

# Fusion 360 Part Setup

Tutorial



# Table of Contents

[MODEL SETUP](#)

[CAM SETUP](#)

[TOOL PATHS](#)

## MODEL SETUP

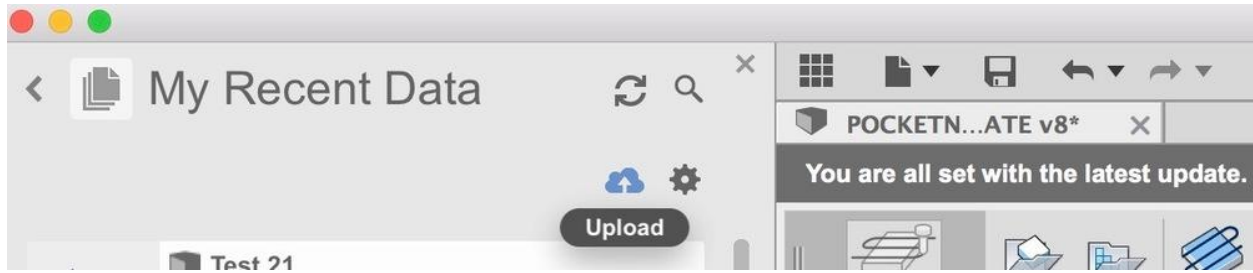
The purpose of this tutorial is to demonstrate start to finish, importing a machineable part to generating toolpaths in the Fusion 360 CAM environment to cutting the part on your Pocket NC.

When setting up any part on the Pocket NC mill, it is recommended that users include the Pocket NC B Axis table and vise models (or other work holding) in the CAM environment. This will help users visualize orientation of the part and table to avoid collisions.

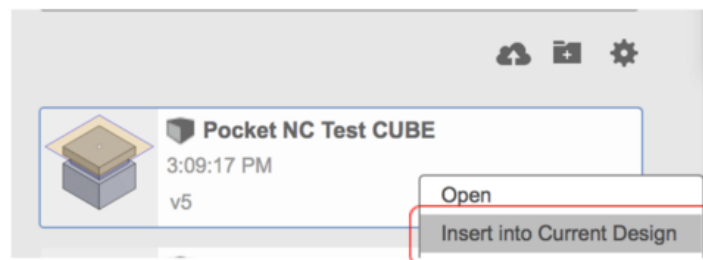
To use the B axis table and vise model, download them [here](#). Also be sure to download the 6 sided cube used in this tutorial [here](#). Because Fusion 360 is cloud based software, users will need to upload both models to Fusion. To do this, first open the data panel.



Then select upload and migrate to the file location.

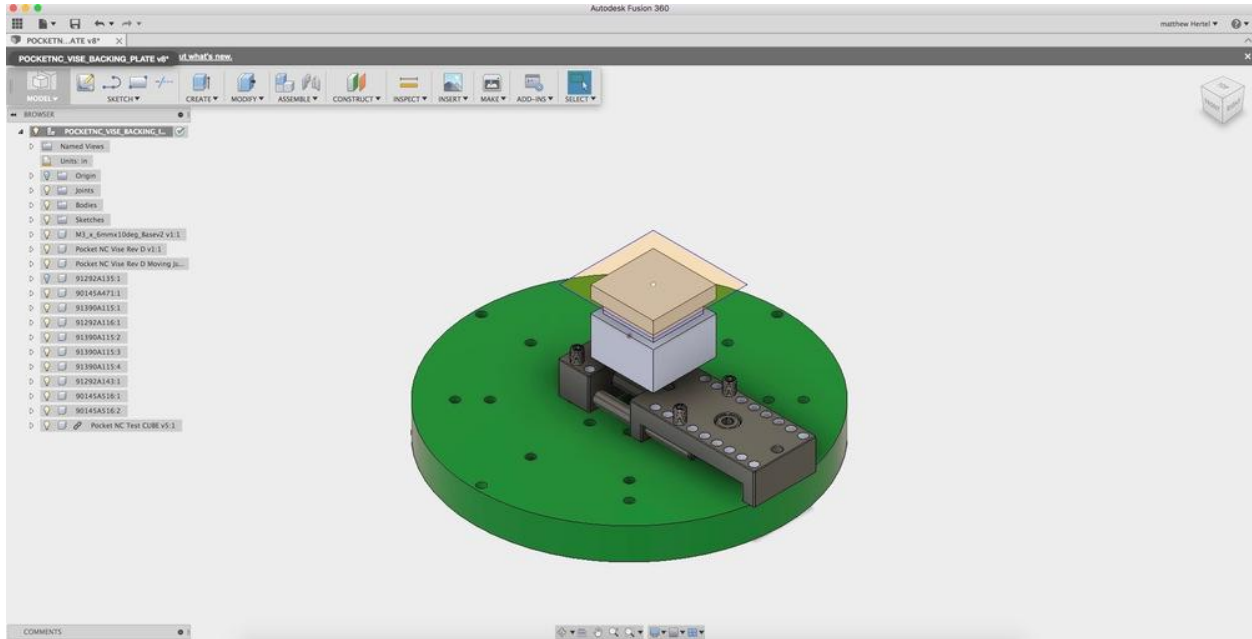


Files uploaded to Fusion will stay within the Data Panel and can be accessed any time in the future from that location. With the files uploaded, open the Pocket NC table and vise model by right clicking the file and selecting open.

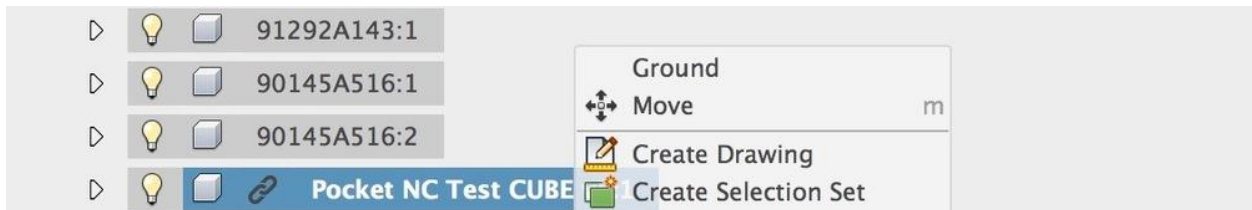


Next import the Pocket NC Test CUBE into the current setup by selecting “Insert into Current Design”

After all the models are in Fusion 360, users will set the orientation of the part relative to the vise or table so it matches that of the setup on the Pocket NC mill. A transparent square located atop the Test CUBE represents the size of the material the Test CUBE will be machined from.



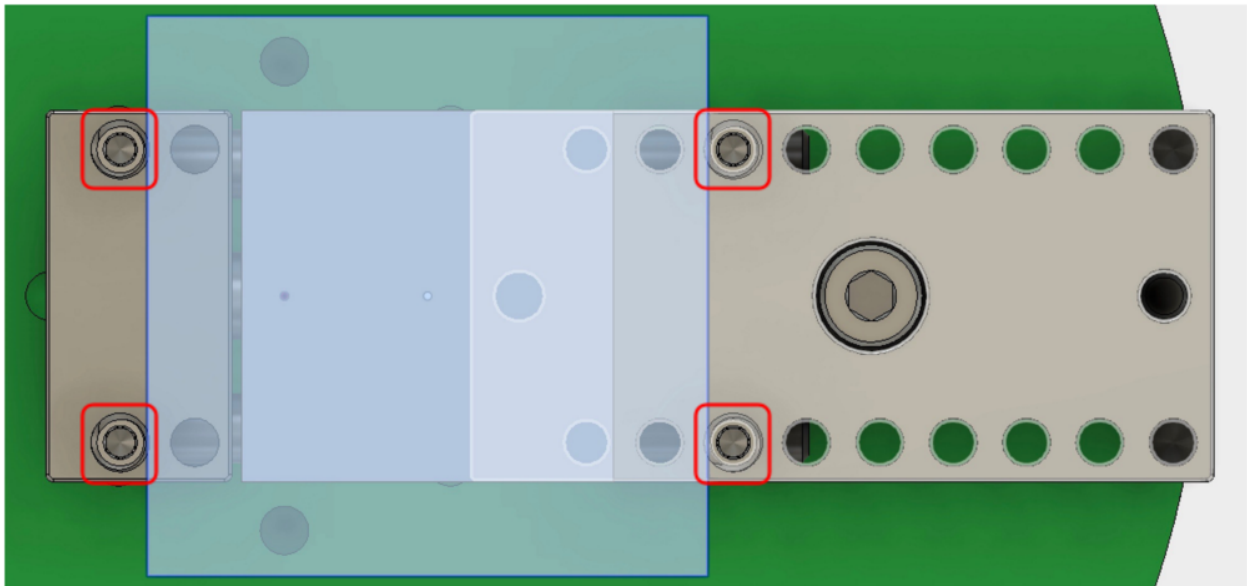
To move the an object within Fusion 360, right click the part in the design tree and select move.

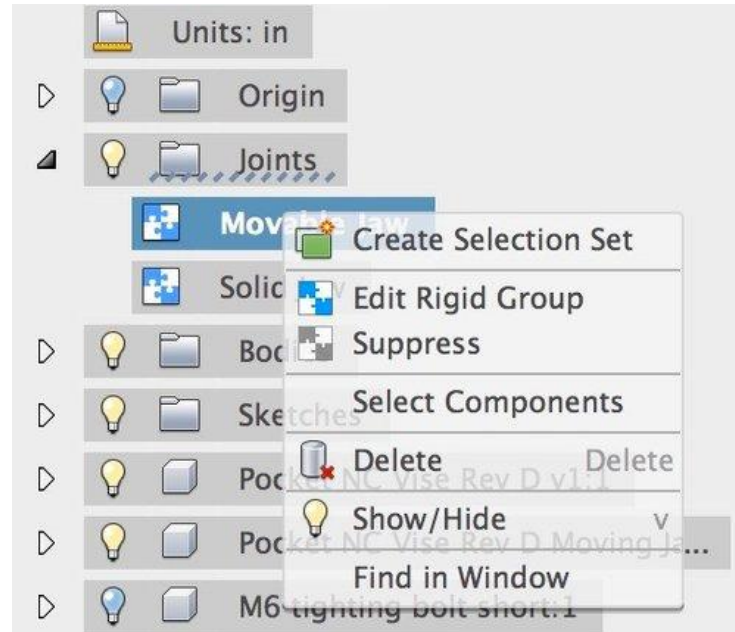


All 4 gripper setscrews on the vise will need to be moved. Start by moving the gripper setscrews on the **long side of the vise** by selecting both components in the design tree and selecting move. The hole spacing for the grippers is 5mm, so the they will have to move a total of 20mm to match the picture below.



