

The following links were used in my Fusion Heart Wax Mold process from Design to Manufacturing (CAM) to physically milling it out on the Othermill (Bantam Tools).

[Instructables - How to Use Fusion 360 with Othermill](#)

[Fusion 360 and Othermill Overview](#) - **IMPORTANT/RECOMMEND**

[Fusion 360 Tool Library](#)

Fusion Shopbot Process

[Using CAM to create a pocket in Fusion](#)

[Long process/Full Video for Fusion with OtherMill](#)

[Support File Formats for Othermill](#)

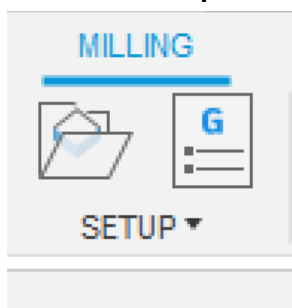
[Fusion CAM Overview](#)

[How to create g.code in Fusion](#)

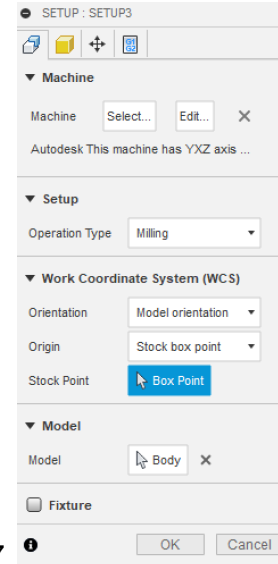
[Fusion 360 Manufacturing Support](#)

**In progress - 3.23.2022

1. Change workspace to Manufacture
2. Click **new setup** in setup drop-down menu



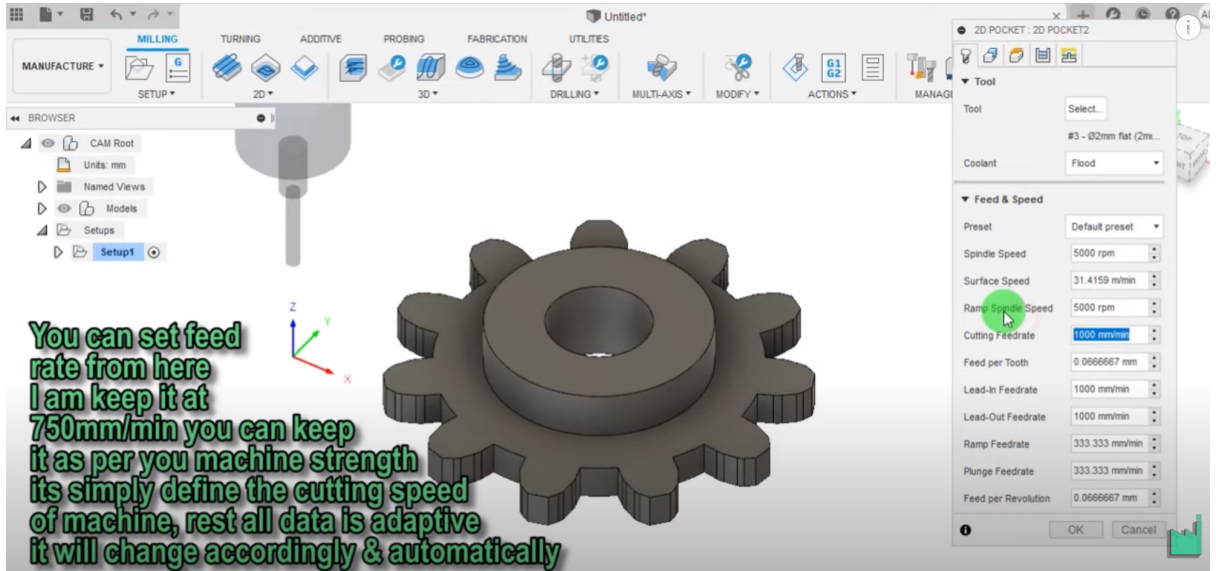
3. In the dialogue box, in the setup tab, select machine as Autodesk Generic 3-axis (many options will pop up, doesn't matter which you select).
4. In the stock tab, change the mode to a fixed-size box
5. Under height, change the model position to **offset from the top**
6. Change the offset to 0
7. Click ok in the dialogue box to complete the setup



