

IM3206-L

Introduction to CNC Programming: Hands-on Autodesk Fusion CAM Workshop for Designers and Engineers

Scott Moyse
Toolpath Labs

Lab Assistants:

Tim Paul - Toolpath Labs
Al Whatmough – Toolpath Labs
Paul Clauss – Toolpath Labs
Philip Mestenhauser – Meta
Rob Lockwood - Meta
Marti Deans – Autodesk
Seth Madore – Autodesk
Wayne Griffenberg – Autodesk
Dan Pacific - Autodesk

Learning Objectives

- Gain a basic understanding of CNC milling processes and the role of CAM in translating designs into machine instructions.
- Identify potential design for manufacturability issues using some of the available toolsets in Autodesk Fusion.
- Learn how to define fixtures, create setups and tools, and then machine simulate your program before postprocessing.

Description

This hands-on lab will introduce engineers and designers to the world of using CAM to program computer numerical control (CNC) machines and its connection to the design process. Through a series of guided exercises in Autodesk Fusion CAM, you'll gain practical experience in setting up stock material and coordinate systems, choosing tools, defining machining operations (including machining boundaries), simulating the toolpaths, and generating G-code for a CNC milling machine. This session will equip you to better communicate with CAM programmers and optimize your designs for manufacturability.

Speaker(s)

Scott Moyse is the RevOps Strategy Manager for Toolpath.com, where he drives outreach initiatives to expand the user base of the Toolpath AI powered quoting and CAM automation product. Collaborating closely with the customer success team, Scott ensures the delivery of valuable content and support to the Toolpath & Fusion user community.

Scott's career began in the UK, where he studied Motorsport Engineering before relocating to New Zealand. He spent over nine years at SMI, starting as design support and quickly moving into CNC programming. His role involved close collaboration with both manufacturing and design teams to develop and implement automated processes, providing him with a deep understanding of both areas. In 2008, Scott transitioned back into full-time design, and by 2009, he was appointed Design Manager. During this time, he implemented and managed Autodesk Vault Professional, significantly improving communication, work allocation, and control over the design review process.



In late 2013, Scott joined Cadpro in New Zealand as a Manufacturing, CAM & Data Management Technical Specialist. There, he focused on Autodesk's manufacturing design and engineering products, with a particular emphasis on CAM. Scott also developed skills in writing Post Processors for Autodesk Fusion, HSMWorks, and Inventor CAM, as well as creating add-ins for Fusion. Over the years he was instrumental in helping to develop and grow the Massif platform, with his final 18 months at Cadpro being focused on launching Cadpro's new website, brand and internal business systems to support the Autodesk Agency model and future Cadpro growth.

Outside of work, Scott is passionate about designing Grumpy Sloth low-profile mechanical keyboards, a lifelong Formula One fan and enjoys running long distances on challenging trails.

CNC Basics

High level CAD to CAM flow

The CAD to CAM to CNC workflow is a process used to turn digital designs into physical products. It starts with the 3D model being brought into the CAM software, which converts it into toolpaths that ultimately guide a CNC machine. Finally, using a post processor, the toolpaths are output in a format the CNC machine understands, so it can precisely cut the material into its final form.

Co-ordinate Systems

In CNC machining, the Machine Coordinate System (MCS) is the fixed reference frame inherent to the machine, while the Work Coordinate System (WCS) is set by the user to align with the specific workpiece. Tool offsets adjust for the physical dimensions of cutting tools—such as length and diameter—to ensure that machining operations are executed accurately within the WCS. Together, the MCS, WCS, and tool offsets enable precise and accurate manufacturing by aligning machine movements with the intended design.

Machine origin

When the machine powers up, it undergoes an indexing or homing process, moving each axis to predefined reference points using limit switches or sensors. This action establishes the machine's zero point, resetting the MCS so the machine knows the exact positions of its axes relative to a fixed origin. By setting the MCS through indexing at each startup, the machine ensures accurate interpretation of coordinates and consistent precision in all subsequent machining operations.

WCS origin

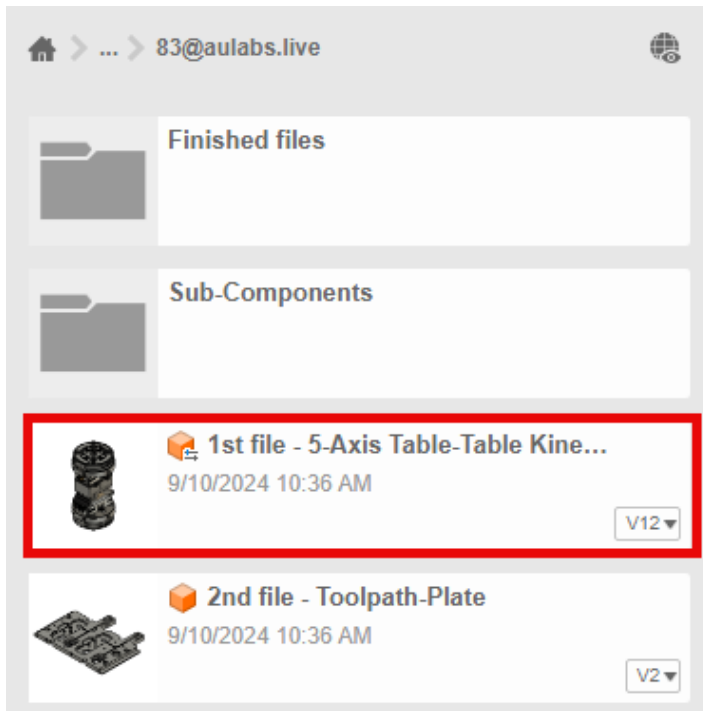
The Work Coordinate System (WCS) is a user-defined reference frame tailored to the specific workpiece. Operators set the WCS by establishing a zero point on the part using techniques like edge finding, probing, or fixture offsets. This customization allows toolpaths to be programmed relative to the workpiece itself, ensuring that machining operations are accurate and repeatable regardless of the part's position on the machine table.

Tool offsets

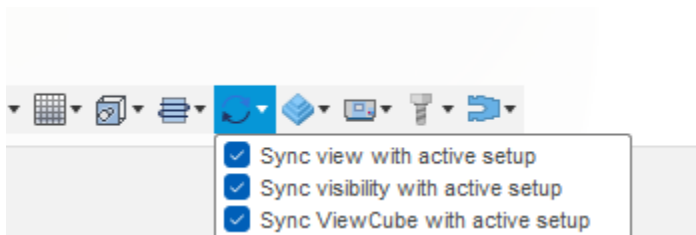
Tool offsets are adjustments made in the machine's control system to account for the exact dimensions of cutting tools, such as their length and diameter. These offsets ensure that the tool's tip is correctly positioned relative to the workpiece, compensating for any variations in tool size or wear. By inputting precise tool measurements, the CNC machine can execute programmed toolpaths accurately, maintaining tight tolerances and consistent quality. Tool offsets are essential for precise machining operations, enabling transitions between different tools and ensuring each cut aligns perfectly with the required design.

Co-ordinates in action

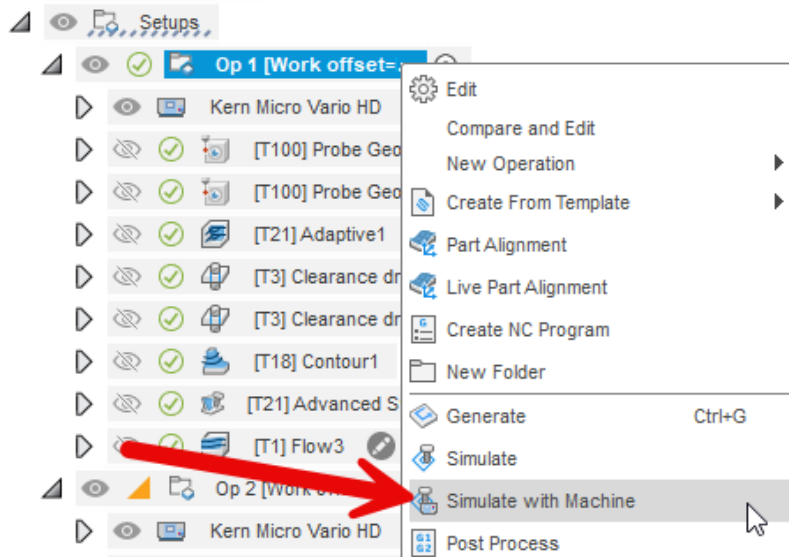
1. From your lab machine folder, open the document named “1st file - 5-Axis Table-Table Kinematics”



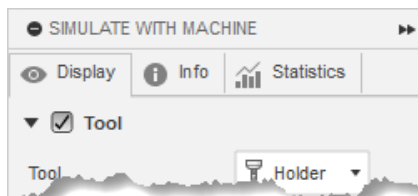
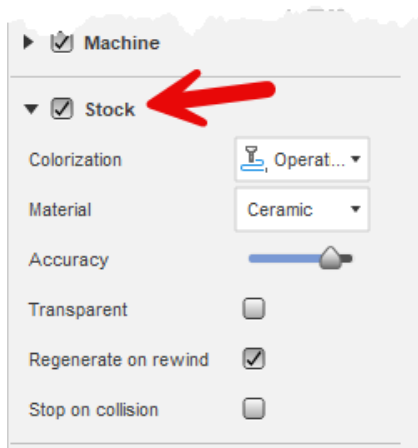
2. Make sure all the **Synchronise active setup** options are enabled in the bottom toolbar of the graphics area.



- Right-click on the “Op1” setup in the browser and select ***Simulate with Machine***

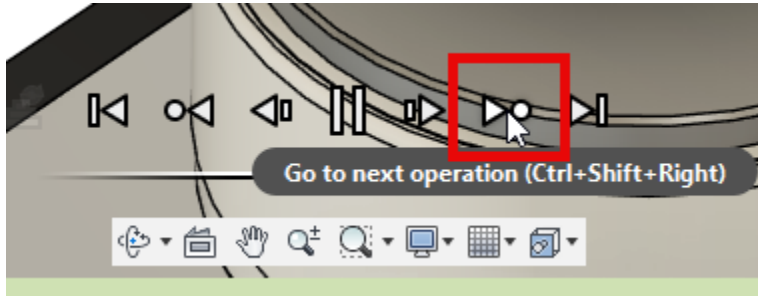


- In the simulation pallet on the right hand side of the screen, enable the ***Stock*** checkbox on the ***Display*** tab. This will allow you to see what the probe is touching.



- Start the simulation by clicking the play button at the bottom of the graphics area.

- Once the Adaptive toolpath has roughed away some material, click the **Go to next operation** button to skip ahead one operation at a time.



As you progress through the toolpaths, take note of the way the linear and rotary axes move together. All the toolpaths are positional 5-axis (just 3-axis toolpaths at different rotary positions), until you get to the Flow toolpath at the end. All 5 axes start moving at the same time, that's a simultaneous 5-axis toolpath.

- Switch to the **Info** tab and take note of the additional B & C axes under the **Machine** group.
- Note how different the XYZ co-ordinates are between the Position and Machine groups. Position is relative to the WCS and Machine is relative to the Machine Coordinate System.
- Click the green tick on the ribbon to exit simulation.

SIMULATE WITH MACHINE

Display **Info**

Position

X position	20.99 mm
Y position	14.318 mm
Z position	70.98 mm

Spindle speed 14006 rpm CW
Feedrate 1638.7 mm/min
Movement Cutting

Operation

Time 0:00:05 1.3%

Machine

Description Kern Micro Vario HD

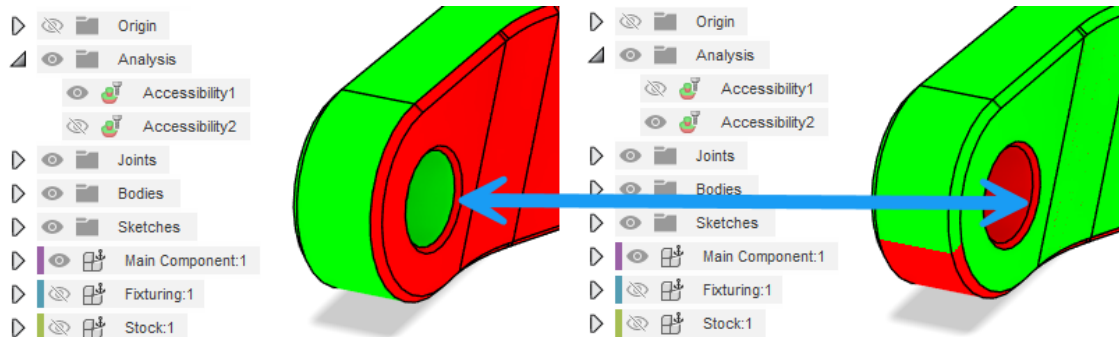
Current Pose

X	73.69 mm
Y	14.421 mm
Z	-147.553 mm
B	-34.8 deg
C	-0.3 deg

Design for manufacturing (DFM)

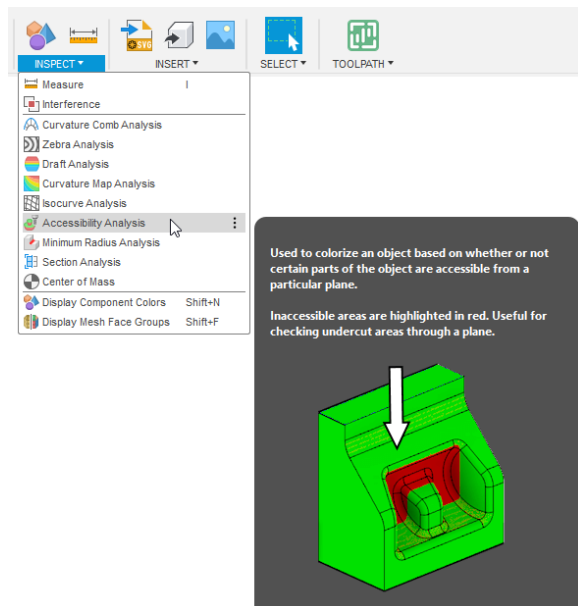
Accessibility analysis

1. Leaving the previous file open, switch to the Design workspace
2. Turn off the visibility of the Fixturing and Stock components.
3. Expand the Analysis folder and turn on the Accessibility analysis'



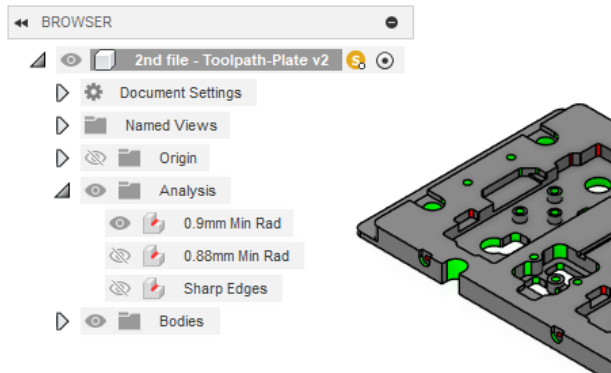
You can see that the internal chamfer isn't accessible from any orientation, it will require a tool and a toolpath that can undercut to get the job done.

These analysis' were created using the Accessibility Analysis tool in the Inspect drop-down menu on the ribbon.