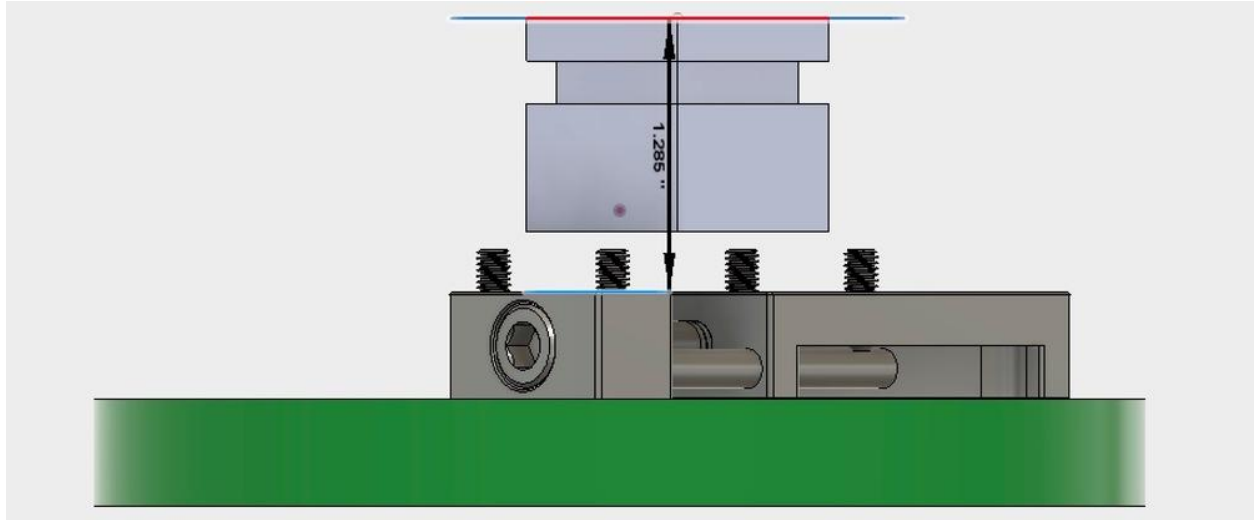
Next, locate the Test CUBE so the transparent square overlaps the gripper setscrews on the right by about 0.01". Note, when material is placed in the vise between the grippers, some of the stock material will be crushed. By overlapping material with the grippers, users will get a better representation of what is actually happening on the machine.





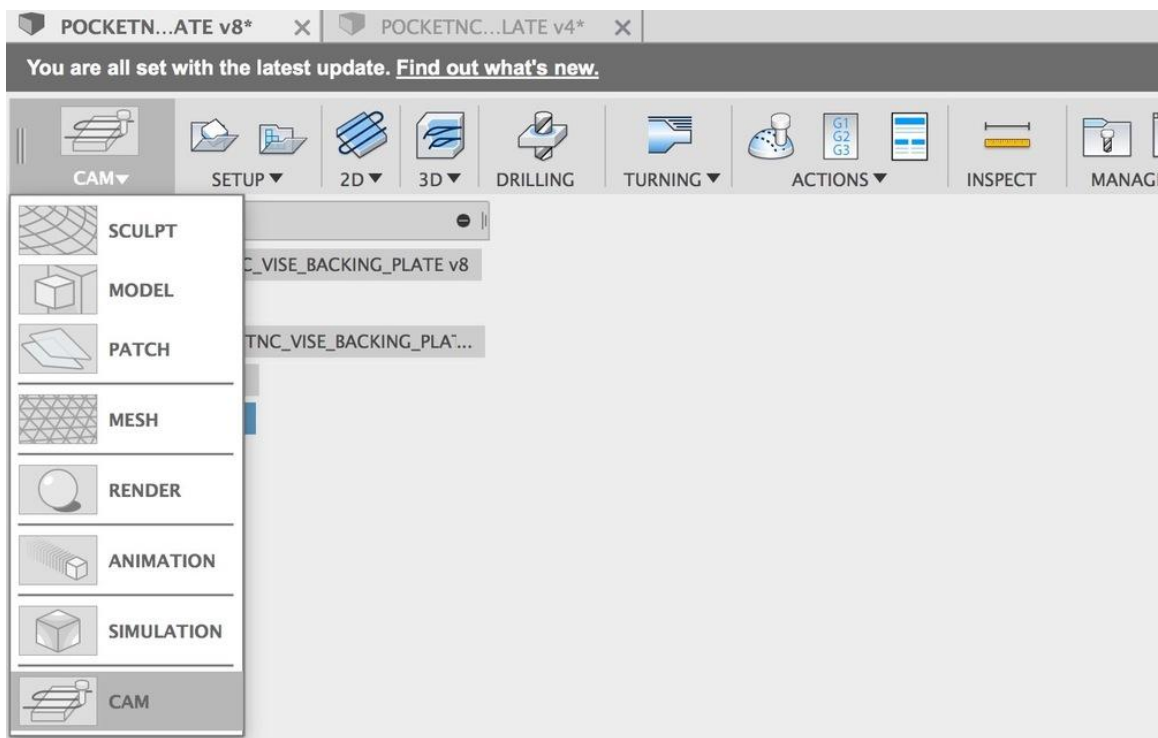
The grippers on the left will also need to be moved. They are mounted to the movable jaw of the vise, this means they can move in any increment. To move the left grippers, first select all components in this assembly by opening the joints dropdown within the design tree, right click “Movable Jaw” and left click “Select Components”. Move the components so that the set screws on the left have a similar overlap with the stock material to the set screws on the right.

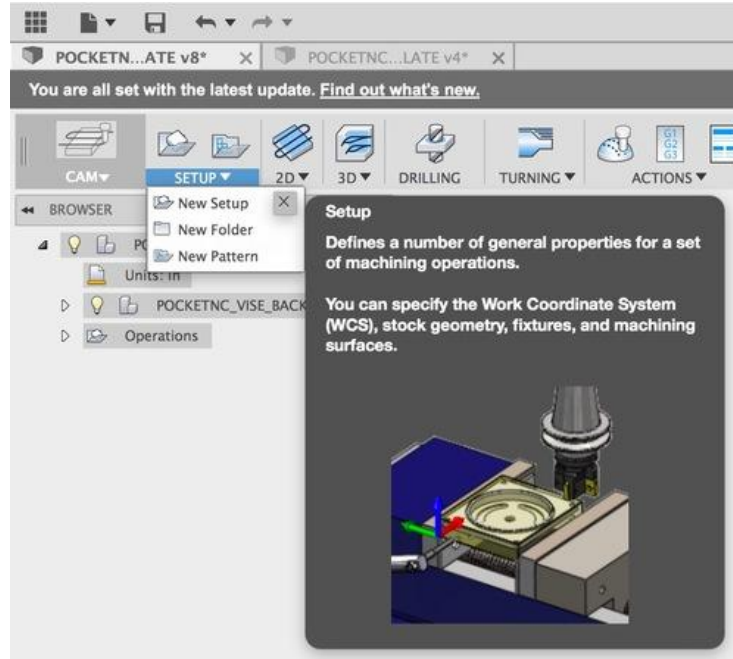
The Assembly is almost complete, the last step to locating the Test CUBE, is setting the height from the vise. The Test CUBE will be made from a 1.5” x 1.5” x 1.5” stock material, this means the top of the Test CUBE must be 1.5” from the vise or less. The bottom of the Test CUBE must also be above the grippers to avoid collisions when machining. To move the Cube in Z select the part within the design and select move. As you can see in the picture, the top of the cube is 1.285” from the top of the vise.



## CAM SETUP

To begin making toolpaths for the Test Cube users must first migrate from the MODEL or SCULPT environment to the CAM environment. To do this select the “Change Workspace” drop down tab at the top of the screen, then select CAM.

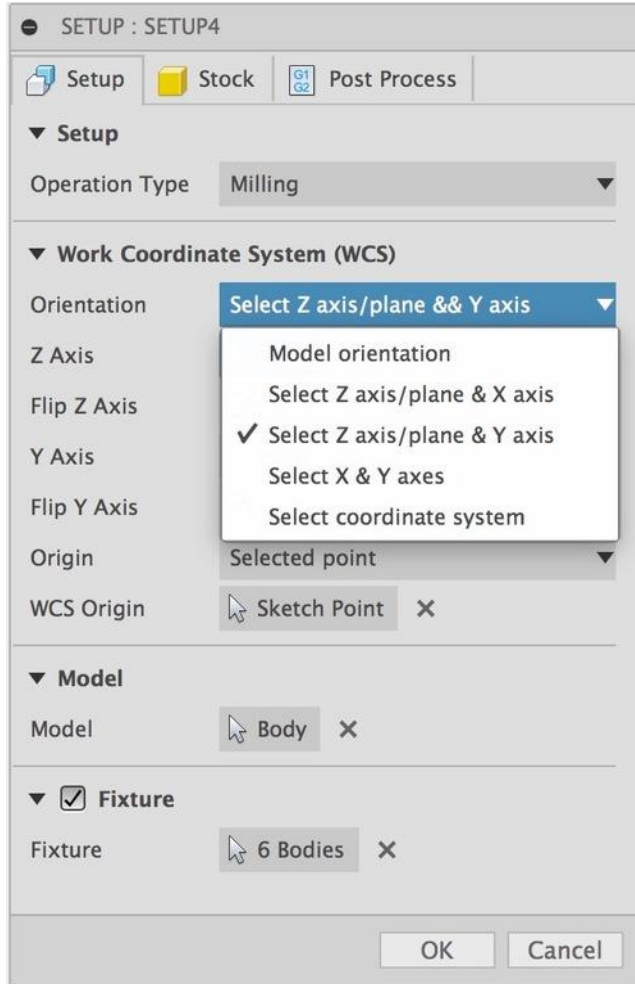




Before toolpaths can be created, a Setup must be created that includes origin location, stock material and a coordinate system. To create a Setup select the setup dropdown and “New Setup”.

