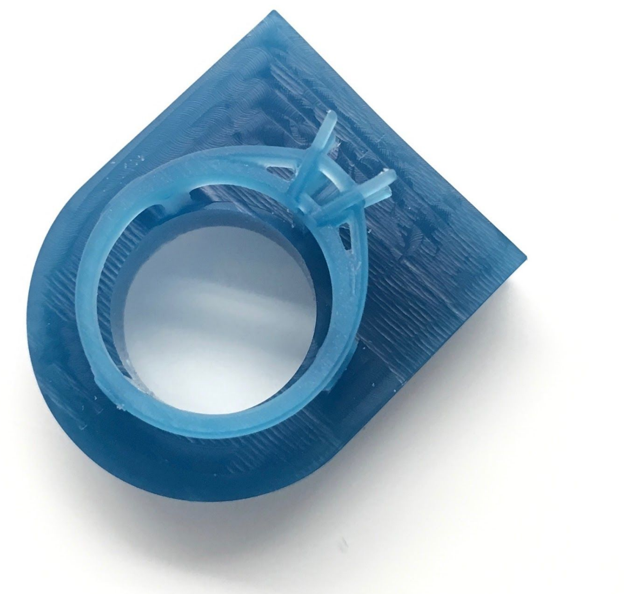


Wax Ring Tutorial

Change Log

About

The purpose of this tutorial is to familiarize Pocket NC users with the process of machining intricate wax parts such as a wax ring. This tutorial will cover importing mesh files into the Fusion 360 modeling environment, reducing the mesh count of the mesh files, converting mesh files to solid geometry, importing solid models into the Fusion 360 CAM environment, programing toolpaths, and outputting machine code(G-code).

Notes

This document is for reference only! Readers and operators take on all liability in the use of this document.

All measurements are in inches.

This document assumes the user knows basic functions of the machine. These functions include setting up the Pocket NC vise, transferring files to and from the machine, loading/running programs, setting up tooling and measuring tool length offsets.

All files mentioned in this document can be found within the zip folder containing this document.

The CNC program included with this document, “Wax Ring.ngc”, is to be used at the risk of the operator or machine owner. Pocket NC does not take responsibility for any damage or injury caused by the use of this or any other shared program.

For proper function of the “Wax Ring.ngc” program, users will need required tooling set up with offsets under the correct tool numbers. See tooling section. Users can also use this tutorial to write programs that incorporates different tooling.

Note that each machine has a slightly different center of rotation and as a result the Fusion part must have its center of rotation changed before using the program. The documentation shipped with each machine includes the offset. If the machine was shipped prior to 2017 see the “B-axis offset Tutorial” for how to check the offset on your machine. Pocket NC changed the alignment process at this point in time which makes checking your value important.

1 Getting Started

The following section will start at the beginning, importing .stl files. To start with posting tool paths, look at Section 2 of this document.

Machining a part from an .stl file requires several steps. First, the file must be converted to a format usable by Fusion 360 CAM. Once the file has been converted users can create a machining setup within the CAM environment that matches that of the real world. Once the setup has been created, tool paths can be created within Fusion 360 to simulate the removal of material. Finally, users can export the file to be used with the Pocket NC mill.

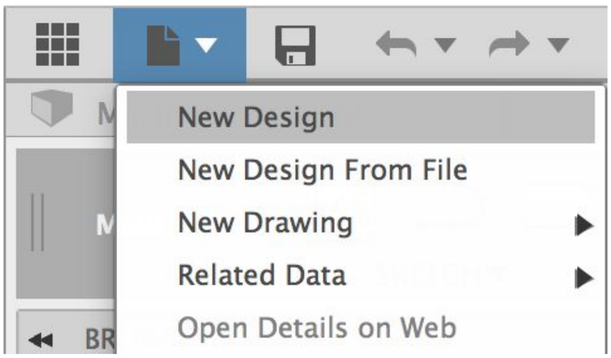
Fusion 360 requires solid files rather than mesh files for its CAM environment. In many cases, mesh files that have been converted to a solid, must also be reduced in mesh count, as a large number of surfaces will cause slower function of the CAM environment.

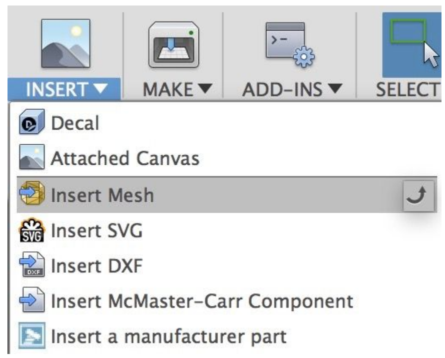
The reduction of mesh surfaces may be of concern as surfaces in a mesh format, quickly become rough. Users should note that a smoothing function exists within the CAM environment to round sharp edges at the intersection of faces. This function can be seen in Section 2 of this document.

1.1 Converting .stl Files

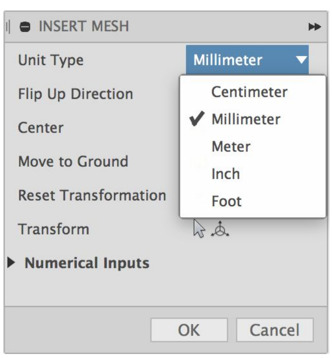
Start by downloading the “Sample Ring.stl” file from the Pocket NC website.

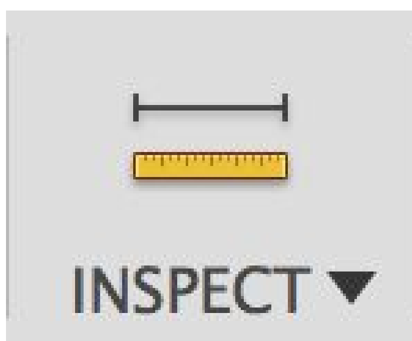
Create a new design, then import the file “Sample Ring.stl”. To do this, select “File,” then “New Design.” Once the new design has been opened, right click in the model area and select “Insert,” then “Insert Mesh.” Finally, migrate to the “Sample Ring.stl” file location, select the sample ring file and click “Open” .

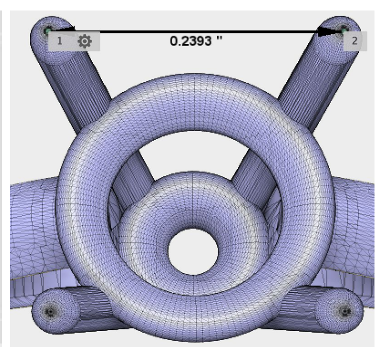




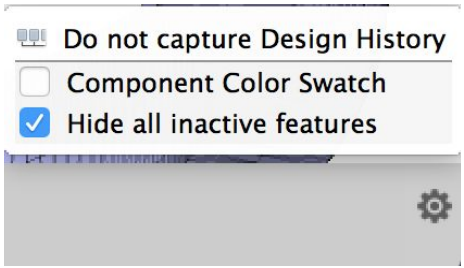
.stl files exist in two formats, English and Metric format. If the geometry is imported using the wrong format, the sizing will be much larger or smaller than expected. Import the Sample Ring using **Metric units**. Using the Inspect icon, users can measure the distance between prongs and check for a correct import.

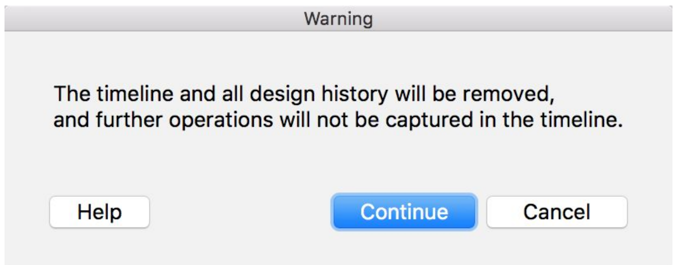




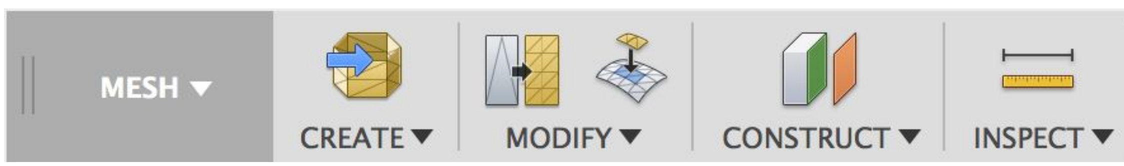


Before the Sample Ring can be used by Fusion 360 in the CAM environment, the file must be converted from a mesh format to a solid format. To convert the file users must first migrate to the Mesh workspace. To do this, open the settings icon located at the bottom right corner of the Fusion 360 screen, then select “Do not capture Design History”. Select “Continue” when prompted.

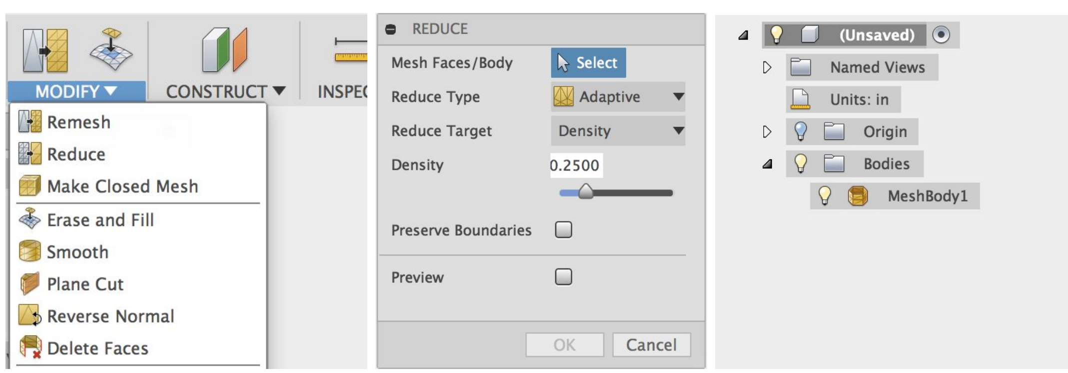




Next, migrate to the “Mesh” work space using the “change Workspaces” tab. Located to the upper left corner of the Fusion 360 screen.

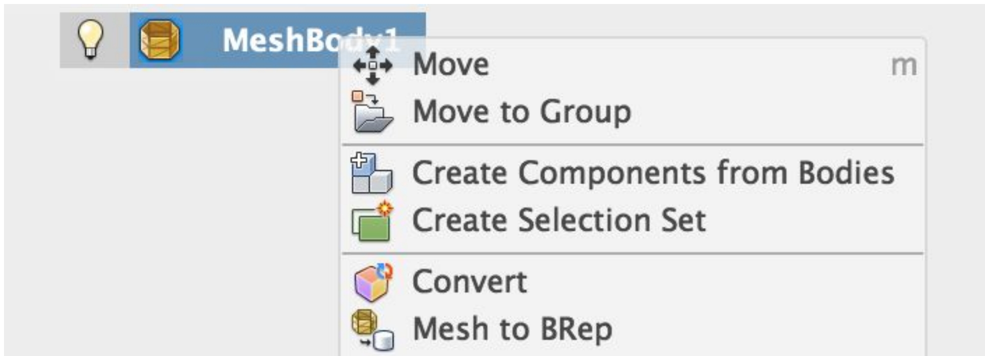


Once in the Mesh environment, reduce the Sample Ring mesh surfaces count by 75%. To do this, select “Modify,” then “Reduce.” Select “MeshBody1” from the browser as the “Mesh Face/Body” to be reduced. Last, select a density of 0.25 then select “ok.” Once complete, the .stl Sample ring will be reduced to a mesh with fewer surfaces. This mesh will retain its original dimensions but the facets composing it will be 4 time larger.

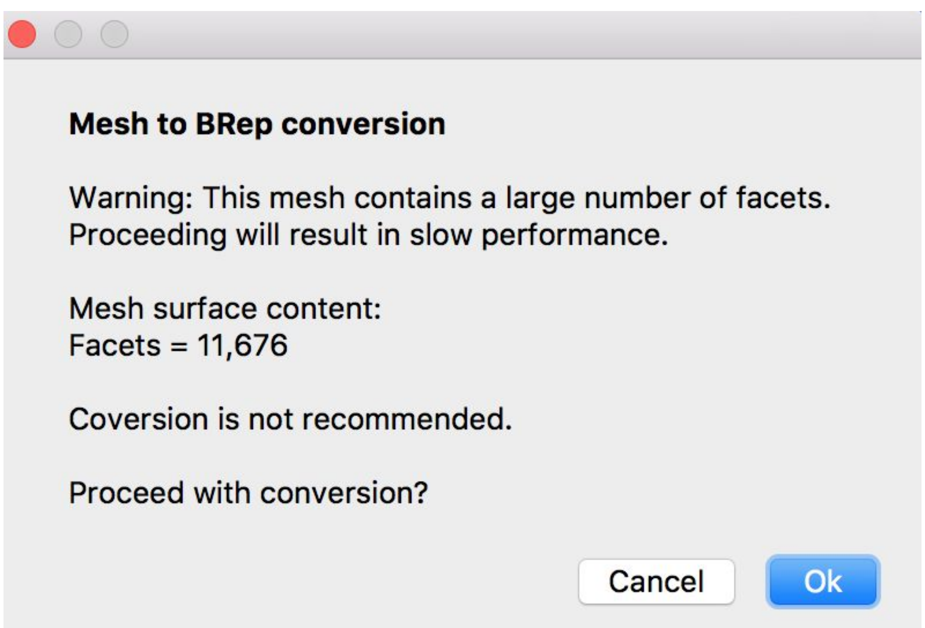


At this point, all work has been completed within the Mesh environment. Migrate back to the “Model” environment.

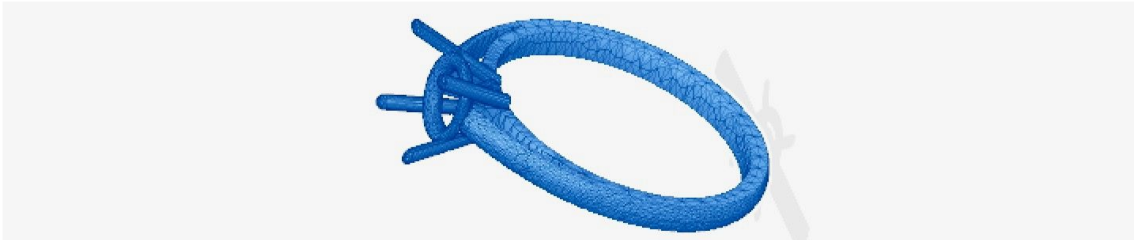
Once in the Model environment, users will need to convert the reduced .stl mesh into boundary surfaces for use in the CAM environment. To do this, select “MeshBody1” within the design tree, right click and select “Mesh to BRep”.



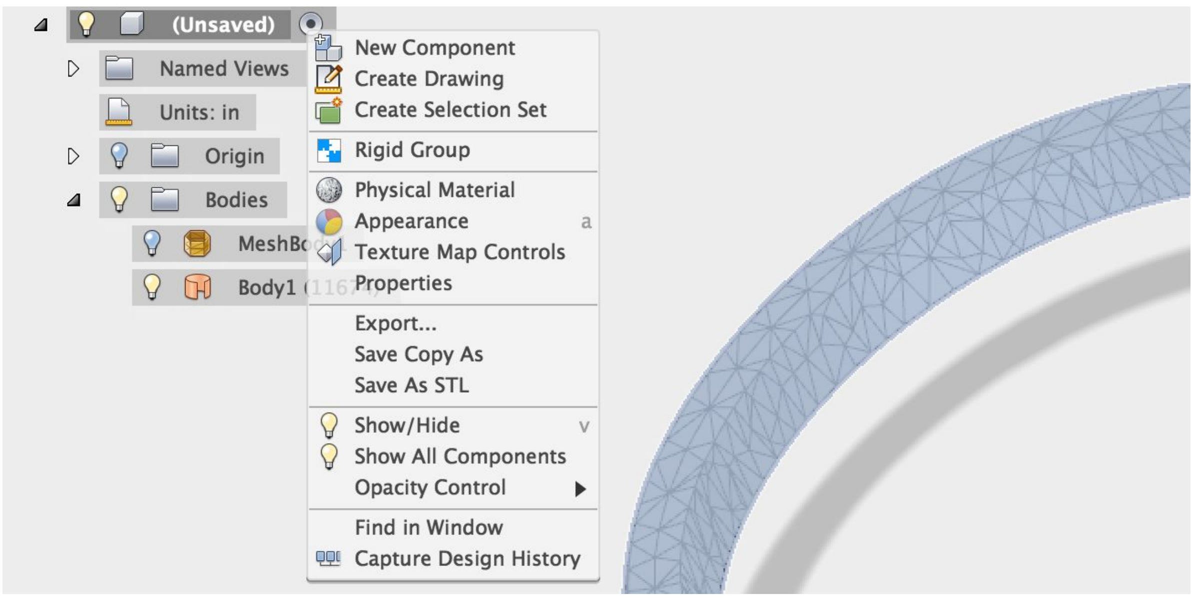
Fusion may produce a warning like the one below saying that the mesh count is high and that proceeding may lead to slow performance. Select “ok” to ignore the warning a proceed with the conversion. The Fusion conversion software has a limit of about 10,000 faces for converting a mesh to a BRep. In this case it is ok to proceed, but the performance may be impacted to some degree.



The converted sample ring should look like the image below. At this time, save the converted part, and turn back on the design history. It is now a solid and ready to be set up and fixtured for programming toolpaths.



To turn on design history right click on the Browser root and select "Capture Design History."



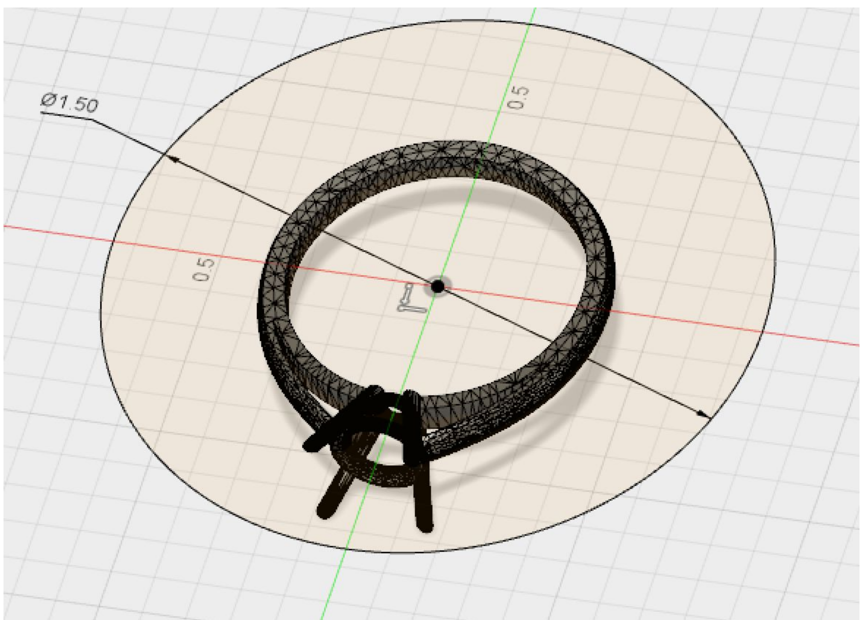
1.2 Part Setup and Fixturing

1.2.1 Create Stock and Supports

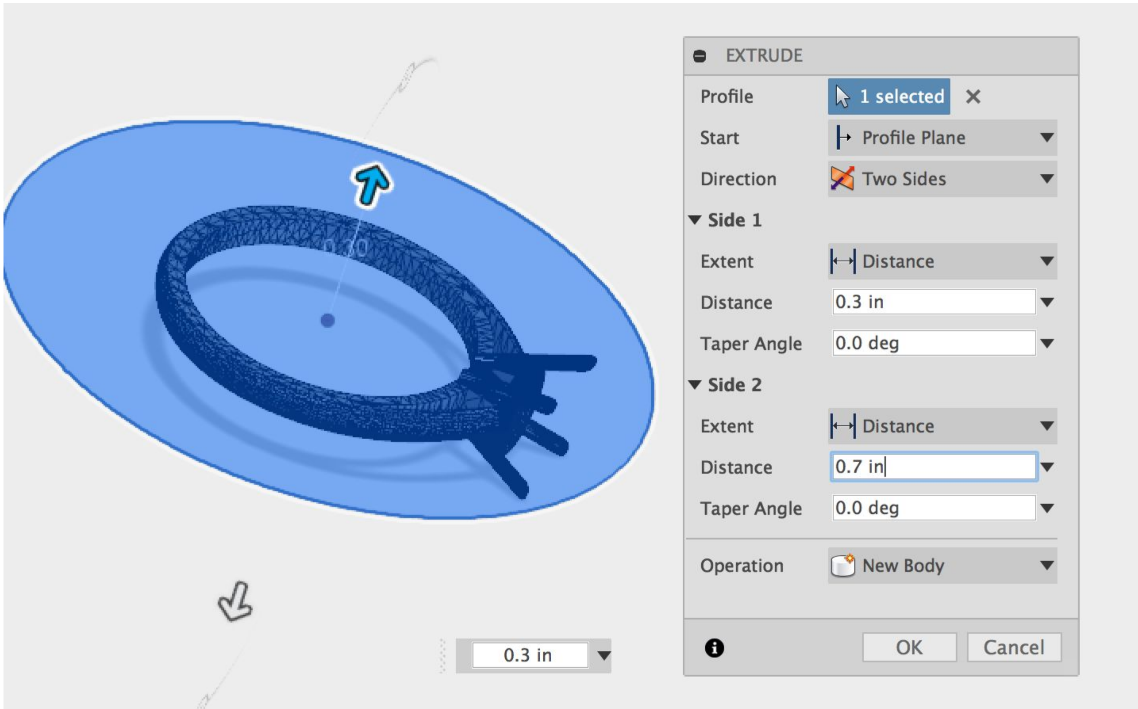
This section will focus on locating the sample ring within Fusion 360 CAM environment for use with the Pocket NC mill. Users should note that the setup used in this tutorial might not be the setup recommended by the jewelry industry. It is only an example, users are free to explore different setups and fixturing.

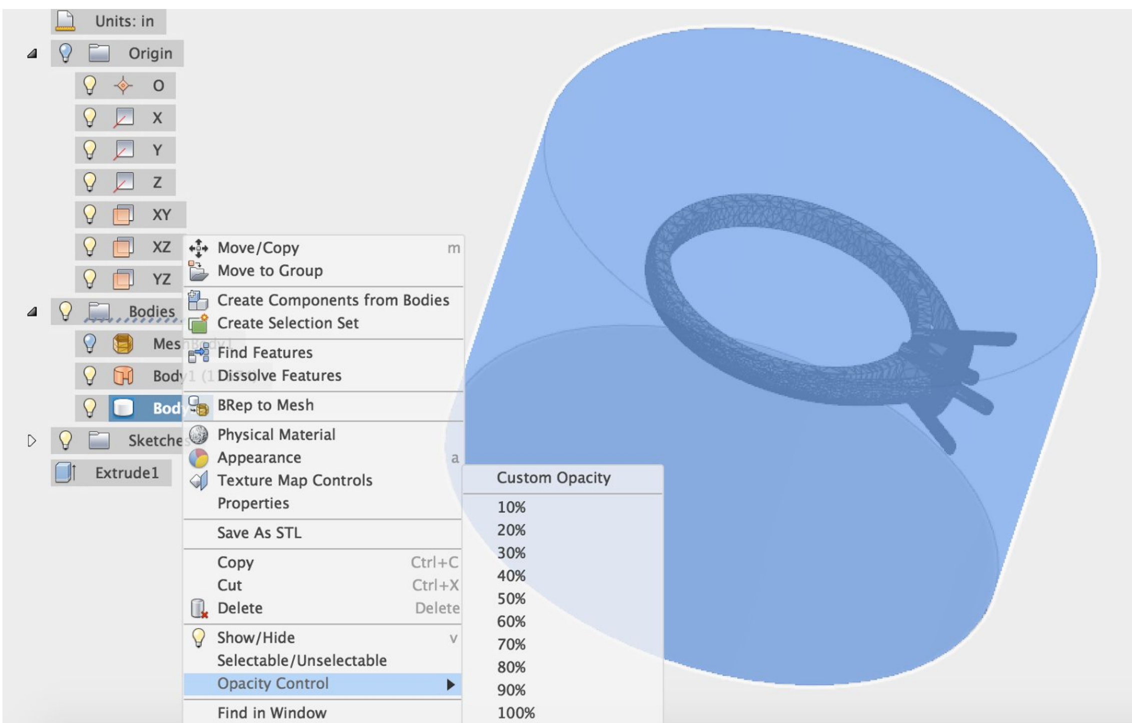
The first step in the setup is creating the geometry that defines the stock material. Secondly the supports that will be left connected to the work by the machining operations will be defined. Finally, reference geometry used to help define the machining steps will be created.

Start by creating the sketch used to define the stock material. The ring will be machined from a cylindrical piece of machinable wax. The diameter of the stock will be 1.5 inches and the height is 1 inch. In the modeling environment, create a new sketch. Select the xz plane, sketch a center-midpoint circle concentric with the ring. Dimension the diameter of the circle to be equal to the diameter of the stock from which the ring is being machined, here 1.5 inches.



Next, select the circle just created and extrude it both directions. Extrude one side 0.3 inches and the other side 0.7 inches. Change the opacity of the body created by the extrude to30% so that the ring is still visible inside of it. The goal of this step is to locate the ring within a block of stock. Because the vise has to grip the base of the material, the ring is located closer to the top of the stock than the bottom.

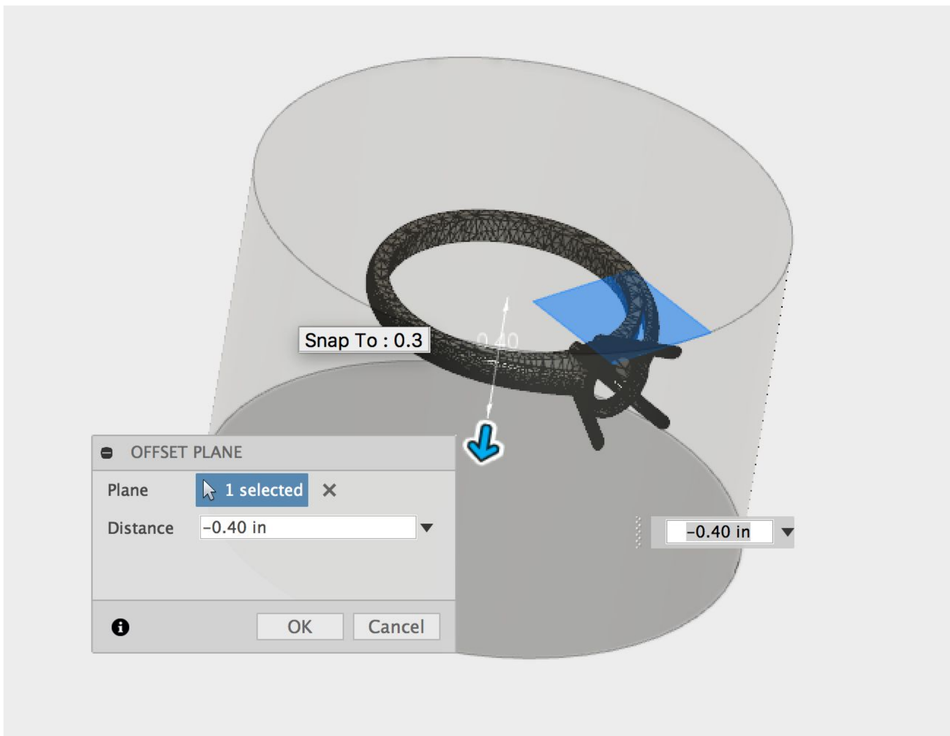




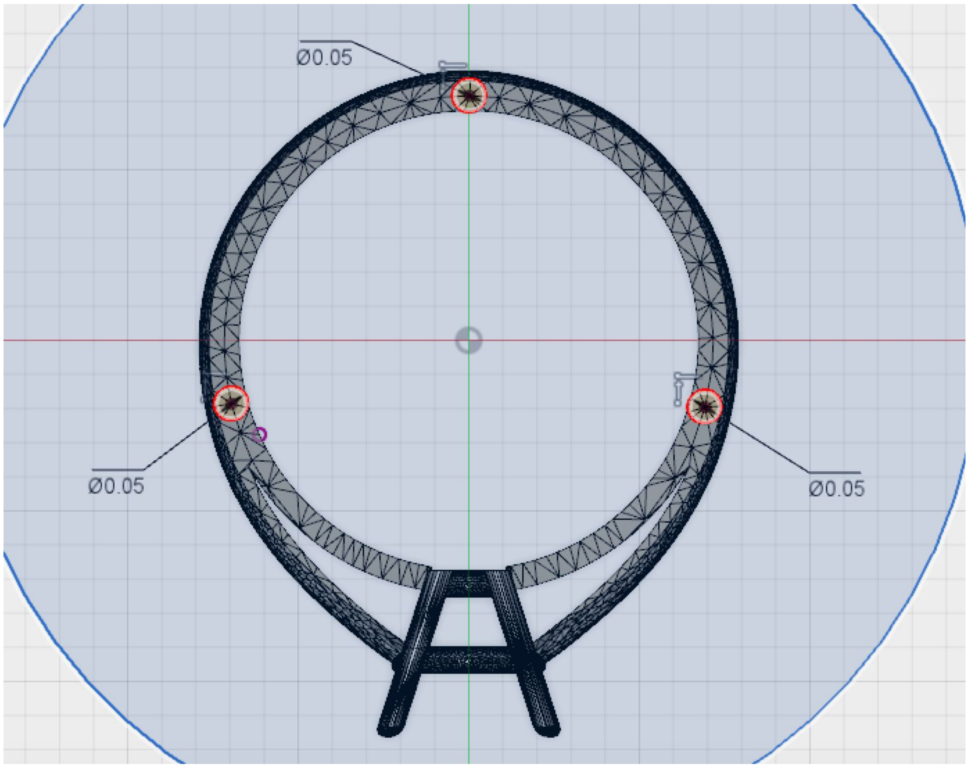
The ring has to be supported while it is being machined so that is maintains its position in space and can be reliably located. In order to avoid cutting away all of the support structure, geometry that will be avoided by the machining process is created.

These supports will have to be removed after the machining is complete in order to separate the workpiece from the waste material. It is desireable to make these supports as small as possible, for ease of removal, while still holding the work rigidly to resist the machining forces so the piece doesn't break while it is being machined.

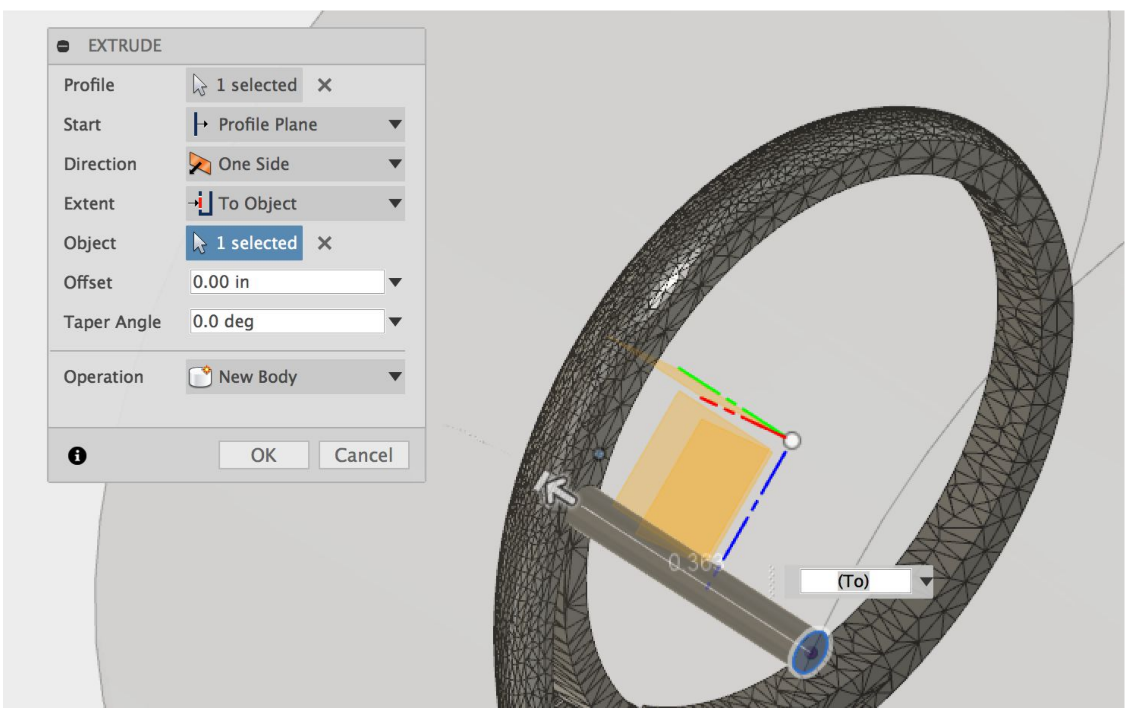
Create an offset plane 0.4 inches below the plane on which the stock circle was sketched.



Sketch three center-diameter circles on that new offset plane by selecting points around the perimeter of the ring that are spaced fairly evenly as shown below. Do not place any supports close to the front of the ring. These will be supports left in place during the machining and would interfere with the machining of the prongs if they were placed in the front.



Extrude the circles up to the bottom of the ring to form the supports. Make sure that the operation chosen is "New Body."



After all of the supports are extruded, the sample ring looks likes this.