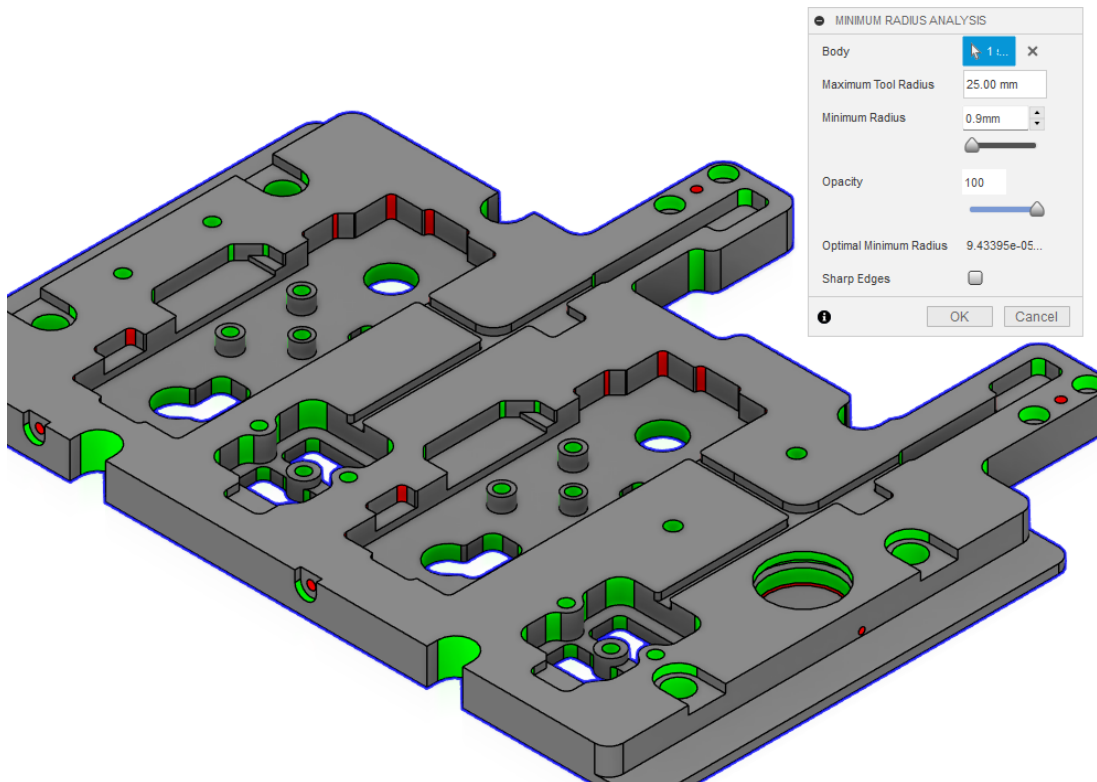


Min rad & sharp edge analysis

1. From your lab machine folder, open the document named “ 2nd file - Toolpath-Plate”
2. Expand the **Analysis** folder in the browser

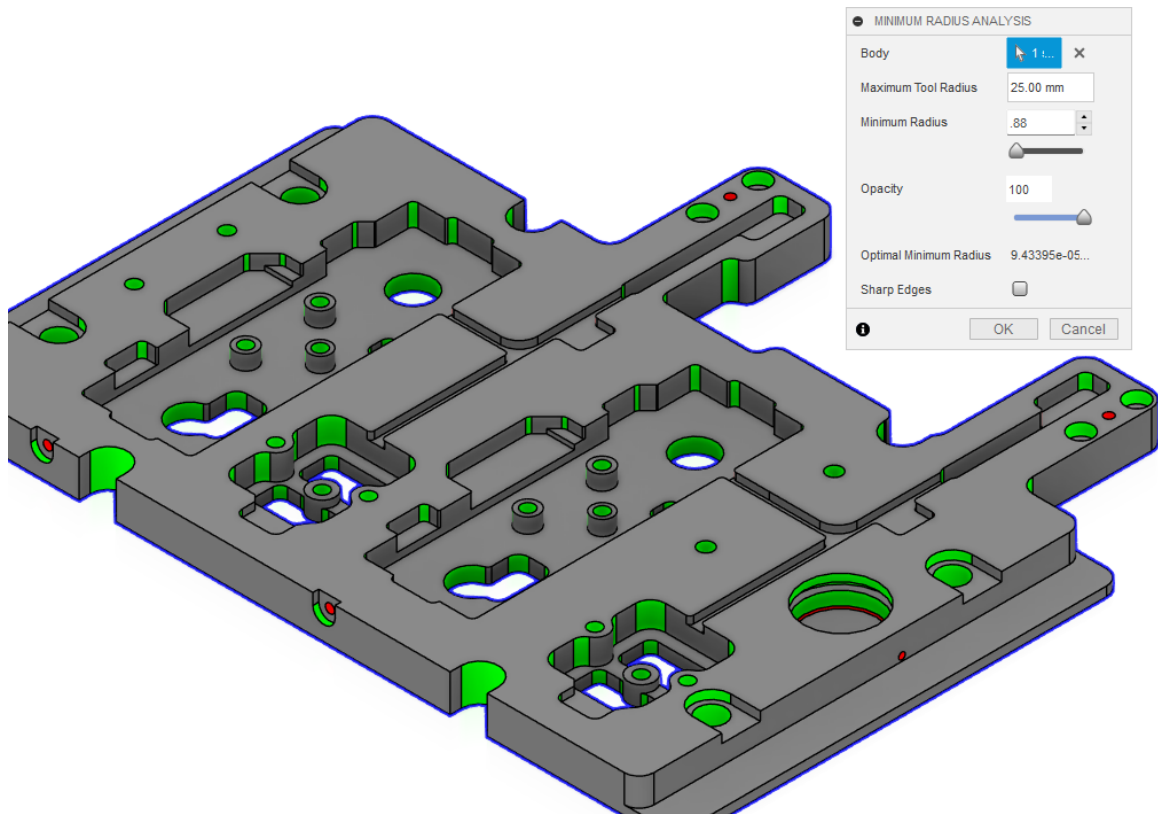


3. Click on the eye icon for the 0.9mm Min Rad analysis.
4. You can right-click and select Edit if you want to see how the analysis is configured.



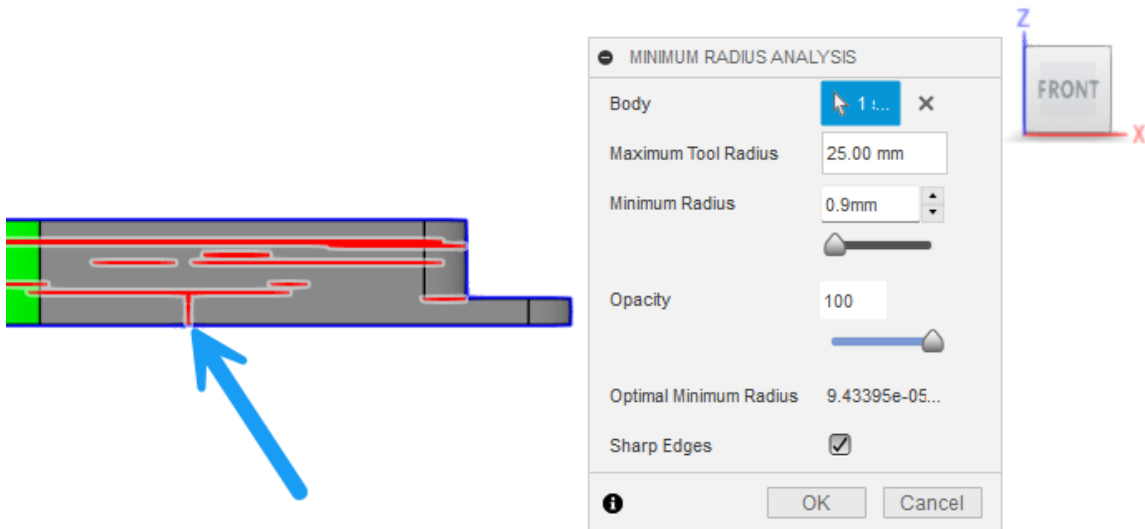
Notice all the small red internal radii. If you keep reducing the minimum value until they all turn green, you will find the minimum radius tool required to finish these pockets.

5. Now turn on the 0.88mm Min Rad analysis in the Analysis folder.

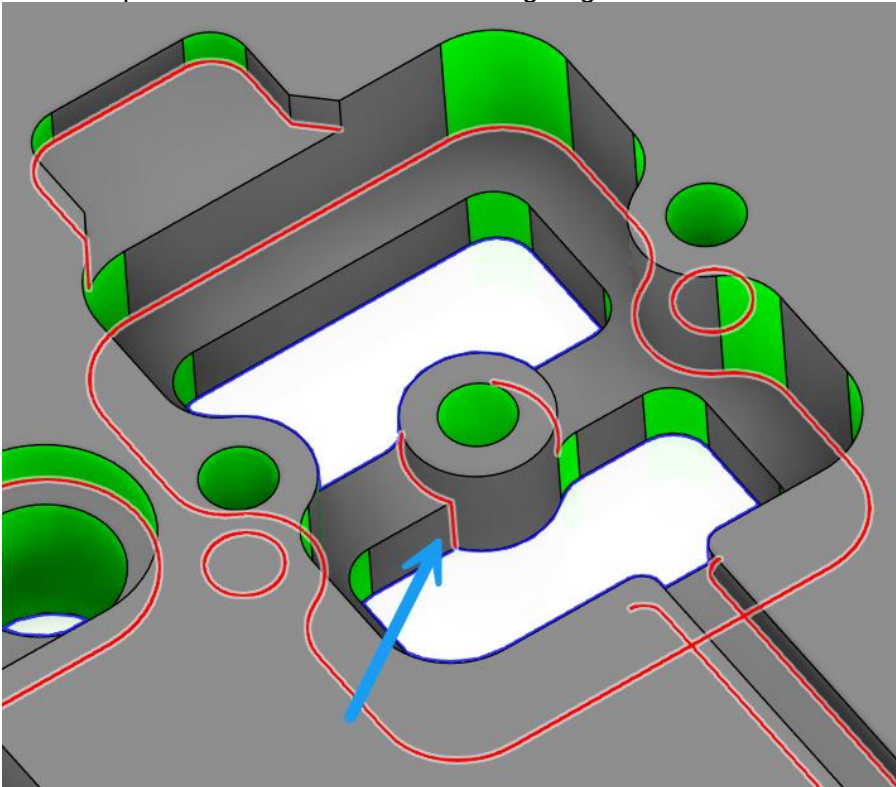


Aside from the red holes that would get drilled and the small fillet at the bottom of the circular pocket, the min radius to finish all the pockets on this side of the part is 0.88mm. So really quite small!

6. Now turn on the Sharp Edges analysis.
7. Click the Front face on the view cube and look for any vertical red lines



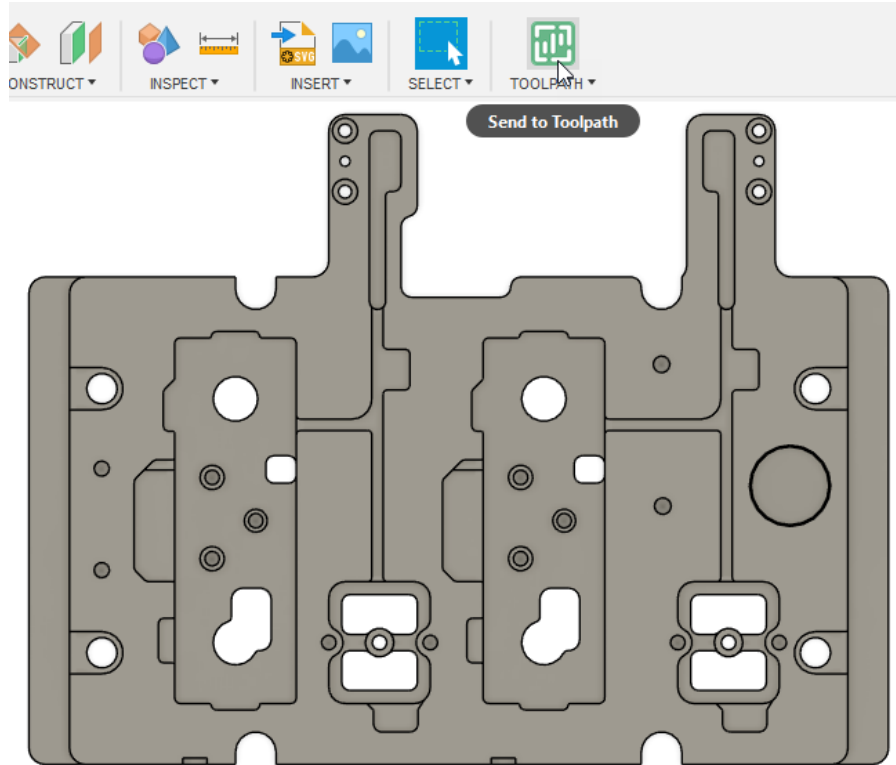
8. Orbit the part around to find the offending edge



It's clear to see here that the designer missed a fillet radius, making this part harder to manufacture.

Cheeky plug for Toolpath

OR instead of all that, you could just send your part to [Toolpath](#) to analyse and get an estimate:



It will go on to assess the cost to machine each feature based on the tooling you have available. Then optionally output all the toolpaths back to Fusion.

TOOLPATH

Dashboard > Parts > 3rd File Toolpath Plate V2 Body1

3rd File Toolpath Plate V2 Body1

Summary Tools 10 Setups 4 Strategy Estimate

Starter Demo Cut Config Export to Fusion

179 %

134.51 x 101.6 x 6.35 mm

Machine type 3-axis

Bounding box 134.51 x 101.6 x 6.35 mm

Stock size 139.7 x 107.95 x 19.05 mm

Part volume 48.1 cm³

Removal volume 38.6 cm³

Modified Features
Some features of this part were modified for machinability.

SETUP 1

- Open Pocket #1
- Open Pocket #3
- Open Pocket #7
- Open Pocket #9
- Blind Hole #5
- Blind Hole #6
- Blind Hole #7
- Blind Hole #8

SETUP 2
Through Pocket #8

3D Surface No

L/D ratio 50.96:1

Depth below top of part 2.03 mm

Feature depth 2.03 mm

Largest tool diameter possible 6.11 mm

Smallest tool diameter required 0.04 mm

ROUGHING TOOLS

1/16" 3F EM

0.0625 in 0.3125 in 0.1875 in

FINISHING TOOLS

1/16" 3F EM

0.0625 in 0.3125 in 0.1875 in

Tight Radius Deviation
A model deviation has occurred to make this feature machinable

Deviation amount 0.08 mm

This is how a sharp internal edge is reported in Toolpath.

Fusion CAM Basics

Thanks to Phil Mestenhauser for providing the epic workholding template. Check out his class [MFG3914 | Templates, Configurations, and Containers for Agile Prototype Machining in Autodesk Fusion](#)

The purpose of this section is to give you some insight into what's required to program a single setup of a part that could be machined on a 3-axis, but with the benefit of a 5-axis machine. You will see some of the challenges programmers come across, but also just how easy it can be to program a 5-axis machine when all things are going in your favour.

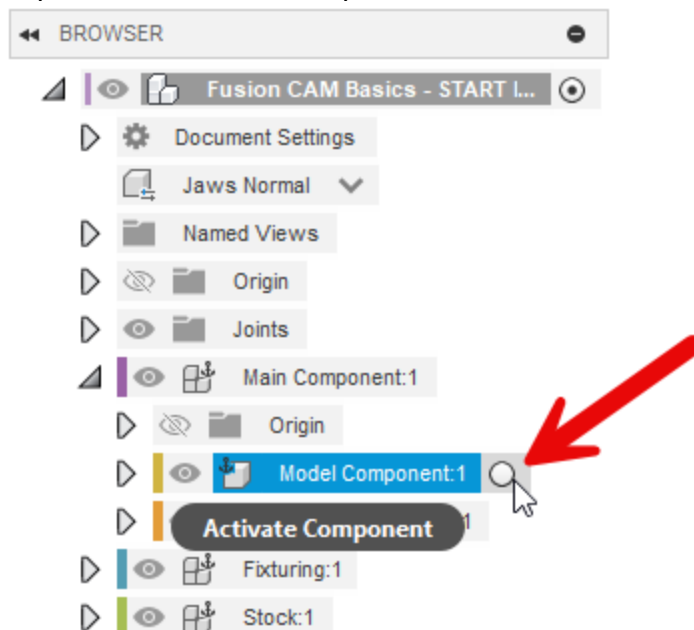
Setup CAD for CAM

We're now going to step through how to setup and program the suspension link part you saw in the earlier dataset.

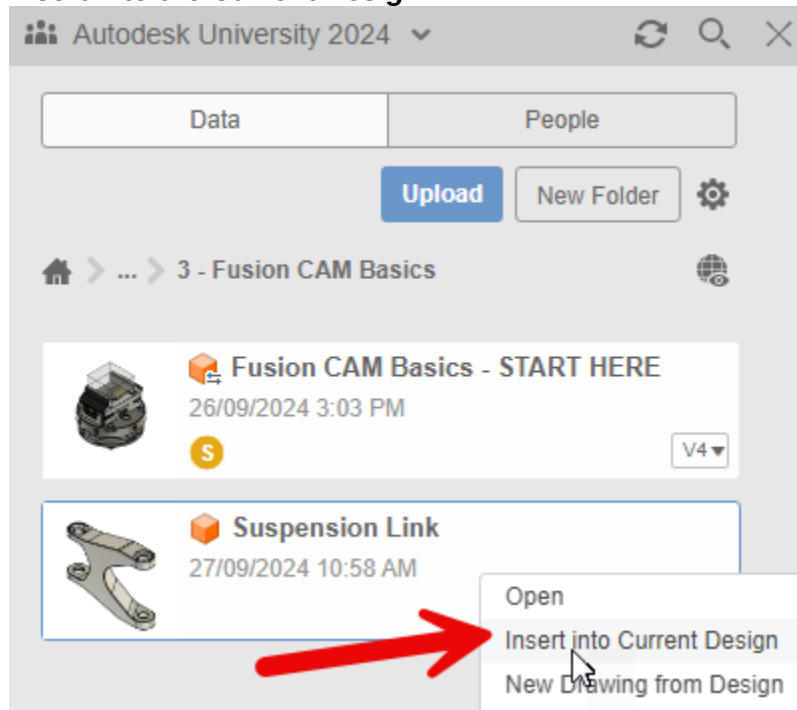
Insert part

We have a workholding template document prepared, typically this would include all the work you repeat each time you program a part. So you don't have to do it over and over again.

