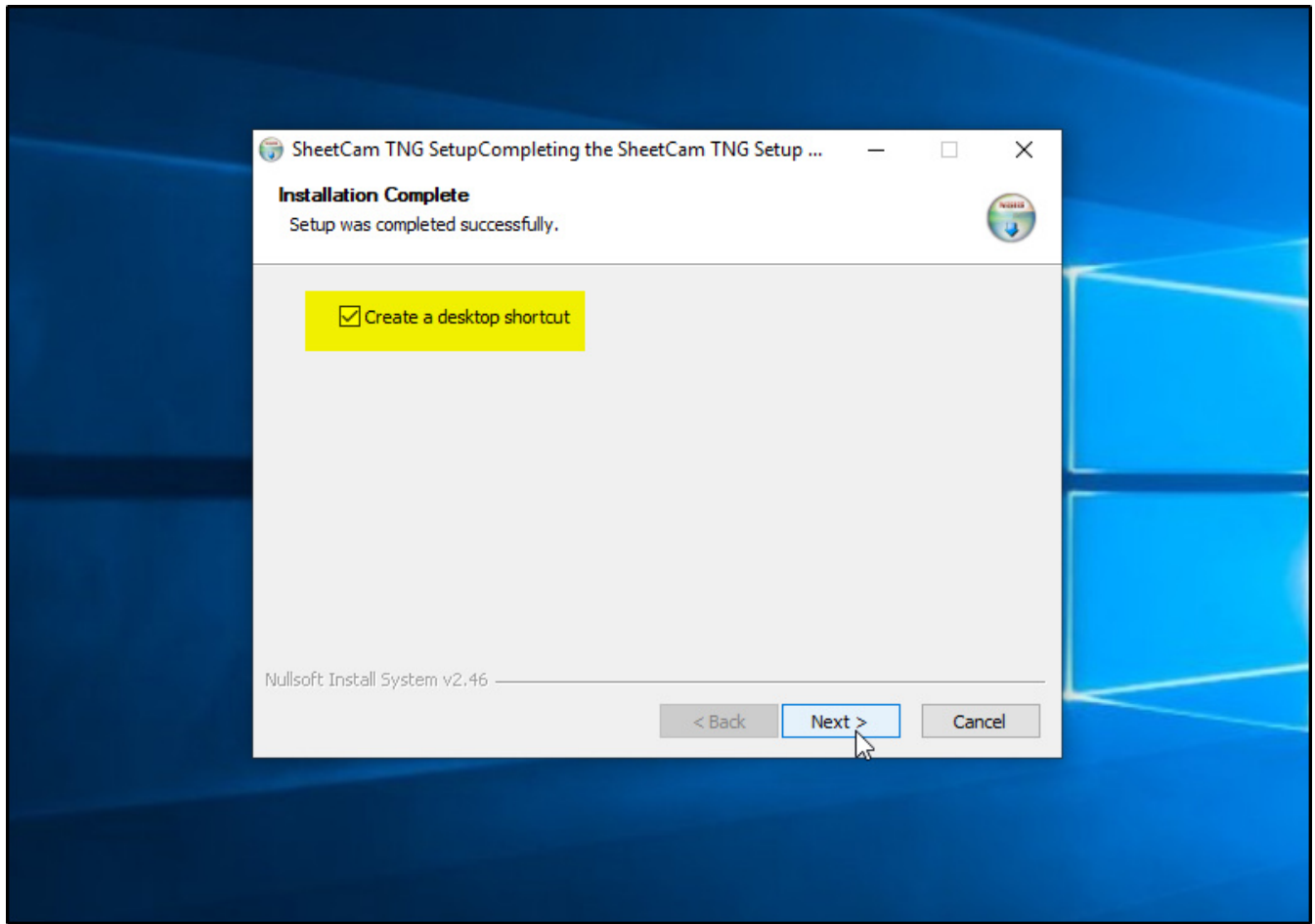
- Navigate to your downloads folder (or the directory you downloaded the installer to) and run the SheetCam TNG installer.



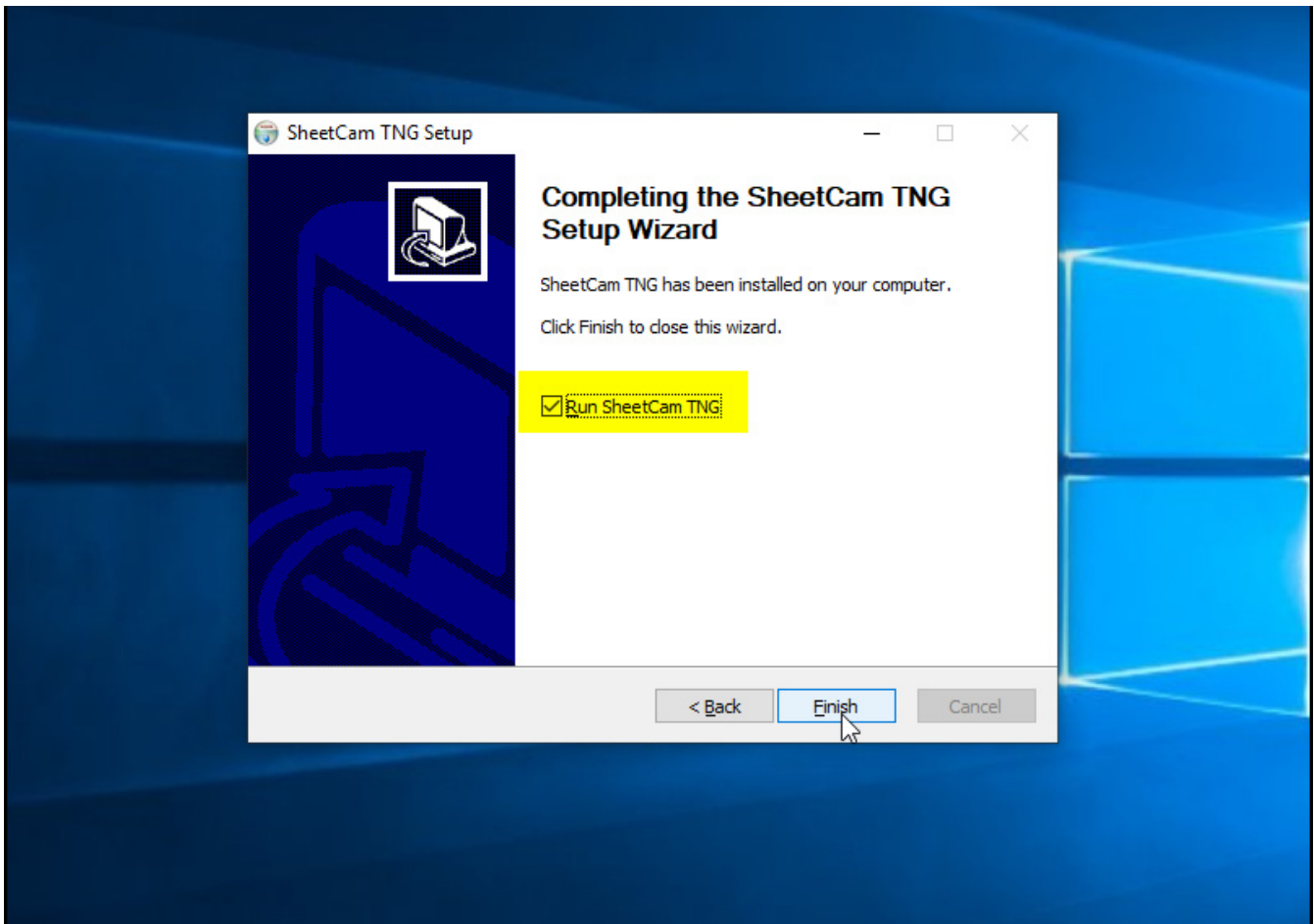
- Click "**Yes**" to begin the installation.



- When the installer asks for an install location, it is highly recommended to use the default location.



- When you get to the prompt shown above, select the **"Create a desktop shortcut"** option.

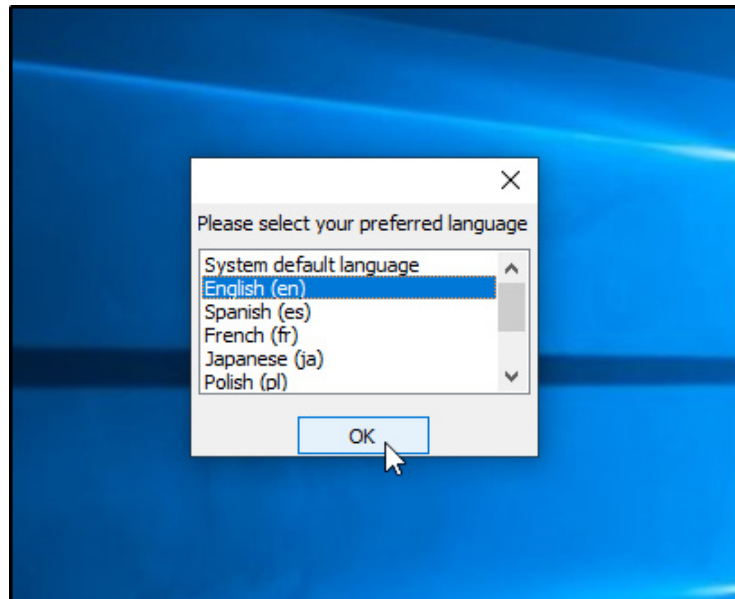


- Select the **"Run SheetCam TNG"** option and finish the SheetCam install.
- Continue with the setup and license procedure in the next section.

## 2. Install License File

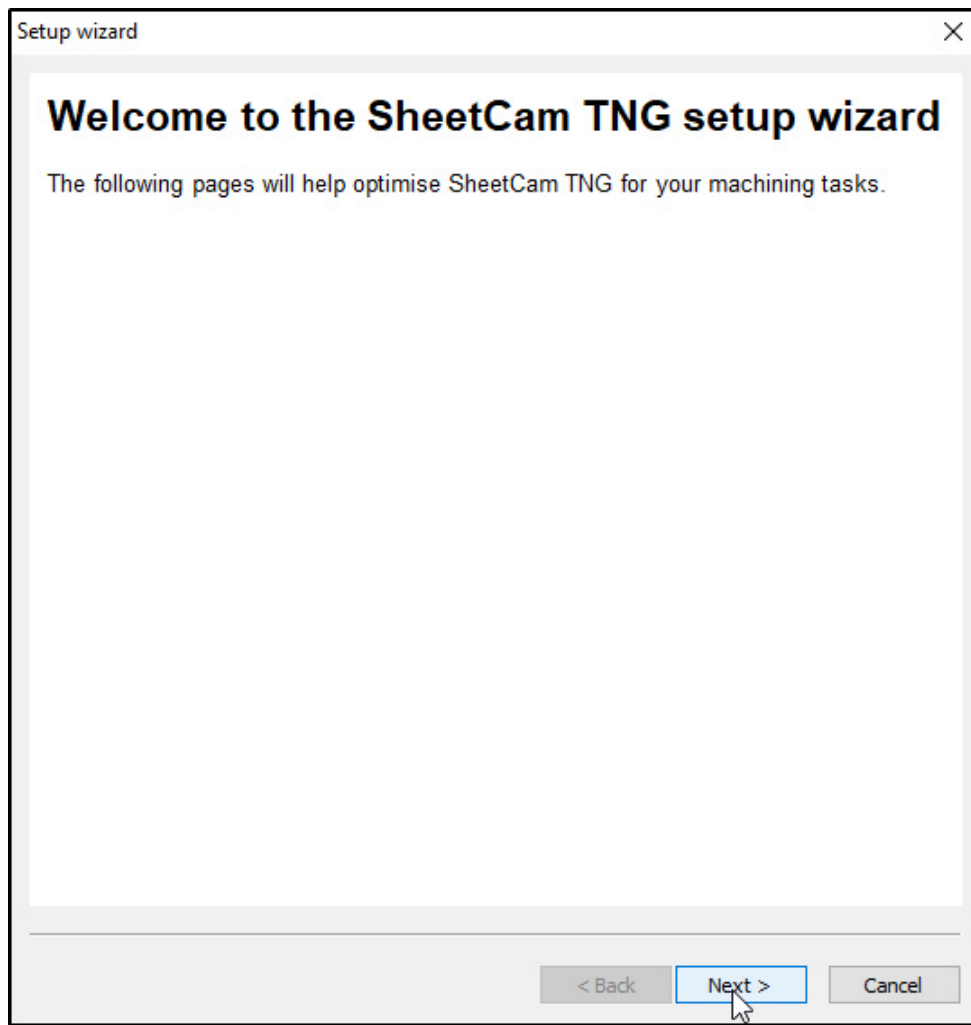
### Software Setup Note

To complete this section you will need your SheetCam license file. This is provided to you by Avid CNC via email after your purchase of SheetCam. It is recommended to save this file to your Downloads folder or Desktop. If you need to purchase a SheetCam license, they are available at `{{ config.build.domain.store }}`/sheetcam-plasma-cam-software-p-445.html.