1.2.2 Construct Reference Geometry:

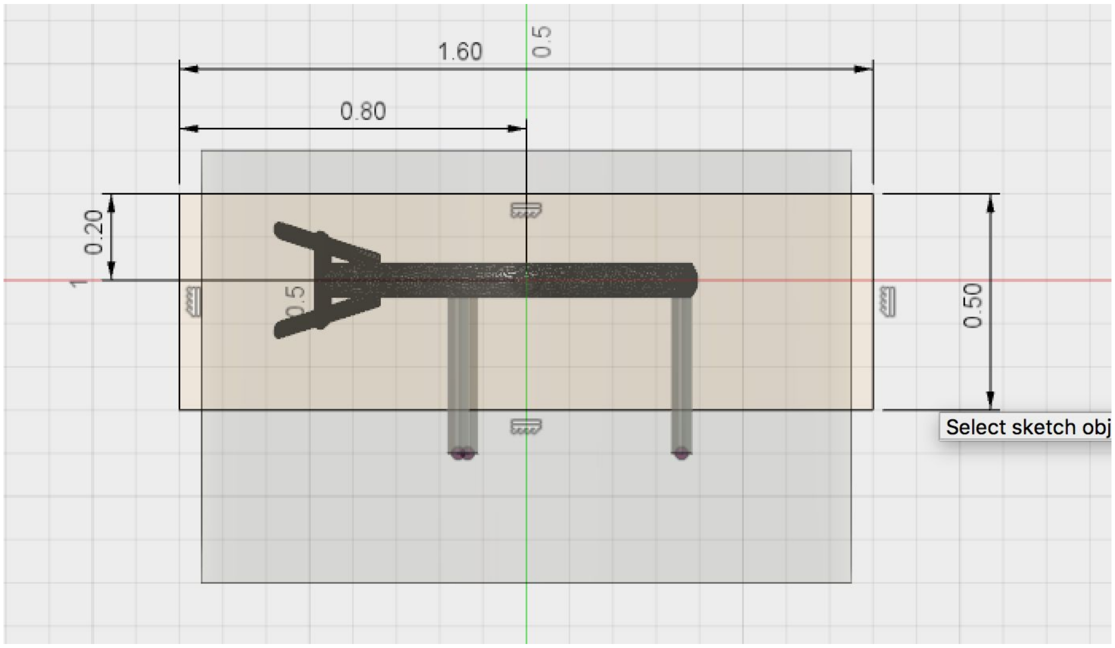
The final pieces of geometry that need to be added are the references needed for the machining operations. In order for the tool to reach all of the geometry on the ring the machining will be done from four basic positions, the top, two sides and the front.

The general machining sequence for any part is to first remove the bulk of the material, then rough out the shape of the part, finally the part is cleaned up with finishing passes. By getting rid of as much bulk as possible at the start, the amount of cutting time with small cutters is reduced. However, care must be taken to leave enough material to support the tool forces of the subsequent steps.

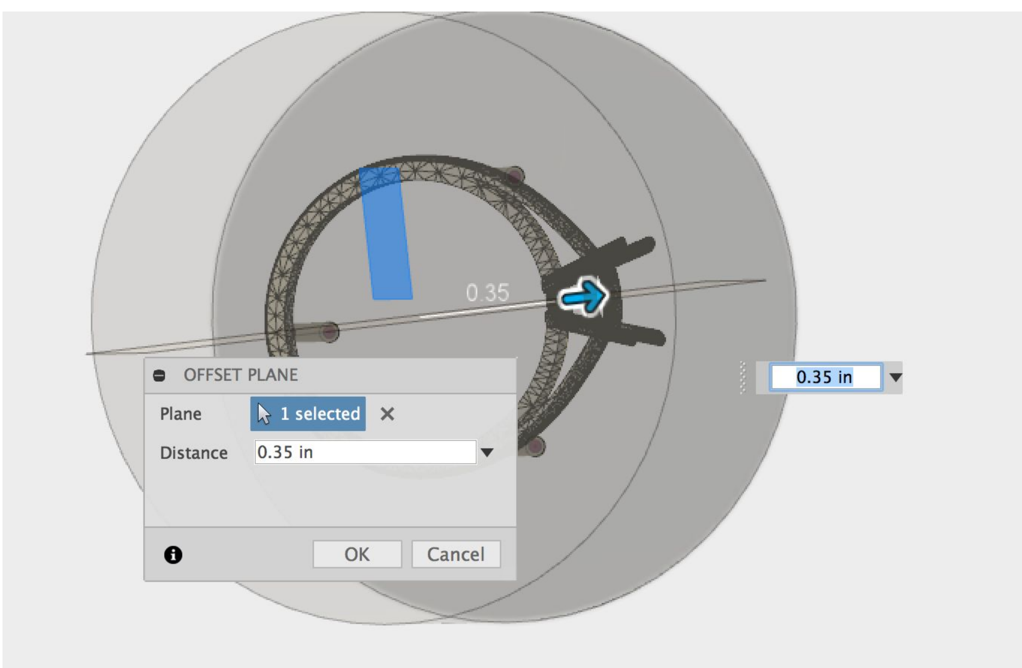
The ring will be machined by removing the excess material from around the perimeter and the middle of the ring. The ring will then be undercut and trimmed to its basic shape on the sides. The front of the ring will then be roughed to shape. The cleanup will be completed by repeating them same positions and doing the final material removal from the top, sides, and front.

In order to avoid wasting time by machining a larger area than needed, as well as to help avoid collisions, some reference sketches will be created.

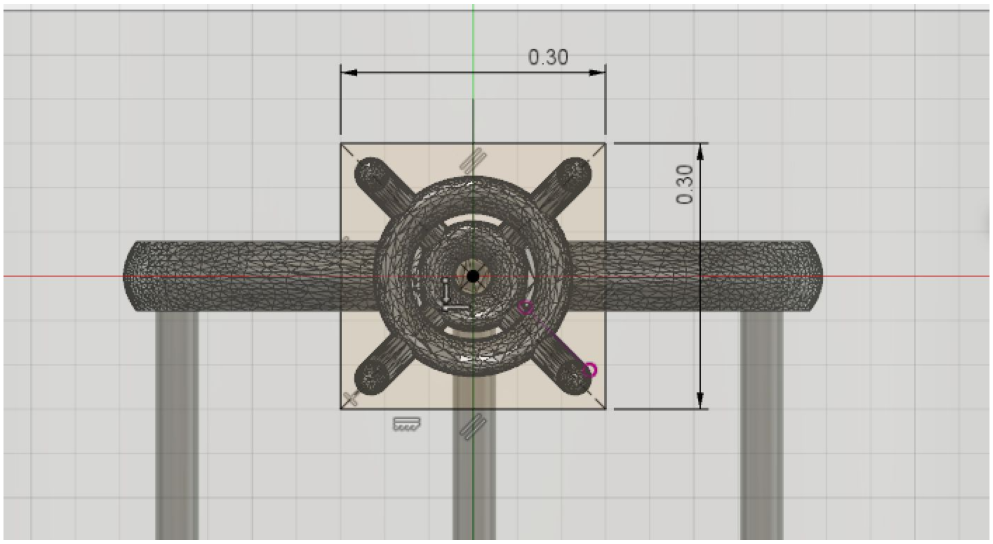
Select the YZ plane and create a rectangular sketch as shown below.



Create a reference plane 0.35 inches in front of the XY plane.



Create a sketch on that new reference plane of a 0.3 inch x 0.3 inch square centered about the middle of the gem setting.

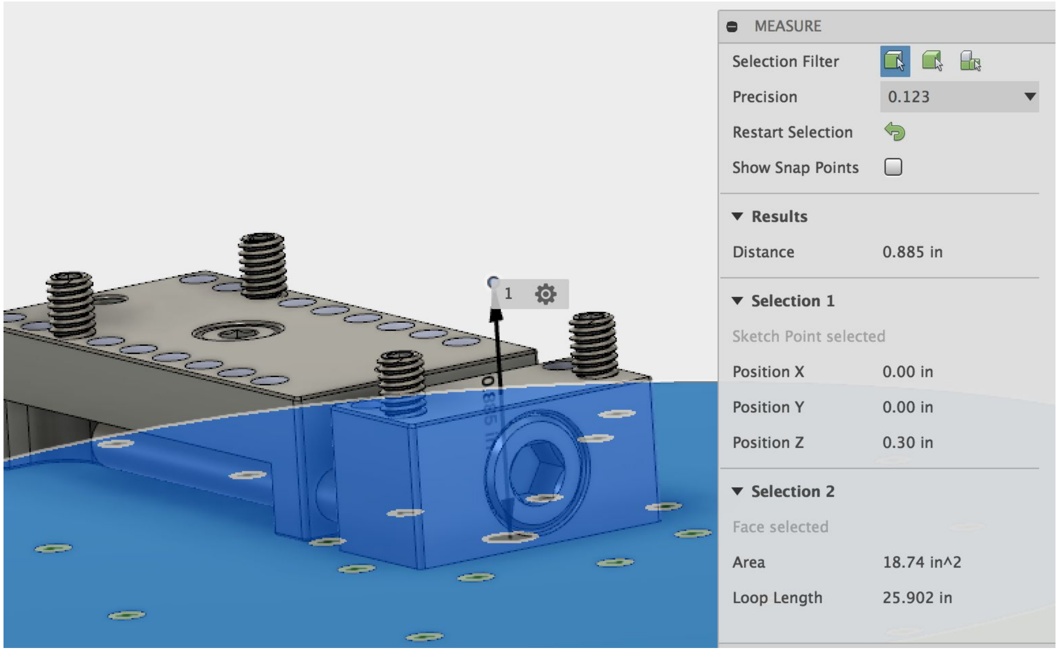


1.2.3 Fixturing the Part

The setup of the ring is now complete. The part now needs to be fixtured in the way in which it will be held when it is cut on the Pocket NC mill. The sample ring will be held in the Pocket NC vise. Users will need to import the "Pocket NC backing plate and vise" and upload the parts to Fusion 360.

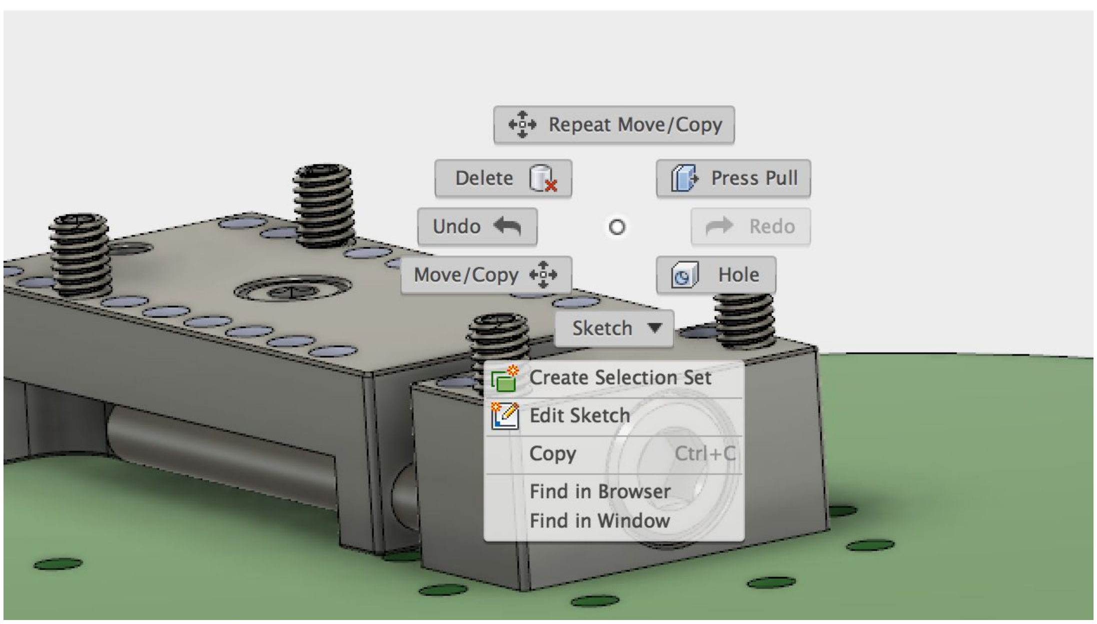
Set B-Table Offset

Open the "Pocket NC backing plate and vise and check if the B table offset is correct for the machine being used. This is the distance from the origin to the B table in the model. The default value is 0.885". This may be measured using the "Inspect" tool.

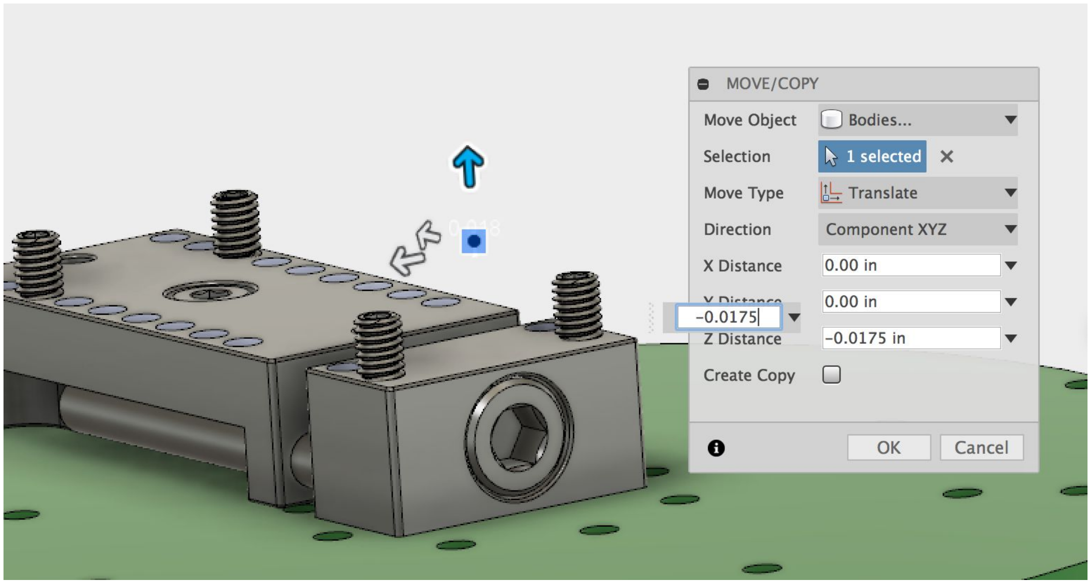


The B-Table Offset, is the point in space about which both the A and B tables rotate. This point in the model must match the point in reality in order for the part to be oriented correctly relative to the machine when the rotary axes are moved.

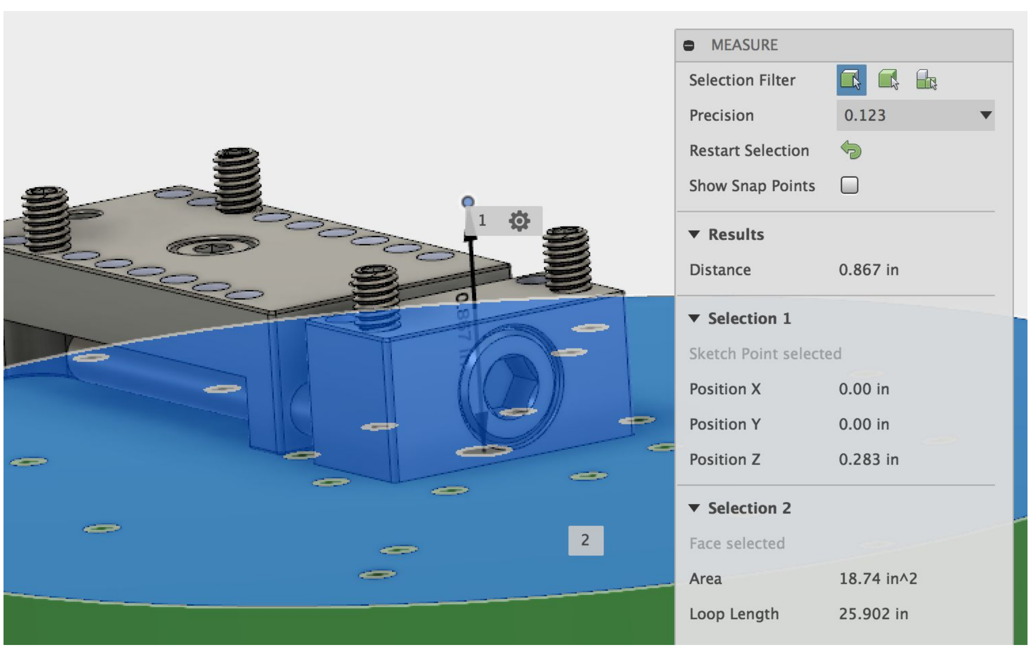
If the location of the offset needs to be changed, start by right-clicking on the origin and selecting "Move."



When the dialog box opens, change the selection filter to "Bodies." Select the origin as the body to be moved. Calculate the distance that the origin needs to be moved. In this case the default origin is 0.885” and the machine B Table Offset is 0.8685. This means that the origin needs to be moved .885 - .8685 = 0.0165” closer to the table.

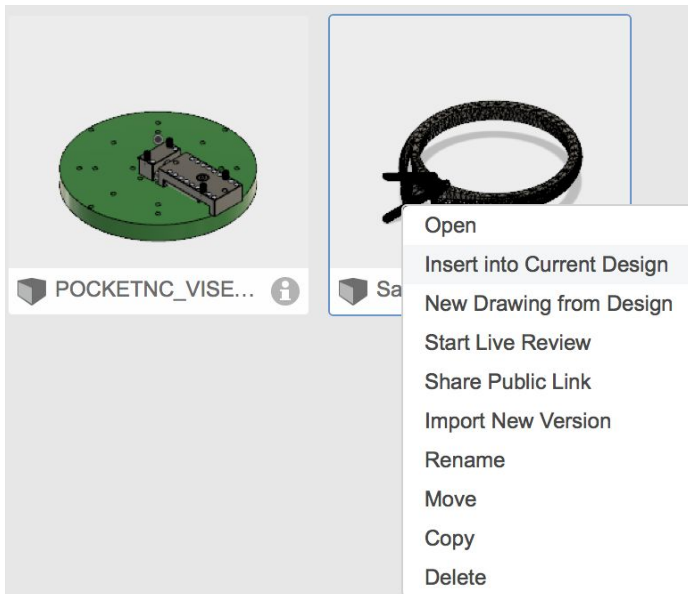


Enter the amount by which the B-table offset in the model needs to be adjusted in order to match the machine’s offset. Click “ok” to apply the change. It is good practice to verify the new location has updated properly before saving the file. Note that the offset will only need to be adjusted in Z direction. It should never be moved in the X or Y axes. Make sure to save the file before proceeding.

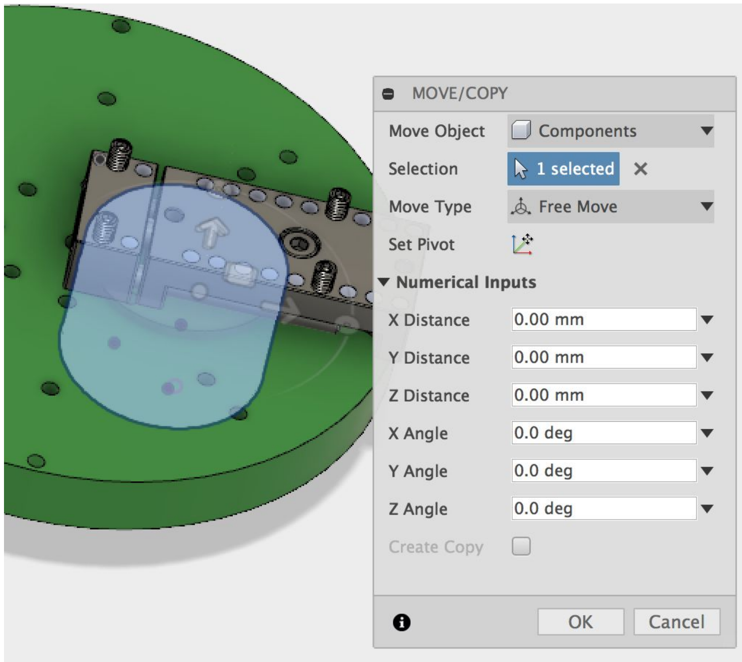


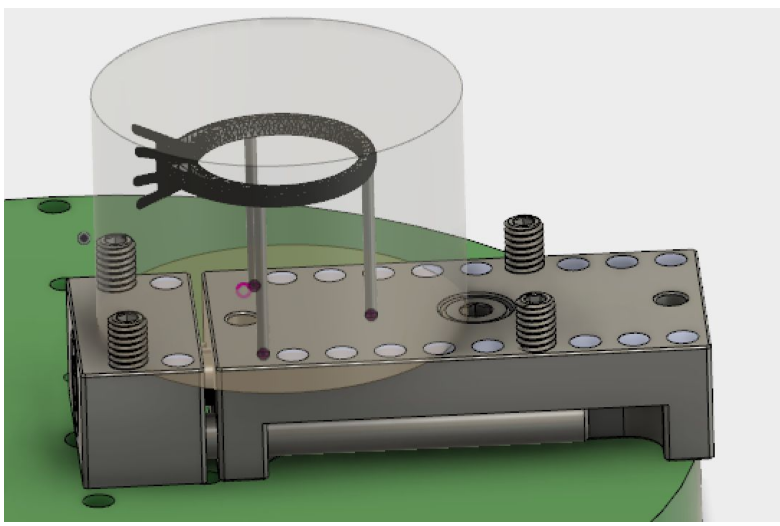
Orient the Part in the Vise

Next the ring model will be inserted into the vise and B table assembly. Open the Data Panel then right click on the ring file that was created earlier and select “Insert into Current Design.”

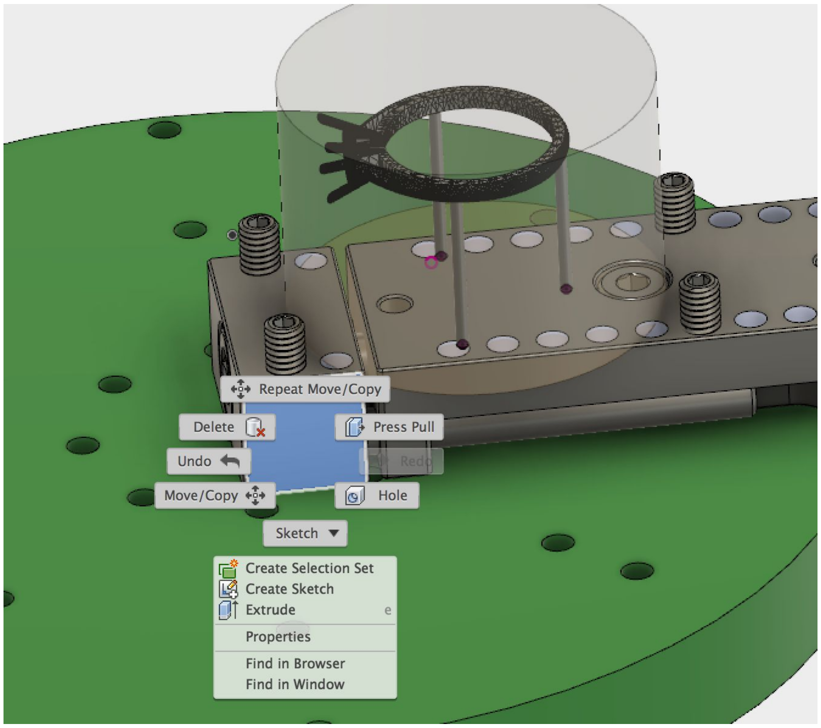


The part will not usually import in the desired position. The first step in the setup is to used the the move commands locate the part in approximately the desired location and orientation.

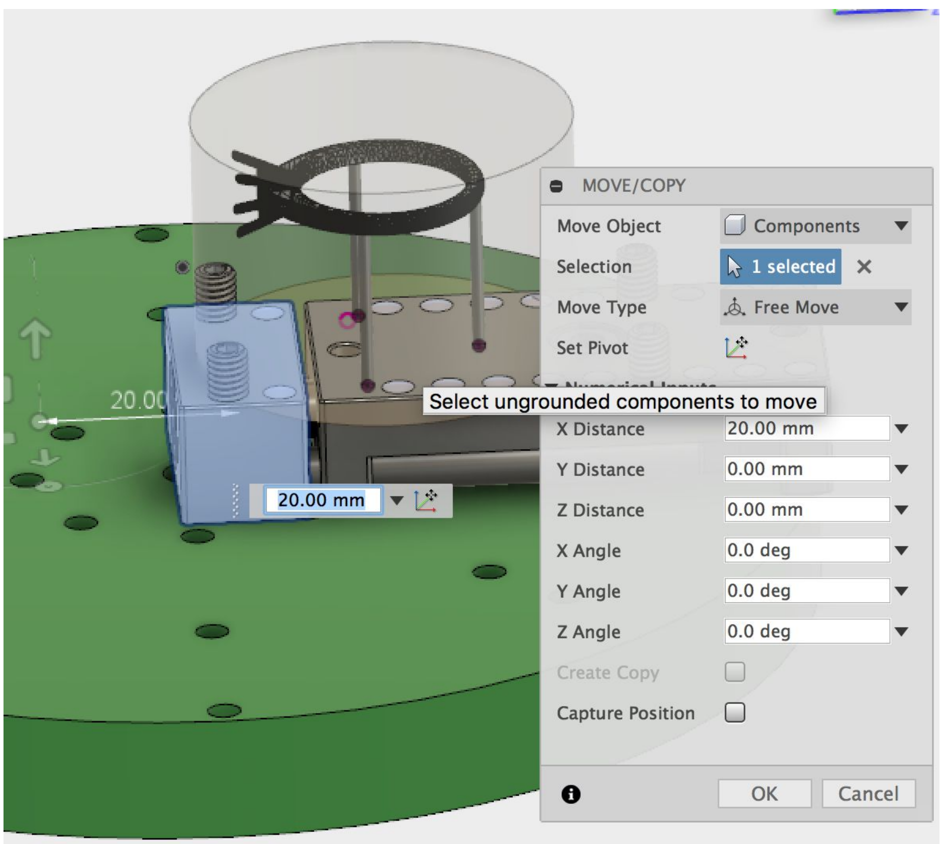




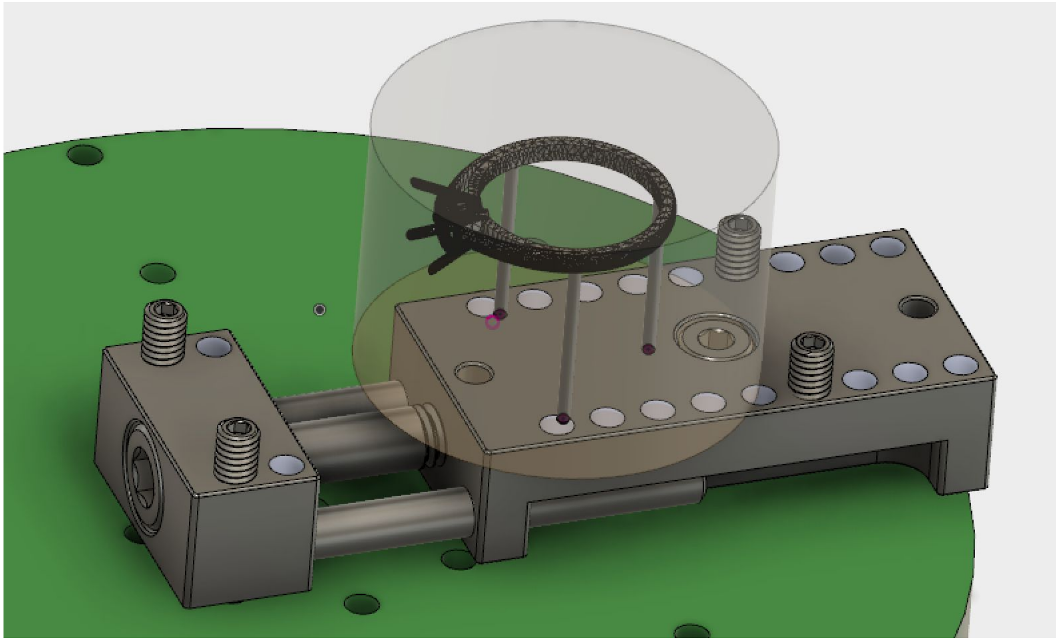
Next the adjustable jaw on the vise(the one with the screw used to tighten the jaws) is opened. Right click on any piece of the adjustable jaw. All of the adjustable jaw pieces are constrained to move together so only one needs to be selected. Click move, and then change the drop down in the options dropdown from “body” to “component.” The arrow pointing in the desired direction of motion may be clicked and dragged or a direction may be selected and the magnitude of the movement may be entered. The jaw is moved 20mm to the left to open it wide enough to hold the stock.



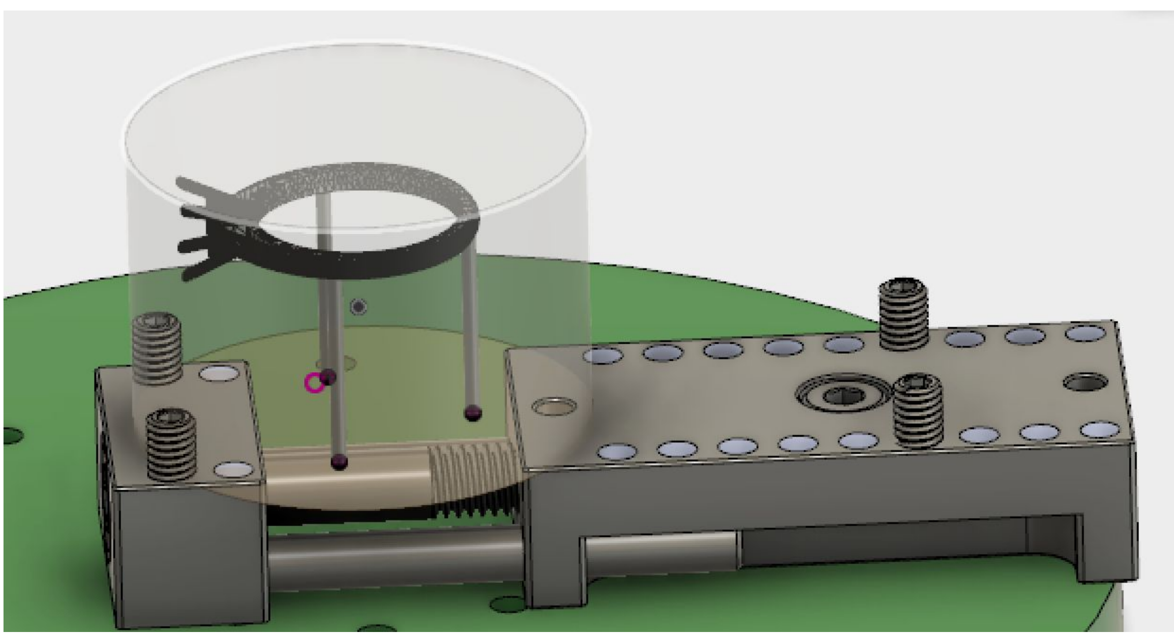
Repeat Move/Copy
Delete
Press Pull
Undo
Redo
Move/Copy
Hole
Sketch
Create Selection Set
Create Sketch
Extrude
Properties
Find in Browser
Find in Window



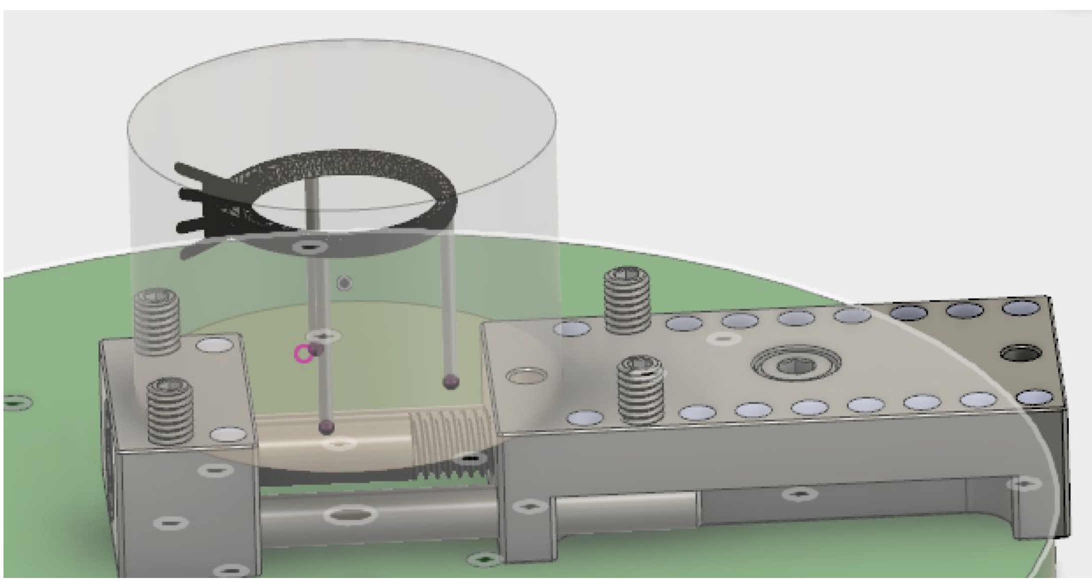
20.00 mm
MOVE/COPY
Move Object
Components
Selection
1 selected
Move Type
Free Move
Set Pivot
Select ungrounded components to move
X Distance
20.00 mm
Y Distance
0.00 mm
Z Distance
0.00 mm
X Angle
0.0 deg
Y Angle
0.0 deg
Z Angle
0.0 deg
Create Copy
Capture Position
OK
Cancel



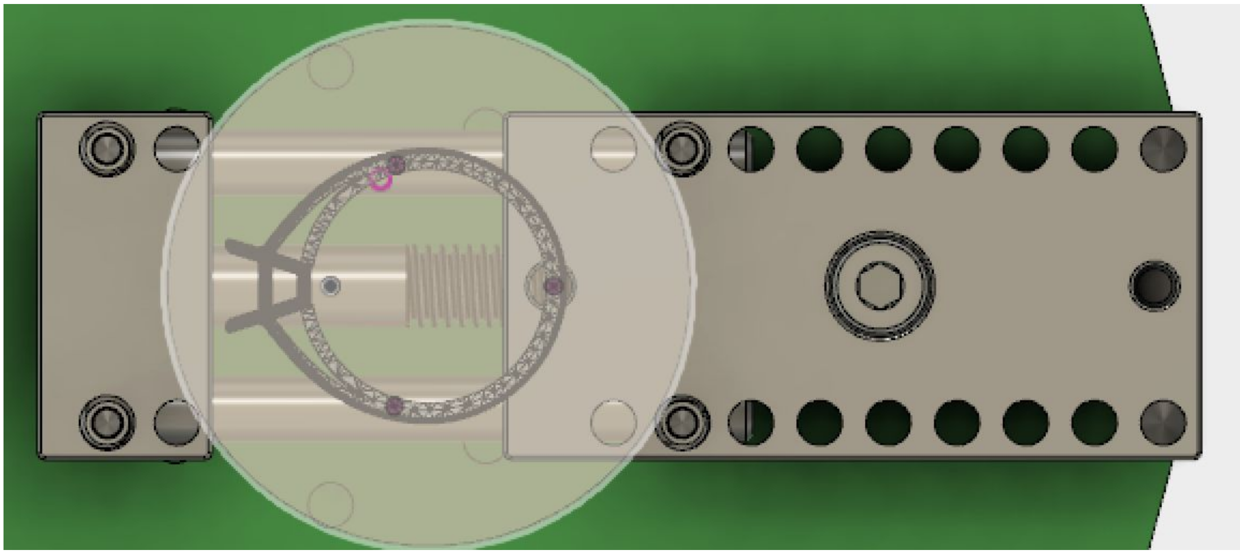
The workpiece is then moved about 20 mm to the left as well so that the grippers on the fixed jaw can be more easily accessed.



