

Supporting Software

Resources for some of the programs you might use in tandem with Mach.

- [SheetCAM Post Processor](#)
- [SheetCAM Training Information & Tutorials](#)
- [Wiggle Lead In \(SheetCAM\)](#)
- [SigmaNEST Oxy or Plasma Process Selection](#)
- [SigmaNest Access Violation Message](#)
- [Post Processors for CAM Software](#)
- [SheetCAM Path Rules](#)
- [Audible Light Stack Solution](#)
- [SheetCAM Freezing on Startup](#)
- [iCCP: known incorrect sRGB profile](#)

SheetCAM Post Processor

Description of the problem:

What are the correct post files for use with SheetCAM?

Solution:

Machine Type	Post File Name
Router	Mach3.scpost
Mach3 Plasma/Oxy Fuel	MachMotion Plasma THC Version4.scpost
Water Jet	MachMotion WaterJet.scpost
Mach4 Plasma	MachMotion Mach4 Plasma.scpost

Router post is included in SheetCAM installation.

To update the SheetCAM Post Processor:

1. Save the post to your computer
2. In SheetCAM, select the options menu item, then machine options
3. Select the post processor tab
4. Click on import post button
5. navigate to the post you save in step 1 and click open
6. click ok button in the machine options dialog box

Keywords: 7684

SheetCAM Training Information & Tutorials

Tips/Notes:

- Using the % character in the filename can cause SheetCAM to crash/freeze when importing.

Links to SheetCAM Resources:

Install:

<http://sheetcam.com/Downloads/akp3fldwqh/SheetCam%20TNG%20setup%20V6.0.19.exe>

Manual:

http://www.sheetcam.com/Download/SheetCam%20TNG%20Manual_A4.pdf

Support Forum:

<http://forum.sheetcam.com/>

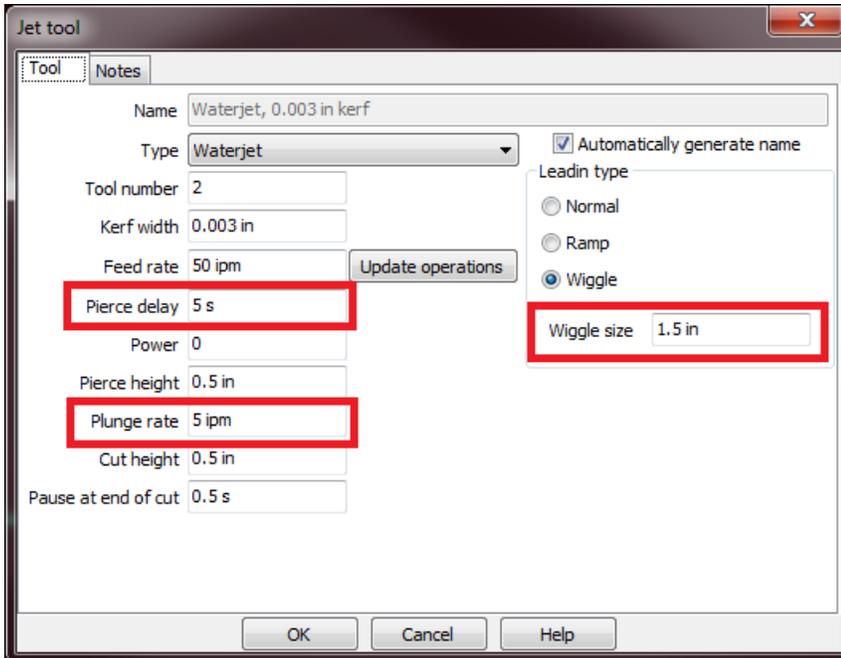
Video Tutorials:

1. <http://www.youtube.com/watch?v=hk7WNZ-erY8>
2. <http://www.youtube.com/watch?v=ODyK90hh3YE>
3. <http://www.youtube.com/watch?v=ue8BtibfeKk>
4. <http://www.youtube.com/watch?v=dekJVgIKCgl>