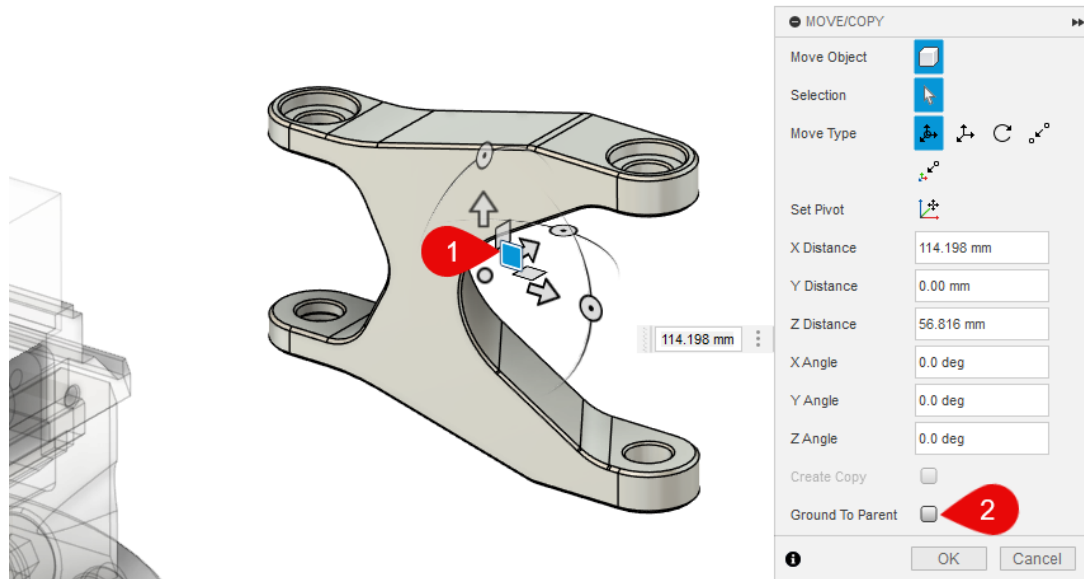
1. Open The "Fusion CAM Basics – START HERE" document
2. Since we don't want to overwrite our 'template', from the File menu, select Save As, and replace START HERE with your initials. In real life, you may save this with the job number and in an appropriate directory within Fusion Team.
3. Expand out the Main Component, then activate the Model Component.



- In the Data Panel, right click on the “Suspension Link – Mod” document and select ***Insert into the Current Design***.

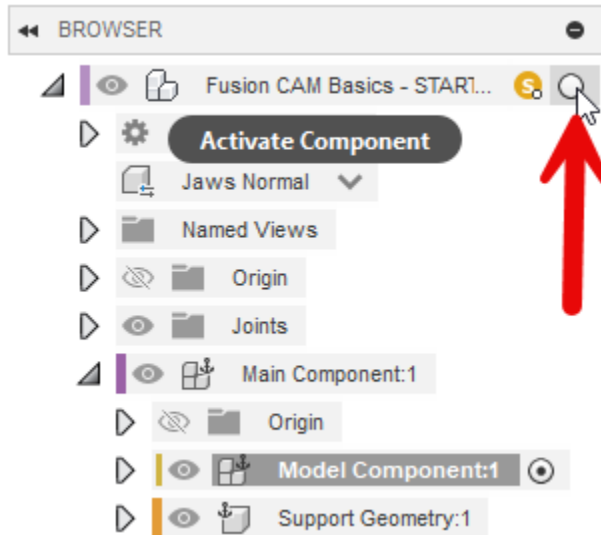


- Use the manipulator to move the component into clear space, so it's not obstructed in anyway. Then make sure you disable ***Ground To Parent*** before clicking **OK**.

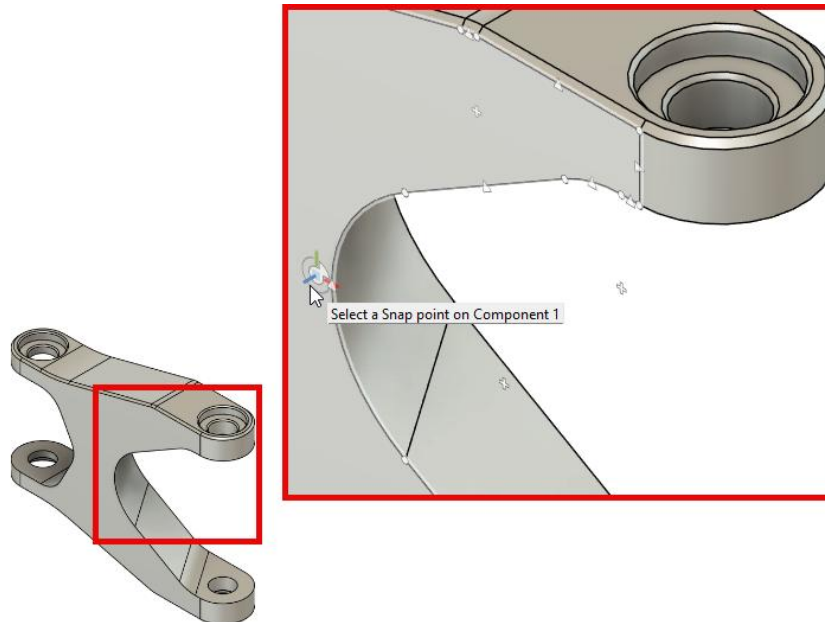


Now we need to position this component into the fixture, in such a way that we have the clearance required to machine as many of the surfaces as possible in the first setup. To do this, we will use a Joint and snap it to a pre-created Joint Origin.

6. Activate the top level Component in the document

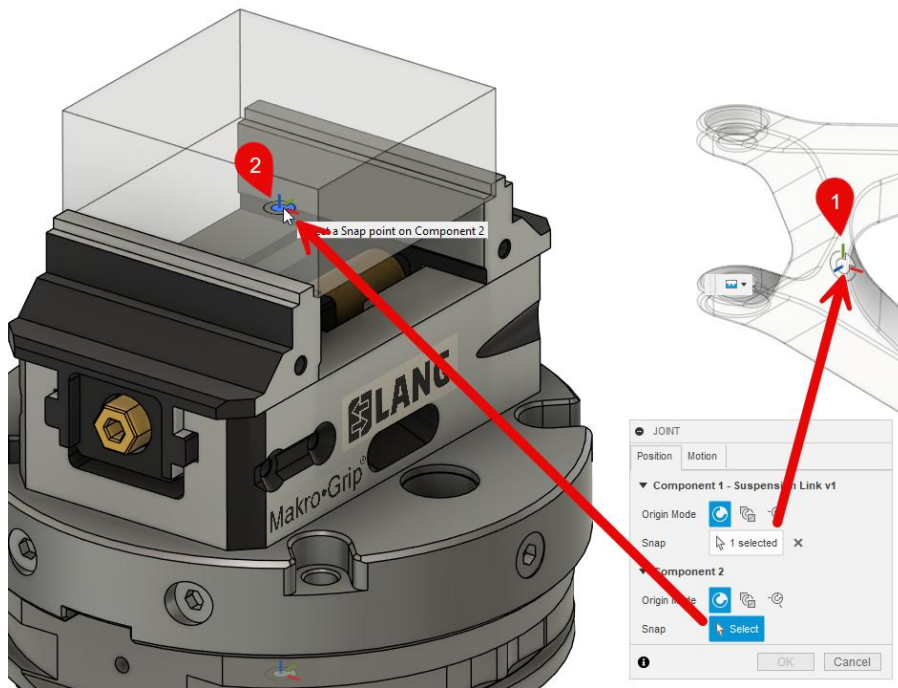


7. Press 'J' for Joint. You're going to join Component 1 to Component 2.
a. Select the following snap point on the underside of the component. Take note of the direction of the blue z-axis on the Joint:

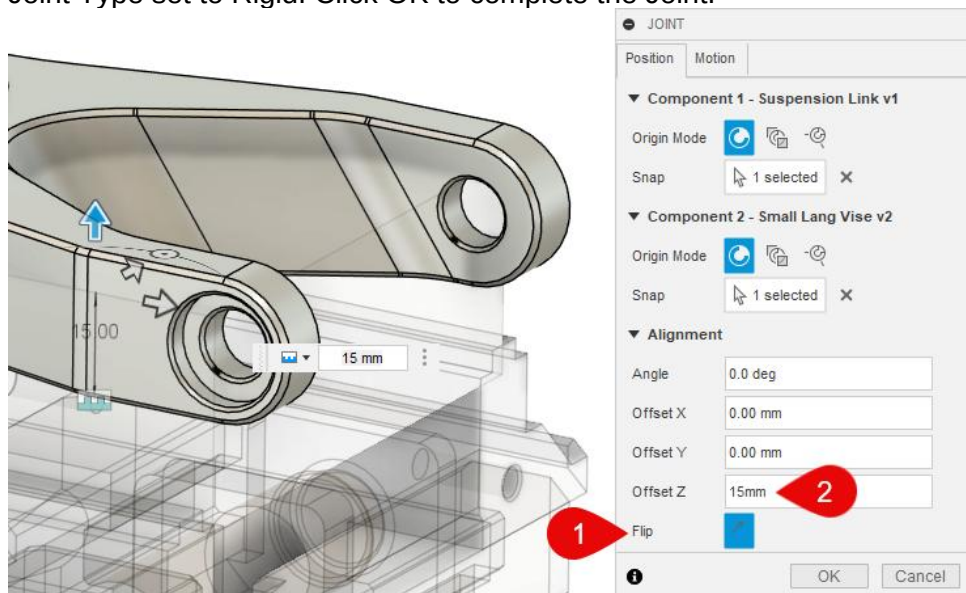


Note, it's the largest face on the part, just in case you have orbited your model. You can hold down CTRL to lock the selection to a single face before selecting the point.

- b. For the second input, you're going to select the Stock Attachment Joint Origin from the graphics area.



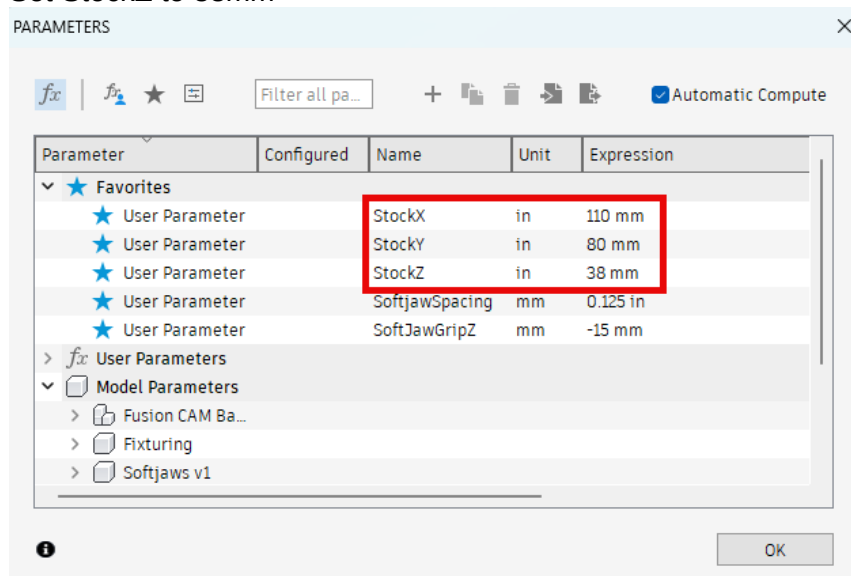
- c. You will notice the component is initially upside down, so toggle the Flip button. Then it needs to be lifted up to clear the jaws and give you the clearance to get a cutter underneath the part to profile it. Enter a 15mm Z offset to achieve this. Switch to the Motion tab and make sure you have the Joint Type set to Rigid. Click OK to complete the Joint.



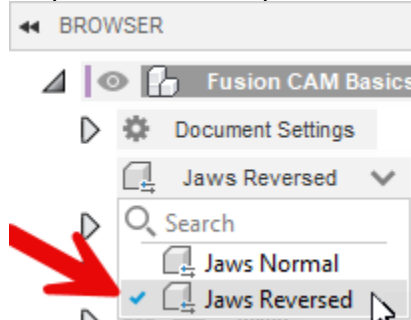
Set stock size

As you can see, the translucent stock body isn't big enough for this component. Let's fix that now.

1. From the **Modify** panel drop down, select **Change Parameters**.
2. Set StockX to 110mm
3. Set StockY to 80mm
4. Set StockZ to 38mm



5. Since the jaws are now overhanging the vise body, we can use our configuration drop-down at the top of the browser. Switch to the Jaws Reversed configuration.

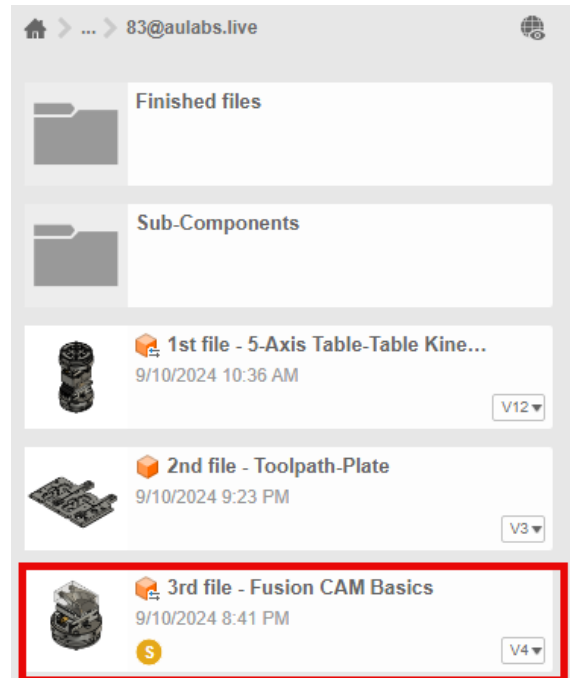


Setup your CAM

Open the “3rd file - Fusion CAM Basics” document from your lab machine folder.

Now you have the CAD all setup, you can go about creating your first setup. You will need to define the following:

- The machine configuration
- If you are milling or turning
- The orientation and position of your work coordinate system
- Which geometry is the Model you will be manufacturing
- Which geometry is the fixture and therefore should be avoided
- Which geometry, if you have modelled it, will be the stock
- Your program number
- The work coordinate offset number you will use when running this program on the machine

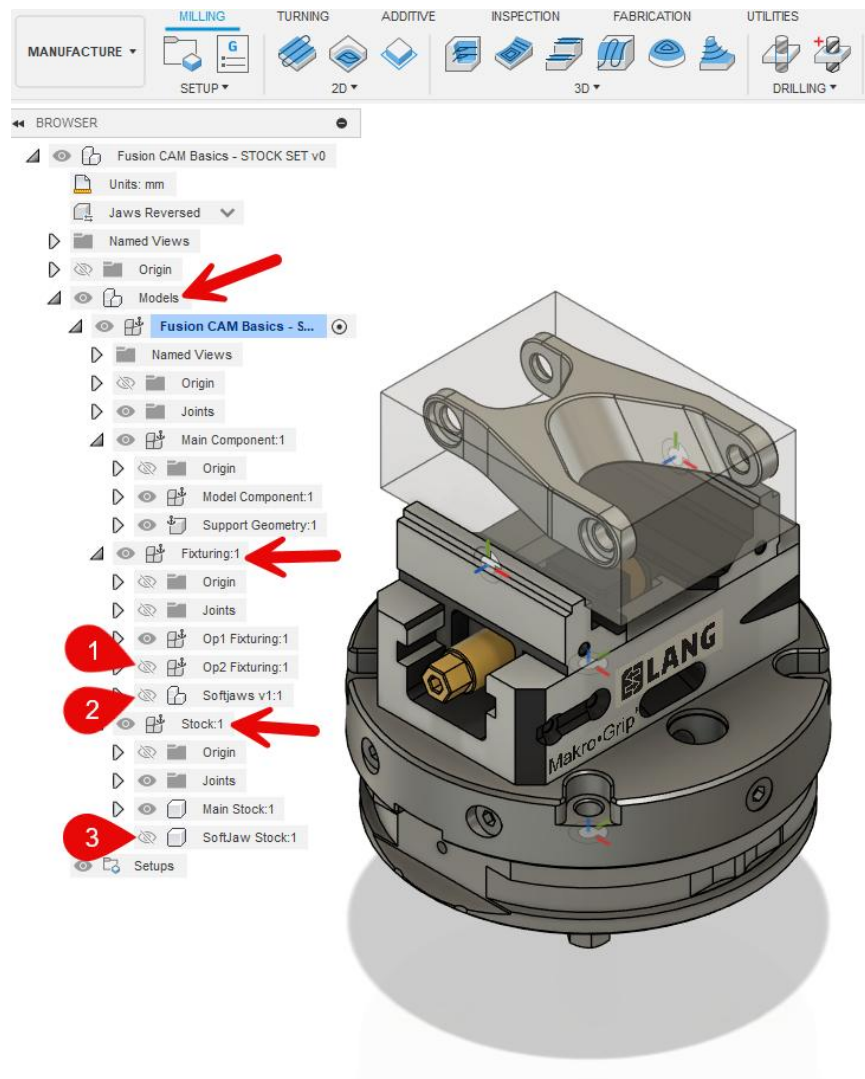


Set Component visibility

Once the document is open, you will find yourself in the Manufacturing Workspace, with all the Components visible.

1. Expand out the **Models** node in the browser
2. Under the **Fixturing** component you will need to turn off the visibility of the
 - a. **Op2 Fixturing** and
 - b. **Softjaws** components.
3. Then under the **Stock** component, turn off the **SoftJaw Stock** component.

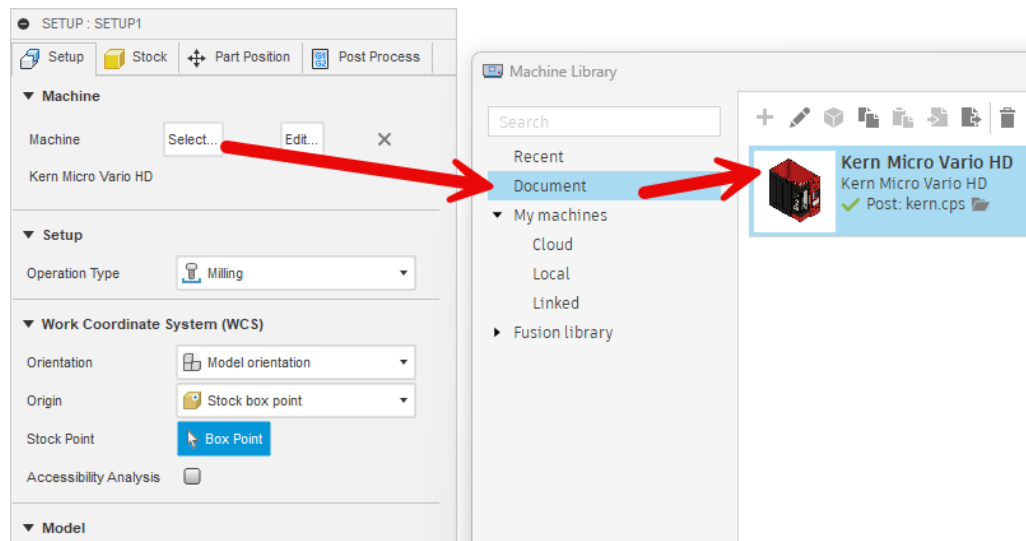
Your result should look like the image below.