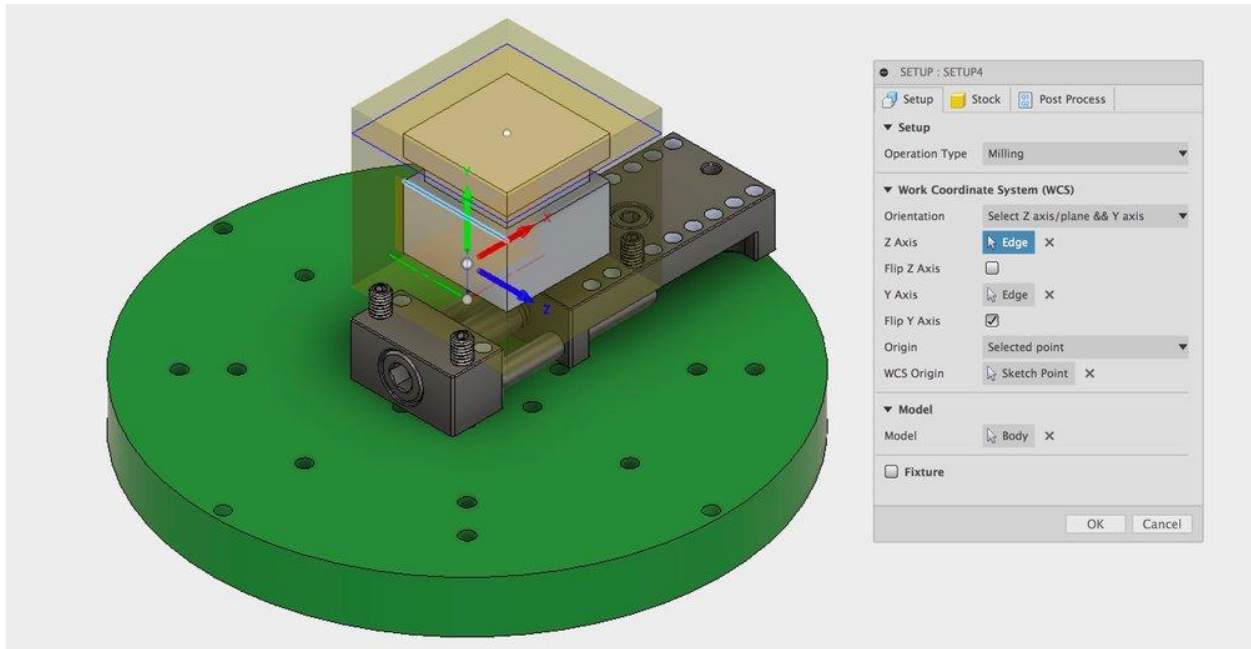
The Setup is a tool for Fusion 360 to understand excess material, objects to avoid and the machine's initial position. The first part of a setup is selecting the work origin and orienting the Work Coordinate System (WCS). To set up the WCS within the Fusion CAM environment, select the WCS dropdown and “Select Z axis/plane & Y axis” next to orientation.



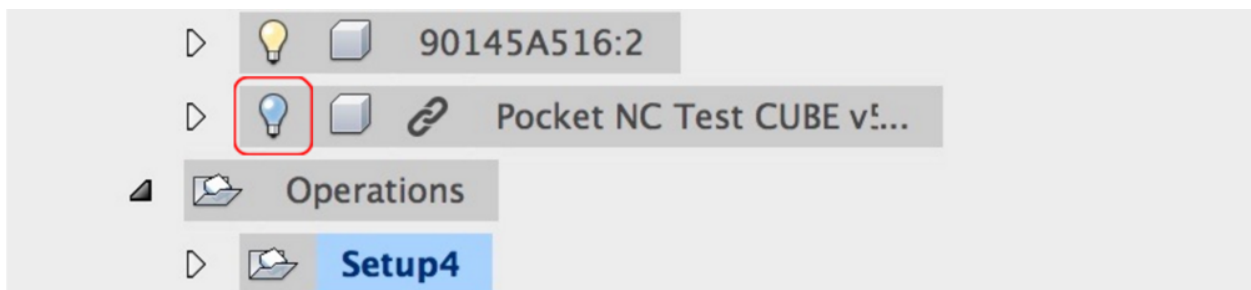


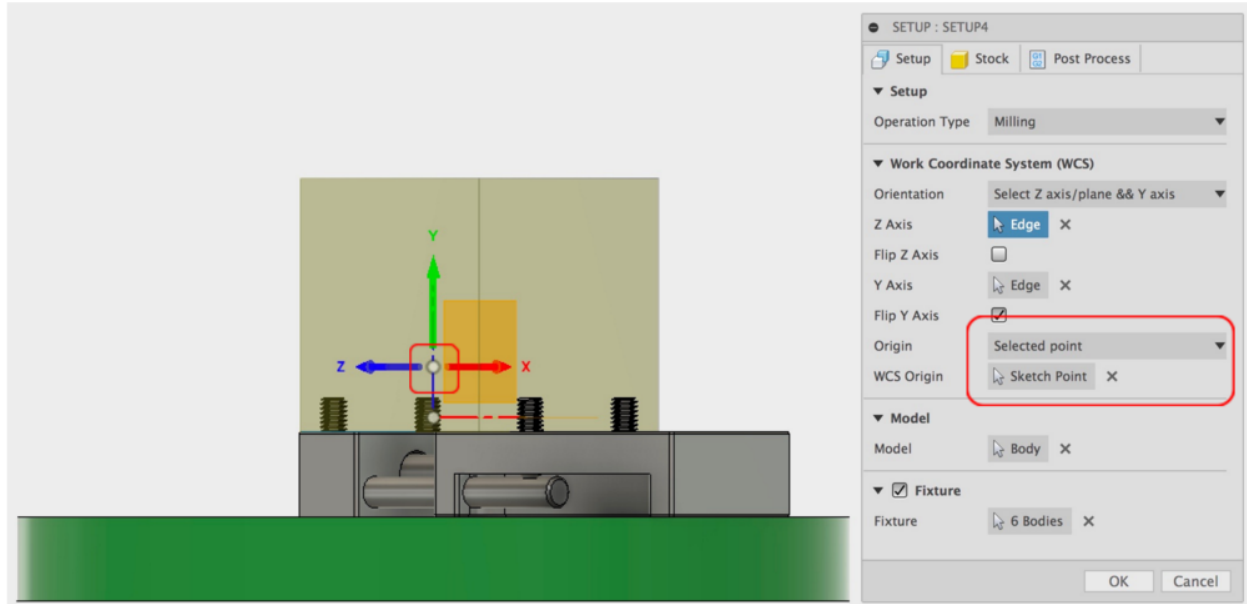
Two pieces of geometry will need to be selected to complete the “Orientation” field, one for the Z Axis and one for the Y Axis. To do this, select the box next to the “Z Axis” labeled “Nothing”, then select geometry (a straight line) on the vise perpendicular to the moveable axis of the jaw and parallel to the work table. Repeat the operation for the Y Axis, selecting a feature perpendicular to the work table. When finished users should have a WCS that looks similar to the image below. If you need to change the directions of the axes, use the check boxes to flip them.

The orientation of the B axis is flat or (perpendicular) to the spindle of the Pocket NC when homed. This needs to be reflected by the WCS. On the Pocket NC, the Y Axis moves the table up and down this must be reflected in the setup for the output code to work with the machine.



Next, you will select the origin of the machine within the model. The origin of the machine is represented by a point in space .885" off the surface of the table at the center of rotation. This point varies slightly from machine to machine and can be changed from the downloaded model to match a specific machine. Before it can be selected any geometry covering the origin must be temporarily hidden. To hide the cube, select the cube model within the device tree, then select the lightbulb to the left of the name.

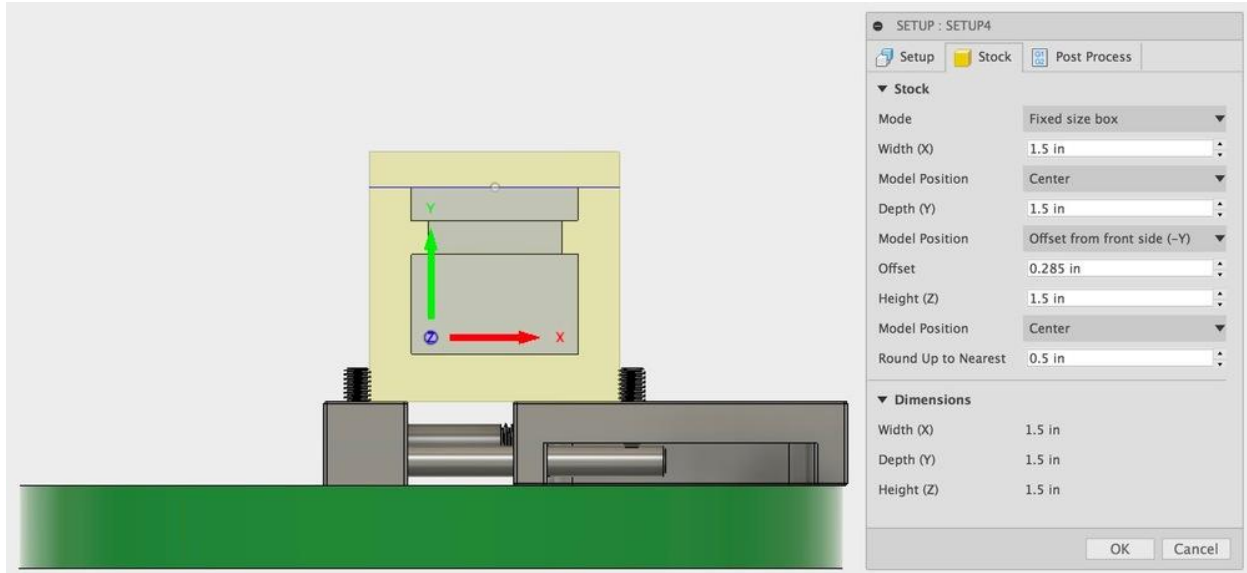




With the origin visible, select “Selected point” from the Origin drop down menu. Then select the box labeled “Nothing” to the right of WCS Origin. Now select the machine origin.