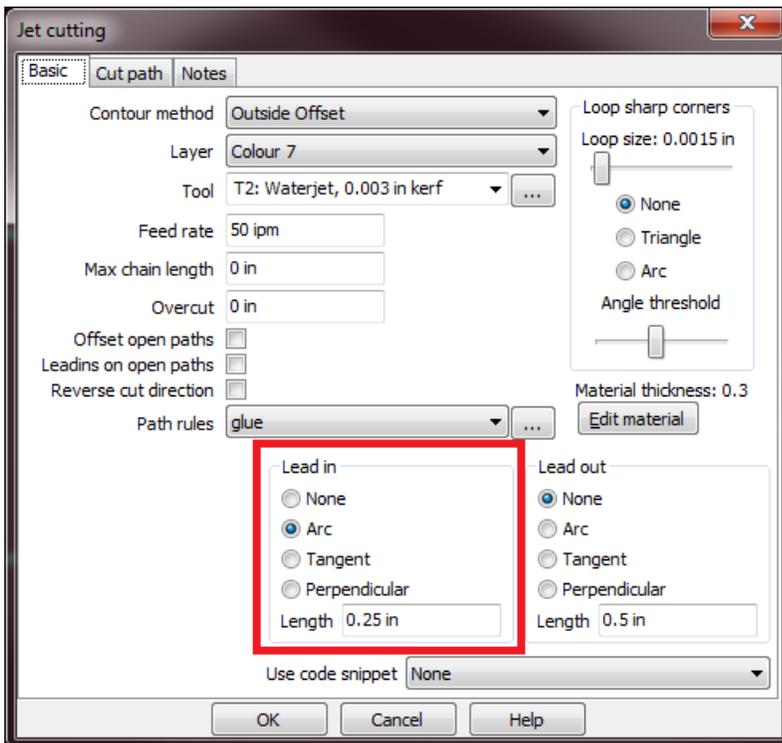
Wiggle Lead In (SheetCAM)

Here are a few screen shots of menus from sheetcam and an explanation of how to set up a wiggle/oscillation:

The number of wiggles is calculated from the pierce time, feed rate and wiggle length.



You also need some type of lead in defined in the operation.



Attached is an example file with wiggle lead in (attachment on right side)