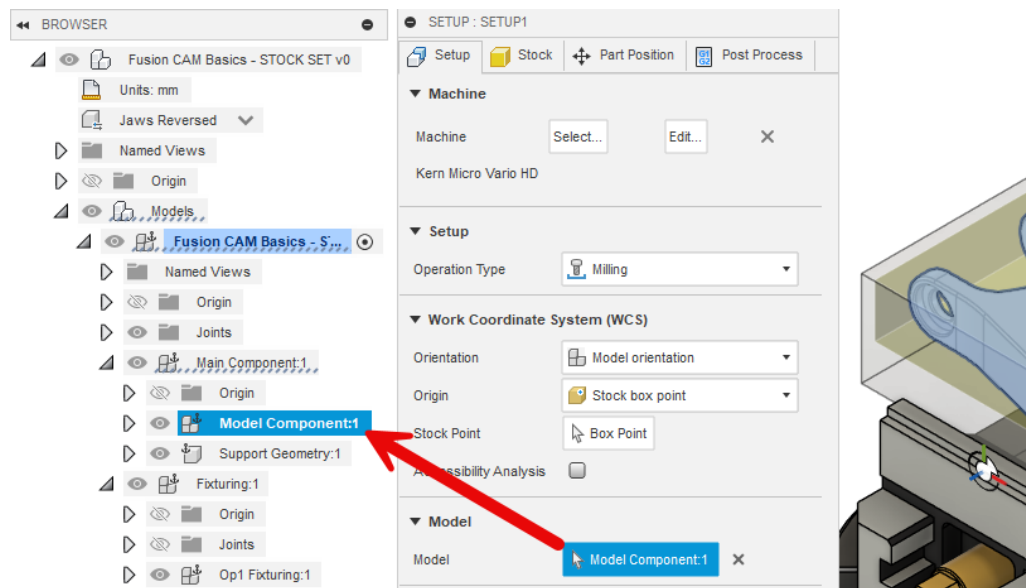
Create your Setup

From the **Setup** panel drop-down on the ribbon, select **New Setup**. Focussing on the Setup tab to begin with:

1. Select the Kern Micro Vario HD machine configuration (it may already be selected automatically, since it's the only machine config in the document)

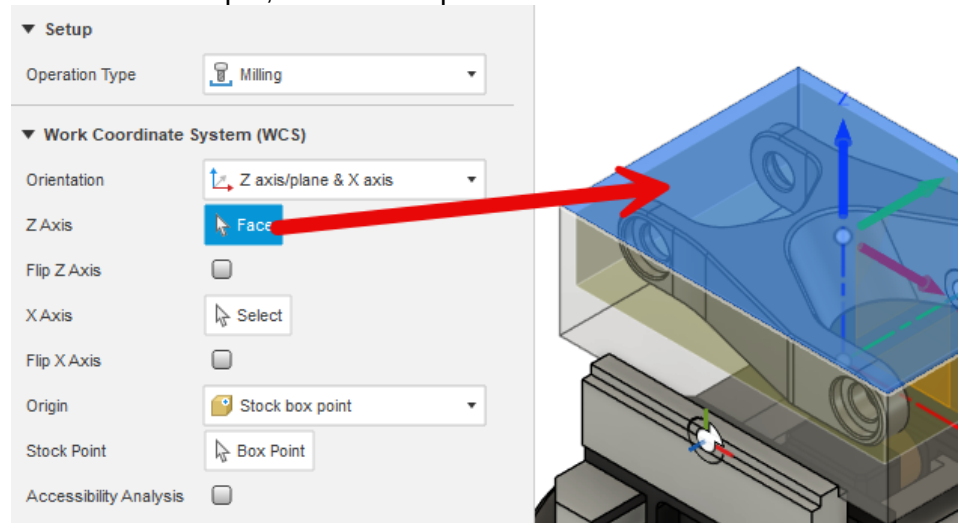


2. Make sure **Operation Type** is set to Milling.
3. Because the yellow stock body is currently dominating the view, let's jump ahead to the **Model** group and select the "Model Component" from the browser (it's nested under "Main Component")

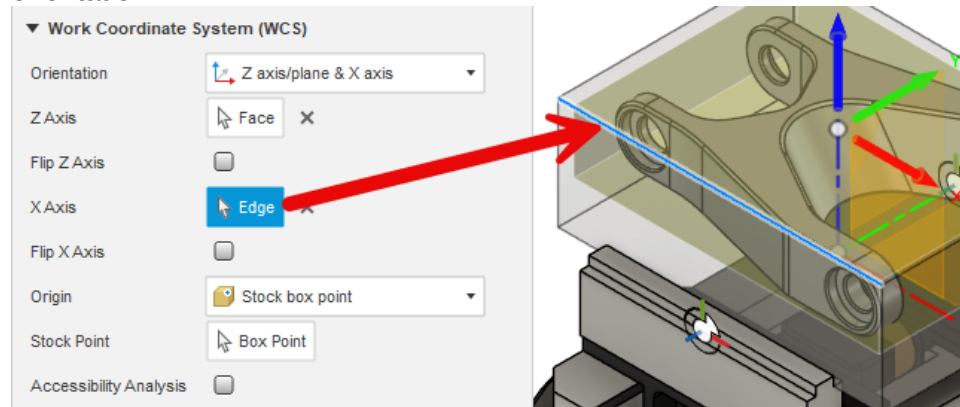


4. Moving back up to the **Work Coordinate System (WCS)** group, what you set here is very important. It's the first place you can make a mistake that may result in crashing your machine. Because of the way this template has been created, along with the use of zero point workholding on the machine, the position and alignment of the WCS is perfect by default. However, that won't be the case for the majority of situations until you have your template game and on machine workflows dialled. Just so you understand the fundamentals let's set it the long way.

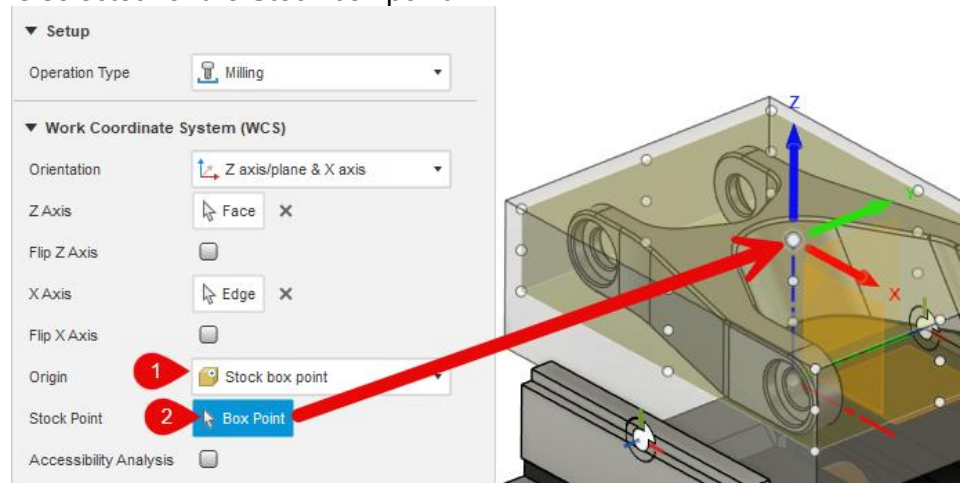
- a. Change **Orientation** to **Z axis/plane & X axis**
- b. For the Z Axis input, select the top face of the stock



- c. Next select the top front edge of the stock model to set the X axis orientation

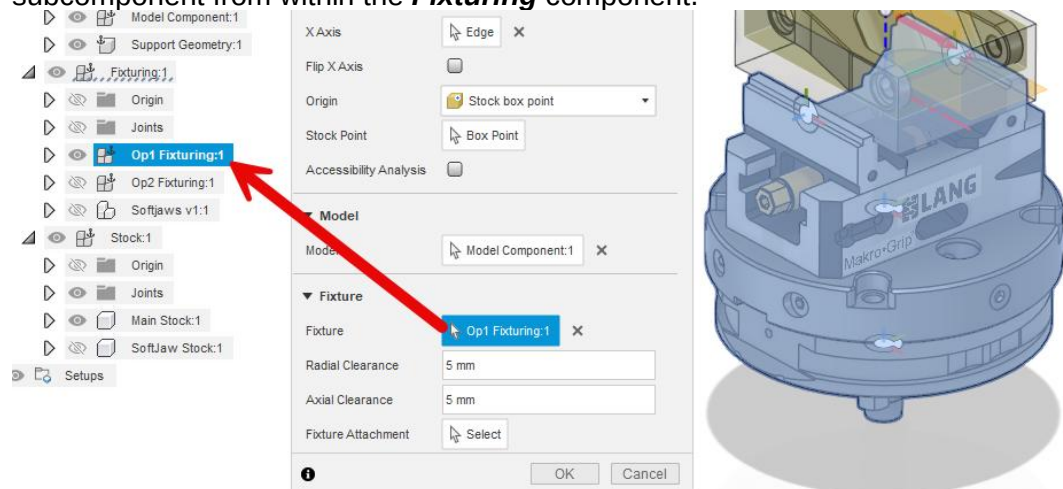


- d. The final step for defining the WCS is to make sure the top center node is selected for the Stock box point.



The node you choose here, is a composite of the measurable references for each of the 3 linear axes. In this case we will be probing all 4 vertical faces of the stock to find the X & Y axis zero points, and the top face to find the Z zero for the WCS. Those inputs result in the WCS being in the center of the stock's top face.

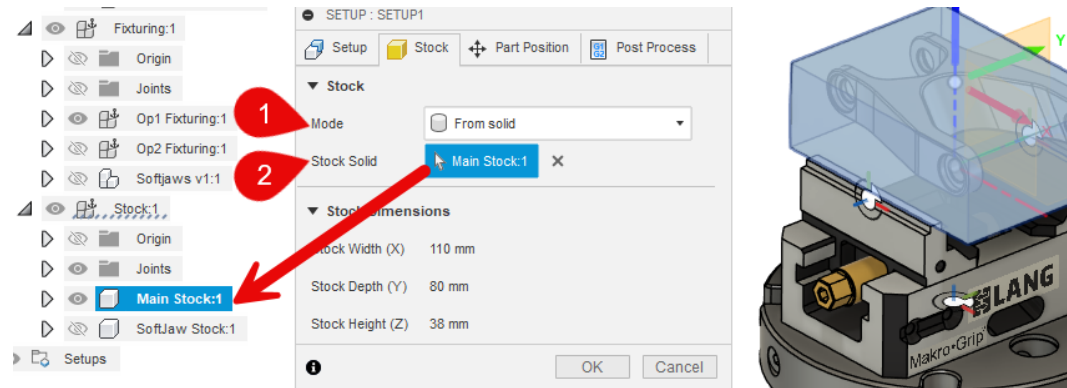
5. The last section of the Setup tab is optional, but strongly recommended. In recent months Fusion has been blessed with a swath of updates which make it's toolpaths far more 'fixture aware' than it was previously. However, all of those new goodies rely on including a low fidelity, but accurate, model of your fixtures in your Setup. This is the 1st Setup for this part, so select the **Op1 Fixturing** subcomponent from within the **Fixturing** component.



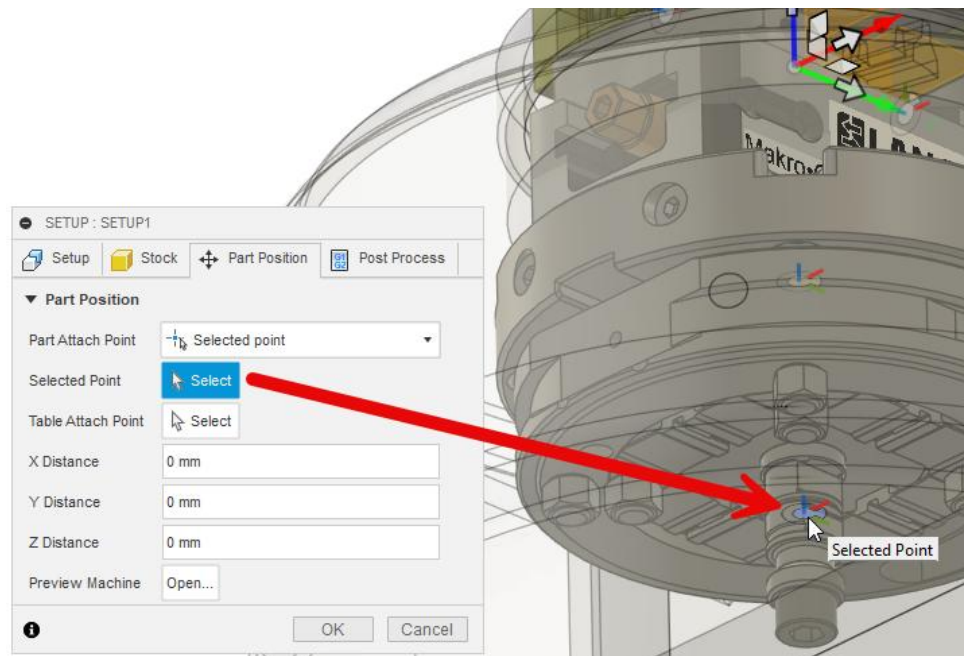
Leave all the clearance values and fixture attachment inputs in their default state. It's worth noting that the fixture's radial and axial clearance values become the default avoidance distance within the Geometry tab of all toolpaths within this Setup.

6. Switch to the **Stock** tab. By default, Fusion will 'shrinkwrap' a bounding box of 'graphical' stock around the model object you selected on the Setup tab. In this

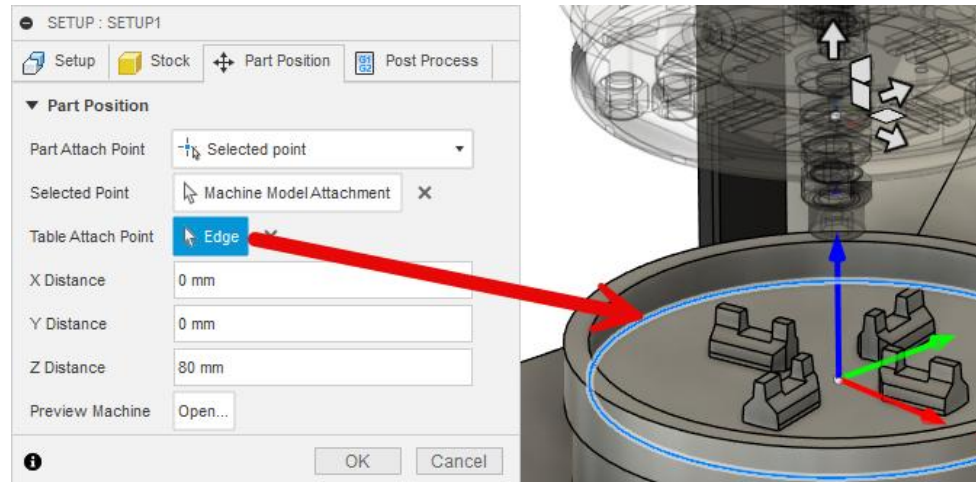
case, the stock is already modelled as part of this CAM template document, switch the **Mode** to **From solid** then select the “Main Stock” subcomponent from the Stock component in the browser.



7. Switching to the **Part Position** tab, you will position the setup onto the machine.
 - a. Change the **Part Attach Point** to **Selected point**
 - b. Orbit your model so you can look underneath your fixture, then select the lowest Joint Origin for the **Selected Point** input.



- c. Orbit back to look down onto the machine table and select the lower inner circular edge of the table as input for the **Table Attach Point**.



In the image above, the Z distance is temporarily set to 80mm so you can clearly see the edge that needs to be selected. But make sure the Z distance is set to 0mm before moving onto the Post Process tab.

8. On the **Post Process** tab, leave everything as default apart from the **Program Comment** field. Enter “My First 5-axis Program”.
9. Click OK to complete your Setup.

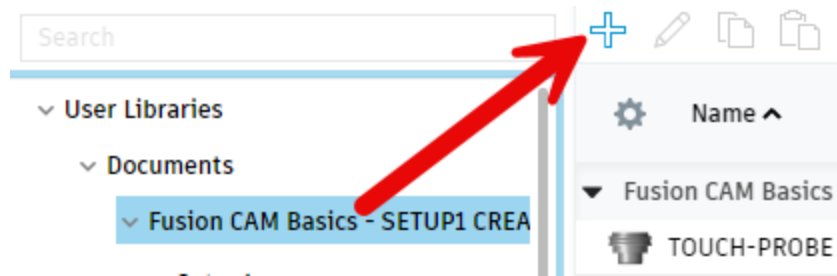
Defining your tools

To machine a part, we need tools. Fusion has a tool library feature, where CAM programmers and machinists can build out sets of tools of different types. They can organise their tools based on the materials they’re capable of machining, as well as store presets for:

- cutting feed rates
- spindle speeds
- axial depth of cut
- radial depth of cut

Presets allow programmers to confidently apply a cutting formula to suit the type of cut they will be performing with the tool for the current toolpath being programmed.