Make your CUBE visible again and if it wasn't already selected, select it as your model.

Last under the setup tab uncheck the fixture box.

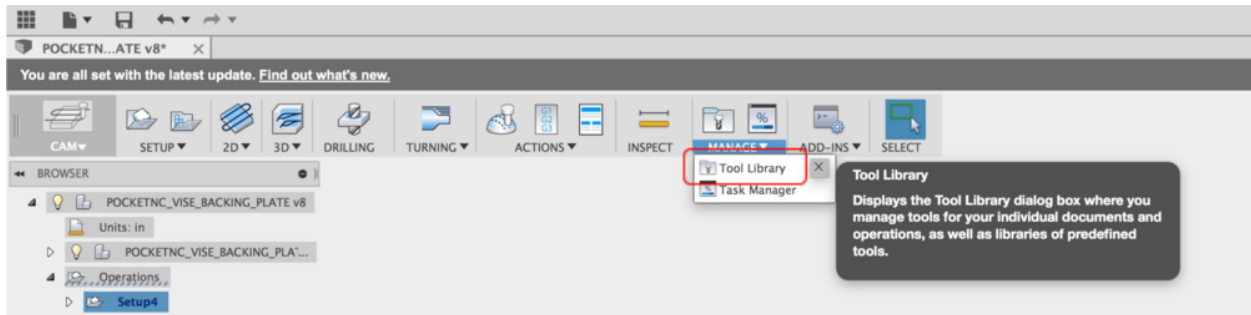


The test cube will be made from a 1.5” x 1.5” x 1.5” block of material so this should be input for “width” “Depth” and “Height”. Because the top of the cube was set 1.285” from the top of the vise the stock offset will need to be about .285” offset from (-Y). When finished with the setup, select ok to save and close the setup box.

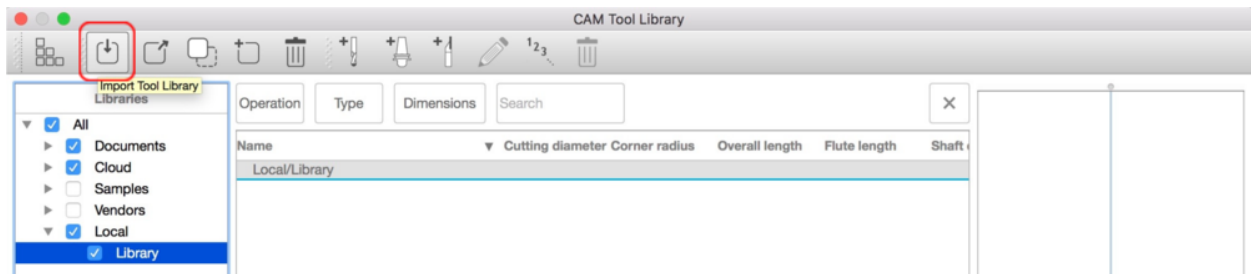
## TOOL PATHS

The Pocket NC uses a set of instructions known as a tool path written in a language called Gcode, to direct the cutting bit and other parts of the machine. This set of instructions is used in the world of machining to remove material rather than add it like 3D printing. This tutorial will help users create three types of tool paths; facing, roughing and finishing, to turn a 1.5” cube into a 1” cube with a few extra features. Roughing toolpaths are used to remove the bulk of excess material from a part, while finishing toolpaths only remove as much as a human hair, sometimes less. Facing tool paths can be used at the beginning or end of a program, sometimes even both, a facing toolpath removes material from the top of a part.

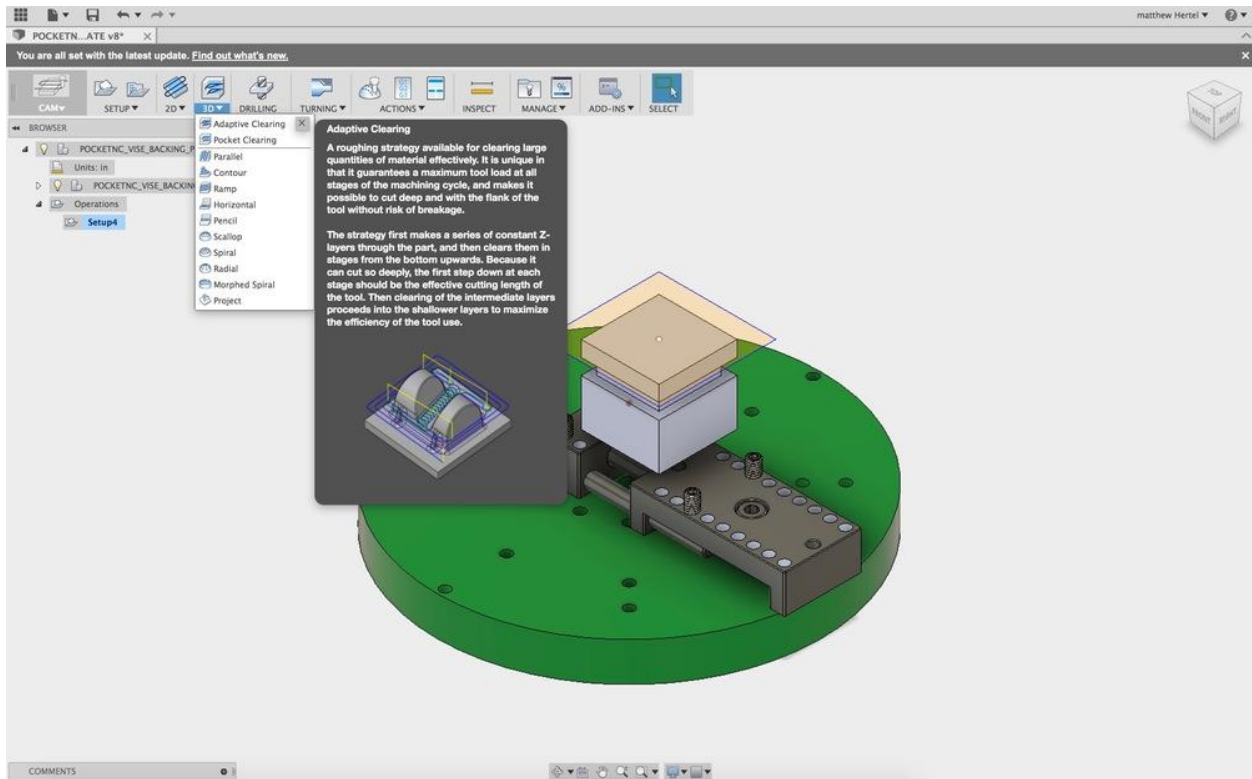
The block can be cut using the 1/8” tool packaged with the Pocket NC. The order of the toolpaths will be; rough top, rough 4 sides, finish top and finish 4 sides. Before users can generate toolpaths they will need to download the Pocket NC Tool library. To install the new tool library, select “MANAGE” then “Tool Library”



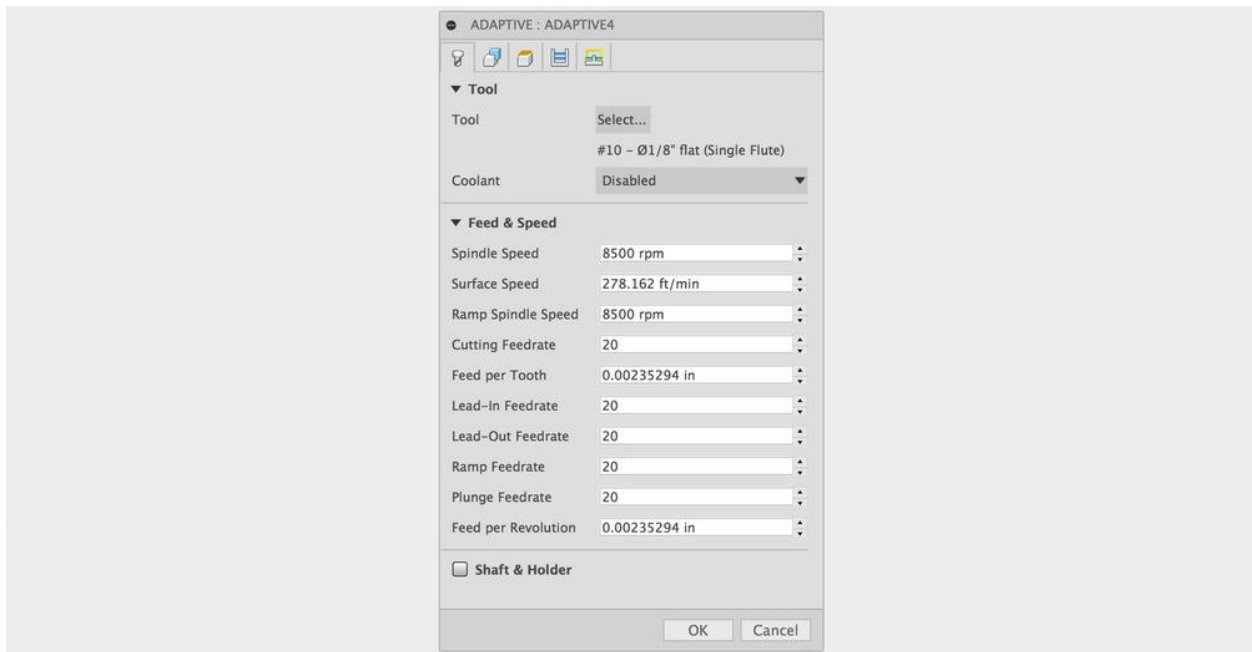
Next, under “Libraries” select “Local” then “Library”. Now use the “Import” tab to select the Pocket NC Tool Library from your downloads location.



