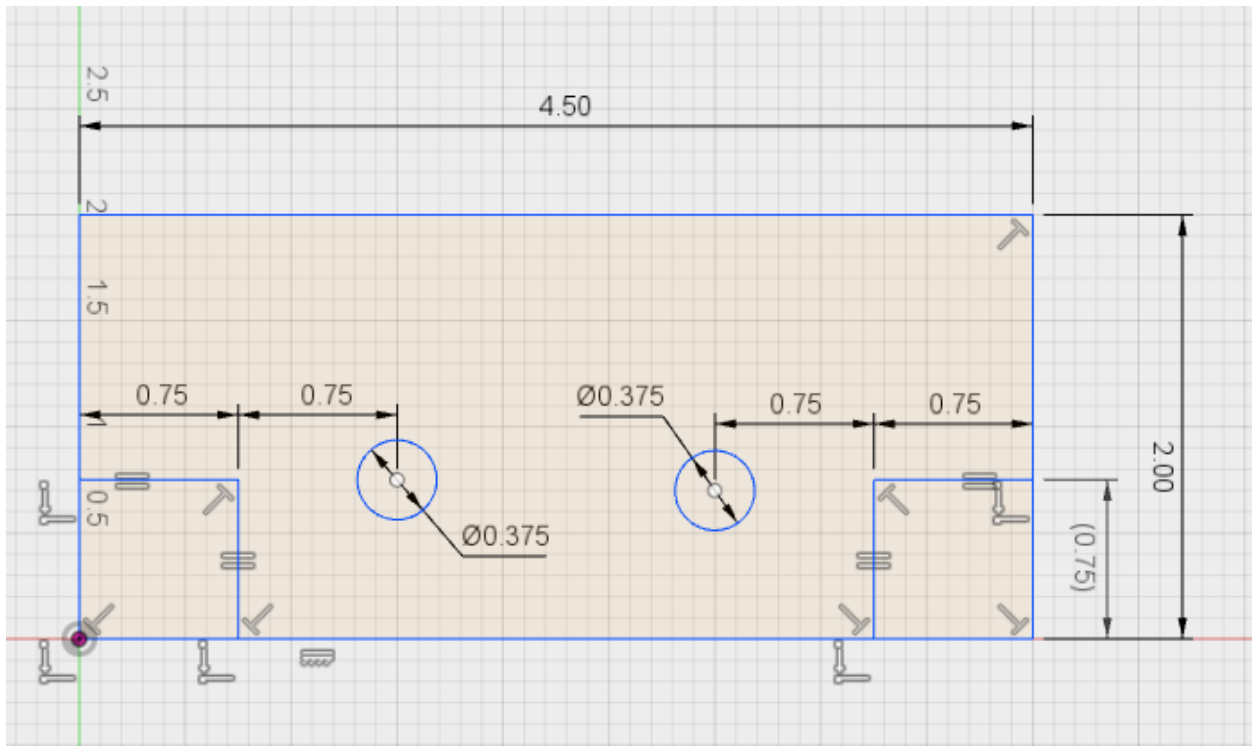


Fusion 360 Workflow Example

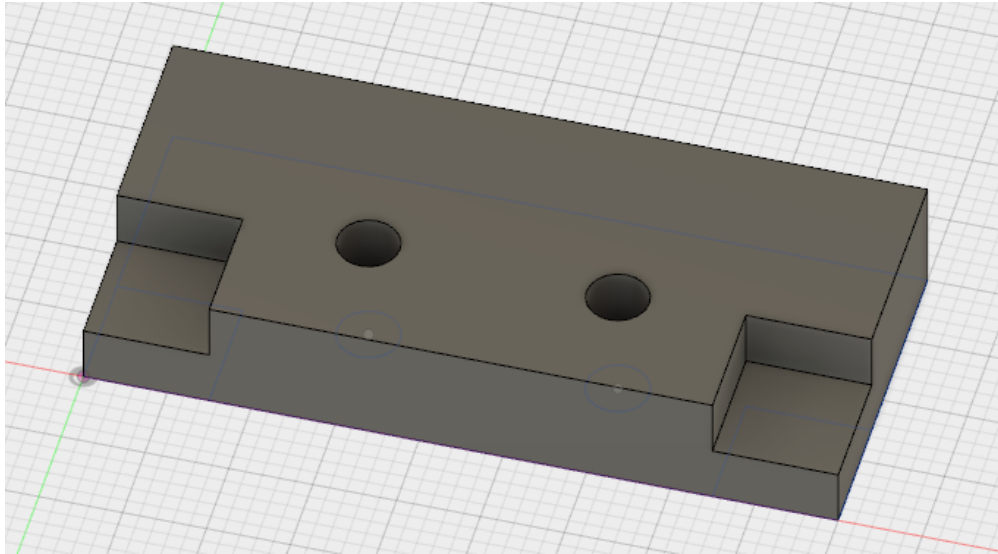
I thought I would share my experience using Autodesk Fusion 360 to mimic the Corel Draw -> VCarve -> Mach 3 workflow process that Michael has posted [here](#) (by the way, it's a great walkthrough on the CNC process, thank you!). Fusion 360 is probably more complex to use, but so far it's been capable of doing everything I asked of it. Not bad for free software (at least, free for 1 year).

So here it goes...