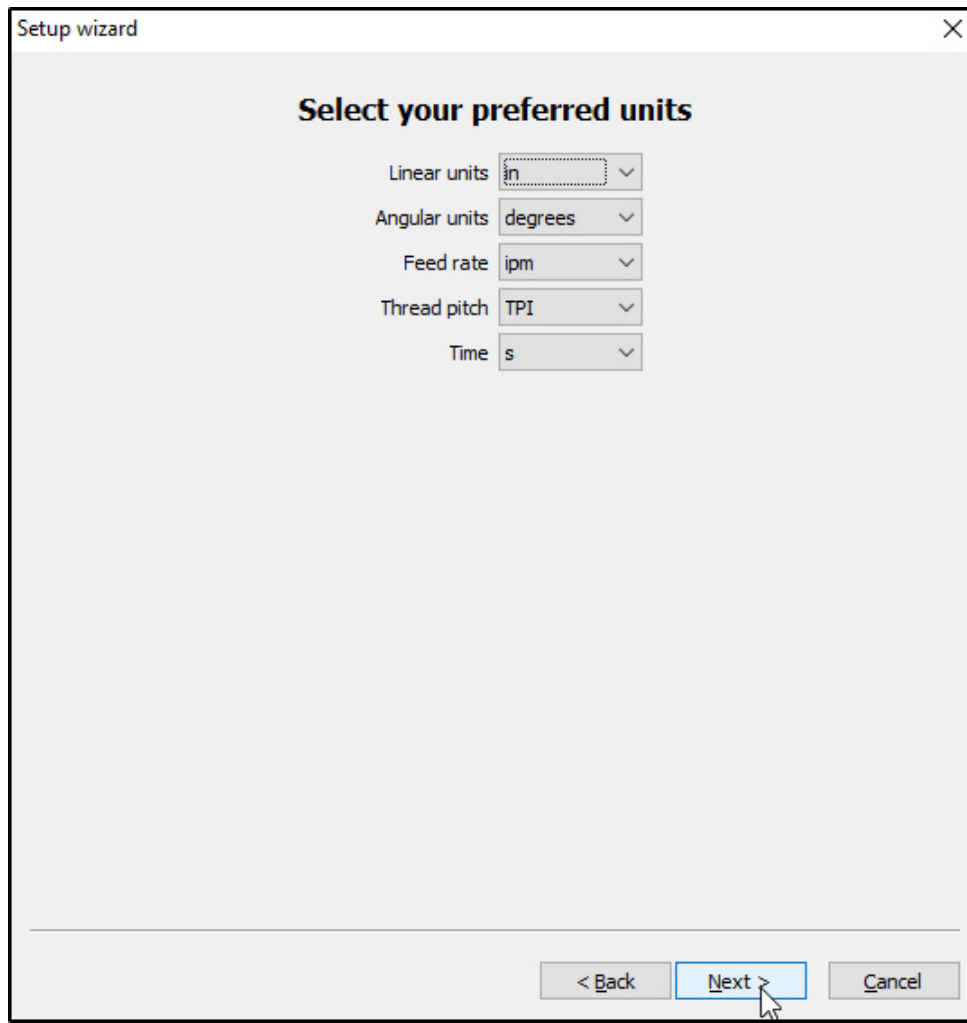


- Select your default language.

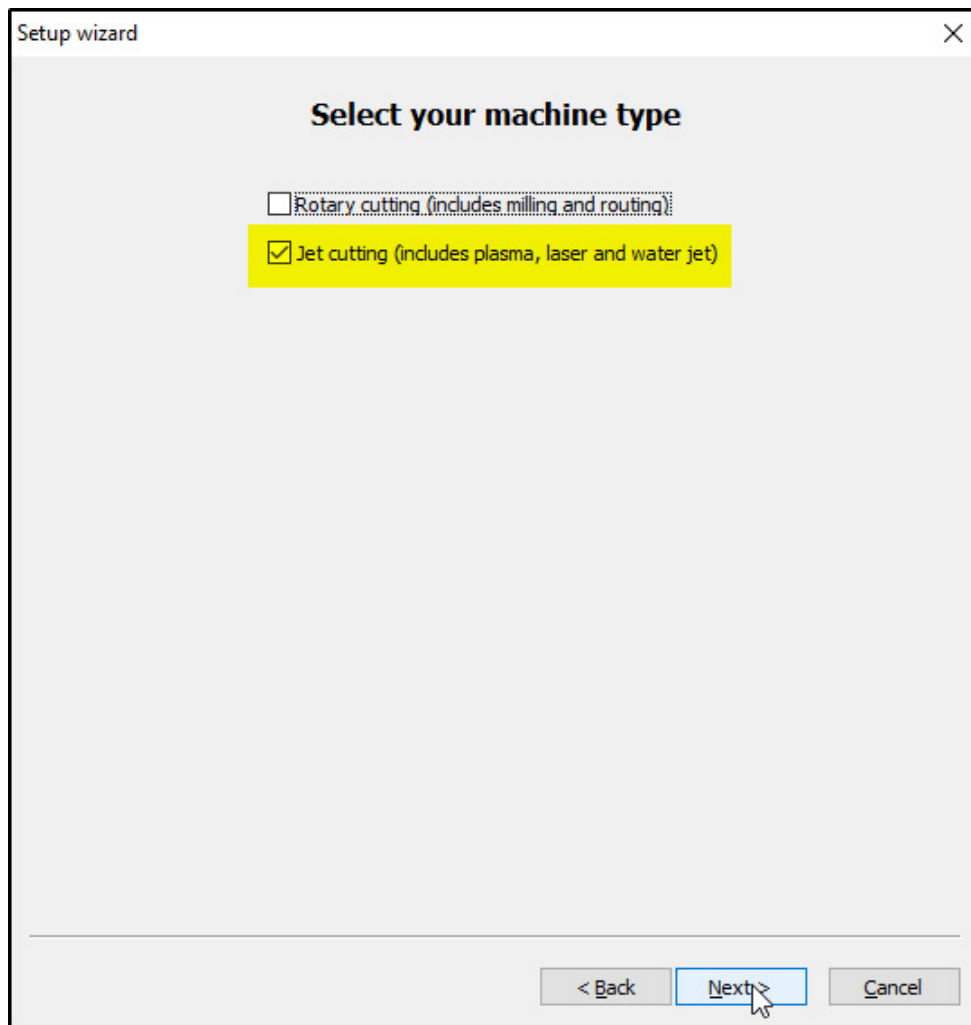




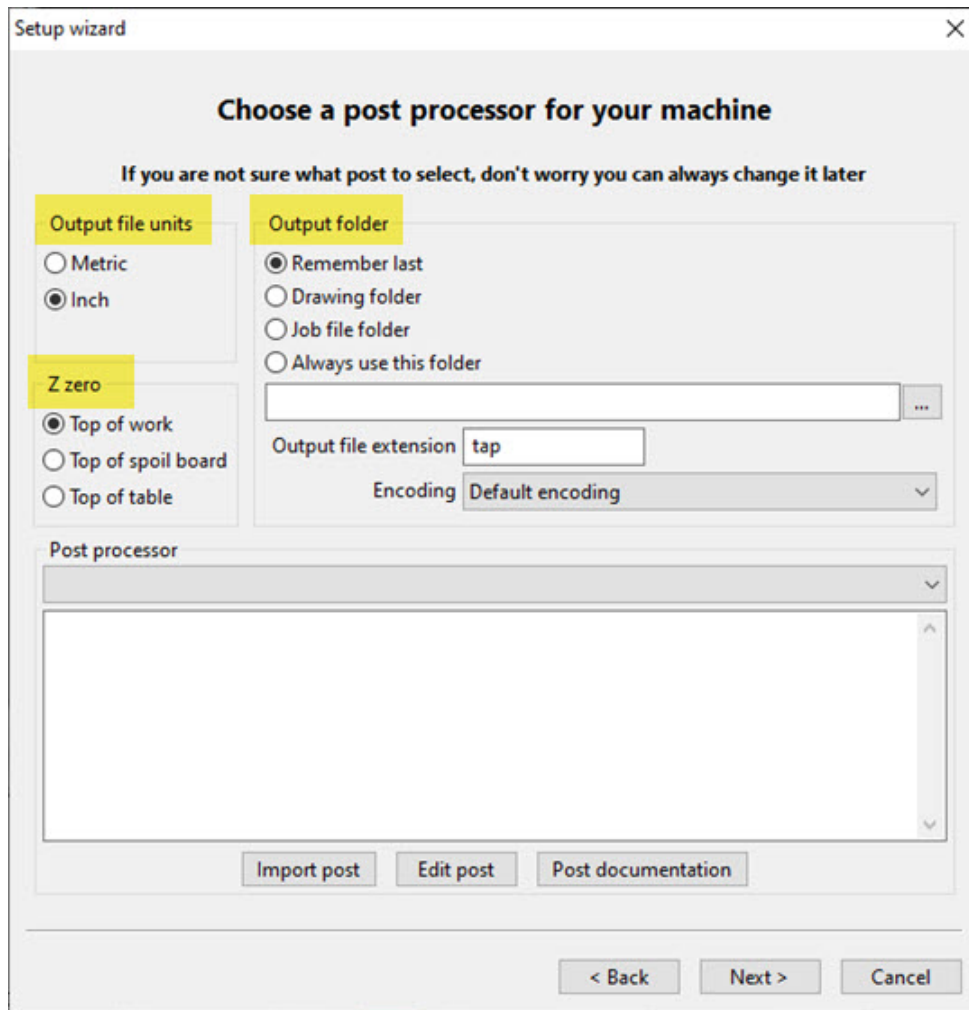
- Use the "**SheetCam TNG setup wizard**" to setup SheetCam for plasma cutting.