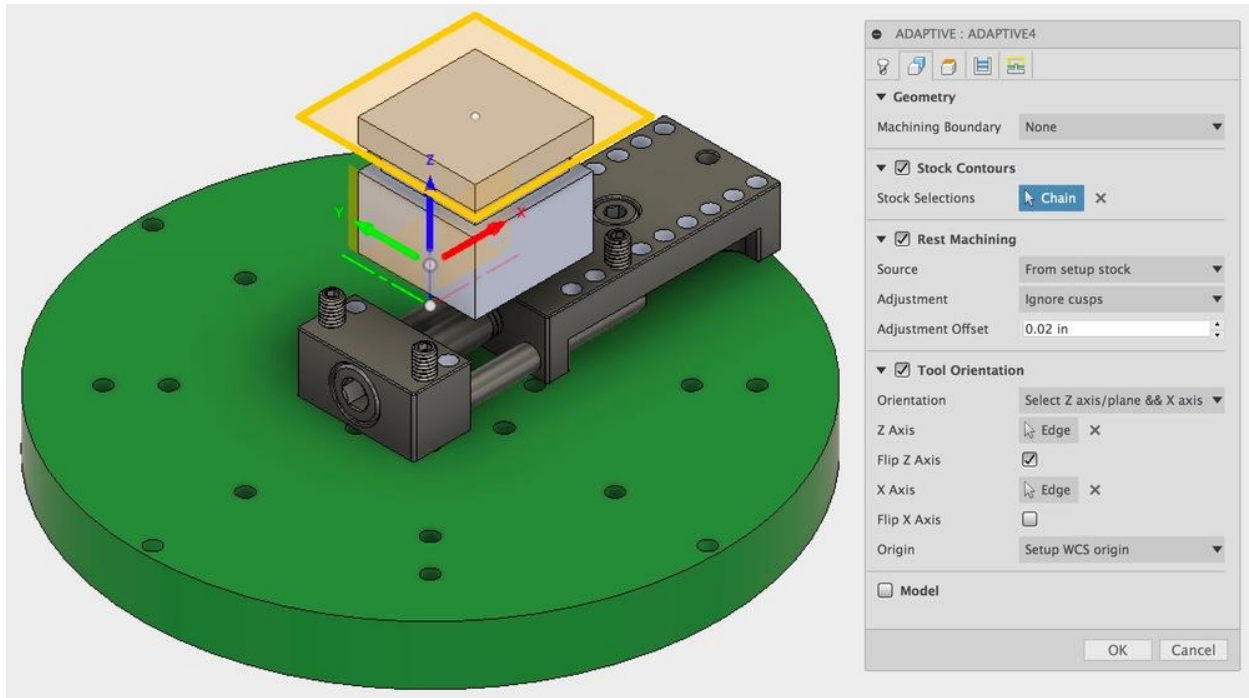
To create the first toolpath select 3D, then “3D Adaptive Clearing”.



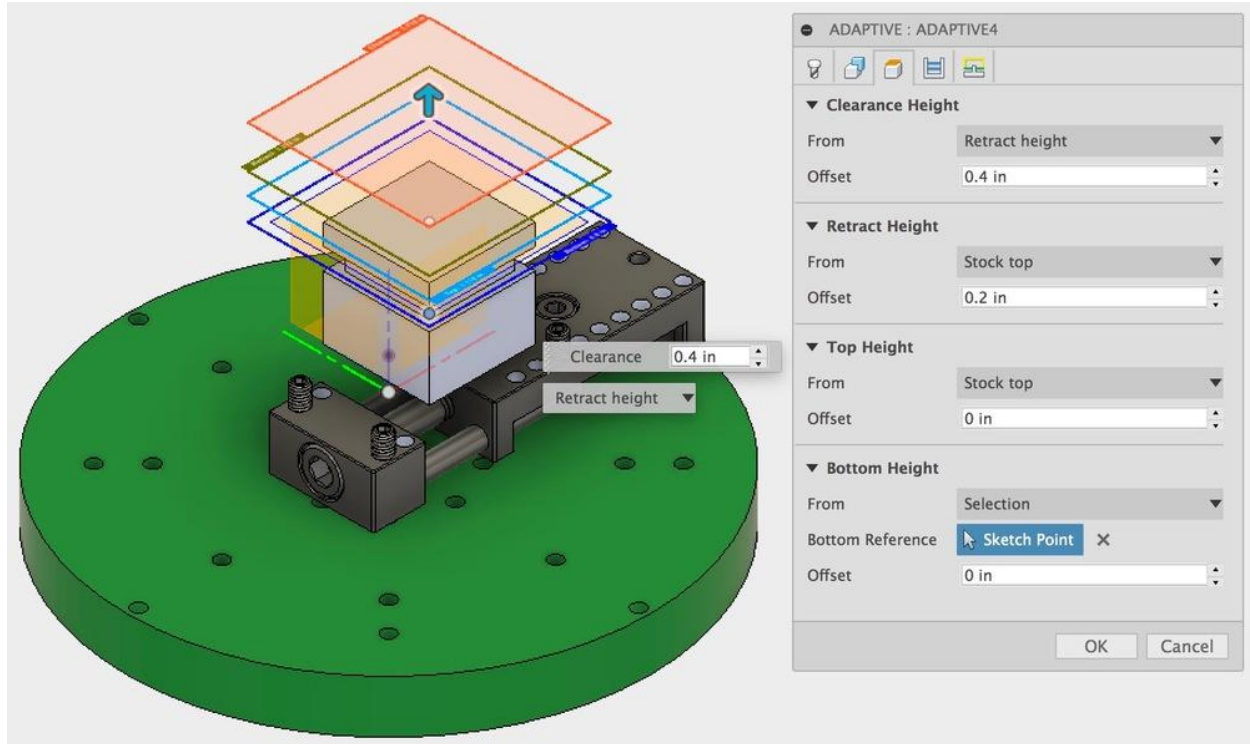
First, select a 1/8" single flute cutting tool from the imported Pocket NC Tool list. Use the settings in the picture below to get the correct feeds and speeds for making the cube in pine.



In the geometry tab, set up the tool orientation so that the coordinate system matches that of the image below. Next select the top outline of the 1.5 inch cube for the stock contour. This geometry will let Fusion know what it needs to machine.

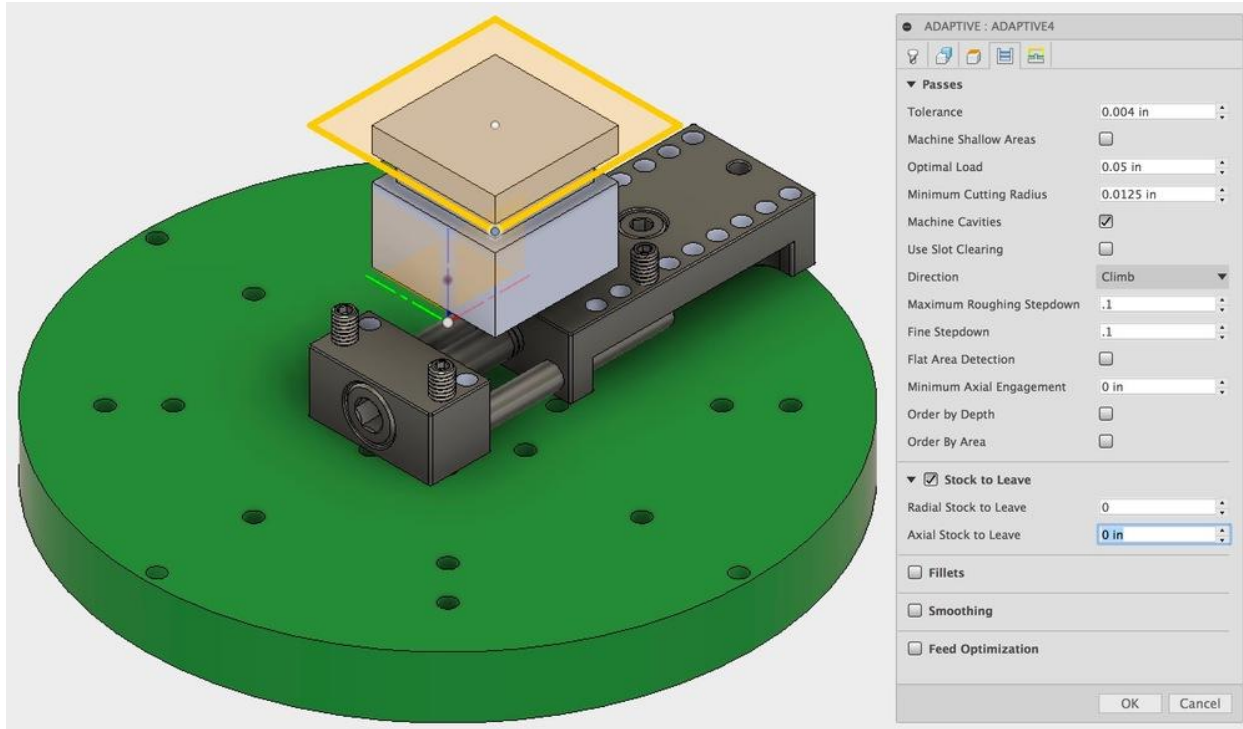


The “Heights” tab is where users will select the depths for machining. For the first operation, select the cube top as the “bottom height”. This will be the maximum depth the tool will cut at. “Top Height” should be the cubes material top, this will not need adjusting if the stock is setup properly.



