



# SheetCam Software Setup Guide

*v2020Q4.1*

# SheetCam Software Setup Guide

This guide provides step-by-step instructions for installing SheetCam CAM Software. This includes 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.

## 1. Install SheetCam

A screenshot of a web browser displaying the SheetCam Downloads page. The browser's address bar shows 'sheetcam.com/downloads'. The page features the SheetCam logo at the top left 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', 'SheetCam TNG Development version', and 'SheetCam TNG for Linux'. Each section provides detailed instructions and download links for various versions of the software.

Downloads

sheetcam.com/downloads

# SHEETCAM

:. Downloads :.

Home Features Downloads Plugins Support Purchase Contact Distributors Scanything Forum

### SheetCam TNG Stable version

As SheetCam develops, more features are added. However in some cases this can introduce bugs. This version is not the most recent but the functions it does have are thoroughly tested. **Note this download is limited to approximately 180 lines of generated code.** You need to purchase a license to remove this limit.

If you are installing on a computer without Internet access, the installer may ask you to install this Microsoft update: [vcredist\\_x86\\_vc2009.exe](#). Windows 2000 users may need to install [Windows 2000 update rollup 1](#) first.

[Download SheetCam TNG V6.0.30](#)

[Show recent changes](#)

### SheetCam TNG Development version

This is the latest version of SheetCam. It has the newest features but also may not be as reliable as the stable version. If you find any bugs, please report them so they can be fixed. **Note this download is limited to approximately 180 lines of generated code.** You need to purchase a license to remove this limit.

If you are installing on a computer without Internet access, the installer may ask you to install this Microsoft update: [vcredist\\_x86\\_vc2009.exe](#). Windows 2000 users may need to install [Windows 2000 update rollup 1](#) first.

[Download SheetCam TNG Development V6.1.74](#)

[Show recent changes](#)

### SheetCam TNG for Linux

To install it, download the file then right-click on it and select 'properties'. Go to the permissions tab and make sure 'execute' is turned on. You can now simply double click on the file to install SheetCam.

If that does not work you can simply unzip the file. The data folder in the zip is a tarball containing everything you need to run SheetCam. Execute run-sheetcam to run SheetCam.

[Download SheetCam setup V6.1.57 32 bit version](#)

[Download SheetCam setup V6.1.57 64 bit version](#)

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

Home Features Downloads Plugins Support

## SheetCam TNG Stable version

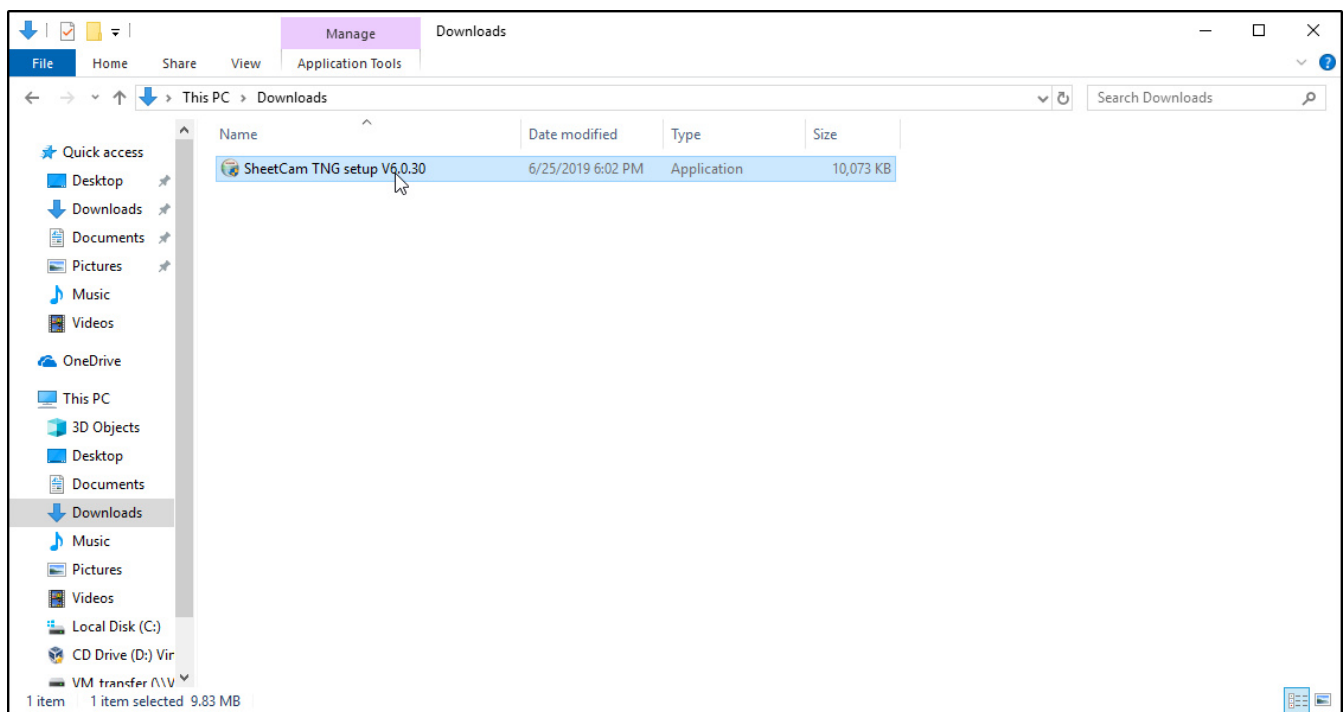
As SheetCam develops, more features are added. However in some cases this can interfere with older versions. **Note this download is limited to approximately 180 lines of generated code.**

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

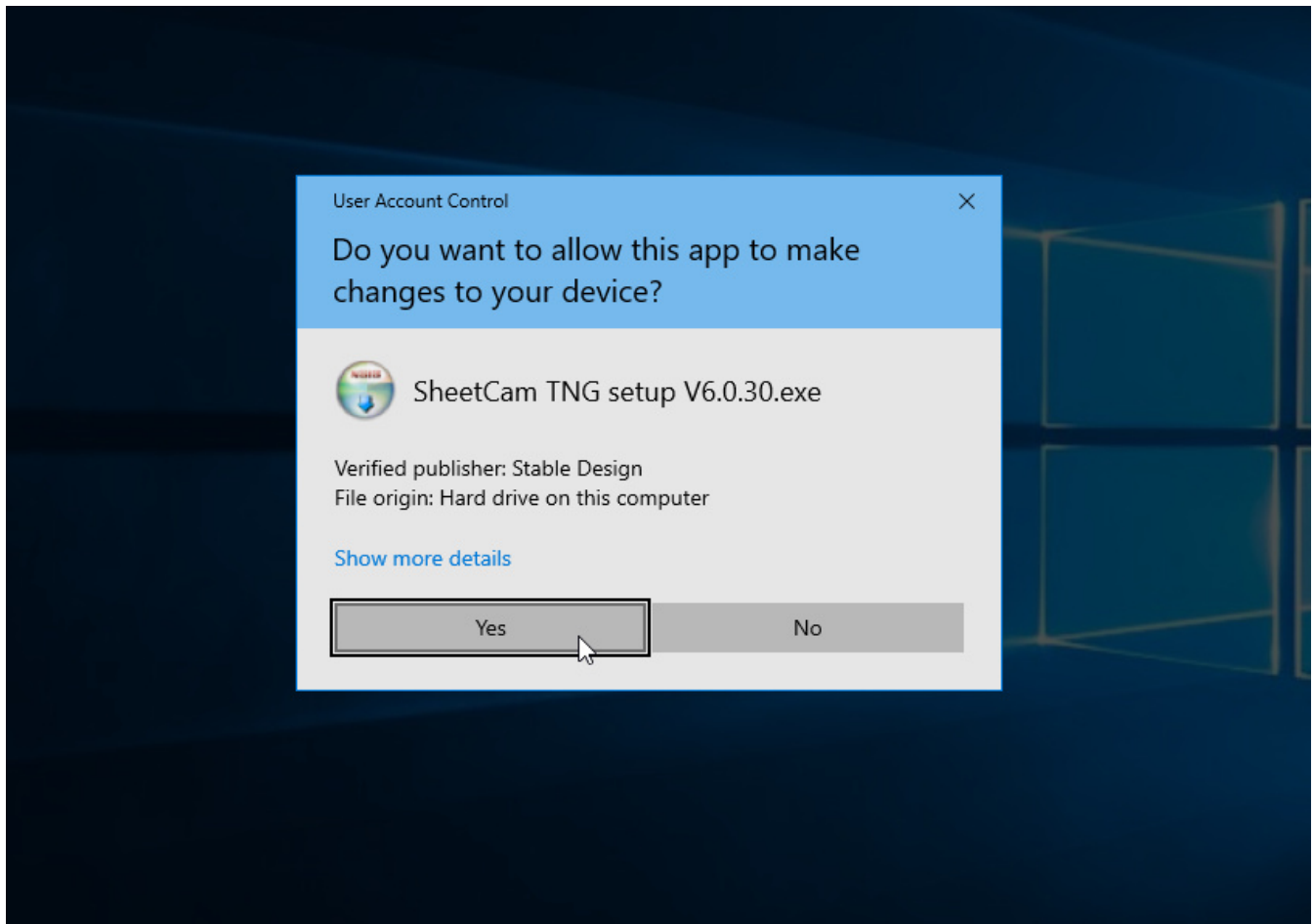
[Download SheetCam TNG V6.0.30](#)

[Show recent changes](#)