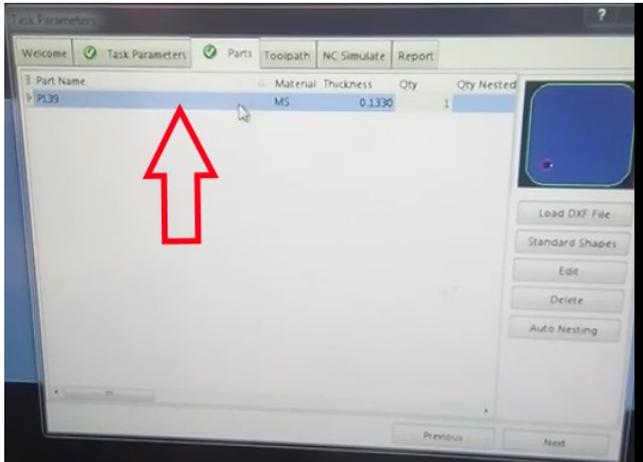
Keywords: 13479

SigmaNEST Oxy or Plasma Process Selection

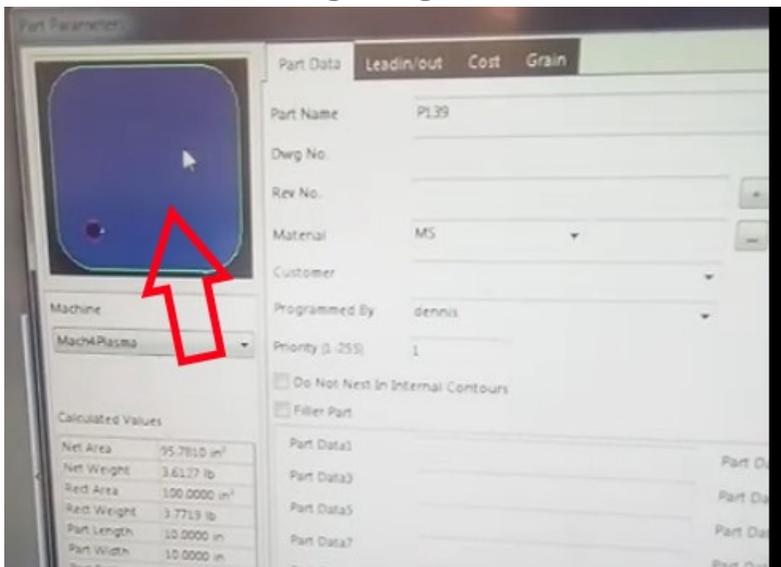
SigmaNEST is set to use tool 1 (plasma) as default process. You can change to OXY per part or set as default in the post.

1. Change process for the part:

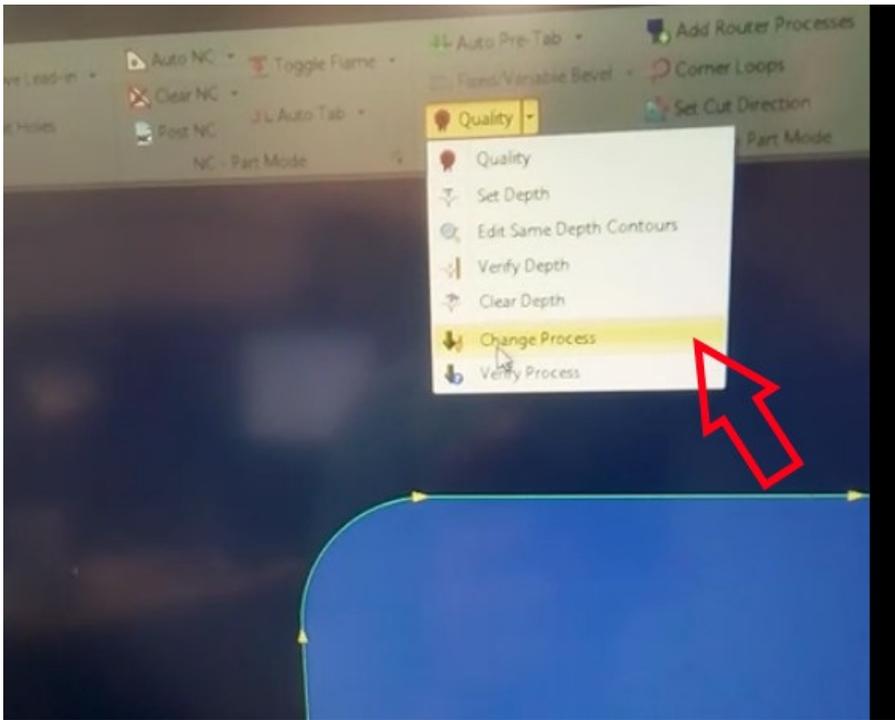
Double click on the part:



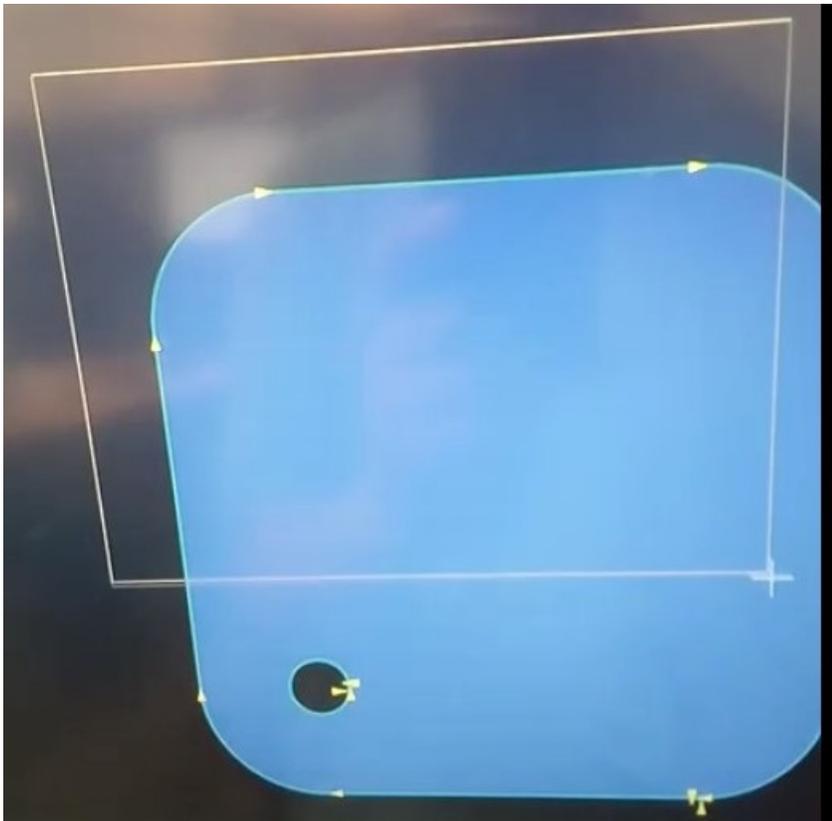
Double click on the design image:



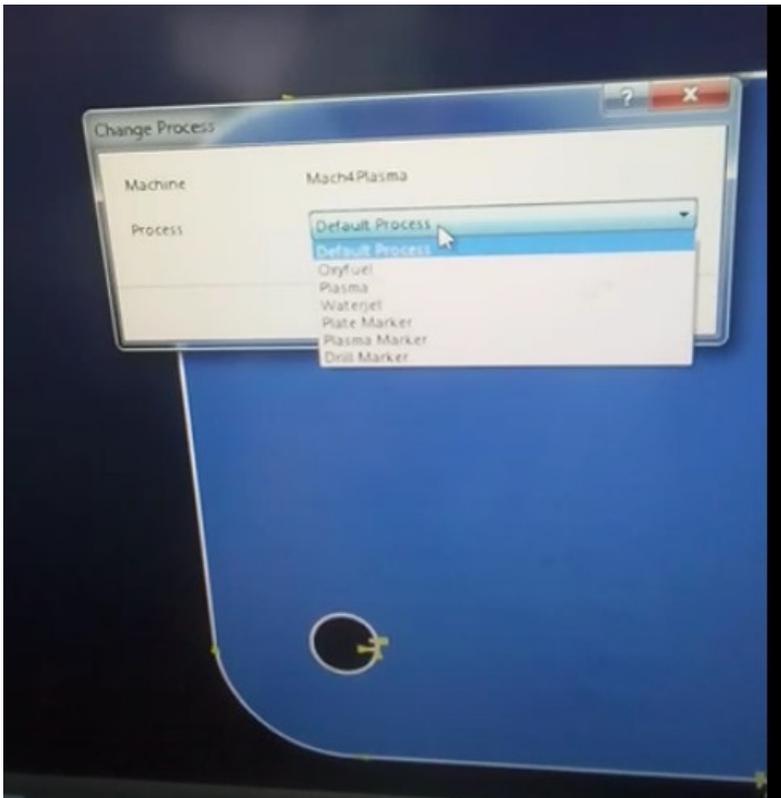
Click the "Quality" dropdown menu option, then select "Change Process":