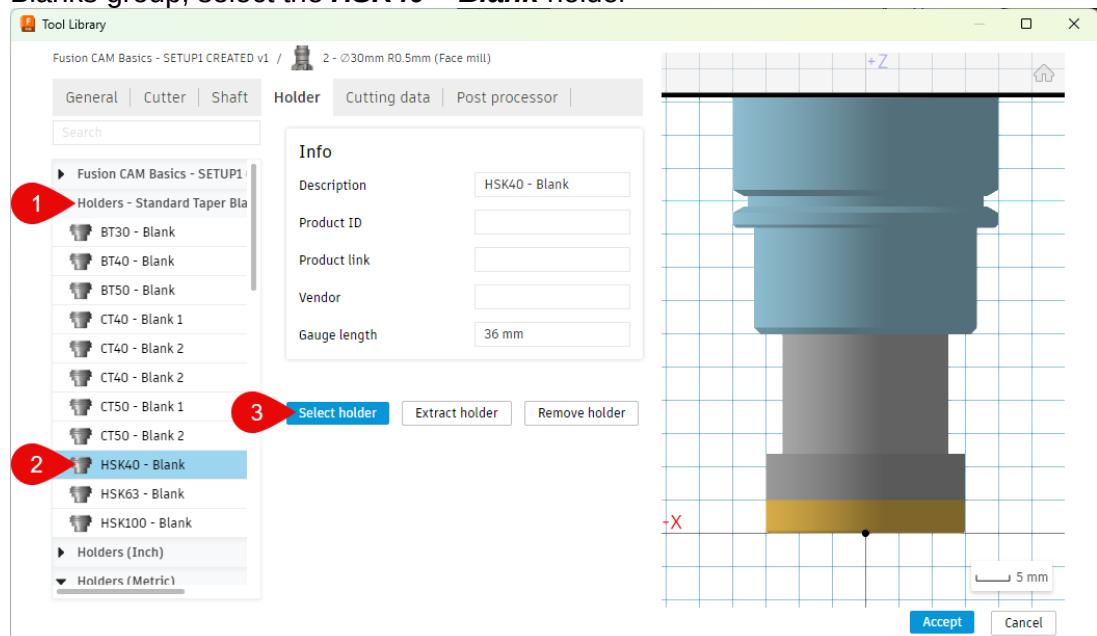
1. From the **Manage** panel drop-down on the ribbon, launch the **Tool Library**. In the left pane, you will see a list of libraries. For this exercise, you’re only going to create tools within your current document. Select the **Fusion CAM Basics** document and you will see the + icon on the toolbar will become active:



2. Click the blue + icon and select the **Face mill** option from the Milling group.
3. Normally you would diligently fill out the details on the **General** tab, but we will skip that for brevity. So switch straight to the **Cutter** tab.
4. Working from the top to the bottom of this dialog, set the following values:
 - a. **Number of flutes** = 5
 - b. **Diameter** = 30mm
 - c. **Shaft diameter** = 25mm
 - d. **Length below holder** = 30mm
 - e. **Shoulder length** = 12mm
 - f. **Flute length** = 5mm
 - g. **Corner radius** = 0.5mm

Note the **Gauge length** is currently equal to 30mm, which is representative of the Length below holder dimension we specified. Once we add a holder, this value will increase.

5. Skip the Shaft tab and go straight to the **Holder** tab. From the Standard Taper Blanks group, select the **HSK40 – Blank** holder



If you go back to the Cutter tab, you will notice the Gauge length value is now equal to 66mm.

6. Moving to the **Cutting data** tab, we can setup a couple of presets. Right-click on the **Default Preset** and rename it to “Mild Steel – Rough”. Then set the following values:
- Spindle speed** = 1500 rpm
 - Feed per tooth** = 0.15mm
 - Ramp feedrate** = 400 mm/min
 - Plunge feedrate** = 300 mm/min
 - Enable **Use stepdown** and set **Stepdown** to 1mm
 - Enable **Use stepover** and set **Stepover** to “tool_diameter * 0.45” / 45% of the tool diameter.

Fusion CAM Basics - SETUP1 CREATED v1 / 2 - Ø30mm R0.5mm (Face mill)

General
Cutter
Shaft
Holder
Cutting data
Post processor

+
edit
undo
redo
trash

Cutting data

Mild Steel - Rough
Mild Steel - Finish

Speed

Spindle speed
1500 rpm

Surface speed
141.37167 m/min
fx

Ramp spindle speed
1500 rpm
fx

Feedrates

Cutting feedrate
1125 mm/min
fx

Feed per tooth
0.15 mm

Lead-in feedrate
1125 mm/min
fx

Lead-out feedrate
1125 mm/min
fx

Transition feedrate
1125 mm/min
fx

Ramp feedrate
400 mm/min

Vertical feedrates

Plunge feedrate
300 mm/min
fx

Plunge feed per revolution
0.2 mm

Passes and linking

Use stepdown
☒

Stepdown
1 mm

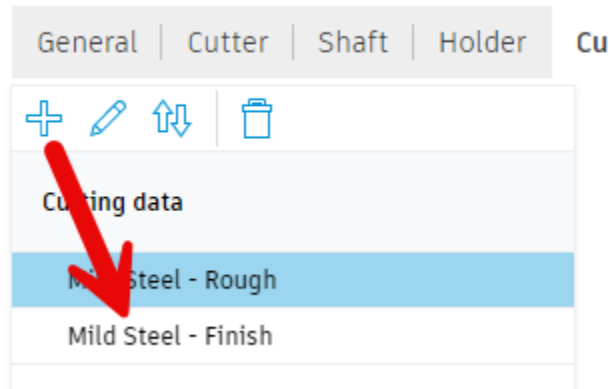
Use stepover
☒

Stepover
13.5 mm
fx

Coolant
Flood

7. Click the blue + icon on the left of the Cutting data tab to create a 2nd cutting preset. Rename the preset to “Mild Steel – Finish”. Using a consistent naming convention across all your tools and libraries is recommended.

Fusion CAM Basics - SETUP1 CREATED v1 /  2 - Ø30



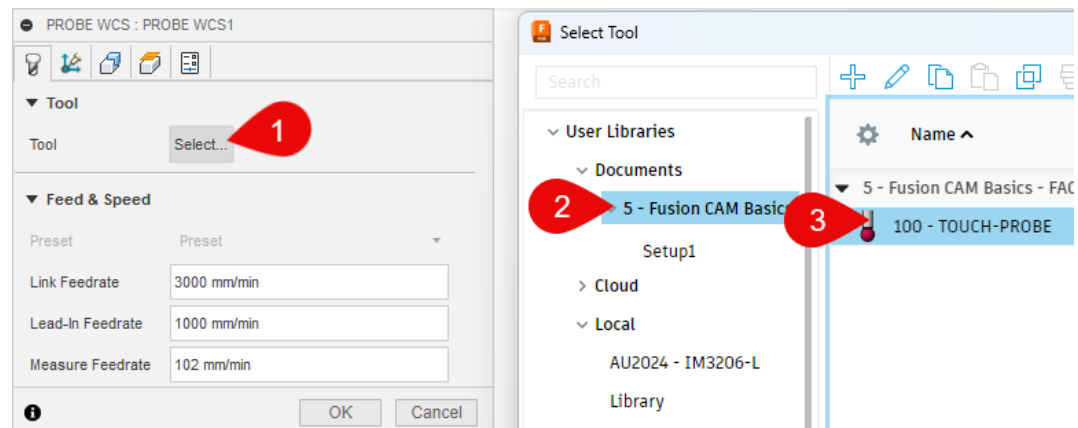
All the settings from the first preset will be copied over. So the following values will need to be changed to suit a finishing scenario:

- a. **Spindle speed** = 2000 rpm
 - b. **Feed per tooth** = 0.08mm
 - c. **Ramp feedrate** = 400 mm/min
 - d. **Plunge feedrate** = 220 mm/min
 - e. Enable **Use stepdown** and set **Stepdown** to 0.2mm
 - f. Enable **Use stepover** and set **Stepover** to “tool_diameter * 0.7” / 70% of the tool diameter.
8. The **Post processor** tab is where you specify the tool number and offset numbers for the CNC control to correctly select the tool from it’s tool changer, then apply the correct length and diameter offsets. It will use this information to correctly position the tool relative to the geometry you want to cut. Leave the default values in place and click **Accept**.

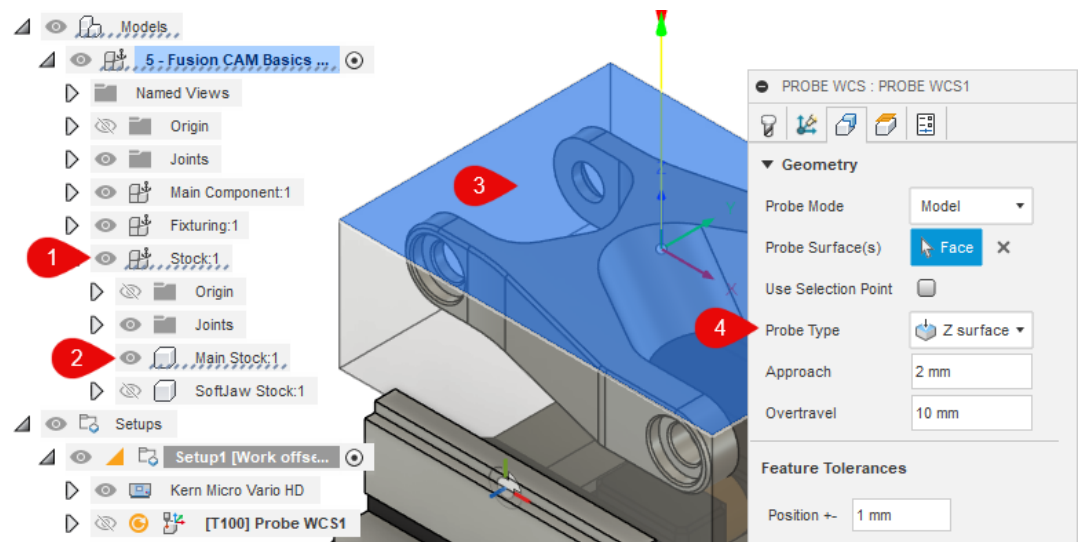
Probing

After the machinist loads the stock into the vise, they need to check and set the location of the stock. You can use Fusion's Probe WCS command to achieve this.

1. From the **Setup** drop-down on the ribbon, select **Probe WCS**
2. Select the TOUCH-PROBE tool from the document library

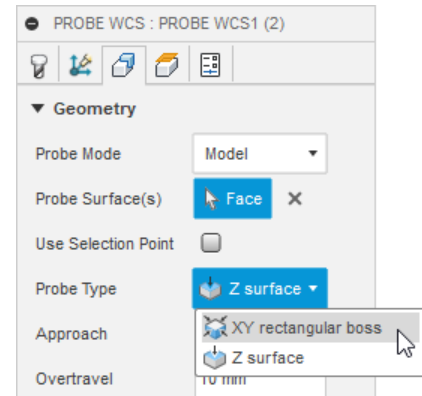


3. On the **Geometry** tab, make sure the **Probe Mode** is set to **Model**.



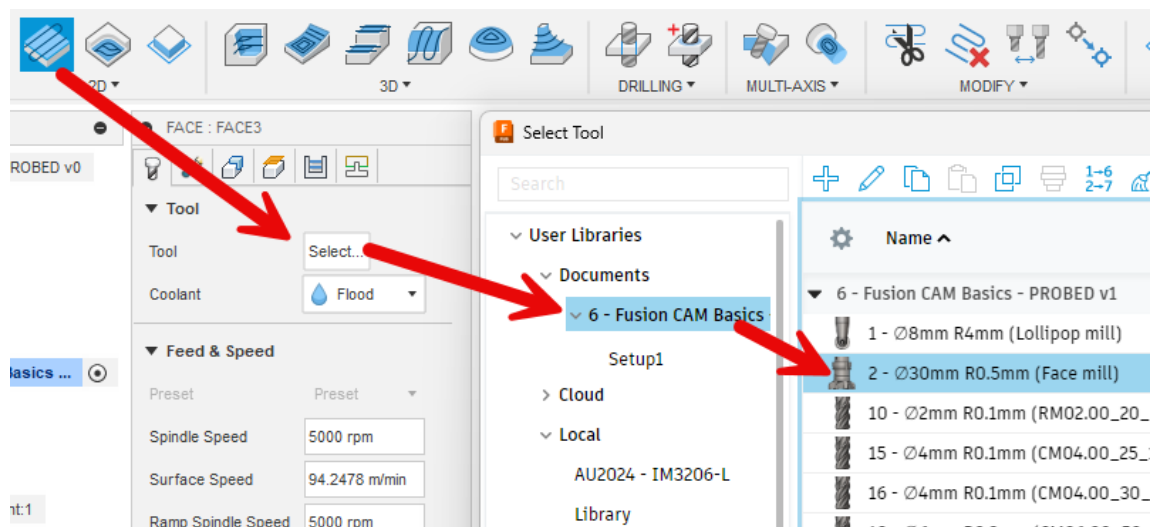
- a. Turn on the visibility of the "Stock" Component, followed by the "Main Stock" component
- b. Select the top face of the stock as the **Probe Surface**
- c. Change the **Probe Type** setting to **Z surface**
4. Click OK

- Right-click on the probe toolpath and select Duplicate (or select the toolpath and use Ctrl + D).
- Edit the new toolpath and change the **Probe Type** to **XY rectangular boss**.
- Click **OK**
- Right-click and rename the toolpaths to “Probe WCS Z” and “Probe WCS XY” respectively.
- Turn off visibility for the “Main Stock” component so it doesn’t get in the way of upcoming toolpath selections.



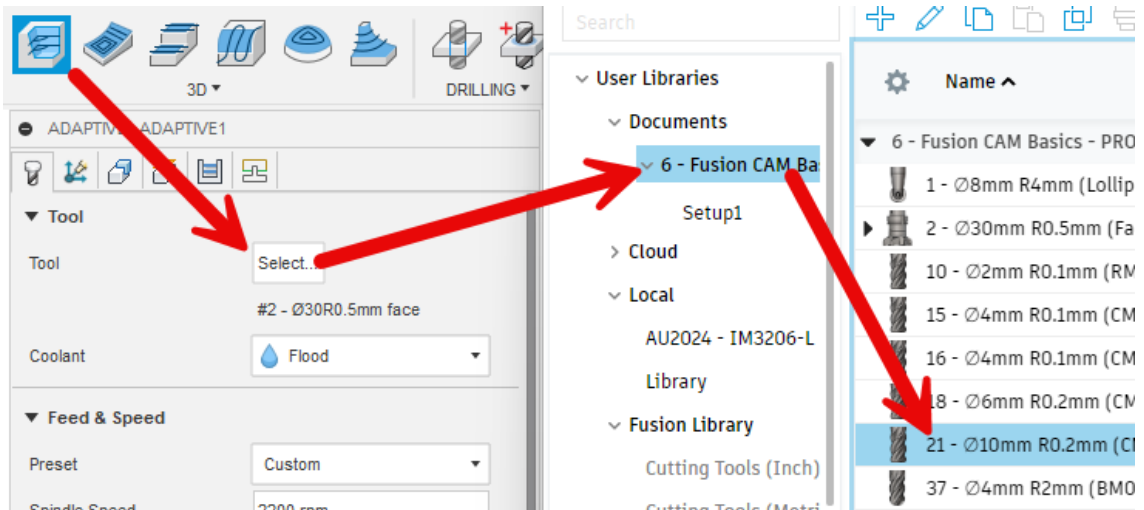
Applying toolpaths

Facing



- Select **Face** from the 2D Milling panel.
- Choose the 30mm diameter face mill you defined earlier from your document library
- Click **OK** and review the resulting toolpath. Feel free to simulate if you have time.
- The cut is quite big since it is a single pass by default. Right-click and **Edit** the toolpath.
 - On the **Passes** tab, enable **Multiple Depths** and **Stock to Leave**. Leave all the settings as default.
- Create a new Face toolpath, notice it will reuse the same tool, so you don’t need to reselect it.
- On the **Tool** tab, within the **Feed & Speed** group, set the “Mild Steel – Finish” **Preset**.
- Click OK to create the final finishing face toolpath.

