CNCing Foam at SCD using Fusion360 CAM

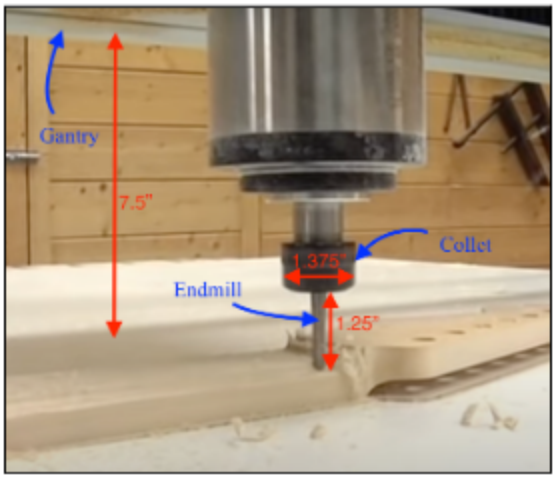
2/24/22 by Christopher Xu (cyx3)

^insert big distraction so you scroll down

Purpose and disclaimers

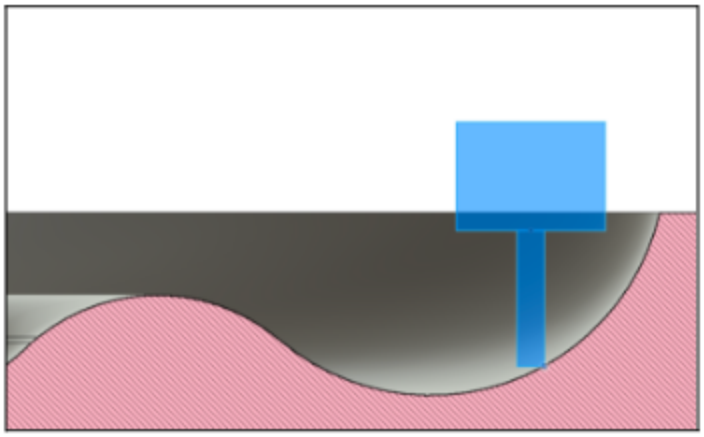
This is not a tutorial; it's more like the procedure section for a middle school science fair project, describing how Tiffany (tl52) and I milled a 2ft. x 2ft. wheel cover mold. I've only cut one real piece so don't trust me too much.

Limitations



The Laguna Swift CNC at SCD has horizontal dimensions of 4ft. x 4ft. The gantry height is 7.5 inches, but that doesn't mean it can cut anything shorter/thinner than 7.5" tall. Endmill clearance means that the realistic maximum foam thickness is probably 6". You could shorten the endmill, but the side of the collet may hit long/steep vertical faces in your piece. See the next part about checking collisions.

