CNC Mill

Table of Contents

Key Info

Machine: Tormach 1100 MX

Machine Info & Specs

Location: IGEN Machine Shop

Book CNC Mill Training Here!

> You must complete Safety Training & CNC Mill Training to use this tool.

Introduction

This document will help you understand some key considerations when designing for and using the IGEN CNC mill. Please read the Manual Mill document before reading this document, as this assumes you have knowledge from that document.

Hazards & Emergency Protocols

- Always wear safety glasses and optionally hearing protection or an apron depending on the situation
- Do not stick your hands anywhere inside the mill enclosure when machining!
- Do not wear gloves when machining
- Don’t leave halfway through milling! Always make sure you’re supervising the mill when it’s cutting.
- End Mills are sharp, handle with care

- Familiarize yourself with what to do in the event that the mill starts doing something you don't want:
    - If it's going off course, but you have plenty of time to stop it, press the spacebar or the "pause" button to pause
        - Go back to your toolpath on your PC. Simulate again, and try to figure out what is going wrong. You should always diagnose and course correct before trying again. If you can't figure it out, ask Zach, Arif, or Jordan for help!
    - if you want the machine to immediately stop, press the big red emergency stop button
        - This will mean that you will need to restart the toolpath and re-origin the machine
        - If you have damaged the table or the tool, make sure to mark the damage and contact a shop supervisor
- Ensure that your stock is securely mounted before pressing start

In an Emergency:

1. Call 9-1-1
2. Call Campus Security – 604-822-2222
3. Call Zach Huser (IGEN Shop Supervisor) - 778-316-3601

Required Software

Solidworks

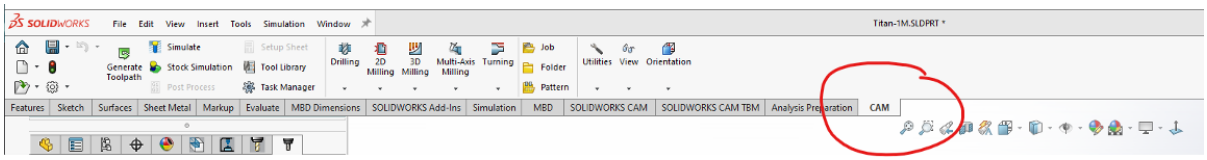
You should have a Solidworks license through UBC or through your design team. The UBC license is available here.

- Click "Download Teaching and Learning Software" and continue to install the Solidworks Student Engineering Kit on your PC.

Autodesk HSMworks

All students have free access to most of Autodesk's products. Here's how to gain access:

1. Head to AutoDesk's Website
2. Click "Get Started"
3. Create a new AutoDesk account or use your existing login
4. You should now see a button saying "Get Educational Access." Click it.
5. Enter the info requested, and then upload a pdf copy of your proof of enrolment letter (available on the SSC) when prompted
6. You should get an email promptly telling you you have educational access.
7. Head back to the same website and make sure you're signed in (top right corner).
8. Now, you should see "Hi ___, Your educational access to Autodesk products is valid…"
9. Scroll down to HSMworks Ultimate and click "Get Product." The latest version is good.
10. Install the software and now when you launch Solidworks you will likely be prompted to sign into your Autodesk account and should see a "CAM" tab in your SolidWorks toolbar



IGEN Tool Library

A list of CNC tools and important information for selecting them is available on this spreadsheet.

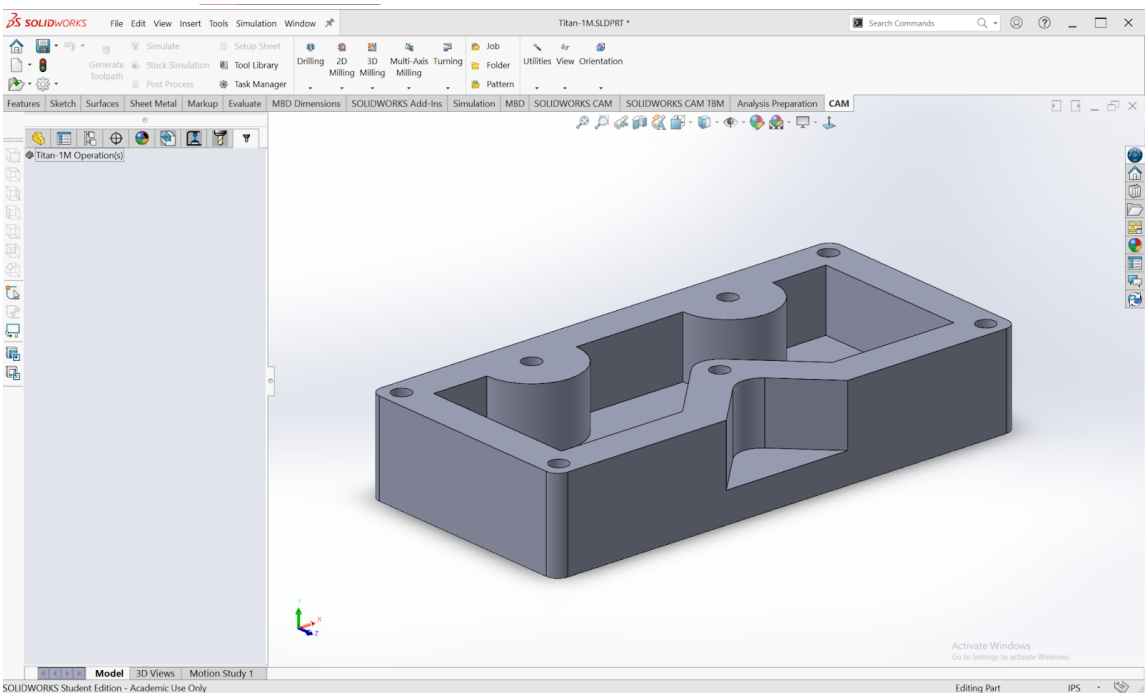
The library can be imported into HSMworks by downloading this file and then navigating to it when making your toolpath.