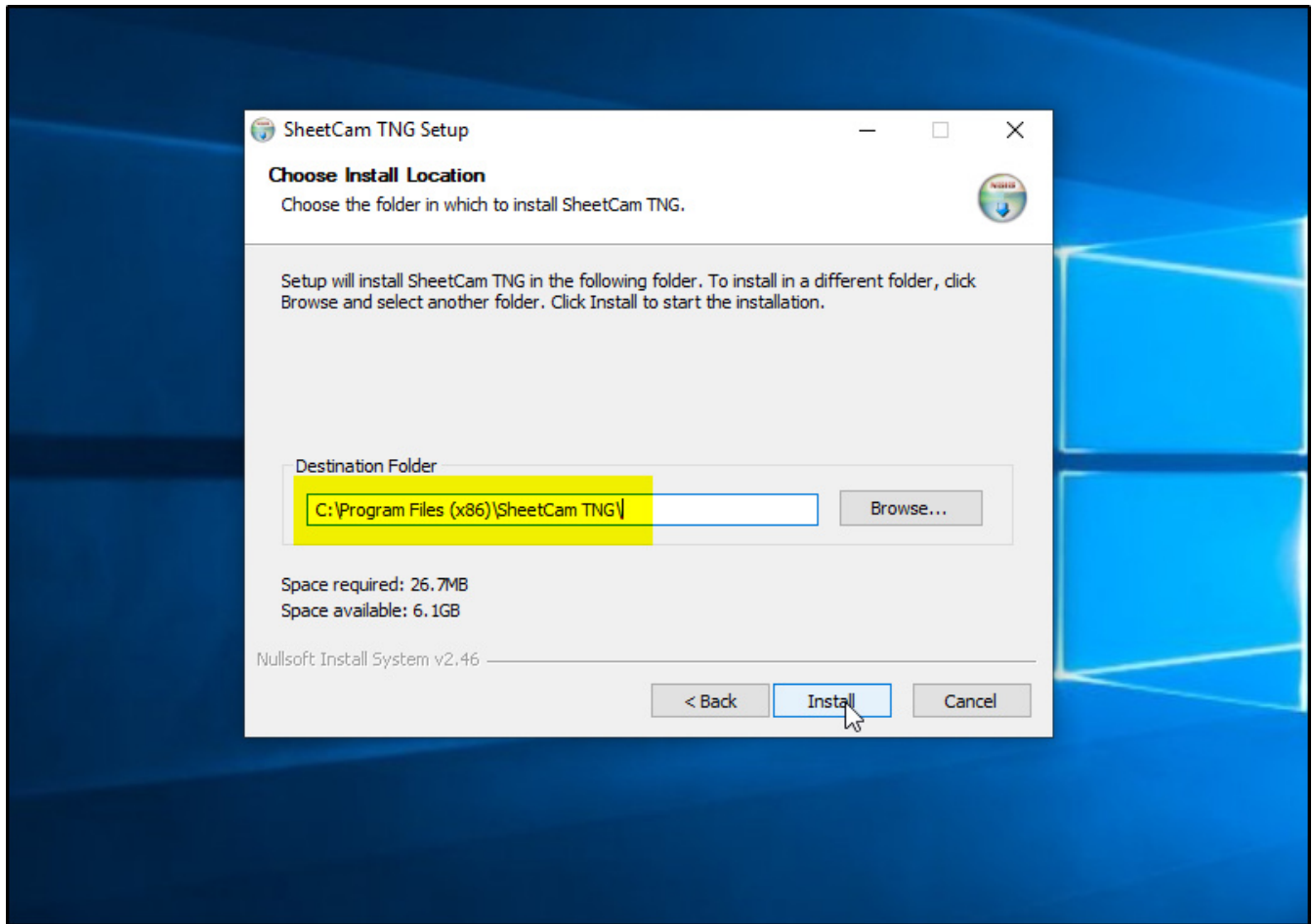
- It is recommended to download "SheetCam TNG Stable version".



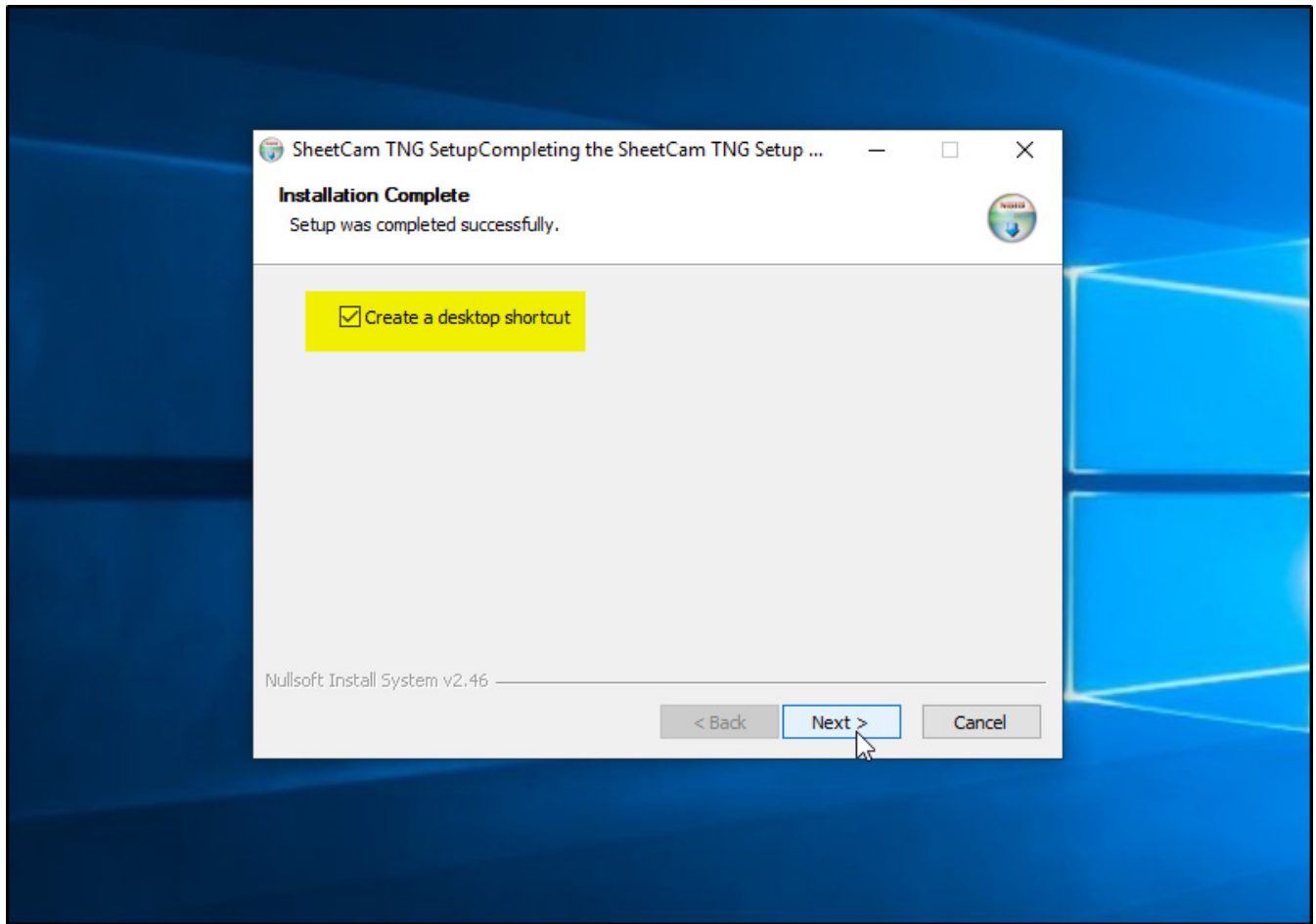
- Navigate to your downloads folder (or the directory you downloaded the installer to) and run the SheetCam TNG installer.



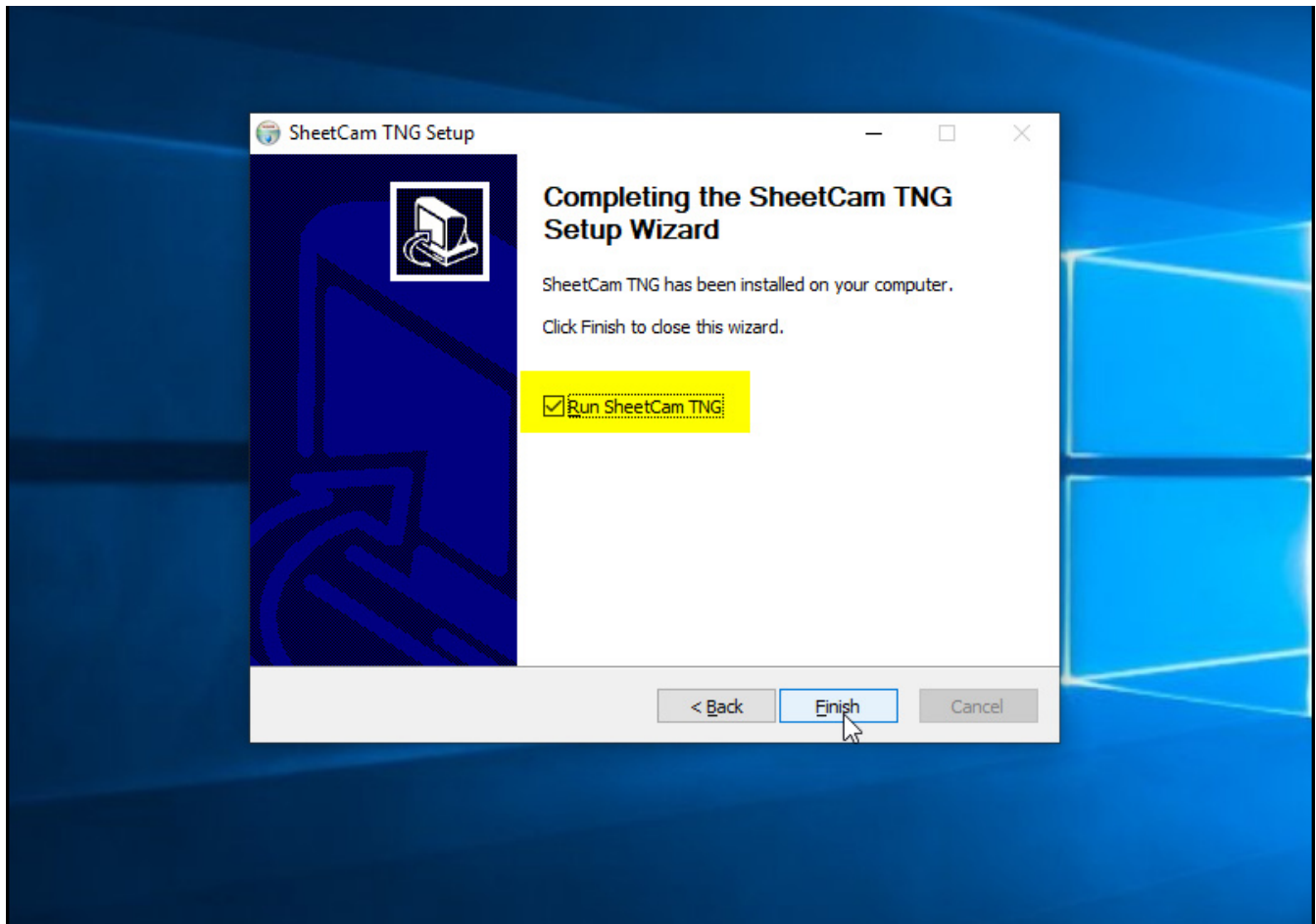
- Click "**Yes**" to begin the installation.



- 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.

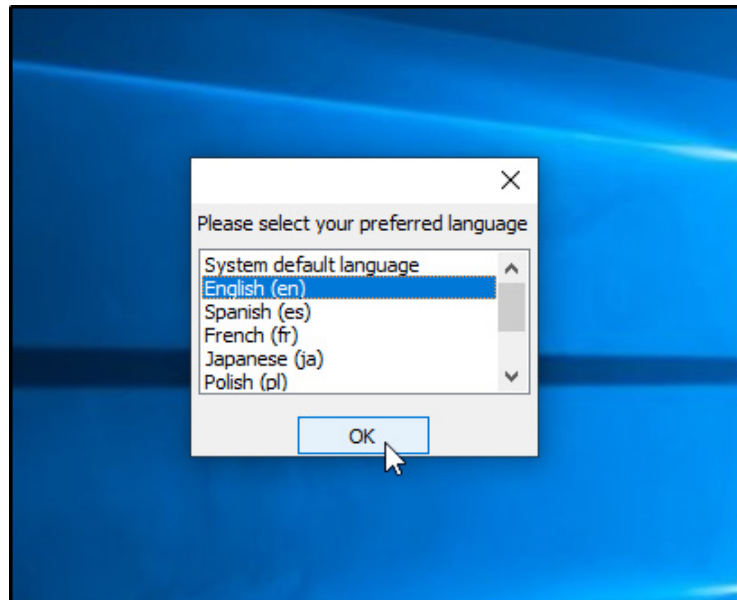


- Select the "Run SheetCam TNG" option and finish the SheetCam install.
- Continue with the setup and license procedure in the next section.