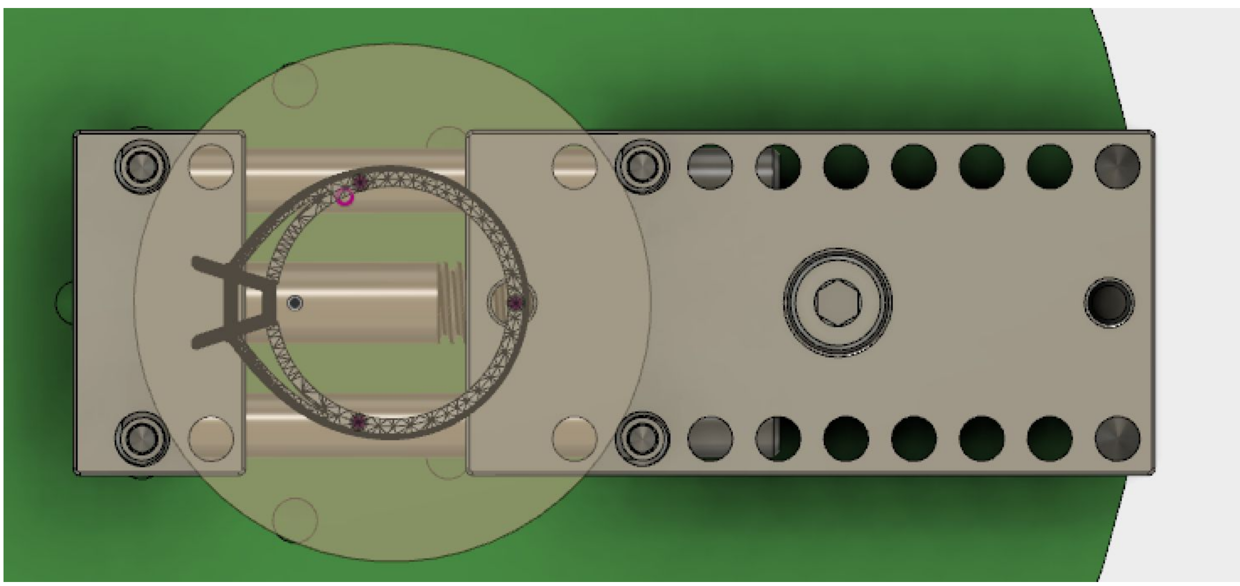
The grippers on the fixed jaw(the one with two rows of 9 holes drilled in it) are moved 20mm to the left. The holes are 5mm apart so the grippers are moved 4 holes to the left. Since the grippers must fit in the holes the adjustment of the fixed jaw is in 5mm increments.



Next, the stock is moved back to the right so that it just barely overlaps the grippers on the fixed jaw. This overlap is to account for the crushing/biting into of the stock material that occurs when the vise is closed. The overlap should be about 0.01in. This overlap will give the user a better representation of what will happen to the material when the vise is closed than setting the grippers and stock tangent.

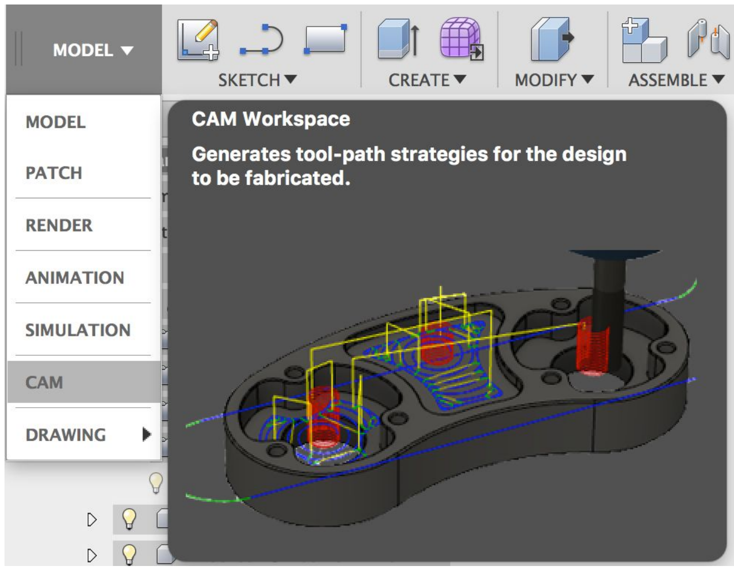


Finally, the moveable jaw is adjusted back to the right so that the grippers on the moveable jaw overlap the stock in the same way at the grippers on the fixed jaw.

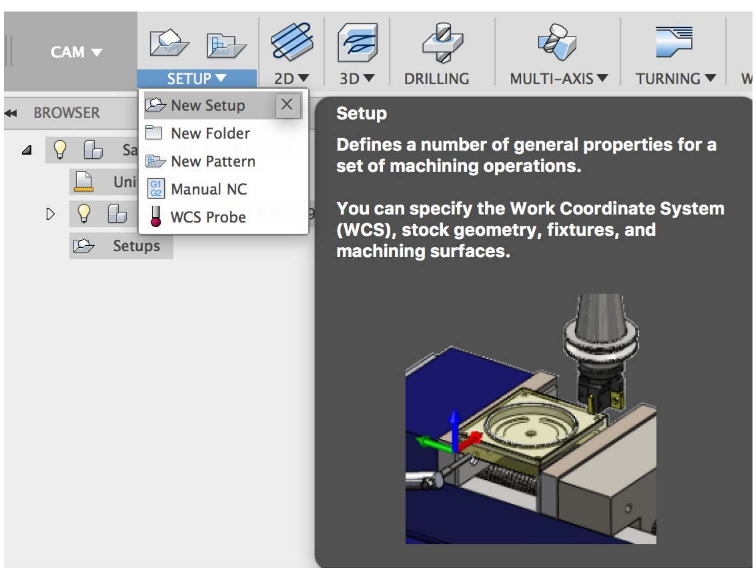


2 CAM SETUP

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.

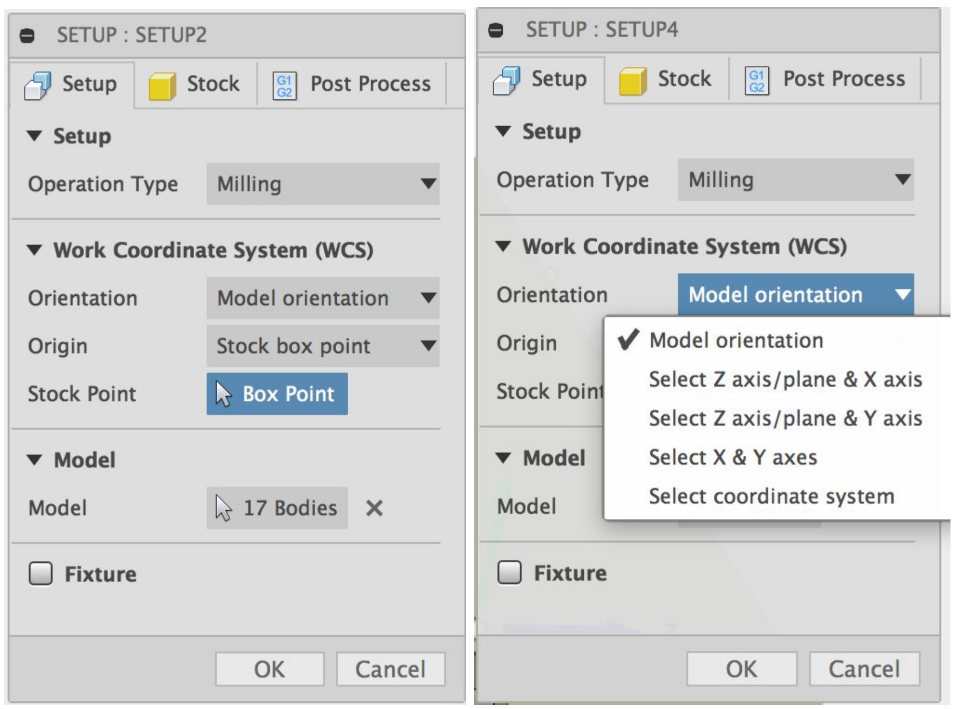


Before any toolpaths can be created a setup has to be created that defines the location of the origin, the stock, the fixtures, and the coordinate system. These features must be chosen in a way that makes it possible to make the physical setup on the machine match the one used to write the toolpaths in the model. To create a Setup select the setup dropdown and “New Setup”.

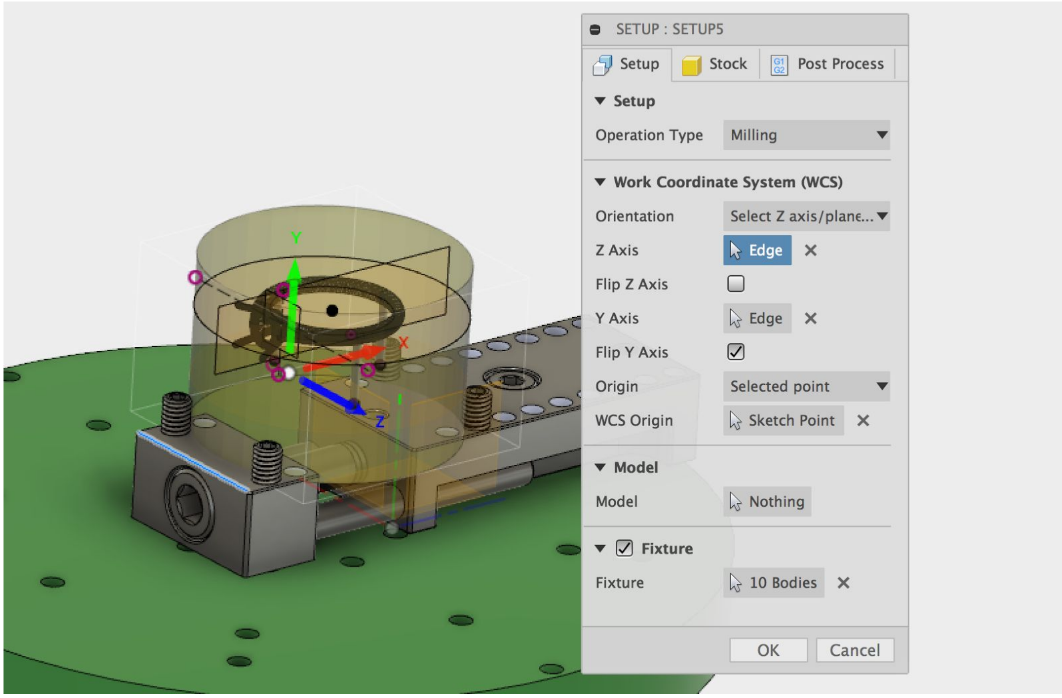


The new setup brings up a dialog box where settings can be changed and model elements can be used to define the origin and coordinate system. The dialog box has three tabs, only the two on the left need to be addressed before writing the toolpaths for the part. First the operation type, coordinate system, and model items need to be identified. Then the stock needs to be defined.

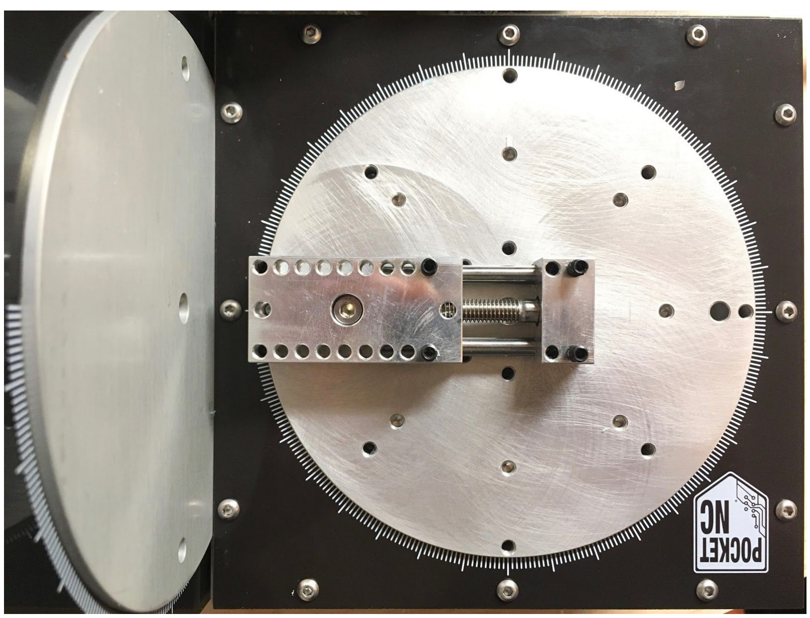
Since the ring is being milled, the operation type is “milling.” Next the work origin is selected and the Work Coordinate System (WCS) is orientated. To begin the setup of the WCS select the WCS dropdown and choose the “Select Z axis/plane & X axis” option for the orientation.



The two pieces of geometry needed to define this orientation are the Z axis or plane and the Y axis. To define these first select the box next to the “Z Axis” labeled “Nothing”, then select a straight line on the vise perpendicular to the axis of motion of the moveable jaw and parallel to the work table (B table). Repeat this selection process for the Y Axis, this time selecting a line perpendicular to the work table (B table). When finished, the WCS orientation should look similar to the one shown below. If needed the direction of the axes may be changed by using the checkboxes to flip them.

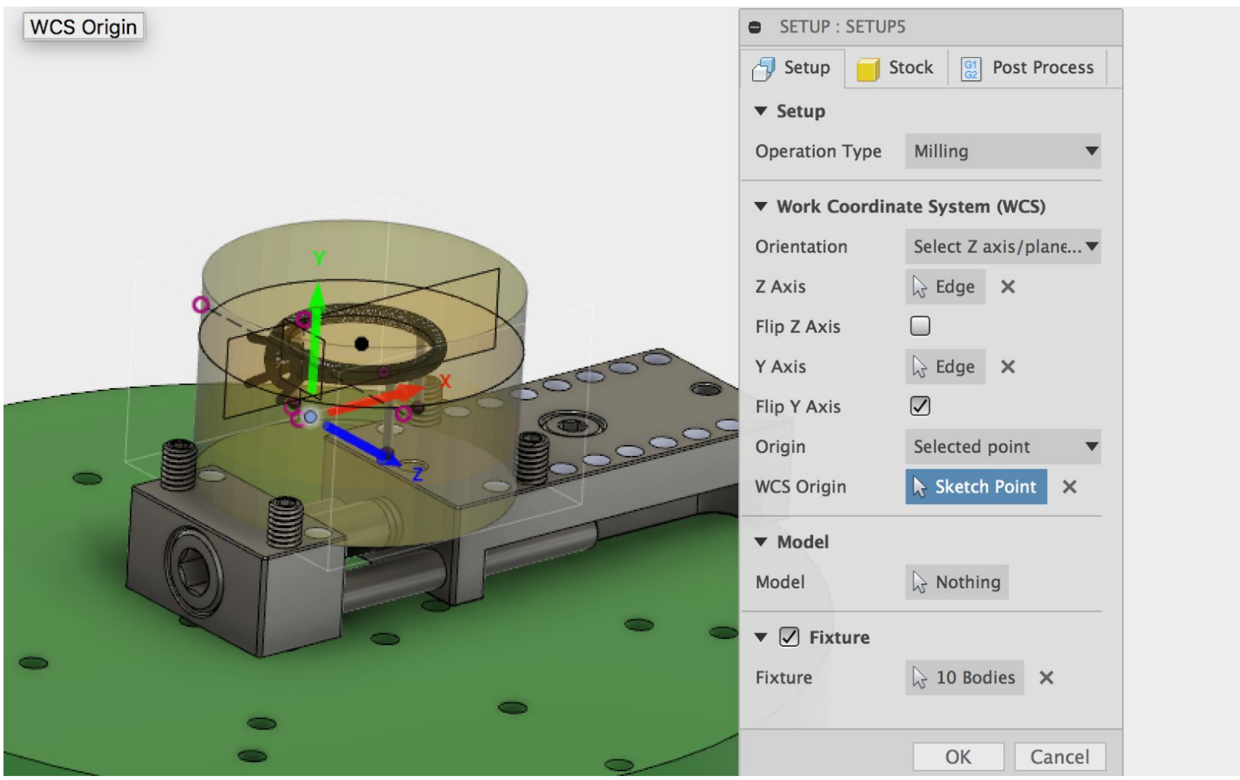


This orientation corresponds to the way the axes are aligned when the machine is the its home position. The B table is flat or parallel to the spindle of the Pocket NC mill in the home position. The Y axis moves the table up and down. The Z axis is parallel to the spindle and the X axis is perpendicular to both Y and Z and moves the A and B tables forward and back. The home position of the machine is shown below for reference. The spindle is to the top of the image.



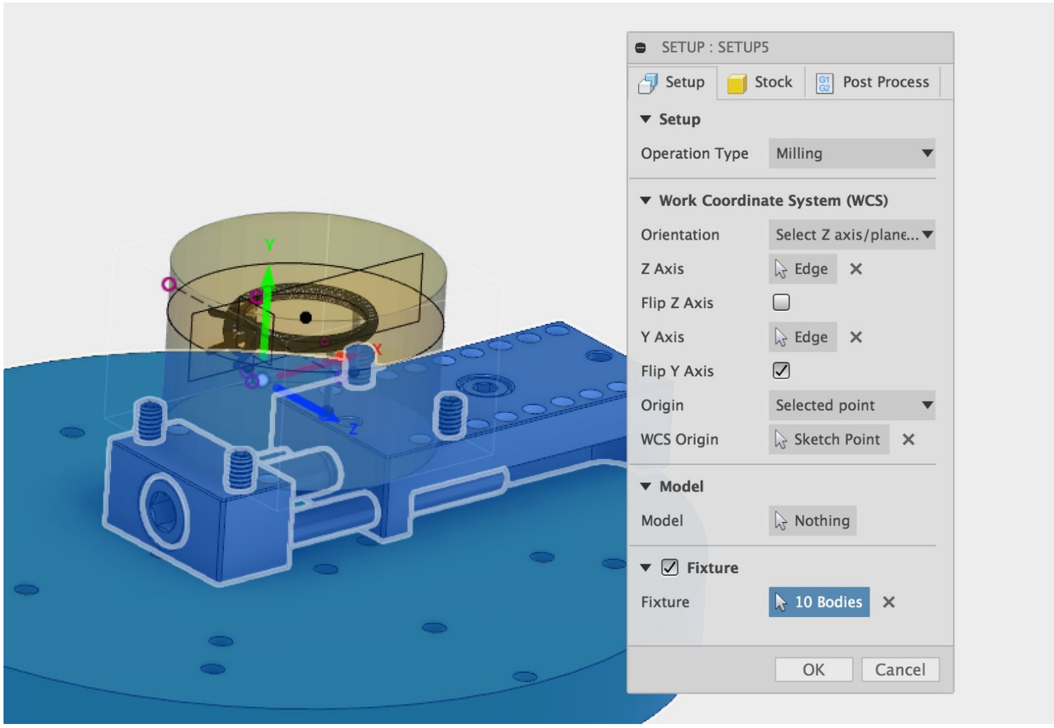
Next the origin of the machine is changed within the model. The machine origin is represented by a point in space 0.885in off the surface of the B table. It is the point in space where the axis of rotation for the A table and the axis of rotation of the B table intersect. This point varies slightly from machine to machine and the distance must be changed to match a specific machine. This distance is called the “B table offset” and can be found in the documentation provided with each machine.

In order to change the machine origin first choose “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.

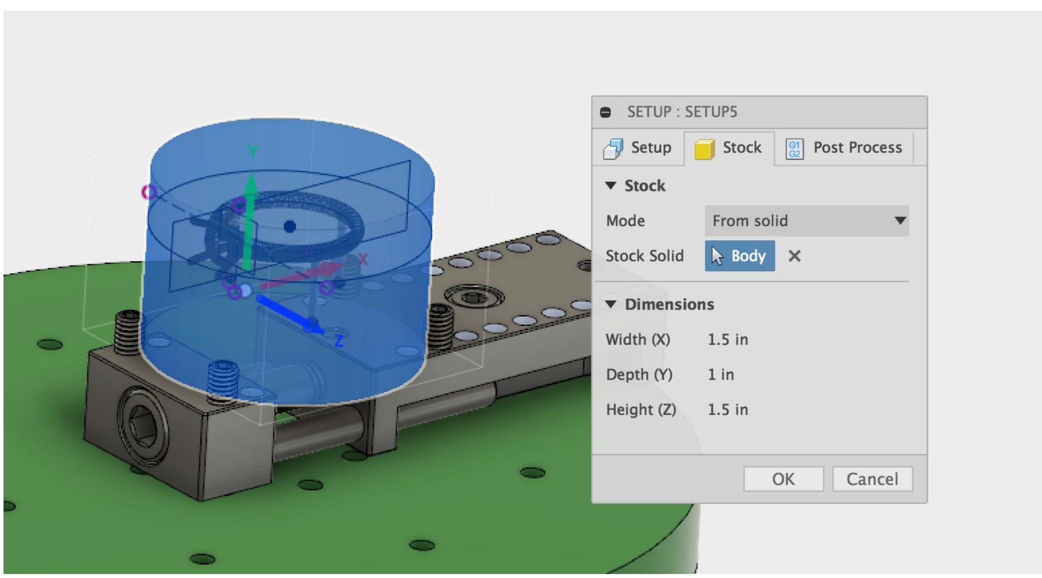


The “Model” tab is left with no selection. The model will be defined for each machining step instead.

Next, check the fixture box. The fixture is the parts of the machining system that are used to hold the workpiece. When a model item is labeled as a fixture the software used to create toolpaths automatically avoids running into it by placing restraints on the space the tool is allowed to cut or cross. This helps prevent breaking tools and damaging the vise. The B table, vise, grippers and screws are all marked as fixture pieces as shown below.



Next the stock size and shape is defined. Click the “Mode” drop down and select the “From Solid” option. Click the box next to “Stock Solid” labeled “Nothing” Then click the cylinder that was modeled earlier as the stock from which the ring is machined.



For this tutorial, nothing needs to be changed under the “Post Process Tab.” This is where you could name the program as well as naming the WCS offset if desired.

When finished with the setup, select “Ok” to save and close the setup.

3 Tool Paths: Generating machinable files

The Pocket NC mill is operated by giving it a set of directions, called a toolpath, that is encoded in a machine control language called G-code. These instructions include spindle speed, coordinate positions, and speed of movement between those positions. Since it is difficult to write complex machine codes by hand, the toolpaths are created in the Computer Aided Machining (CAM) environment of Fusion 360 then converted to machine code that the Pocket NC mill is designed to execute using a processor. This is similar to the process of writing and compiling programs in other computer languages.

Programs are written in a sequential series of steps, called operations. Each operation contains one toolpath. Each toolpath uses a specific tool and accomplishes a different machining process such as roughing, facing, or drilling. Each operation has a unique set of conditions including spindle speed, surface speed, and feed rate. These parameters are changed depending on the machining operation being performed, the tool being used, the material being cut, and the surface finish that is desired.

When a program is run on the Pocket NC mill, tools may have to be changed or the work may have to be repositioned in between operations.

The wax ring shown in the following tutorial will use the following parameters:

Material:
Machinable Wax
Diameter 1.5” Height 1”
The wax ring test part is designed to be cut from wax using the Pocket NC mill. Without changing the parameters, harder materials may cause damage to the Pocket NC mill as feeds and speeds have been selected specific to wax.

Runtime:
~1 hour

Tooling:
The ring will be cut using the ⅛” shank tools made by Harvey Tool and available from Pocket NC. The specific tools required are a ⅛ inch 2 flute end mill and a 1/32 inch ball nose end mill.

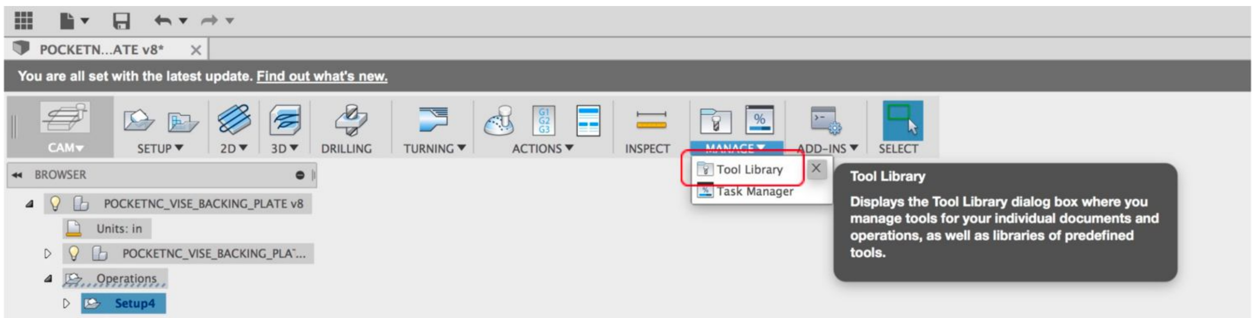
Tool Number 6
⅛ inch 2 flute flat 45 deg helix (Harvey Tool # 24208)

http://www.pocketnc.com/45-degree-square-helix/18th-inch-45-degree-square-helix-2-flu
te

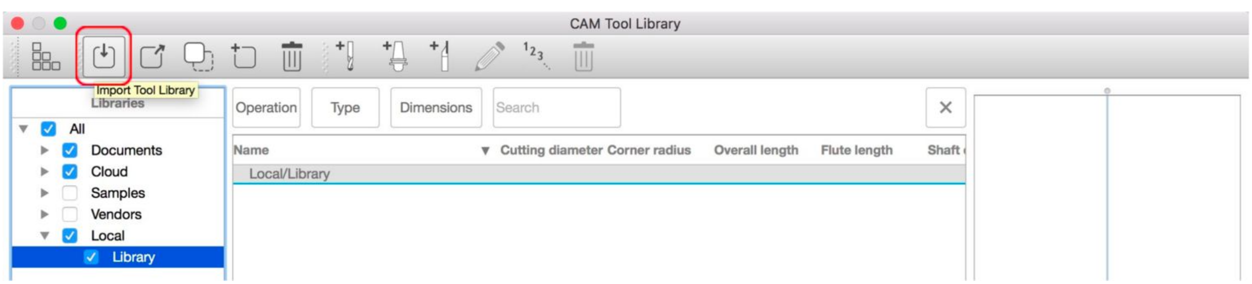
Tool number 12
.031 inch 2 flute 2 deg ball nose (Harvey Tool #29831)
http://www.pocketnc.com/tapered-rib-cutters/10-degree-164th-tapered-rib-cutter

Fixture:
The wax should be held in the Pocket NC vise using threaded set screws to grip the sides of the wax blank.

Before the toolpaths can be generated, the Pocket NC Tool Library needs to be downloaded. To install the new tool library, select “Manage” then “Tool Library.”



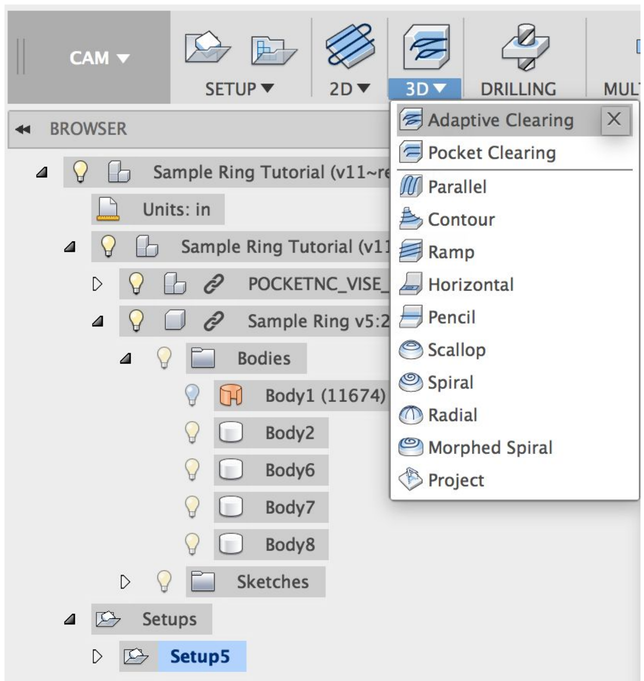
Next, under “Libraries” select “Local” then “Library.” Now use the “Import” tab to select the Pocket NC Tool Library from your downloads location. Close the window to save changes and exit the library manager.



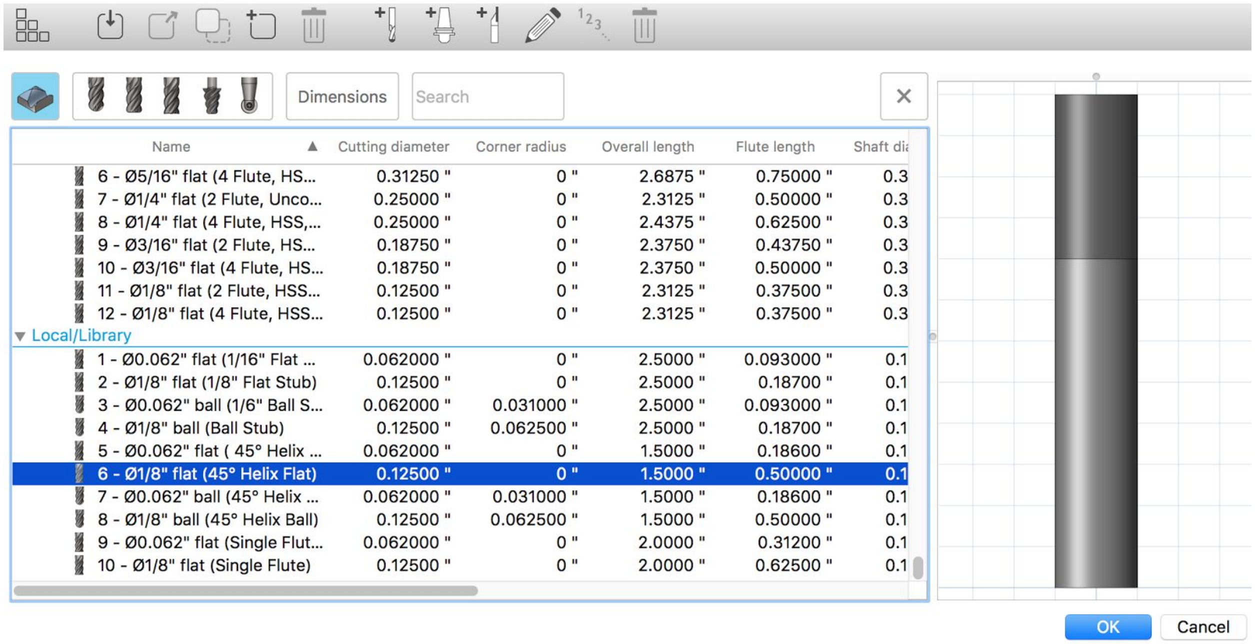
The first step in machining the ring is removing the bulk of the excess material. Adaptive toolpaths are efficient for removing excess material because they keep the cutting tool in contact with the work as much as possible and minimize the number of amount of time needed to hog out rough shapes. To create the first toolpath, select “3D” then “Adaptive Clearing.”

Operation 1: Adaptive Clear Top

The first operation uses adaptive clearing to remove as much of the material as possible in an efficient manner. Start by selecting 3D tab in and then click on “Adaptive Clearing.”