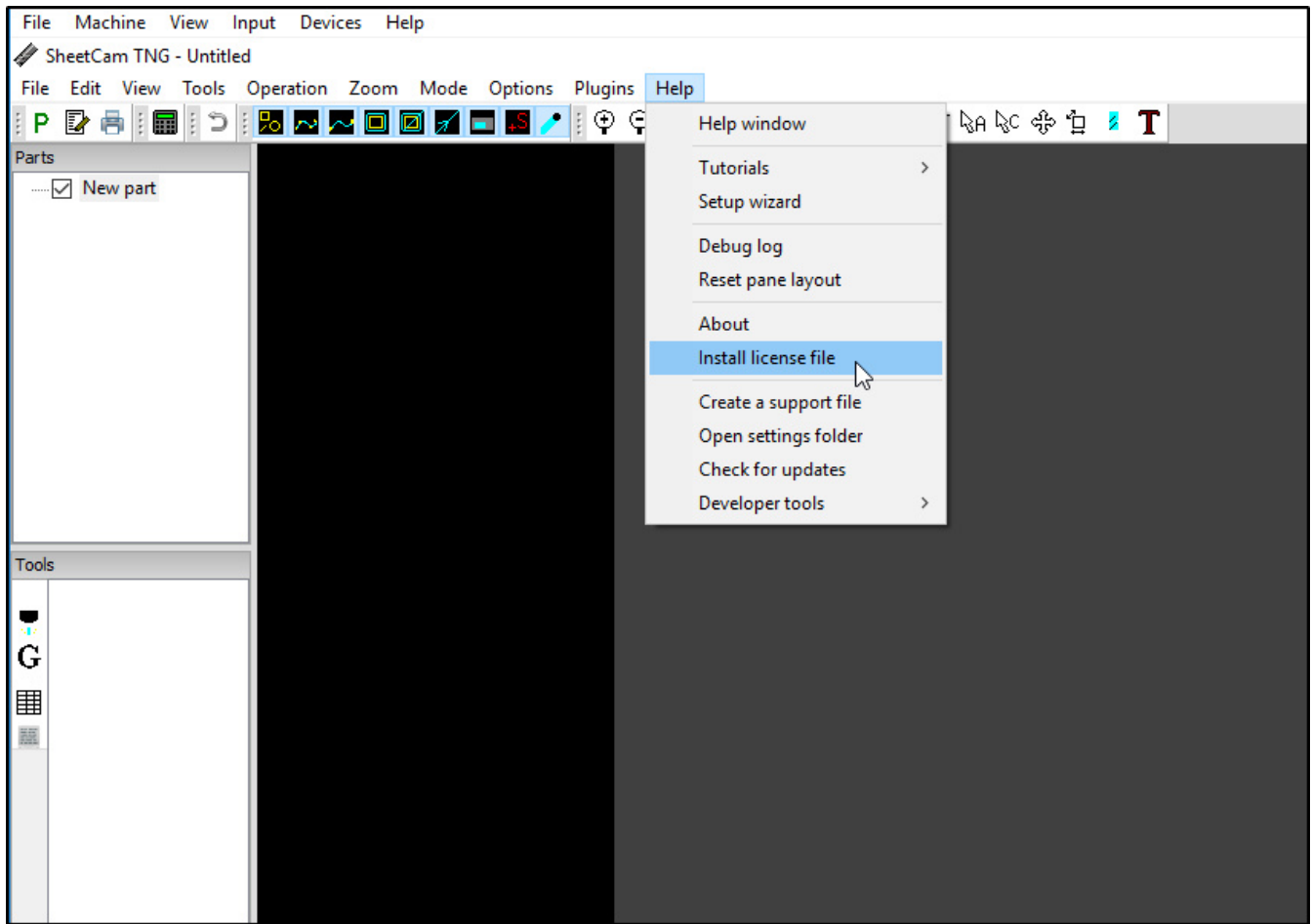
- Select your preferred units.



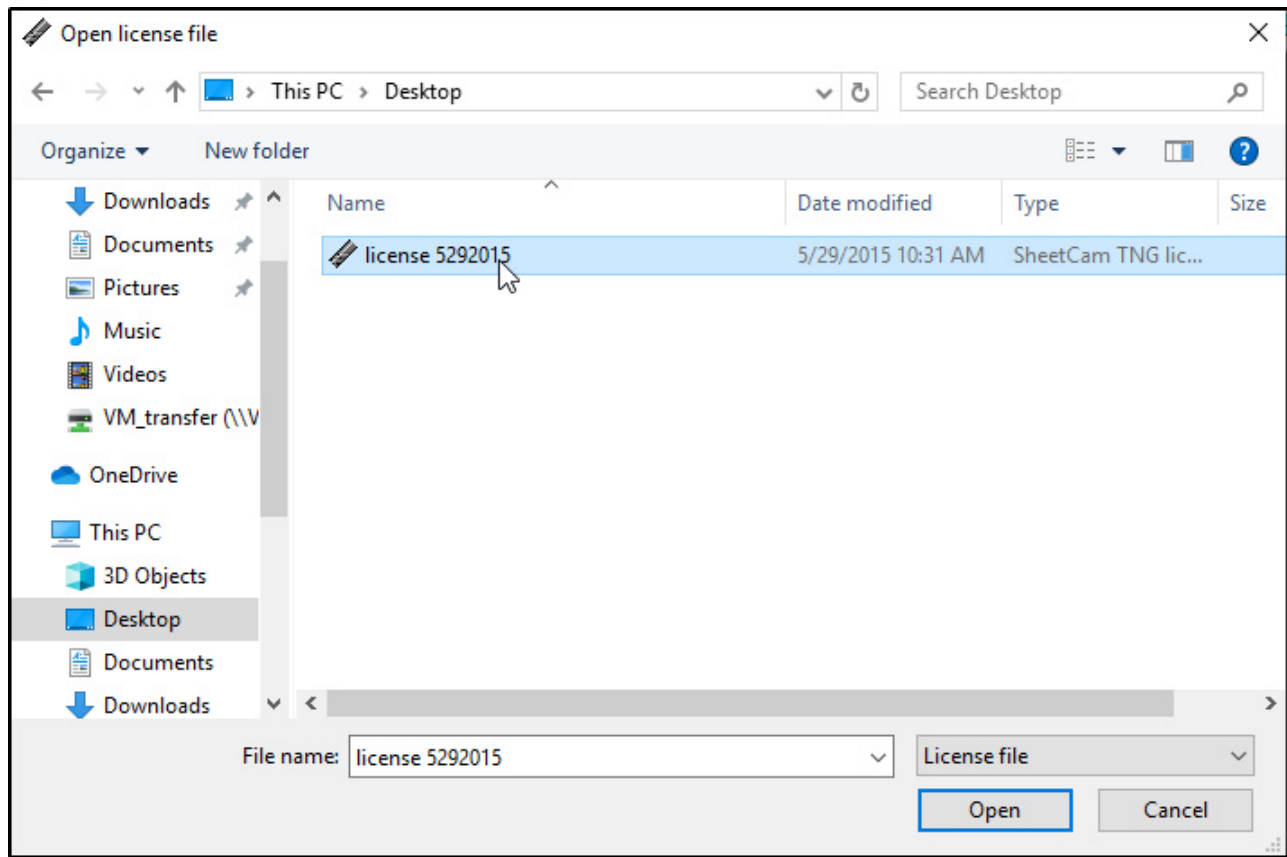
- Select "**Jet cutting**" machine type for plasma machines.



- Choose your selections for "Output file units", "Z zero", and "Output folder". The post processor selection will be set in Section 3.



- After completing the setup wizard, navigate to **"Help > Install license file"**.



- Select your license file and click **"Open"**.
- **Close SheetCam before continuing with the next section.**

### **i** SheetCam License

If you have not already downloaded your SheetCam license, follow the instructions in the email that was sent by Avid CNC after you purchased SheetCam.

### 3. Install Post Processor and Tool Tables

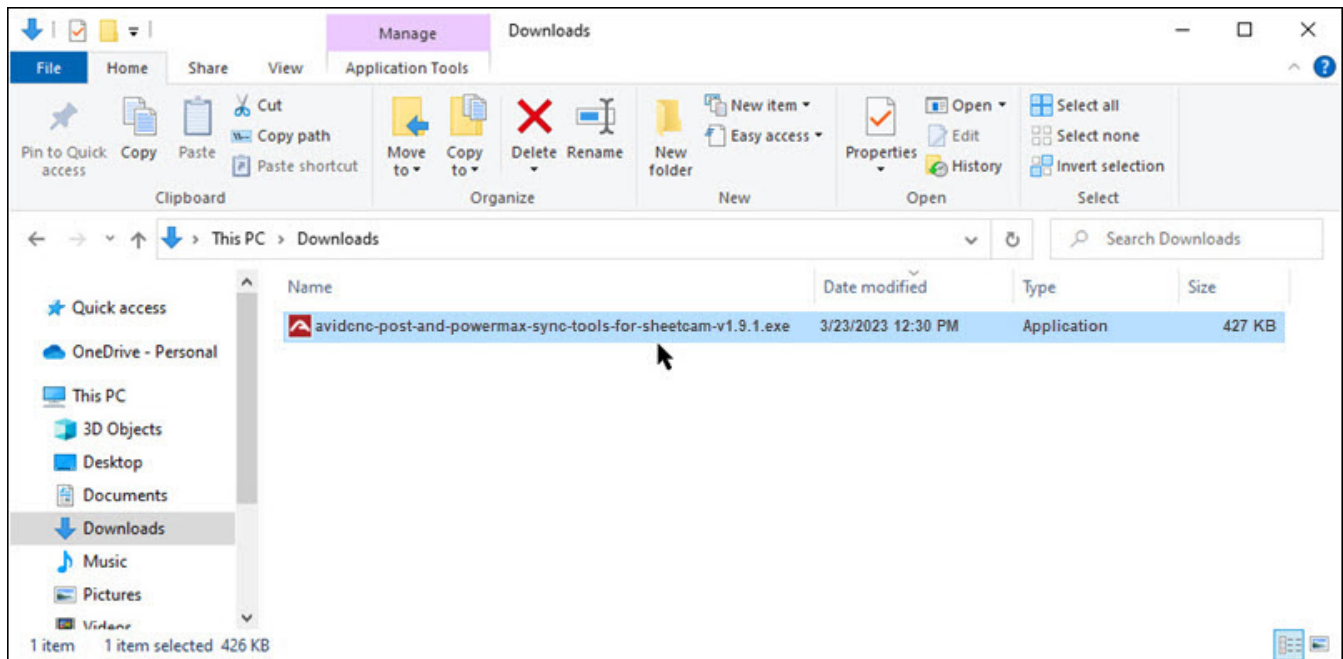
In this section you will download and install the Avid CNC Mach4 post processor for SheetCam and tool tables for Hypertherm torches. Begin by downloading the installer for these items. ({{ config.build.domain.support }}/instructions/software/downloads/sheetcam)

#### **i** SheetCam Usage

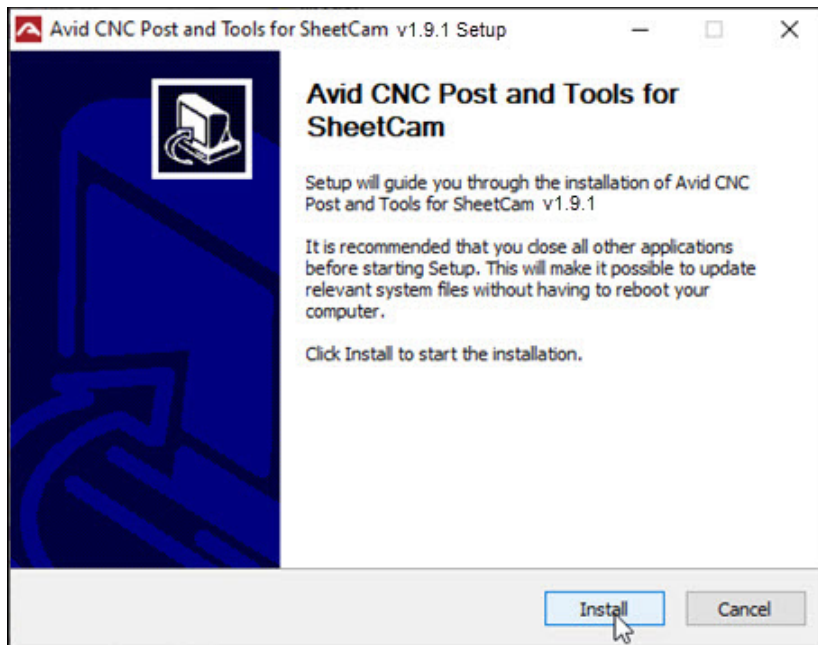
If you will not be using the Avid CNC tool tables, you will need to Contact Us ({{ config.build.domain.store }}/contact\_us.php) for instructions to add the required SheetCam code snippets and path rules.

#### **i** Installation Note

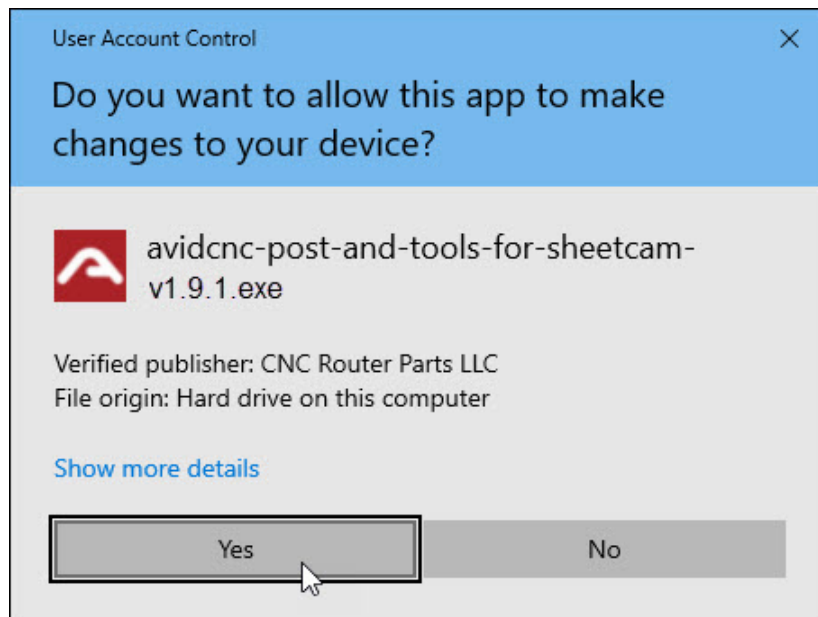
Make sure the SheetCam software is closed before continuing with this section.