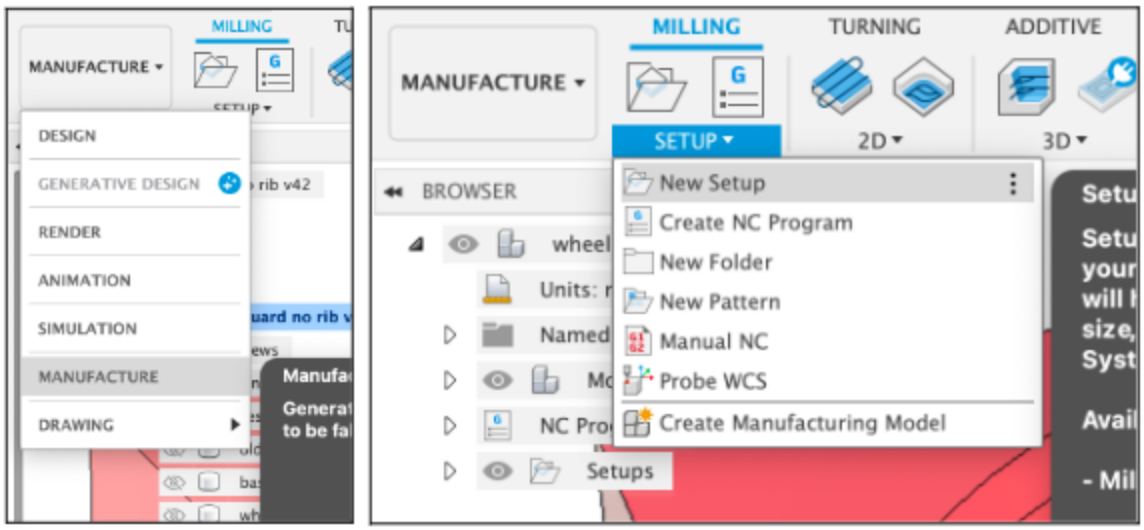
Process



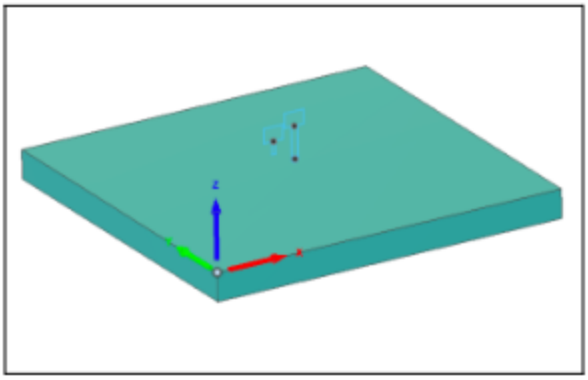
1. In your CAD, make sure the collet doesn't hit the final piece.
   - Here's my method: measure the dimensions of the collet (about 1.375" dia.) and the endmill to **create a sketch of them in the CAD**. Drag it around as if it were cutting the bottom of the part and see if any non cutting part hits the piece. If it does, try using a longer endmill or clamping the endmill at a higher point so that it sticks out more.

- Make sure the longer endmill is still short enough to fit under the 7.5" gantry without hitting the stock foam.

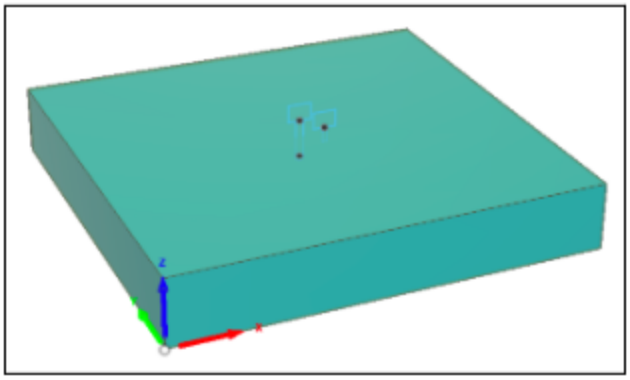
2. If you need to mill at depth greater than would fit, slice the piece horizontally into multiple layers in a way that fits. Update in the CAD so the mold consists of multiple bodies, and proceed with the next steps for each body.
3. **Model the stock piece**. Measure your foam's dimensions to make a block in CAD.
4. **Manufacturing workspace -> Setup**
    a. Where to click:



    b. Machine: select the "**Autodesk Generic 3-axis XYZ axis on head**" in the Fusion360 library.
    c. Setup Operation Type: **Milling**
    d. Work Coordinate System: Make sure the stock model is visible. Use any of the selection options to place the **zero at the corner of the stock mounted closest to the CNC controller**.
    1) For most pieces, use the tallest part of the stock.
    2) For tall pieces with little clearance between the endmill and the stock, use the board the foam is mounted on so zeroing Z will be easier.



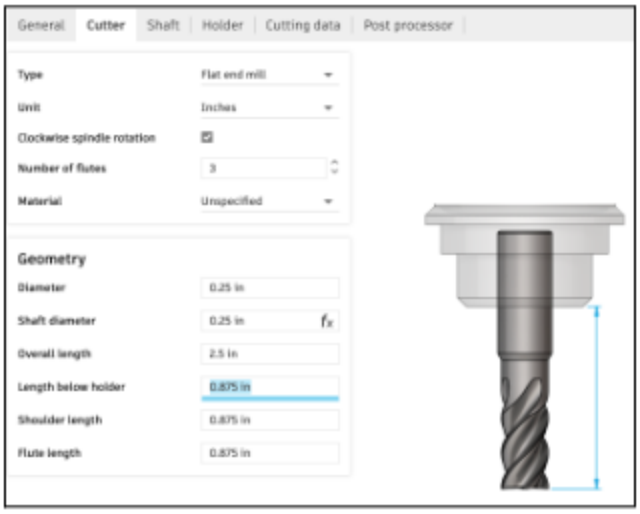
Case 1 (most common)