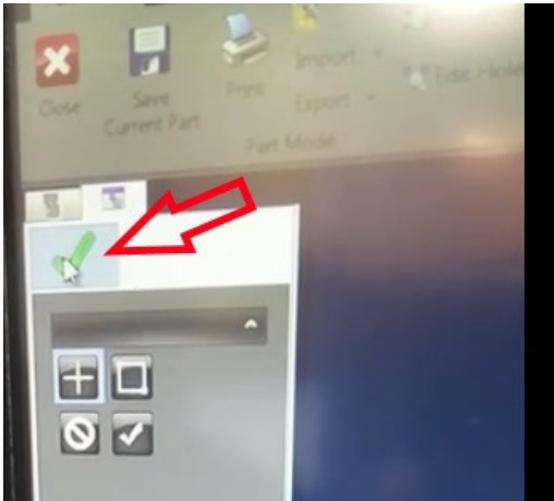
Click-hold, and drag to expand the box over the area to be changed (note in image below the entire shape has not yet been selected):



Once part selected (above), right-click in the part and a window will come up to change the process. Use the dropdown to make your selection:



Once the selection has been made (above), click the ok button, then over in the right side of the window, click the green check mark. You can then click the close menu option:



Changes are now applied and will be posted once you re-nest and generate gcode.

2.0 Change Default Process in Post

This change is made in the post file. For example:

C:\Mach4\Post\SigmaTEK\Mach4\MachMotion\MachMotion.pst

Look for "processes available" section (around line 200). there is a default tools number. change the number to match the default tool wanted from the list below that line.

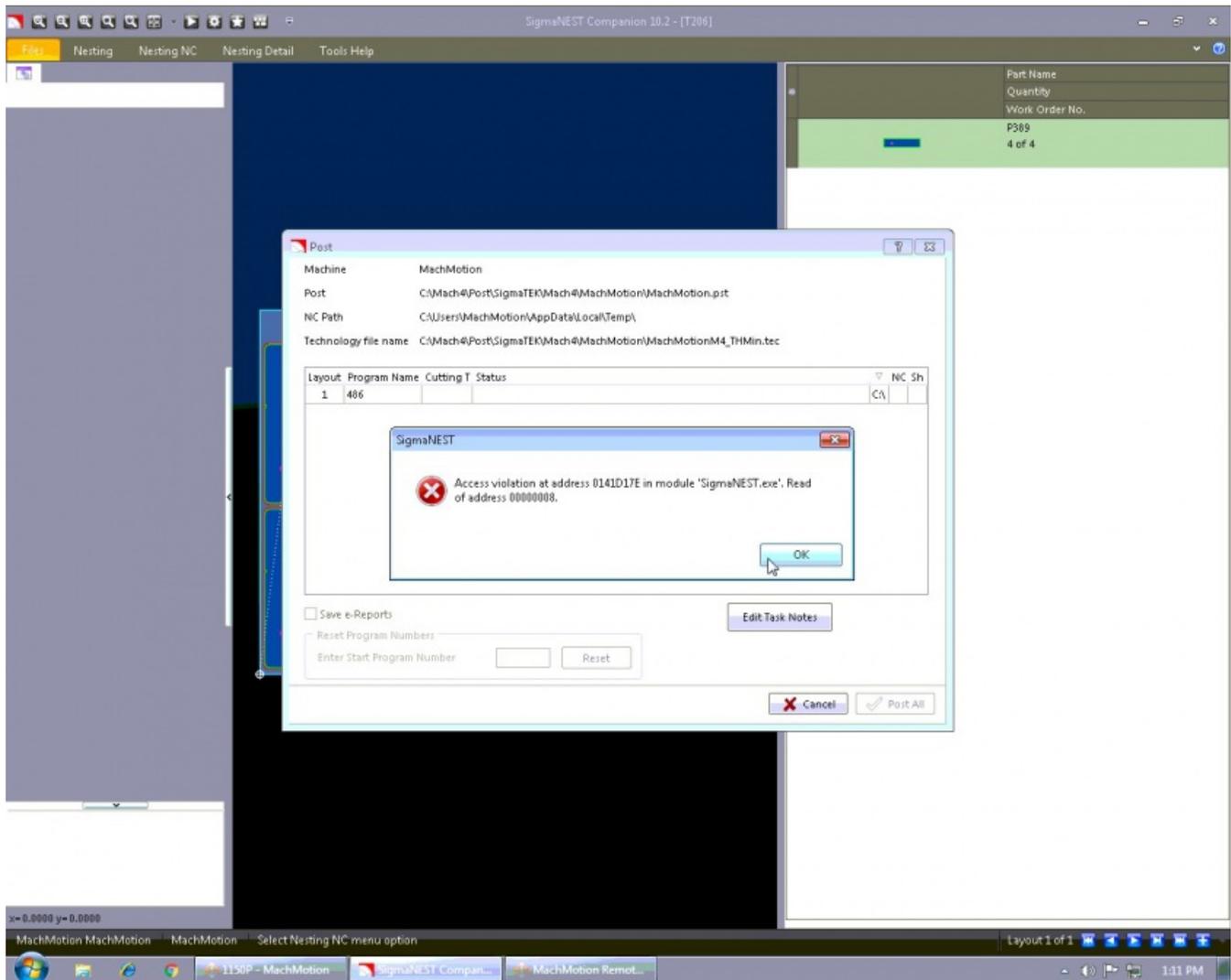
Keywords: sigmanest, oxy, torch, plasma, default, process