- Run the installer you just downloaded. The name may vary depending on your download selection.



- Click **"Install"** to begin the installation.



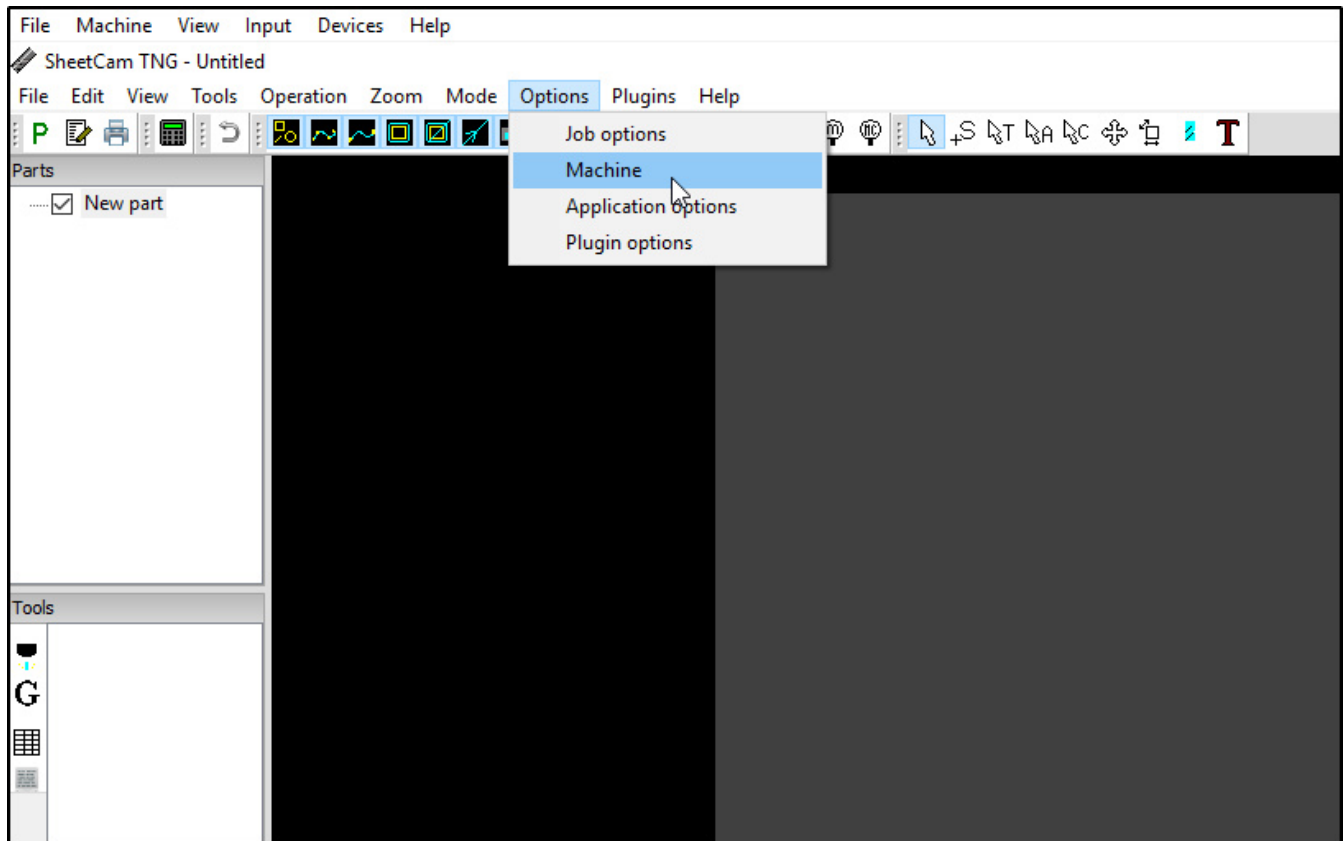
- Part way through the install, you'll be asked about making changes to your device. Click **"Yes"** to allow the plasma tools files to be installed correctly.
- Once installation has finished, continue to the next section.



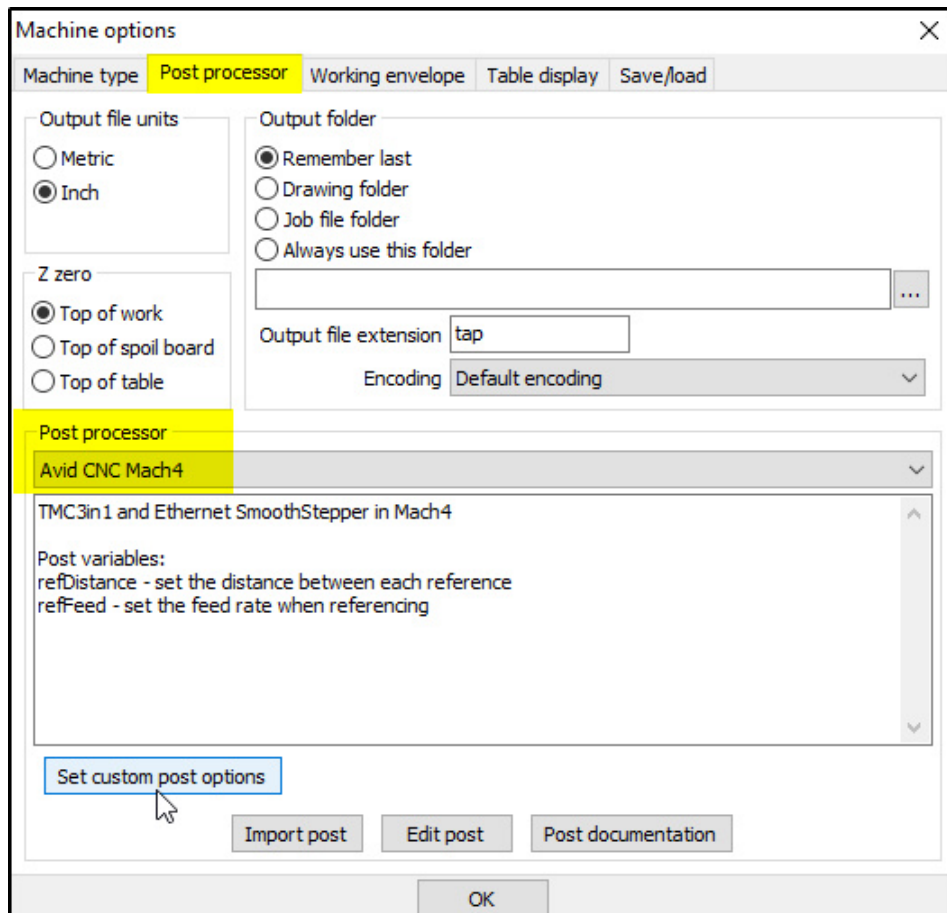
## 4. Post Processor Setup

In this section you will configure your post processor setting for your machine. This is a critical step to ensuring programs created in SheetCam work correctly.

### Post Processor Selection



- Open SheetCam.
- Navigate to **"Options > Machine"**.



- On the **"Post Processor"** tab, verify **"Avid CNC Mach4"** is the current post processor.
- At the bottom of the screen, select **"Set custom post options"**.

## Post Processor Settings

Post settings		×
Distance between references	0 in	1
Reference feed rate	19.685 ipm	2
Slow Probe Height	0.25 in	3
Touchoff, 1 = Mechanical 0 = Ohmic	0	4
End program Z park (machine coordinates)	-0.5 in	5
OK		

### 1. Distance between references

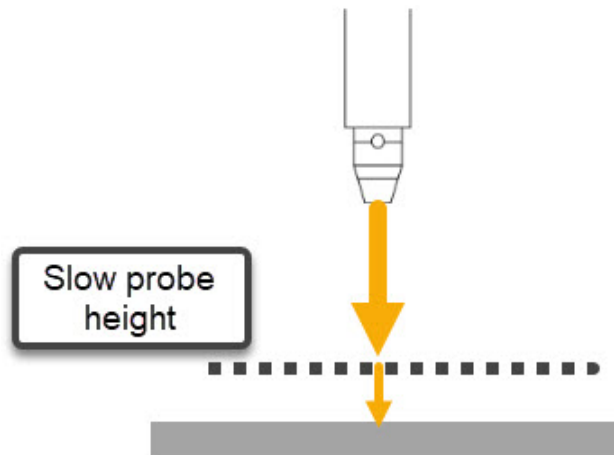
Sets a radius around the most recent probe where subsequent pierces will start without a probe. This value can be increased to reduce program run time on flat, stable (usually thick) material that will not have a significant height difference between pierce locations.

### 2. Reference feed rate

Sets the feedrate during the probe portion of a move. Increasing this from the default will reduce run time but also reduce the accuracy of the Z positioning.

### 3. Slow Probe Height

Sets a height above the last material Z position that the torch will rapid down to before starting the slow probe move.



The first probe in a program will always be full height at slow speed to initially find the material. Using the Slow Probe Height greatly reduces the run time of programs with many pierces.

#### 4. Touchoff type

This dialog sets the touchoff type - Ohmic or Mechanical. For general operation, Ohmic touchoff is strongly recommended.

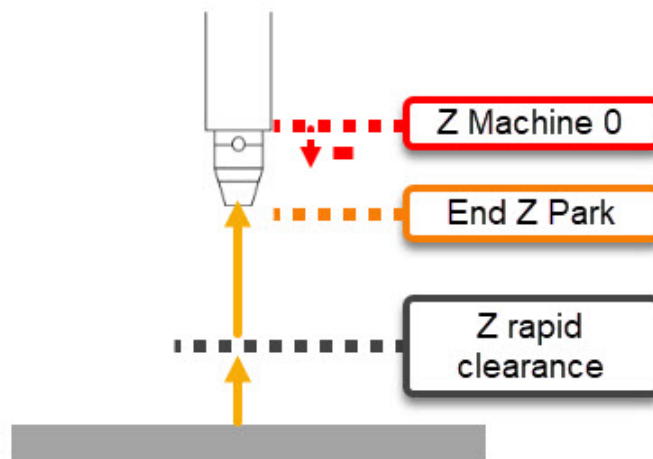
##### **Note**

Using Mechanical touchoff requires opening the Ohmic Protection Box and changing a jumper position.