## 2. Install License File

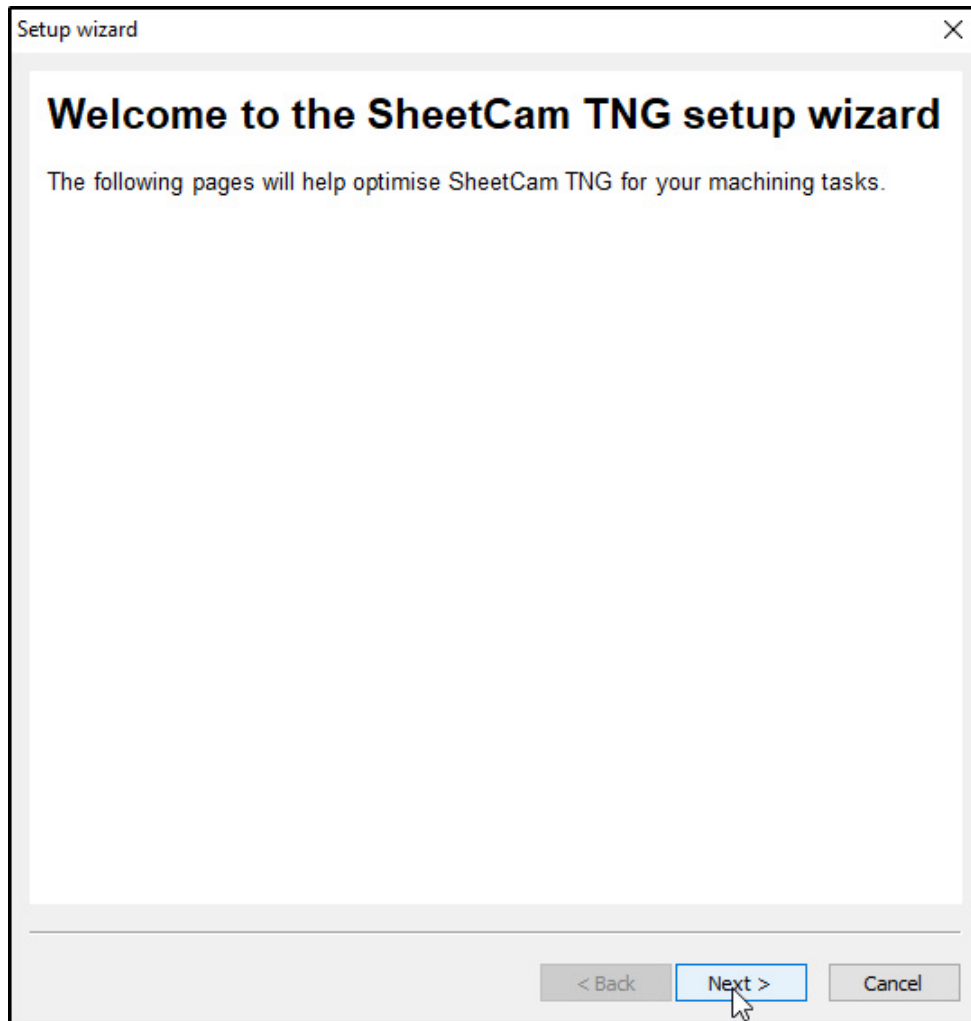
### Software Setup Note

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 <https://www.avidcnc.com/sheetcam-plasma-cam-software-p-445.html>.

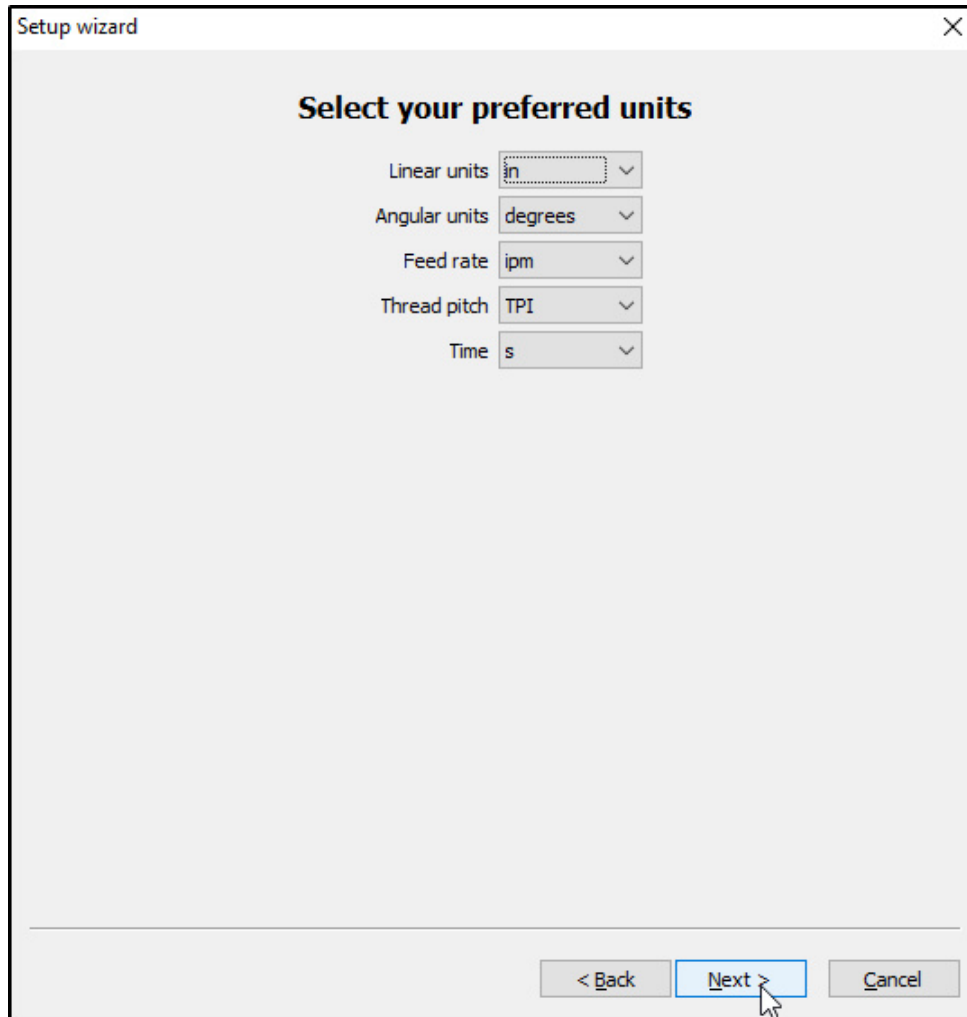


- Select your default language.