SigmaNest Access Violation Message

This error message seems to happen when the same file is loaded into the Mach4 software.

Recommendation is to make sure that all gcode files are closed out of the Mach4 software and try again. If still having the issue, save the SigmaNEST file then restart SigmaNEST and Mach software (ensure the mach software does not autoload the same gcode file, if so then close it out).

If still having issues, contact SigmaNEST Support (513-595-2002), or contact MachMotion support and we can work them. Internet access to the control will be needed. Had an instance where they updated some read/write privileges to paths/folders/directories that sigmanest uses.



Keywords: 15369

Post Processors for CAM Software

Overview

Our Mach software runs gcode and can be in various file types such as .txt, .nc, and .tap.

Our software does not convert the drawings such as DXF into gcode. A CAM software package is needed for that. For many of our systems, SheetCAM or SigmaNEST Companion is used.

If you are not using our CAM options you will need a post-processor from your CAM software vendor to output into g-code format for Mach. MachMotion does not provide the post processor but this page should help assist you in obtaining the needed post. Machmotion also offers a service of updating your current post-processor. Please contact us for more details and pricing information.

Common CAD CAM Packages

CAM Package	Description	Cost
Fusion 360	Basic	Low
BobCAD	Basic	Low
MasterCAM	Advanced 3D Options	Medium
SigmaNest	3D Options Amazing Nesting	High

Integrating to a Post

Many companies already have the needed post so you can contact them directly and see. Some posts are available already at Mach software vendor's site:

<https://www.machsupport.com/downloads-updates/cam-post-processors/>

The following documents can help your CAM vendor determine the fanuc style g-code needed:

<https://support.machmotion.com/books/software/page/mach4-g-code-and-m-code-reference>

Lathe: <https://support.machmotion.com/books/software/page/mach4-lathe-g-code-and-m-code-reference>

Attached to this page are example g-code files that can help them as well. The THC test file is for Plasma.

Some common items are:

Mist can be turned on with M7 and Flood/Coolant with M8. M9 turns these off.

Spindle on FWD is M3, REV is M4. M5 turns it off. S command sets the speed: M3 S500.

M6 is for tool change with a T for the tool number to use: M6T3 for example.