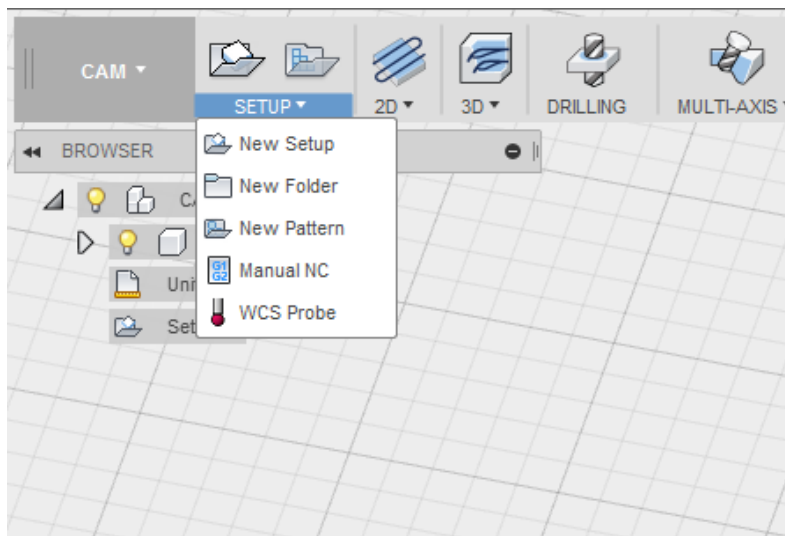
I started by creating a sketch of the Z-stop block. I won't spend any time describing how to sketch or navigate in F360 – there are tutorial videos included that do a far better job than I could! Suffice it to say, I was up and running after a 15 minutes of tutorial.



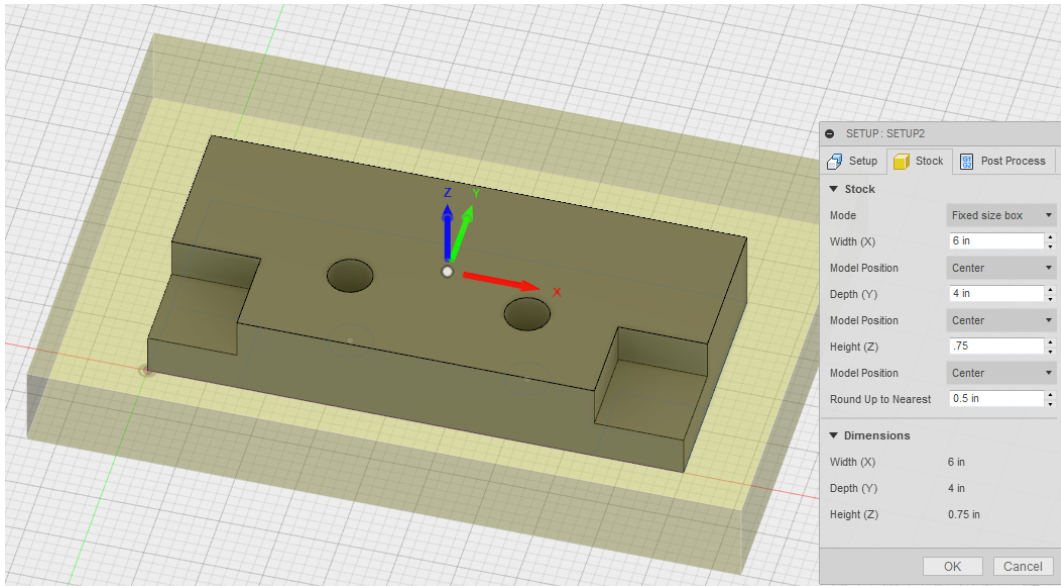
Here's where the workflow starts to deviate from VCarve... In the next step, I created a 3D model using the extrude command (again, great F360 videos showing how to do this), extruding the main body to 0.75" and the corner sections to 0.375. Looks pretty good!



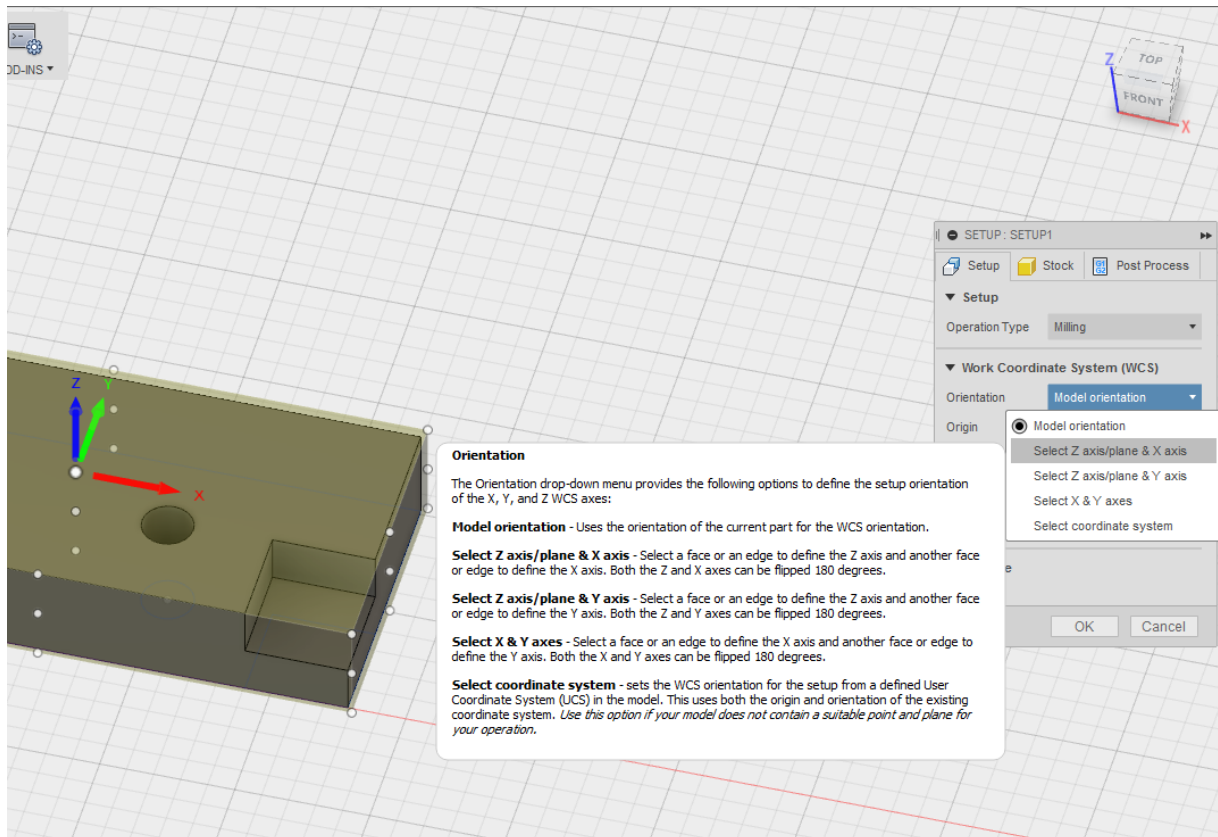
Once you have the 3D CAD model built, you switch to CAM mode in F360... and here is where it starts to get a little trickier, perhaps, than VCarve. Quick Disclaimer: I'm still very much a novice in CAM – so others jump in if you see better ways of doing things. Anyway, the first thing to do is create a CAM setup:



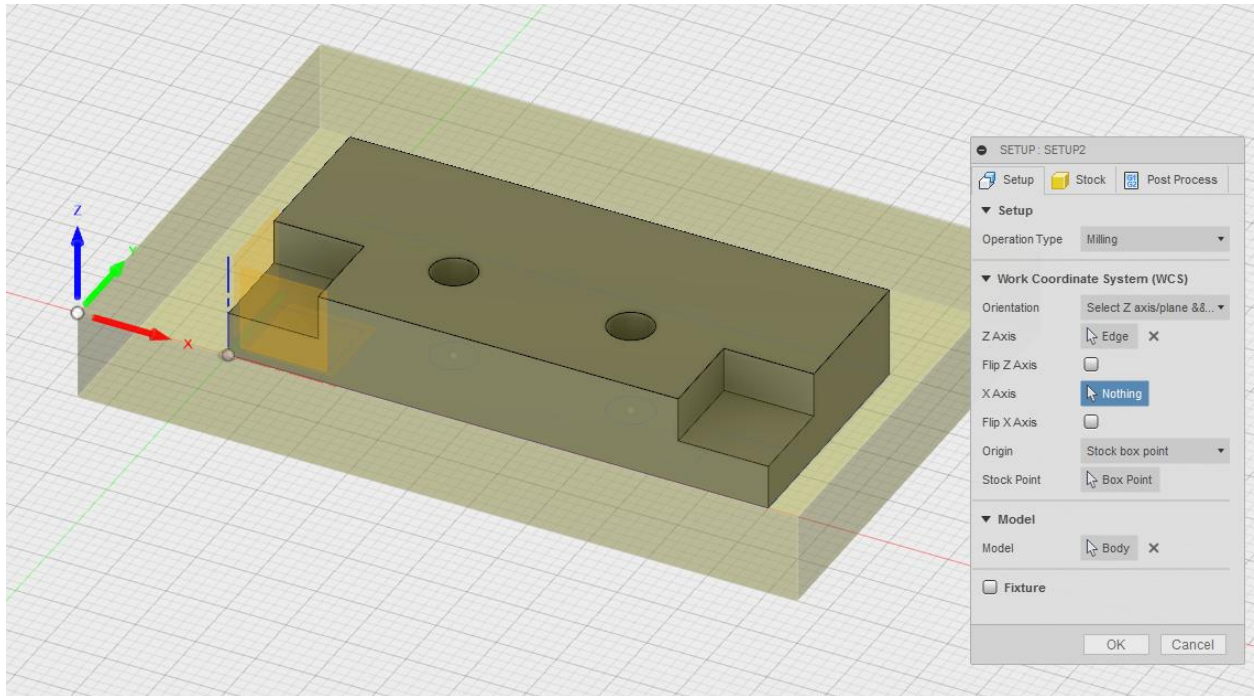
Next, I set the stock size via the "Stock" tab in the Setup menu. I'll use the same dimensions as the Vcarve walkthrough, 6"x4"x0.75":



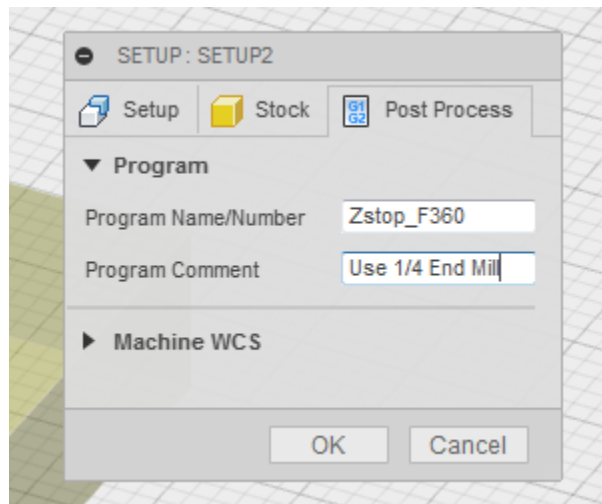
Next, I set the work coordinate system to match the KRMx02. To do this, I hit “Select Z axis/plane & X axis,” and then clicking on a Z edge (any z edge will work)



