Team 3176 White Paper

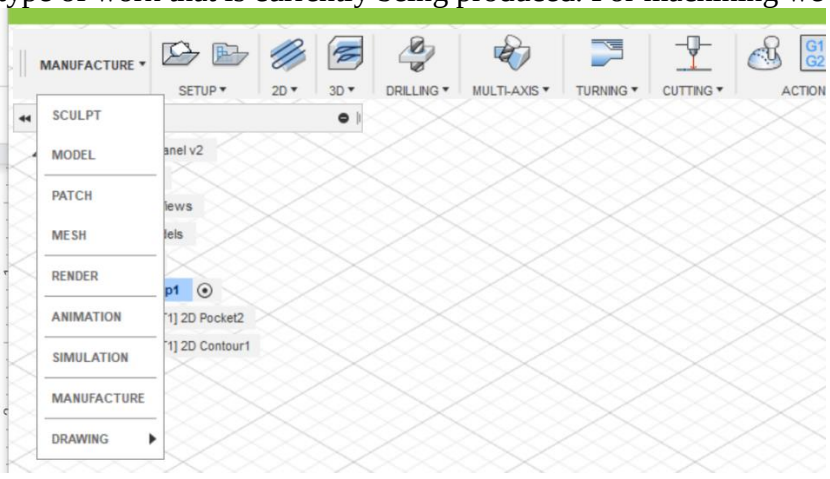
Title: Fusion 360 CNC Pathing	Author(s):Michael Dill
Sub-team/Function: Prep. for Fabrication	Subject: CNC Pathing
Date:3/1/19	White Paper Number:01

Abstract

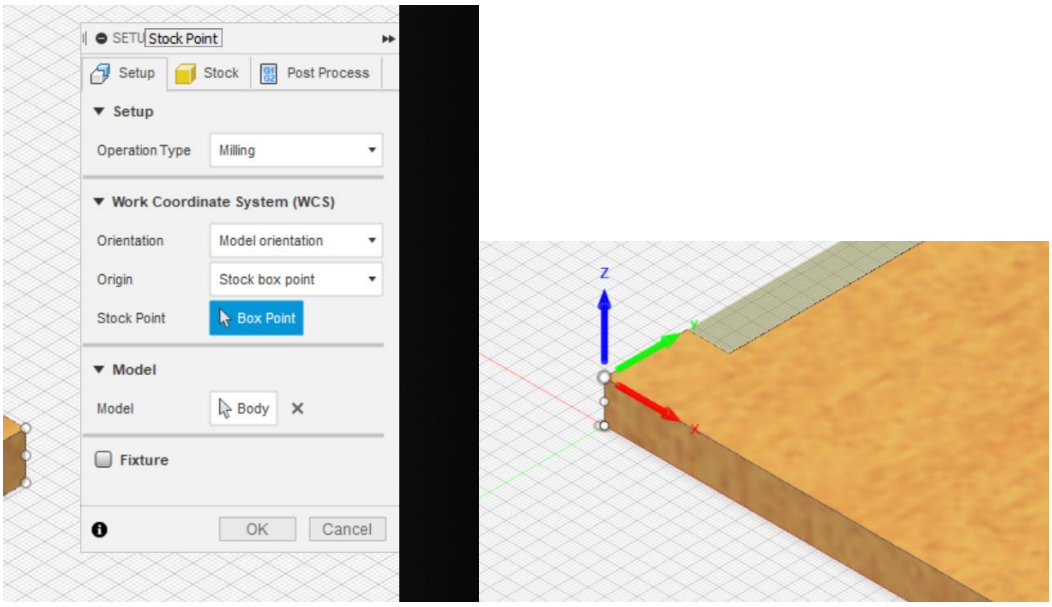
With the recent acquisition of the carbide 3d 3 axis cnc router it is important to know how to take a product from Computer aided design file(CAD, stl, ipt) into a usable machine path. To do this several pieces of software and physical tools are needed. You will need a copy of carbide motion, a part designer, internet access and calipers.