Planning Your Operation

Once you have modeled your part in SolidWorks, you need to figure out how you are going to machine your part. What stock, workholding method, and end mills will you be using?

Similar to machining on the manual mill, you need to think quite a bit about what piece of material you're going to use, how you're going to hold it without interfering with anywhere you need to remove material, and what cutting tools you'll use to machine the part. Again, if you're unsure of any of these things, read the manual mill document.

1. Select your piece of material, and decide how you are going to hold it. Similarly to the manual mill, the CNC can accommodate a few sizes of vises on its table. It also has t-slot grooves that accommodate the same toe clamps used on the manual mill.
2. Select the endmill(s) you'll use to machine your part. You may need to use ball-nose (round), or corner-radius end mills depending on your part. Let's pretend we are making this part here:



Creating a Toolpath

Now we need to go ahead and create a CAM toolpath. This stands for Computer Aided Manufacturing. Essentially, a toolpath tells the CNC where to cut material away from your piece of stock (stock=material that will become your part).

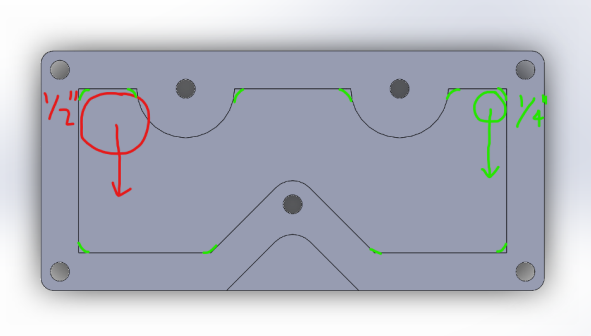
Autodesk HSMworks is the program we will be using to create this toolpath.

Workholding Considerations

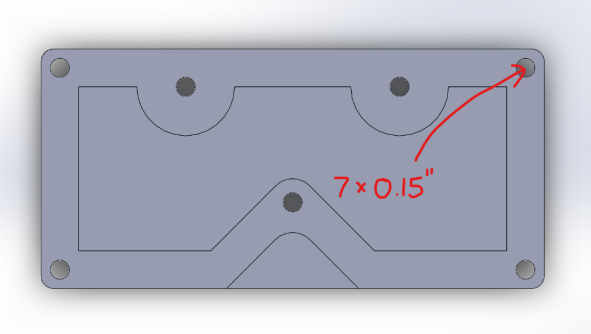
We've found a piece of aluminum stock that is 4.5"*2.25"*1.25", which is a little bigger than our model in each dimension. Given that our stock has two parallel faces, we will hold it in the vise. Given that we need to leave access to the sides of the part to machine these down, we'll have to hold the piece near the bottom. Also, looking at the interior pocket of the part, we can see a few limitations of the mill that we'll have to think about:

Selecting an End Mill

As we can see, using a ½" diameter end mill wouldn't allow us to get into those corner areas. Using a ¼" endmill, we can machine the entire pocket, but we won't be able to hit those sharp interior corners. This is simply a limitation of the mill, as it uses rotating cutters that will always produce some radius on an internal corner.

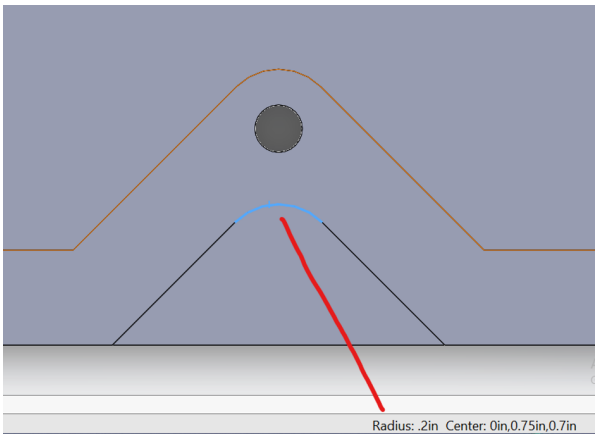


Next, we have these 7 0.15" diameter holes. We can opt to either drill these holes using the closest diameter drill bit we have (5/32" or 0.1562"), or we can machine the holes with an endmill smaller than 0.15".



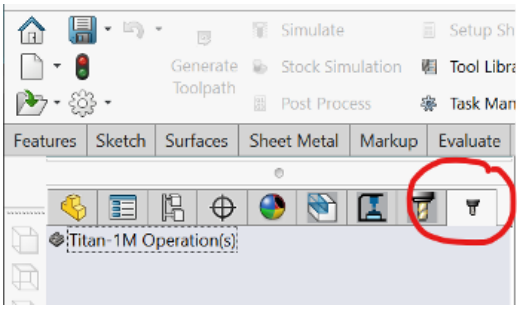
Finally, we need to make sure our endmill can machine the feature on the right.

Given that the radius of our ¼” diameter endmill is 0.125”, which is less than 0.2, we can machine this feature with our selected endmill.