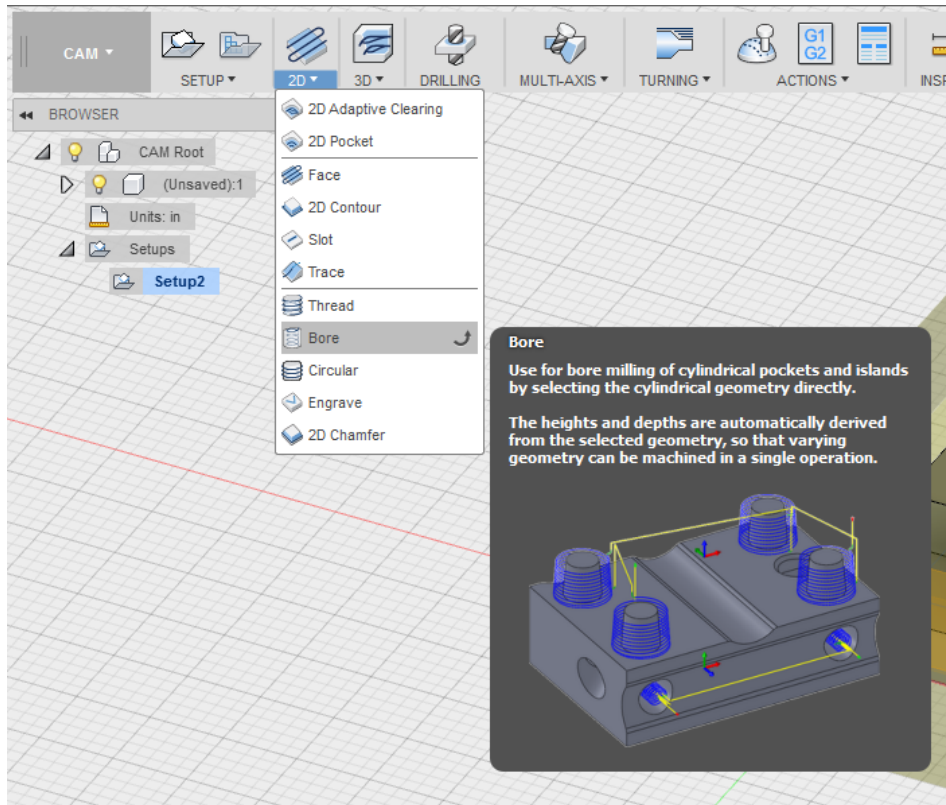
Next, I set the origin point by selecting “Stock box point” from the menu, then clicking on the lower left corner, on the top stock plane.



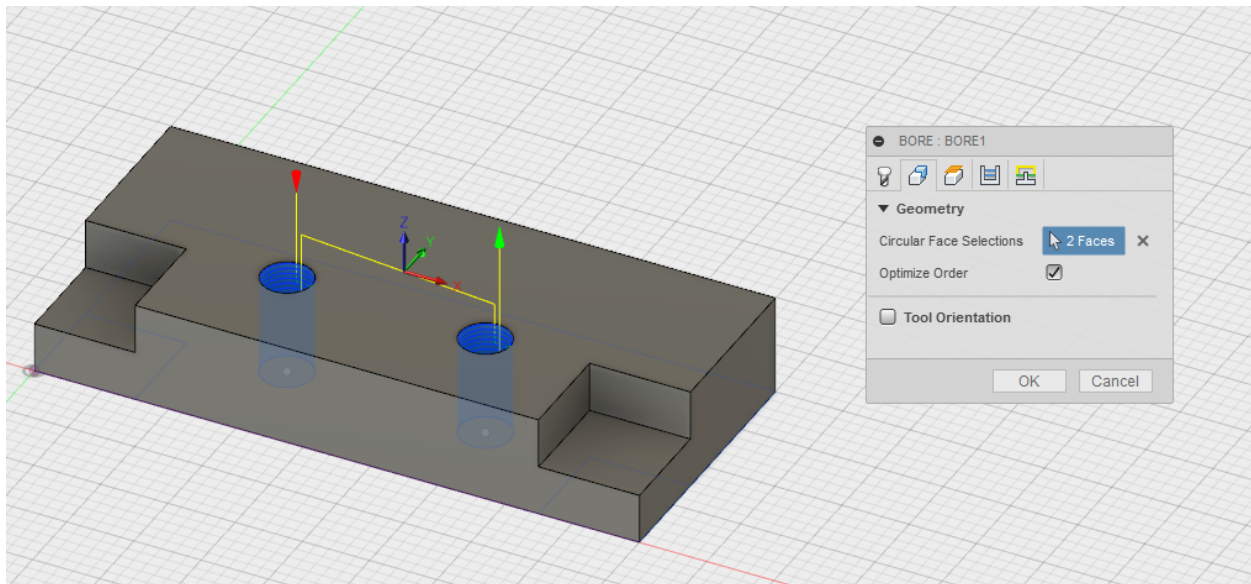
Finally, I type in some descriptions in the Post Process fields:



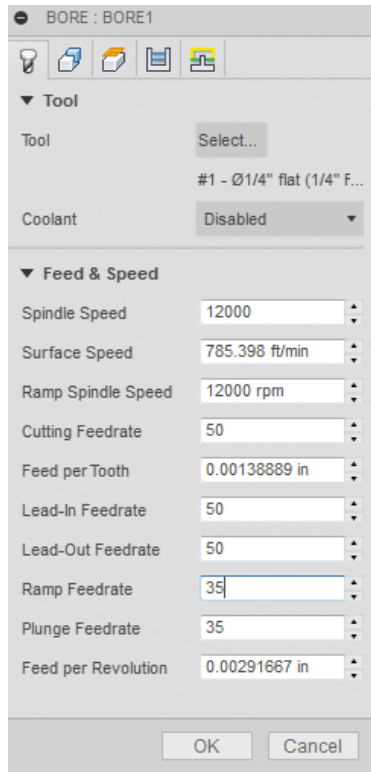
Ok, now to set the tool paths. I'll start with the through-holes tool path. To do this, I select 2D > Bore