How to Create a Path

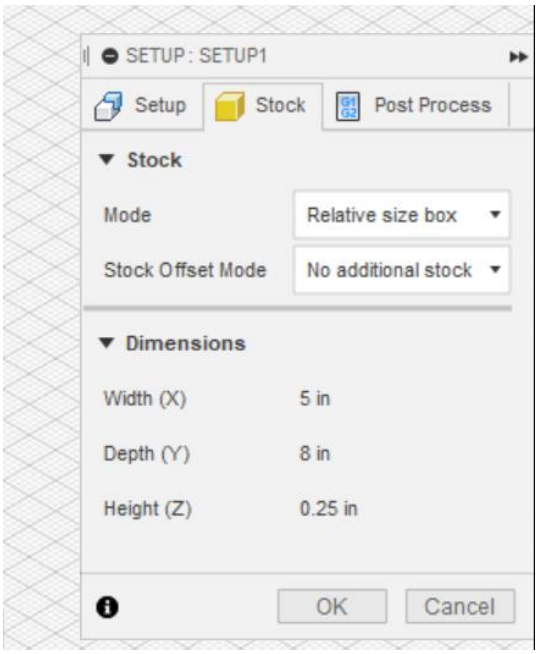
Fusion 360 is the program that is currently in use for creating CNC router routes and Gcode for our team. To begin the process you first need to gain a copy of the ipt you are trying to machine. Note: .iam is not compatible. With the file imported into Fusion 360 we need to go into the dropdown menu for the type of work that is currently being produced. For machining we choose the manufacture tab.



With the manufacture tab now selected all the tools on the top panel change to what is needed for machining a part. The first step on the process is to create a setup. This defines the orientation of the part relative to the machining surface.

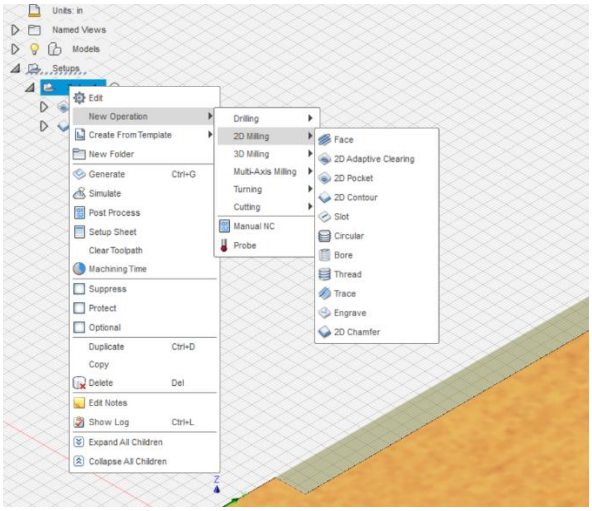


Be sure the Z and X axis are set correctly. While still in the setup screen be sure to select no additional stock under the stock tab.



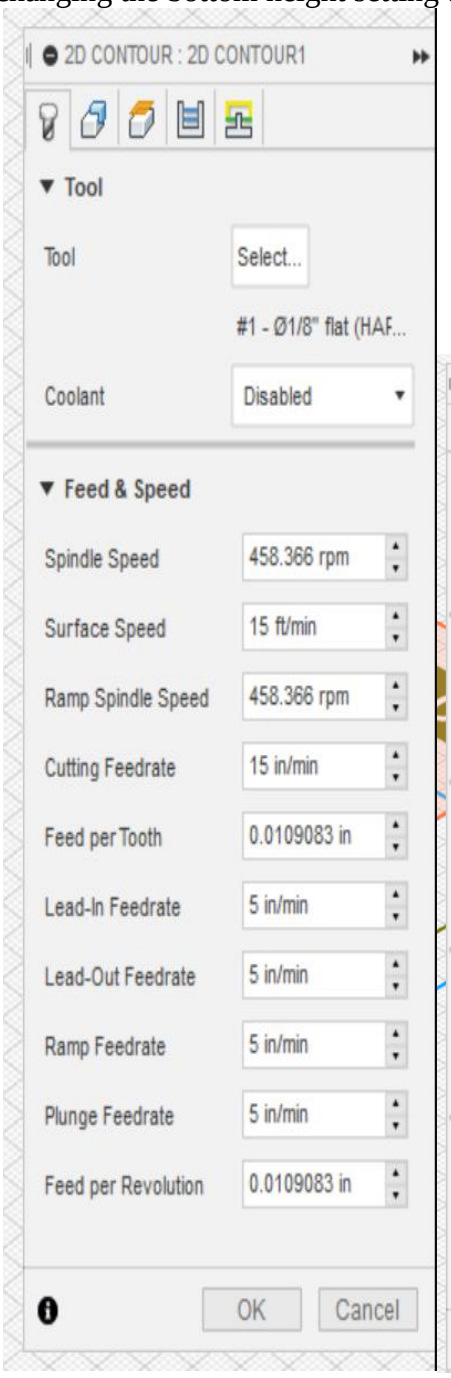
After the additional stock is removed it is important to select the origin point for the cut, we use the bottom left hand corner. One the setup is created you can now start to create the actual cutting paths. For the majority of cuts needed the contour tool is the one that is you need to use.

