Fusion 360 Post Processor For PlasmaC

Introduction

For those interested in using Fusion 360, here is a post processor that is designed to work with PlasmaC. I have done some testing and it seems to work but there might be cases where it does not. Please use with caution and report any problems and I will try to fix them. If you have limited experience using Fusion 360, please see the tutorials online as this guide is focused on using the PlasmaC post processor only and will require you to have some basic knowledge of Fusion 360. All the pictures are from Fusion 360 running on a Mac. The Windows version will look different but should have the same options. As far as I know, there is no Linux version of Fusion 360.

The Plasma C post processor was created from the existing Fusion 360 LinuxCNC post processor and the Fusion 360 Jet Template. This post processor is specific to PlasmaC in that it does not create any Z movements. I have added features to improve cutting small holes (roughly defined as less than 37mm). Hypertherm's Jim Colt and others have posted numerous times about ways to improve small hole cutting and I have incorporated three of these ideas that seem to get a lot of mentions. First, slow the cutting speed. In PlasmaC, this also has the added effect of suspending torch height adjustment. Second, the lead in for the cut starts at the center of the hole and then proceeds in an arc to the perimeter of the hole. Third, once the hole is cut, the torch turns off and continues to move along the hole perimeter another 4mm.

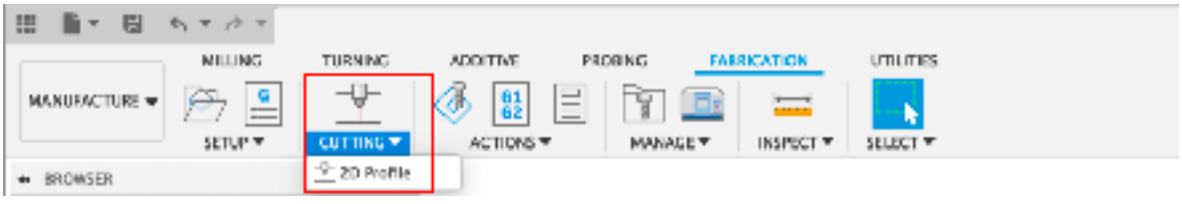
Another addition to the post processor that makes it specific to PlasmaC is the ability to use the built-in PlasmaC hole center punch function. The post processor does not require you to specifically locate the center of any holes; rather, it automatically finds the center point of a selected hole, moves the torch to that point, and then uses the PlasmaC center punch function.

All drawings should be created on the XY plane. The post processor should work with both metric and imperial settings.

The Fusion 360 PlasmaC post processor is used as a personal post processor. Please see https://knowledge.autodesk.com/support/fusion-360/learn-explore/caas/sfdcarticles/sfdcarticles/How-to-add-a-Post-Processor-to-your-Personal-Posts-in-Fusion-360.html for instructions. The post processor file is attached to this post.

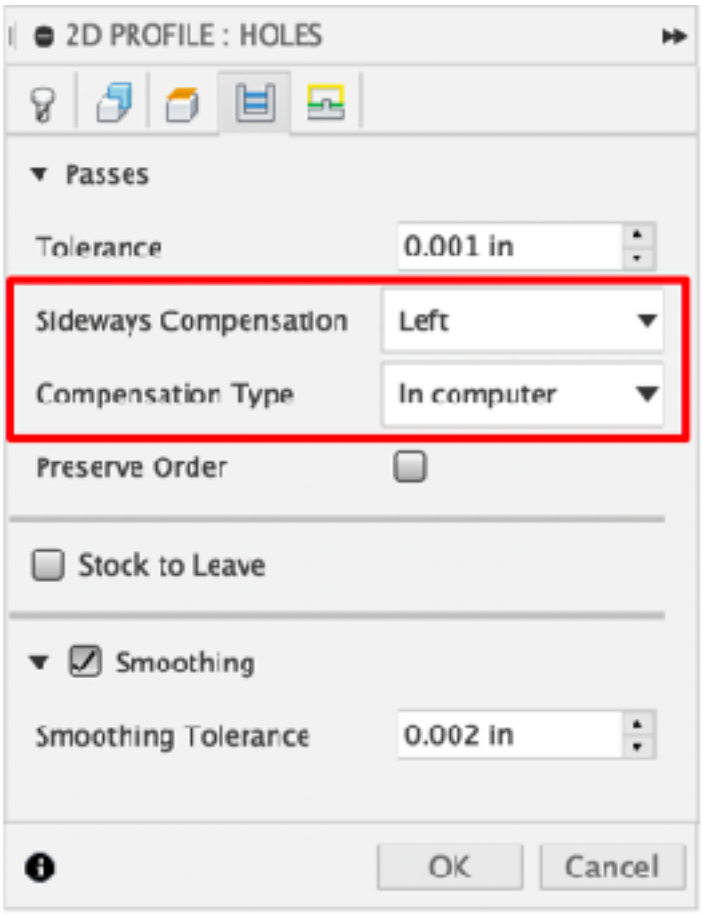
Small Hole Cutting

To access the small hole cutting features, create a new 2D Profile:



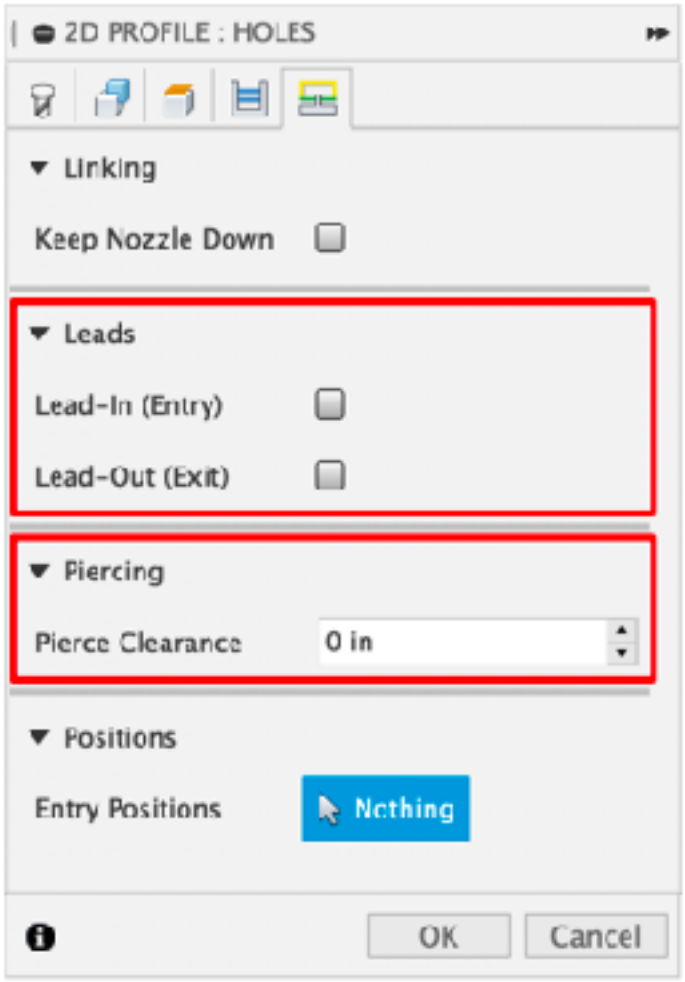
On the Geometry tab, be certain that you select only the holes where you want to use the special features in this 2D Profile. All other cutting should be created in one or more separate 2D Profiles.

Next, you need to set the following on the Passes tab:
Sideways Compensation: Left
Compensation type: In Computer



On the Linking tab set the following:

Lead-In: Unchecked
Lead-Out: Unchecked
Pierce Clearance: 0



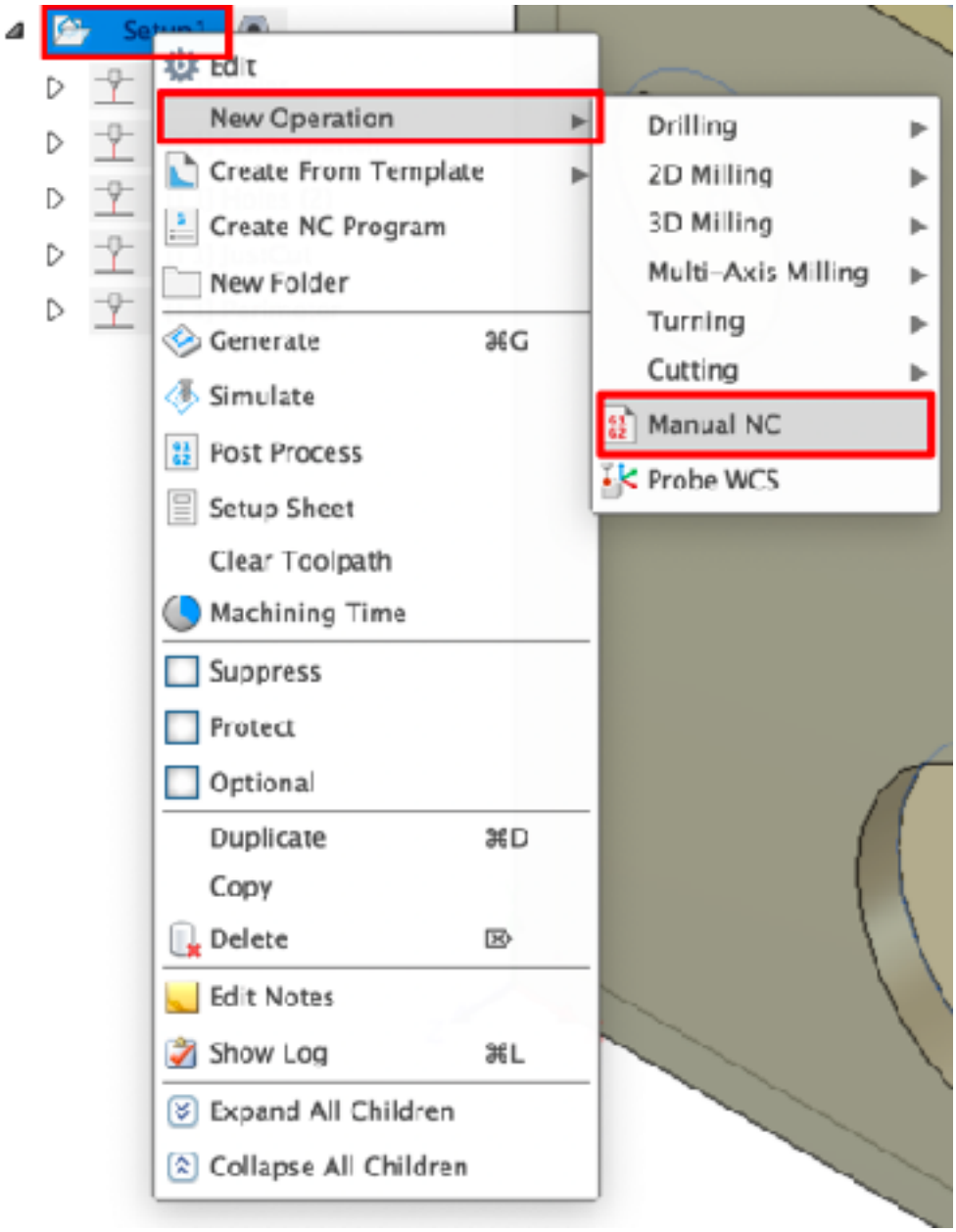
The post processor detects that you have unchecked the Lead-In box and creates all the special features. The other settings are necessary to make the special features work correctly. While these features are really designed for improving the cutting of small holes, you could use them for any size hole if you wanted.

More 2D Profiles can be created in this same setup, if desired.

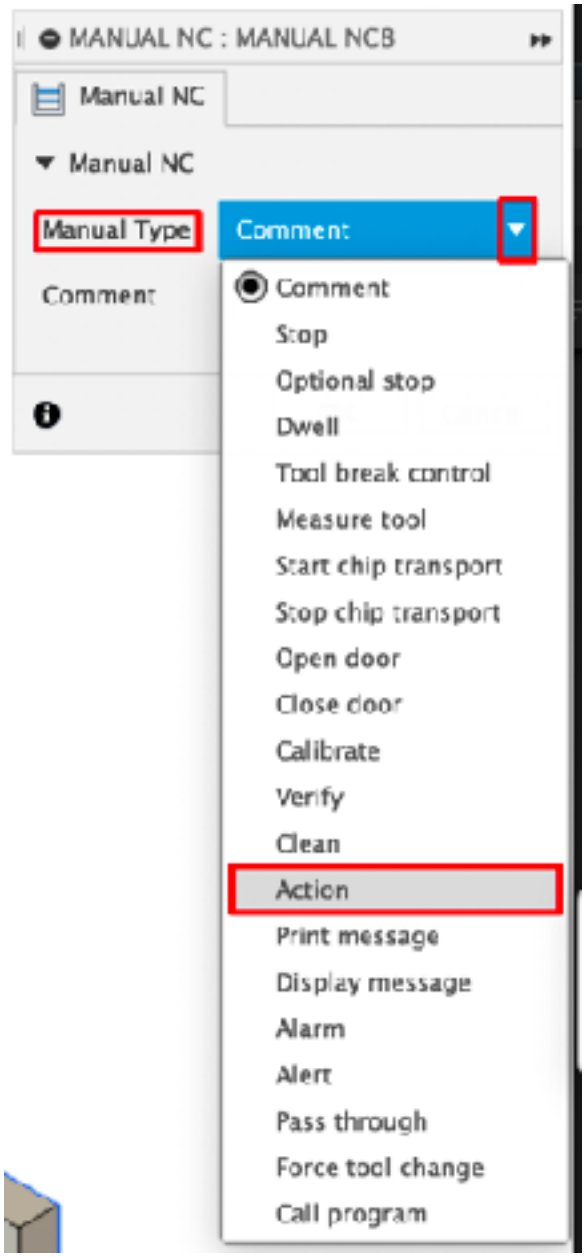
Center Punching

Two manual operations need to be created to turn on and off the center punch feature. These operations can be saved as templates and easily used in the future.