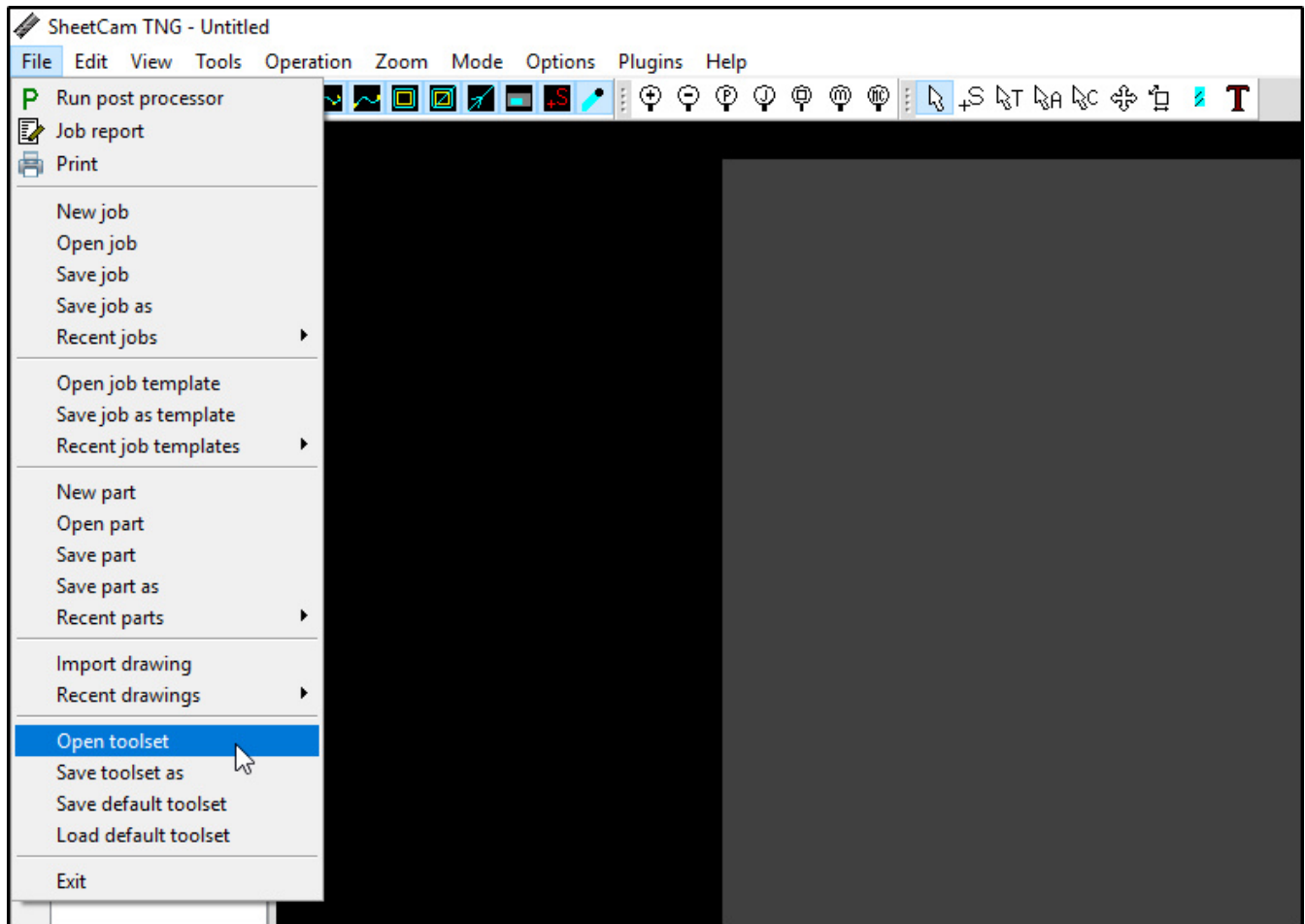
For more information please refer to the **Plasma Getting Started Guide**.

#### 5. End Program Z Height

Sets a machine coordinate location to park the Z after a program. This is used to drive the Z up and reduce water splashing at the end of a cut.



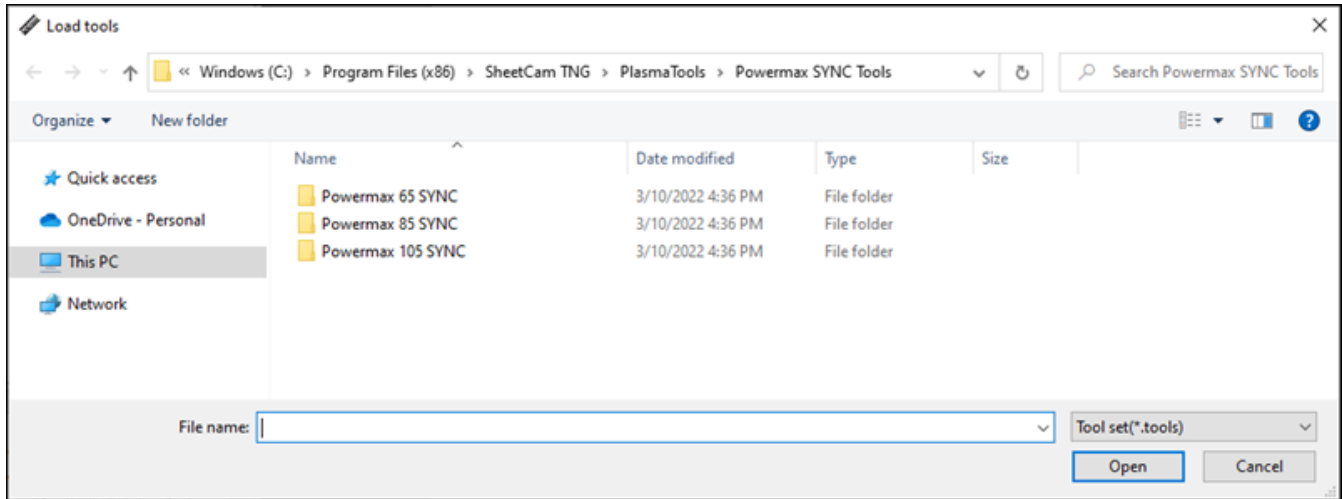
## 5. Choosing a Toolset



- To load a tool set, navigate to "File > Open toolset".

## Torch Option

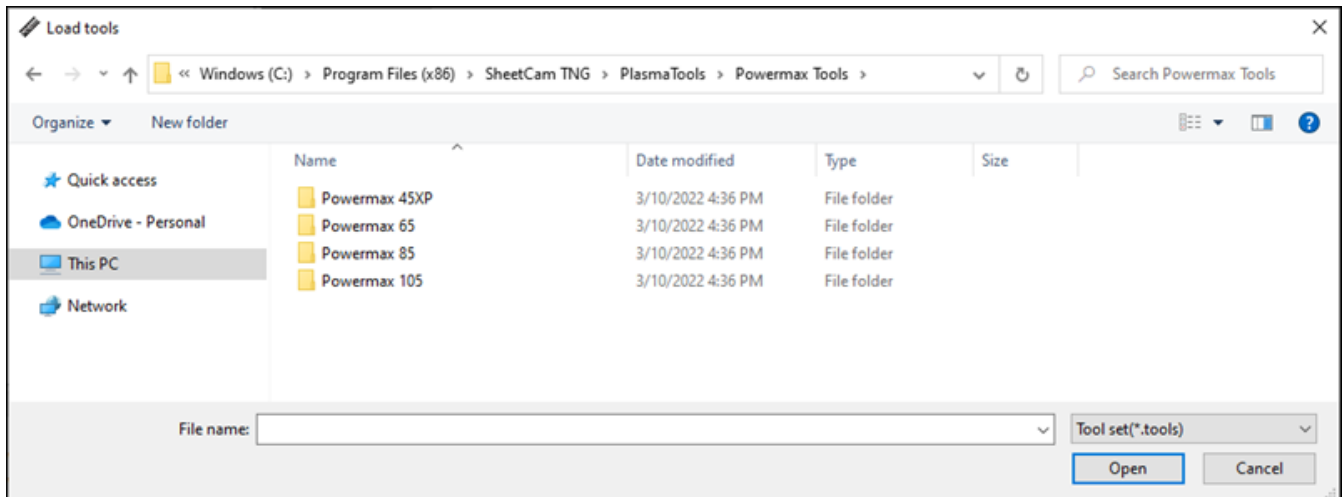
For Hypertherm Powermax SYNC torches



- Open the folder for your specific Hypertherm plasma system.

## Torch Option

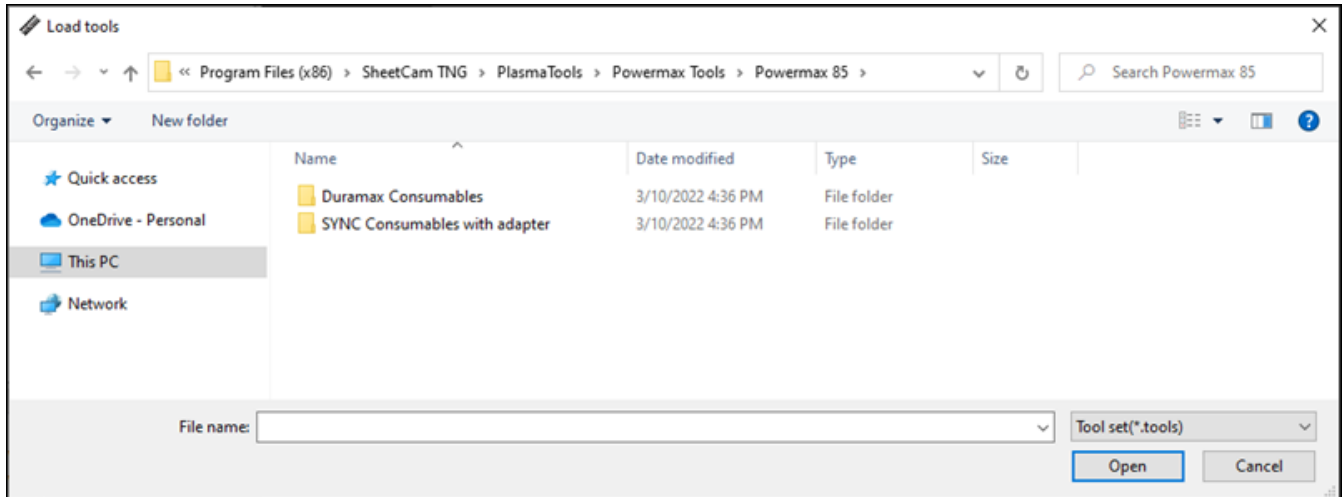
For Hypertherm Powermax Duramax torches



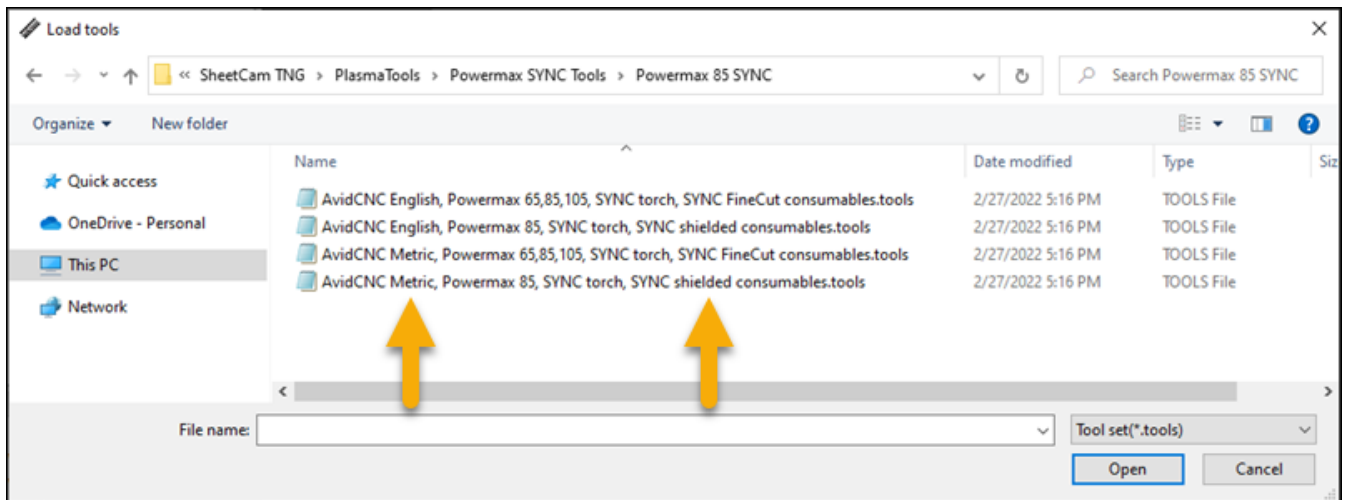
- Open the folder for your specific Hypertherm plasma system.