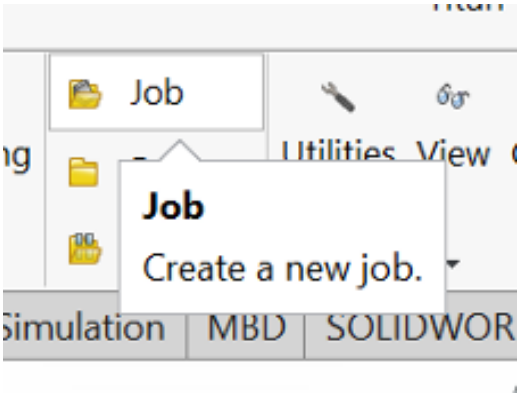


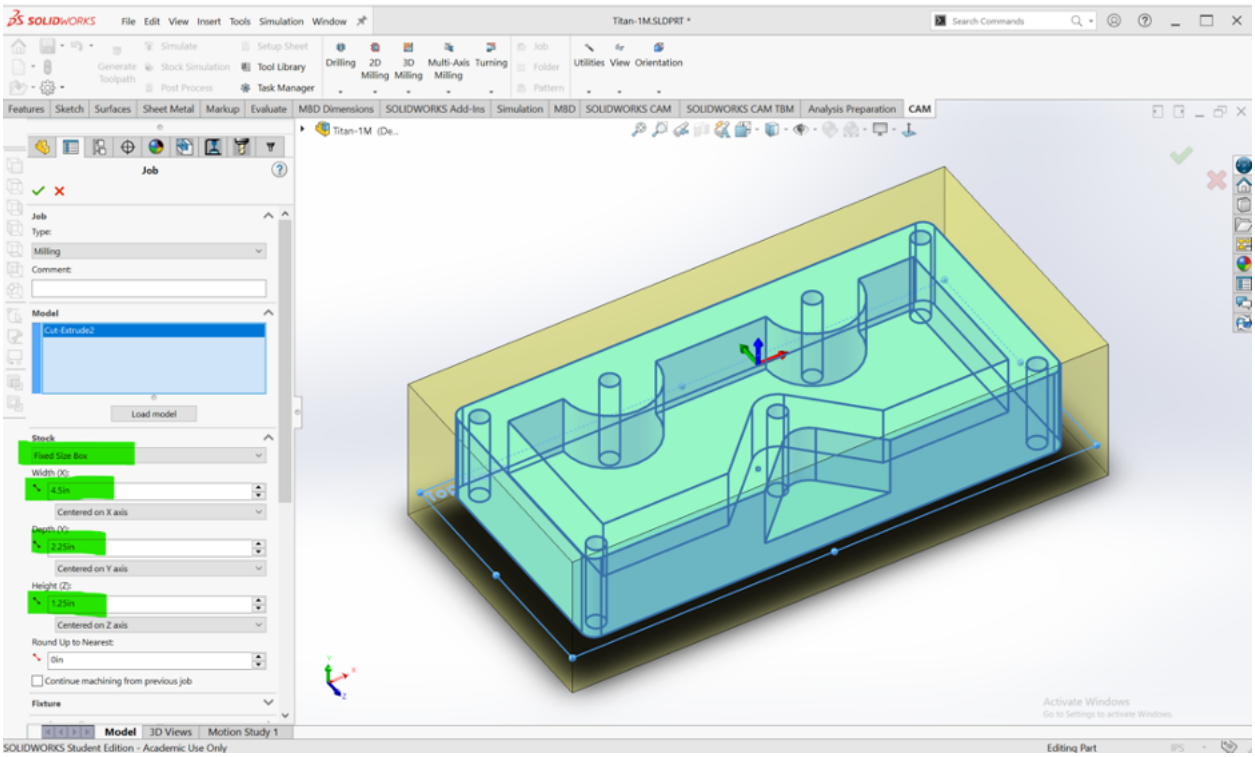
Using HSMWorks

Now we’re ready to create our toolpath. Start by navigating to the last CAM tab on the left window in SolidWorks

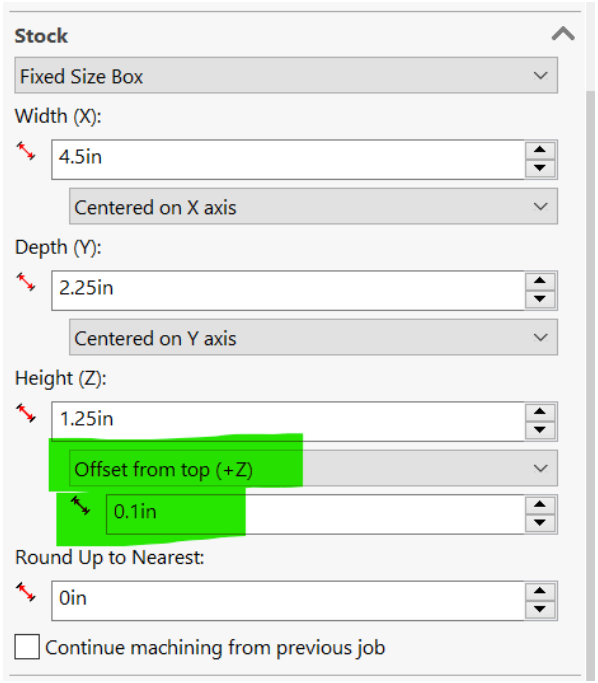


Create a new job, and specify the size of your stock:

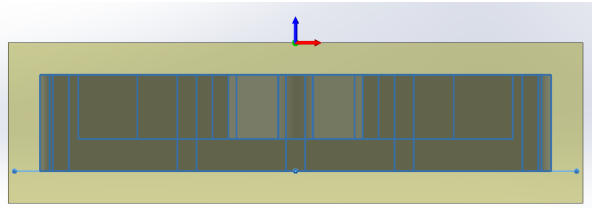




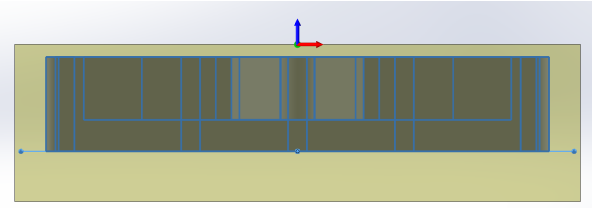
Given that we want to hold the stock from the bottom edge, we'll want to position the part higher up within the stock:



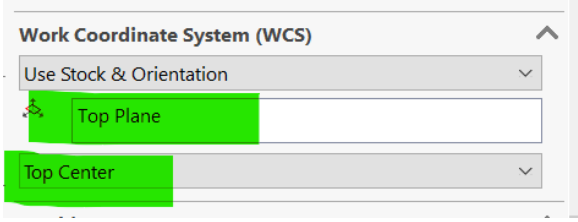
Before:

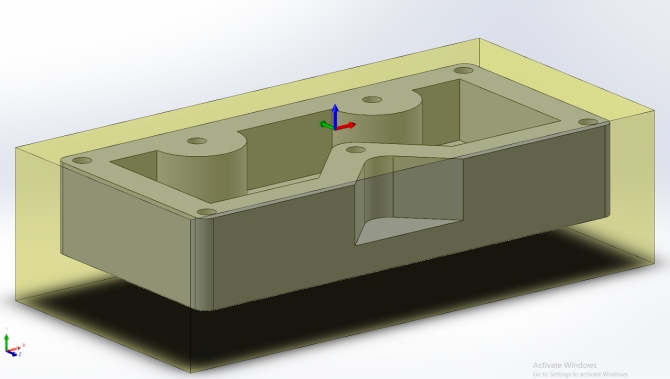


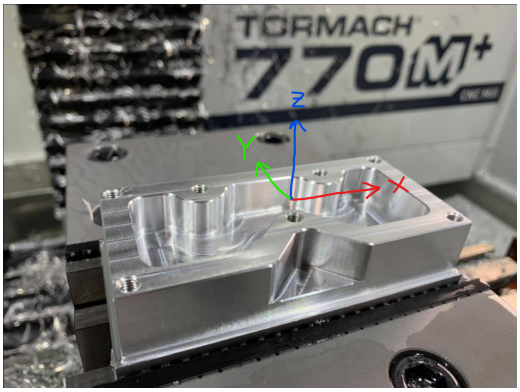
After:



Next, and very importantly, we want to determine where our origin and work coordinate system axes are. The origin is the one point that acts as “zero” for the machine, where x=0, y=0, and z=0. Let’s say that the top center is the origin (as shown above). Essentially, when you tell the CNC to cut something out on the left side of your part, it will move to (-2, 0, 0). If you’re doing a simple part that won’t need multiple operations it’s usually good to pick the top center. Next, you need to ensure that the coordinate axes that the CAM program knows are accurate to the “real world” machine.





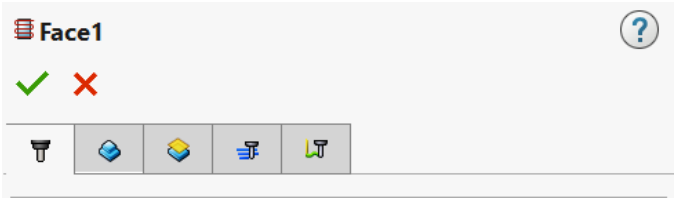


As shown above, positive x is right, positive y is back, and positive z is up.

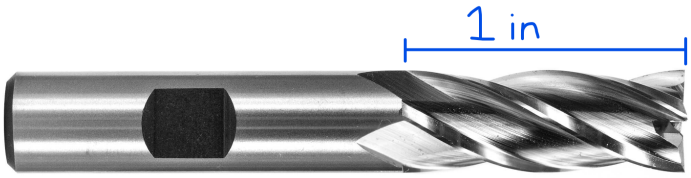
Now that our job is created, we can start creating parts of our toolpath. Operations are separated into a few categories:

3D	2D	Drill
Move X, Y, and Z simultaneously	Move Z separately from X and Y	Drill holes

Looking at the HSMworks interface on the left, every operation is separated into the same five categories. Here’s a description of what each category governs, as well as some things to look out for. **Note that if you hover over anything in HSMworks it will give you more information to look at!**



1. Tool
    a. Select your tool from the downloaded IGEN Tool Library
    b. Igen has a CNC tool table as mentioned previously. The spindle RPMs and feedrates come pre-loaded with the tool, so you dont need to worry about changing them for now.
2. Geometry
    a. Specify what geometry you want to cut
    b. If you want to cut out a certain shape, this is where you'll select the contours you want to cut. Sometimes you may want to limit the area that a toolpath generates to a certain area, which can be defined by a sketch. You can select that sketch here.
3. Heights
    a. Specify what the "top", "bottom", and other heights are that you want the operation to cut down to
    b. Sometimes you'll want to cut a little deeper to ensure that you get all the material off after you've flipped a part in the mill
4. Passes
    a. Specify how much material you want to take off in each pass in X and Y (step-over) and each pass in Z (step-down). There are several other more advanced options here and you'll likely spend most of your time on this tab.
        i. Generally, you want to keep step-over low (around 1/5th to 1/8th of the end mill diameter), and step-down a little larger. Looking at an endmill, you can see that the cutting edges only go up part of the endmill:



ii. Ideally, you want to use more of this cutting length by setting your step-down to around ¾” in this example. The reason for this is that you are using more of the cutter so that it will wear evenly. If you set your step-over small, the tip of the cutter will dull much faster than the cutter edge higher up.