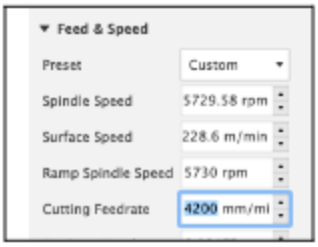


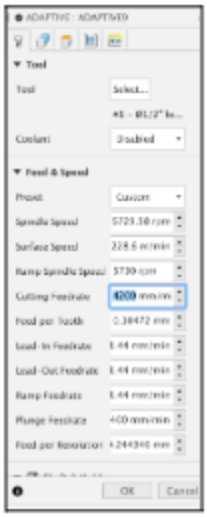
Case 2

e. Model: select your final milled piece
f. Stock (2nd tab): **From solid** and select your stock model
g. Ignore the rest and click [OK]

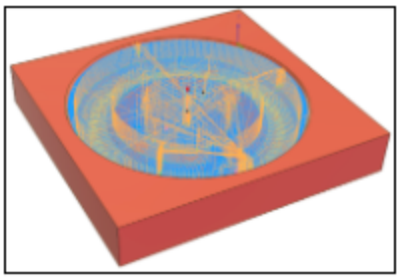
5. Create and generate 3D operations
   a. First, use **Adaptive Clearing** to rough out the majority of the piece. When selecting a tool, make sure to measure and enter "Length below holder". We had success with a **½" diameter ball end** using the Plastics-Roughing preset with Feed Speed = **4200mm/min**. Might take a few mins to generate after clicking [OK].



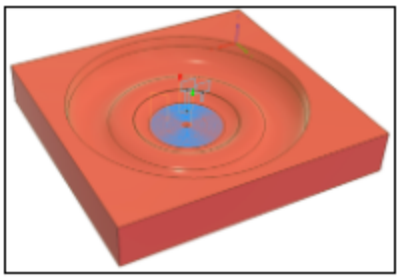




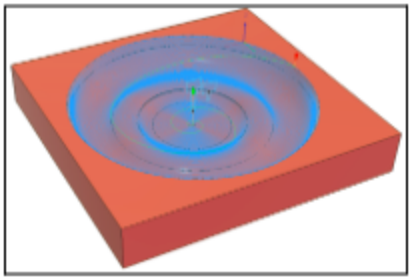
   b. Smoothing: Right click the new Adaptive operation on the left. Choose the options in **Create Derived Operation** that match the geometry of the piece and what endmill you want to use. [Tip: You can hover over the options to see an explanation of each.] Make sure to select the correct endmill while editing the new operation, but otherwise the default options seem to work fine. If one smoothing operation does not properly smooth the entire piece, right click on the last operation and do another Create Derived Operation to form a chain.
   We used **Flat** with a flat endmill for the middle part of the wheel guard and **Scallop** with a ball endmill for the rounded walls.



Adaptive

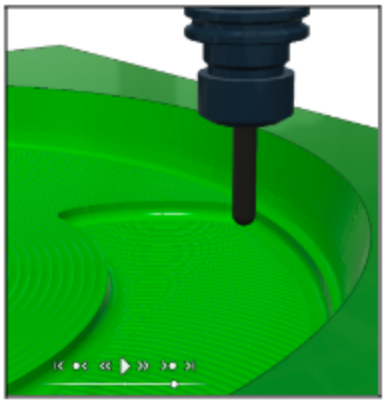


Flat

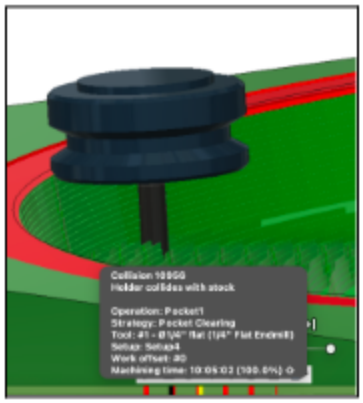


Scallop

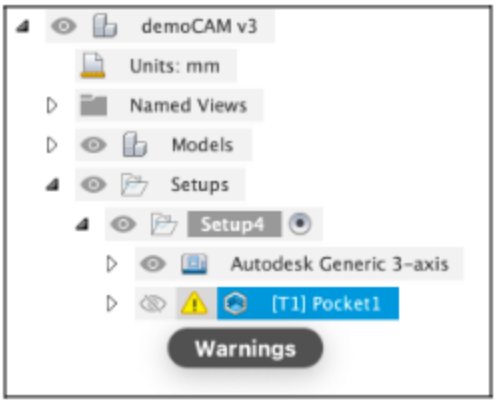
c. Right click on each operation and **Simulate** them. There should be a green bar on the bottom, and any red bands means that there is some collision. Edit the operation (right click) and check the **Shaft & Holder** box, select Detect Tool Length. Generate it again, and the software should tell you how long the endmill should be for nothing to hit. Possibly go back and edit the CAD, or make the endmill longer.