Outputs 0-5 can be turned on and off with M-codes. See the following table for reference:

Custom M-Codes	Functions	Default Output
M200	Output 0 on	Y2
M201	Output 0 off	
M202	Output 1 on	Y3
M203	Output 1 off	
M204	Output 2 on	Y4
M205	Output 2 off	
M206	Output 3 on	Y5
M207	Output 3 off	
M208	Output 4 on	Y6
M209	Output 4 off	
M210	Output 5 on	Y7
M211	Output 5 off	

Machine Type Requirements

Mach3 Waterjet and Mach4 Plasma require the following:

- Absolute IJ Arcs (For example, "G90.1 G03 X1.0192 Y2.0162 I1.0192 J1.4866")
- Absolute positioning (G90)

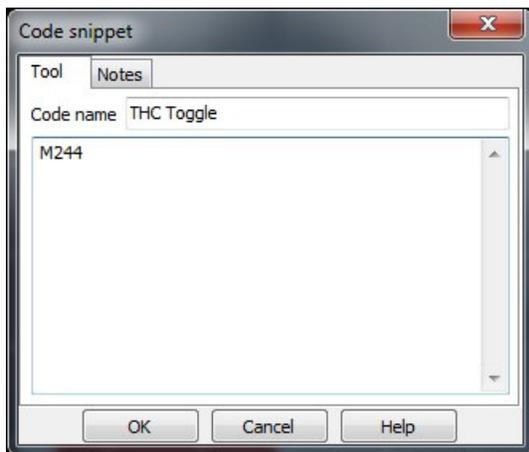
SheetCAM Path Rules

SheetCAM path rules can be setup for things like turning off THC control on small parts and slowing down on corners.

Creating Code Snippets:

Note: The info below is for enabling/disabling the torch height control. You can create custom code in macros to use for other purposes but that is beyond the scope of this document.

1. Attached is M244.m1s file and should be saved to the c:\mach3\macros\plasma\ directory (the path may be a little different depending on your setup and profile name). If M244 already exists in your directory, change the attached file name to an available number and use that new number for the rest of the setup where we have 244. The code in this file simply toggles the state of the THC button on the plasma screen, effectively turning THC control on and off.
2. Under tools menu option, click the G code option.
3. Set the Code Name and enter the filename of the code to run in the box below, without the extension:



4. Click OK button

Creating Path Rules:

Rules to prevent diving (Turn THC Off) and Rounded Corners:

Under the options menu is job options. In there select the path rules tab and you can edit those rules parameters for your liking. The idea is they turn the torch height control off at certain points to prevent it from diving, or slow down the feedrate for corners to prevent rounding.

- **Rule: On lead out**

Feed rate: 100%

Start Code: Code: THC Toggle

Distance Before: .2 in

End Code: Code: THC Toggle

- **Rule: On small shapes**

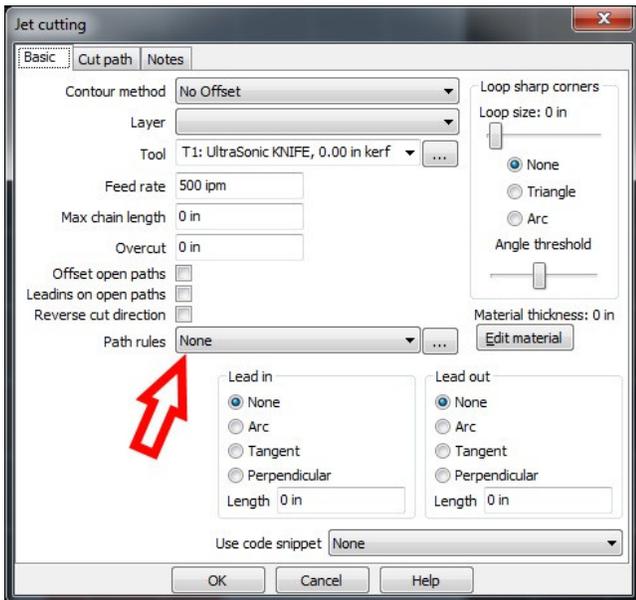
Smaller than: 2 in
Feed Rate: 75%
Start Code: Code: THC Toggle
End Code: Code: THC Toggle

• **Rule: On all corners**

Tighter than: 45 degrees
Distance Before: 0.2 in
Feed Rate: 100%
Before Code: (nothing)
Distance After: 0.2 in
End Code: (nothing)

Using Path Rules

In order for the path rule to be used, it must be selected in the operation in sheetcam. Double click on the existing operation or when setting a new operation, select the Path rule to be used in the dropdown menu:



Audible Light Stack Solution

Suggestion is they get a lightstack with audible option. The sheet below has info on that. They can goto/call automation direct for more.