iii. Climb milling is preferable in CNC machining

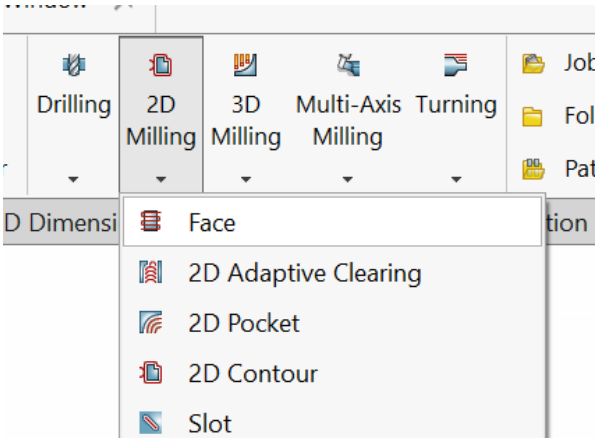
iv. Stock to leave is a great option if you want to come back later with a different tool and perform a finishing pass. Otherwise, make sure you leave this option off so the part comes out just the same size as you designed!

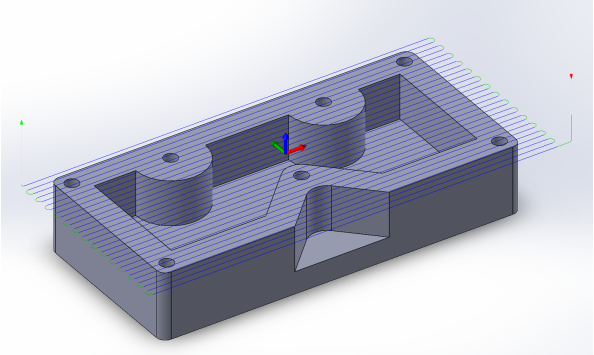
5. Linking

   a. Specify how you want to link the various cutting paths together - retracting, moving over, coming back down, and starting to cut. These aspects are controlled in the linking tab.

      i. Generally, you want to keep the tool down “most.” Keep lead-ins and lead-outs on, and be careful with helix diameters when machining pockets. Helixing is sort of a downward spiral that your tool will take when entering a pocket. If your pocket is smaller than the helix diameter, you may notice that the tool just won’t enter that pocket. If this is the case, decrease the helix diameter.

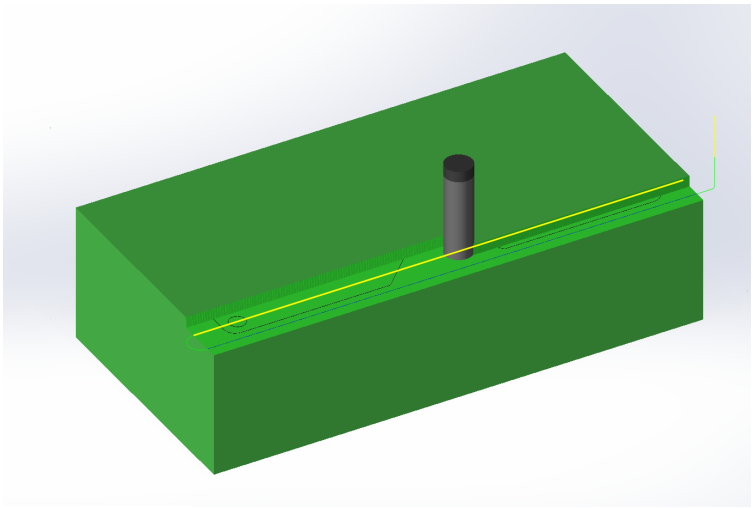
As a first step, it’s usually a good idea to face the top of your stock down to the top of the part:

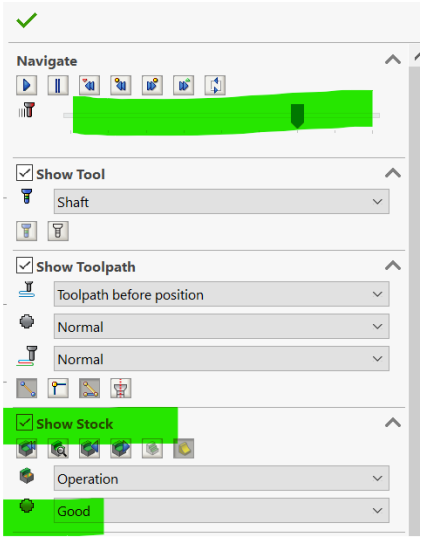




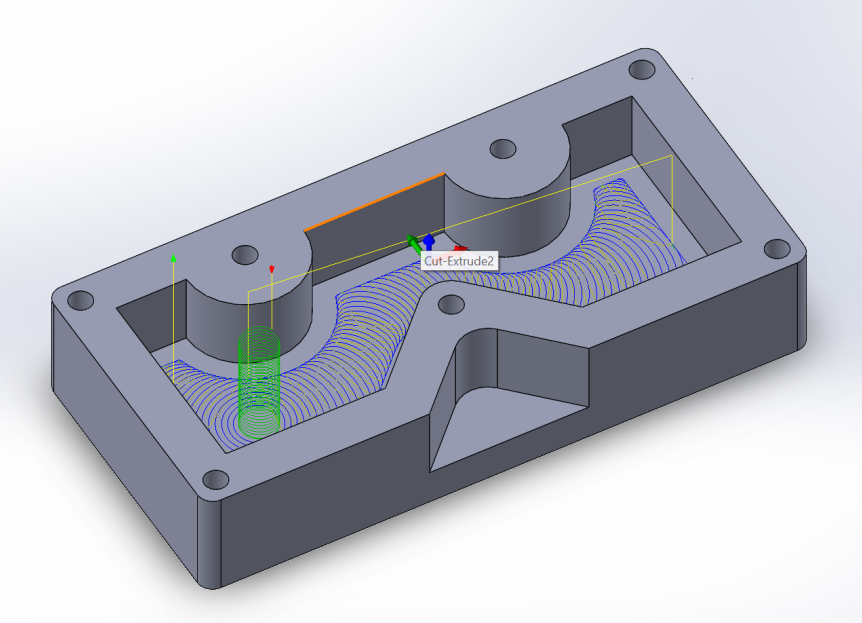
And voila! We have a toolpath. The blue line represents the center of the bottom of the end mill as it cuts. Next, we'll look at one of the most useful and important aspects of this process: **simulating your toolpath:**

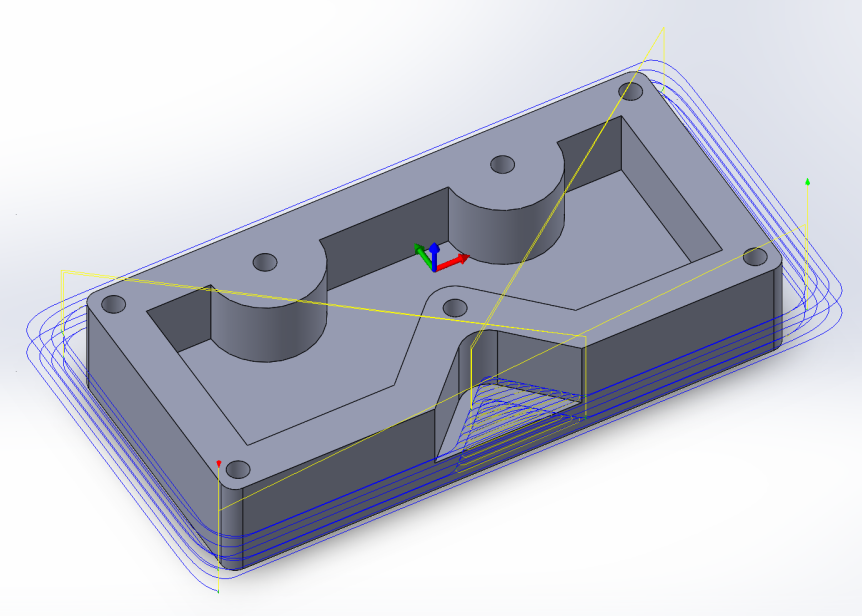
Hit the play button, and use the slider to control the speed of the simulation.