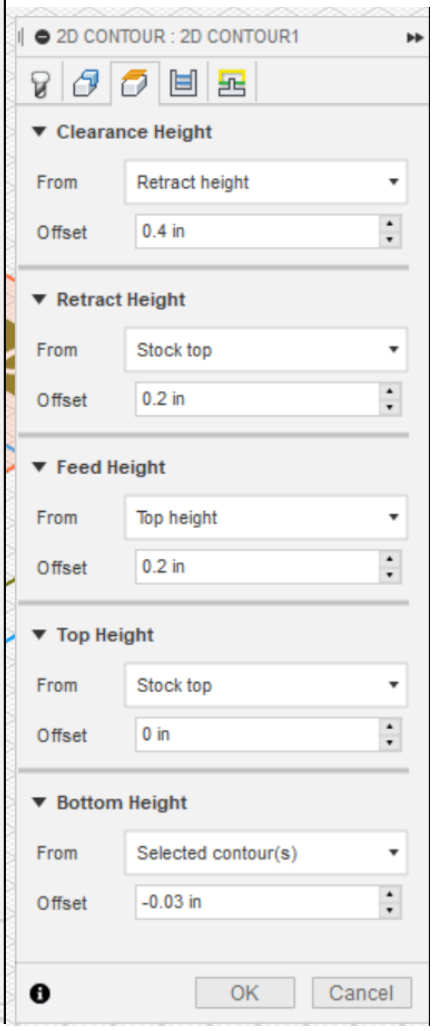
The contour tool can trace and interior or exterior outline of a shape.



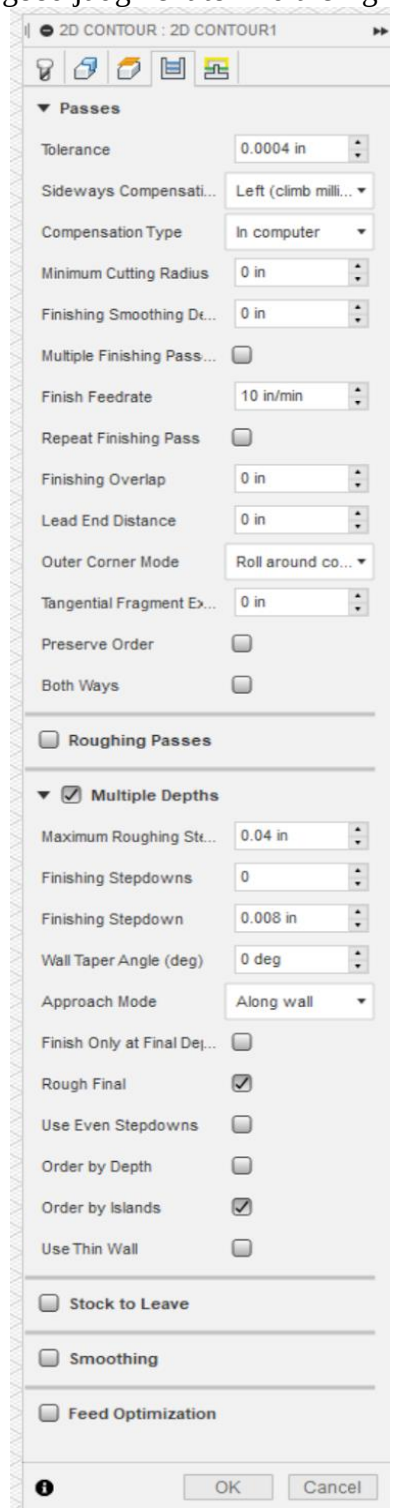
Once the contour is selected you can select the bit needed. For cutting metal and other hard materials the 1/8in bit is the one that is the best to use.