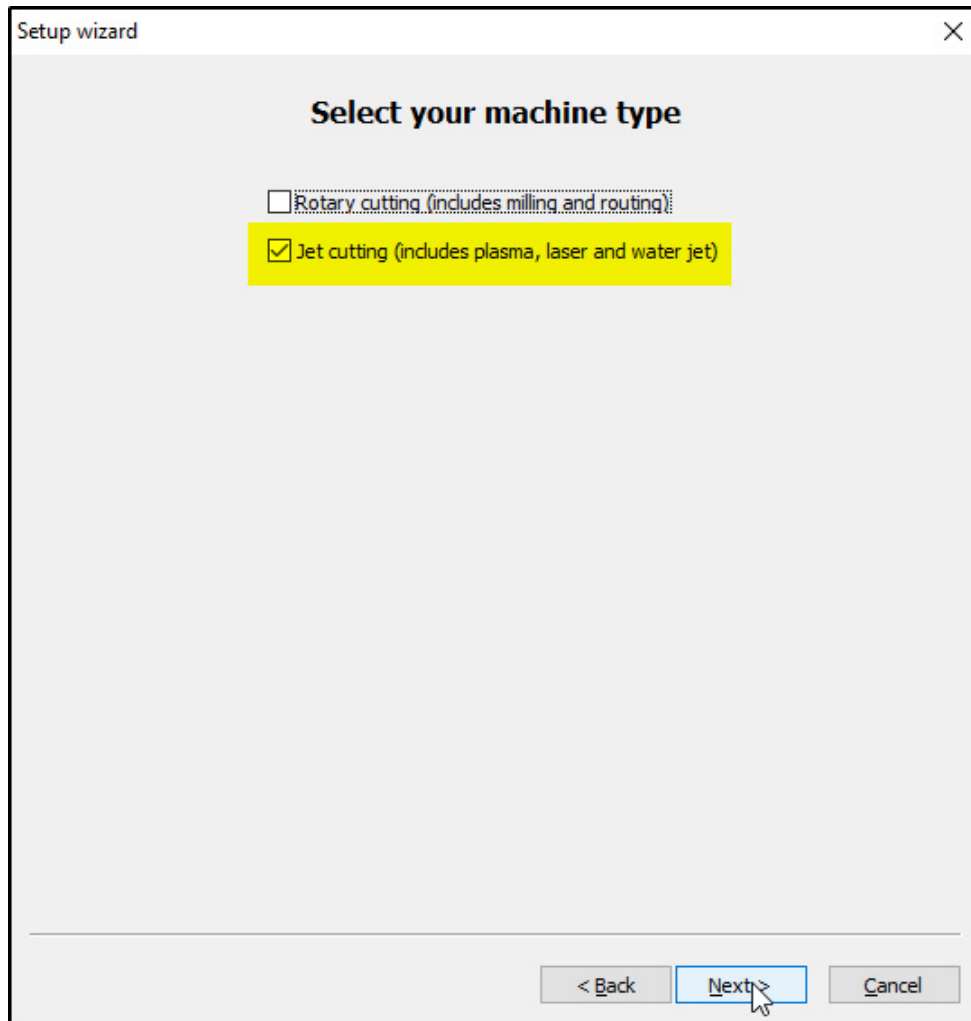




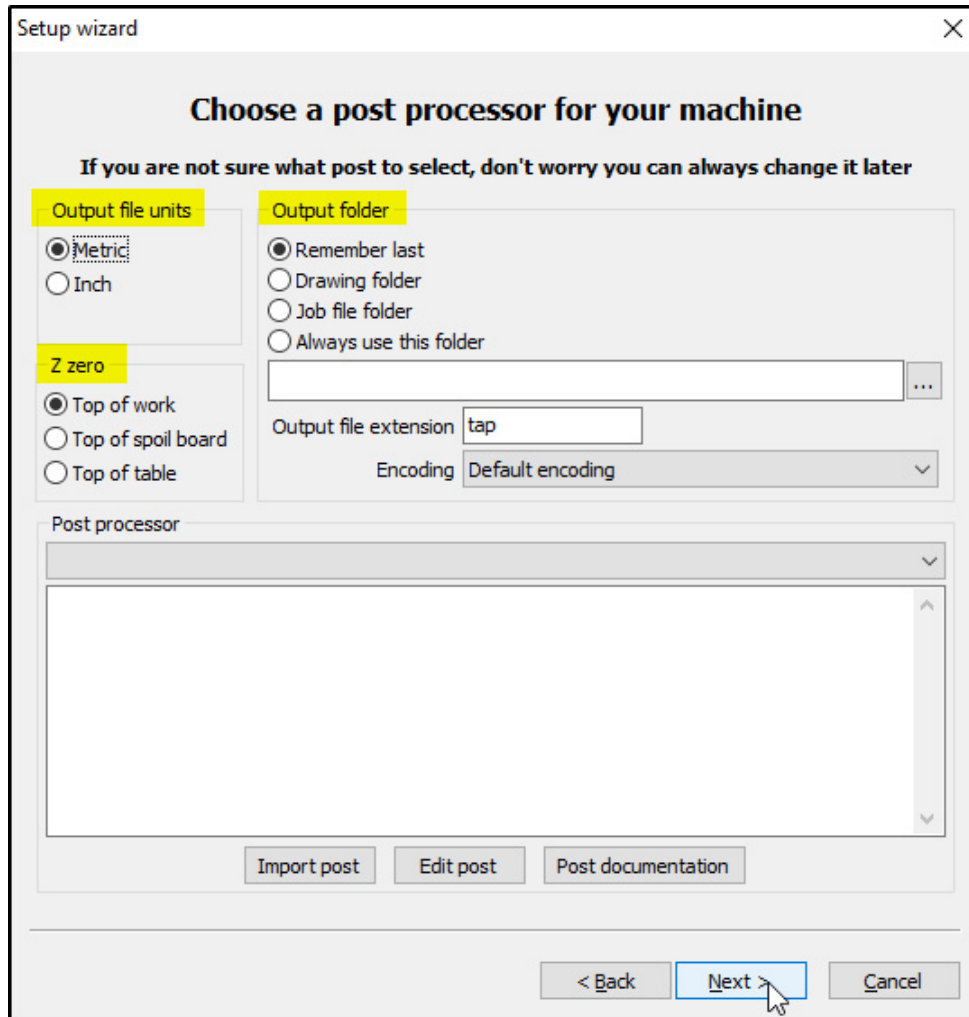
- Use the "SheetCam TNG setup wizard" to setup SheetCam for plasma cutting.



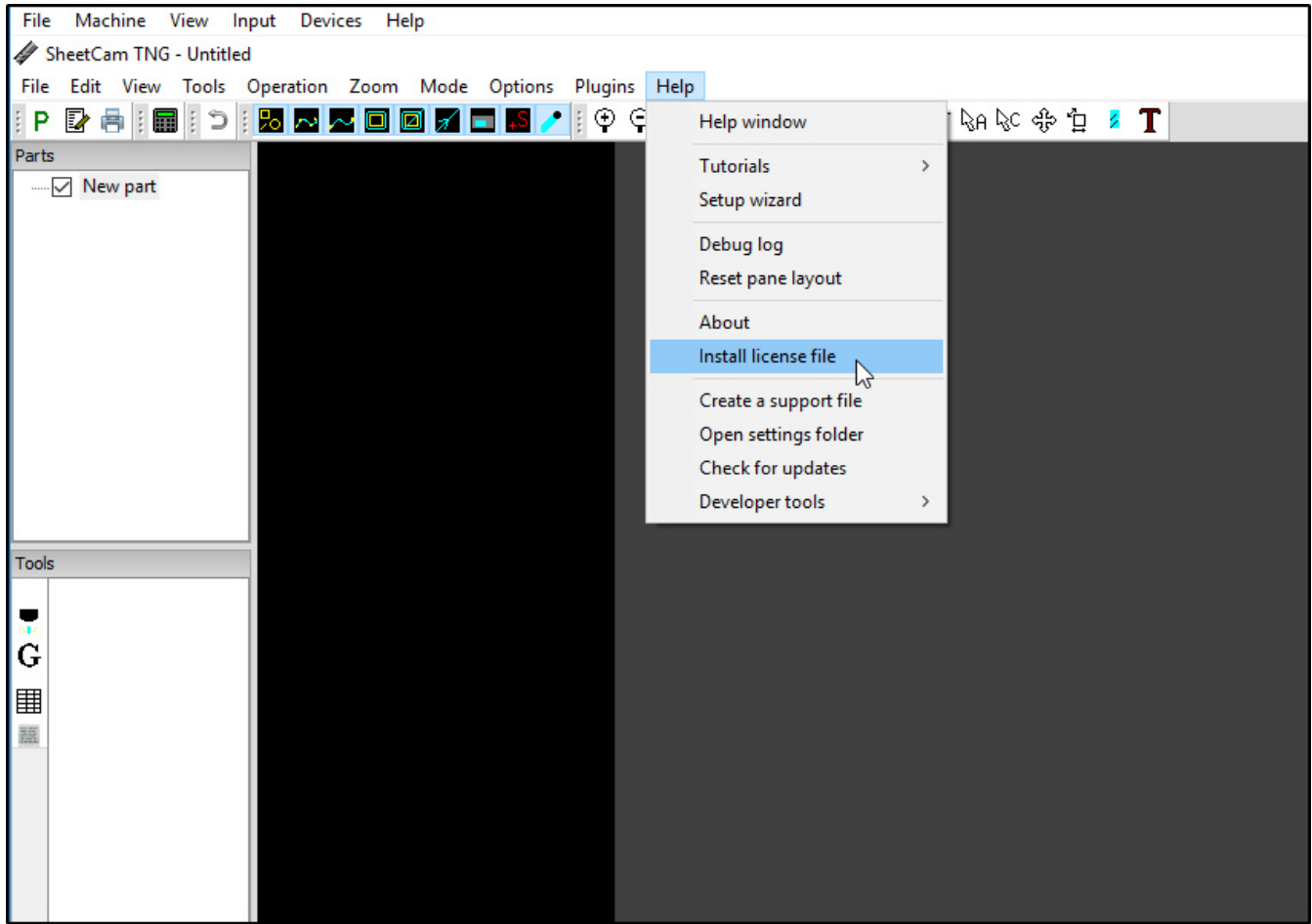
- Select your preferred units.



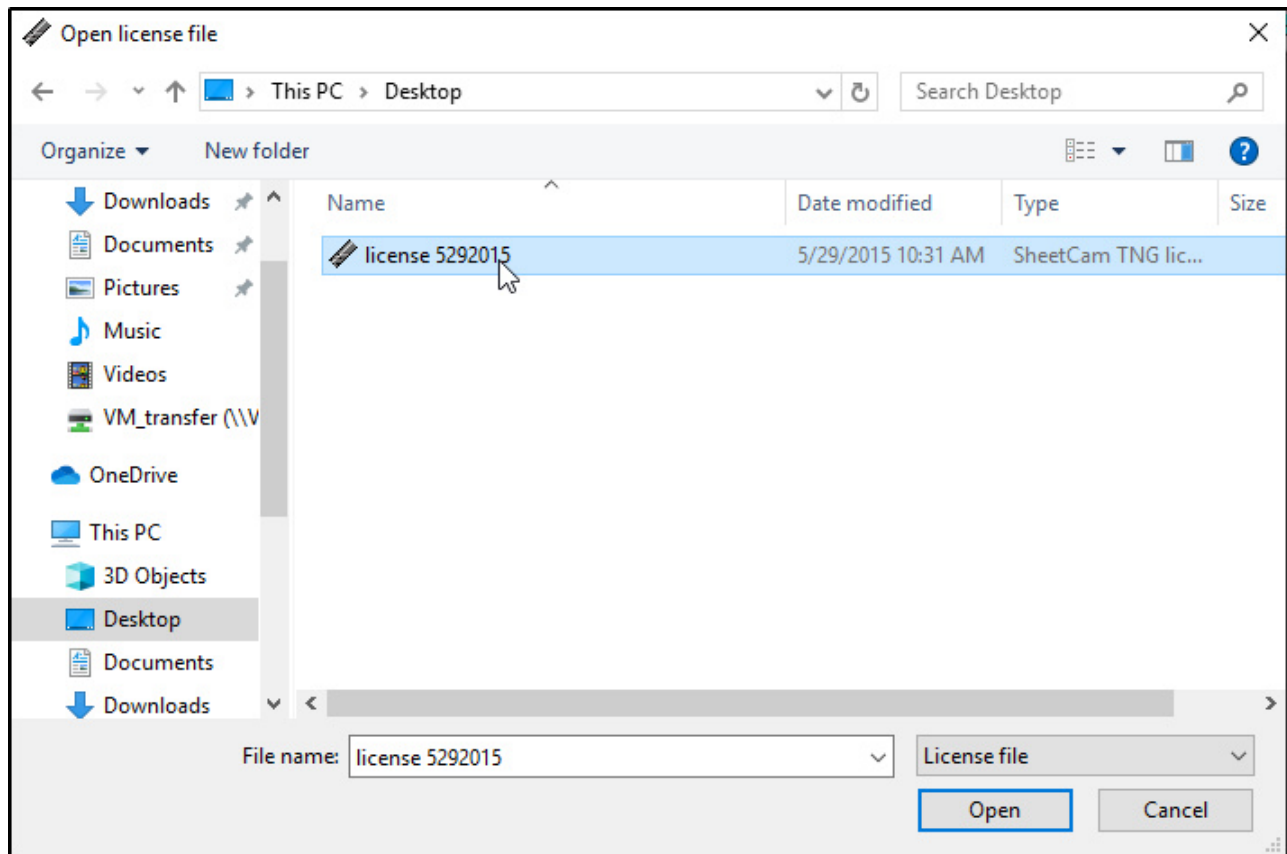
- Select "**Jet cutting**" machine type for plasma machines.



- 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.