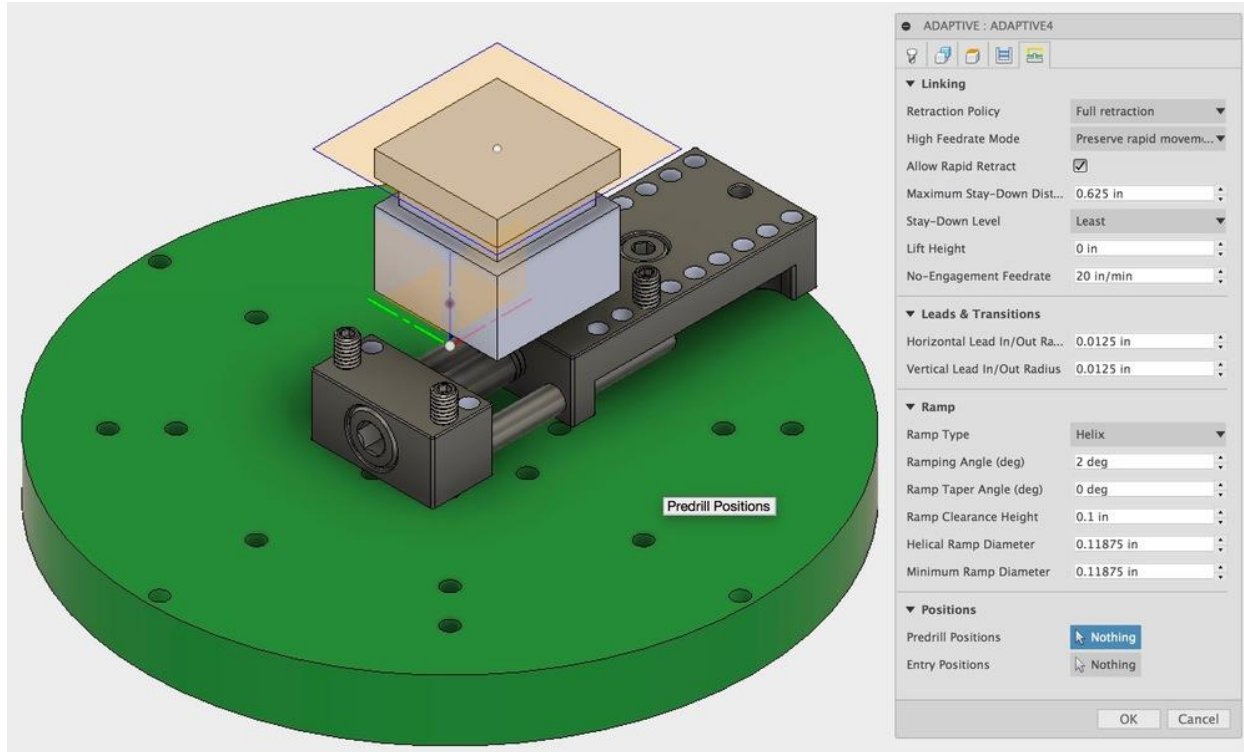
The “Passes” tab controls how much material is removed at any given time. Three values need to be adjusted to get a proper cut; "Optimal Load", "Maximum Roughing Stepdown" and "Stock to Leave". Adjust them to the settings below or fine tune them as you see fit.



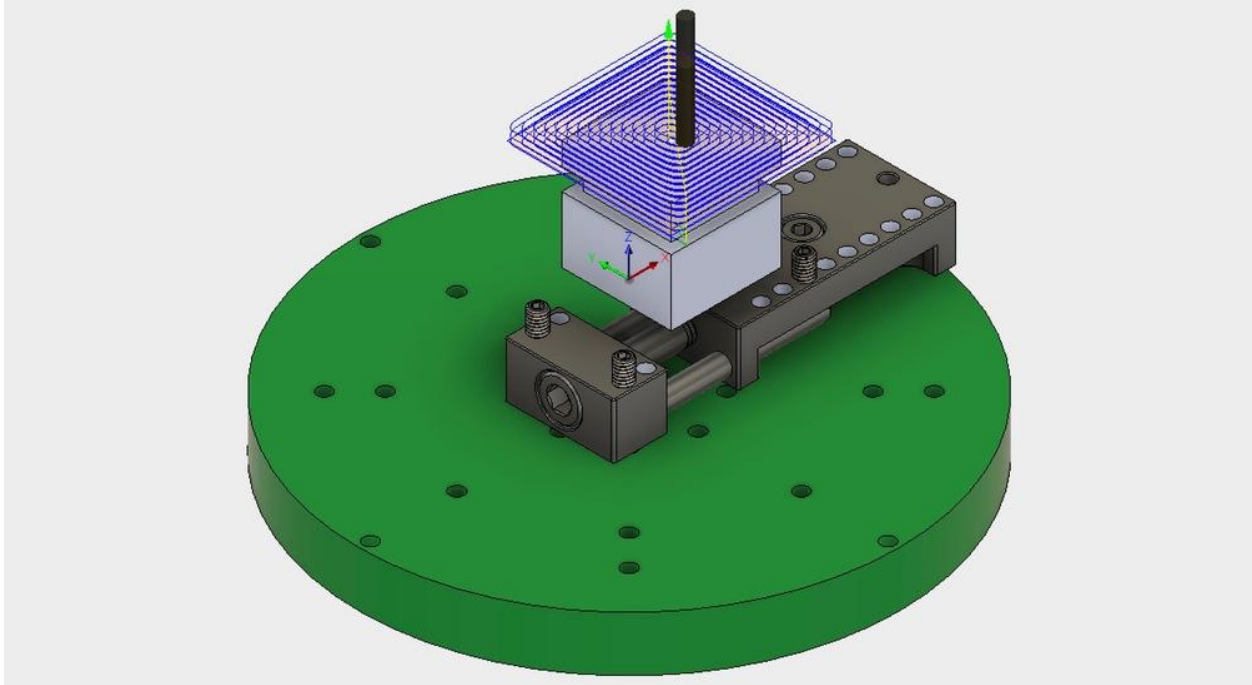


The last tab, "Linking" is mostly for fine tuning of a tool path. For example, "No Engagement Feedrate" can be altered from 20 in/min to 40 in/min to save time when the tool is in a retract and not cutting. A "Ramping Angle" of 2 deg is good for Aluminum because the cube is made from pine, A ramping angle of 10-15 deg may be more appropriate.

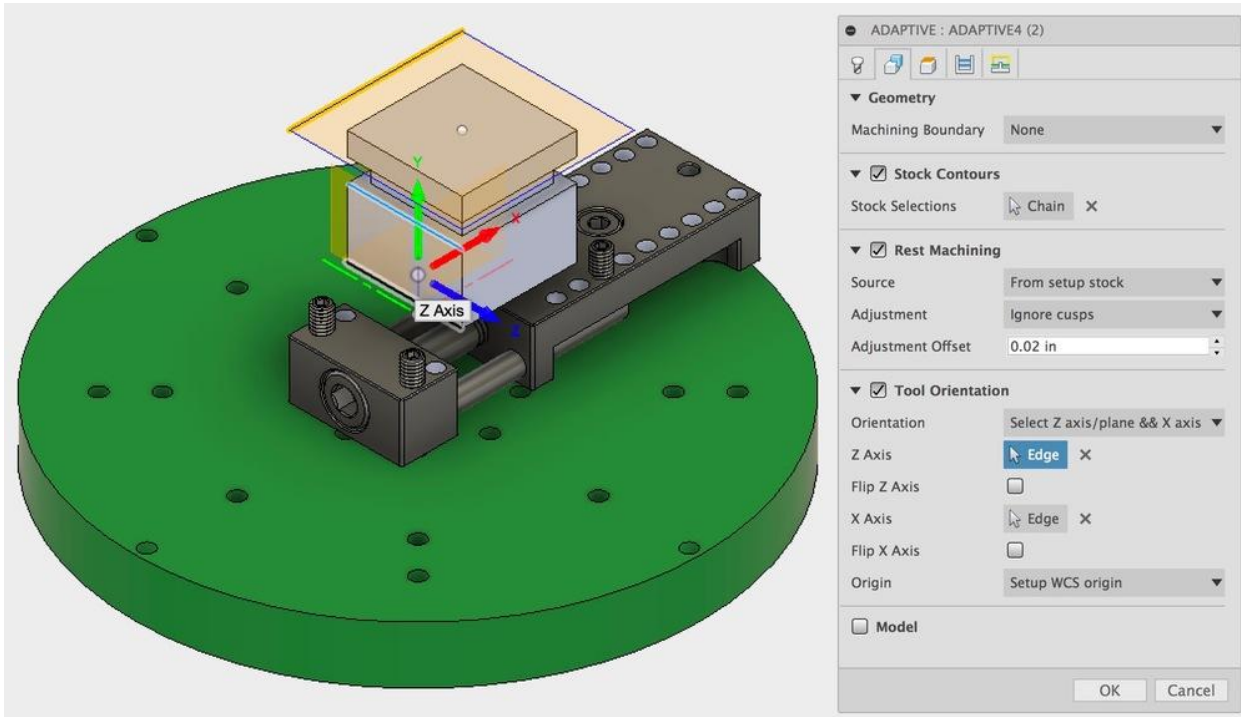
When finished making changes select "OK" to save changes.