For Lexan the Feeds and Speeds need to be set according to Appendix I.
With Feeds and Speeds now set the maximum depth for the contour cut needs to be set this is done by changing the bottom height setting to match the actual thickness of the done by changing the bottom height setting to match the actual thickness of the material.

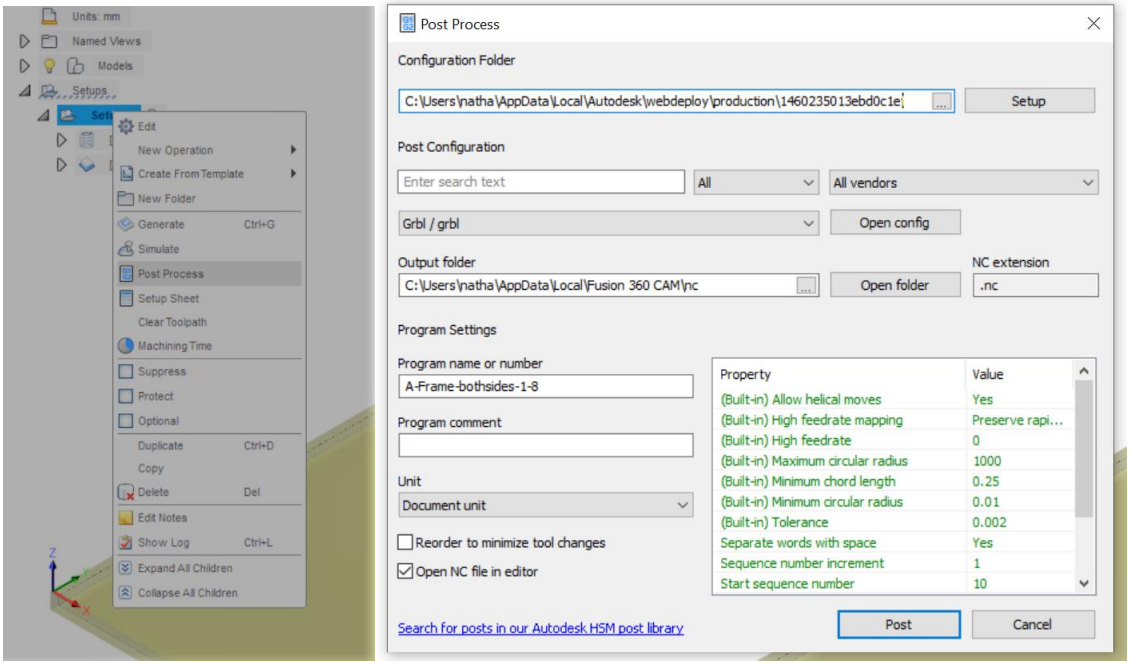




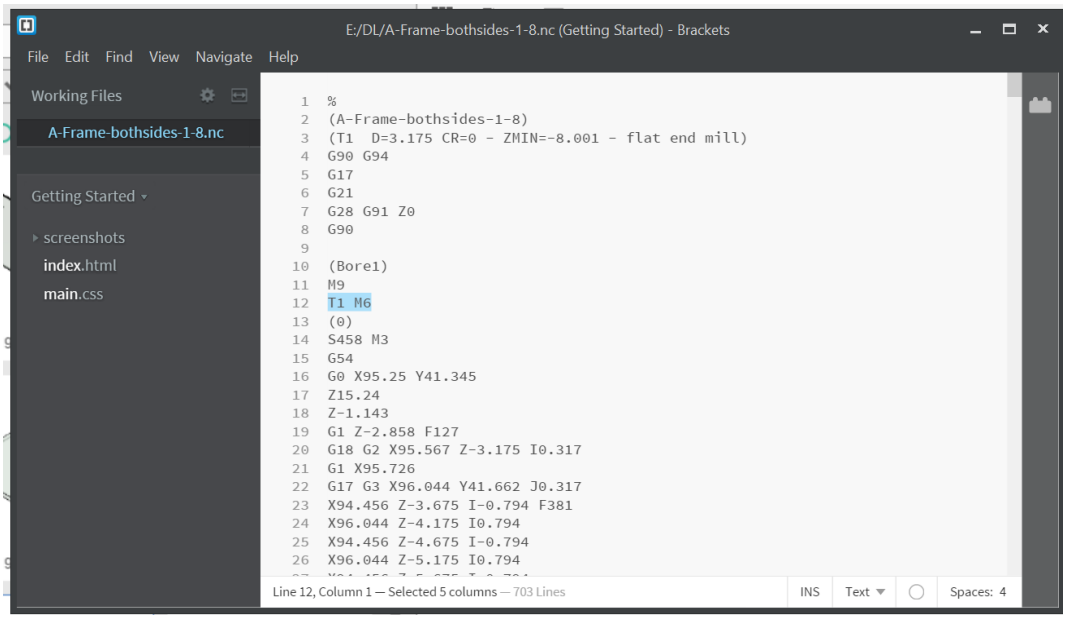
This is important due to the fact that the thickness of the material is often not the same size that is shown in CAD. Before exiting the contour creation box it is necessary to change the number of depths that the bit goes through before reaching the bottom of the material. For metal 10 thousands of an inch is normally standard. For other materials it is best to search online and use good judgment to find the right cutting depth.



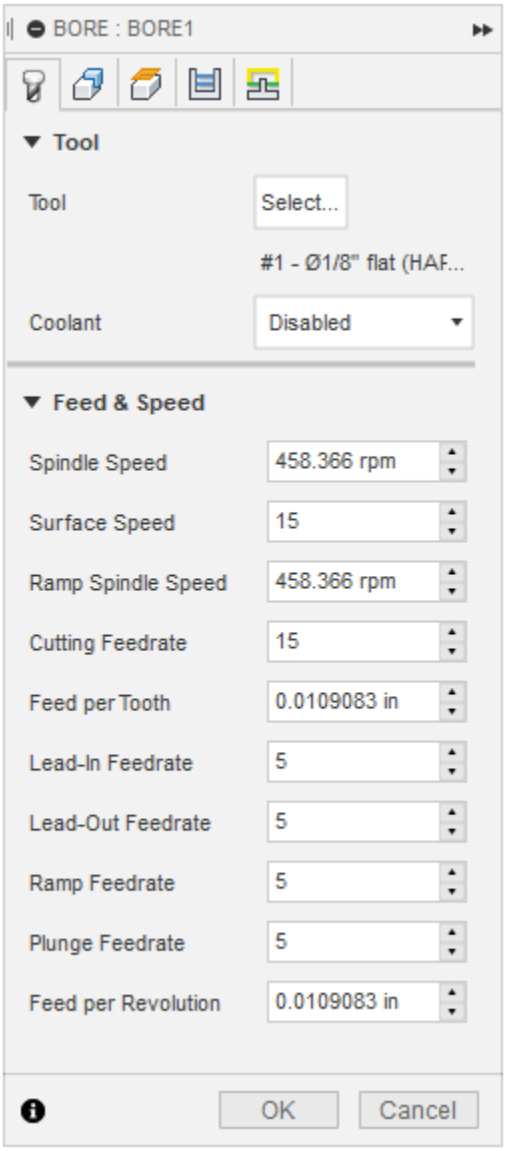
Exporting the completed pathing of the part is a easier part of the process. The first step is to change the document units to millimeter. This is needed in order for the cutting software to more completely understand the instructions given to it since it was coded with metric natively. Once the document units are set it is necessary to re-generate the toolpaths. Right clicking on the setup tab allows for an option called post processes which will open the export screen.



Our router rus GRBL as the Gcode language. This allows for communication between the router and the computer running the Gcode from this point when post is pressed the file output folder and name can be set. After the file goes through exports a Gcode editor will appear. We need to delete line 12 which is a command that sets the speed of the router. With our configuration this line will cause a error if left in.



APPENDIX I



LEXAN - WOOD - Router Speed 3
Polycarbonates-Router Speed 2.5

DELETE LINE 12