endmill long enough :)



uh oh



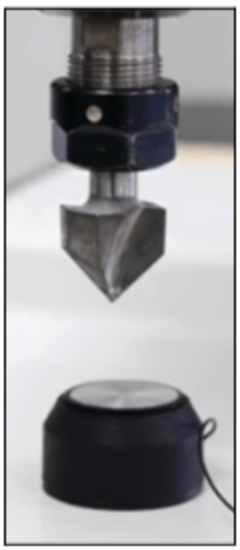
click warning for required length

6. Post process into an .mmg file
    a. Download the [Laguna iQ & Swift postprocessor for Fusion360](). Alternate [link](). For MacOS, move the .cps file into **/Users/ <username>/Autodesk/ Fusion 360 CAM/Posts**.
    b. Right click on each of the operations on the left (like Adaptive1 or Scallop2), click **Post Process**. Select the Laguna Post processor and rename the output something that has a recognizable first 4 characters. Longer names will be truncated on the CNC controller. Each operation will have its own file. Remember which bit each one uses.
7. Save all of the **.mmg files to the Laguna CNC usb drive** (silver 8GB). Rishi (cargo fast) says that's the only one that works, so let's trust him.
8. Plug in the usb drive into the controller and turn the machine on. The first prompt should be asking you to home the machine, so do that. Then use the ±XYZ buttons to move the cutting head somewhere out of the way, so you have space to clamp down the piece of foam onto the bed.
9. Foam securing
    a. Put the flatter side on the bottom. May need to surface to make both sides flat (see Surfacing).
    b. If the foam is less than 1.75 inches thick, put the block directly on the OSB as squarely as eyeballs possible and clamp the top of the foam to the rails on the CNC bed.

c. With thicker foam, drive **wood screws from the bottom of the OSB** up into the foam, then **clamp the OSB to the CNC bed**. Make sure none of these are in the way of the toolpath. Try to space the screws so that the screw heads fall into the grooves on the CNC bed (27"? apart).
d. Push on the foam to check if it's secure

10. Surfacing
    a. Optional: you might want to flatten one surface because it was cut with hand saws
    b. Put in a flat wide bit that looks like this —>

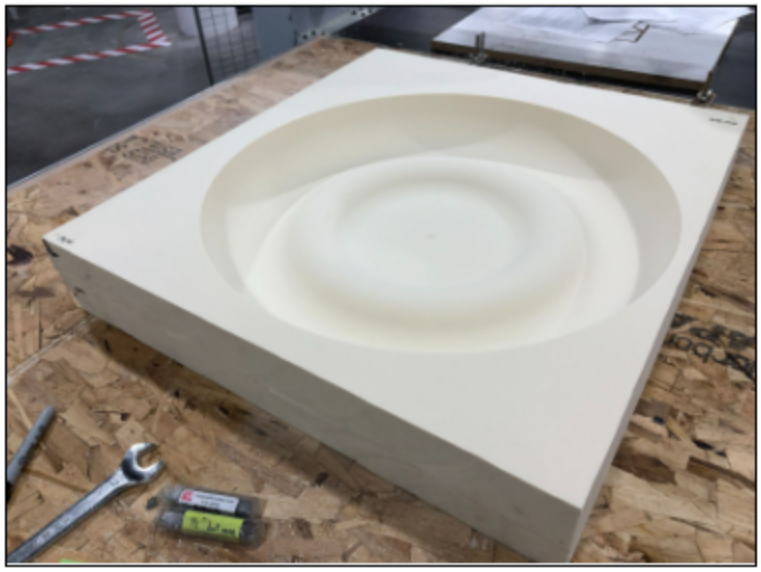
    c. Once the foam is clamped/screwed, press [ON/OFF] to start spinning the spindle. Use the CNC controller buttons to move Z down to the material then start **sweeping around in the XY plane** to level the top surface.
    - For a 1" diameter bit, try spacing the sweeps in increments of 25mm.
    - You can go pretty fast on low density foam, but be careful about tearing up the foam underneath.
        - If that happens, move XY slower (toggle H/L button) or increase the RPM (menu button->machine setup->toolset->delete button->number pad buttons).
    d. Alternatively, generate a Face operation in Fusion360. Probably the conventional and more efficient way but we haven't tried. Main reason is sometimes you don't know how far down to cut.
11. Zeroing
    a. Put in and secure the correct endmill. Measure the length extending down from the collet to make sure it matches the length in Fusion360.
    b. Bring the Z axis all the way up
    c. XY zeroing: Use the controller buttons to position the endmill directly above the zero point of the piece, matching the setup in Fusion360. Press the **XY->0 button**.
    d. Z zeroing:
        i. If the zero is set on the top of the stock, move XY closer to the center of the stock. If the endmill moves down, it should hit the foam.
        ii. If zero is set at the bottom, move XY off the stock but close to the zero position. If the endmill moves down, it should hit the OSB board under the foam.