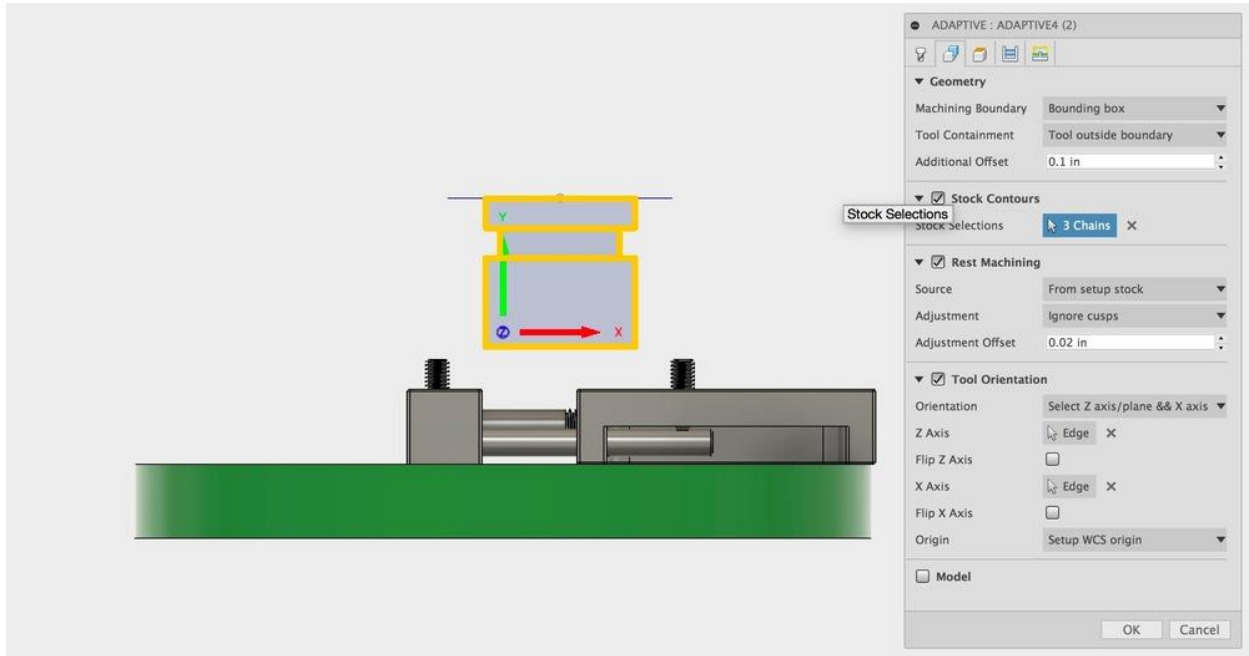
The tool path should look something like the image below removing all material within the boundary. This toolpath can be processed into gcode and sent to the machine.



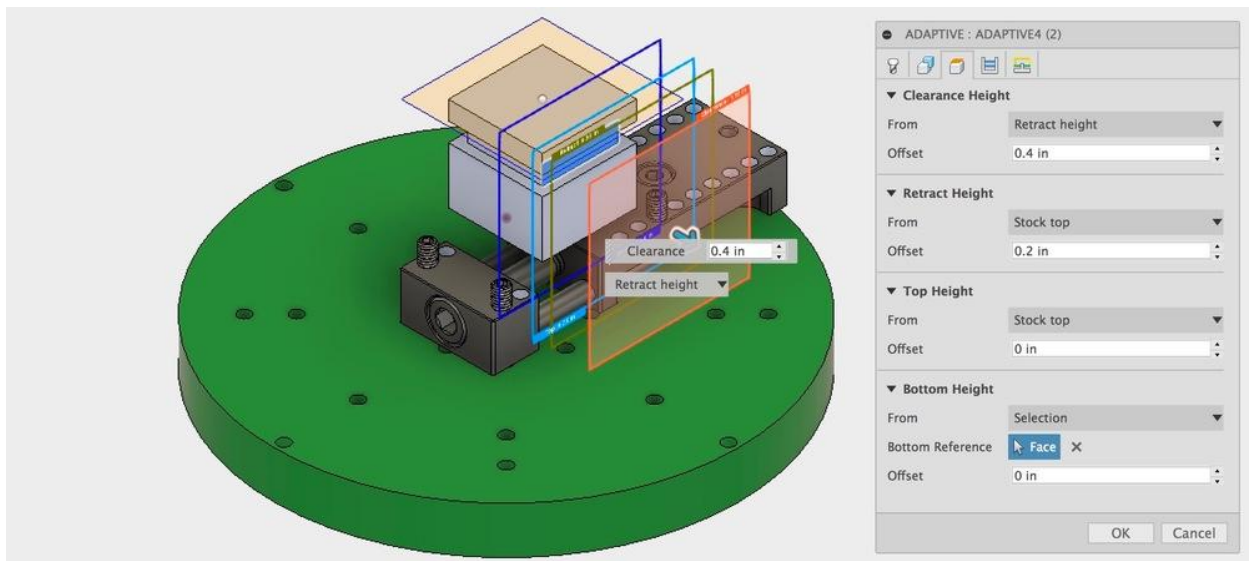
To cut the remaining sides of the cube, repeat the process by creating a new 3D adaptive toolpath and changing the orientation of XY&Z for each tool path.



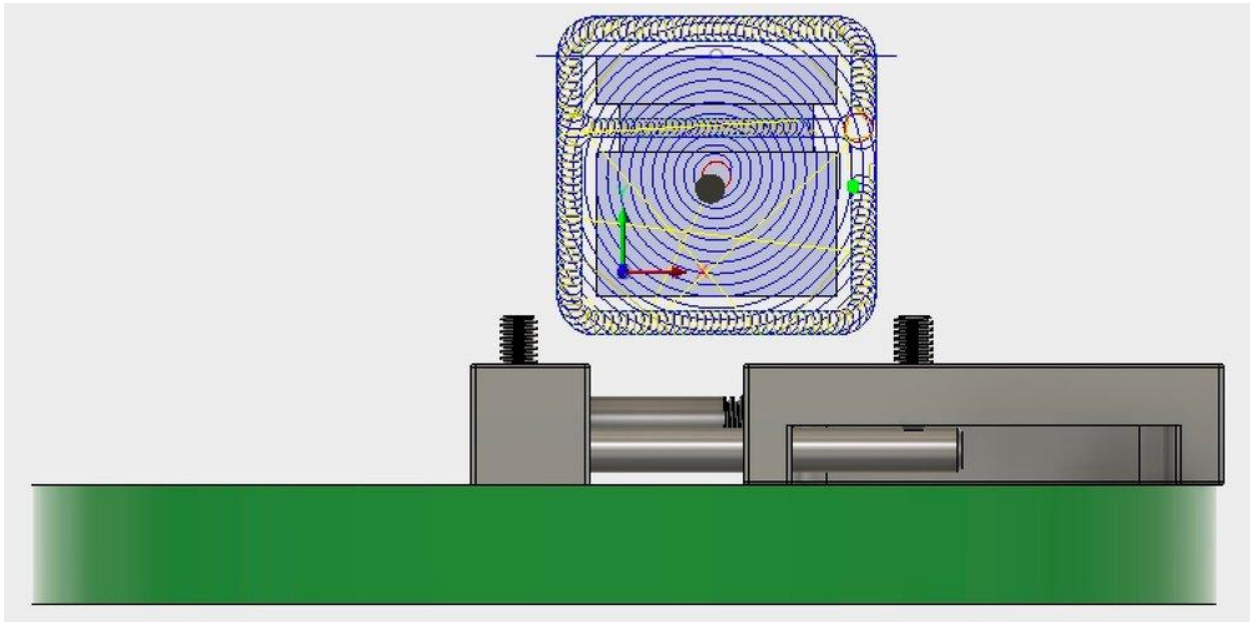
Set "Machining Boundary" to Bounding box and "Tool Containment" to Tool outside boundary with an offset of ~.1" (be sure to check tool paths for visual collisions before running). From the side, select all three surfaces for the "Stock Contour field".



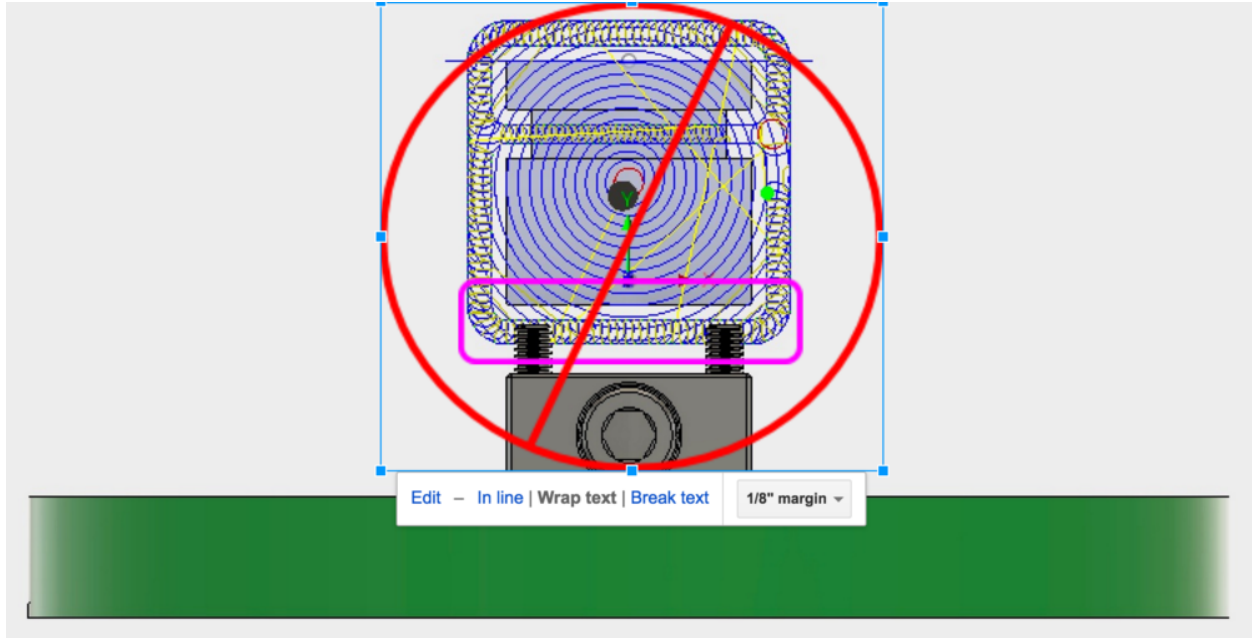
Set the Bottom height on the side to the lowest desired machinable height.