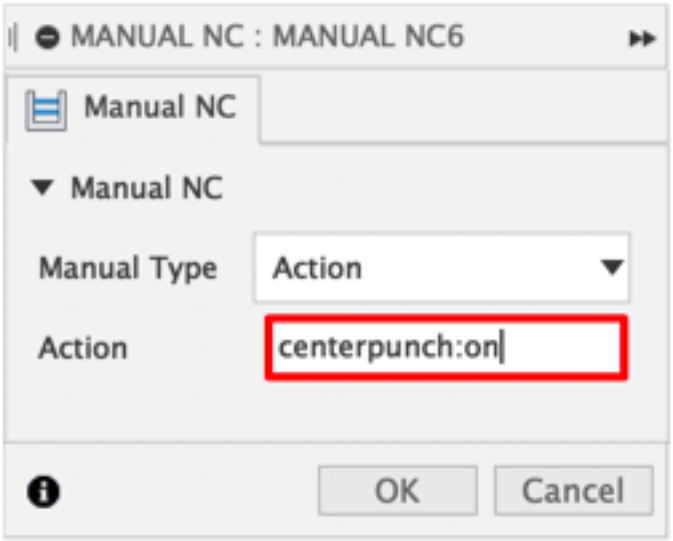
Begin by right clicking on the current Setup, selecting New Operation, Manual NC.



From the Manual Type dropdown box, select Action.



In the Action field enter:
centerpunch:on

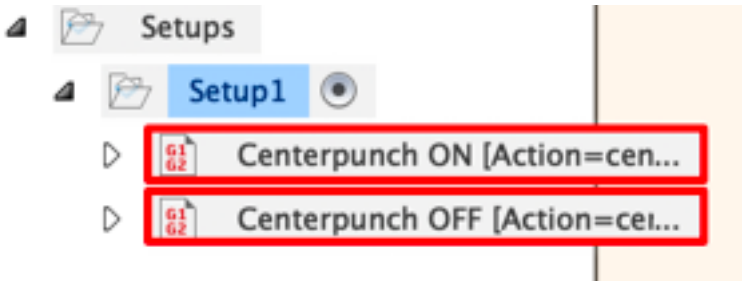


Click OK and then rename the operation (two single clicks on the name) to:
Centerpunch ON

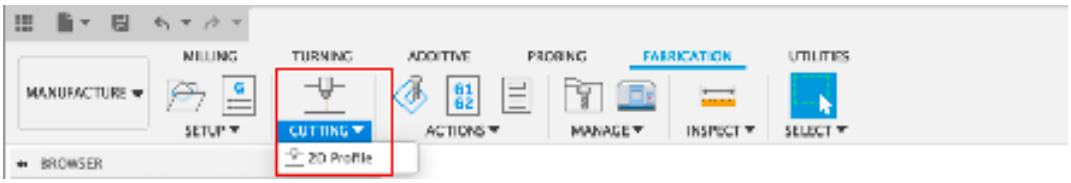
Repeat the above process but this time in the Action field enter:
centerpunch:off

Rename this operation to:
Centerpunch OFF

You should now have something that looks similar to this:

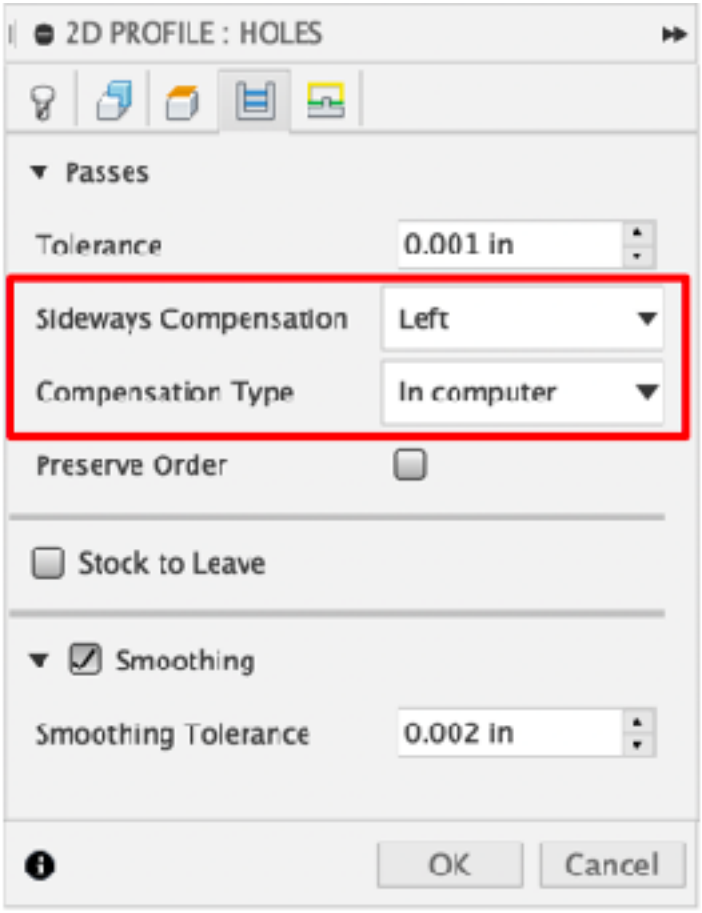


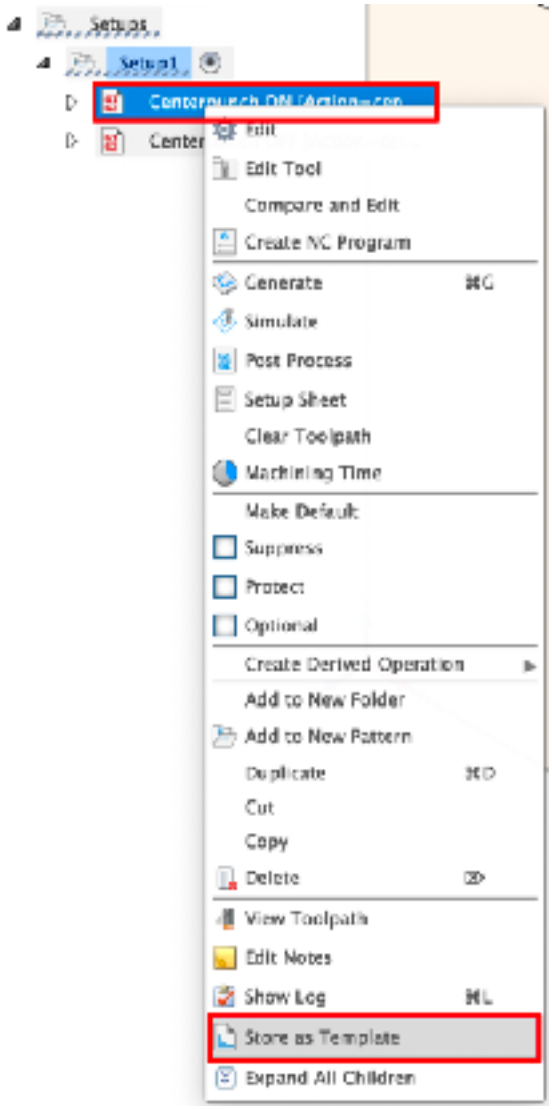
Now, save each of these for future use by right clicking on the operation, and selecting Store as Template. Store them in the default location with the names Centerpunch ON and Centerpunch OFF.
To access the center punch features, create a new 2D Profile:

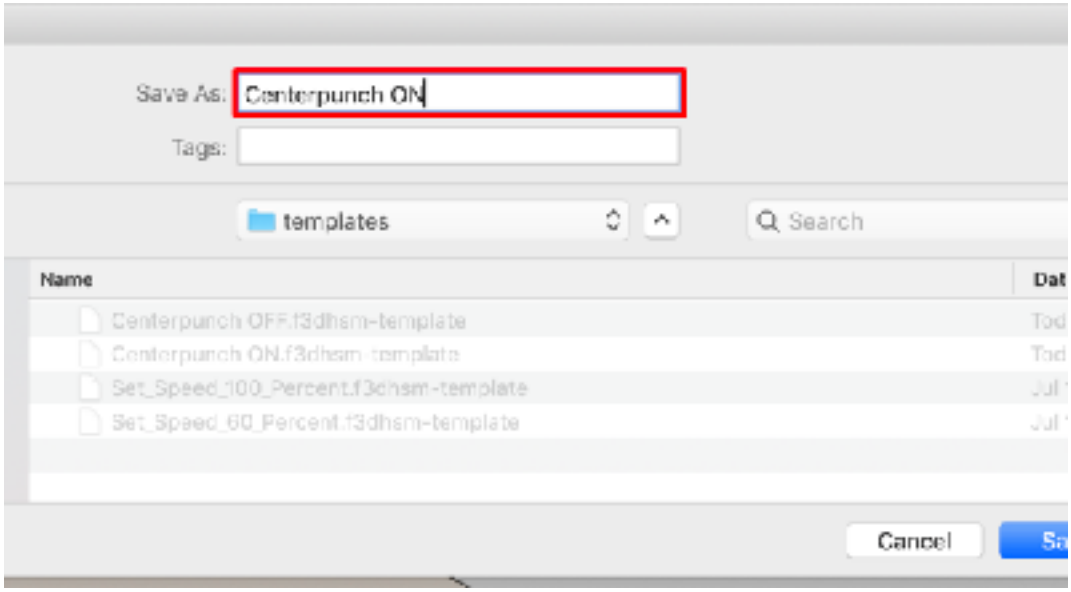


On the Geometry tab, be certain that you select only the holes where that you want to center punch in this 2D Profile. All other cutting should be created in one or more separate 2D Profiles.

Next, you need to set the following on the Passes tab:
Sideways Compensation: Left
Compensation type: In Computer

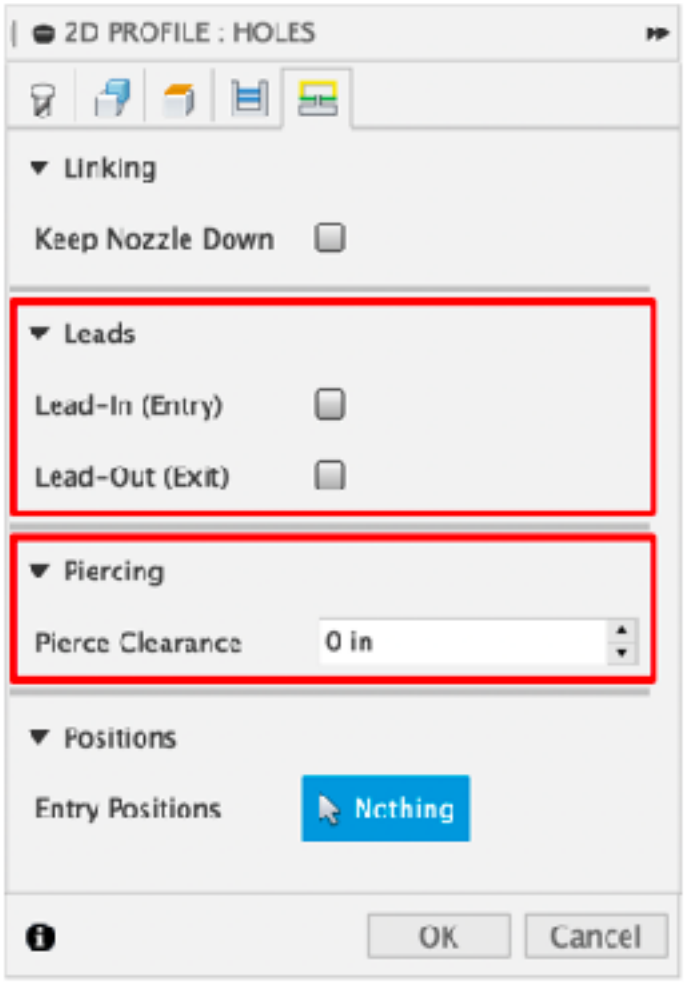



Setups
Setup1
Edit
Edit Tool
Compare and Edit
Create NC Program
Generate ⌘G
Simulate
Post Process
Setup Sheet
Clear Toolpath
Machining Time
Make Default
Suppress
Protect
Optional
Create Derived Operation
Add to New Folder
Add to New Pattern
Duplicate ⌘D
Cut
Copy
Delete ⌦
View Toolpath
Edit Notes
Show Log ⌘L
Store as Template
Expand All Children


Save As: Centerpunch ON
Tags:
templates
Search
Name
Centerpunch OFF.f3dhsm-template
Centerpunch ON.f3dhsm-template
Set_Speed_100_Percent.f3dhsm-template
Set_Speed_60_Percent.f3dhsm-template
Cancel

On the Linking tab set the following:

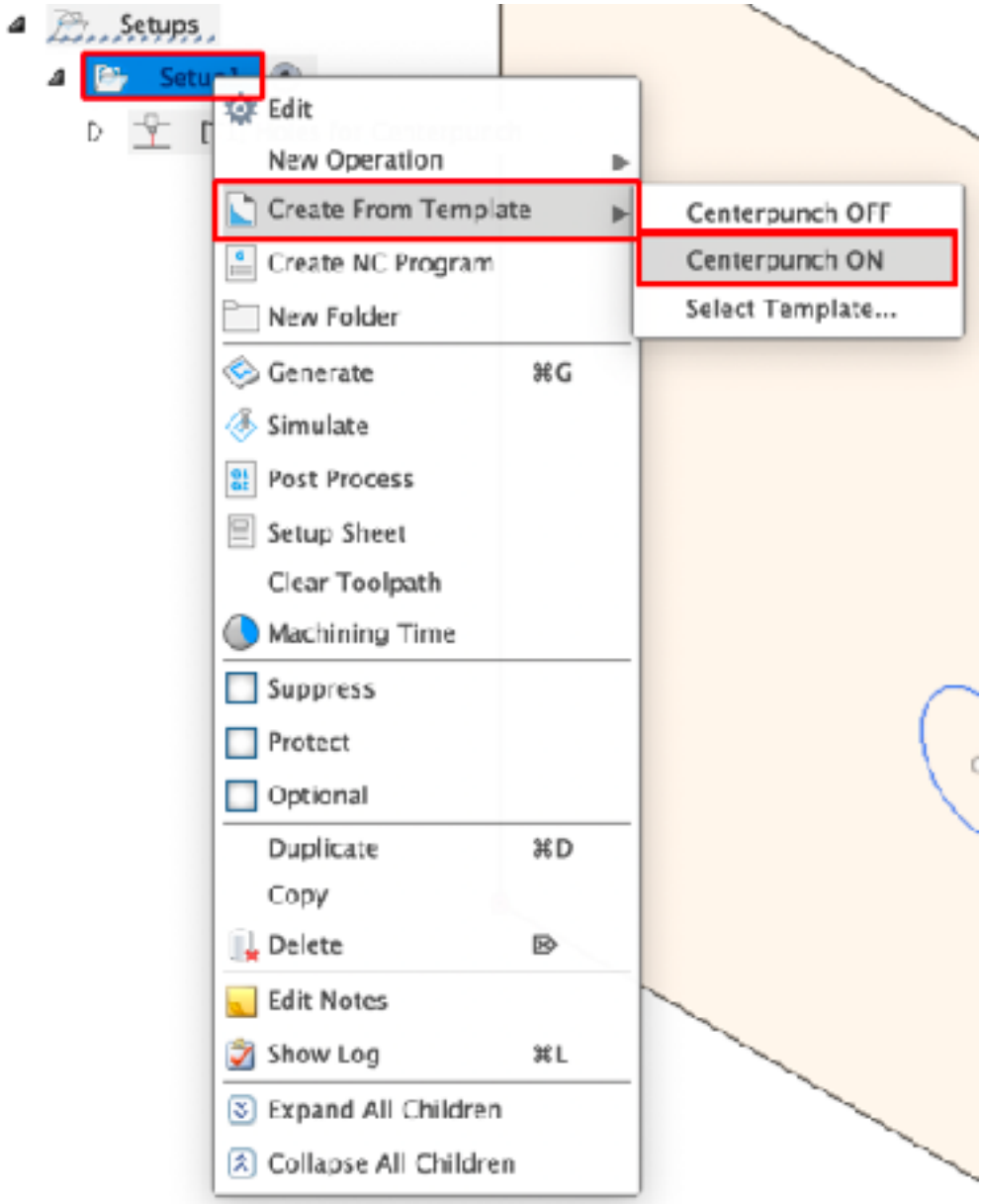
Lead-In: Unchecked
Lead-Out: Unchecked
Pierce Clearance: 0



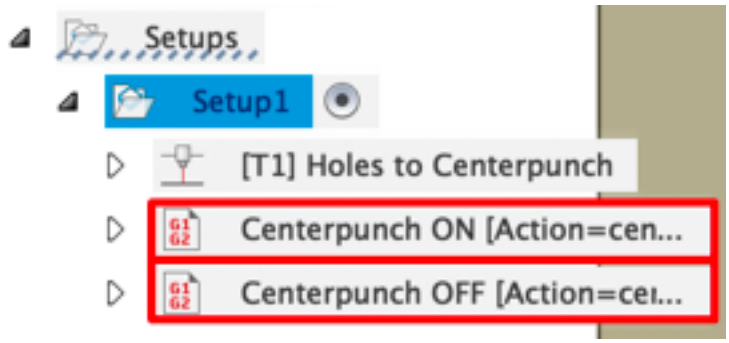
Click OK to create the operation.