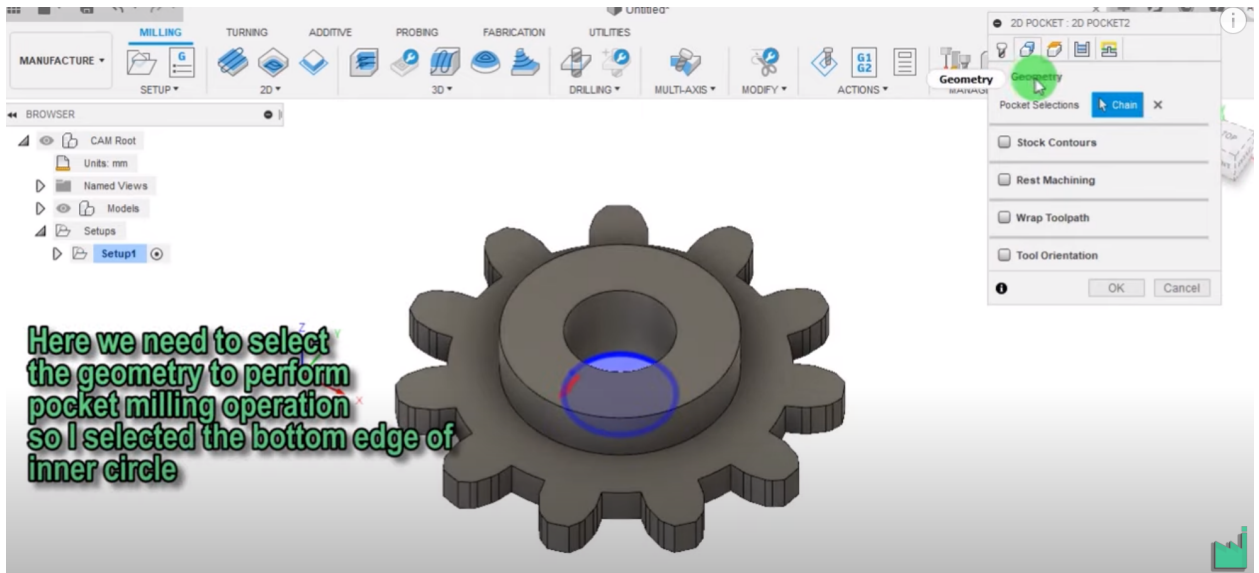
- a. This image is the dialog box referenced in steps 3-7

WARNING - STEPS 8-15 will vary between 2D and 3D toolpaths depending on your design, as well as you may have different tools for your milling process. The following information can help you understand how to navigate this process.

8. In the toolbar, under the 2D tab, you can click pocket clearing for the following example. You may need to choose the 3D tab for your particular design. (Pocket Clearing is only in the 3D tab).
9. In the dialogue box, in the tool tab, select the tool as “1/8” R1/16” (1/8” Ball End Mill)” from the other mill tool library. You will need to install the [other mill tool library](#). **(Extract only othermill library after downloading the fil. Right click on loca library to import only that file).**
 - a. If you need to use a smaller bit (i.e. 1/64”) for finer detail, you will need to complete a second toolpath
10. The geometry tab will be used to select the best path for yourself in the 2D process, but it may be unchecked in the 3D process.
11. Click ok in the dialogue box to complete the pocket clearing