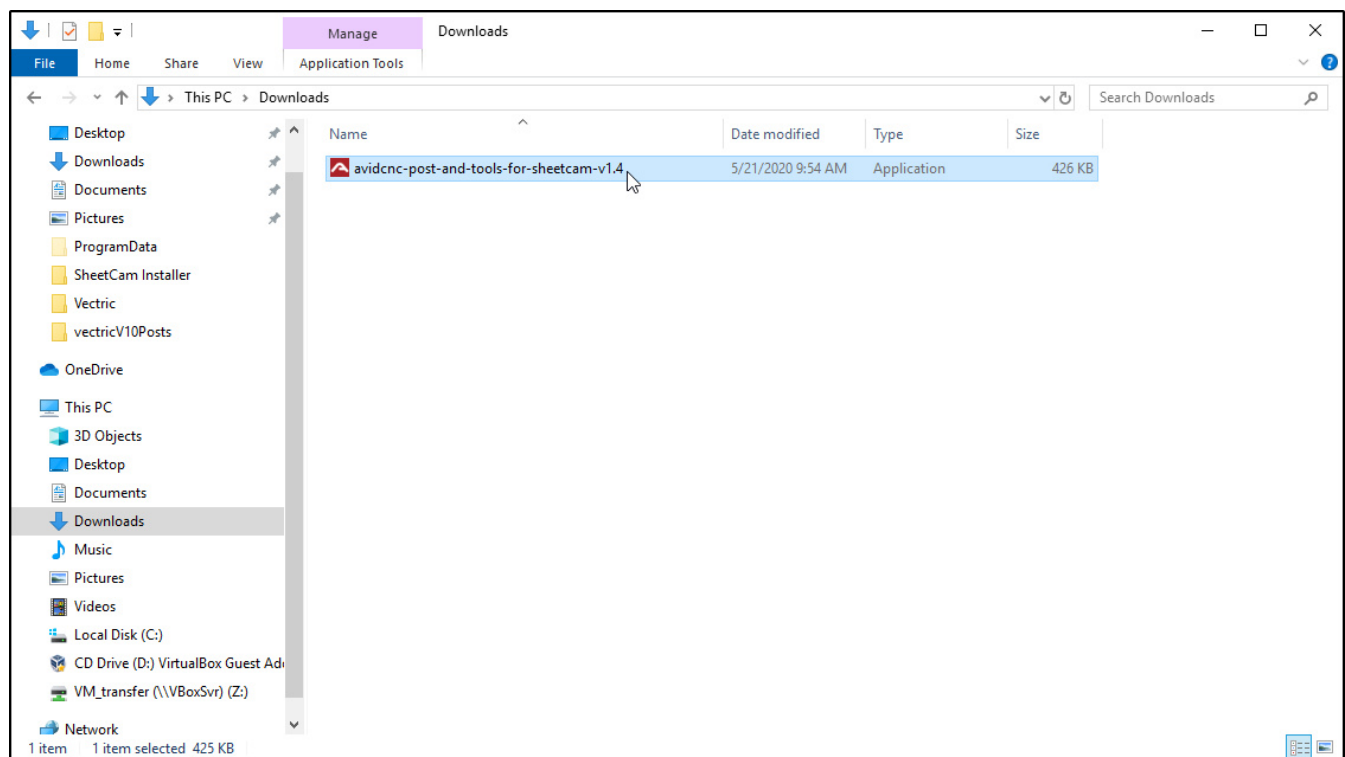
(<https://www.avidcnc.com/support/instructions/software/downloads/sheetcam>)

#### SheetCam Usage

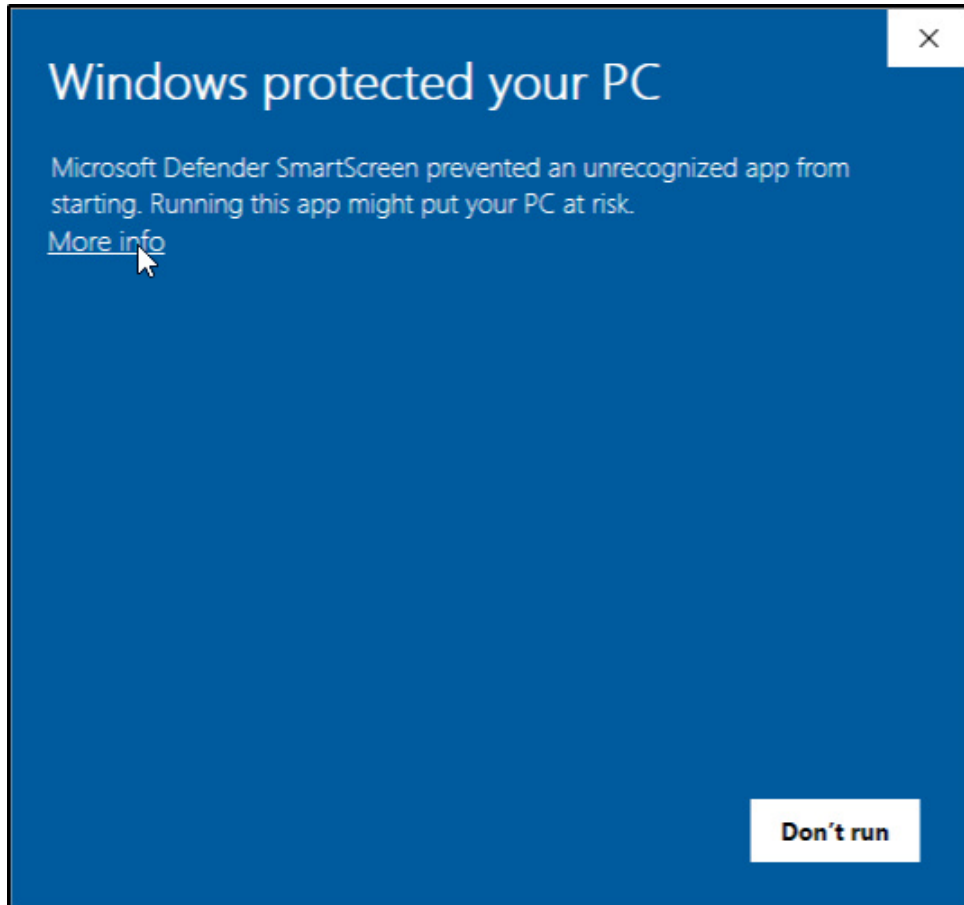
If you will not be using the Avid CNC tool tables, you will need to [Contact Us](https://www.avidcnc.com/contact_us.php) ([https://www.avidcnc.com/contact\\_us.php](https://www.avidcnc.com/contact_us.php)) for instructions to add the required SheetCam code snippets and path rules.

#### Installation Note

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

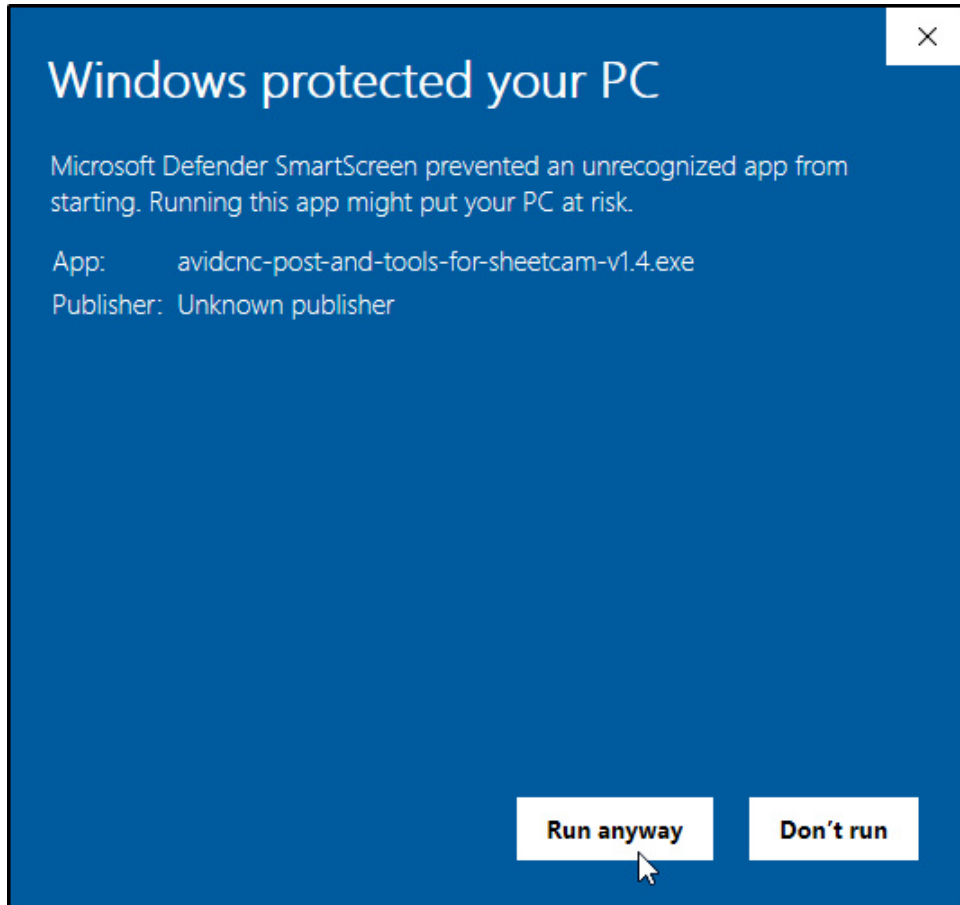


- Run the "avidcnc-post-and-tools-for-sheetcam" installer you just downloaded.

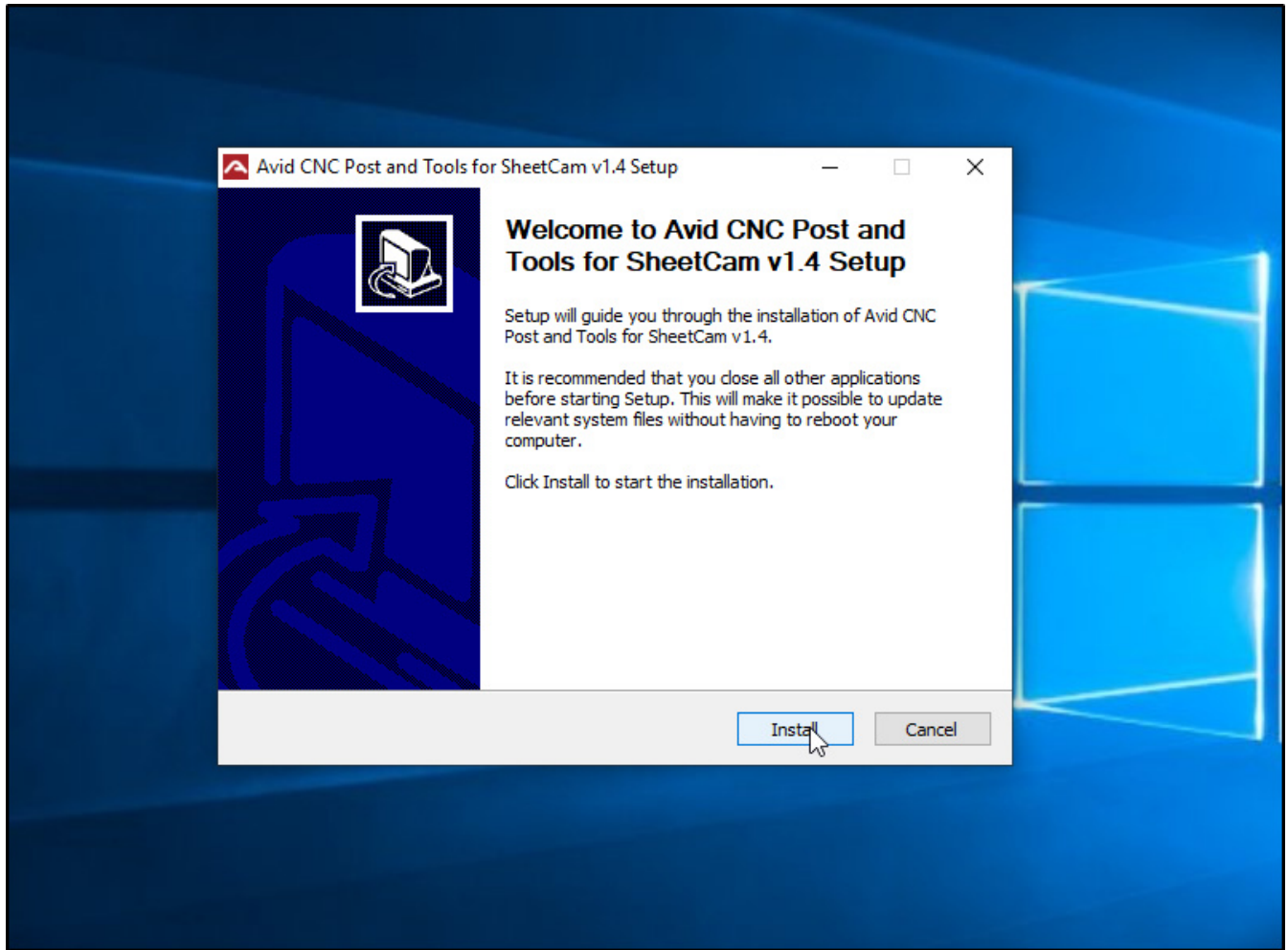


- If the warning message pictured above appears, click "**More Info**".