<https://cdn.automationdirect.com/static/specs/patlite40mmLR4.pdf>

<https://cdn.automationdirect.com>

Or

If they want to use their lightstack and add audible, then they could go with this:

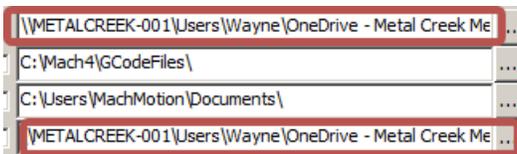
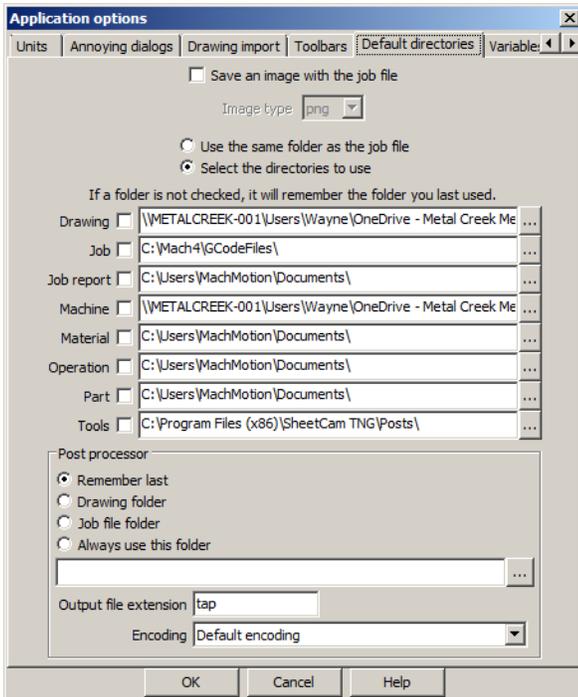
https://www.automationdirect.com/adc/shopping/catalog/pushbuttons_-z-_switches_-z-_indicators/22mm_plastic/audible_buzzers/ecx2071-24r

But then they have to add Mcode to turn the output on to their post processor.

SheetCAM Freezing on Startup

If SheetCAM gets stuck on the loading screen it can be caused by a permissions conflict if the customer has mapped to a network drive or external PC.

Start Sheet CAM and wait, it could take up to 5 minutes to load. Once the application loads, navigate to application settings and look for any file locations that might be mapped to an external location.



Locate this location on the network, it most likely will display this when you attempt to access it:

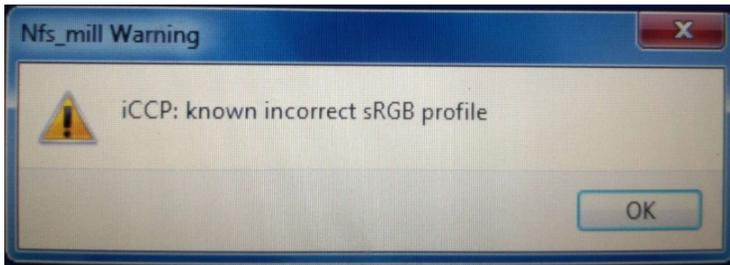


Have the operator log into this and it should fix the issue.

iCCP: known incorrect sRGB profile

Description of Issue

Issue occurs when a PNG file was created using a different module of libpng than the library that is used to read/display it. Watch for it when using mixed version software, older software on a newer OS, and custom image files from customers. A warning box will display when the bad image is attempted to be used. Generally does not affect the ability to run, but will show up every time the image is accessed and can be quite annoying.



Description of Solution

Image can be fixed using free software [ImageMagick](#). When installing, be sure to install legacy tools. The following command line can be used to fix image: `magick convert image.png image.png`