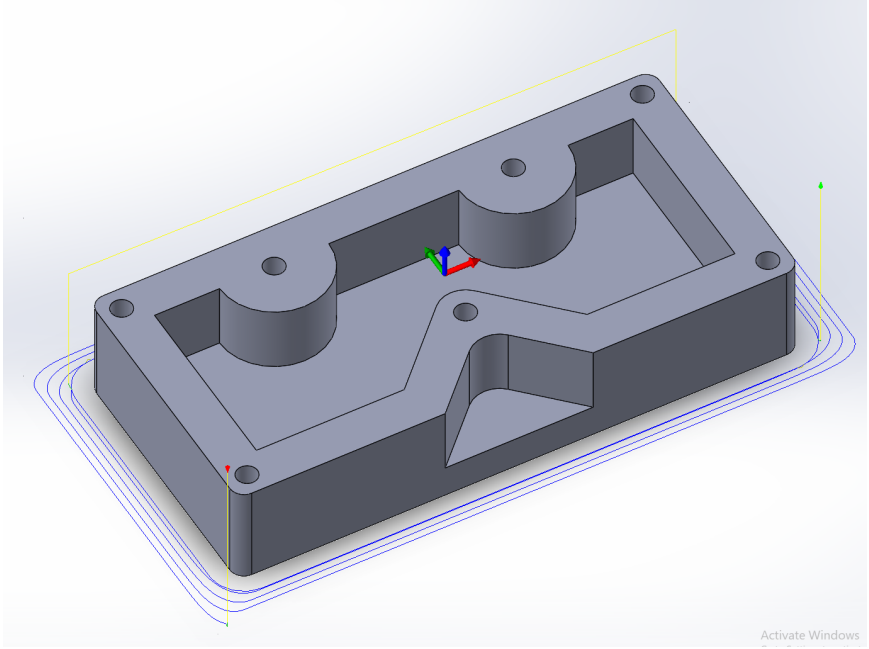




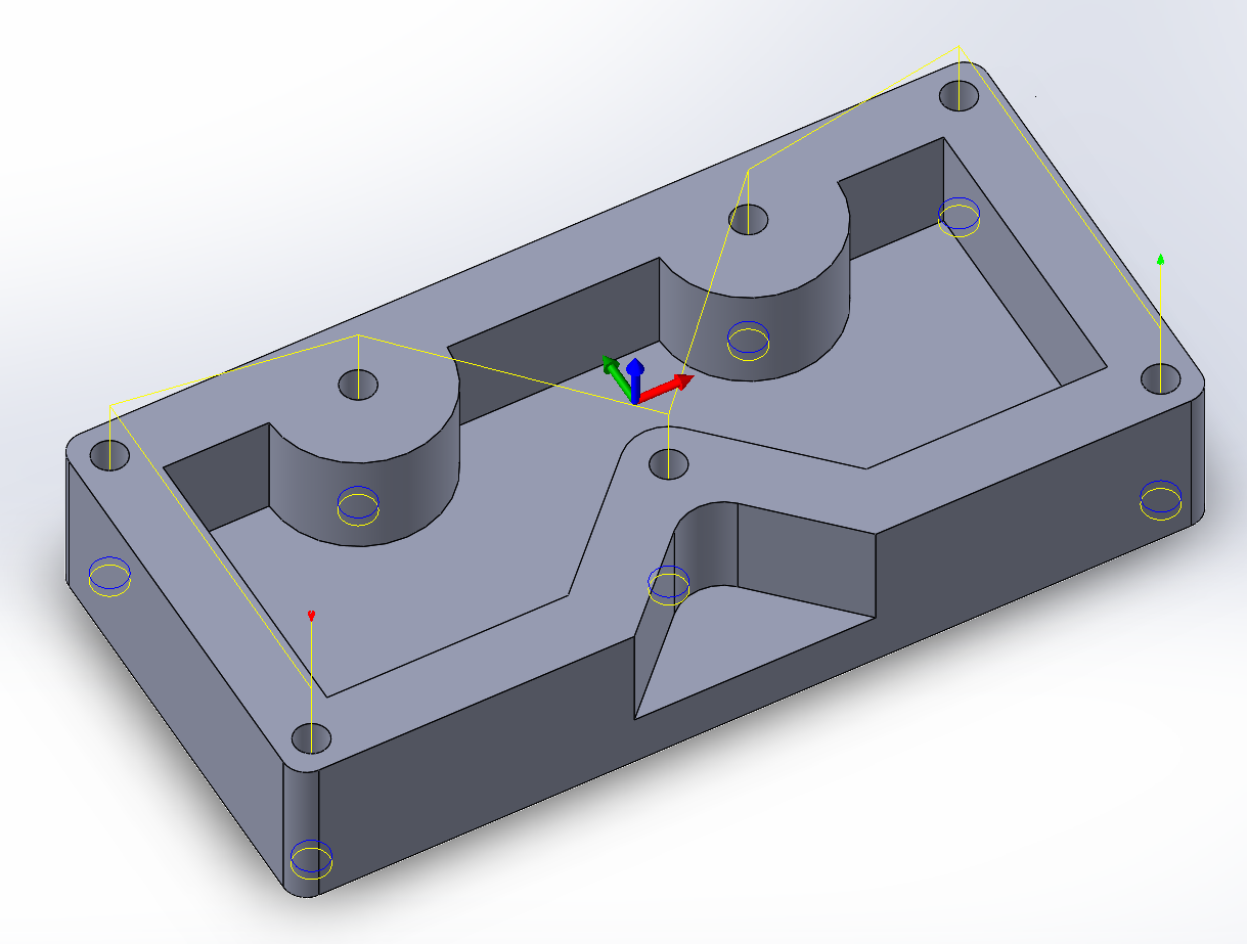
Next we can create a few 2D adaptive clearing operations:


Cut-Extrude2

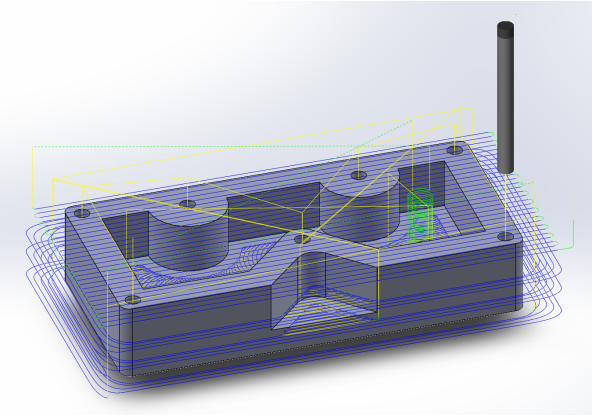


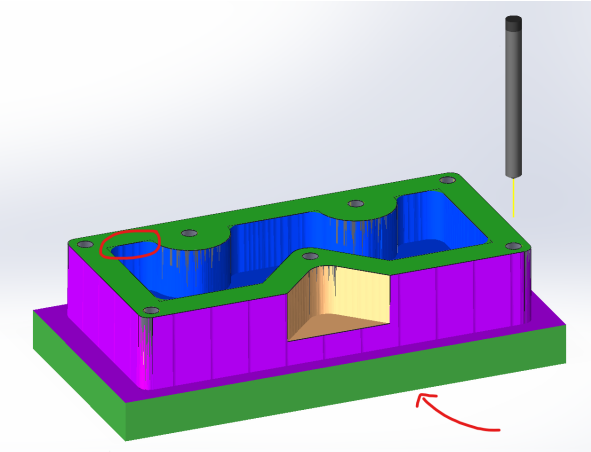


And finally a drill operation, selecting the bottom edges of each hole:



You'll want to simulate often, as it will help you see mistakes or oversights in your CAM toolpath planning, and allow you to fix these while you still can!