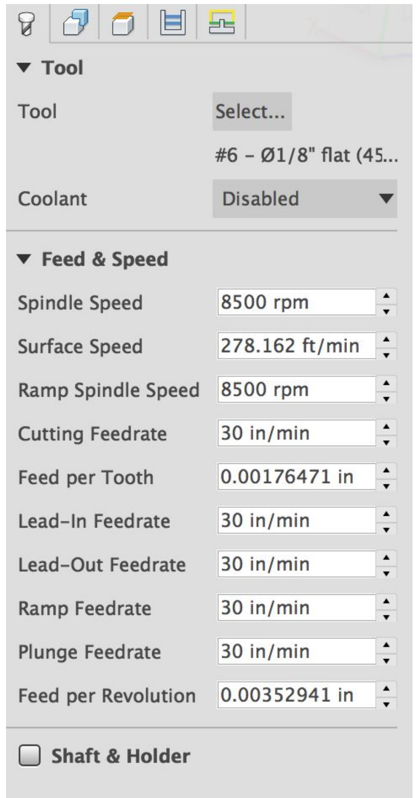


A dialog box will open where the parameters for the toolpath are entered. The first parameter to be set is the tool. Click on the “Select” button under the tool tab. This allows a tool to be chosen from the tool library. Scroll all the way to the bottom of the list of tools and select tool 6, ⅛ inch flat 45 deg. end mill, out of the “Local Library” which is the Pocket NC tool library.



Name	Cutting diameter	Corner radius	Overall length	Flute length	Shaft di
6 - Ø5/16" flat (4 Flute, HS...	0.31250 "	0 "	2.6875 "	0.75000 "	0.3
7 - Ø1/4" flat (2 Flute, Unco...	0.25000 "	0 "	2.3125 "	0.50000 "	0.3
8 - Ø1/4" flat (4 Flute, HSS,...	0.25000 "	0 "	2.4375 "	0.62500 "	0.3
9 - Ø3/16" flat (2 Flute, HS...	0.18750 "	0 "	2.3750 "	0.43750 "	0.3
10 - Ø3/16" flat (4 Flute, HS...	0.18750 "	0 "	2.3750 "	0.50000 "	0.3
11 - Ø1/8" flat (2 Flute, HSS...	0.12500 "	0 "	2.3125 "	0.37500 "	0.3
12 - Ø1/8" flat (4 Flute, HSS...	0.12500 "	0 "	2.3125 "	0.37500 "	0.3
Local/Library					
1 - Ø0.062" flat (1/16" Flat ...	0.062000 "	0 "	2.5000 "	0.093000 "	0.1
2 - Ø1/8" flat (1/8" Flat Stub)	0.12500 "	0 "	2.5000 "	0.18700 "	0.1
3 - Ø0.062" ball (1/6" Ball S...	0.062000 "	0.031000 "	2.5000 "	0.093000 "	0.1
4 - Ø1/8" ball (Ball Stub)	0.12500 "	0.062500 "	2.5000 "	0.18700 "	0.1
5 - Ø0.062" flat (45° Helix ...	0.062000 "	0 "	1.5000 "	0.18600 "	0.1
6 - Ø1/8" flat (45° Helix Flat)	0.12500 "	0 "	1.5000 "	0.50000 "	0.1
7 - Ø0.062" ball (45° Helix ...	0.062000 "	0.031000 "	1.5000 "	0.18600 "	0.1
8 - Ø1/8" ball (45° Helix Ball)	0.12500 "	0.062500 "	1.5000 "	0.50000 "	0.1
9 - Ø0.062" flat (Single Flut...	0.062000 "	0 "	2.0000 "	0.31200 "	0.1
10 - Ø1/8" flat (Single Flute)	0.12500 "	0 "	2.0000 "	0.62500 "	0.1

Adjust the setting under the tool tab so that they match the ones in the image below. The spindle speed and feed rates are selected based upon the size and type of cutter being used as well as the type of material being cut. See this tutorial on the Pocketnc website for more information about the selecting the right feed and speed for your material and cutter. These feeds and speeds are for machining wax using the ⅛ inch flat end mill. Milling different materials or using a different cutter would require changing these parameters.

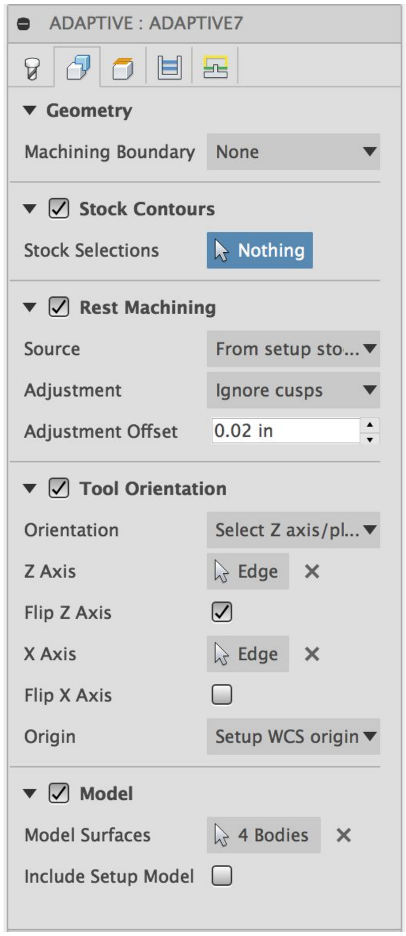


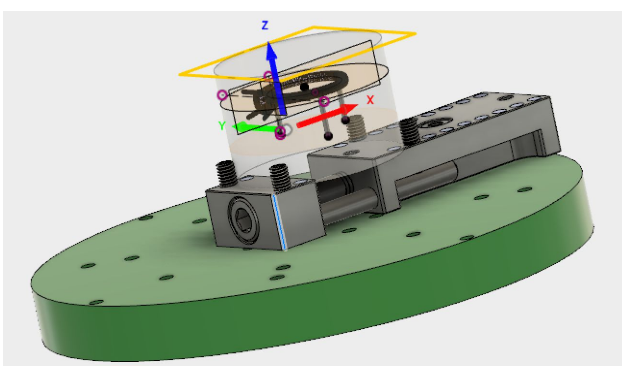
Click the geometry tab and update it to match the settings shown below. The “Stock Contour” check box is selected. This lets the program use the geometry of the round block, which was defined as stock earlier to define the toolpaths. Because the cylinder was selected as the stock in the setup, it does not need to be selected again here.

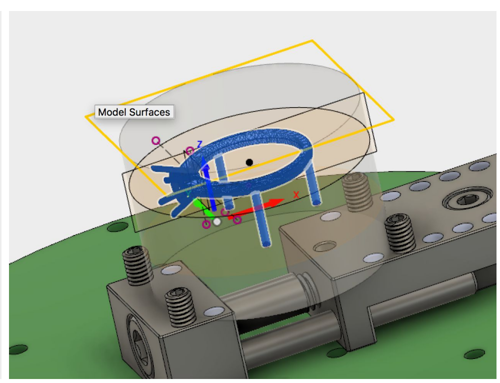
Make sure that the “Rest Machining” check box is selected. The settings below should be the default options once it is selected.

Select “Tool Orientation”. Change the orientation type to “Z axis/plane & X axis”. Select Geometry on the vise so that the orientation of the tool matches what is shown. It is critical that this orientation be correct since the post processor uses the work coordinate system and the tool orientation to drive the A and B rotary axis movements.

Click “Model” then select the ring and the three supports as the model surfaces. A simple way to do this is to click the “Model Surfaces” button then select the appropriate bodies from the browser. The selected surfaces highlight in blue once they are chosen.



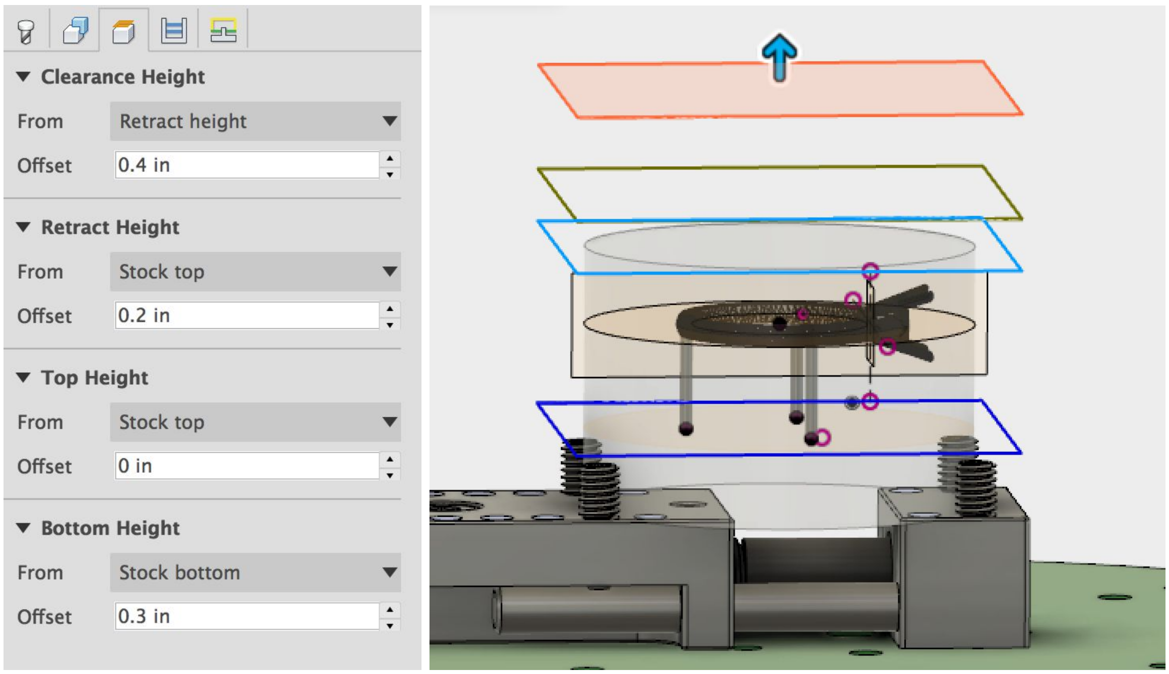




Next, select the “Heights” tab to set the top, bottom, retract and clearance heights. These heights define the space in which the tool is allowed to operate. The retract plane defines the height to which the tool is retracted during drilling or pocketing operations. The clearance plane is the height to which the tool must be retracted before rapid movements or rotation of the tables is allowed. These planes must be selected appropriately so that collisions are avoided.

The top plane is used to define the top surface of the material being milled. The bottom plane is used to define how far below the bottom plane of the part the tool will be allowed to cut. The bottom plane of the part is used as a reference for the bottom plane offset. The bottom plane is defined so that the tool stays above screws on the vise. This avoids collisions as well as makes sure that enough stock remains for the the vise to grip securely.

The planes are shown below, top to bottom in the figure: clearance, retract, top, and bottom planes in orange, green, light blue, and dark blue respectively.



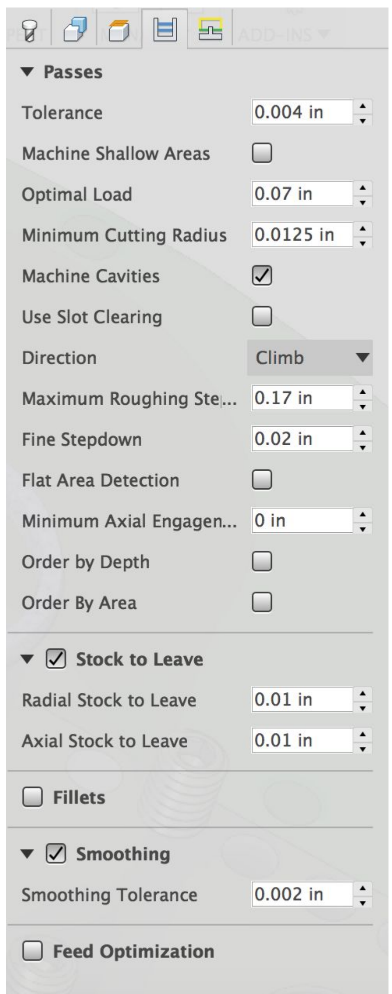
Next, the tool pass settings are defined. These settings control how closely to the model the geometry will be cut, how deep of cuts to make, and how much material to engage in each pass. In this example, the optimal load is increased from the default setting, the maximum stepdown and final stepdown, amount of stock left, and smoothing tolerance are all reduced from the default values.

When the adaptive toolpath is used, Fusion calculates a toolpath that keeps load on the cutter constant. This increases efficiency and reduces tool wear. The maximum stepdown is the depth of each pass. This program will take steps of 0.17 inches and then finish with a pass 0.02 inches deep.

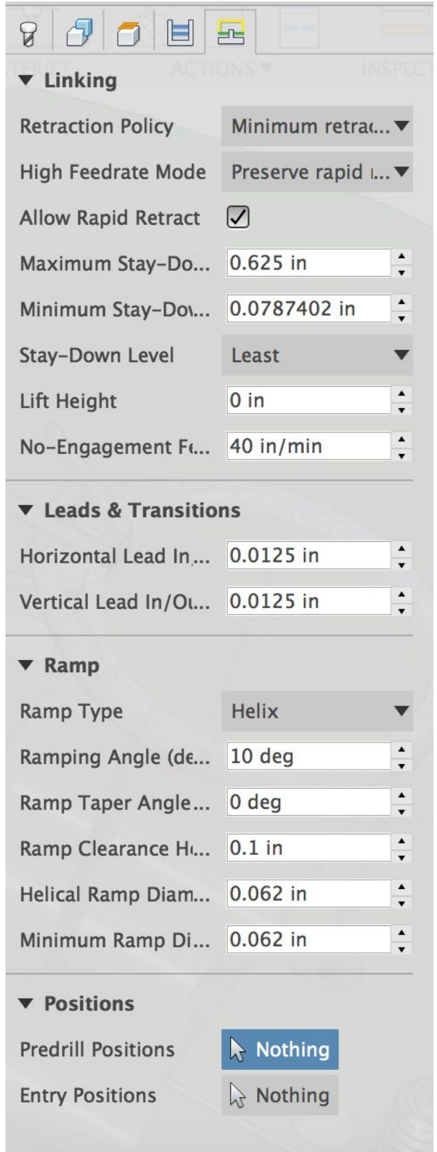
For a roughing operation the optimal load on the cutter is about 60% of its diameter. This helps lengthen the tool life by allowing the tip of the cutter to engage the work at an angle that increases its cutting efficiency and reduces wear.

The amount of stock left controls how much material is left for cleanup operations. It is not desirable to remove all of the material with an adaptive clearing operation because the surface finish will be poor. In general, to get a better surface finish the last passes must be very light (low amounts of material removal) and the cutting direction must be controlled more closely (different methods leave different marks on the finished part). Adaptive clearing is good for removing lots of material efficiently, but it leaves a poor surface finish that has to be cleaned up.

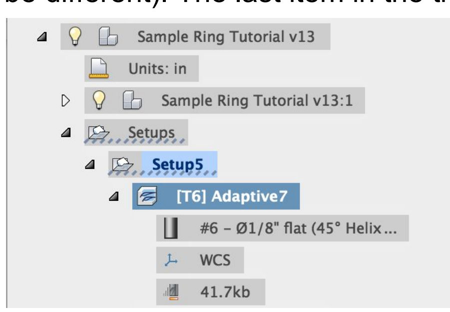
The model of the ring has lots of facets, but a smooth part is desired. Smoothing tells the toolpath software to ignore sharp edges and corners below a specified radius and allows the cutter path to not follow the contour of the part exactly. Smoothing is talked about in more detail in the finishing operations.



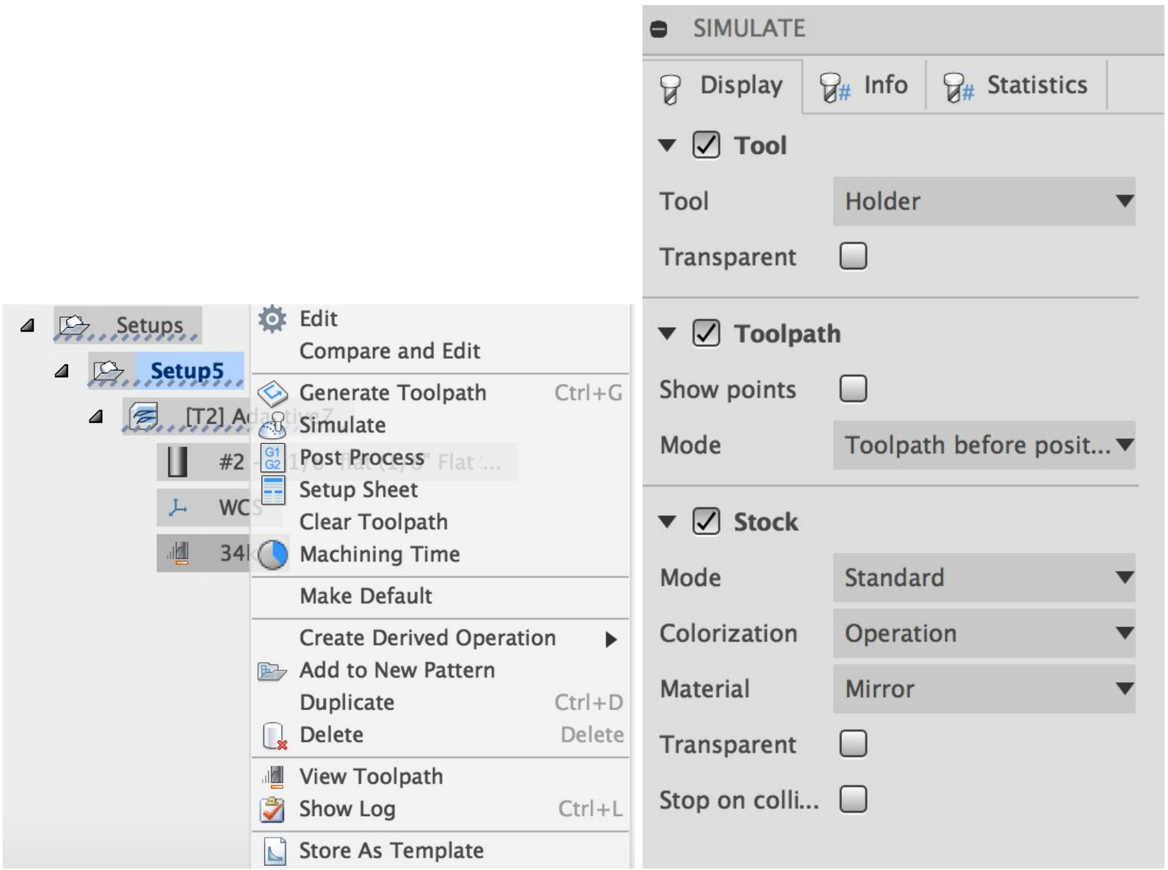
Finally, under the linking tab, the angle of the helical ramp is increased to 10 degrees and the radius of the helical ramp is decreased to be one half of the radius of the cutter being used. Once the settings are correct, click “ok” to save and apply the settings.

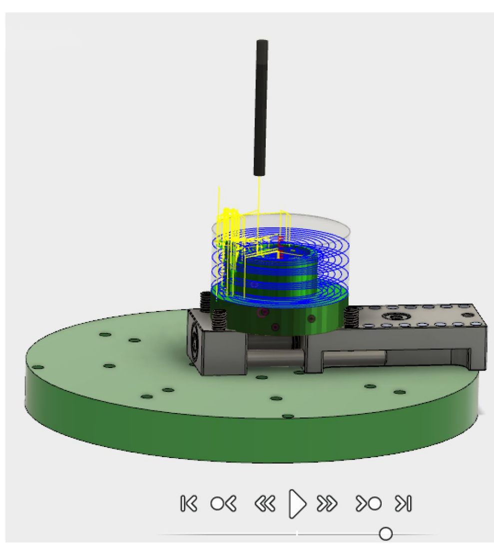


Once the settings dialog is closed, Fusion will automatically generate a toolpath based upon them. The bottom of the tree in the browser should now look like this (setup and toolpath numbers may be different). The last item in the tree is the new toolpath.



Right click on the toolpath and select “Simulate”, when the new dialog box opens select the “Stock” check box to show the stock. Use the controls at the bottom of the window to control playback or to change the playback speed. The “Info” and “Statistics” tabs on the “Simulate” dialog box can be used to view other information about the program such as machining time and collision detection.

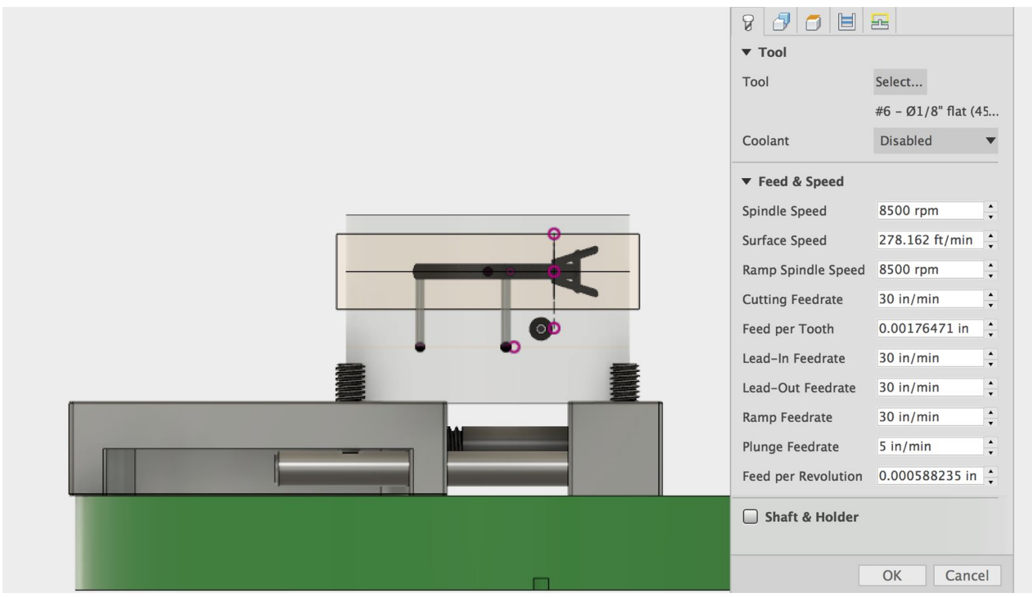




Operation 2: Adaptive Clear Side 1

The second and third operations use adaptive clearing to remove bulk material from the sides and below the ring. As with Operation 1, start by selecting 3D tab and then click on “Adaptive Clearing.”

Tool number 6 may already be selected. If it is not, hit the “Select” button and choose it from the library as described in Operation 1. The settings may have auto populated from the previous operation. Confirm that they match the ones shown below.

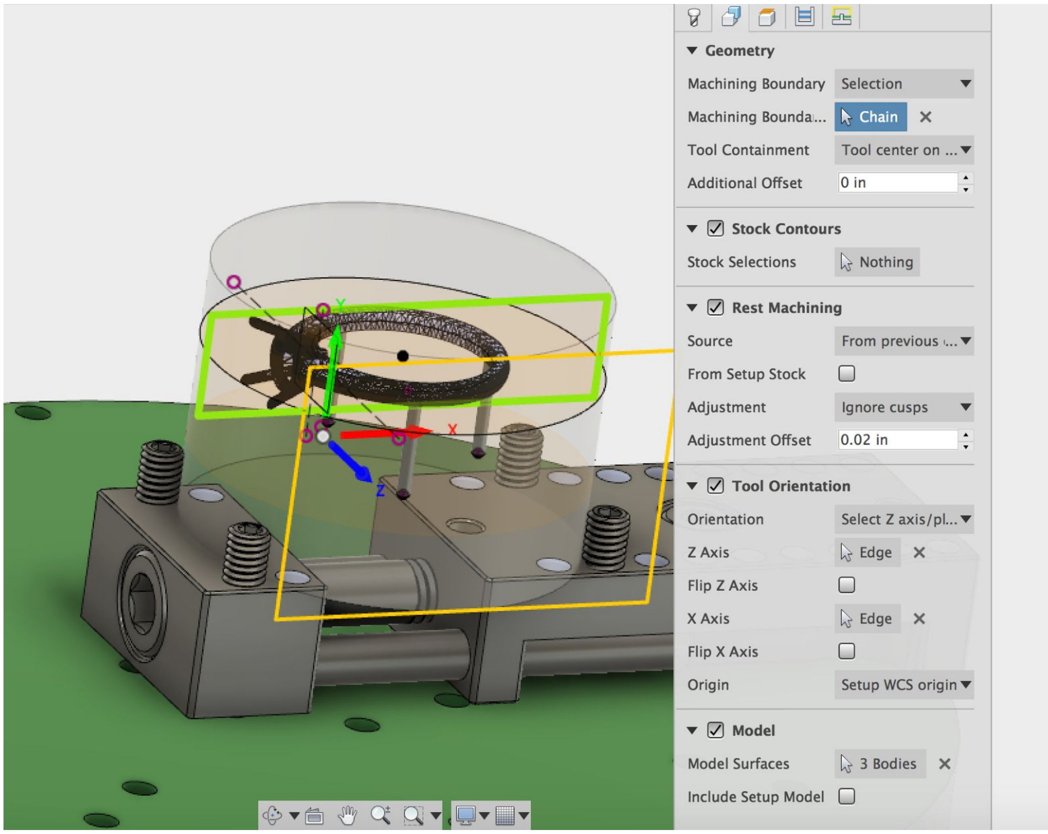


The geometry settings and the orientation of the WCS for this operation are shown below. The machining boundary is the perimeter of the space in which the tool is allowed to operate. Change the boundary type to “Selection” then choose the 1.6 x 0.5 inch rectangle that was sketched earlier. Set tool containment to “Tool Center on Boundary” and additional offset (the amount by which the tool is allowed to violate the perimeter) to 0 inches.

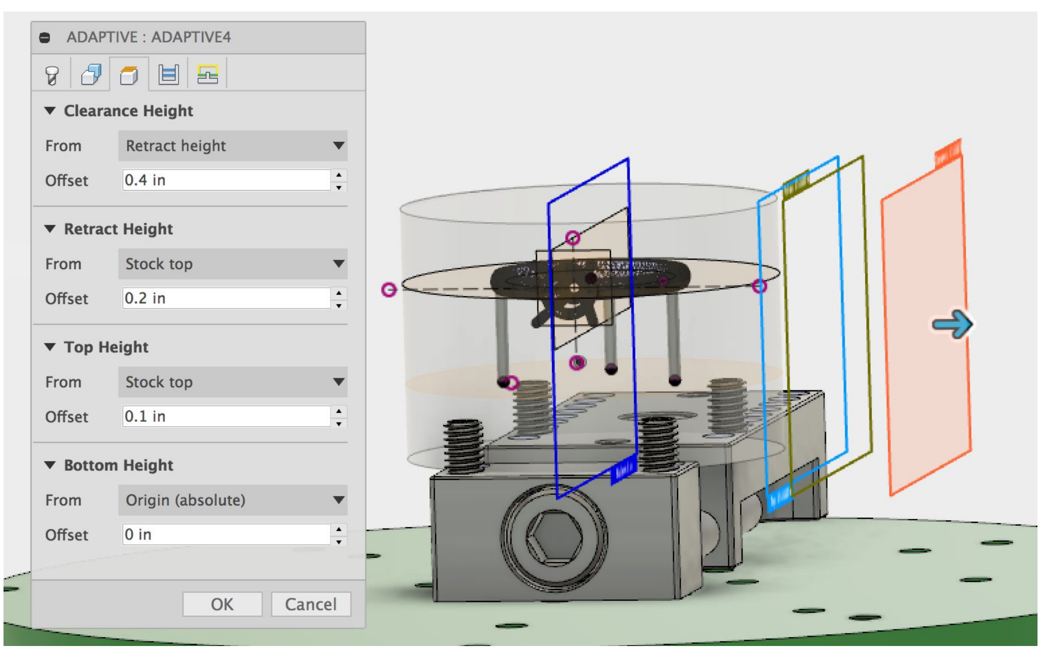
Click on the “Rest Machining” checkbox and choose “From Previous Operations”.

Orient the tool coordinate system as shown by selecting appropriate edges of the vise.

Select the ring, and the supports on the back and side that is being machined as the model bodies.

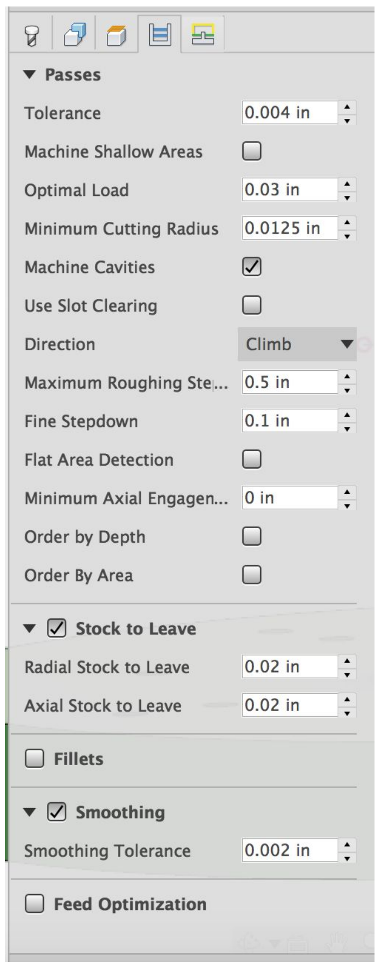


Set up the heights as shown. The bottom is coplanar with the midline of the ring. It is referenced from the world coordinate system origin.

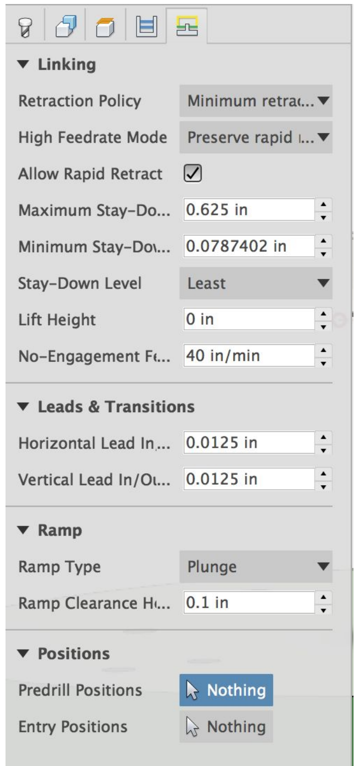


The tool pass settings are shown below. The optimal load is set to about 25% of the cutter diameter for this operation. This will cause the toolpaths to be less efficient at material removal but it will keep the load on the tool low so that the finish is better and prevent damage to the ring that is being machined. Since there will be cleanup passes later, 0.02 inches of stock will be left.

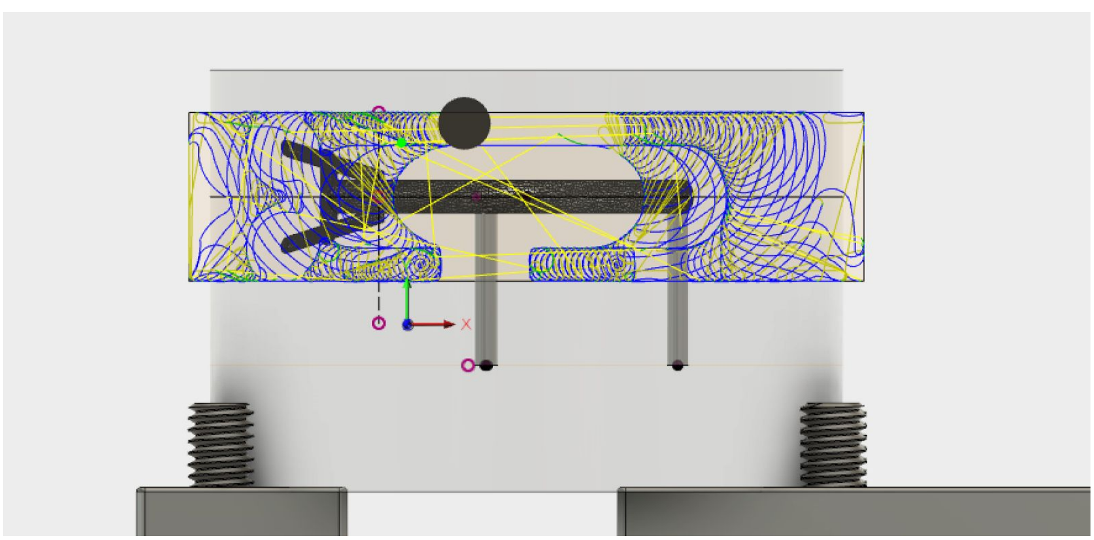
The max stepdown is set to the 0.5 inches (the length of the flutes on the cutter) because it allows the toolpaths to be more efficient. When the stock is repositioned the Fusion software is not able to calculate correctly where the stock has already been removed. Since the previous operation removed most of the material from the outside of the ring there is nothing to machine near the outside of the stock. In order to avoid wasteful toolpaths that only "cut air," the large stepdown is used to skip over most of the previously machined area as quickly as possible.