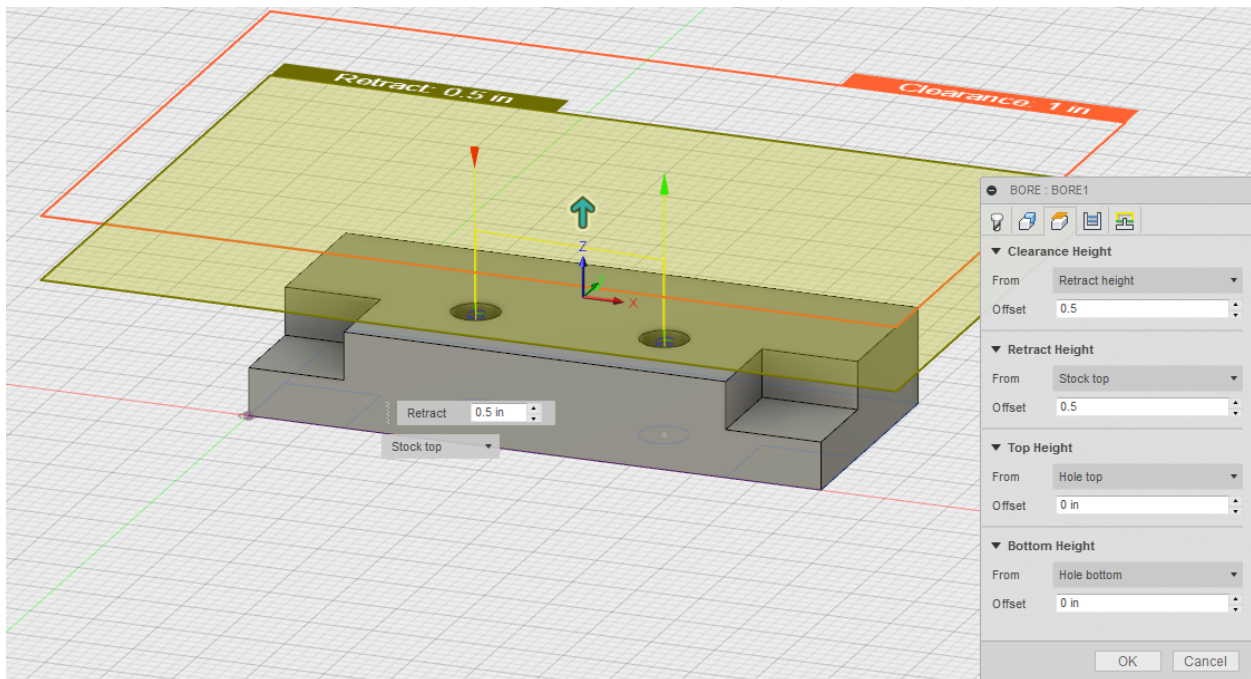
... and then click anywhere on the holes. You'll see the simulated toolpath pop up.



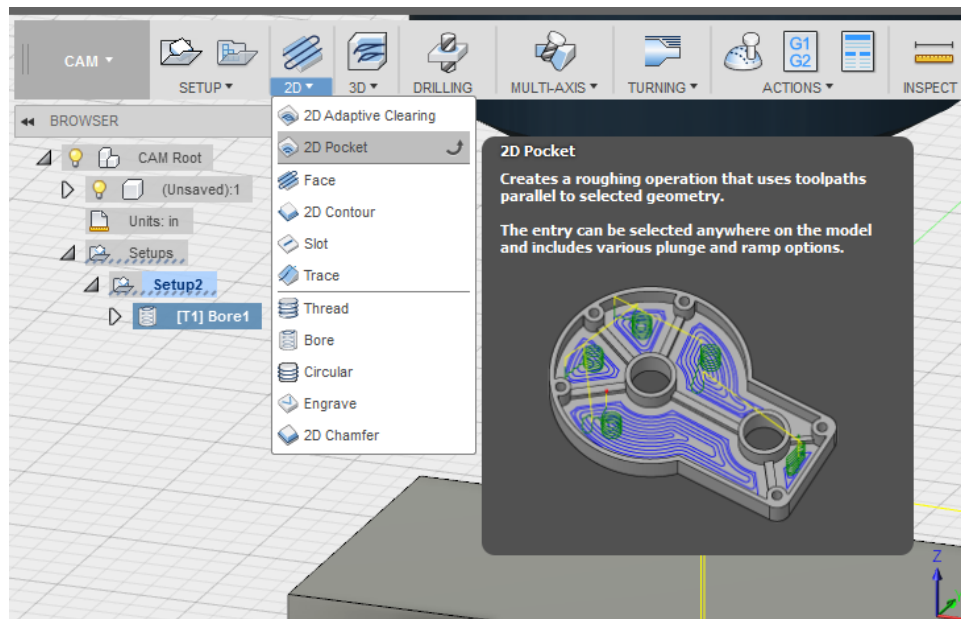
Next, I set the parameters for the holes (bit, feed rates, etc). Note: I don't have a SuperPID set up yet, so I'm just leaving Spindle Speed at the defaults. I set the cutting federate, lead-in federate, lead-out federate, ramp federate, and plunge feedrates. The others fields are automatically calculated



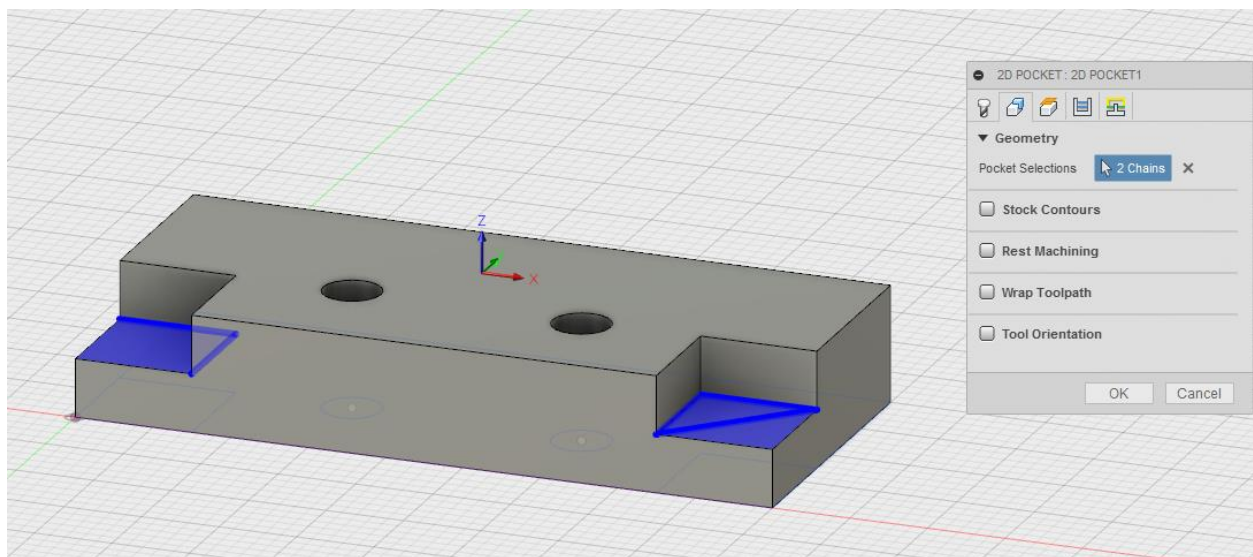
The only other tab I change is the "Heights" tab. I feel that the defaults are a little too close to the stock, so I bump the clearance and retract height to 0.5" which is very conservative.