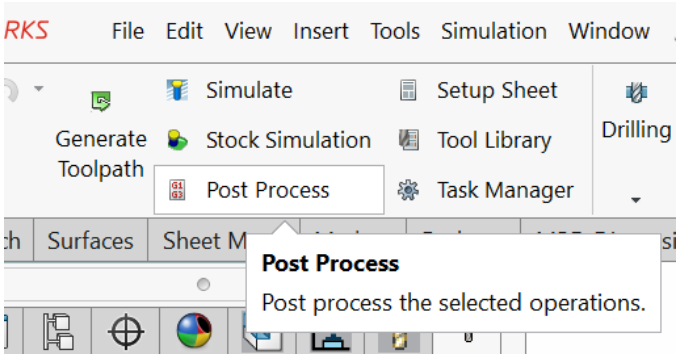


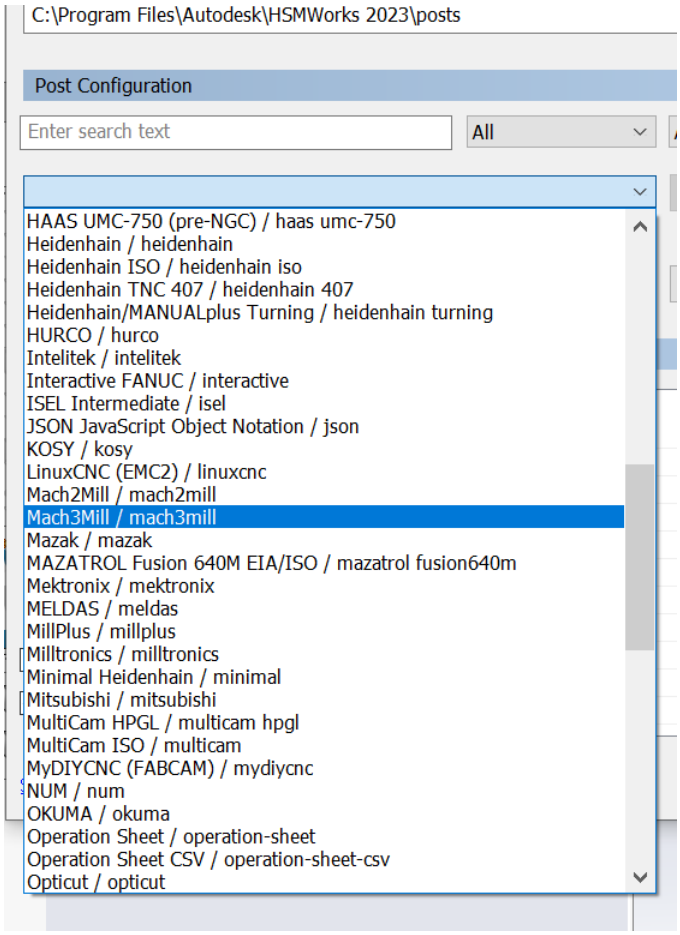


Here you can see that the sharp interior corners aren't quite reached, and that we still have the remaining block of aluminum underneath the part that our clamp is gripping.

Sending the Toolpath to the Mill

When we are happy with our toolpath, we can export it to a usb stick:





Select Mach3Mill as the post processor, and click “post.” Then drag the .NC or .TAP file to a flash drive and bring it over to the CNC. Plug it into the Tormach and turn the tormach on using the rotating switch on the right side of the machine.

Make sure to reference x, y, and z before continuing. This moves all the way to the end of the machine’s travel and trips a limit switch, which lets the machine know where in its travel it is. Now, you can install your stock in the vise, mark the top center as accurately as you can, and load the endmill into the mill.

If you need to manually change the tool, there is a little button on the left of the ATC. If you press and hold this, the tool and collet will pop out of the CNC. Make sure you’re holding the tool with your other hand so it doesn’t drop and get damaged! Loading in a new tool works the same way, by holding the button while pushing the collet up and into the spindle. Make sure to align the two large notches in the collet with the divots in the spindle. When you release the button, it will suck the collet in, and that’s it.

Begin manually moving the tool until the bottom center of the end mill is close to the mark you made. To move around manually, you can use the keyboard up, down, left, right, page up, and page down.

To dial in the origin you’ll need to switch from continuous jog mode to stepping mode, where your keyboard inputs will move the tool in discrete increments which you can select on the touchscreen. Try to get as close as possible visually in X and Y, and zero X and Y. Then turn on the spindle at a low speed (2000 rpm) and incrementally lower the z until you are very close to cutting the top of the stock, decreasing the increment as you get closer. As soon as you start shaving off a tiny bit of metal, zero the Z as well. Now your origin is set! You can turn off the spindle and move the tool up and out of the way.

You’re now ready to hit cycle start, close the doors, and watch to make sure that the CNC is doing what you expect. If at any point it’s doing something wrong or looks like it’s going to start machining the vise, pause the toolpath or smack the big red button if things are really bad. Pausing will allow you to potentially recover, but there will be a delay after pressing it, whereas the big red button is for emergencies and will immediately halt the machine’s motion.

After Machining

After making your part, clean up the chips by putting them in the designated chip container. This gets dumped in the metal recycling by shop supervisors when full. It is acceptable to leave your tool in the machine since the next person may use the same tool and can avoid having to reinstall it. If you used the vise to hold your part, leave it in place as the next person will likely use it. If you clamped your workpiece directly to the table, replace any toe clamps or other fixturing you used and store any material you put underneath your stock back where you got it.