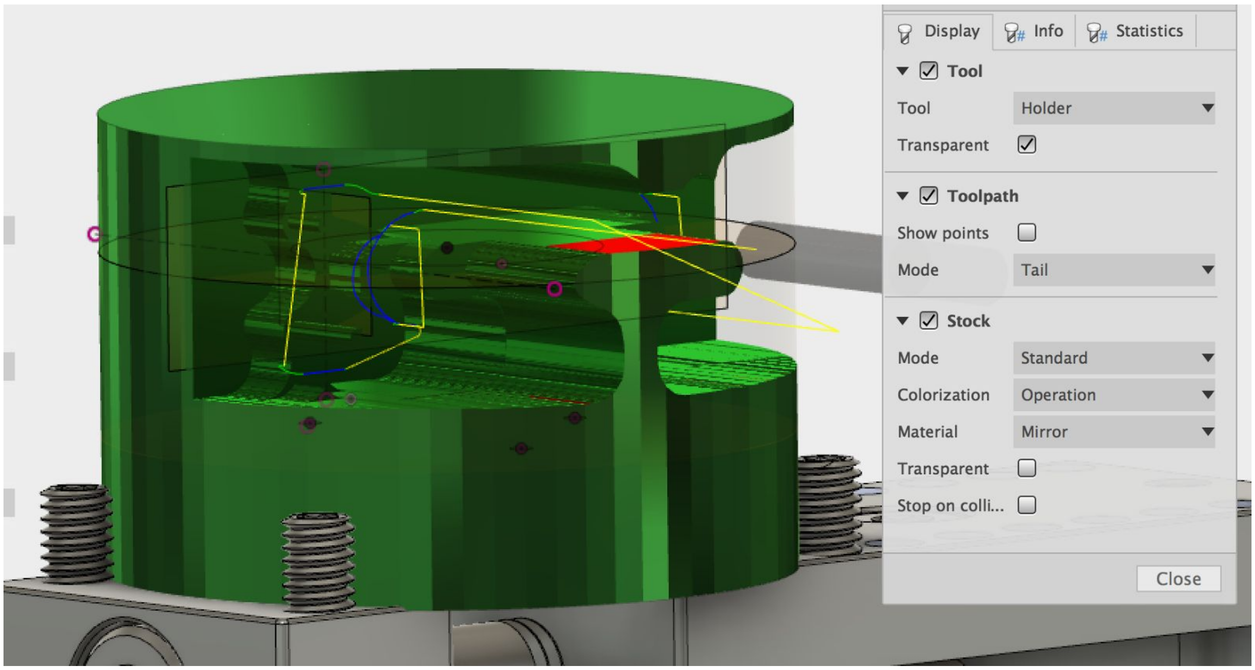
The settings for the linking are shown below. Most of these should be the default settings. The Ramp type may have to be changed from helical to plunge. The “No Engagement Feed Rate” will also need to be updated. These settings control how the cutter is allowed to move when it is not engaged.



The toolpath is shown below. It clears the material from around the sides and underneath the ring on one side. The next toolpath will do the same thing on the opposite side of the ring.



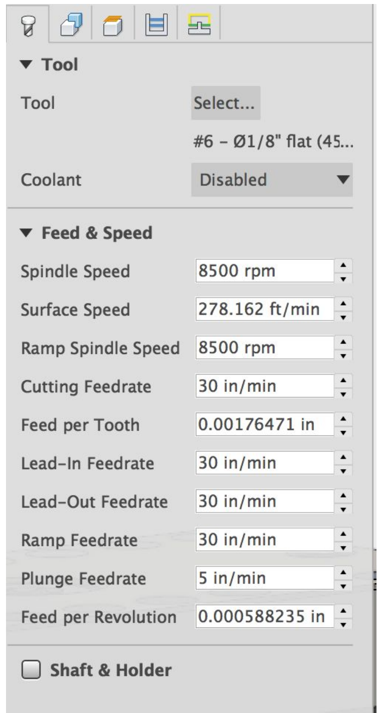
The simulated toolpath is shown below. Note that this simulation is only of this toolpath it does not include the previous operations. This will cause the Fusion to create collision alerts. These collisions are with stock that has already been removed. This can be confirmed by running the simulation on both operations. This will be done later. Take note of the collision reports but as long as they are in places where the stock is removed by the first operation they may be ignored for now.



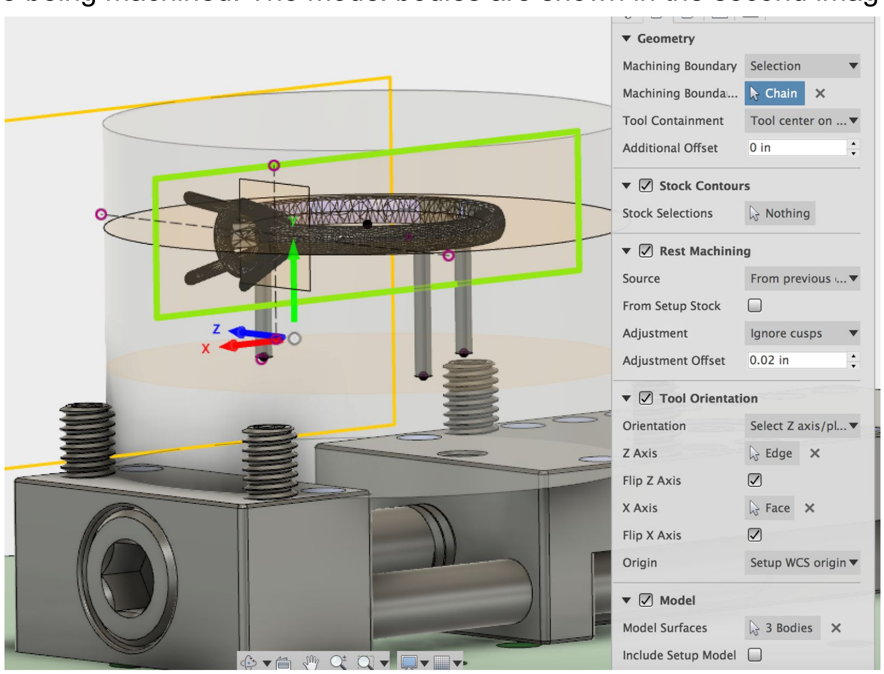
Operation 3: Adaptive Clear Side 2

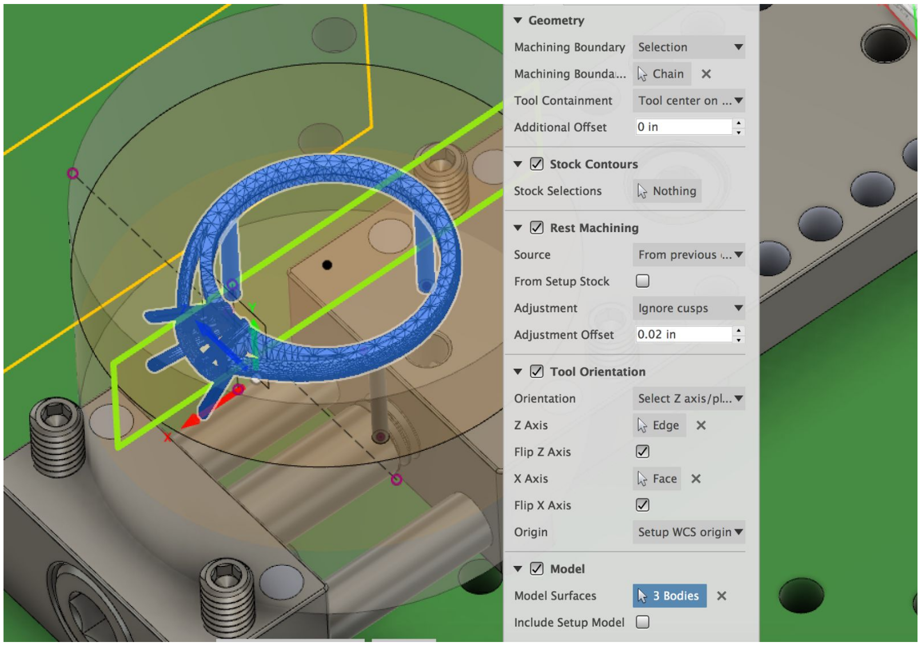
Operation 3 removes the material from the other side of the ring. It is the same as Operation 2, just 180º opposite.

The tool settings for the Operation 3 are shown in the image below. The same tool, number 6, (1\8 inch flat 45º helix) is used.

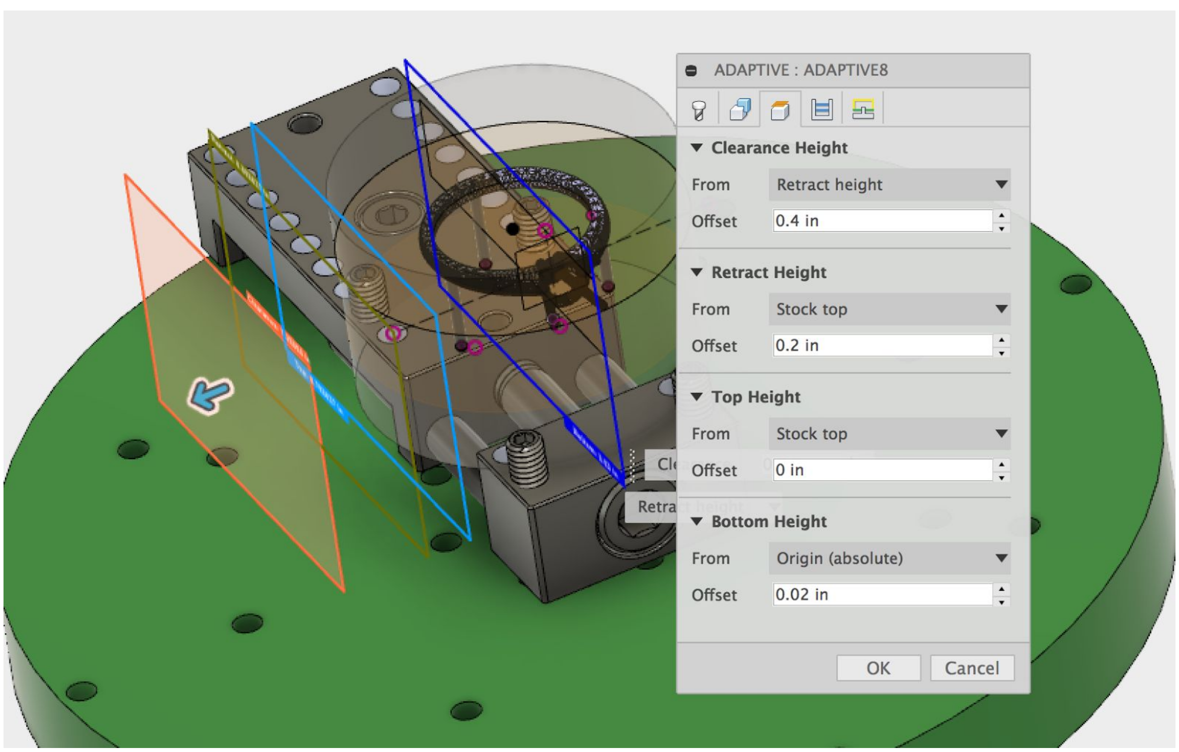


The geometry is setup as shown below. The rectangular sketch on the midplane of the ring is selected as the machining boundary. The “Tool Containment” is set to “Tool center on boundary.” The check box for “Rest Machining” is selected and the previous operations are used to calculated the stock left to remove. Set the “Tool Orientation” so that it matches what is shown by selecting appropriate edges on the vise as the X and Z axes. The “Model Surfaces” are the ring, the support in the middle, and the support on the side being machined. The model bodies are shown in the second image.



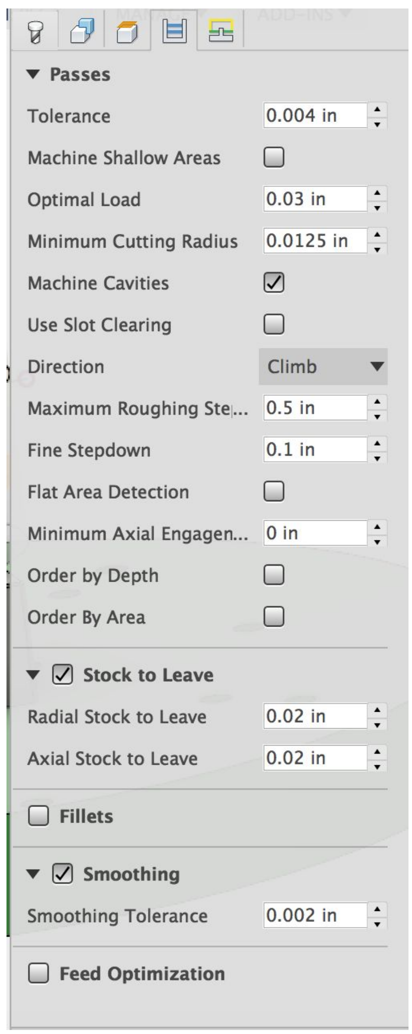


The heights are set as shown below. The bottom height is constrained to the model origin, the retract and top heights are constrained to the stock top surface, and the clearance height references the retract height.



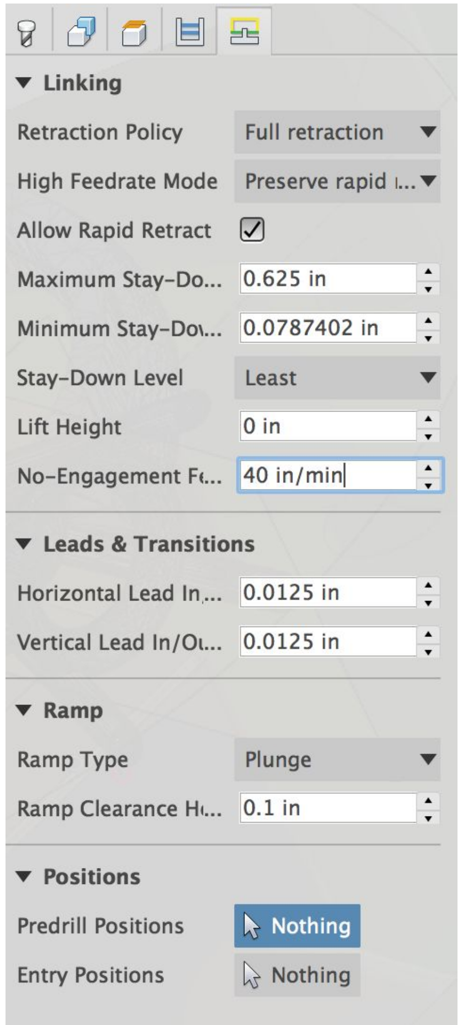
The toolpath settings are adjusted as shown below. The optimal load is set to about 25% of the cutter diameter for this operation. This will cause the toolpaths to be less efficient at material removal but it will keep the load on the tool low so that the finish is better and prevent damage to the ring that is being machined. Since there will be cleanup passes later, 0.02 inches of stock will be left.

The max stepdown is set to 0.5 inches (the length of the flutes on the cutter) because it allows the toolpaths to be more efficient. When the stock is repositioned the Fusion software is not able to calculate correctly where the stock has already been removed. Since the previous operation removed most of the material from the outside of the ring there is nothing to machine near the outside of the stock. In order to avoid wasteful toolpaths that only “cut air,” the large stepdown is used to skip over most of the previously machined area as quickly as possible.

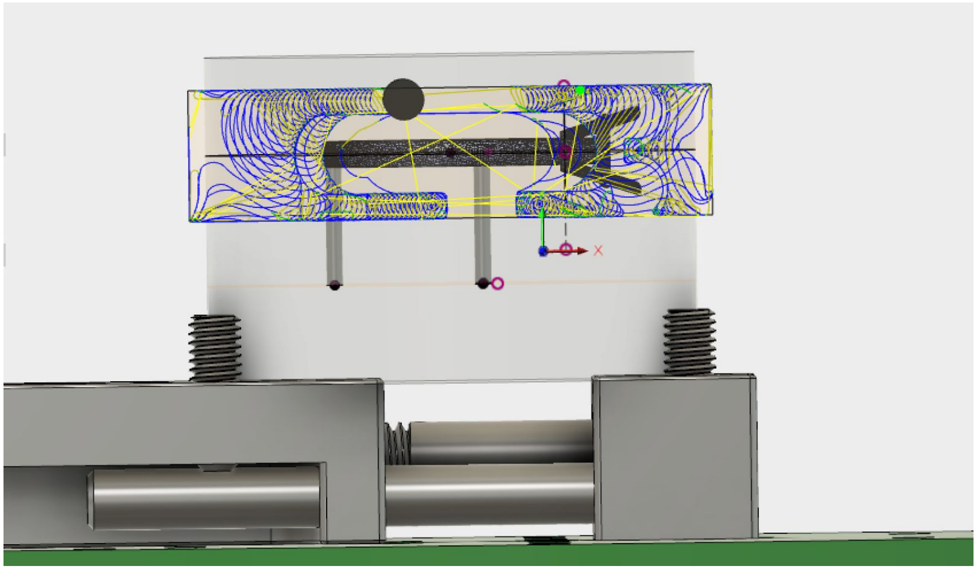


The settings for the linking are shown below. Most of these should be the default settings. The Ramp type may have to be changed from helical to plunge. The “No

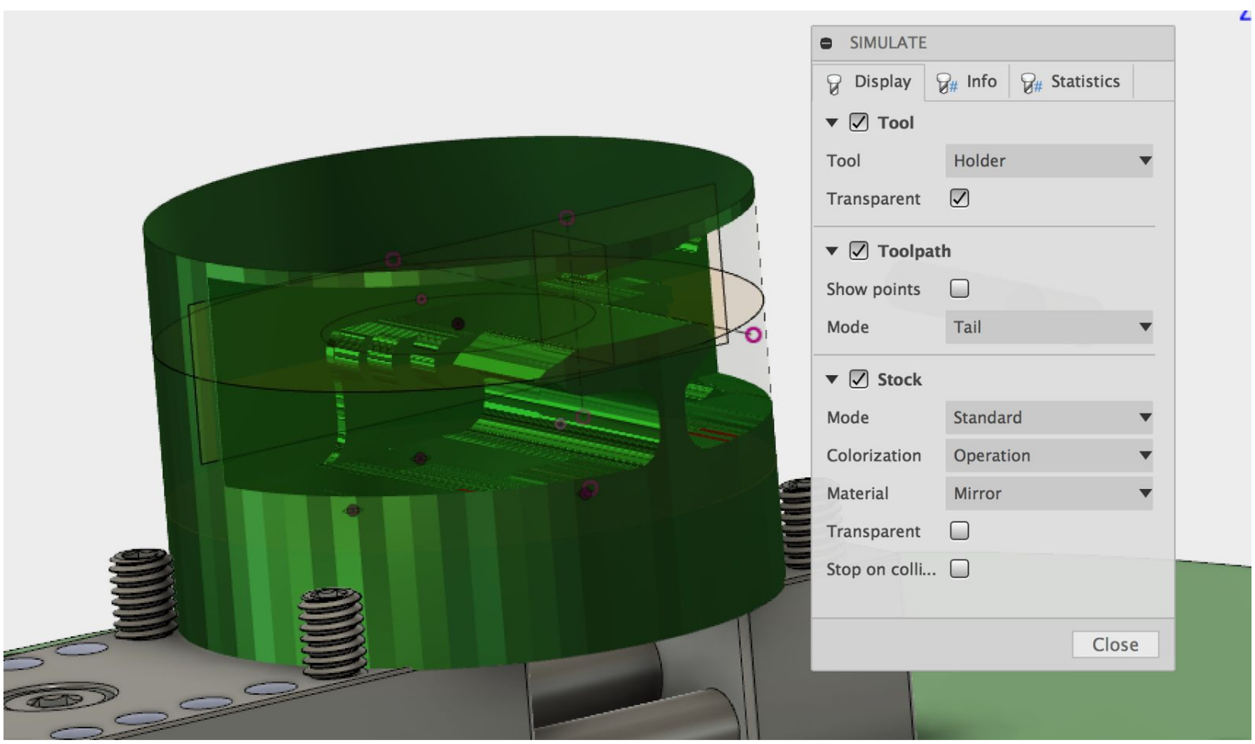
Engagement Feed Rate" will also need to be updated. These settings control how the cutter is allowed to move when it is not engaged.



The toolpath is shown below. It clears the material from around the sides and underneath the ring on one side. This is the same thing Operation 2 did on the other side of the ring.



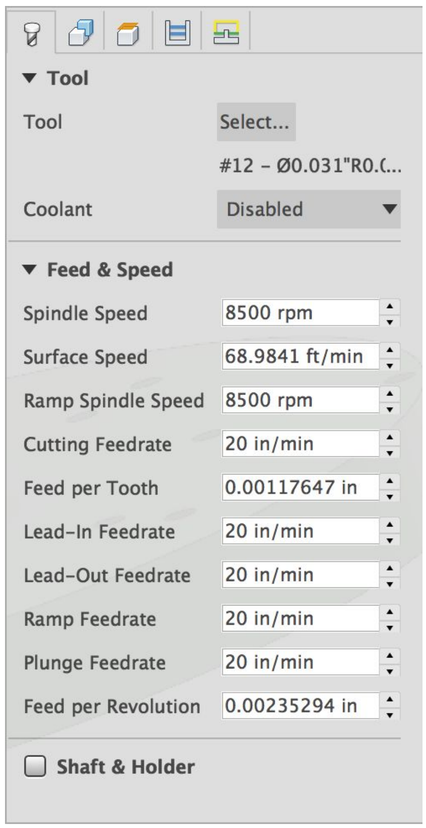
The simulated toolpath is shown below. This is accessed by right clicking on the toolpath in the browser, and selecting “Simulate”. Note that this simulation is only of this toolpath it does not include the previous operations. This will cause the Fusion to create collision alerts. These collisions are with stock that has already been removed. This can be confirmed by running the simulation on both operations. This will be done later. Take note of the collision reports but as long as they are in places where the stock is removed by the first operation they may be ignored for now.



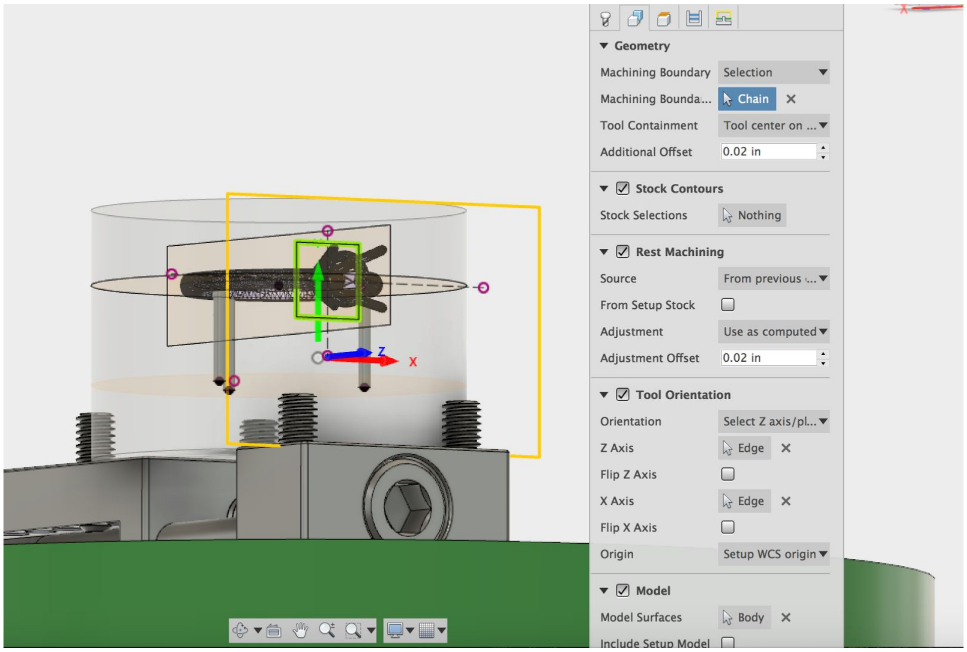
Operation 4: Adaptive Clear Setting

Operation 4 clears the bulk of the material away from the gem setting and prongs on the ring. This is done with an adaptive toolpath that keeps a constant tool load in order to remove the material efficiently. The tool is changed from the ones used in the previous operations to Pocket NC tool number 12 (1/32 inch 2º taper). This will allow access to smaller areas that that need to be milled.

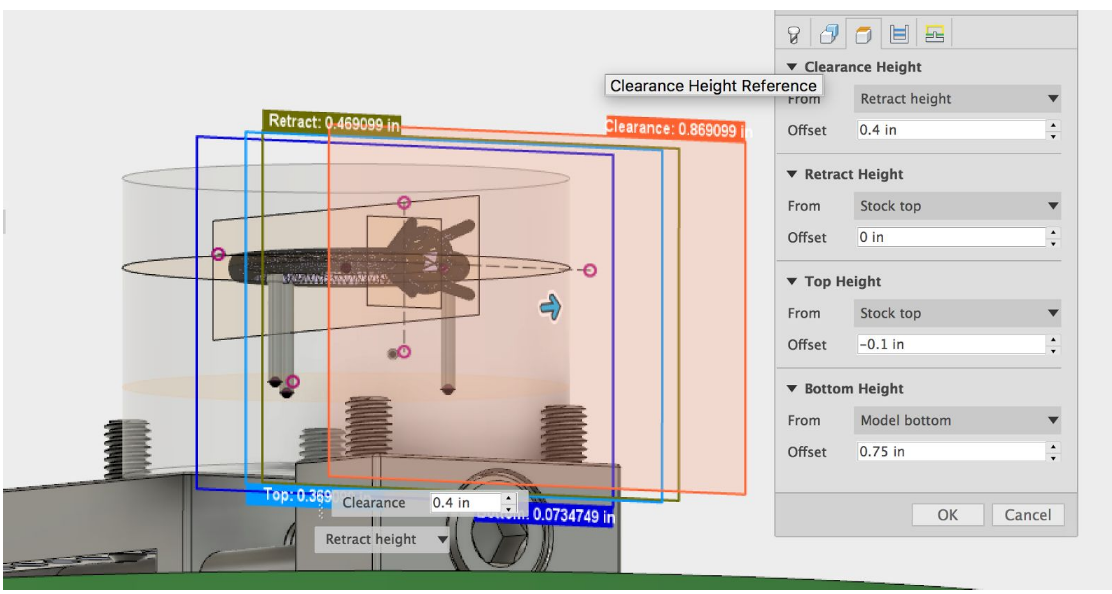
The settings for the tool are shown below.



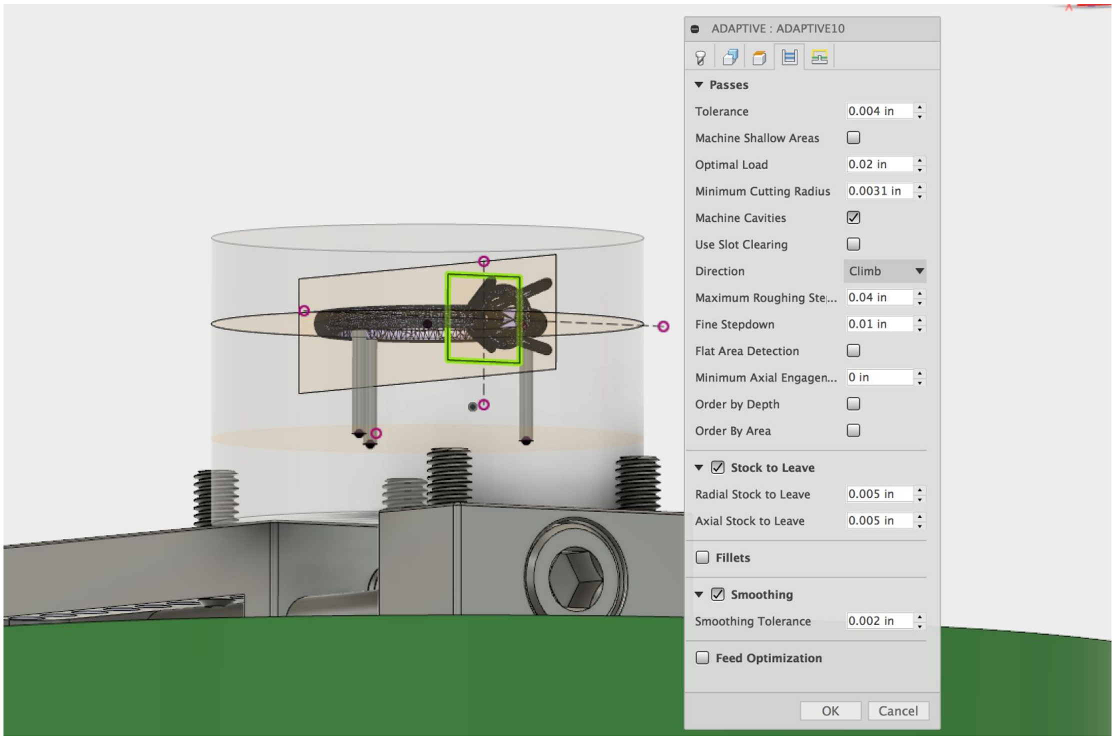
The toolpath geometry is set up as shown. The square that was sketched behind the gemstone setting is selected as the machining boundary. The “Tool Containment” is set as “Tool center on boundary.” The tool orientation is established as shown by selecting appropriate edges on the vise as the X and Z axes. The model surface selection is the ring. Select this by clicking the “model surface” “selection” tab and then selecting the body of the ring out of the browser.



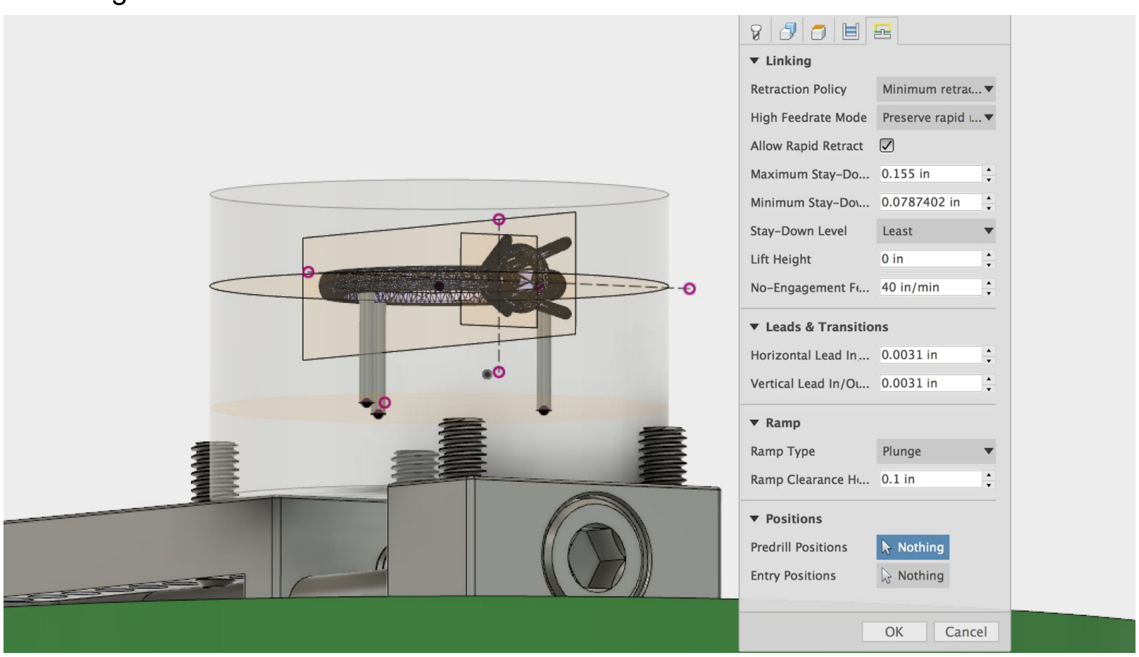
The heights are set up as shown below. The bottom plane is coincident with the back of the gem setting and top plane is 0.1 inches inside the stock surface. In most cases the top plane must lie outside of the stock surface, but since the material has been removed already it is more efficient to let the tool rapid feed closer to the cutting depth.



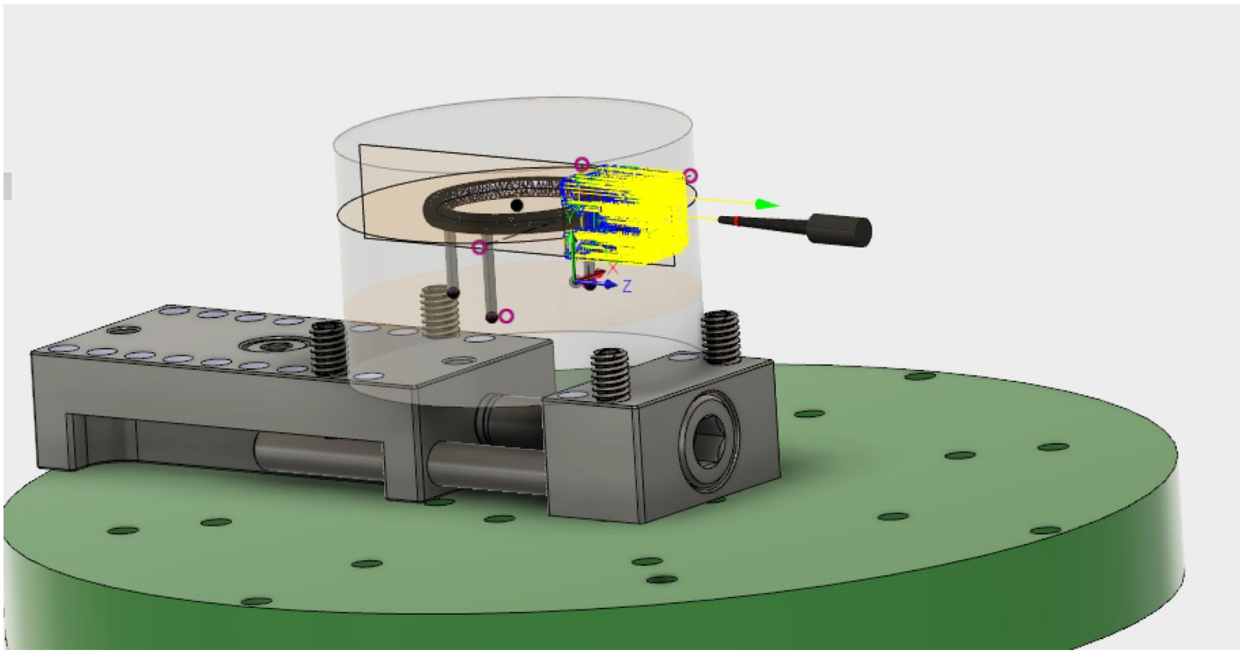
The tool-pass settings are adjusted to the values shown. The optimal load is set to about 60% of the tool diameter. The maximum and final step downs are smaller than the ones used in previous operations. The amount of stock left is only 0.005 inches instead of 0.02 inches as it has been with the previous several operations.