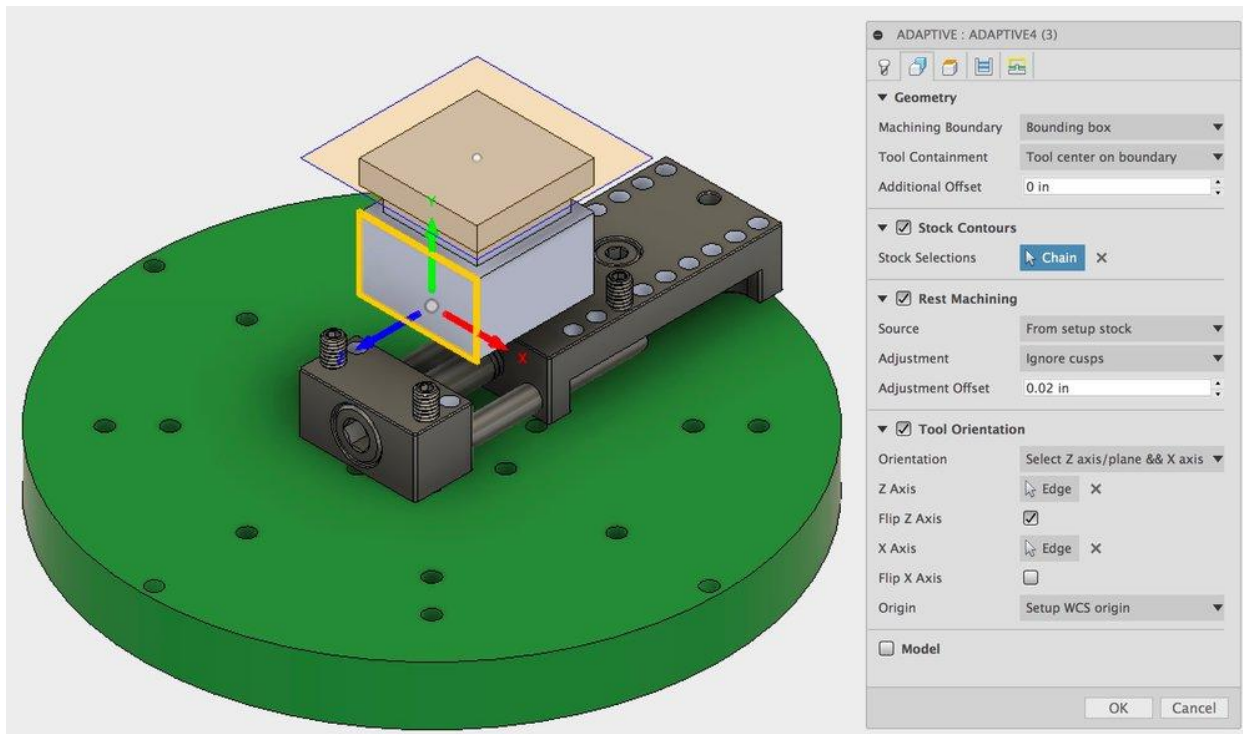
The toolpath should look like the image below. Before Posting a tool path, users should check for collisions with the Pocket NC vise using the Simulate feature within the ACTIONS dropdown.



In the image below, the toolpath intersects the gripper jaws inside the pink box. This will most likely break a tool. Be sure to check for such collisions.

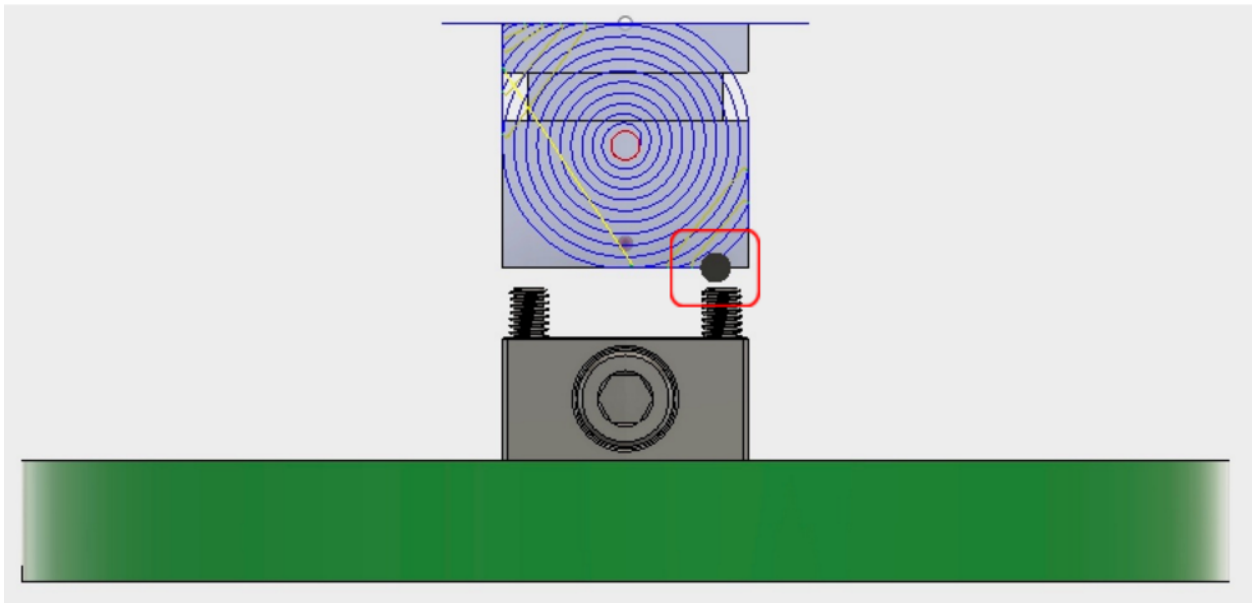


By switching the Tool Containment to "Tool center on boundary" from "Tool outside boundary" the user can limit how far off the edge the tool will travel. The next image show the results of this change.

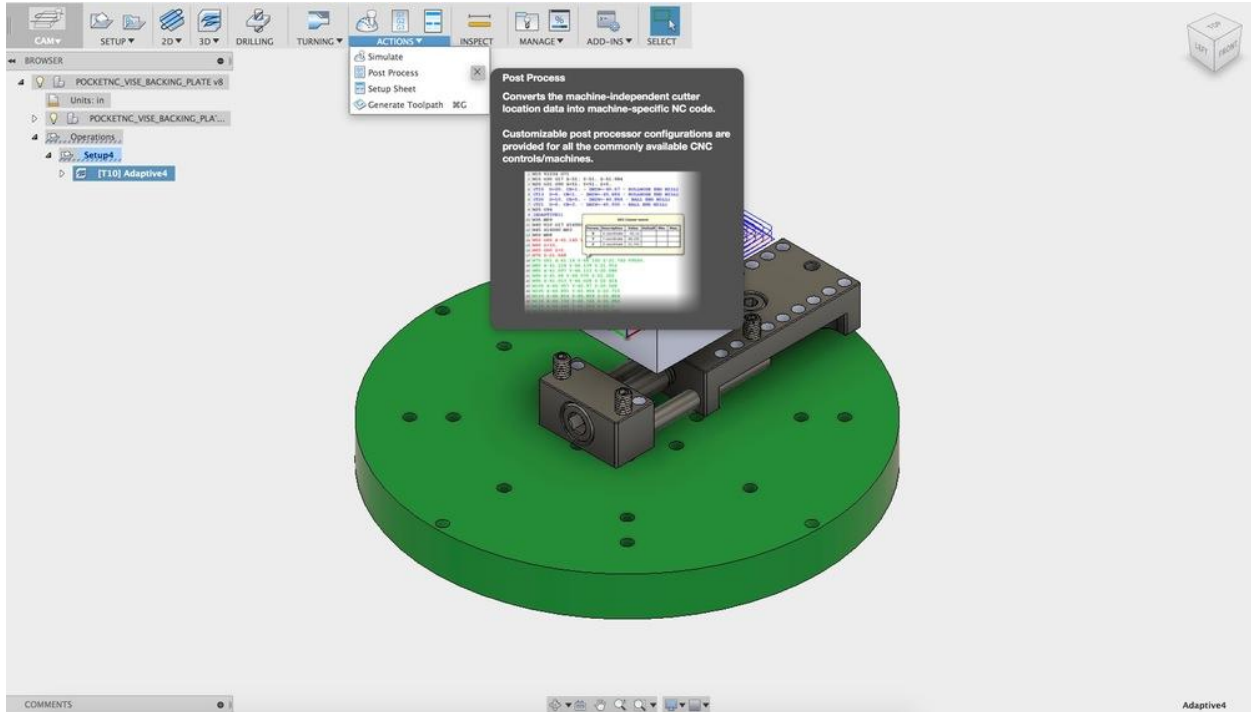




In the picture bellow, the tool does not travel off the material more than half its diameter resulting in a toolpath that does not hit the grippers. It is always good practice to keep excess room in a area of low clearance for a first run.



To create Gcode from the adaptive toolpath, select the "ACTIONS" dropdown then select "Post Process".



Select the pocket nc.cps post from the Post Processor list, then select ok. Fusion will prompt for a file location and name. Save the file to a flash drive with any name. Be sure to end the file with ".ncg" this is the file extension that the Pocket NC mill uses.