On to the .75" corner pockets now. For this, I select 2D > 2D Pocket.

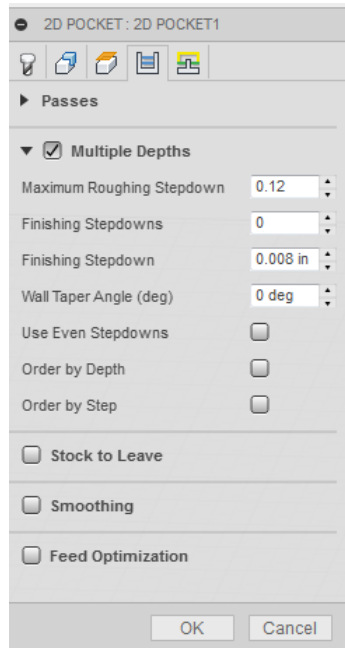


Then click on the two top faces of the pockets that are to be cleared.

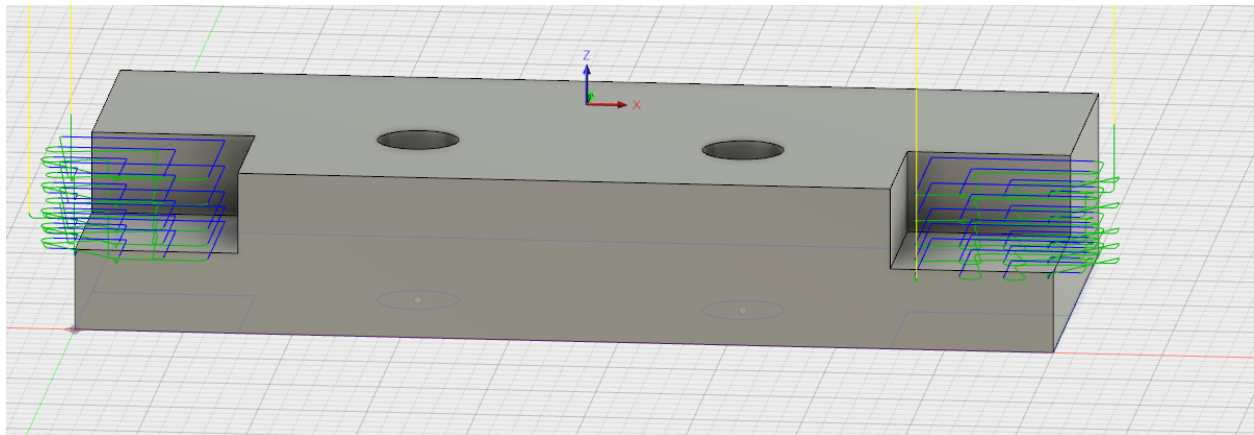


Then set the parameters (should default to the same settings as the holes toolpath).

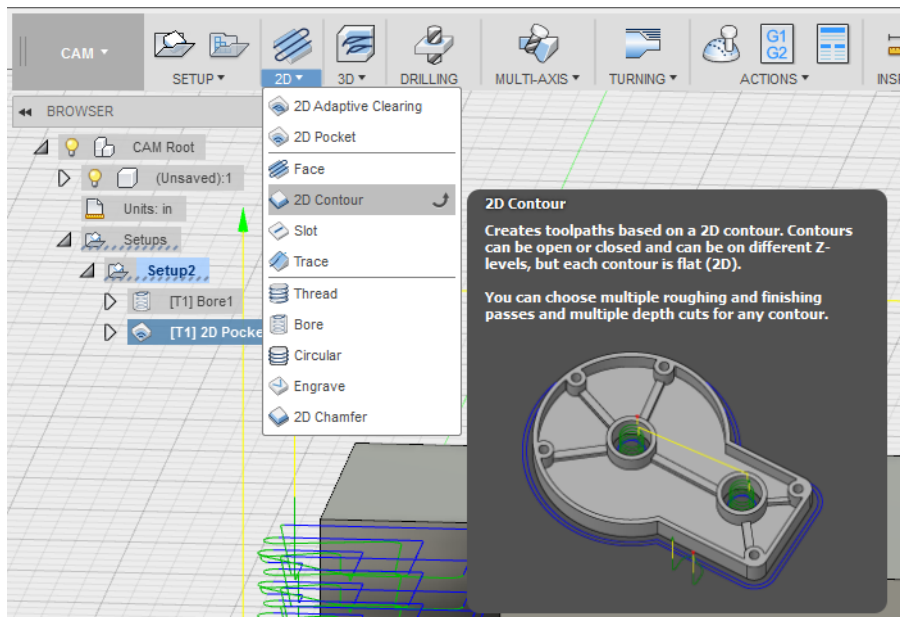
One thing you should be certain to check is on the “passes” tab. By default, “Multiple depths” is not selected. Make sure to set this up, or else it’ll try and take the entire 3/8” chuck in one pass... probably not a good idea. I set the depth of each path to 0.12”. This might be on the conservative side.