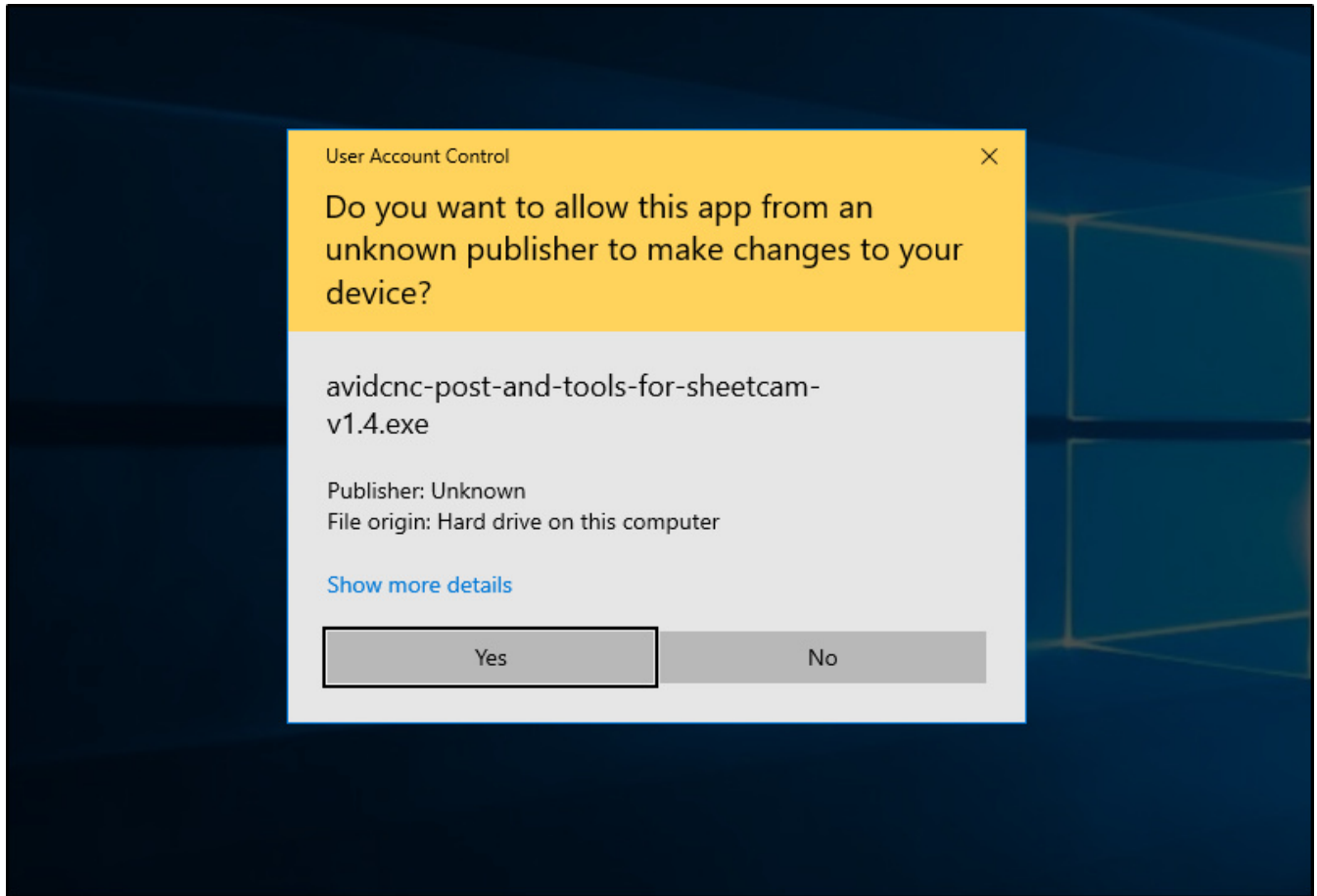




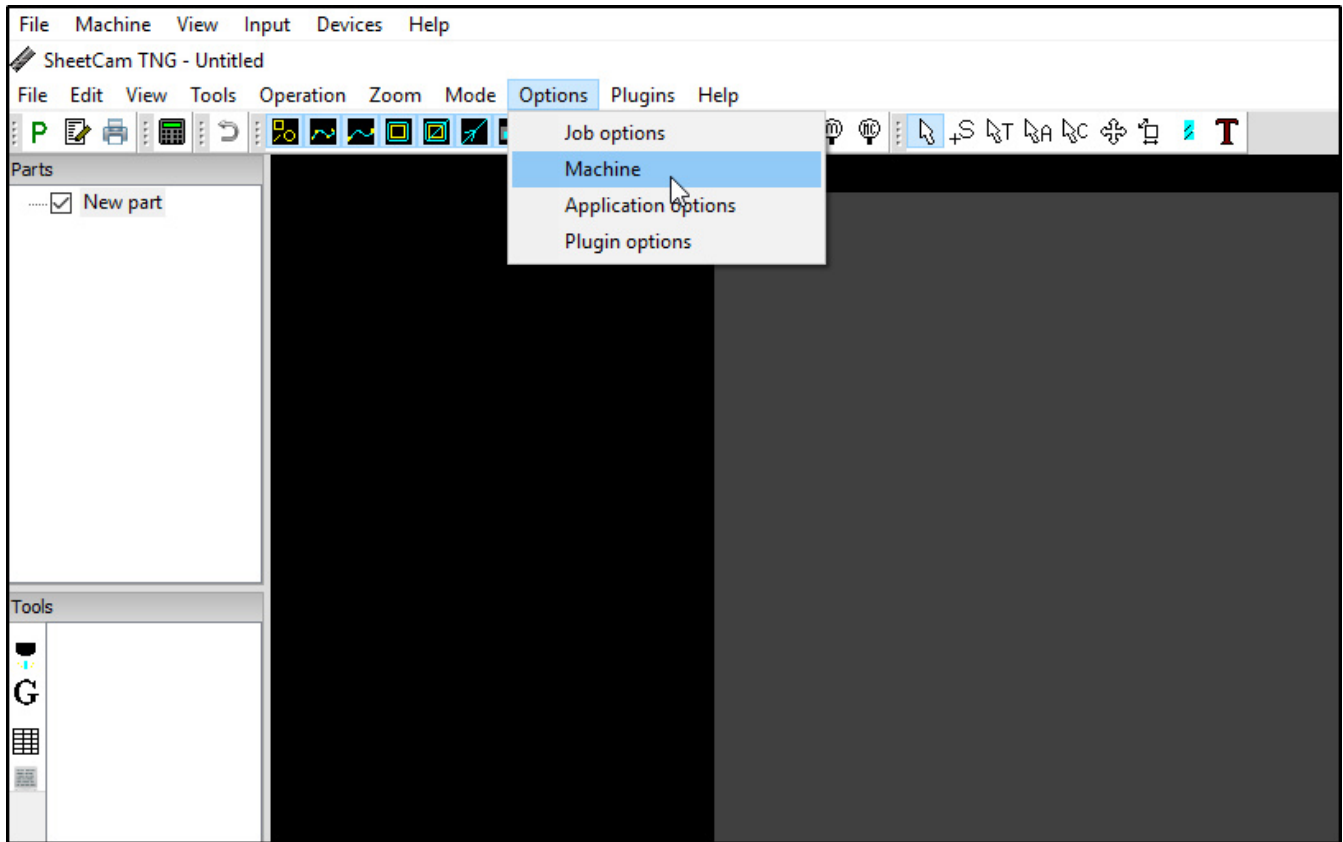
- Click "**Run anyway**".



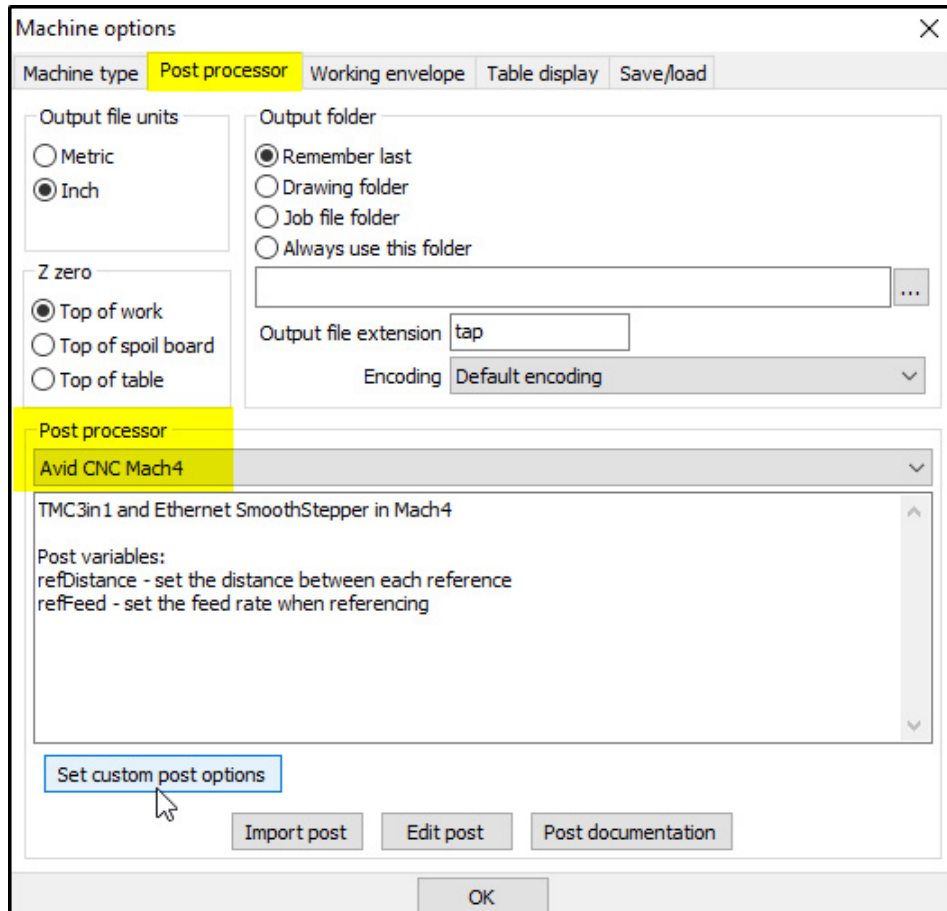
- Click "**Install**" to begin the installation.