## Torch Option

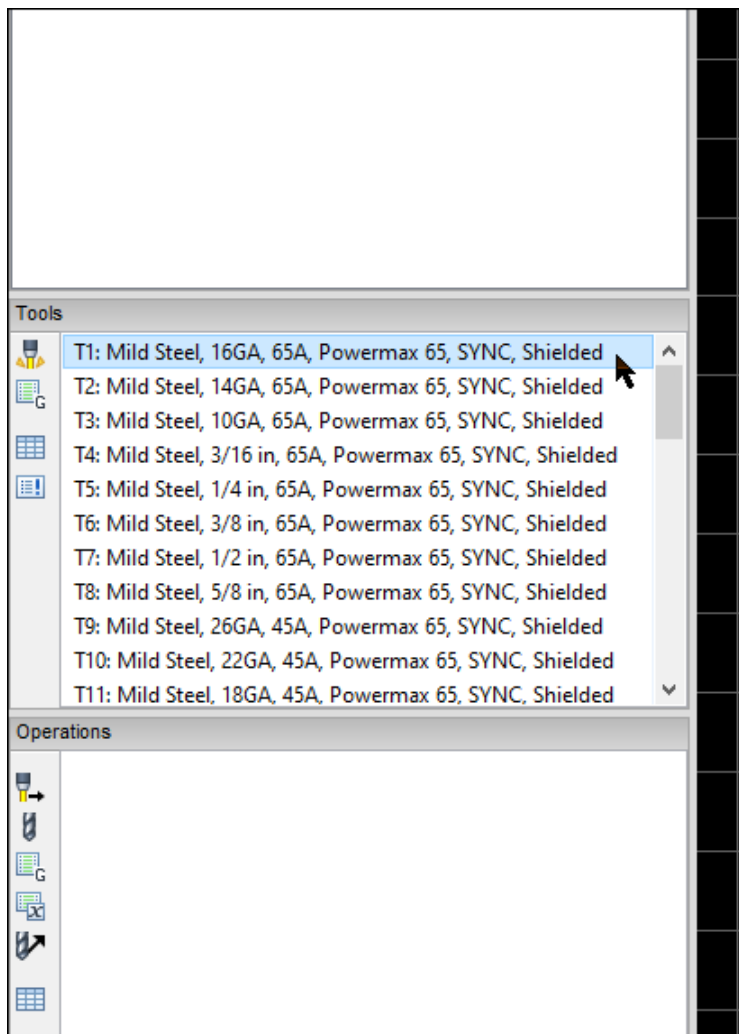
For Hypertherm Powermax Duramax torches only



- Open the folder for the type of consumables you are using with the Duramax torch.
  - **Duramax consumables** - a consumable stack of retaining cap, shield, nozzle, electrode, and swirl ring.
  - **SYNC consumables with cartridge adapter** - a single piece cartridge with adapter.



- Select the appropriate toolset for your desired units (English or Metric) and the type of mechanized consumables you are using.
  - **Shielded** - these are standard mechanized consumables.
  - **FineCut** - these are specific consumable sets designed for improved fine feature cutting.



- After selecting your tool set, the **"Tools"** pane will show a list of possible tools.
- Double click on one of the tools.



Jet tool

Tool Notes

Name: Mild Steel, 10 GA, 45A, PM 45XP, Duramax Shielded

Type: Plasma

Tool number: 3

Path rules: None

Kerf width: 0.057 in

Feed rate: 129 ipm Update operations

Pierce delay: 0.4 s

Pierce height: 0.15 in

Plunge rate: 100 ipm

Cut height: 0.06 in

Pause at end of cut: 0 s

Use leadin settings:

Target Tip Volts: 128

AD1 Delay after Arc OK Time: 1.06 s

Lead in:  None,  Arc,  Tangent,  Perpendicular, Length: 0 in

Lead out:  None,  Arc,  Tangent,  Perpendicular, Length: 0 in

Leadin type:  Normal,  Ramp,  Wiggle, Wiggle size: 0.3937 in

Start at the centre of circles smaller than: 0 in

Use code snippet: None

OK Cancel Help

- You will see the tool is pre-populated with the values from the Hypertherm cut charts and Avid CNC calculations.
- Using the AvidCNC Mach4 post processor, these values will be included in your GCode and automatically populated in Mach4 when the program is executed.

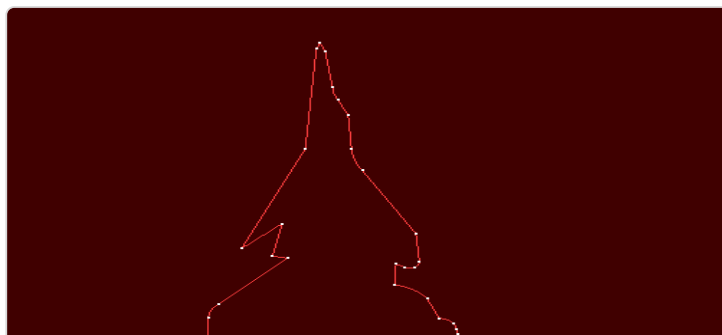
## 6. SheetCam Drawing Import Settings

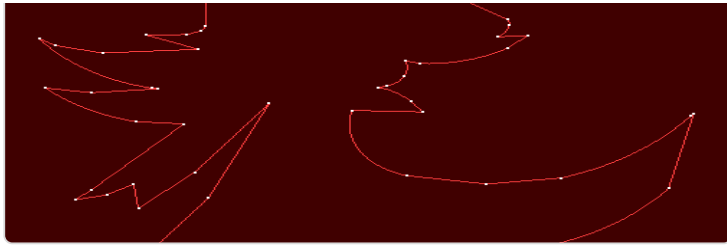
There are settings in **Options -> Application Options -> Drawing Import** that SheetCam uses to smooth and simplify vectors when drawings are imported. More detail on the settings is available in **Help -> Help Window -> Menu Items -> Options Menu**.

The Avid CNC Post and Tools installer changes some of the settings from default to better match the capabilities and needs of plasma cutting. Plasma is a relatively rough cutting method that cannot achieve very fine tolerances. Smoothing the imported vectors will result in smaller file size (1/3 - 1/2 reduction in gcode lines) and smoother motion. Smoothing will result in geometry differences that are smaller than the normal variation in kerf width throughout a cut. The final products of these two files will be indistinguishable.

Setting	SheetCam Default	Avid Default
Import link tolerance	0.0004" (0.01mm)	0.0039" (0.1mm)
Max detail reduction error	0.0008" (0.02mm)	0.0157" (0.4mm)
Arc fitting tolerance	0.0039" (0.1mm)	0.0157" (0.4mm)

Below is an example of SheetCam defaults (left) and Avid CNC defaults (right).



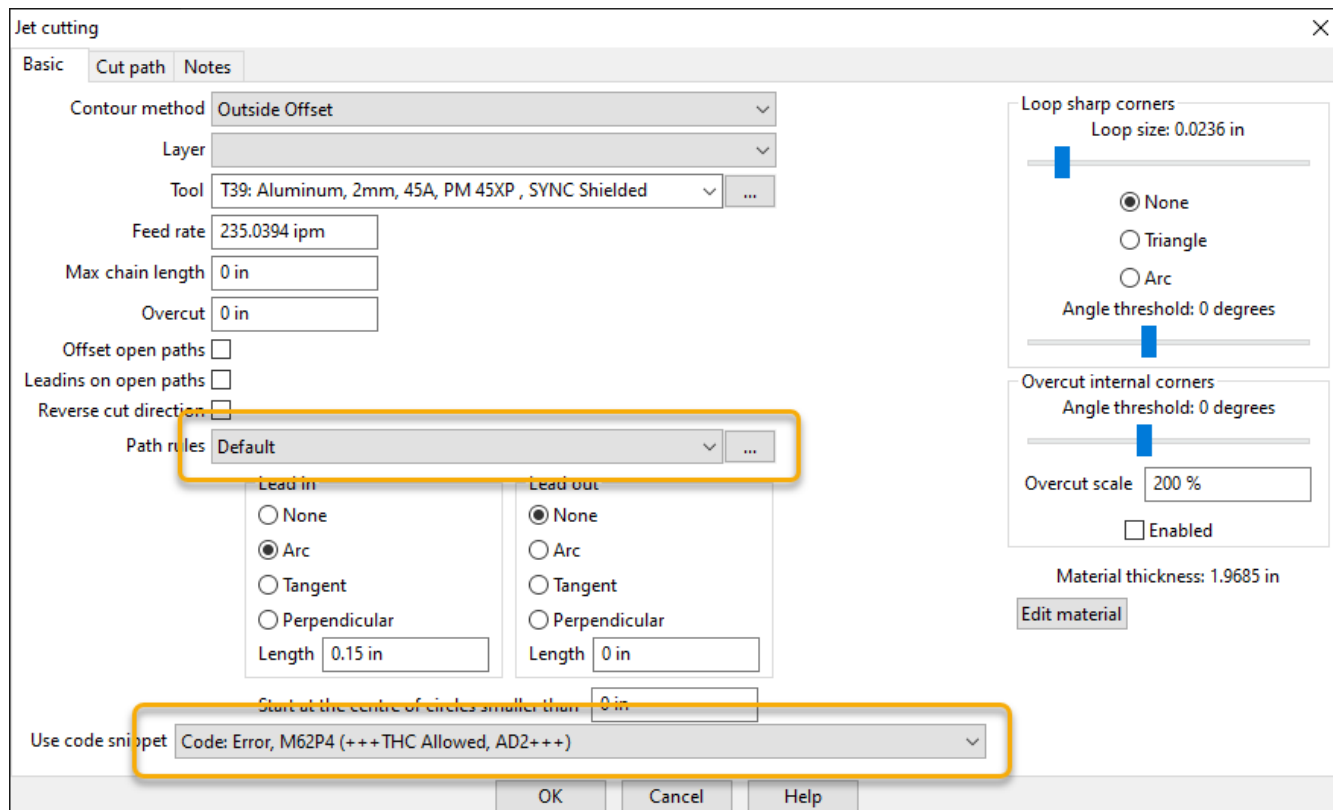


## 7. Operation Setup

When a new Jet Cutting operation is created, the Path Rules should be set to Default and the Code Snippet should be set to "M62P4 (+++THC Allowed, AD2+++)".

### Note

The code snippet may say "Error, M62P4..." initially. This is OK and can be ignored for the first instance.



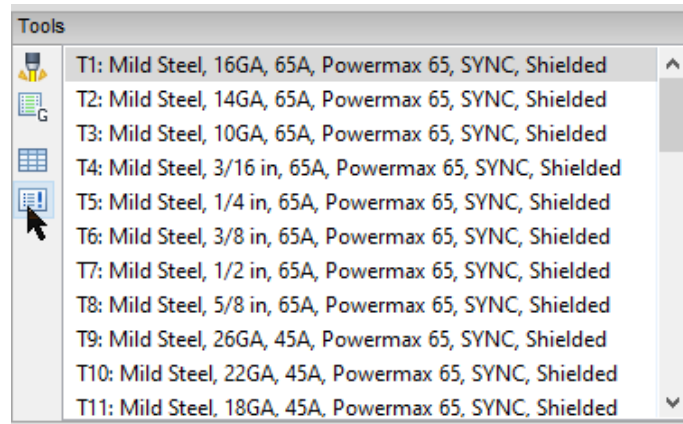
The Code Snippet is necessary to ensure that THC is allowed in the operation. The Path Rules will then allow/inhibit THC for certain features.

Note that if you don't want THC active at all during a specific operation within a gcode program, it is important to also disable the Cut/Path rules for that operation to avoid M62P4 being added automatically on a feature.

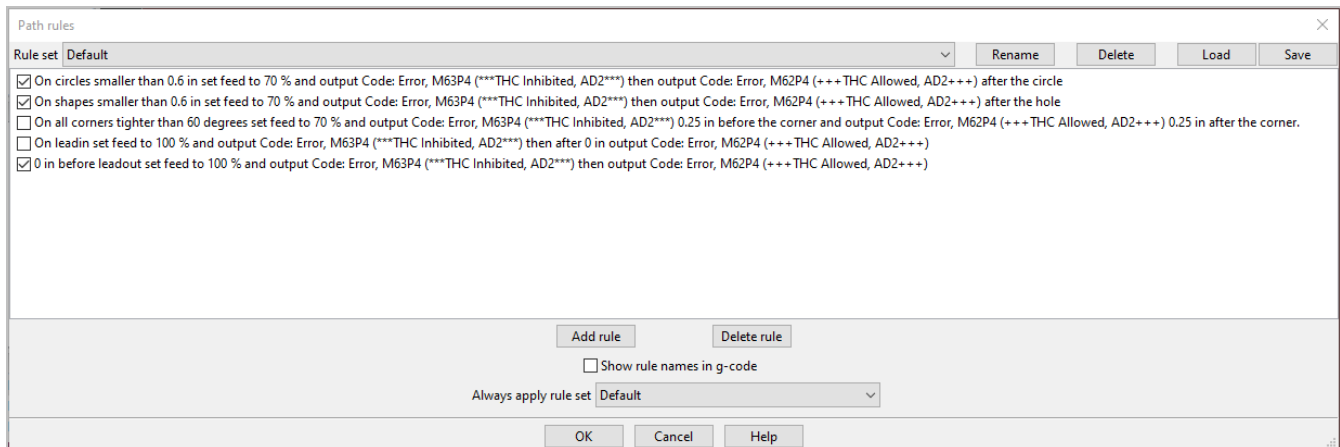
	Use Code Snippet	Use Path Ruleset	Enabled Path Rules
THC wanted	M62P4 (+++THC Allowed, AD2+++)	Default	Yes
THC not wanted	M63P4 (**THC Inhibited, AD2**)	None	None

## 8. Using Cut Rules

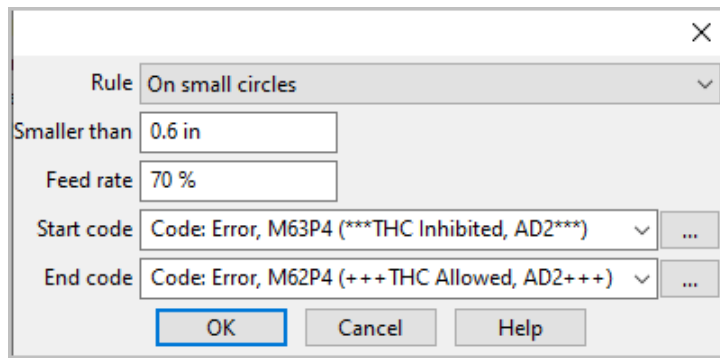
Path or Cut Rules in SheetCam serve as instructions for the post processor to modify the G-Code output for specific toolpath features. The most common uses of Path Rules are preventing dives by inhibiting THC or improving cut quality by locally reducing the feedrate.