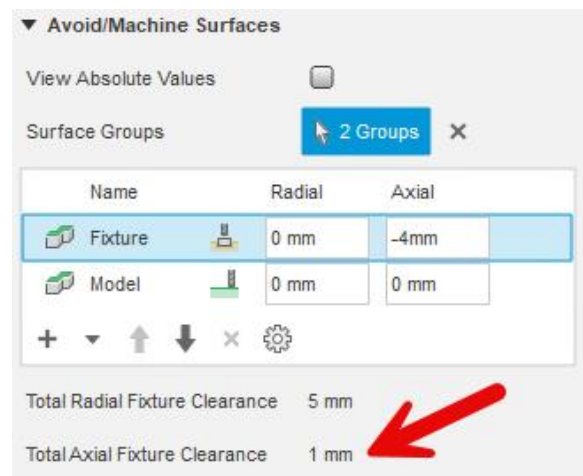
3D Adaptive



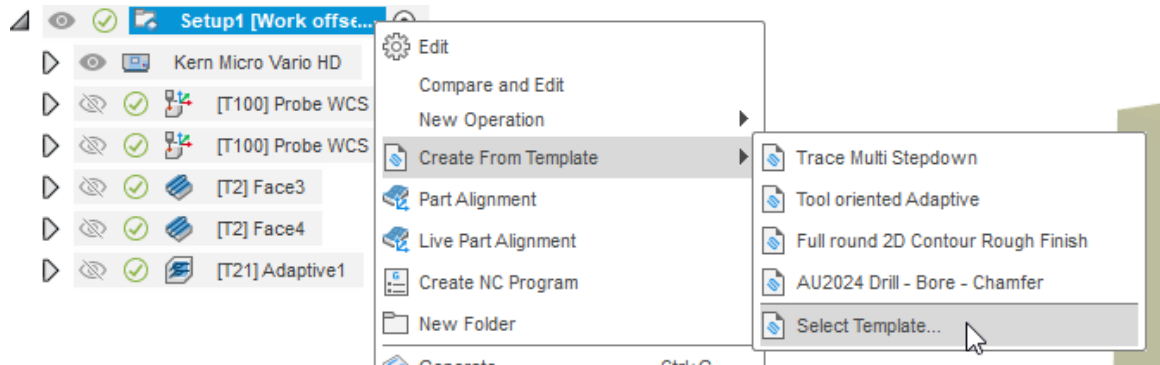
1. Select **Adaptive Clearing** from the 3D Milling panel.
2. Choose tool 21, a 10mm diameter bullnose tool
3. Set the “Aluminum – Adaptive” Preset on the Tool tab.
4. Since this tool has a 0.2mm corner radius, on the **Geometry** tab, you need to set the **Ignore Stock Less Than** parameter to 0.2mm.
5. Click OK to generate
6. Study the stock result

We need to remove as much material as possible here. The goal is to remove all the material needed, so you can access as many features as possible in a single setup. But, some material needs to remain for rigidity purposes. In this case, we need to avoid the vise jaws, but remove all material in the Z on the internal features of the part so we have the required clearances for upcoming toolpaths.

7. Edit the Adaptive toolpath, on the **Geometry** tab, under the **Avoid/Machine Surfaces** group, set the **Axial** clearance for the **Fixture** surface group to -4mm
8. Switch to the **Heights** tab. Then change the following settings in the **Bottom Height** group:
 - a. Enable **Stock Bottom**
 - b. Add a -1mm **Offset**.
9. Click **OK**

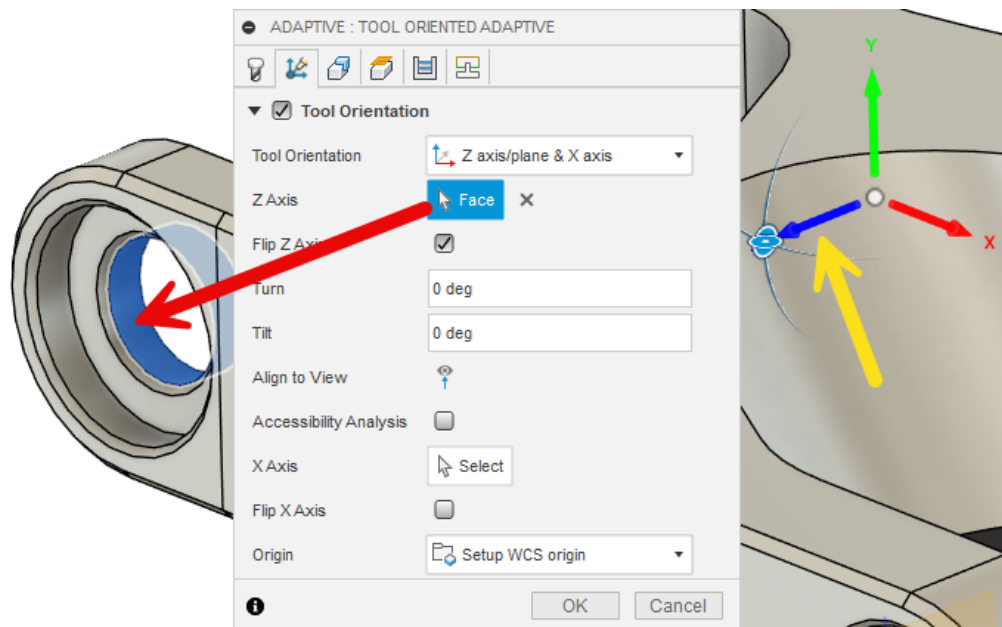


We can use 3D Adaptive from a couple of other orientations to help improve access for future toolpaths. This time you can use a pre-created template which will help save some time.

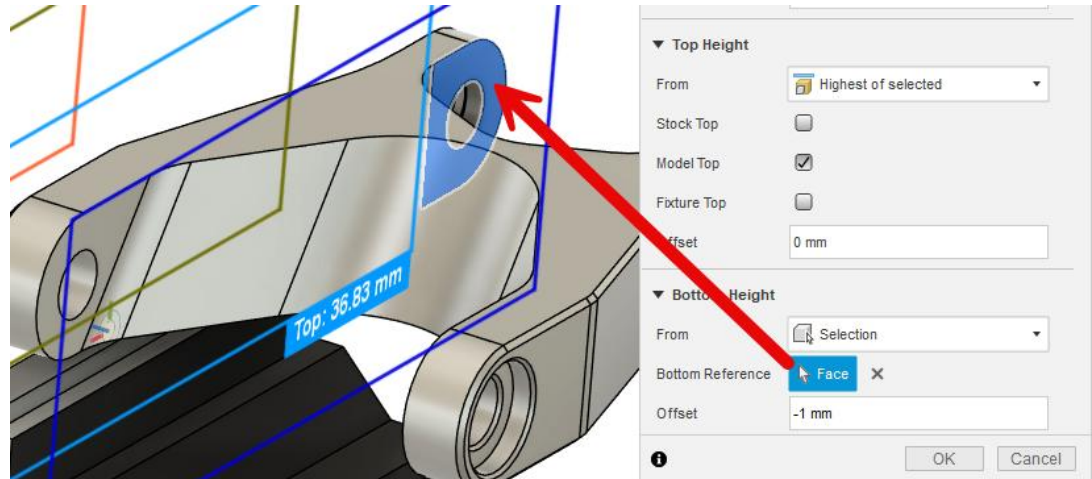


1. Right-click on your setup, then from the **Create from Template** flyout, click **Select Template...**
2. Navigate to the **Local** library folder, highlight "Tool oriented Adaptive" then click **Select**. This will insert two new Adaptive Clearing toolpaths at the end of your setup.
3. Double click to edit the first of the two toolpaths.
4. Switch to the **Multi-Axis** tab and set the **Z Axis** input to one of the hole faces.

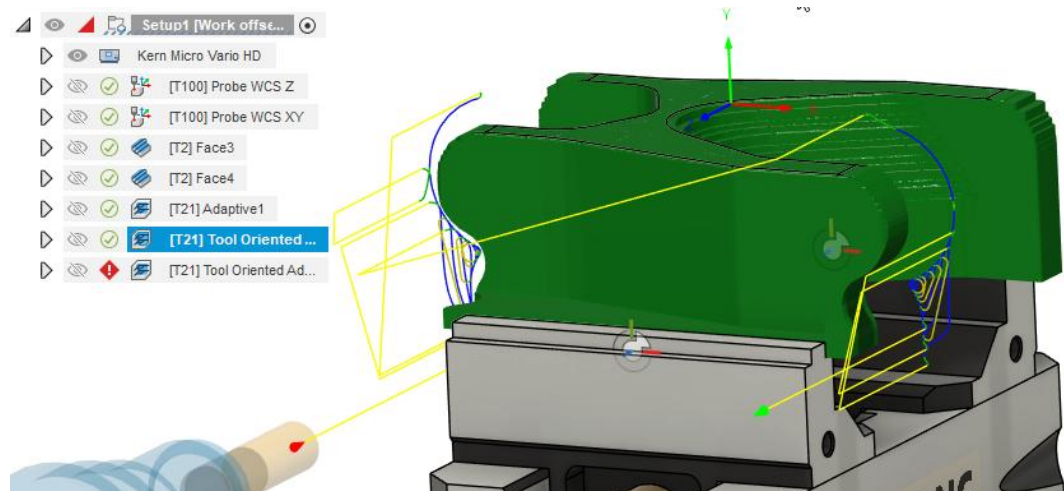
Pay attention to the Z axis (blue arrow) direction, it should be pointing towards the bottom of the tool. Use **Flip Z Axis** if it isn't. The machine will position it's two rotaries so this blue arrow will be pointing to the sky when it machines this operation.



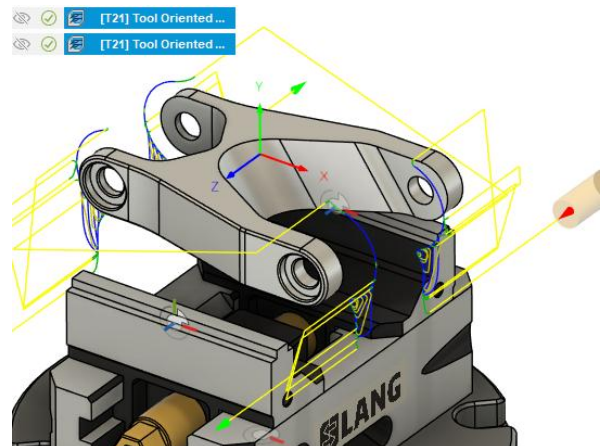
- Switch to the **Heights** tab, within the **Bottom Height** group, select the inner face of the narrowest pivot. You may have to orbit your model to see it for selection.



- Click OK to generate the toolpath. The result should look something like this:



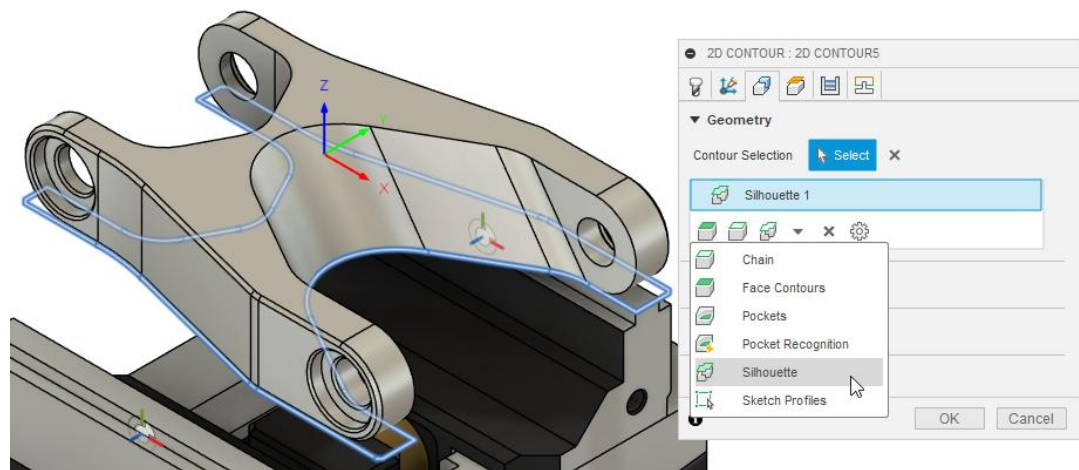
- Now repeat the same thing for the opposing side. Select the same face for the Z Axis input. The **Flip Z Axis** setting is already flipped in the template, compared to the first one. The bottom height reference will be the opposing face to the first one.



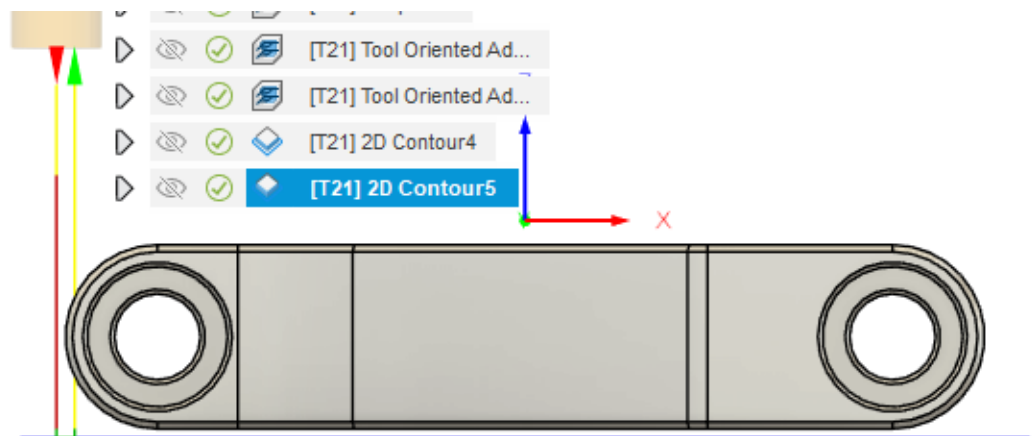
Profile with 2D Contour

Now you need to clean up the periphery of the part. Fusion has various contour selection options at your disposal. This part has full round fillets, which means there is no clear edge chain to select to machine out the overall shape of the part.

1. From the **2D Milling** drop-down on the ribbon, select **2D Contour**.
2. Use the same tool again, you will see it's already selected.
3. Switch to the **Geometry** tab, from the Geometry Selections drop-down, select **Silhouette**. Accept all the default settings in the resulting dialog and click **OK**.

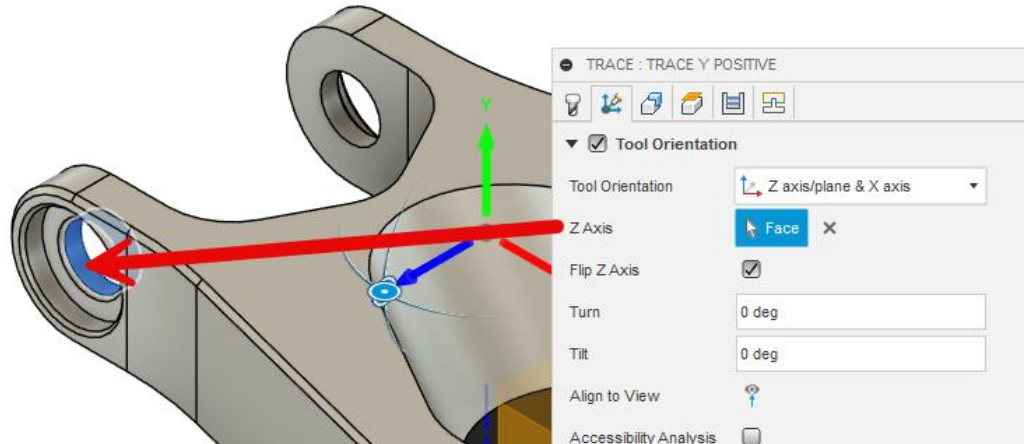


4. Switch to the **Heights** tab, then apply a -1mm **Offset** to the **Bottom Height**. This will make sure the tool fully finishes the vertical walls of the part. As you can see in the image below, the resulting toolpath is sitting 1mm below the bottom of the part. You may need to press F8 to toggle the view of the in process stock.

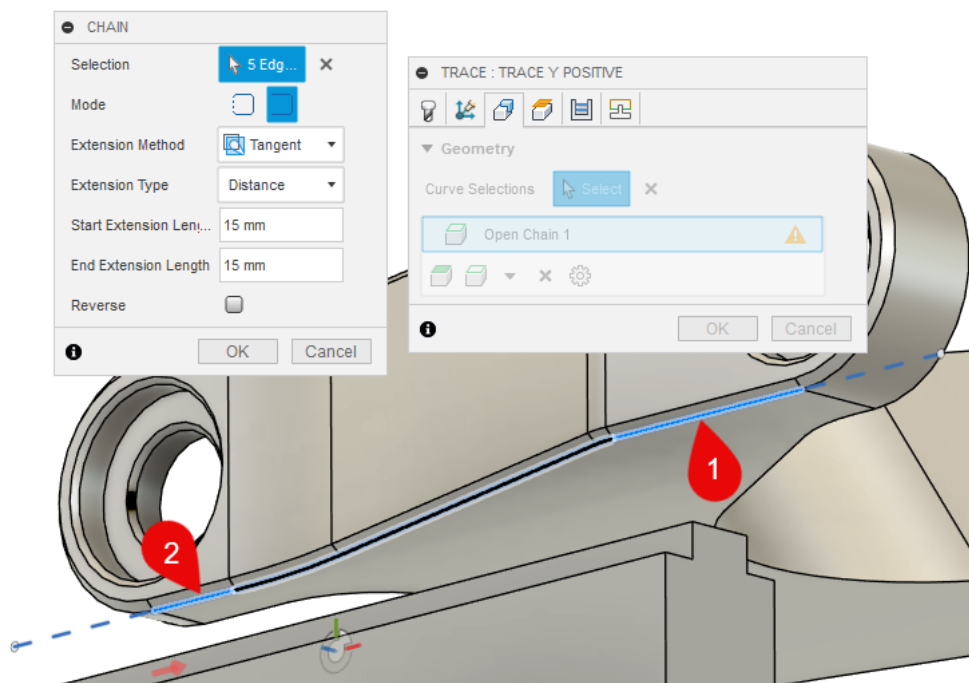


Trace

1. Right-click on the setup, from the **Create from Template** flyout, click **Select Template...** then choose the “Trace Multi Stepdown” template from the local library and click **Select**.
2. Double click to edit the first Trace toolpath. Switch to the **Multi-Axis** tab and select the same hole face as earlier to satisfy the **Z Axis** input.