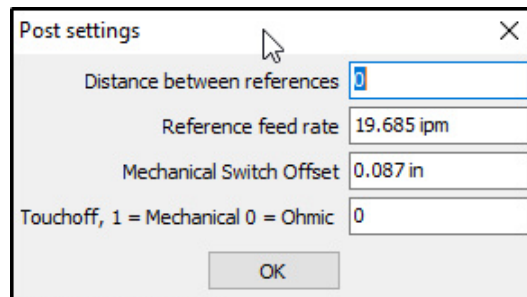
- Part way through the install, you'll be asked about making changes to your device. Click "**Yes**" to allow the plasma tools files to be installed correctly.



- Once the finished the installation, open SheetCam.
- Navigate to "**Options > Machine**".

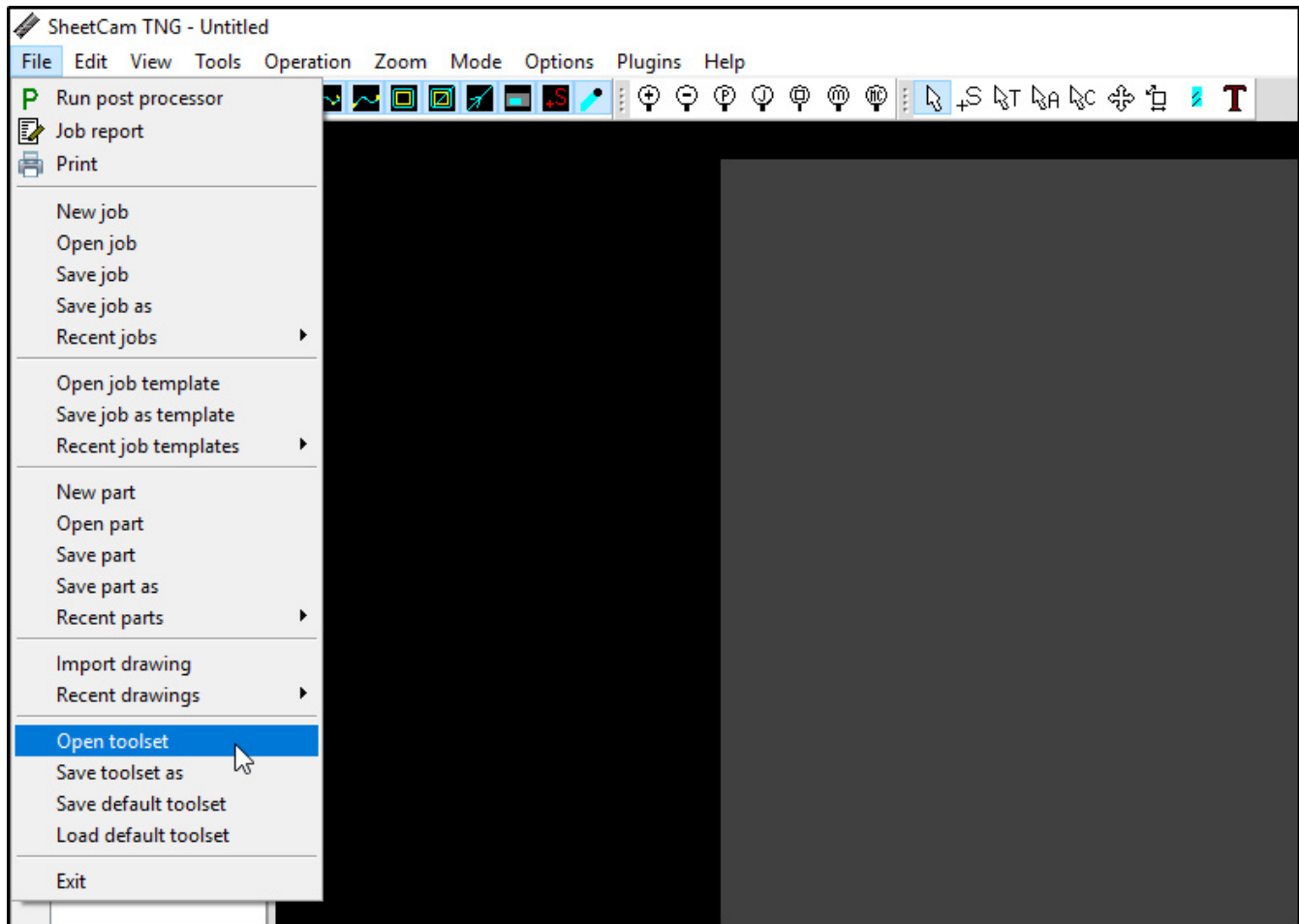


- On the "Post Processor" tab, verify "Avid CNC Mach4" is the current post processor.
- At the bottom of the screen, select "Set custom post options".

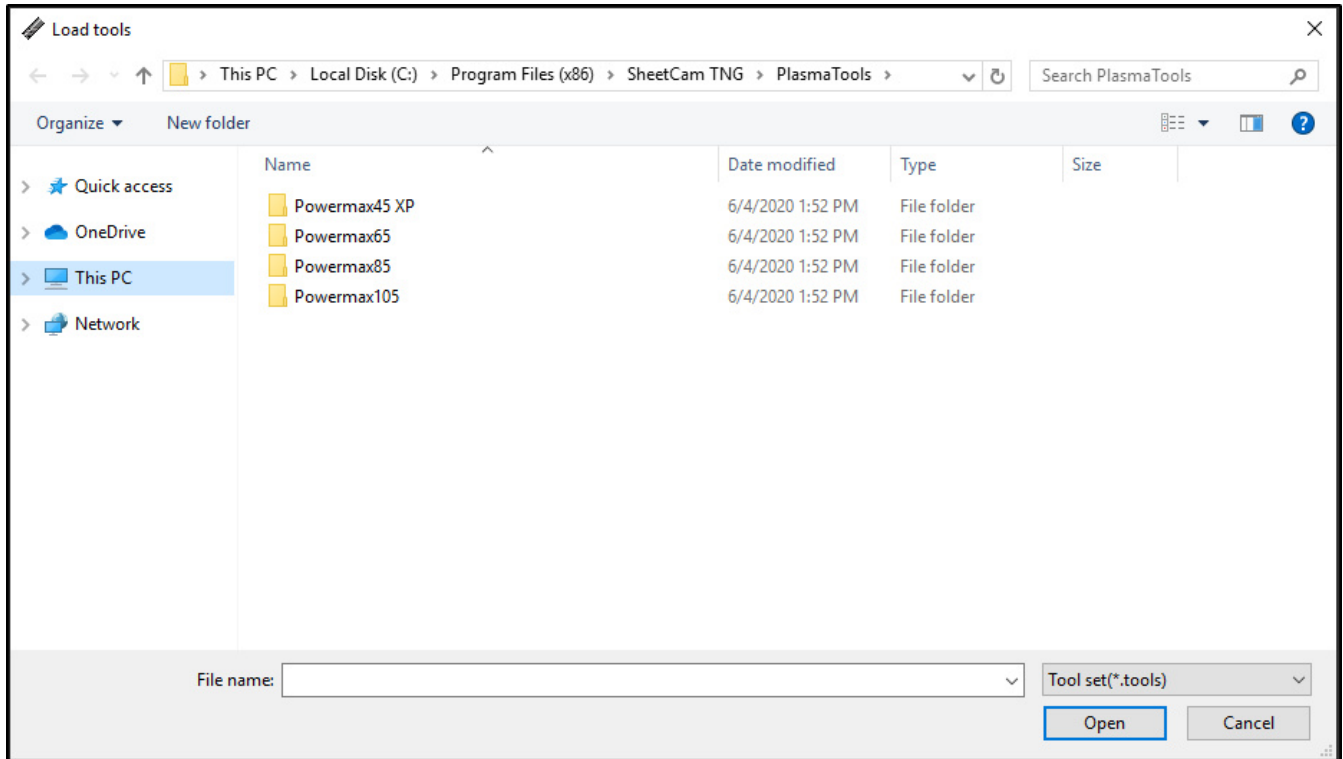


You will see four different options for customizing the post processor.

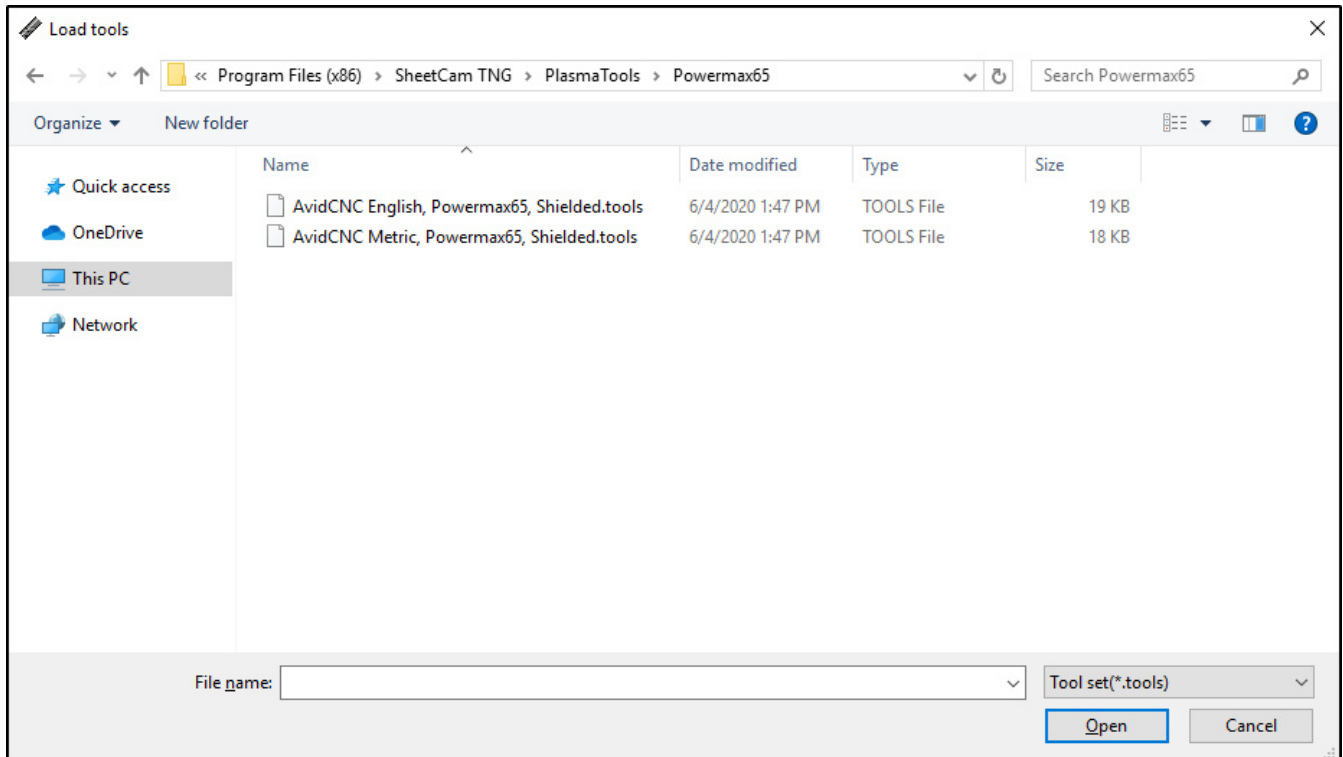
- **Distance between references:** Distance between pierces where a second probe move (reference) will not take place. The default value of "0" will reference before all pierces.
- **Reference feed rate:** Feed rate for probing reference moves.
- **Mechanical switch offset:** Applicable only for mechanical touch offs, this is the travel of the switch plunger before the probe signal is activated.
- **Touchoff:** Select if mechanical touch off (set value to 0) or ohmic probe touch off (set value to 0) is to be used.