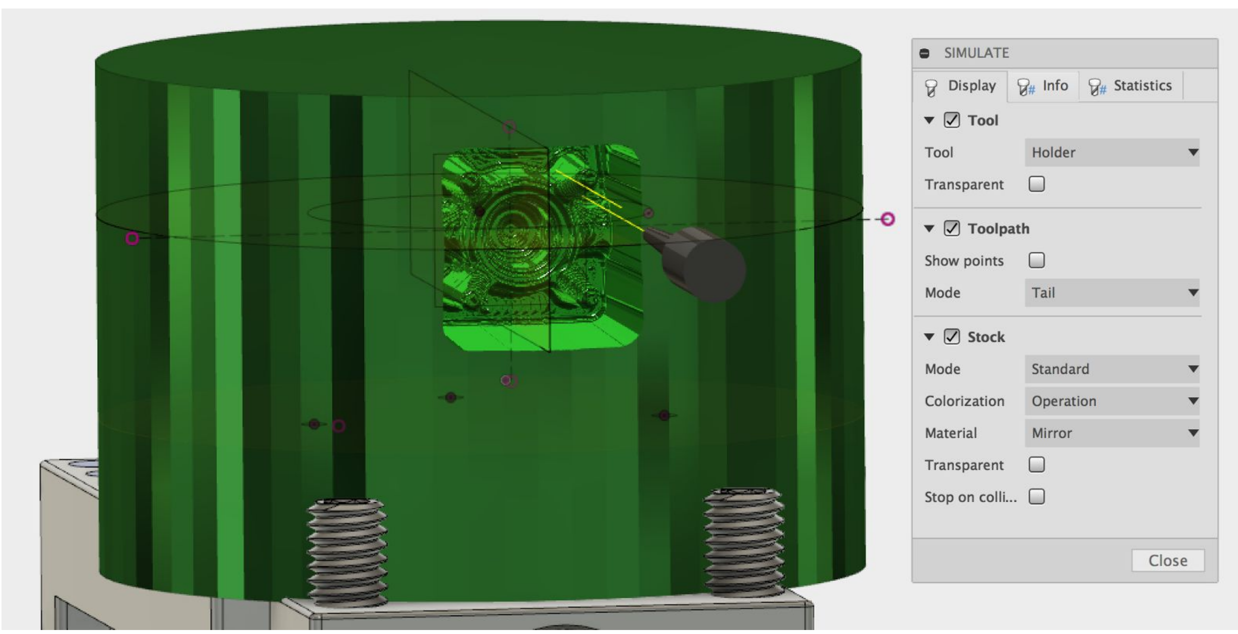


The linking settings are shown below. Most of these values should be the default settings.

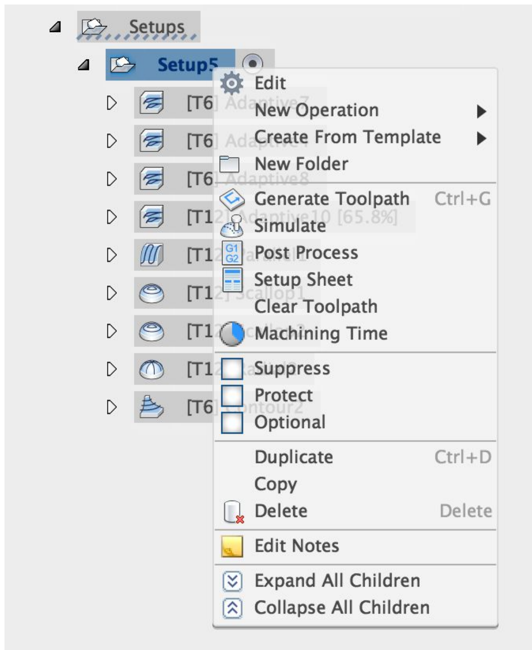


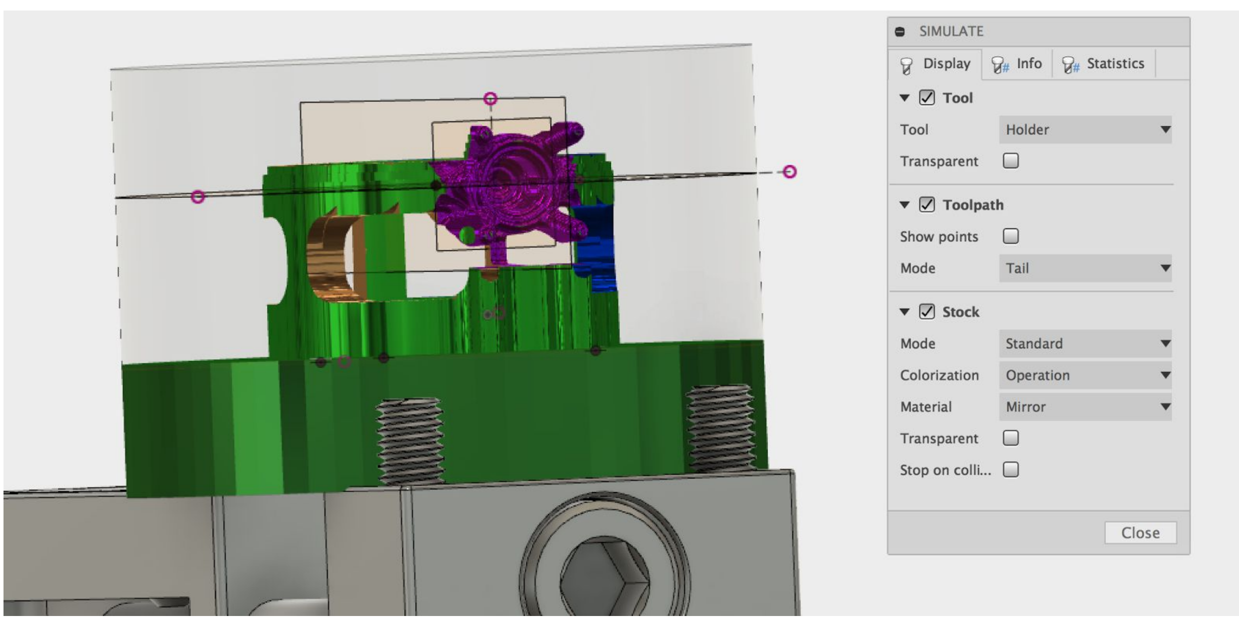
The toolpath and simulated toolpath are shown below. As with the previous two operations, some collisions may be predicted. Verify that the collisions are only in areas where the stock has already been removed by previous operations. If that is the case, the collisions may be ignored.





The first four operations may now be simulated if desired. Simulating the toolpaths is a good way to confirm that they are removing the material in the way that is expected and desired. In order to simulate all four of the tool paths right click on the setup, then select “Simulate.” When the simulation opens choose to display stock. There are several options for toolpath display. In the image below the toolpaths are hidden so that the workpiece can be seen.



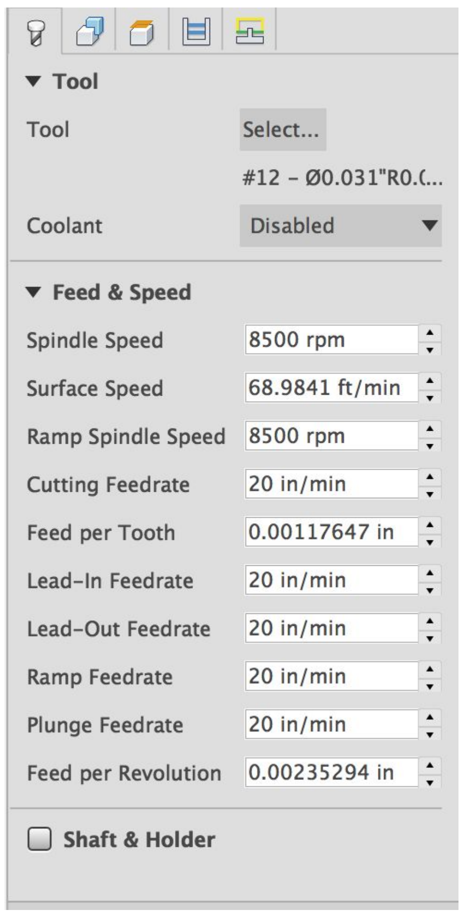


Operation 5: Parallel Clear Top

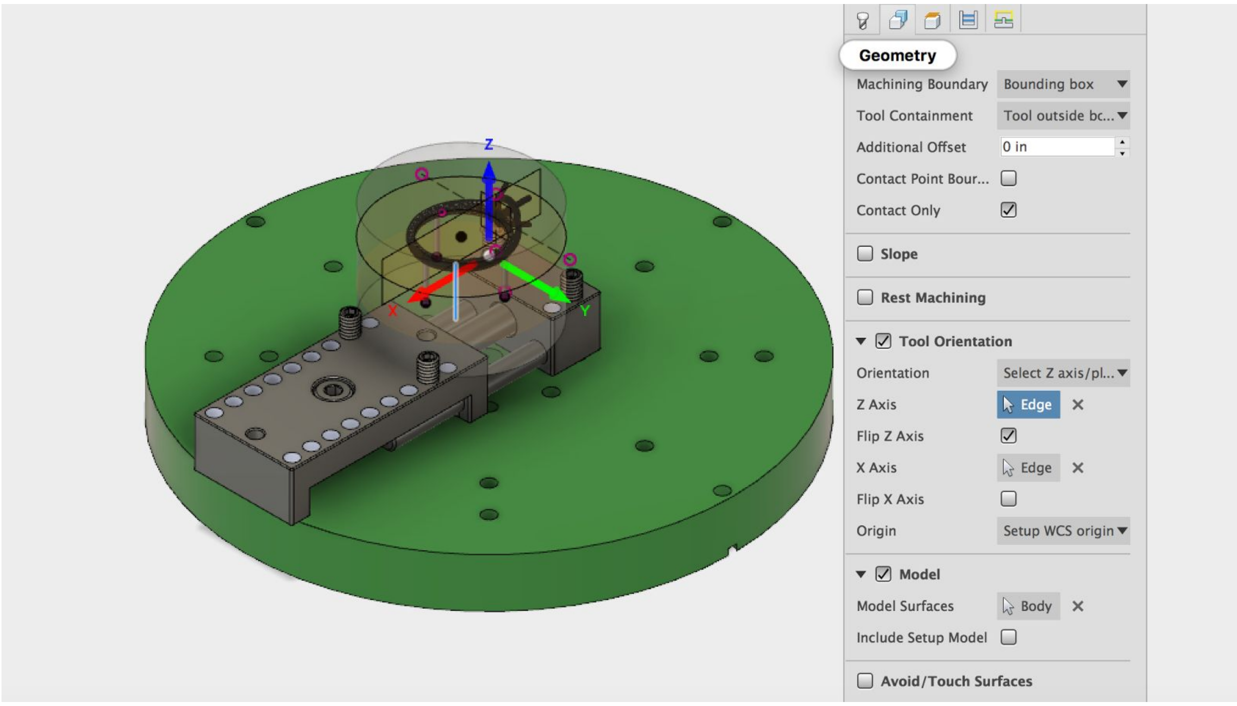
Operation 5 will be used to clear the stock that was left behind by the adaptive clearing on the top of the ring. The type of toolpath that will be used is parallel. This will cut using straight lines that are parallel to the Y axis of the machine and perpendicular to the X axis. The contour of the part will be followed in the Z axis.

This toolpath is chosen in order to minimize the force being placed onto the prongs of the setting as well as to create a good surface finish on the top of the part.

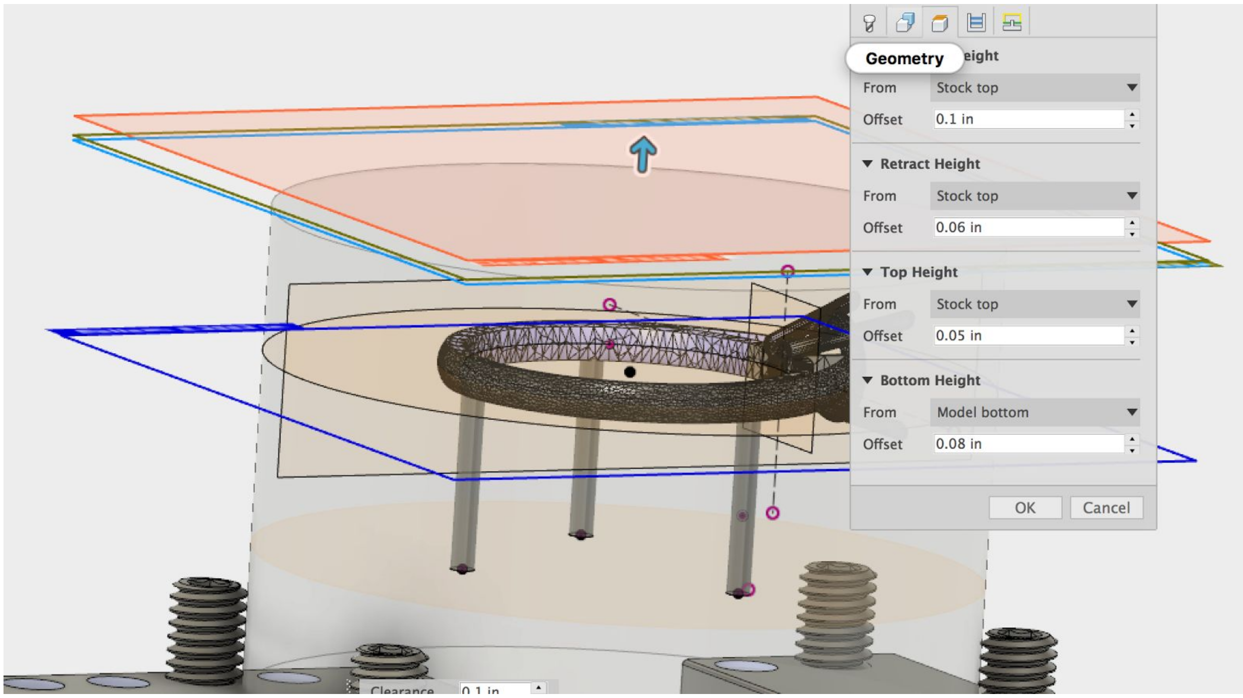
The same tool used in the previous operation, 1/32 inch 2º taper (Tool number 12) is used for this operation. The tool settings are shown in the image below.

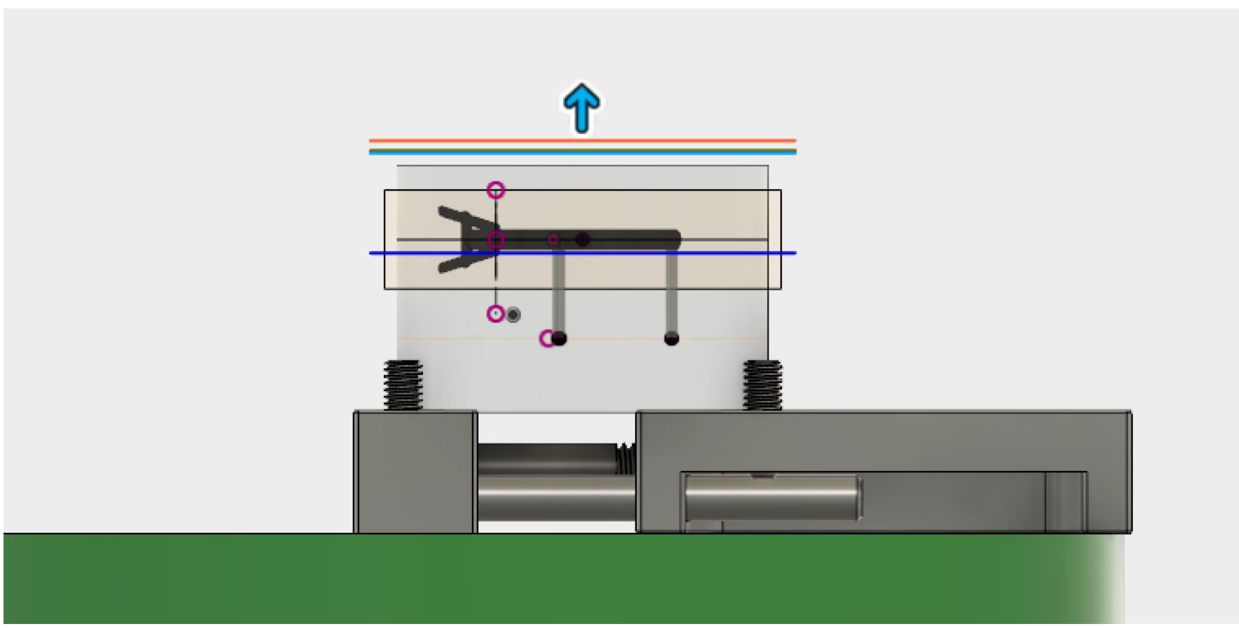


The geometry settings are the next thing to be adjusted. The “Machining Boundary” is set as a “Bounding Box.” This allows Fusion to auto-generate a box that bounds the model geometry being machined. The “Tool Containment” method is selected as “Tool outside box”. The “Contact Only” check box is selected in order to allow the toolpaths to skip over the hole in the center of the ring. The tool orientation is established by selecting edges of the vise as the X and Z axes. The ring is chosen from the browser as the model surface to be machined.



The heights are set as shown below. The bottom height is set so that it is just below the bottom of the ring as shown in the second image below. Due to the limited length of the tool being used (0.75” overall) the the toolholder will collide with the part if the bottom plane is more than the tool stickout below the highest point on the part.

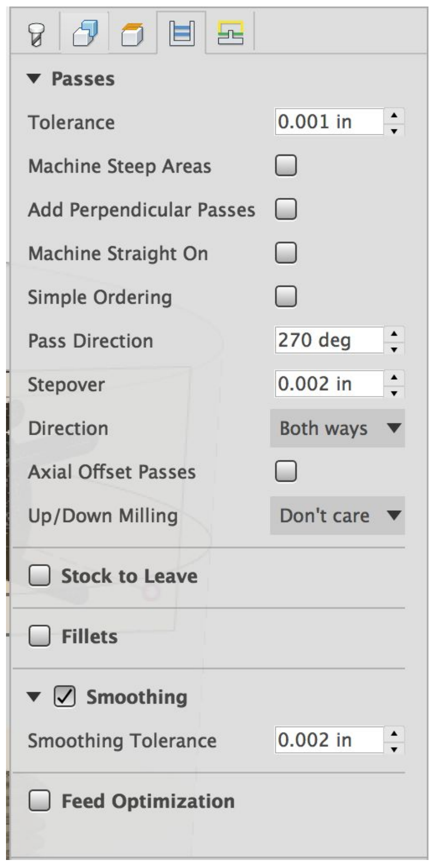




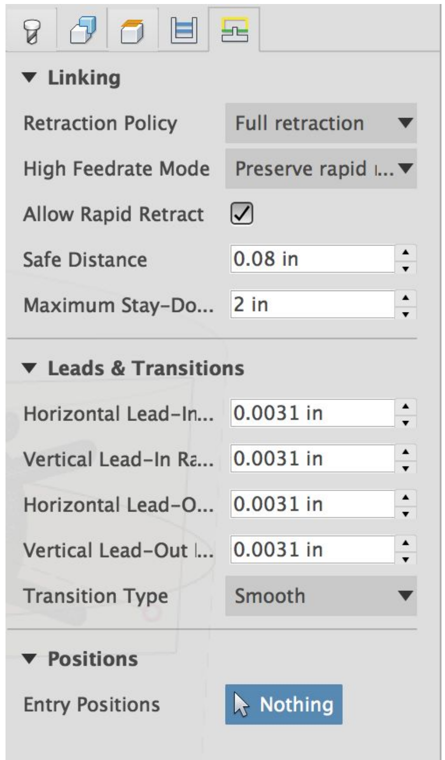
The tool pass settings are shown below. Note that the tolerance is tighter than it has been for previous operations. The pass direction controls the way which the ring is cut. The settings shown will start at the tips of the prongs and work back toward the ring. This is desirable because it means there is more material on the prongs to support the cutting forces when the tips are being milled. Cutting the opposite direction runs a greater risk of breaking the the prongs off.

The stepover is very small in order to produce a good surface finish and the level of detail desired. If the finished product is rougher than desired this value may be further reduced at the expense of a larger file size and longer machining time.

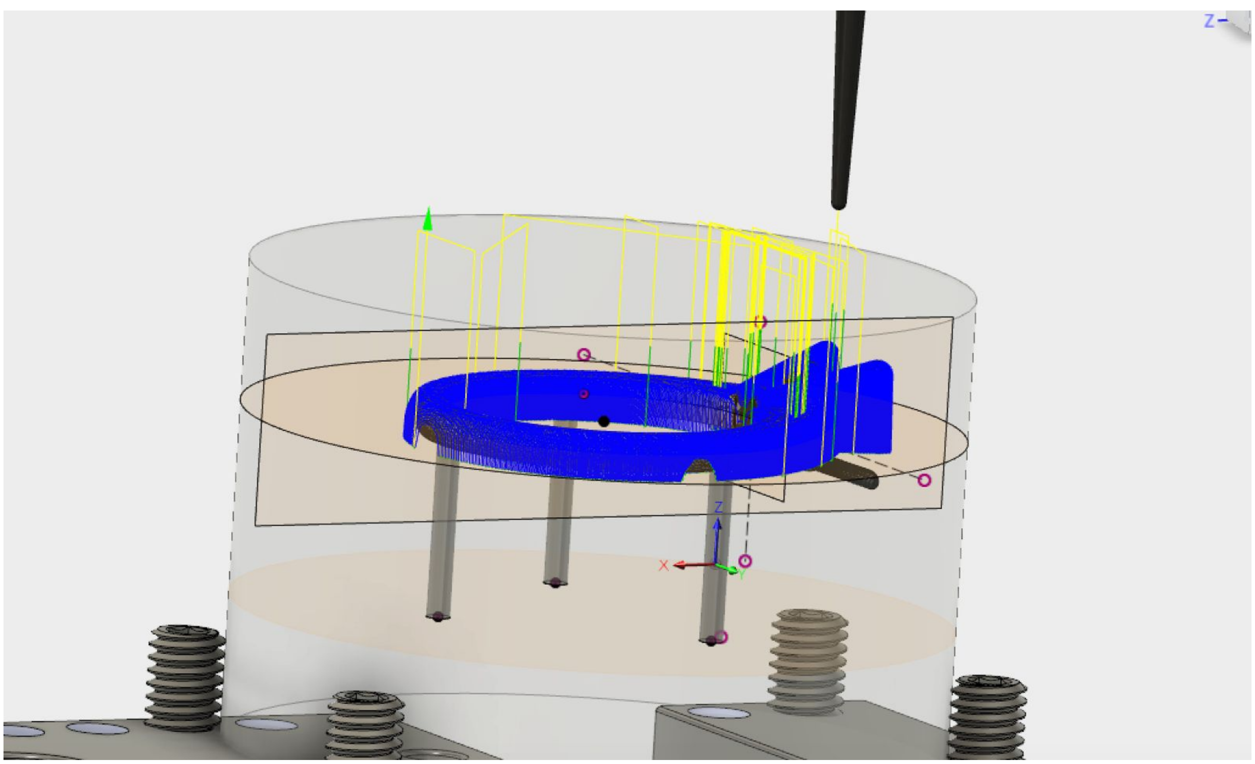
The smoothing tolerance is used to allow Fusion to ignore some of the irregularities on the surface of the ring due to it being a multi-surface body. Because the ring is made up of about 11,000 triangular facets, its surface is far from smooth. The smoothing tolerance allows the tool to follow a path with a specified minimum radius instead of having to follow every contour on the surface.

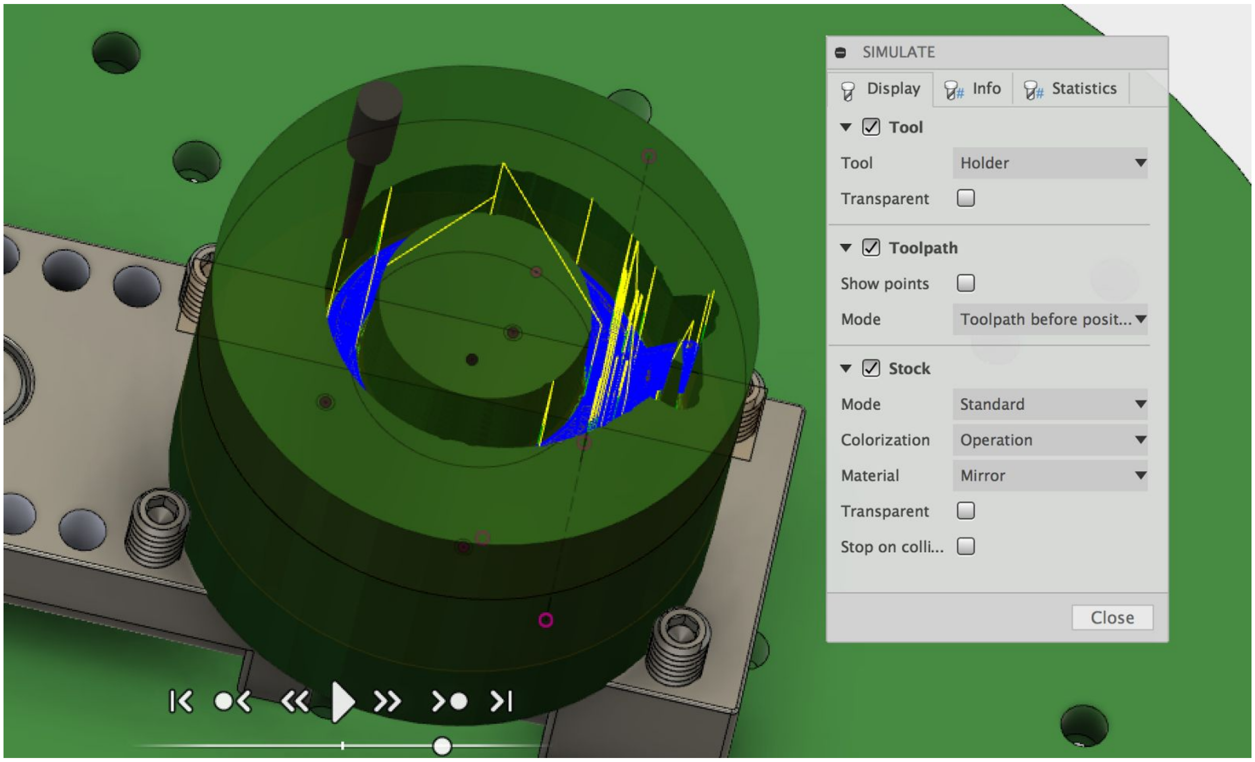


The linking settings are the final settings to adjust before generating and simulating the toolpath. All of the settings below should be the defaults.

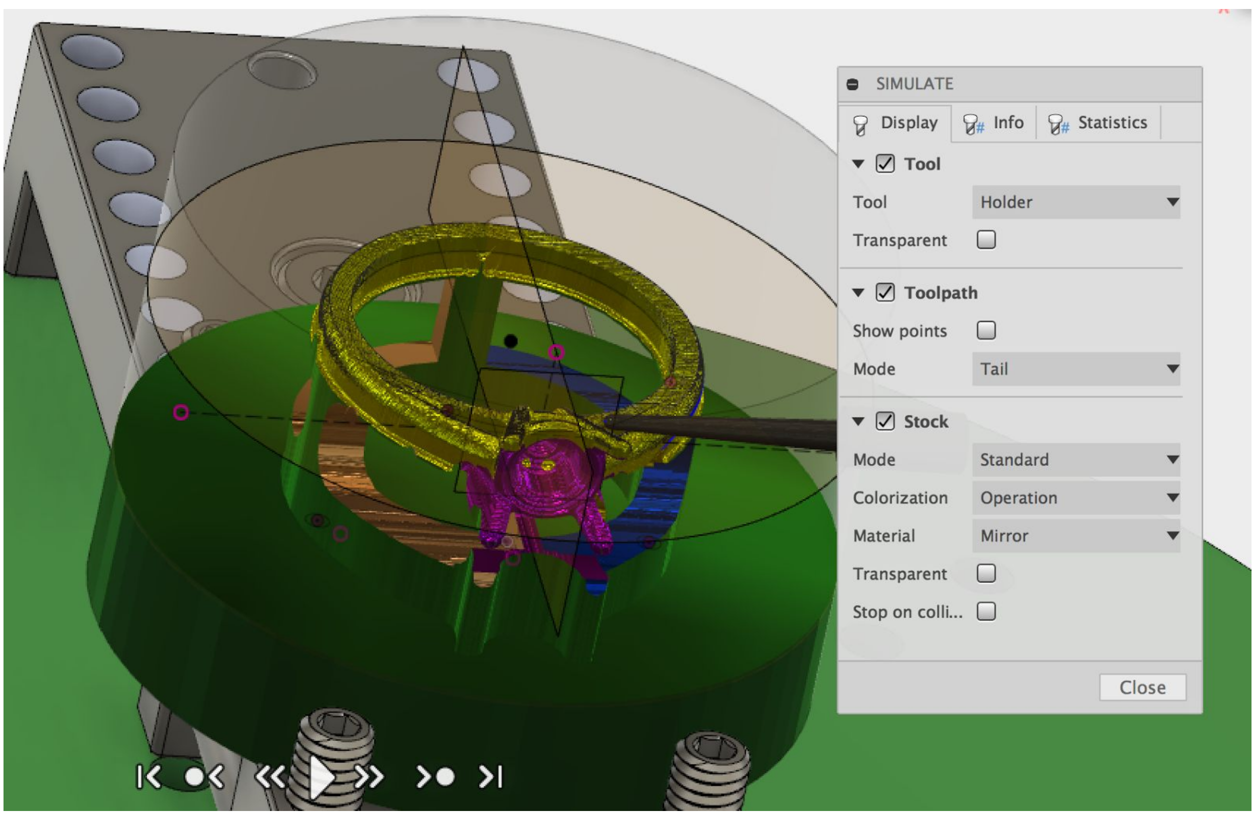


The parallel toolpath is shown below. The simulation of just the parallel toolpath is shown in the second image. The third image is the simulation of the first five toolpaths.





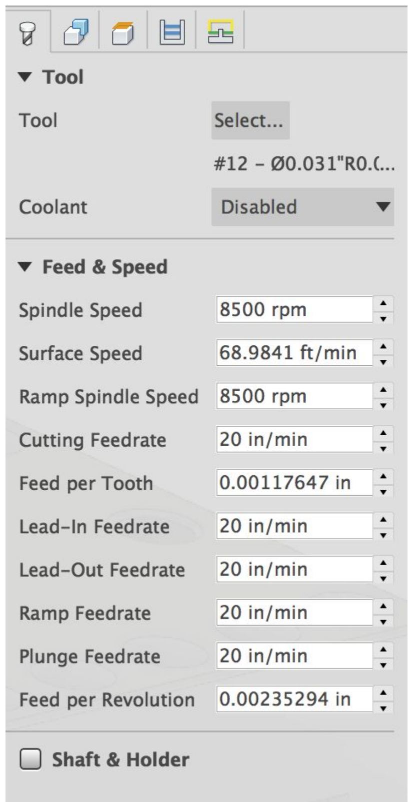
SIMULATE
Display
Info
Statistics
Tool
Tool
Holder
Transparent
Toolpath
Show points
Mode
Toolpath before posit...
Stock
Mode
Standard
Colorization
Operation
Material
Mirror
Transparent
Stop on colli...
Close



SIMULATE
Display
Info
Statistics
Tool
Tool
Holder
Transparent
Toolpath
Show points
Mode
Tail
Stock
Mode
Standard
Colorization
Operation
Material
Mirror
Transparent
Stop on colli...
Close

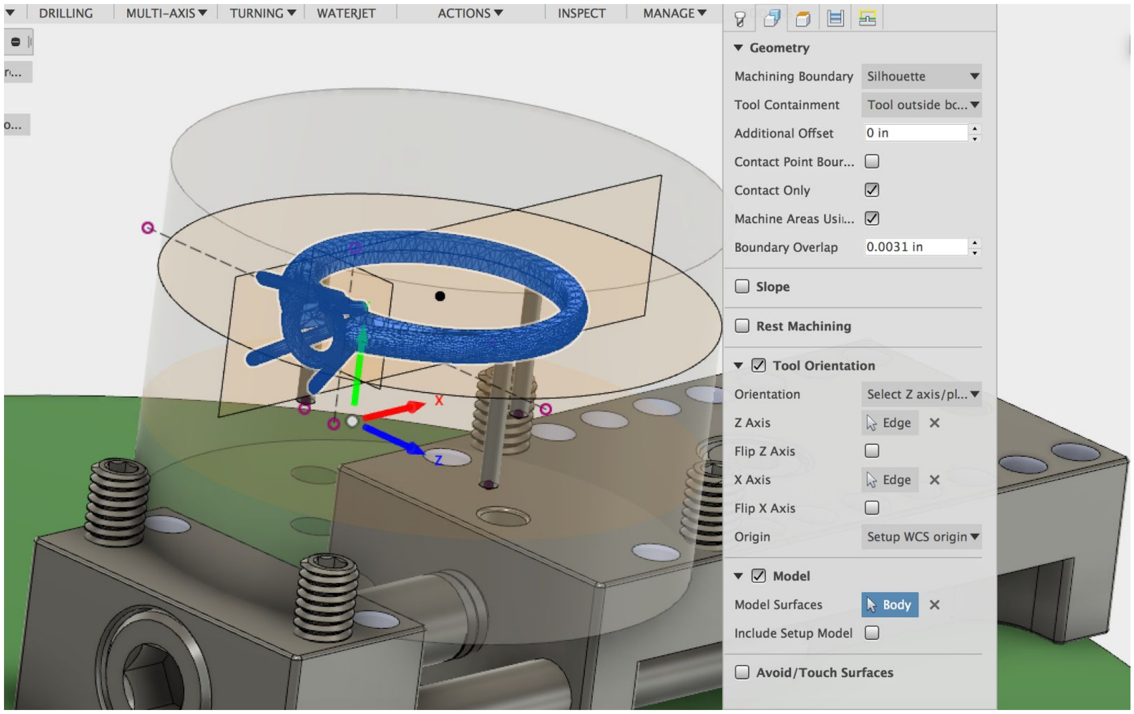
Operation 6: Scallop Side 1

Operation 6 is a scalloping path that removes the material left on the sides of the ring which was roughed by the adaptive clearing. The scalloping method creates toolpaths that are a constant distance apart. This allows the a good surface finish to be created on on complex curves and in areas that cannot be easily reached by other types of toolpaths like contour or parallel. Once again, tool number 12 is used. The tool settings are shown below.