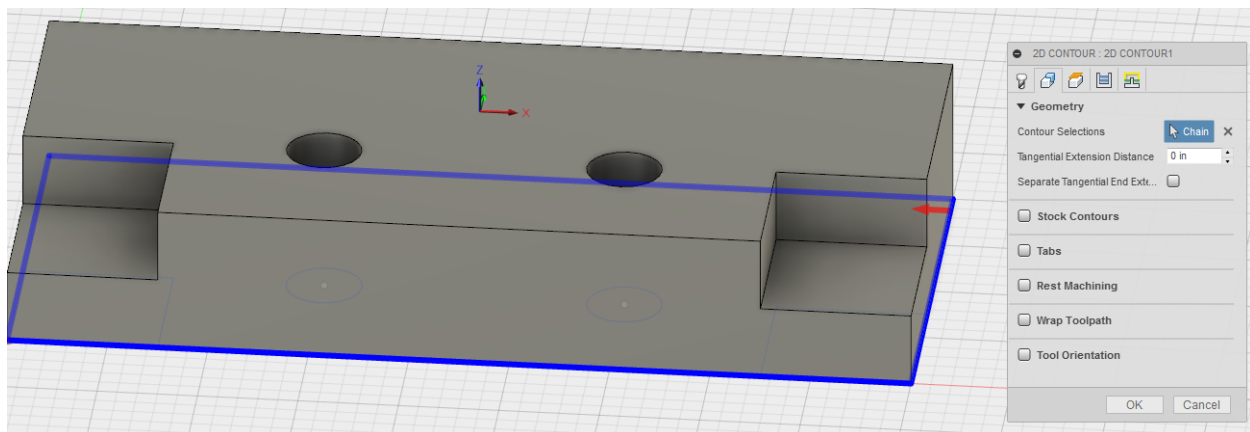
After this, click OK and preview the toolpath.



For the final toolpath, select 2D > 2D Contour

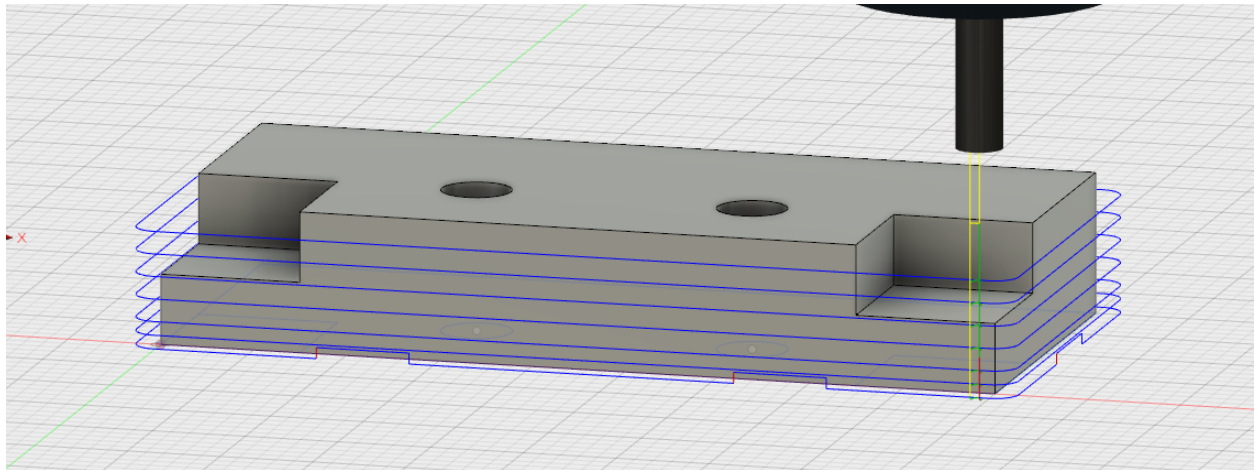


Select the lower edge of the outline:

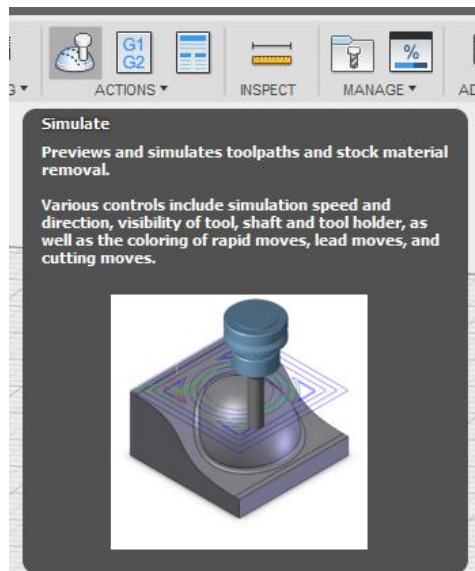


In the Geometry tab, select the "Tabs" checkbox. I usually stick with the defaults unless it looks really funky.

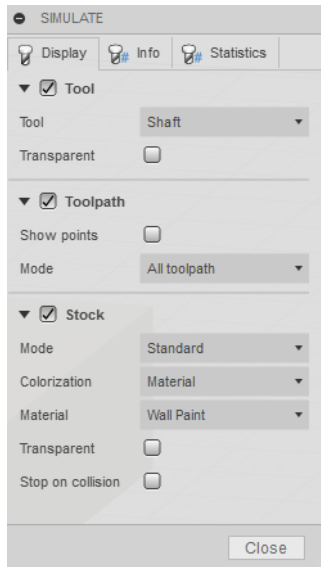
Finally, set Multiple Passes to 0.12" as in the previous step. Preview the toolpath (you can see where the tabs are on this illustration):