- To open the cutting rules editor press the Cut Rules button on the left of the Tools menu on the left of the SheetCam application window, or use the drop down menu Tools -> Cutting Rules.

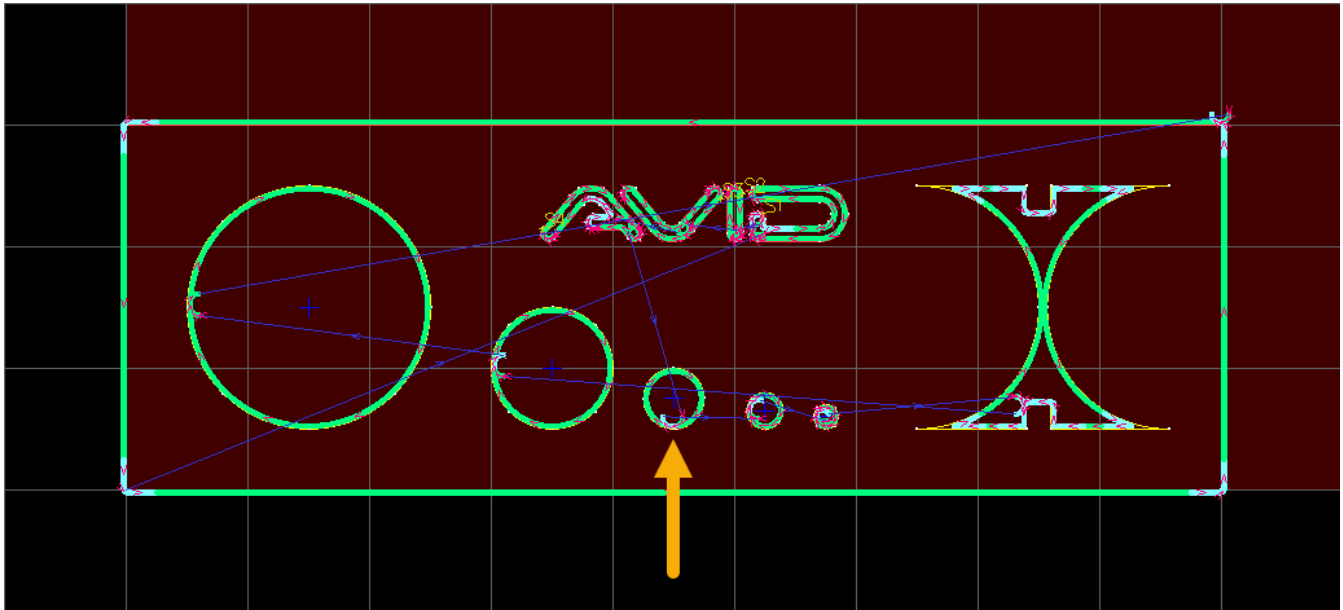
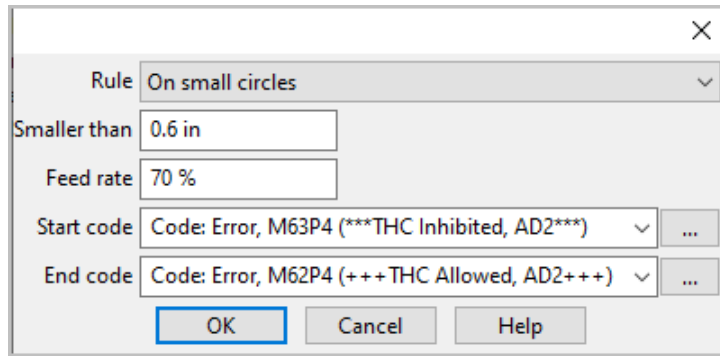


- The Path Rules window will then be displayed. The default rule set will have been installed by the "**AvidCNC for SheetCam**" installer.
- To disable or enable path rules click the checkbox to the left of the path rule name.

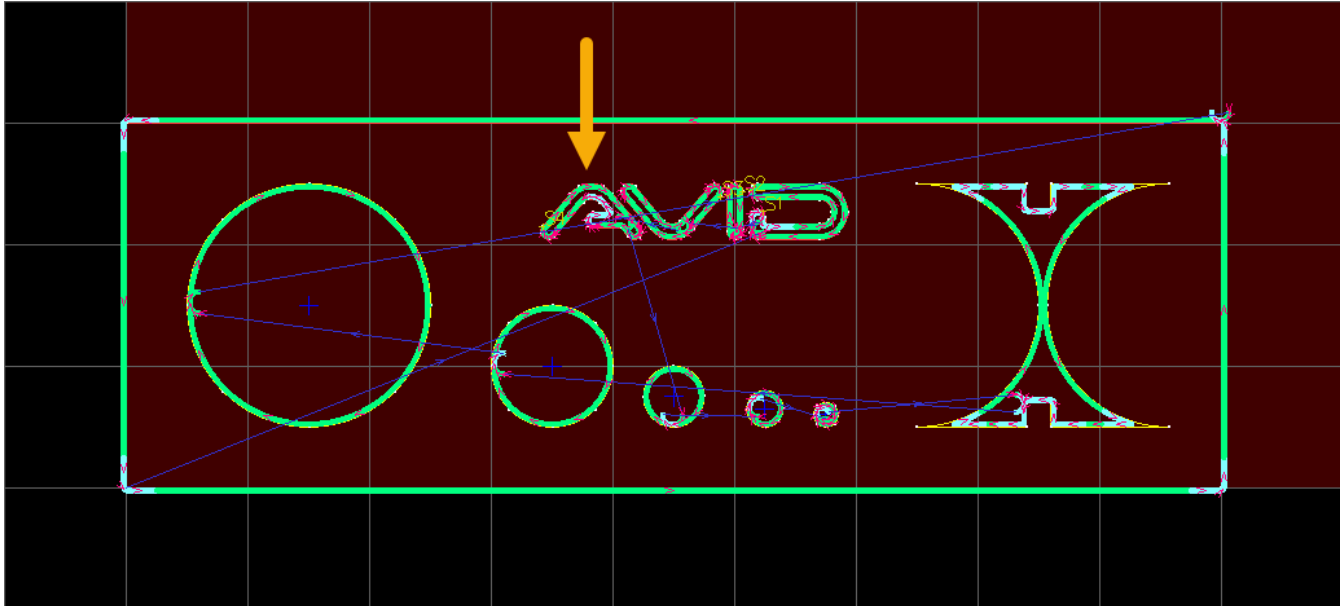
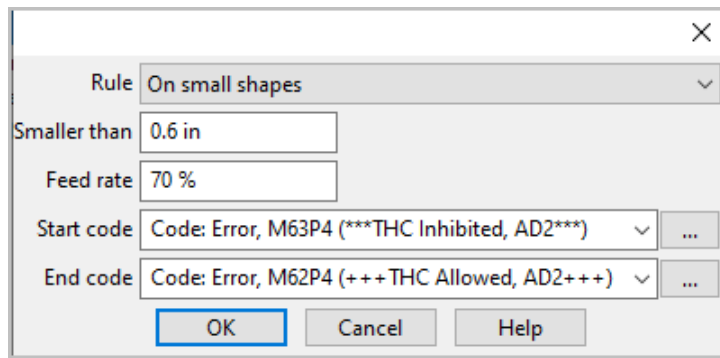


- To edit a path rule double click on the name of the rule, this will open the rule options. The rule parameters may be edited and saved by clicking "OK." "Start code" and "End code" are the M codes which will be inserted into the g-code program at the beginning and end of the path rule's effect.
- Additional types of path rules may be implemented by clicking the "Add rule" button at the bottom of the Path Rules editor window.

Descriptions of the Avid CNC default path rules are below

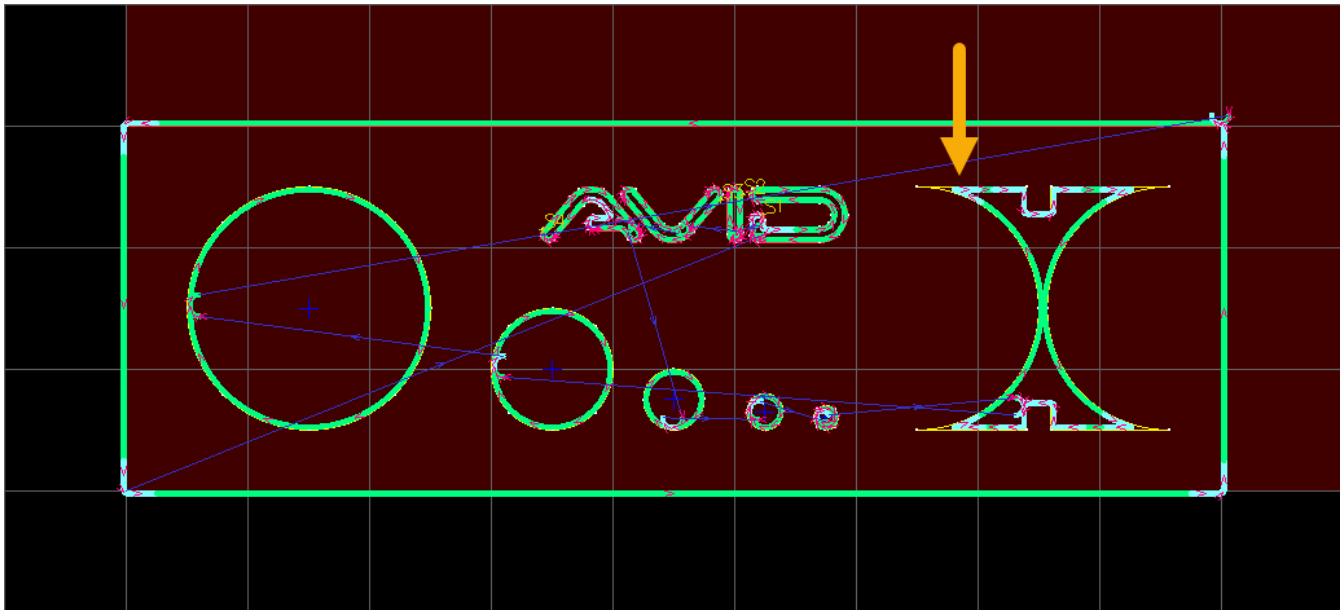
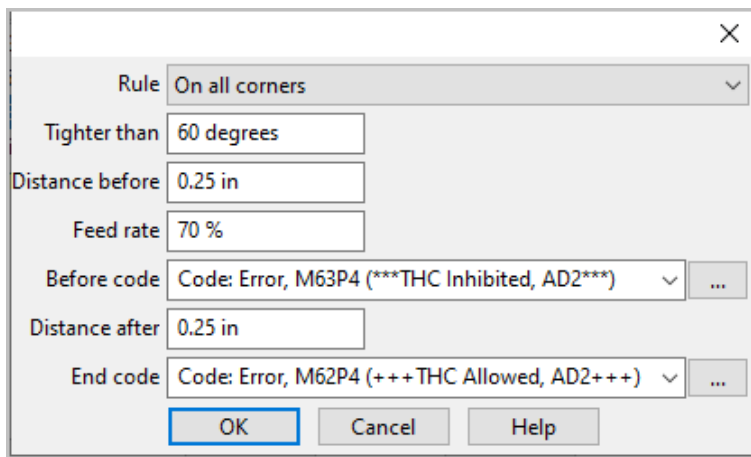


- **Enabled** by default.
- **Function:** disables THC for holes with a diameter under the "Smaller than" value. Reduces the feed rate to the set percentage.
- For most materials the height change across a small circle is too small to really need THC motion, but the tip voltage is unstable during the cut.
- It is useful to prevent unwanted torch dives caused by unstable tip voltage during short cuts.
- Reducing the feedrate for small circles can help reduce the bevel of the cut.