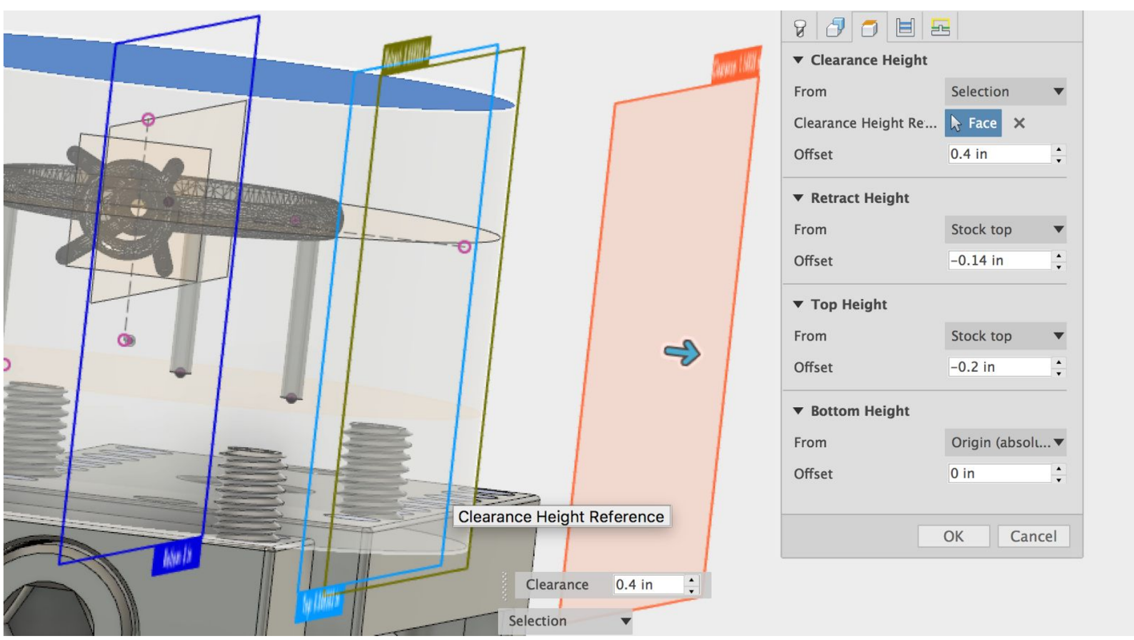


The geometry settings are shown below. The “Machining Boundary” chosen is “Silhouette.” This allows Fusion to calculate the boundary based on the silhouette of the part. The “contact only” and “machine areas using boundaries” buttons are selected. This will keep the toolpaths that are generated in contact with the part.

Set the tool orientation so that it matches the image below. Select appropriate edges on the vise as the X and Z axes in order to orientate it. Choose the ring out of the browser as the model surface to be machined.



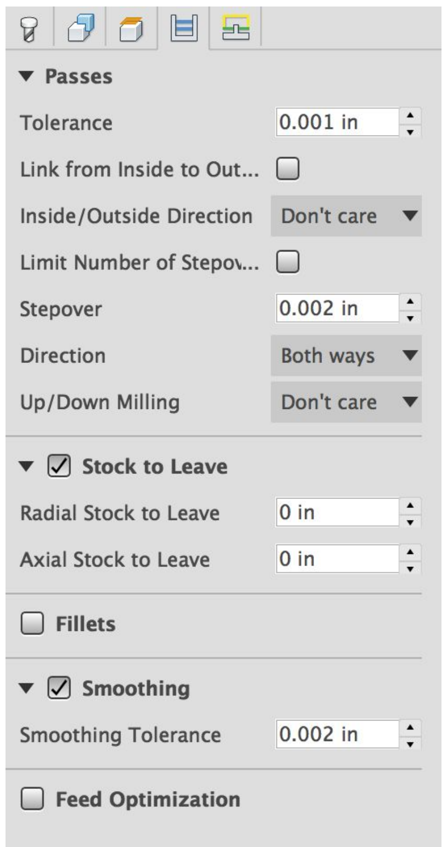
The heights are set as shown. Both the top height and the retract height are set to be within the stock envelope because the material in that area has been removed so collisions will not take place.



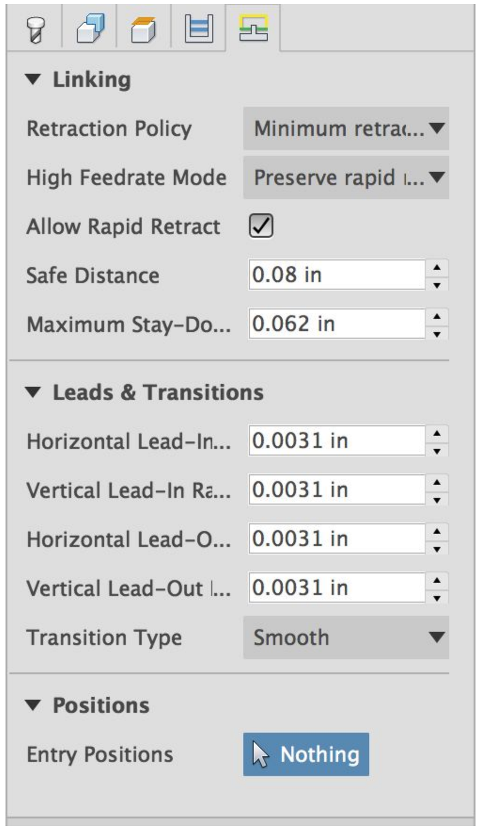
The tool pass settings are shown below. Note that the tolerance is the same as the one on previous operation and the tighter than the ones used for the clearing operations.

The stepover is very small in order to produce a good surface finish and the level of detail desired. If the finished product is rougher than desired this value may be further reduced at the expense of a larger file size and longer machining time.

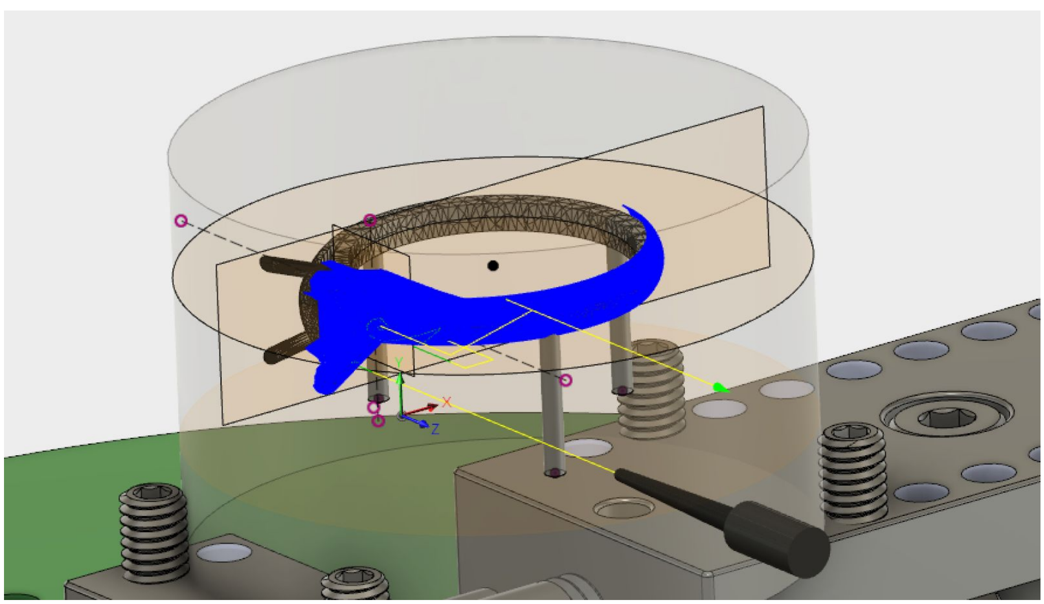
The smoothing tolerance is used to allow Fusion to ignore some of the irregularities on the surface of the ring due to it being a multi-surface body. Because the ring is made up of about 11,000 triangular facets, its surface is far from smooth. The smoothing tolerance allows the tool to follow a path with a specified minimum radius instead of having to follow every contour on the surface.

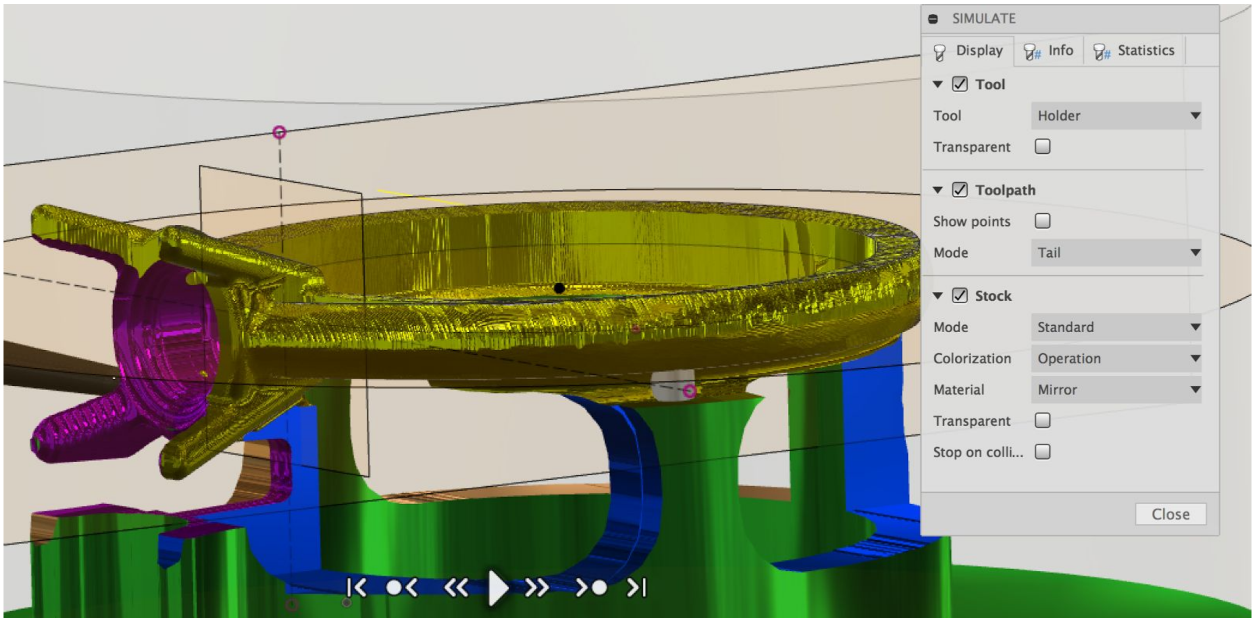


The linking settings are the final settings to adjust before generating and simulating the toolpath. All of the settings below should be the defaults except for changing the retraction policy to “Minimum retraction” instead of “Full Retraction.”



The toolpath and simulation are shown in the two images below.

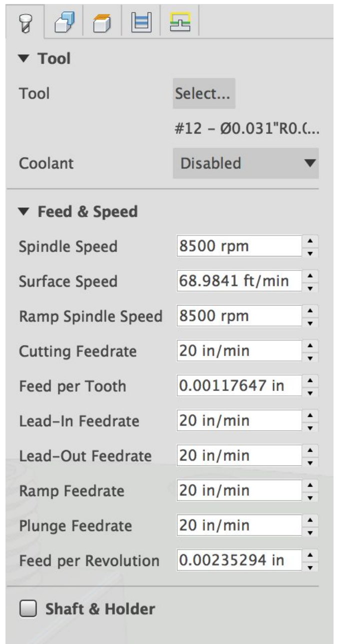




Operation 7: Scallop Side 2

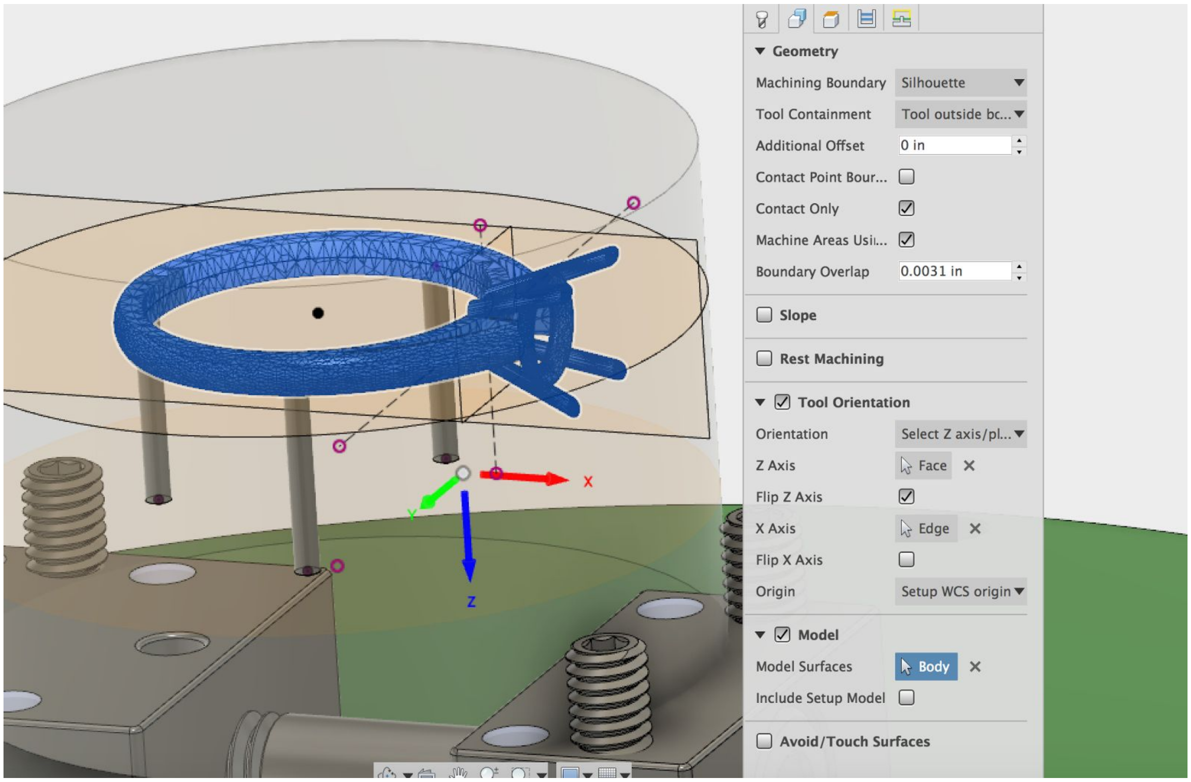
Operation 7 is a mirror of Operation 6to clean up the other side of the ring. Again it uses a scalloping toolpath to remove the last 0.02 inches of material. The goal is to clean up the surface and leave a good finish.

The tool settings are shown below. Pocket NC tool number 12 is used for this toolpath.

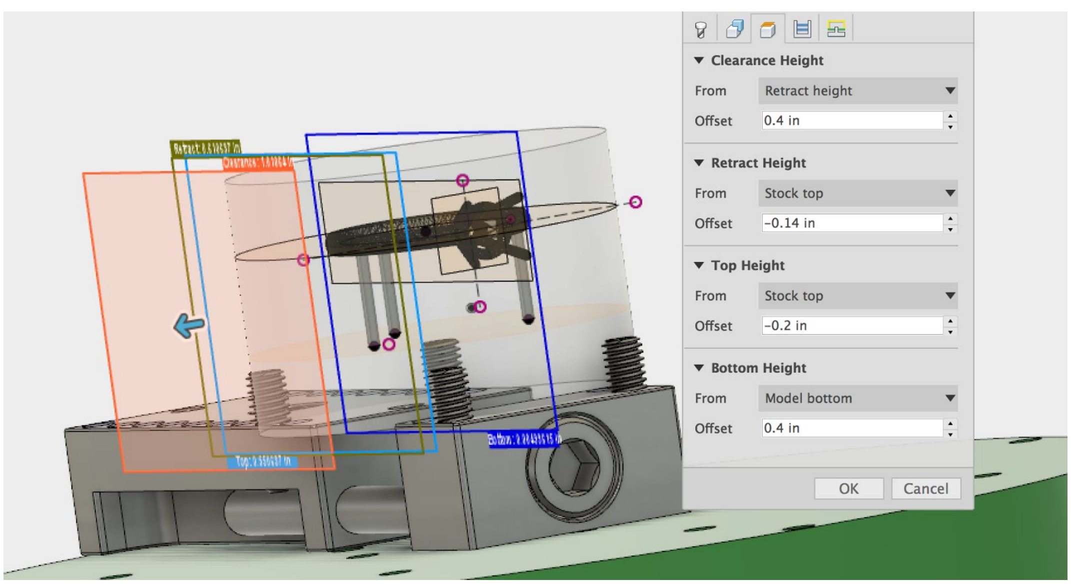


The geometry settings are shown in the figure below. The “Machining Boundary” chosen is “Silhouette”. This allows Fusion to calculate the boundary based on the silhouette of the part. The “contact only” and “machine areas using boundaries” buttons are selected. This will keep the toolpaths that are generated in contact with the part.

Set the tool orientation so that it matches the image below. Select appropriate edges on the vise as the X and Z axes in order to orientate it. Choose the ring out of the browser as the model surface to be machined.

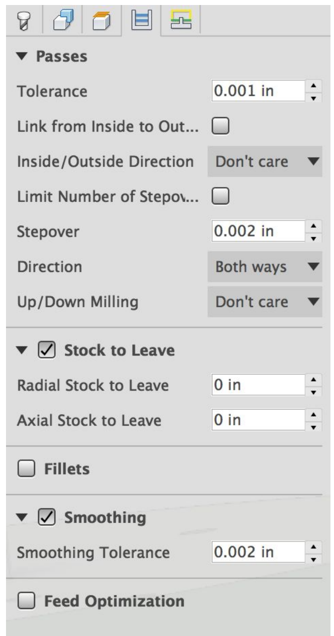


The heights are set as shown below. Both the top height and the retract height are set to be within the stock envelope because the material in that area has been removed so collisions will not take place.

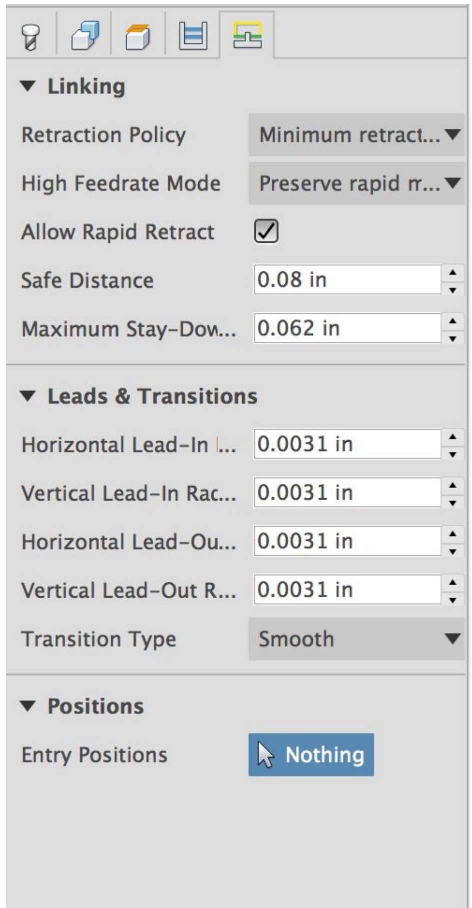


The tool pass settings are shown below. Note that the tolerance is the same as the one on previous two operations and the tighter than the ones used for the clearing operations. The stepover is very small in order to produce a good surface finish and the level of detail desired. If the finished product is rougher than desired this value may be further reduced at the expense of a larger file size and longer machining time.

The smoothing tolerance is used to allow Fusion to ignore some of the irregularities on the surface of the ring due to it being a multi-surface body. Because the ring is made up of about 11,000 triangular facets, its surface is far from smooth. The smoothing tolerance allows the tool to follow a path with a specified minimum radius instead of having to follow every contour on the surface.



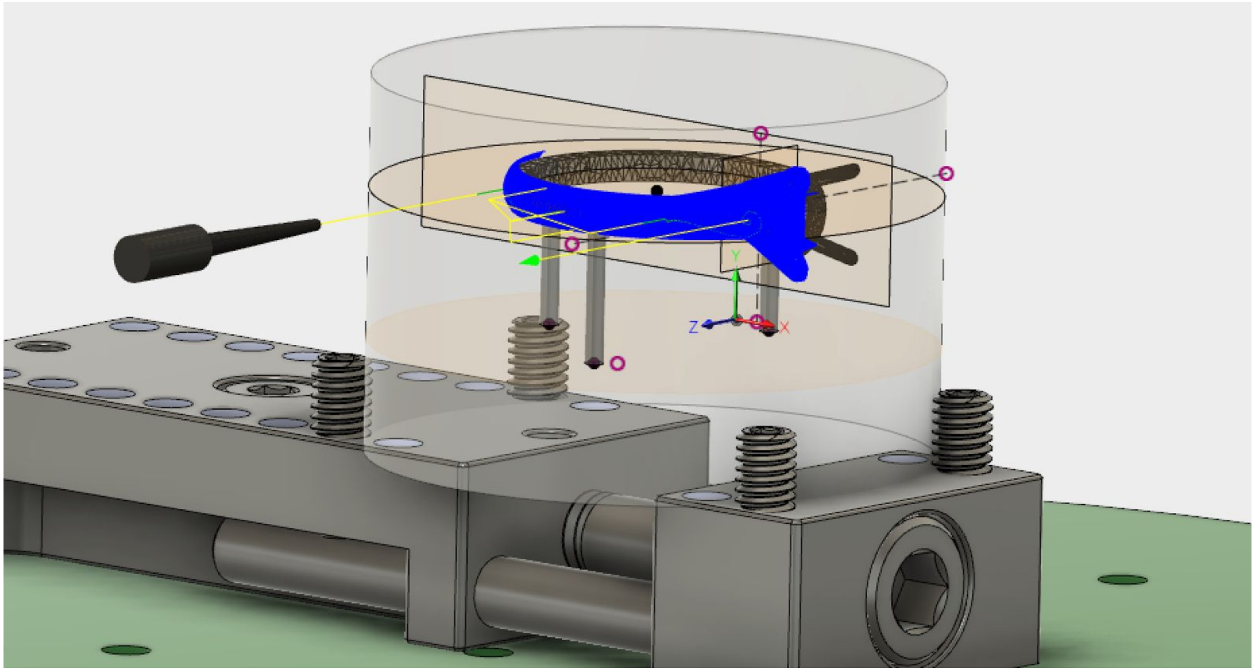
The linking settings are the final settings to adjust before generating and simulating the toolpath. All of the settings below should be the defaults except for changing the retraction policy to “Minimum retraction” instead of “Full Retraction.”

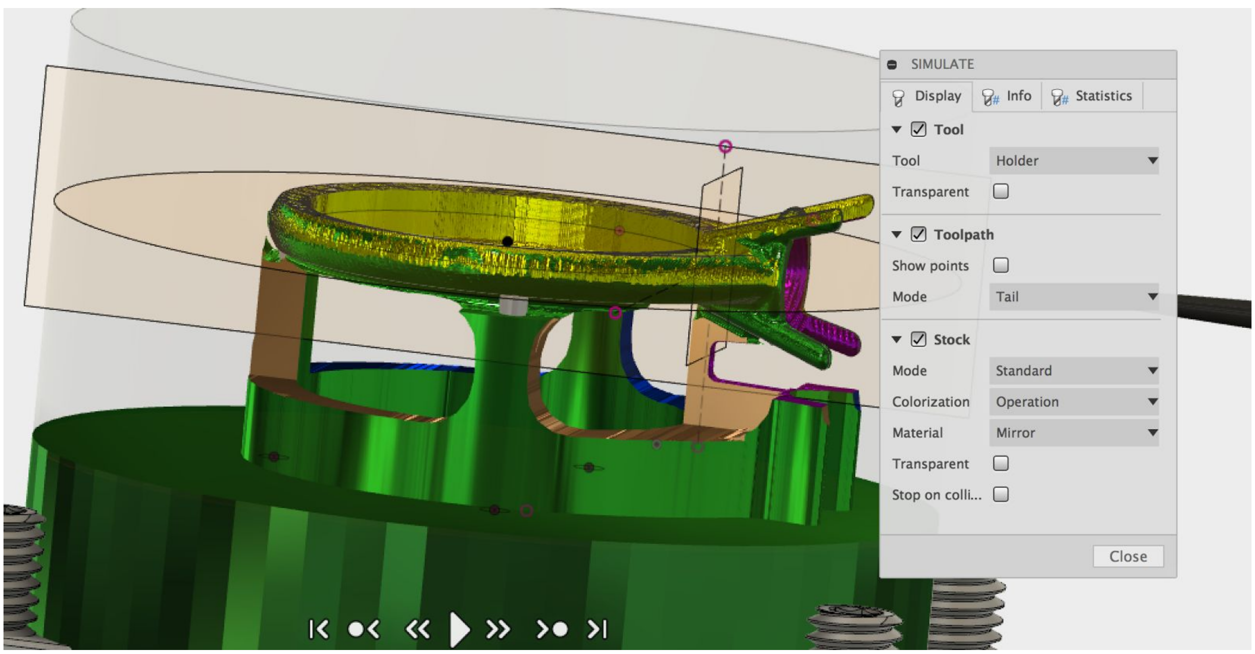


The toolpath and simulation are shown in the two images below.

Test Ring

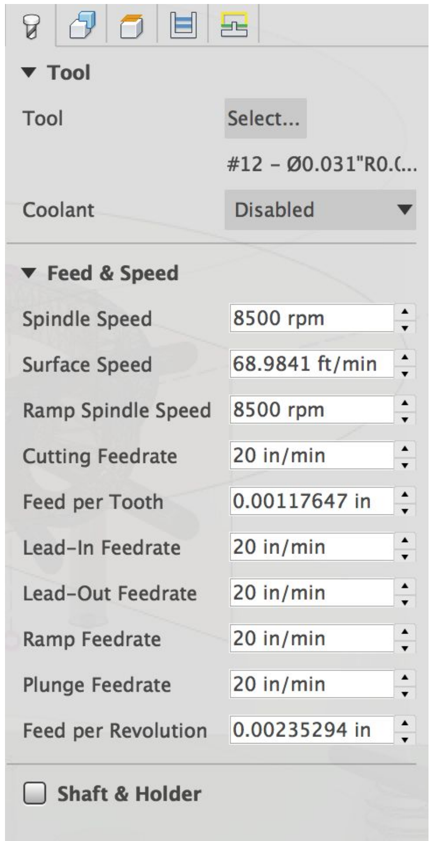
Wax ring test part
Rev A 1/15/17
http://www.pocketnc.com





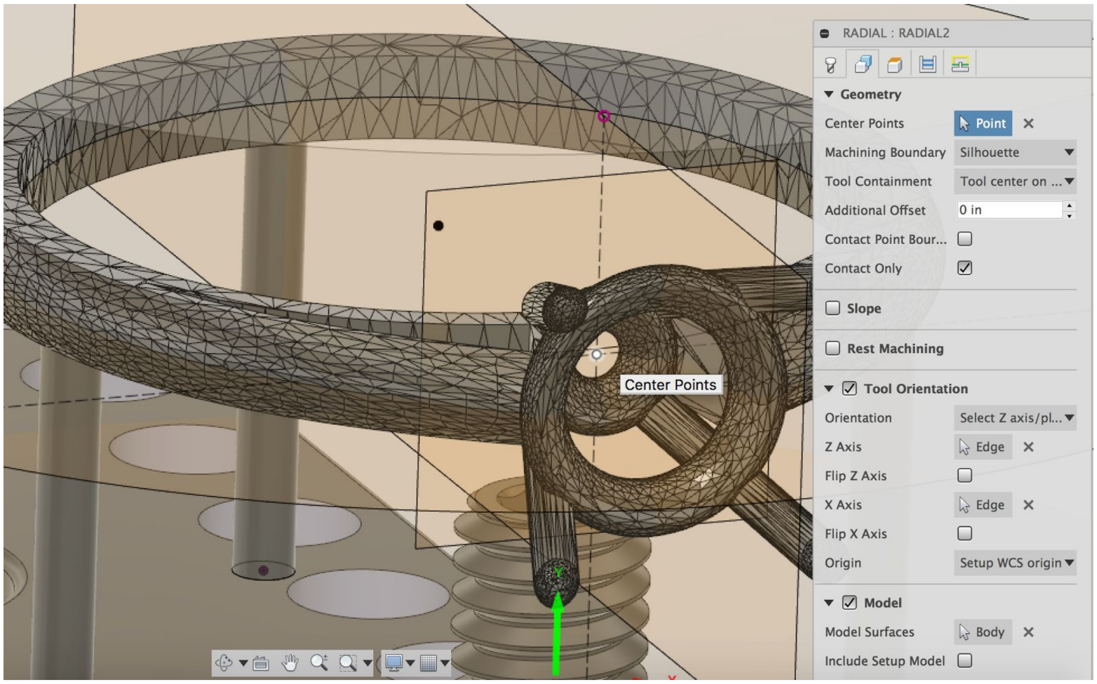
Operation 8: Radial Clear Setting

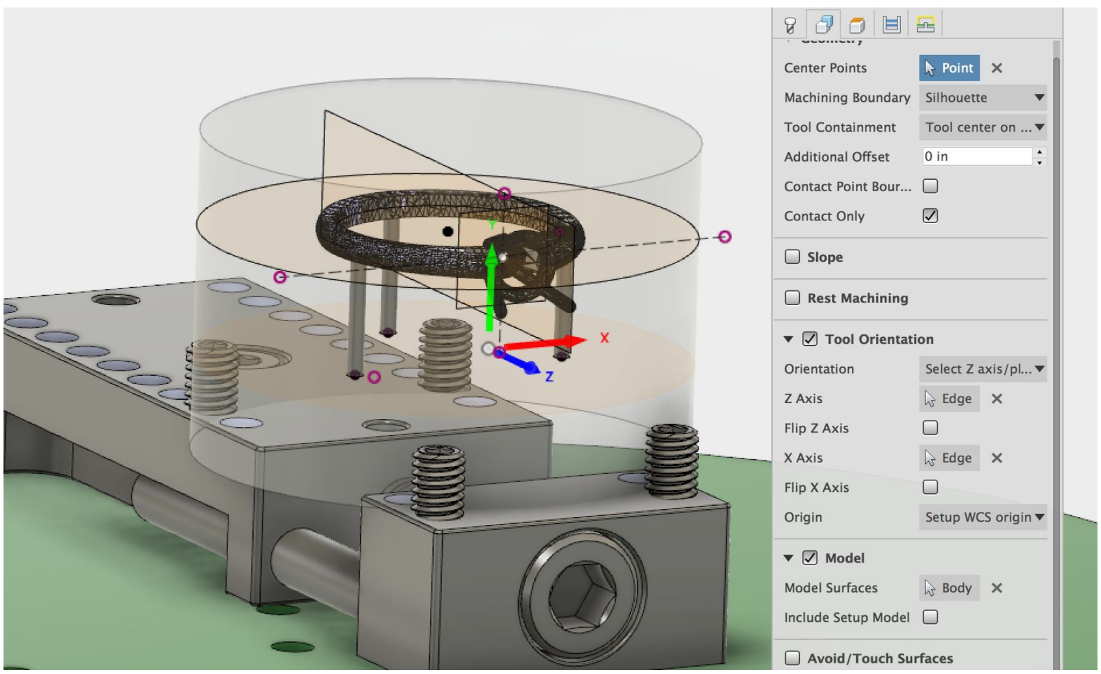
The second to last operation is to clean out the stone setting and remove the last bit of material (about 0.05 inches) from the prongs and setting. The type of toolpath used is radial. All the cuts will start from a center point and proceed outward. These angular distance between each cut is constant. Tool #12 will be used for this operation. The tools settings are shown below.