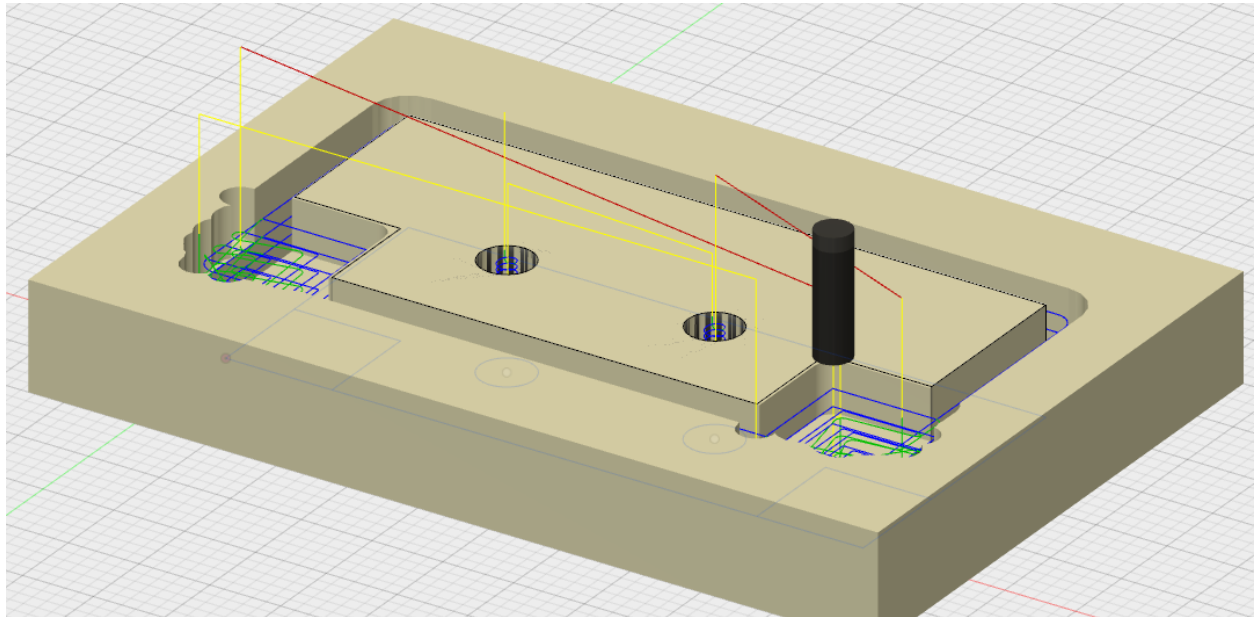
After all the toolpaths are set, I like to use the simulate button to watch the virtual CNC cutout.



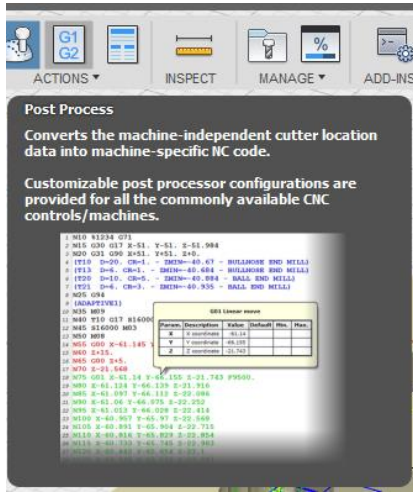
I like the following settings for simulate:



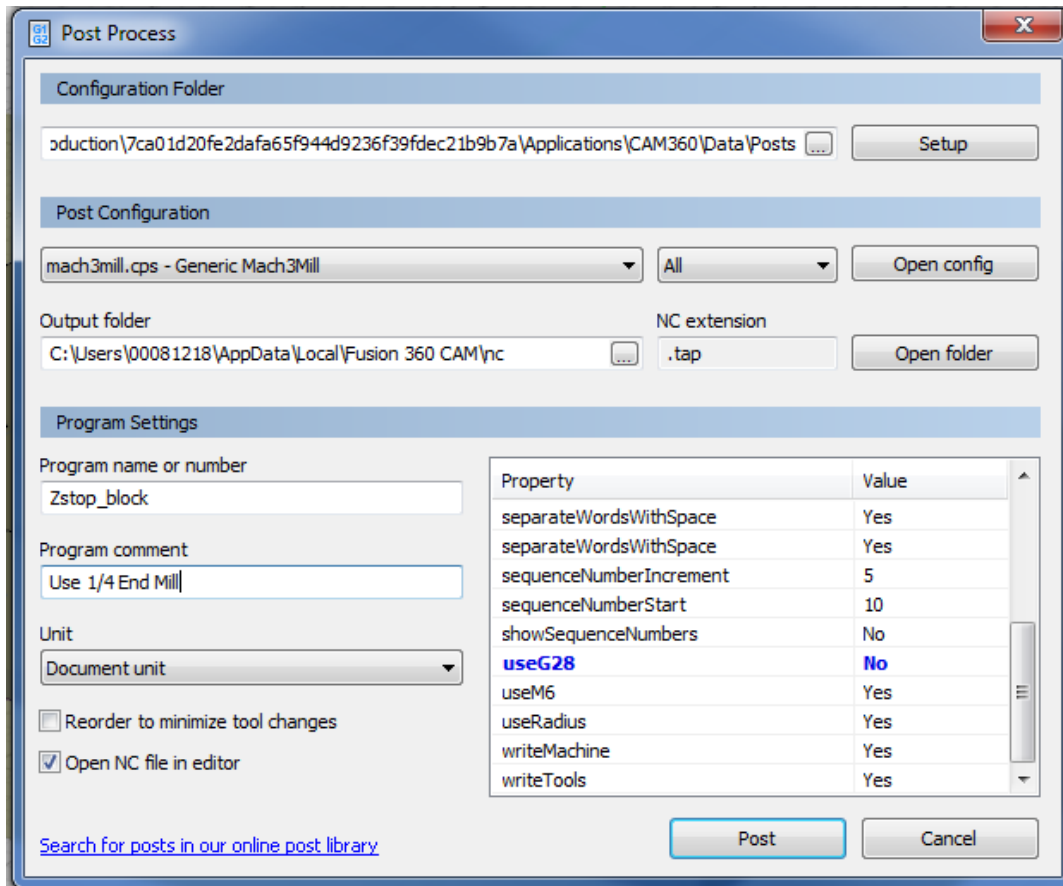
Finally, hit play and watch the program virtually run:



For the last step, I create the G-Code by clicking on the post-processing button:



In the post-process window, I select “mach3mill.cps” from the post configuration menu, and select “No” on use G28. The rest is all default, unless you want to change where your output folder is.



And that's it! The rest of the process is the same for Mach 3. One thing to note is that F360 attaches the ".tap" extension to the g-code file, but it's really just a text file. You might have to tell Mach 3 to look for tap files when you go to load your G-code.