iii. Hold the zeroing puck (right pic) and press the **menu and on/off buttons at the same time** to automatically bring the Z down. Try to hit menu slightly before on/off because on/off alone starts the spinning. As soon as it starts moving, touch the top of the puck to the endmill and it should immediately lift back up. This is a check to see if the puck works and it won't keep going down during the actual zeroing process.

iv. Hold the zeroing puck to the top surface of wherever you set the zero and press menu+on/off again. Wait for the machine to bring Z down and back up. Check to see that the Z value on the controller becomes 0 at the bottom.

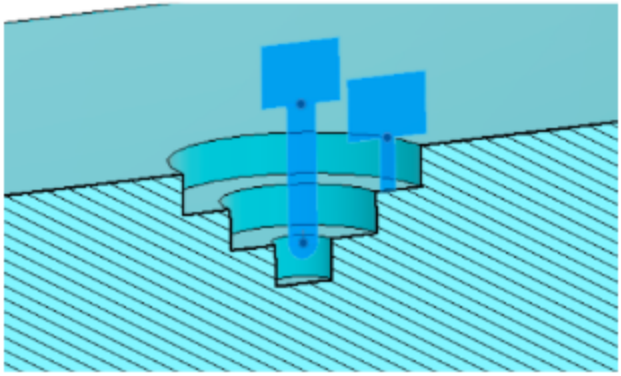
12. Actually running the files: **Press run/pause and udisk read**. Scroll down to your first operation and press start a few times, keeping the default settings. **Tap your GRIT card** to refresh the timer. After a countdown, the operation should start. Get a shop vac to vacuum the chips as it goes, unless your piece is flat enough that you can use the orange vacuum attachment.



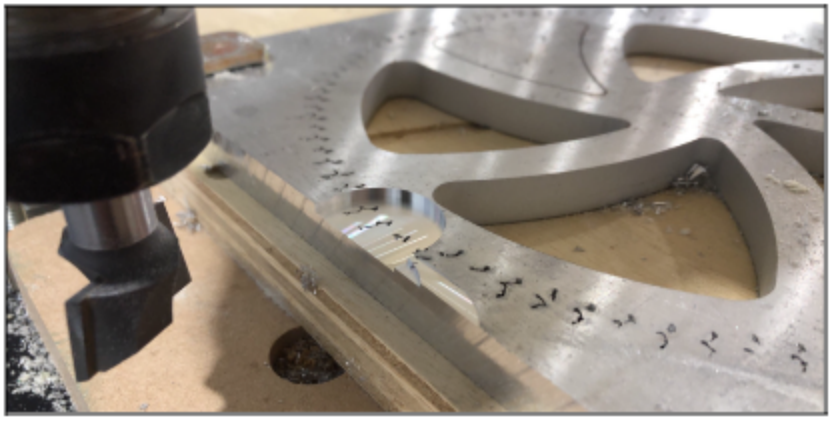
yay!

Possibly useful ideas

- If the foam is too thick to fit the entire endmill, first dig a hole with a shorter endmill. The hole might need to be stepped in order to fit the shorter endmill deeper:



- If you complete an operation on the CNC but have to leave, try to record the XY zero position by drilling a hole in an unused area of the stock piece at a known position. We used the controller buttons to drill a hole with the ½" ball endmill at the (10, 10) and (590, 650) positions and labeled them so if the CNC gets rezeroed, we can find the XY zero again.
- Fun fact, you can also cut aluminum by moving slower and spinning really fast (16000 rpm):



A few of the helpful people at SCD who know CNC:

- Neil Pearse (npearse@illinois.edu), Laboratory and Equipment Operations Manager
- Matt Blessing (mbless2)
- Rishi Mohan (rishinm2)
- Gabe Tavas (gtavas2)
- Hongbo Yuan (hongboy2)