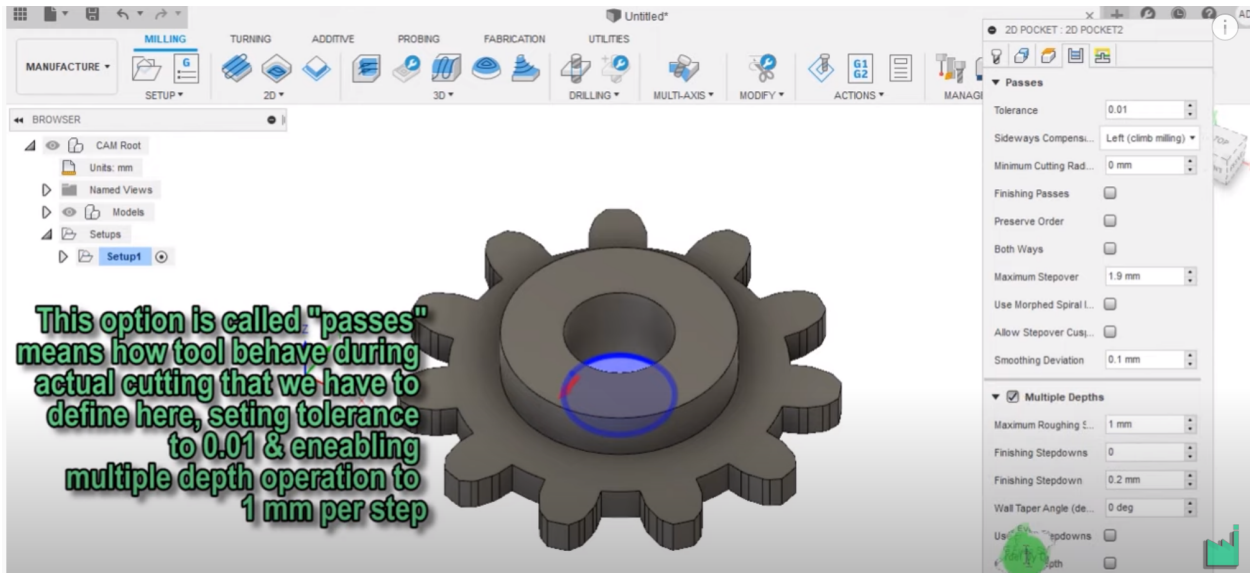


a.



b.

c.

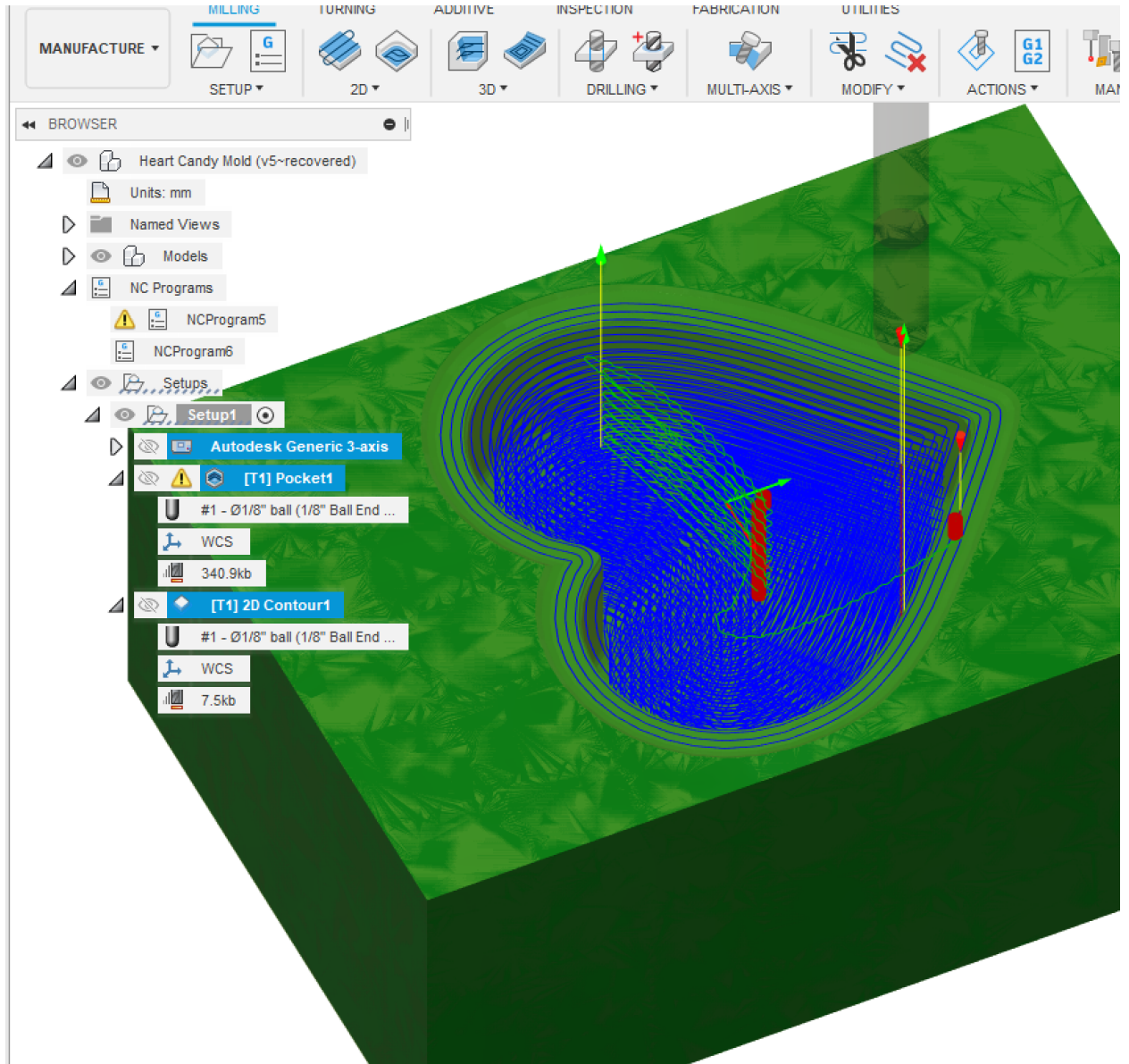


d.