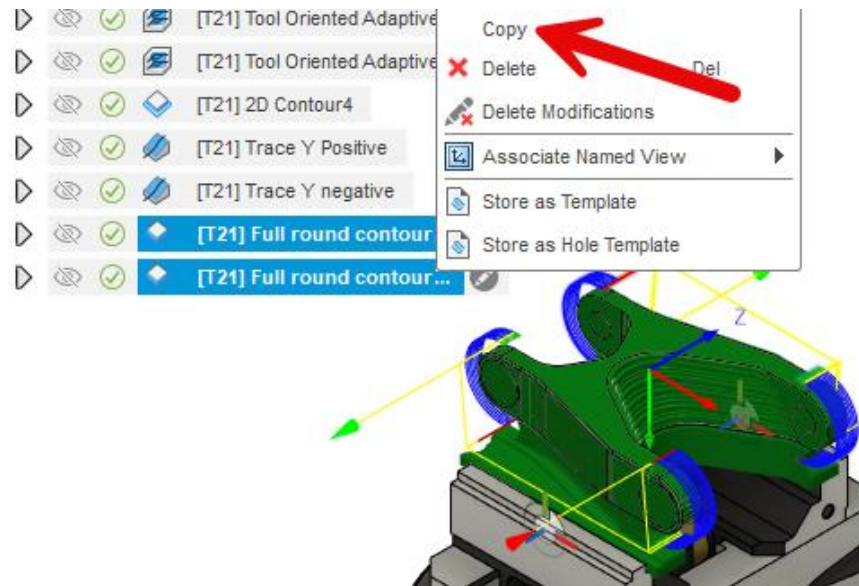
3. Moving onto the **Geometry** tab, you will see there is an existing **Open Chain**, with a warning symbol. Right-click on it and select **Edit**. Notice some of the inputs have been preset, but the dialog requires a selection to be satisfied. Select the two blue highlighted edges shown below, then Fusion will auto-chain the edges between them.



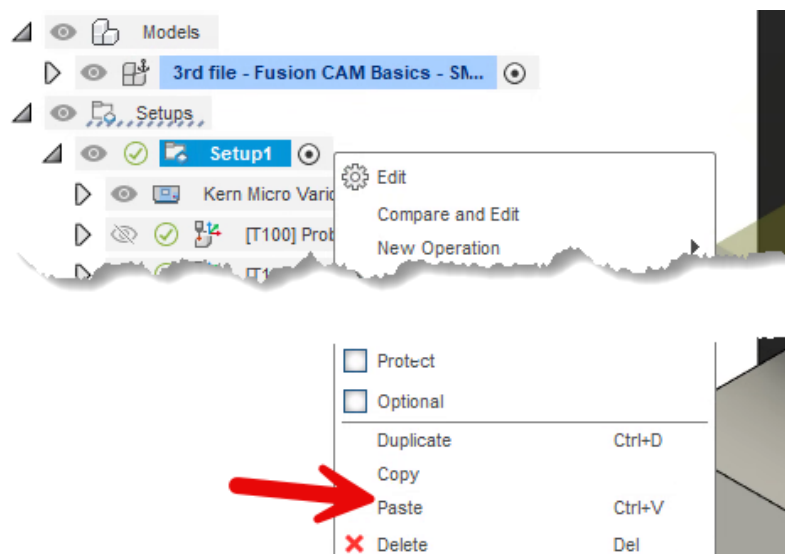
4. Click **OK** on both dialogs to generate the toolpath.
5. Repeat the same process for the opposite side of the part.

2D Contour Full Round

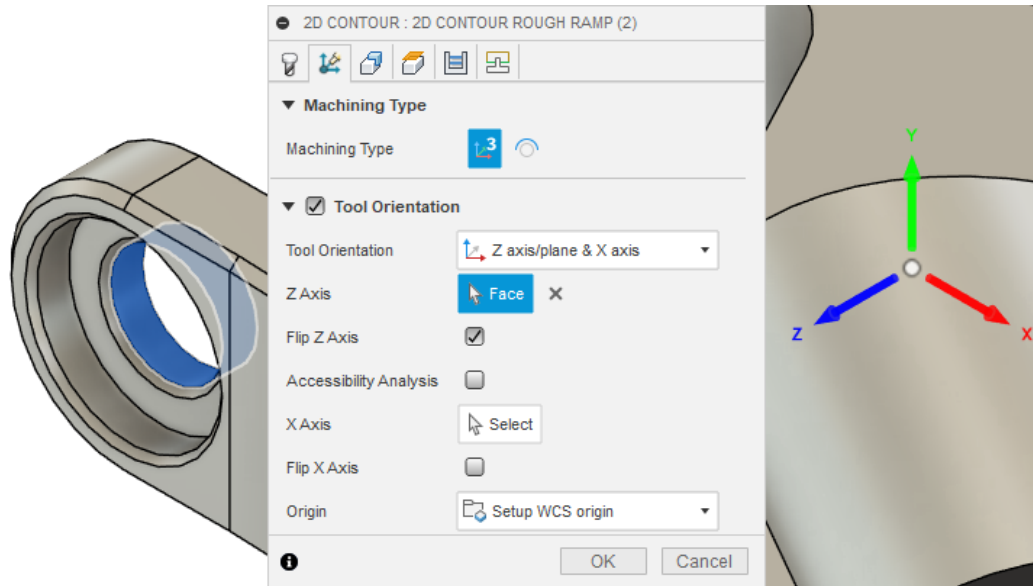
1. Open the “4th file - Full round template” document.
2. Shift select the two Full round contour.. toolpaths at the end of the browser, right-click and select **Copy**.



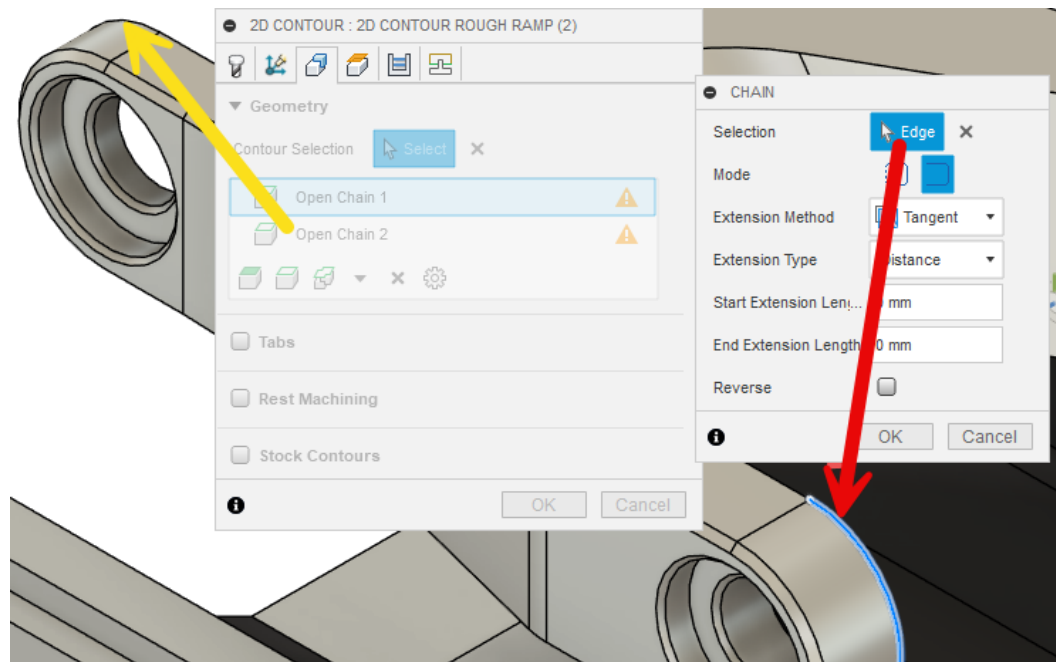
3. Switch back to the “3rd file – Fusion CAM Basics” document
4. Right-click on the setup and select Paste.



5. Edit the first of the pasted toolpaths and switch to the **Multi-Axis** tab.
 - a. Select the cylindrical hole face and make sure the blue Z Axis arrow is pointing in the direction shown below.

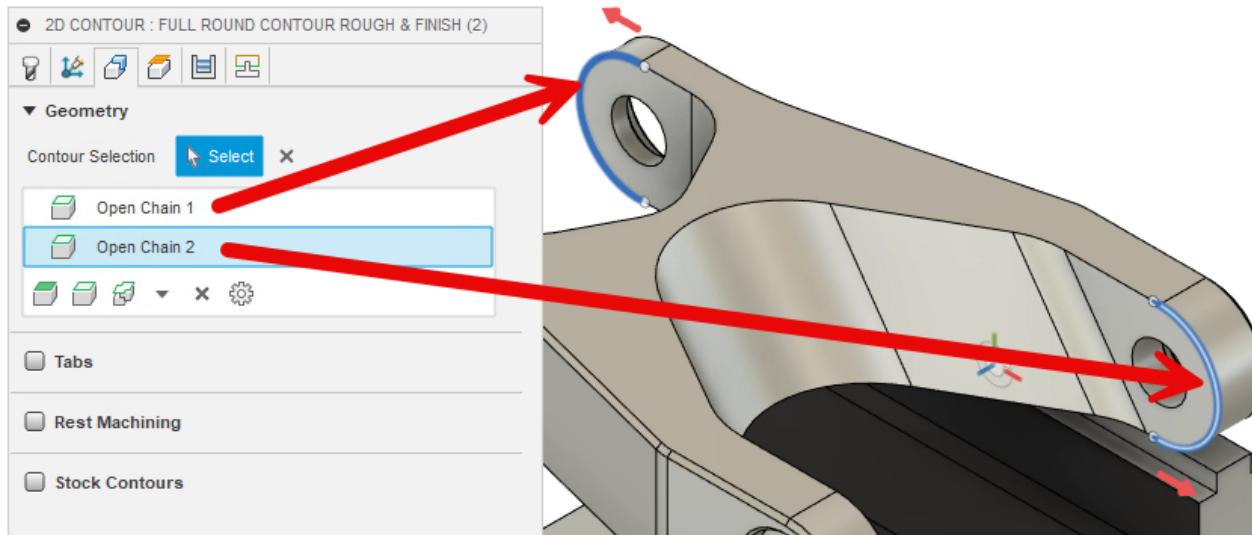


6. Right-click on "Open Chain 1" to edit it, then select the highlighted edge chain in the image below (red arrow). Then click **OK**.

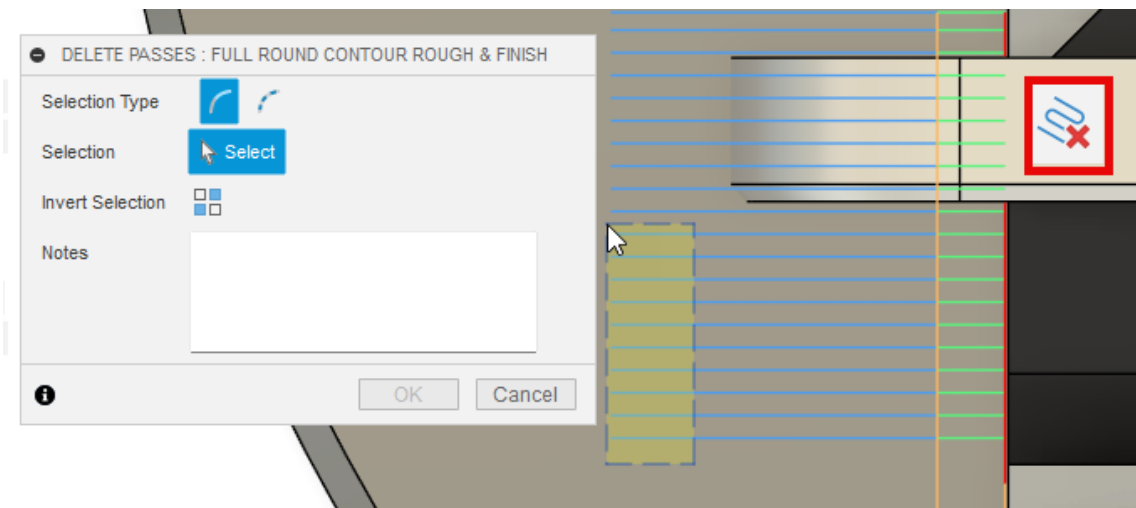


7. Repeat for "Open Chain 2", but select the edge the yellow arrow is pointing at (above). Click **OK** to generate the toolpath.

8. Edit the 2nd toolpath
9. On the **Multi-Axis** tab, select the same face you have been to set the tool orientation.
10. Moving onto the **Geometry** tab edit the open chains:
 - a. Edit **Open Chain 1** and select the edge shown below. Click **OK**. Repeat for **Open Chain 2**.

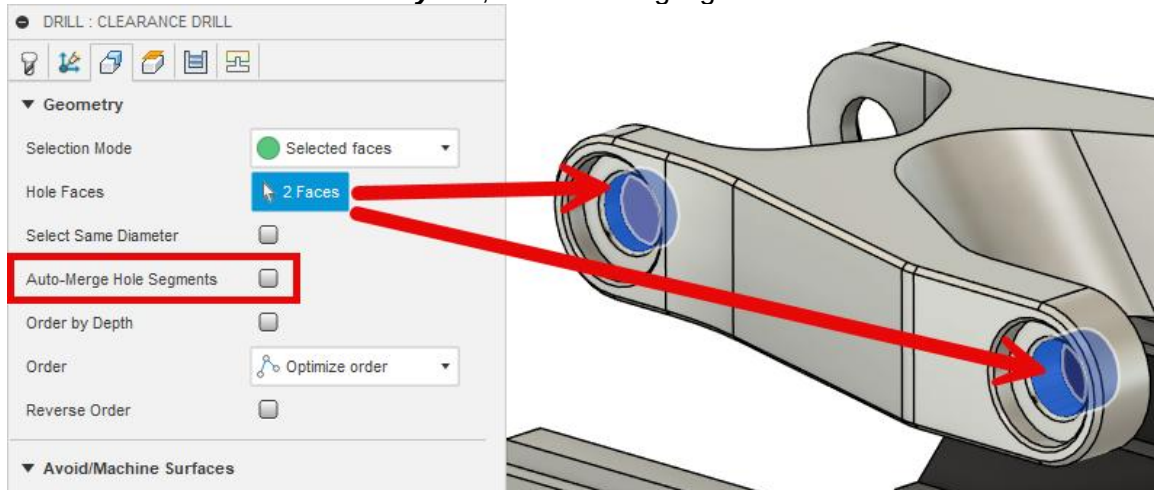


- b. Click **OK** on the toolpath dialog to generate the toolpath.
11. As a bonus step, one at a time, highlight each toolpath. Then use the **Delete Passes** command on the **Modify** panel, to delete all the passes which are air cutting on these toolpaths. The “right to left” window selection method is being used in the screenshot below to mass select the passes. It will select anything the window ‘touches’.