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

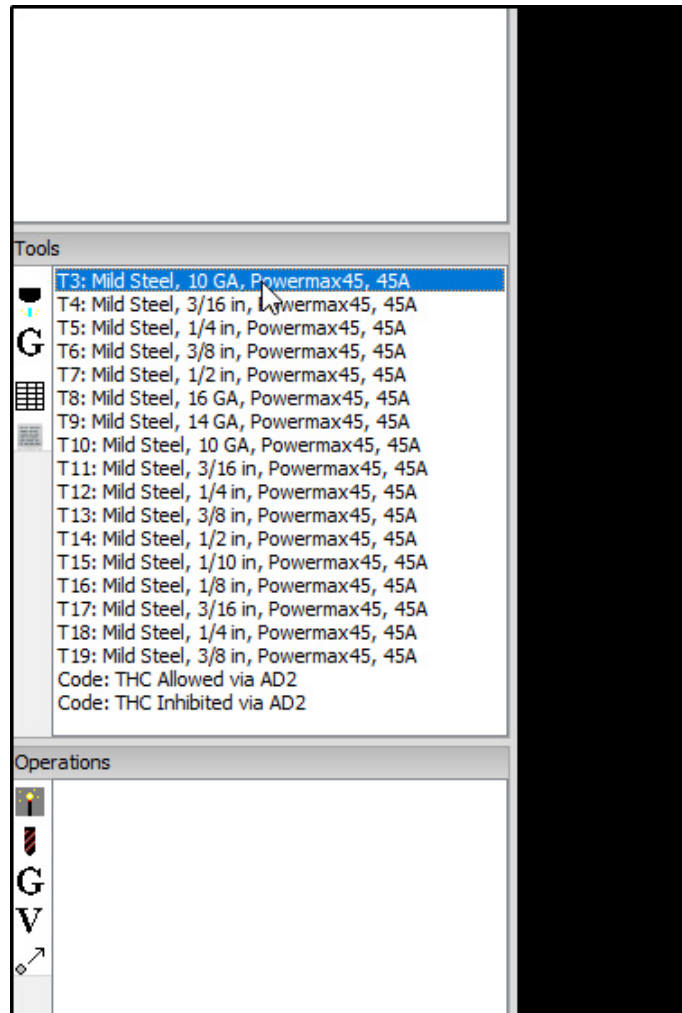


- Open the folder for your specific Hypertherm plasma system.

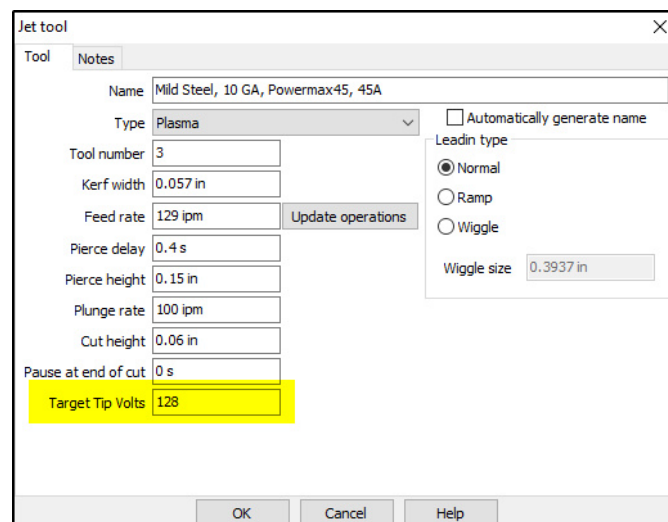


- Select the appropriate toolset for your desired units, english or metric.





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

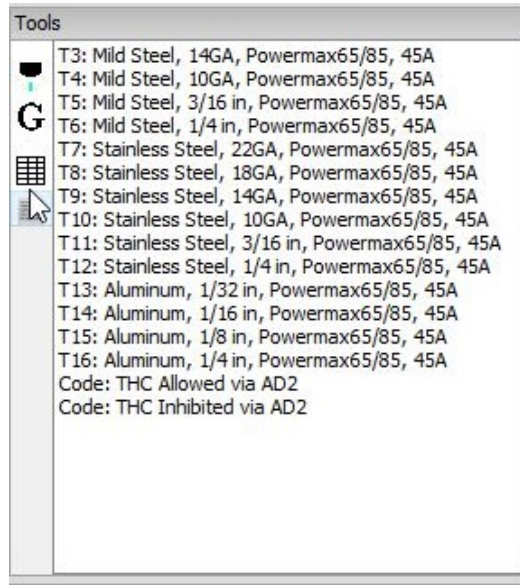


- You will see the tool is pre-populated with the values from the Hypertherm cut charts.

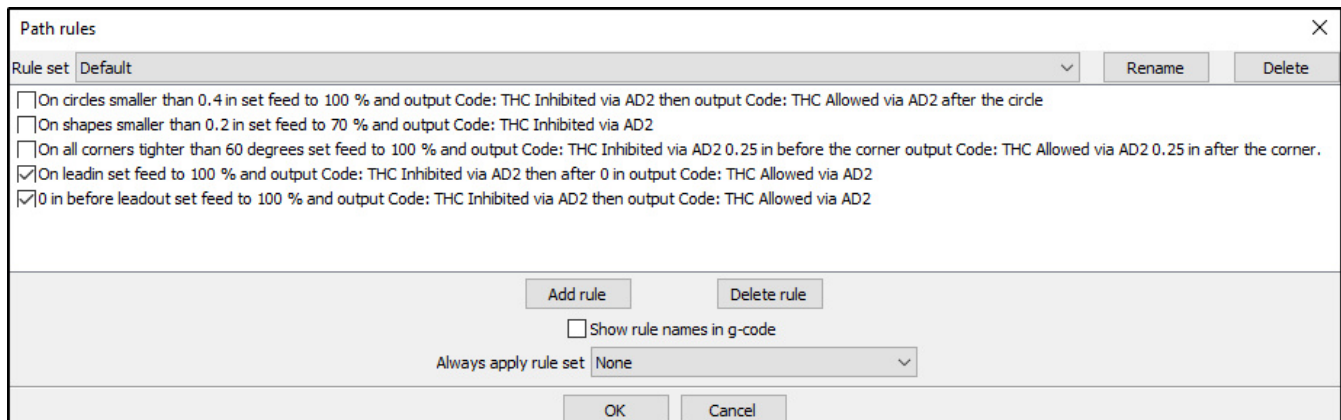
- At the bottom you will see the "**Target Tip Volts**". Using the AvidCNC Mach4 post processor, this value will be included in your GCode. Once loaded into Mach4 the target tip volts will automatically be populated when the GCode program is executed.



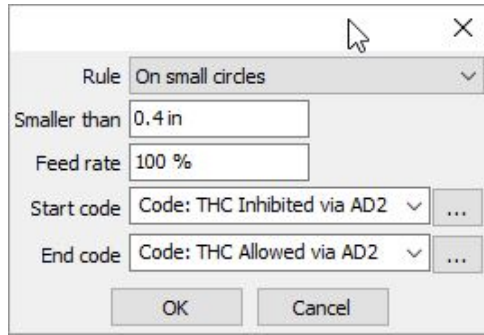
## 4. Using Cut Rules



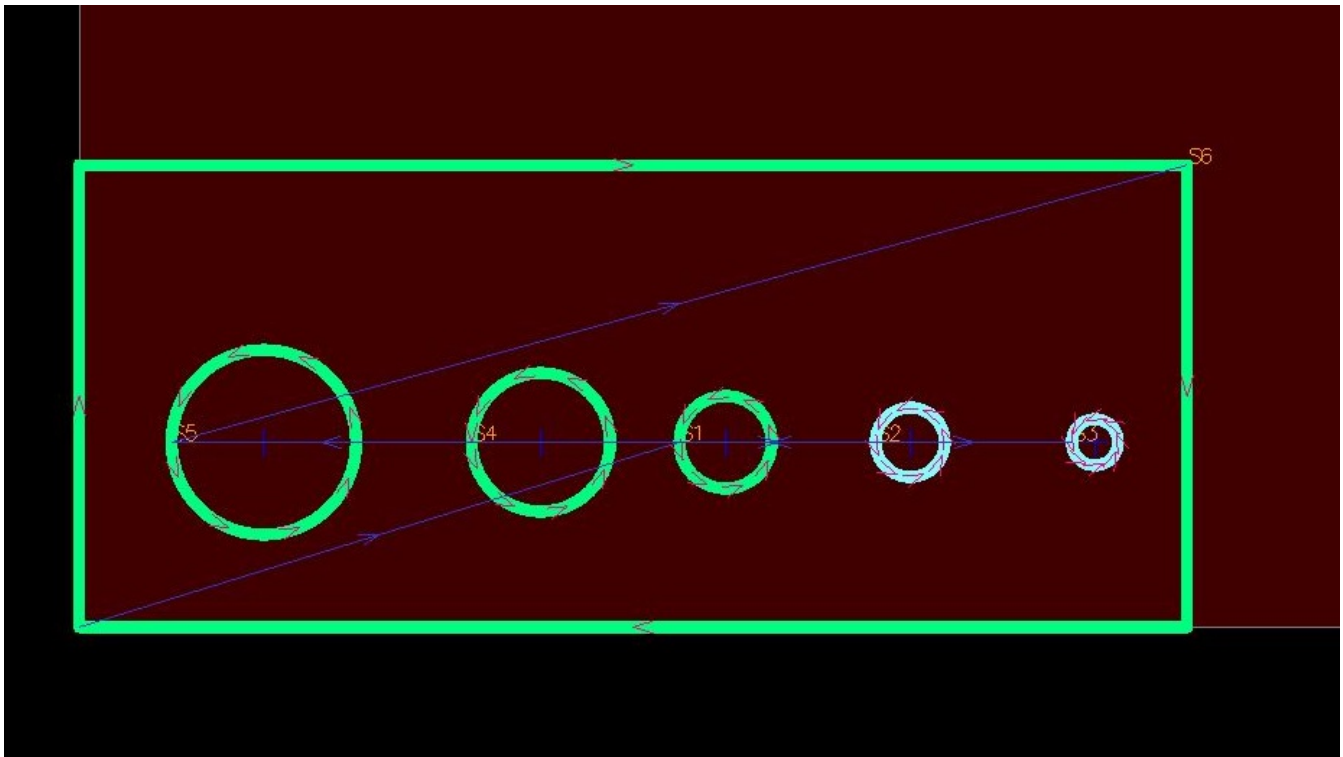
- To open the cutting rules editor press the Cut Rules button on the left of the Tools menu on the left of the SheetCam application window, or use the drop down menu Tools -> Cutting Rules.



- The Path Rules window will then be displayed. The default rule set will have been installed by the "AvidCNC for SheetCam" installer. By default the only path rules which will be enabled turn THC motion off during lead in and lead out moves. 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.



- 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.