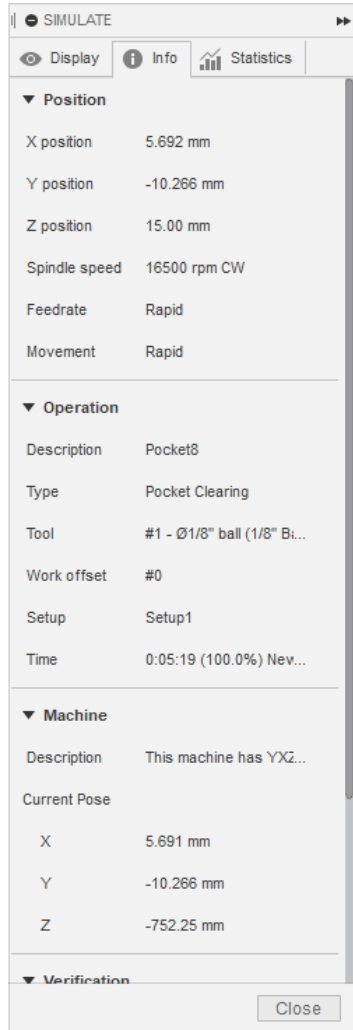
12. Click yes on the warning that appears

13. The previous steps can be done for other toolpaths, such as a finish pass using a 2D contour.

- a. The image below shows an example of two toolpaths: 2D pocket as roughing pass using an $\frac{1}{8}$ in ball nose bit, and a 2D contour as a finishing pass using an $\frac{1}{8}$ in ball nose bit.

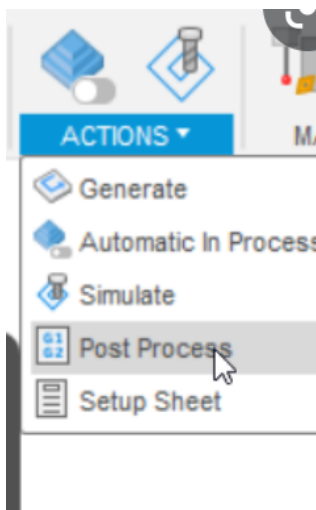


14. Under your pocket toolpath on the left side of the screen, you should simulate your pocket to see how it is going to work. Under the Info tab of the simulation window, you will see an estimated length of time for your file to be milled. Furthermore, you can uncheck the Machine view to get an easier view of your simulation too.