(Note: These are the same settings that you use for the small hole cutting special features.)

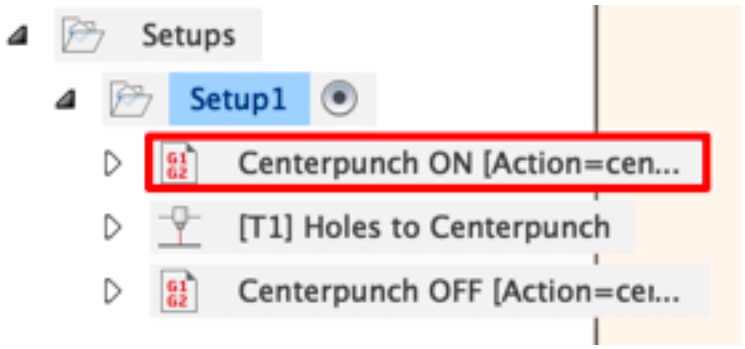
We will now use the Centerpunch ON and Centerpunch OFF templates we created. Start by right clicking on the current Setup, selecting Create From Template, and then Centerpunch ON.



Repeat this process selecting Centerpunch OFF. You should now have something that looks like this:



Now, click and drag the Centerpunch ON operation above the 2D Profile you just created (I changed the name to “Holes to Centerpunch” for clarity).

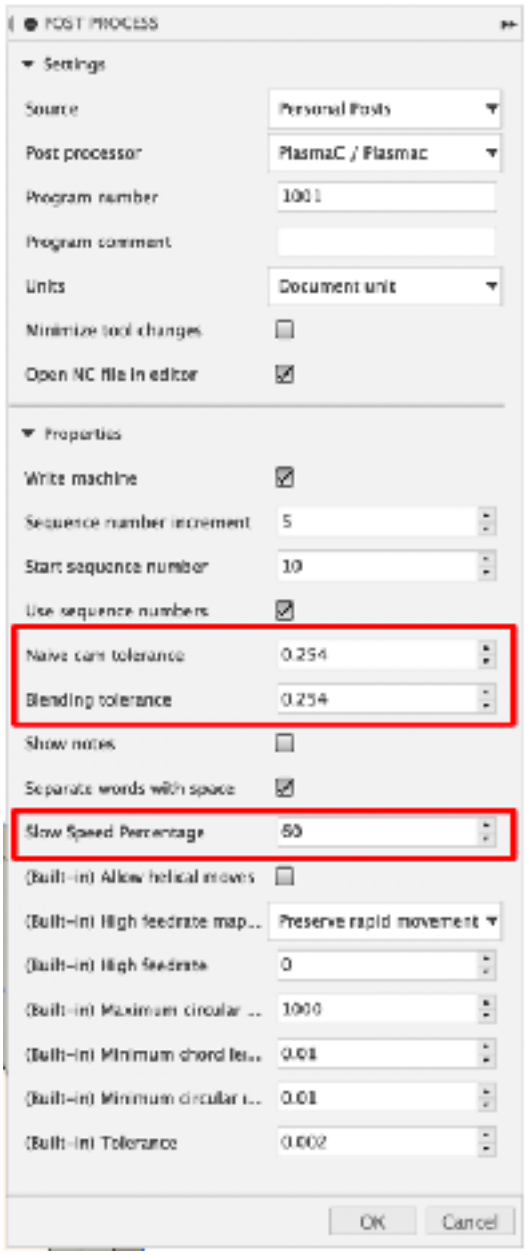


More 2D profiles can be created in this same setup, if needed.

Note: As is pointed out in the PlasmaC manual, not all machines will work for center punching.

Post Processing

Once you have created the 2D Profile(s) for your part(s), you are ready to run the post processor and generate the g-code.



There are 3 properties specific to PlasmaC/LinuxCNC that you need to address:

> **Naive cam tolerance** and **Blending tolerance** values are used with the G64 P(Blending tolerance) Q(Naive cam tolerance) code and placed in the beginning of the file. Please see http://linuxcnc.org/docs/html/gcode/g-code.html#gcode:g64 for a description of how these are used. These values are in mm.

Slow Speed Percentage: This value is only used for the small hole cutting feature. This is a percentage of the “normal” cut rate. Generally, 60% is a good starting place.

For reference, I have also included a Fusion 360 archive with a test piece that has three 2D Profiles in one setup: one for cutting 4 different sized holes, one for center punching 2 holes, and one for cutting the perimeter. If you look at the g-code file created by the post processor, you will see it includes notes when we are invoking these special features as well as when we return to normal operation.