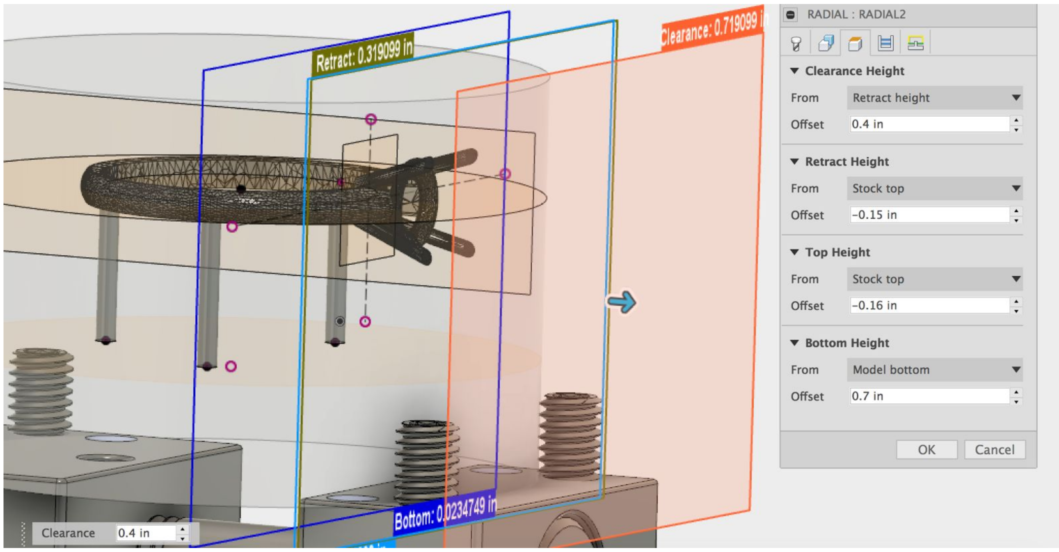
The geometry settings are adjusted next. The centerpoint of the circular hole at the back of the setting is used as the point from which all the cuts start. The “Contact Only” button is selected which allows the toolpaths to skip over the areas that have already been machined.

Set the tool orientation so that it matches what is shown below by selecting appropriated edges on the vise as the X and Z reference plane. Select the ring as the body to be machined.



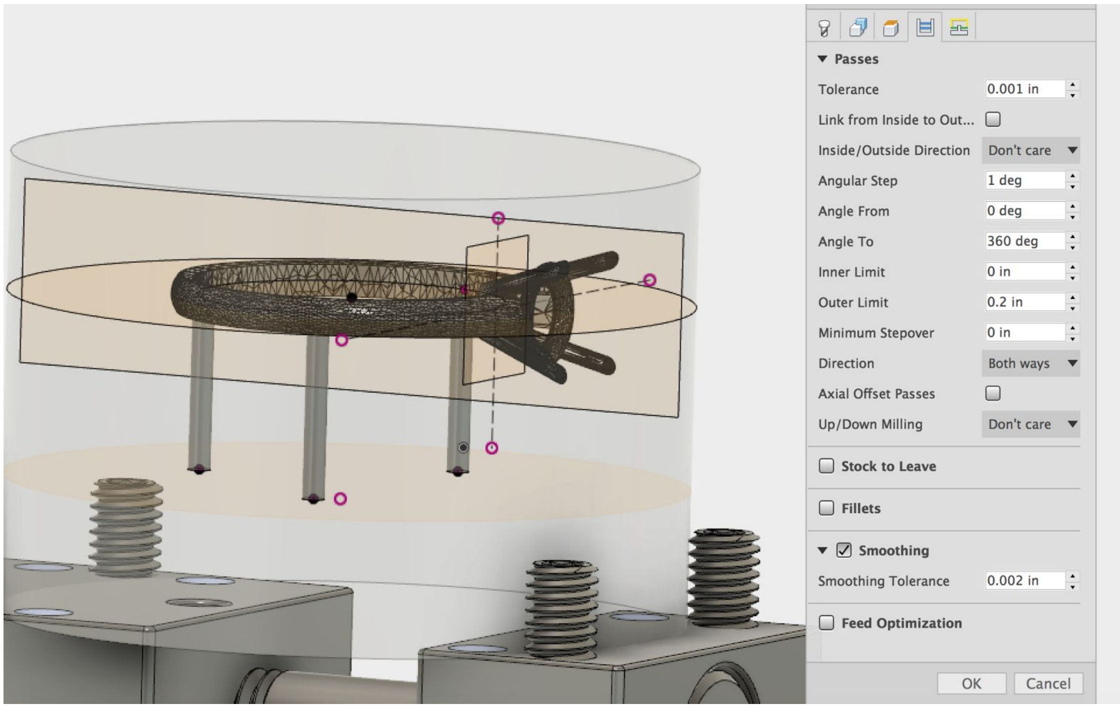


Next, the heights are set as shown below. Both the top height and the retract height are set to be within the stock envelope because the material in that area has been removed so collisions will not take place.

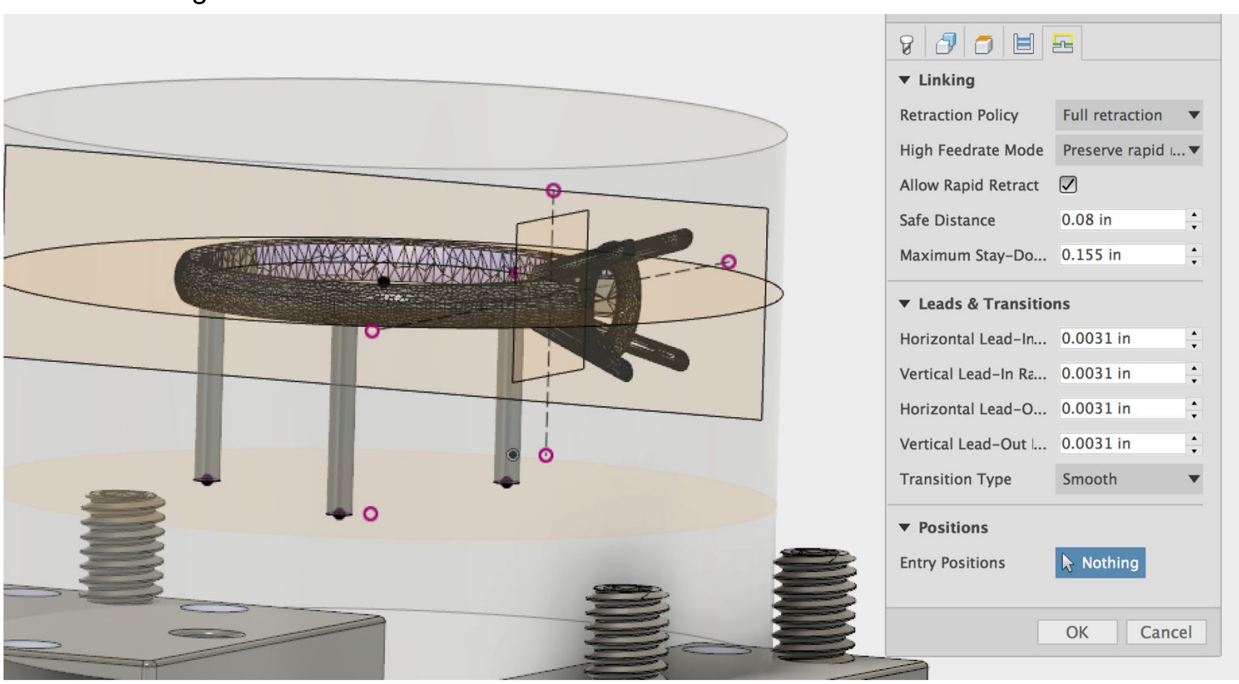


The tool pass settings are shown below. Note that the tolerance is the same as the one on the previous three operations and tighter than the ones used for the clearing operations. The stepover is very small in order to produce a good surface finish and the level of detail desired. If the finished product is rougher than desired this value may be further reduced at the expense of a larger file size and longer machining time.

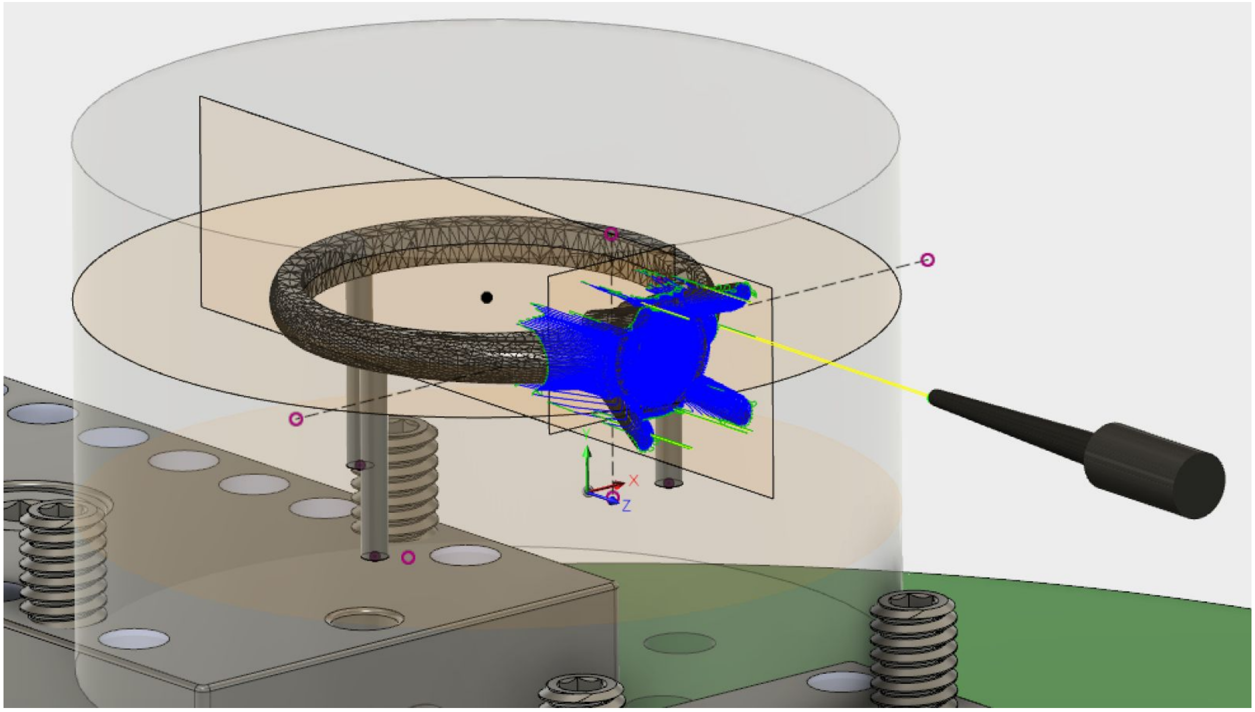
The smoothing tolerance is used to allow Fusion to ignore some of the irregularities on the surface of the ring due to it being a multi-surface body. Because the ring is made up of about 11,000 triangular facets, its surface is far from smooth. The smoothing tolerance allows the tool to follow a path with a specified minimum radius instead of having to follow every contour on the surface.

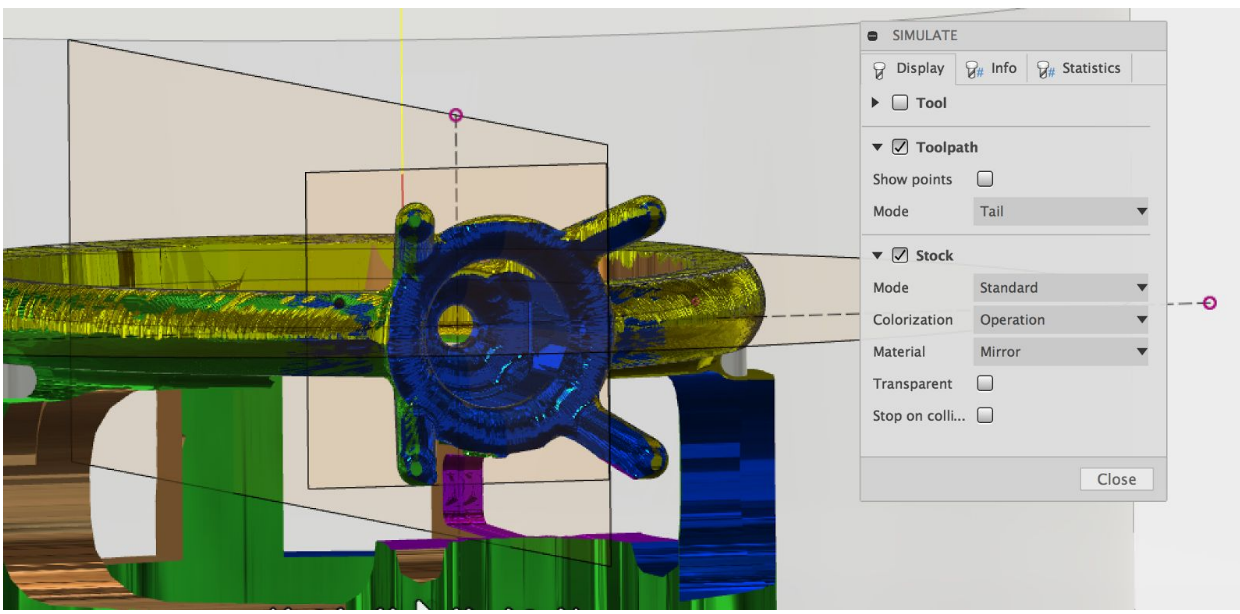


The linking settings are the final settings to adjust before generating and simulating the toolpath. All of the settings below should be the defaults.



The toolpath and simulation are shown below.





SIMULATE
Display
Info
Statistics
Tool
Toolpath
Show points
Mode
Tail
Stock
Mode
Standard
Colorization
Operation
Material
Mirror
Transparent
Stop on colli...
Close

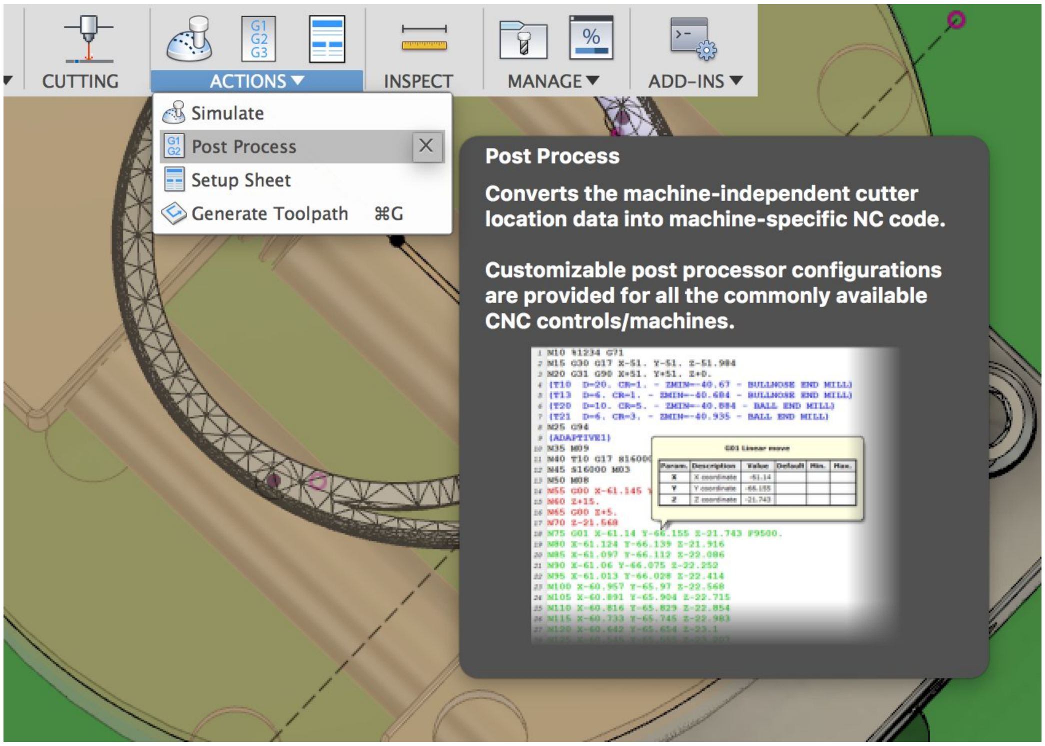
4 Generating G-code from toolpaths: Post processing

The Pocket NC mill can't run directly off the toolpaths produced by Fusion 360. In order to make them useable they have to be converted into code that the mill can read, called G-code. The conversion between the toolpaths generated by CAM software and G-code is called Post Processing.

Most CNC machines have slight differences in the format of the code as well as how certain commands are encoded. This requires each type of CNC machine to have its own post processor that creates G-code that can be read by that particular machine.

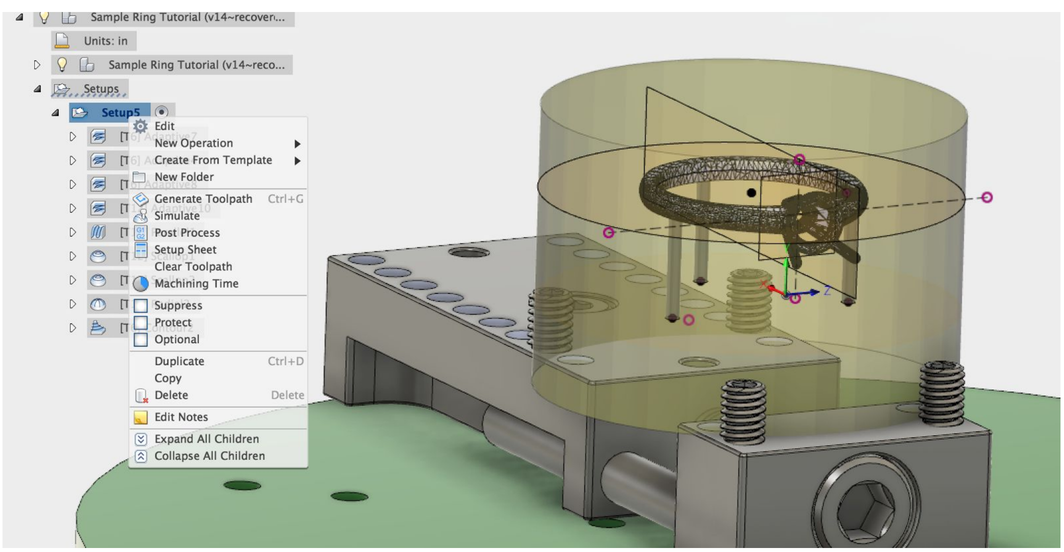
The Pocket NC post processor in Fusion 360 automatically creates the code to orientate the A and B rotary axes based upon the tool orientation defined for each operation and the work coordinate system that defines the machine orientation. The post processor also automatically places a pause at the start of each program as well as between operations if the tool is changed. For example if this part was processed into one code file, there would be a pause at the start of the program as well as a pause between operations 3 and 4 when the tool changes from the ⅛ inch end mill to the 1/32 inch end mill.

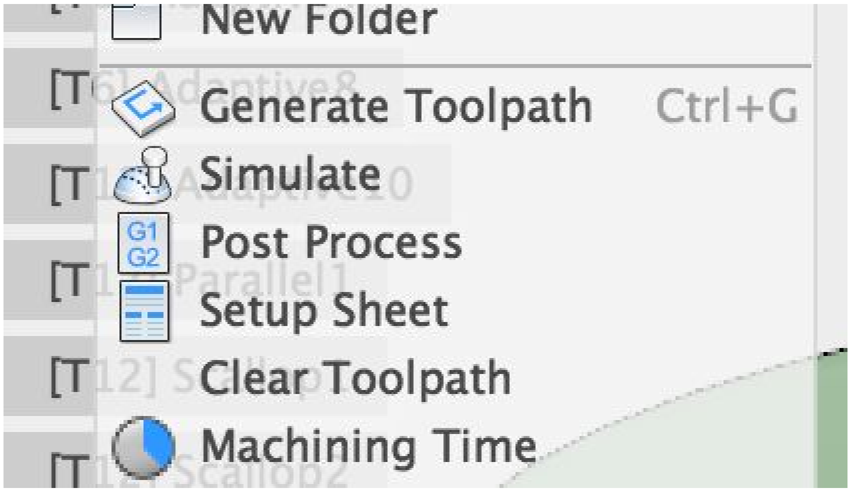
This part could be written in one file by clicking on the "Actions" tab in the CAM toolbar and selecting "Post Process". Select the ""pocket nc.cps - Generic Pocket NC post processor" out of the list. It can be located quickly from the long list of post processors supported by Fusion 360 by beginning to type the name into the processor selection window. Fusion then will prompt for a filename and location to which the file will be saved. Any file name may be used, but it must end with ".ngc" because that is the only file extension that the Pocket NC software can use.



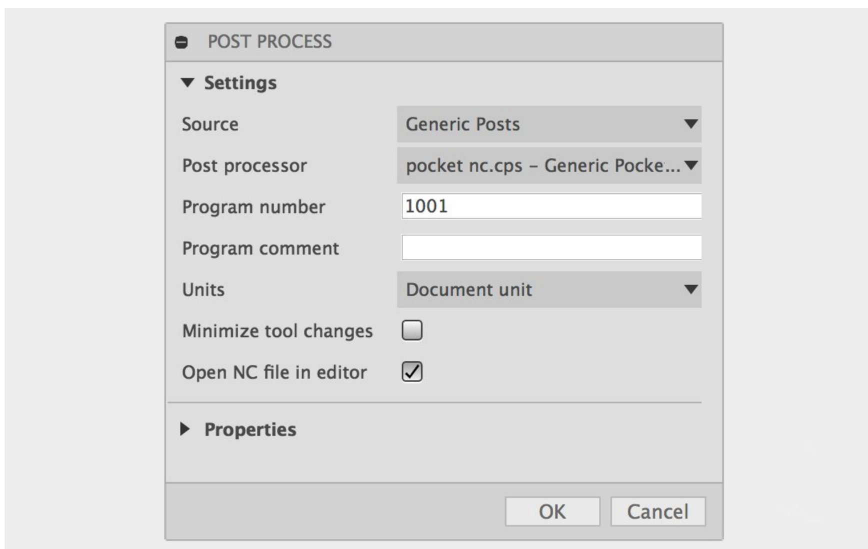
While the ring may be saved as one program file, it is recommended to use as separate program for each tool. For the ring, this will mean two programs; one for the first three operations with the ⅛ inch end mill and another program for the last 5 operations done with the 1/32 inch end mill.

Right click on the set up in the browser tree. Select “Post Process.”

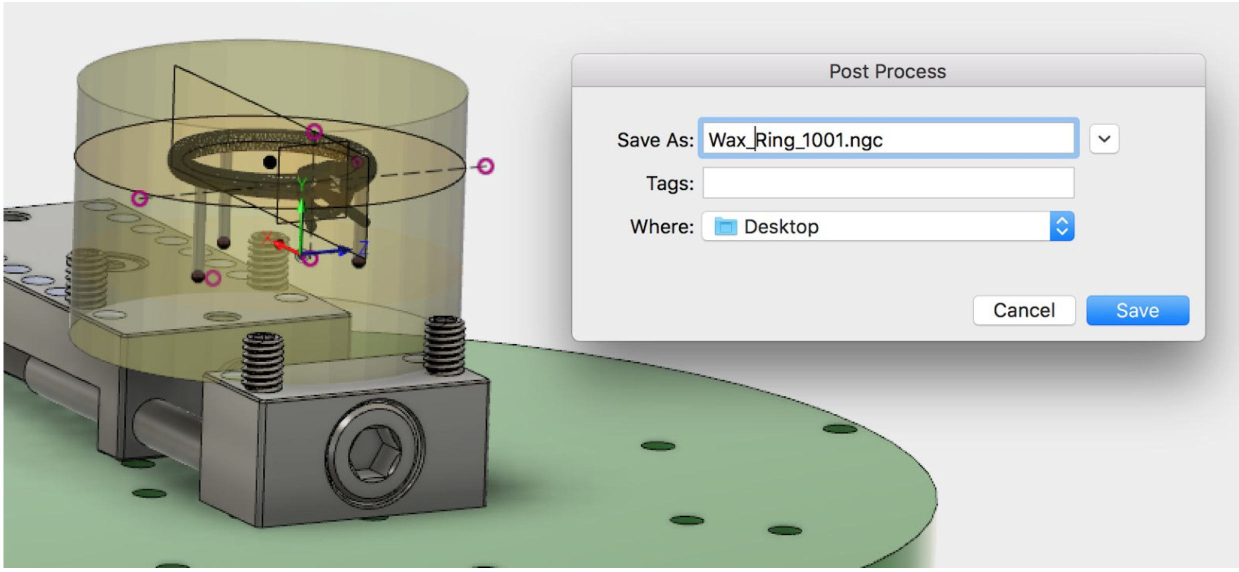




Choose the “pocket nc.cps - Generic Pocket NC” post processor. The name of the program may be changed if desired. Comments may be added if desired. Click “OK.”



Choose where to save the NC file, and what name to use then click “Save.” The filename must end in .ngc because that is the file extension that the Pocket NC mill uses.



The NC file can be viewed and edited with Brackets if you are using a Mac, or Notepad on PC.

```
%
(AXIS,stop)
(1001)
(WAX RING)
N10 G20
N15 G90 G94 G40 G17 G91.1
N20 G53 G0 Z0.
(ADAPTIVE7)
N25 M9
N30 G49
N35 M5
N40 G53 G0 X2.5 Y2.5
N45 M0
N50 T6 M6
(45 HELIX FLAT)
N55 S8500 M3
N60 G54 G0
N65 G53 G0 X2.5 Y2.5
N70 A90. B0.
N75 M9
N85 G0 X-0.481 Y-0.3935
N90 G43 Z1.2043 H6
N95 G0 Z0.4693
N100 G1 Z0.4568 F30.
N105 Y-0.3932 Z0.454
N110 X-0.4807 Y-0.3923 Z0.4514
N115 X-0.4804 Y-0.3909 Z0.449
N120 X-0.4799 Y-0.389 Z0.447
N125 X-0.4793 Y-0.3867 Z0.4455
N130 X-0.4786 Y-0.3841 Z0.4446
```

The NC code is now ready to be used in your Pocket NC mill. It is recommended that you save the file to a flashdrive (USB memory stick) and plug the flashdrive directly into the mill. The file can be loaded and run from there using Machine Kit.

Set the tool length offsets, load the stock, and home all axes on the mill. Plug the flashdrive with the ring-code on it into the Pocket NC mill if you have not already done so. In Machine Kit, click “File => Open” then navigate to the “media” folder on the beaglebone. Use the back arrow button in the upper right hand corner of the window to navigate back through the directory. Open the flash drive, select the ring file, and click “open.”

Push the play button in Machine Kit to start the program. The first line in the code is a pause so you will need to immediately push the yellow pause button on the mill or the pause button in Machine Kit in order to start the program.

This concludes the tutorial on how to program the wax ring for machining on the Pocket NC mill. Thanks for reading.

Happy milling!

Quick Reference

Material:

Wax Cylinder:

Diameter: 1.5 inches **Height: 1 inch**

The wax ring test part is designed to be cut from wax using the Pocket NC mill. Harder materials may cause damage to the Pocket NC mill as feeds and speeds have been selected specific to wax.

Runtime:

~1 hour

Tooling:

Tool number 6

⅛ inch 2 flute flat 45 deg helix (Harvey Tool # 24208)
http://www.pocketnc.com/45-degree-square-helix/18th-inch-45-degree-square-helix-2-flute

Tool number 12

.031 inch 2 flute 2 deg ball nose (Harvey Tool #29831)
http://www.pocketnc.com/tapered-rib-cutters/10-degree-164th-tapered-rib-cutter

Fixture:

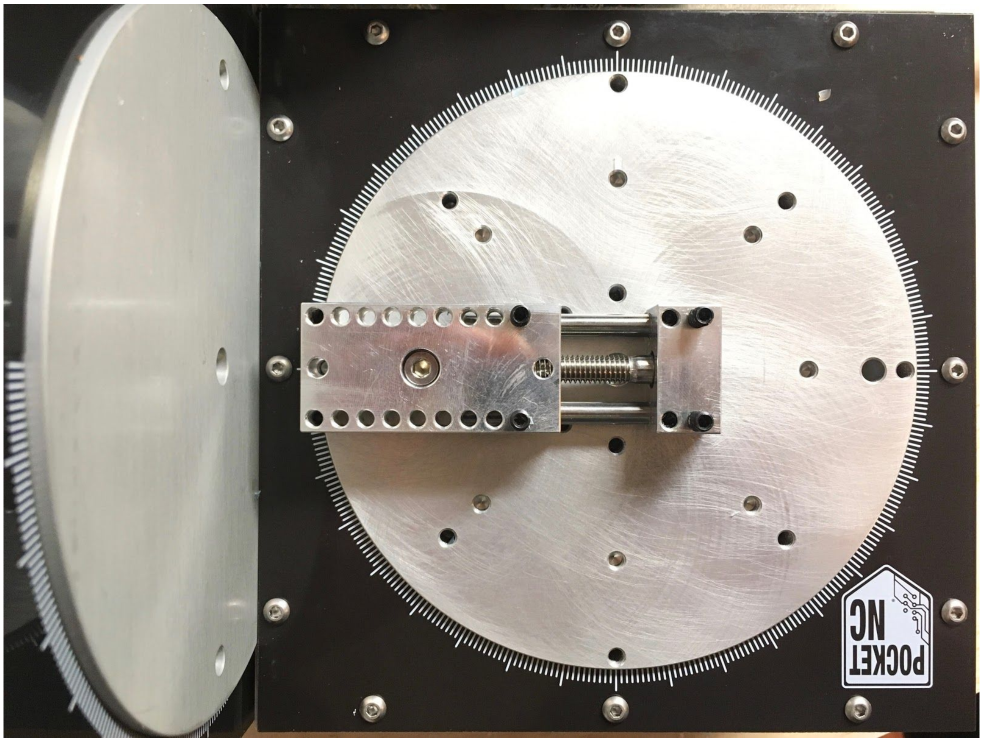
Wax should be held in the Pocket NC vise using threaded set screws to grip the sides of the wax blank.

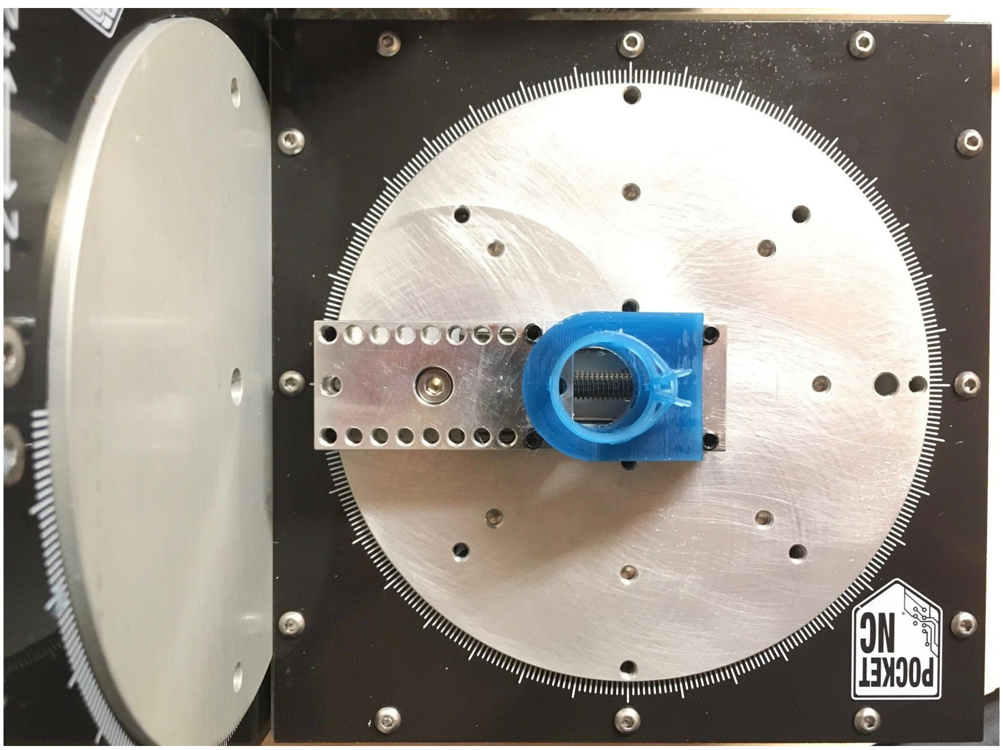
Test Ring

Wax ring test part

Rev A 1/15/17

http://www.pocketnc.com





Notes:

Starting the Program

Set the tool length offsets, load the stock, and home all axes on the mill. Plug the flashdrive with the ring-code on it into the Pocket NC mill if you have not already done so. In Machine Kit, click “File => Open” then navigate to the “media” folder on the beaglebone. Use the back arrow button in the upper right hand corner of the window to navigate back through the directory. Open the flash drive, select the ring file, and click “open.”

Push the play button in Machine Kit to start the program. The first line in the code is a pause so you will need to immediately push the yellow pause button on the mill or the pause button in Machine Kit in order to start the program.

Tool changes

The wax ring uses only two tools. The first tool (number 6) should be loaded before starting the program. The second tool (number 12) is loaded after tool number 6 has completed its operations and has come to stop in the tool change position. Tools are installed and uninstalled using the green handled 3mm hex tool. NOTE, a sufficient whack may be required at the base of the tool holder to remove tooling. Use the back of the green 3mm hex tool for this.

Workholding

The material should be held tight using the Pocket NC vise with M4 set screws. Use the provided 5mm wrench to tighten the vise jaw.