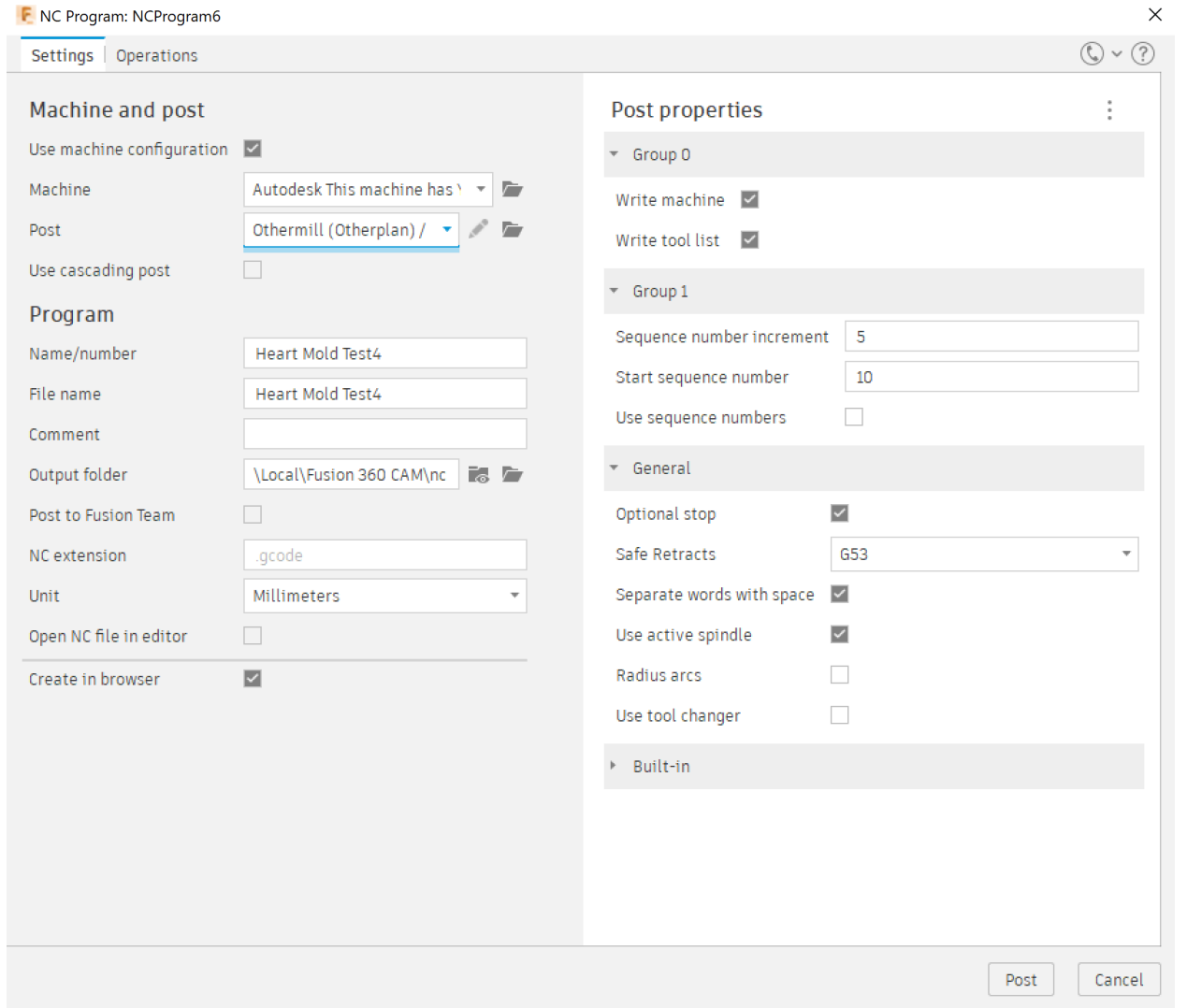
a.

15. In the Action tab of the toolbar, you will choose **Post Processing**.

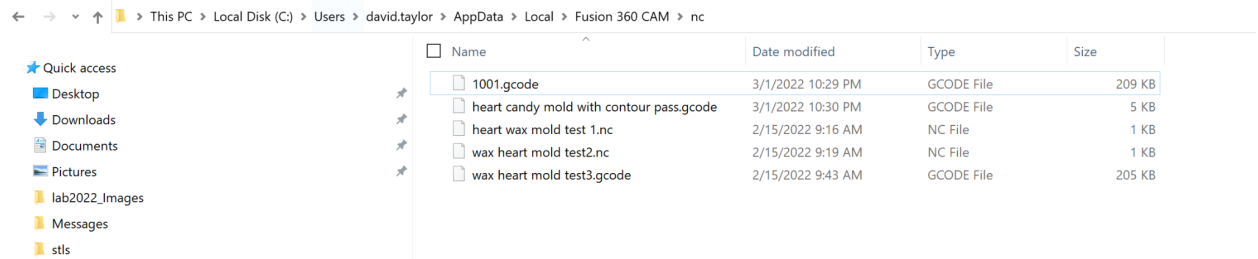


16. Post Process will transform your file into .g-code, which is a computer numerical control (CNC) programming language.

17. Under Post Process, you need to set the Post to **Othermill (OtherPlan)**, you may need to select this using the folder. **(You can also use the “Search” feature in the top and search “other” when prompted.)** Next, name your file under the Program and make sure the Unit is Millimeters. Please pay attention to where the Output Folder (see image “b” below) is located due to this being the location your .g-code file will be saved. Please see the example image “a” below. **Notice the NC extension; it should say “gcode”.**



a.



b.

15. Load your .g-code into engproj from your computer and then download the file onto an OtherMill desktop computer.

16. Next, please load your file in the Bantam Tools software.
18. You need to put in the material sizes and move your file location to fit the center of your material.
19. Please select the appropriate "wax" bit (ends with "-wax").
20. Check to see how long your estimate file will take to be milled on the OtherMill.

If there is an "initializing" error, power the milling machine off and turn back on.

The file must end in gcode, not "nc".

You do not need to "home" the bit or locate the tool. This is done automatically.

Allow cleanup wizard when prompted (will move the board forward).

Vacuum all the shavings and area around the drill bit.