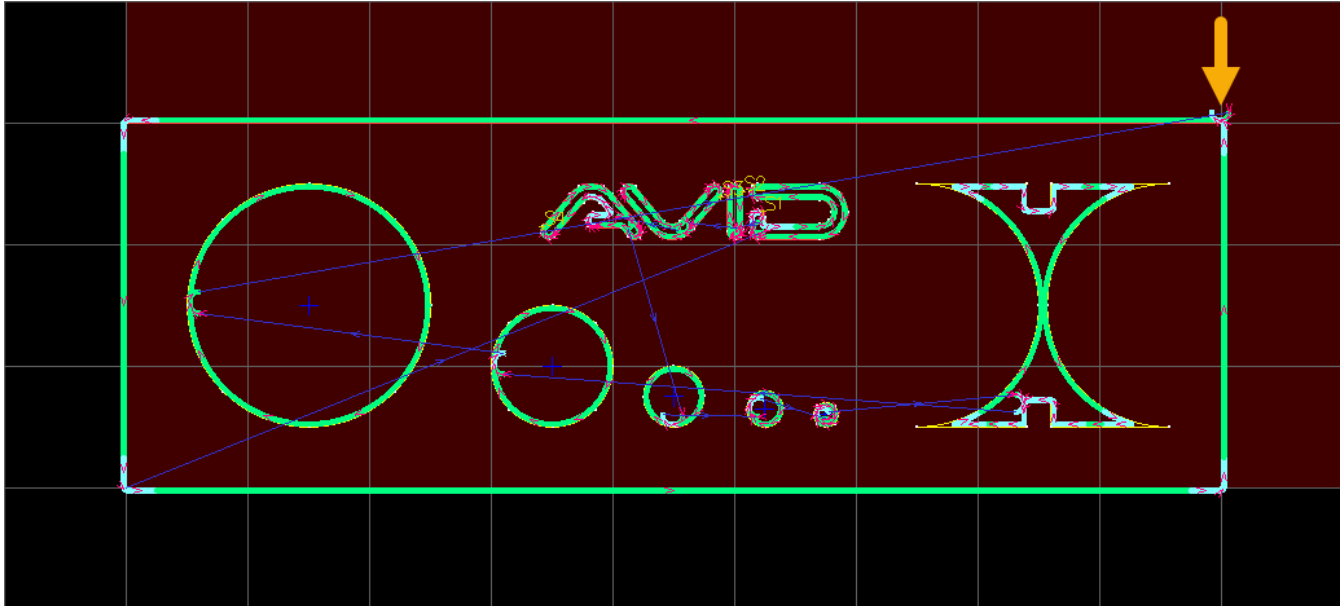
- **Enabled** by default.
- **Function:** disables THC for closed profiles with an *equivalent diameter* under the "Smaller than" value. Reduces the feed rate to the set percentage.
- **Equivalent Diameter** is defined as the diameter of a circle with a circumference equal to the shape's perimeter.
- For most materials the height change across a small shape is too small to really need THC motion, but the tip voltage is unstable during the cut.
- It is useful to prevent unwanted torch dives caused by unstable tip voltage during short cuts.
- Reducing the feedrate for small shapes can help reduce the bevel of the cut.



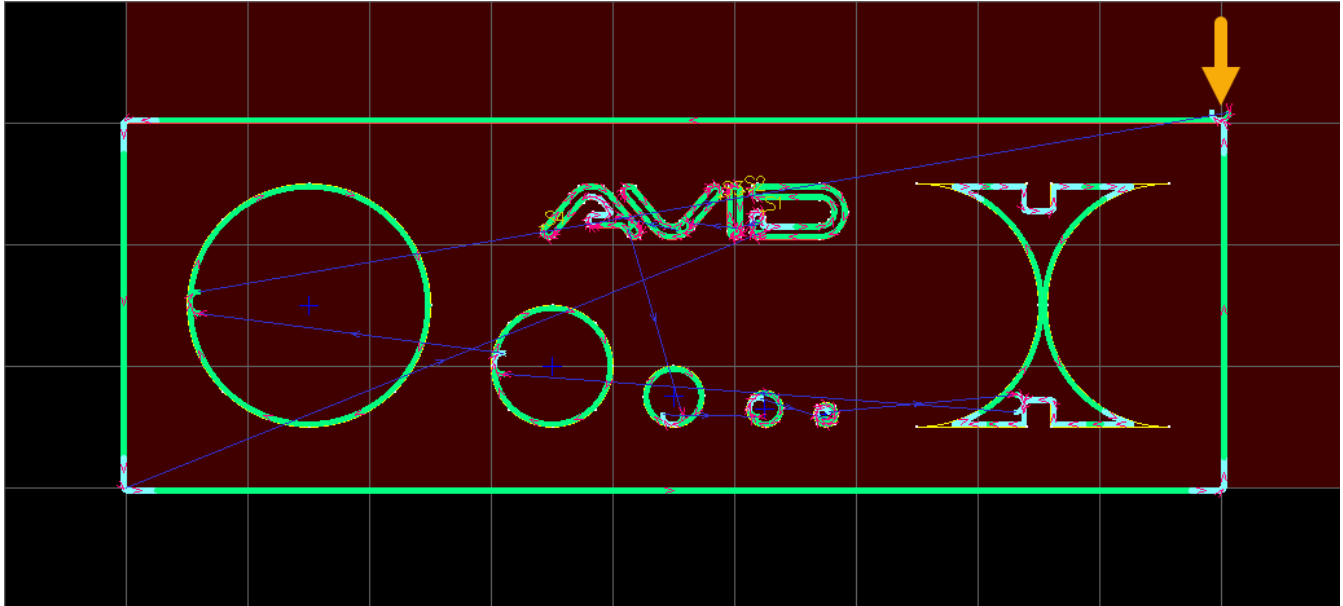
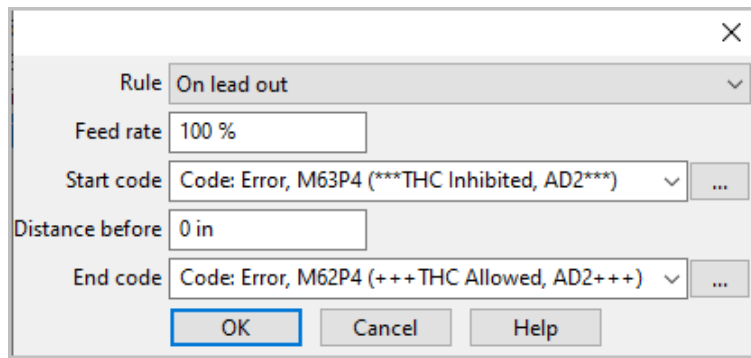


- **Disabled** by default.
- **Function:** disables THC for a set distance before and after tight corners. Reduces the feed rate to the set percentage.
- When a cut makes a tight corner it requires the machine to slow down. This causes the plasma arc to grow and the measured tip voltage to increase.
- This rule can prevent unwanted torch dives caused by artificially high tip voltage in tight corners.
- In most cases, this rule is not needed and can hinder THC's ability to track the material properly.

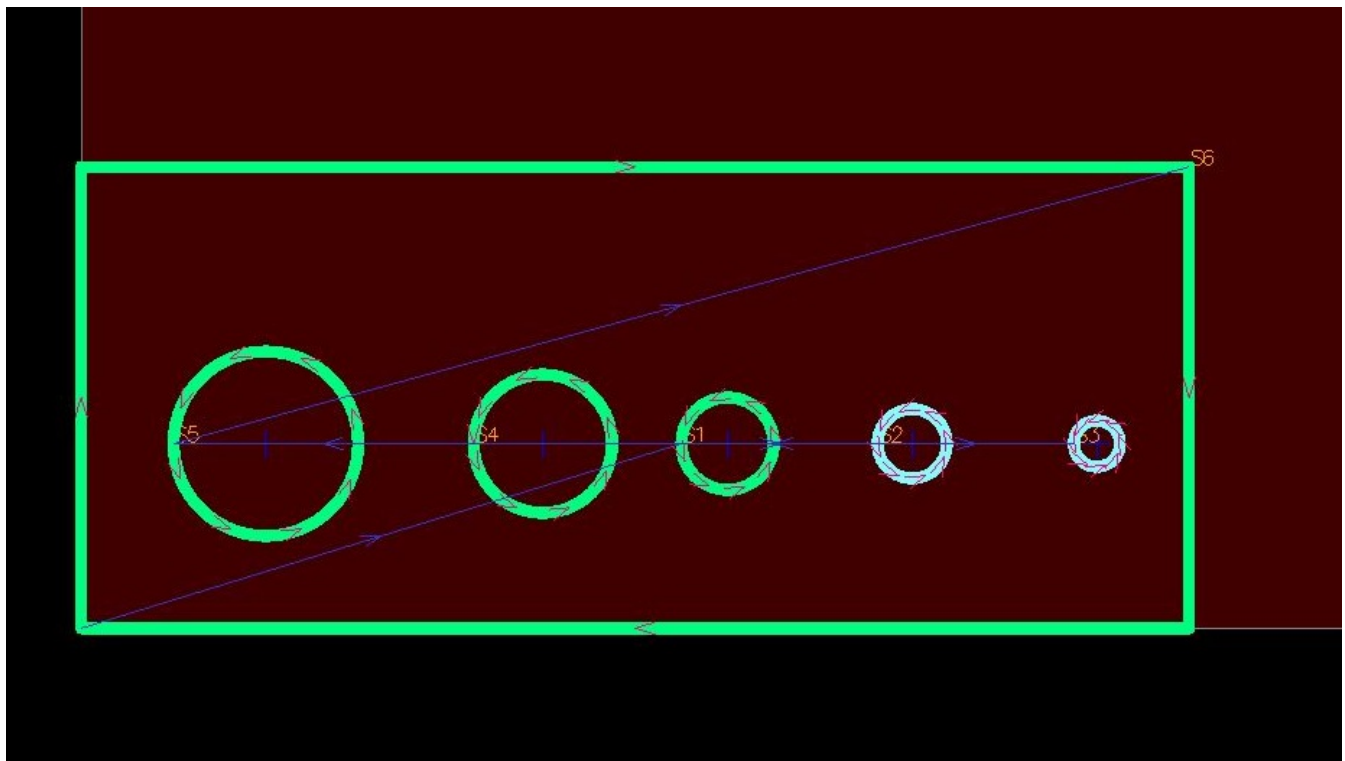
Rule	On lead in	⌵
Feed rate	100 %	
Start code	Code: Error, M63P4 (**THC Inhibited, AD2**)	...
Distance after	0 in	
End code	Code: Error, M62P4 (+++THC Allowed, AD2+++)	...
<input type="button" value="OK"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>		



- **Disabled** by default.
- **Function:** disables THC during lead-ins.
- During lead-ins the tip voltage is typically unstable and above the target.
- This rule can prevent unwanted torch dives caused by artificially high tip voltage.
- In most cases this rule is not needed due to redundancy with other THC Anti Dive settings.



- **Enabled** by default.
- **Function:** disables THC during lead-outs.
- During leadouts the tip voltage is typically unstable and above the target.
- This rule can prevent unwanted torch dives caused by artificially high tip voltage.



- While M codes will not show up in the tool preview, any path rules which change the feedrate percentage will highlight the effected operations in blue instead of the normal green.