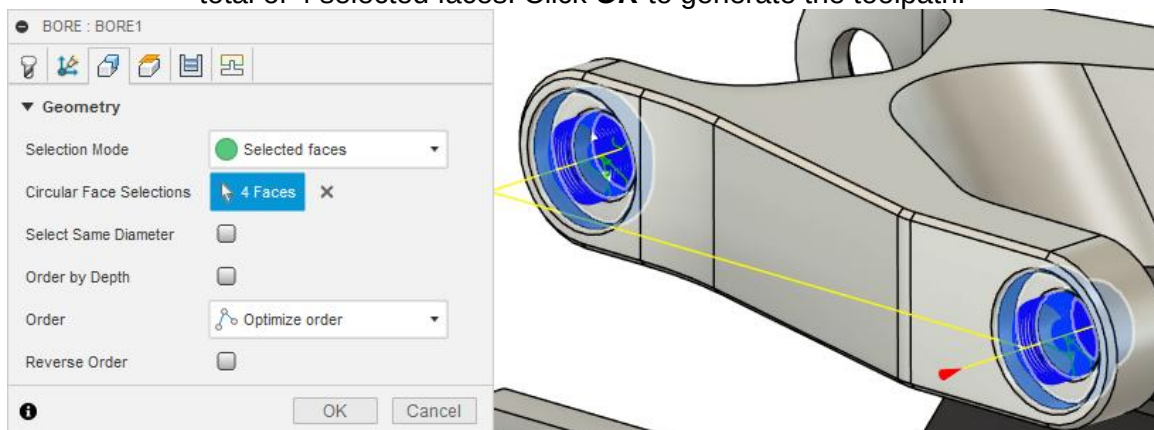


Holes

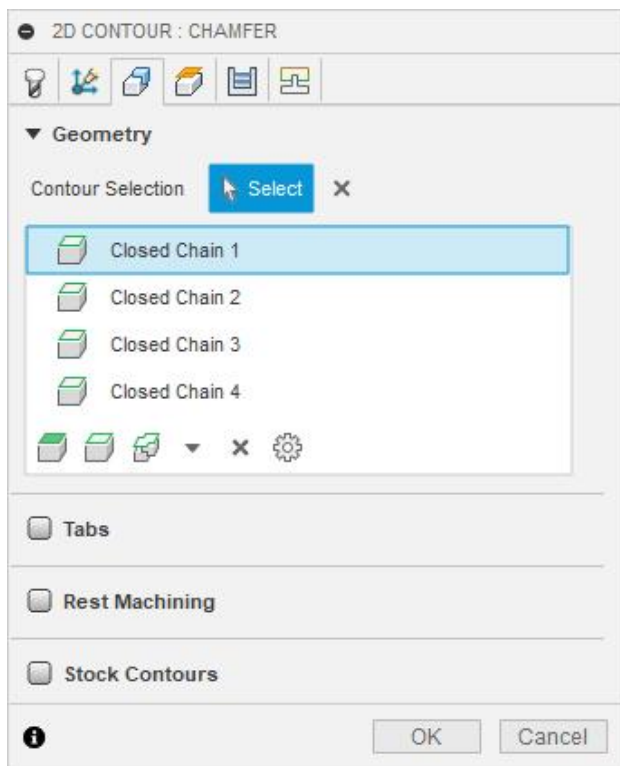
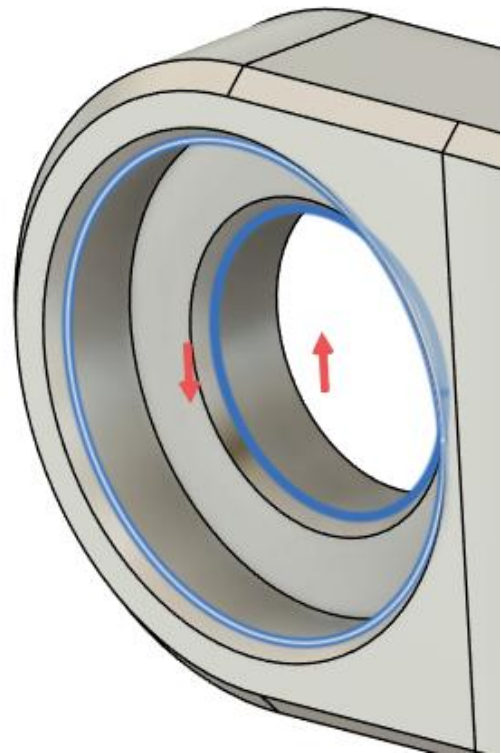
1. Right-click on the setup, from the **Create from Template** flyout, click **Select Template...** then choose the “AU2024 Drill - Bore - Chamfer” template from the local library and click **Select**.
2. Edit the “Clearance drill” toolpath:
 - a. Set the tool orientation in the same way you have for the previous ones.
 - b. On the **Geometry** tab, select the highlighted hole faces shown below:



- c. Enable **Auto-Merge Hole Segments**. This adds the additional concentric cylindrical faces to the hole definition, making sure the top of the hole is correct. Click **OK**.
3. Right-click and **Duplicate** the “Clearance drill” toolpath. Edit the new one:
 - a. Toggle **Flip Z Axis** on the **Multi-Axis** tab.
 - b. Clear the **Hole Faces** selection from the **Geometry** tab by clicking the X
 - c. Select the opposing hole faces to drill the other side of the part.
 - d. Click **OK**
 4. Edit the “Bore1” toolpath:
 - a. Set the tool orientation in the same way you have for the previous ones.
 - b. On the **Geometry** tab, select both cylindrical faces in both holes, for a total of 4 selected faces. Click **OK** to generate the toolpath.



5. Right-click and **Duplicate** the “Bore1” toolpath. Edit the new one:
 - a. Toggle **Flip Z Axis** on the **Multi-Axis** tab.
 - b. Clear the **Circular Face Selections** selection from the **Geometry** tab
 - c. Select the opposing faces to bore the holes on the other side of the part.
 - d. Click **OK** to generate.
6. Edit the “Chamfer” toolpath:
 - a. Set the tool orientation in the same way you have for the previous ones.
 - b. On the **Geometry** tab clear the **Contour Selection** by clicking the black X to the right of the input box
 - c. Select the bottom edges of each chamfer across both holes in line with this tool orientation. For a total of 4 closed chains:

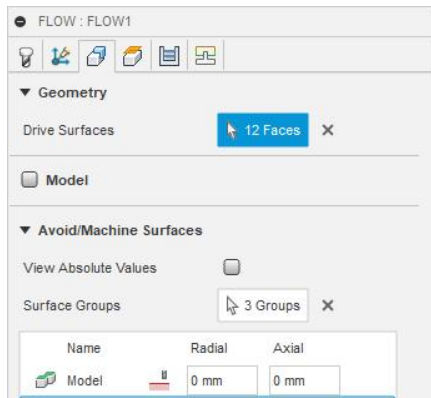
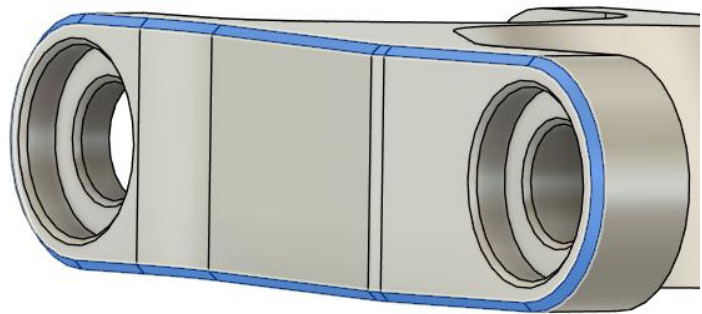


- d. Click **OK** to generate the toolpath.
7. Right-click and **Duplicate** the “Chamfer” toolpath. Edit the new one:
 - e. Toggle **Flip Z Axis** on the **Multi-Axis** tab.
 - f. Clear the **Contour Selection** selection from the **Geometry** tab
 - g. Select the 4 chamfer edges for the holes on the opposite side of the part.

Flow

Since the modelled chamfer on the side of the part is not planar, we can't use 2D Contour to machine it without the chamfer mill overcutting the part. As a result you will need to 3D surface this one.

1. From the **3D** panel on the ribbon, select **Flow** from the drop-down menu.
2. Select Tool #37 from the document library, it's a 4mm diameter ball end mill.
3. On the **Multi-Axis** tab, set the tool orientation again, in the same way you have previously. Remember to make sure the blue WCS arrow is pointing towards the bottom of the tool.
4. Moving to the **Geometry** tab, select the 12 faces which make up the 3D chamfer. If you accidentally select an incorrect face, just click it again to deselect it. Zoom in and pan to make it easier to select the smaller faces.



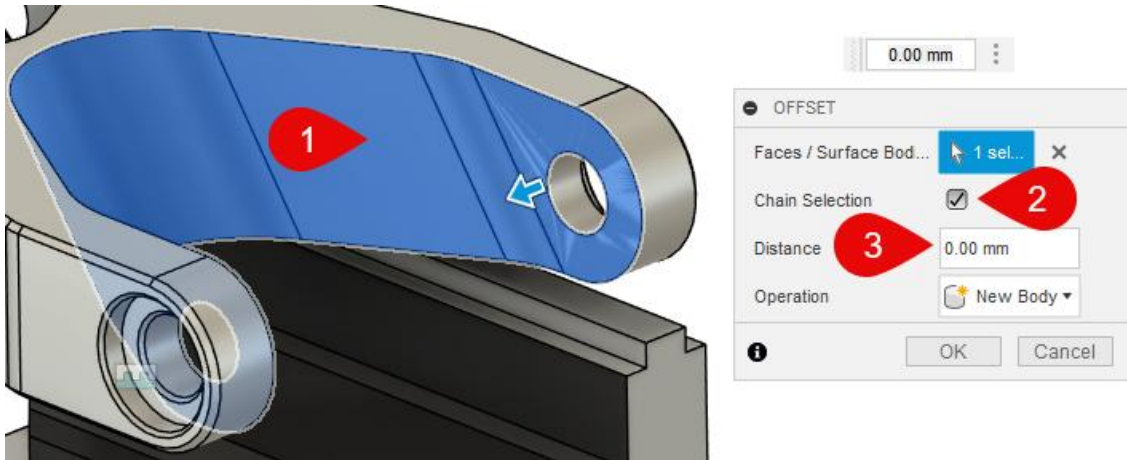
5. Switch to the **Passes** tab and set the **Stepover** to 0.5mm.
6. Click **OK** to generate the toolpath.
7. Duplicate the Flow toolpath.
 - a. Edit it and flip the z axis on the Multi-Axis tab.
 - b. Clear the Drive Surfaces on the Geometry tab.
 - c. Select the 12 3d chamfer faces on the opposite side of the part.
 - d. Click OK to generate the toolpath.

Surface modelling for manufacturing

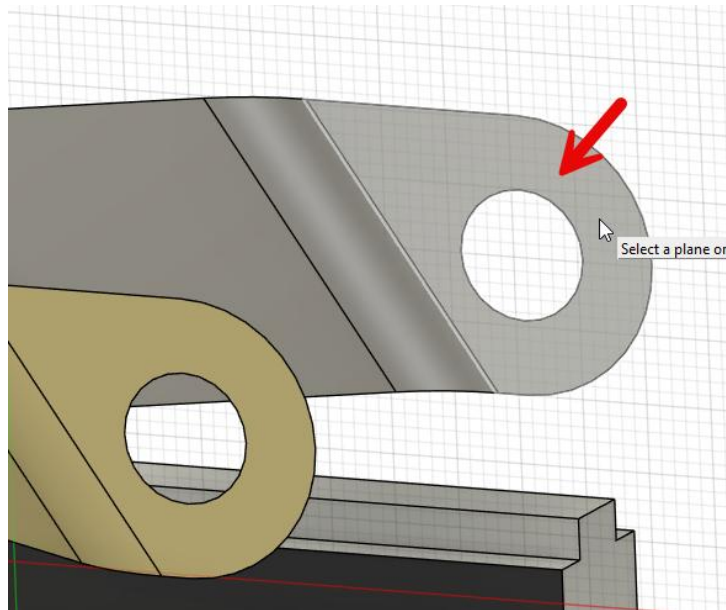
Now you need to finish angled face in the middle of the part. But the modelled geometry is quite inconvenient for toolpathing since it has a couple of holes in it, as well as full round edges. A far superior toolpath will be generated, if the faces are remodelled into a clean surface body.

1. Switch to the Design Workspace
2. Turn off the Stock component visibility
3. Change to the Surface tab
4. From the Create drop-down on the ribbon, select Offset.

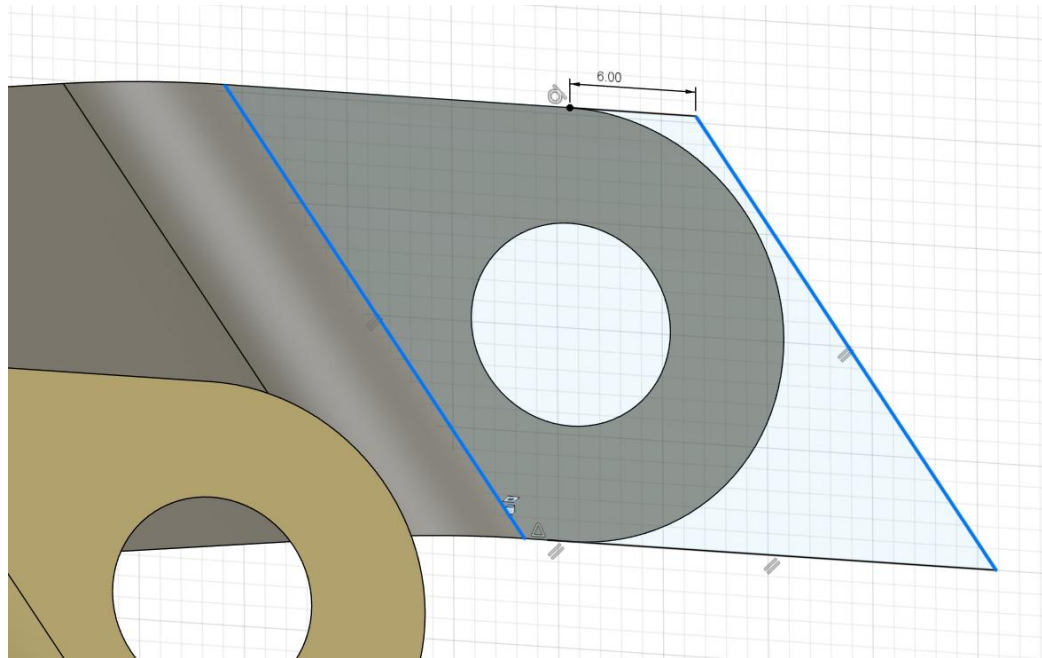
- a. Select the highlighted face in the image below
- b. Make sure Chain Selection is enabled
- c. Set Distance to 0mm
- d. Click OK



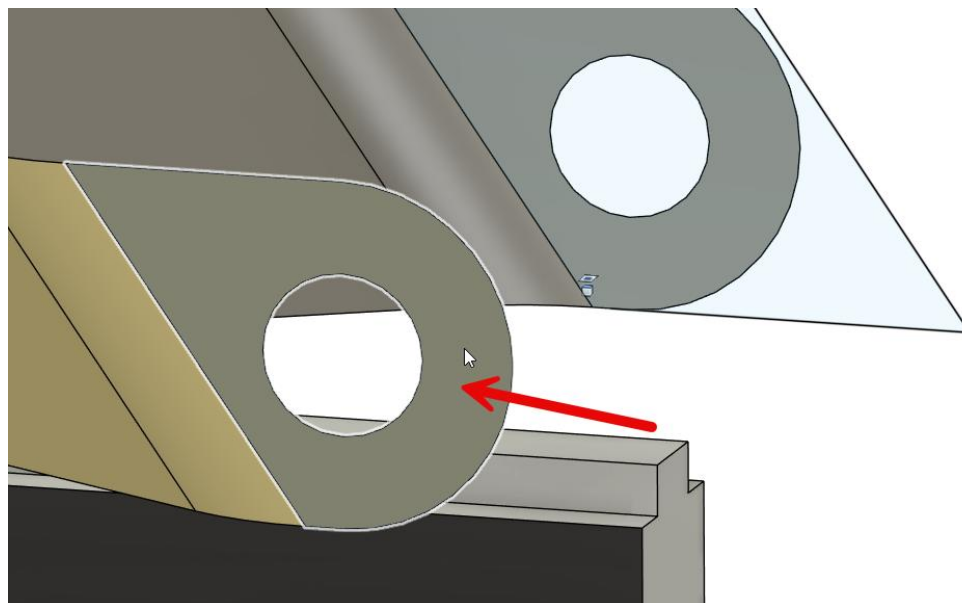
5. Turn off the Main Component visibility
6. Create a sketch on this face



7. Sketch 3 lines to form the following closed profile

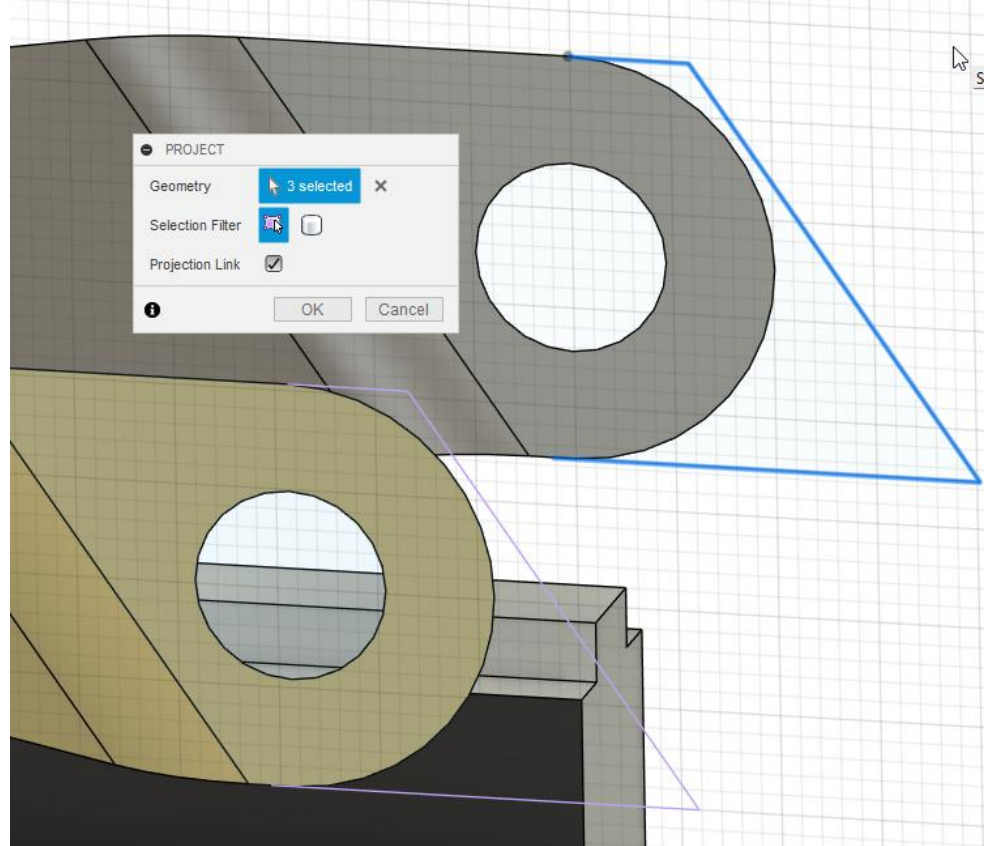


8. Make sure the horizontal lines are colinear with the horizontal edges of the surface. Also make sure the diagonal line is parallel to the highlighted surface edge shown above.
9. Dimension the length of the line shown above at 6mm.
10. Finish the sketch.
11. Create a new sketch on this face



12. Press P, to Project.

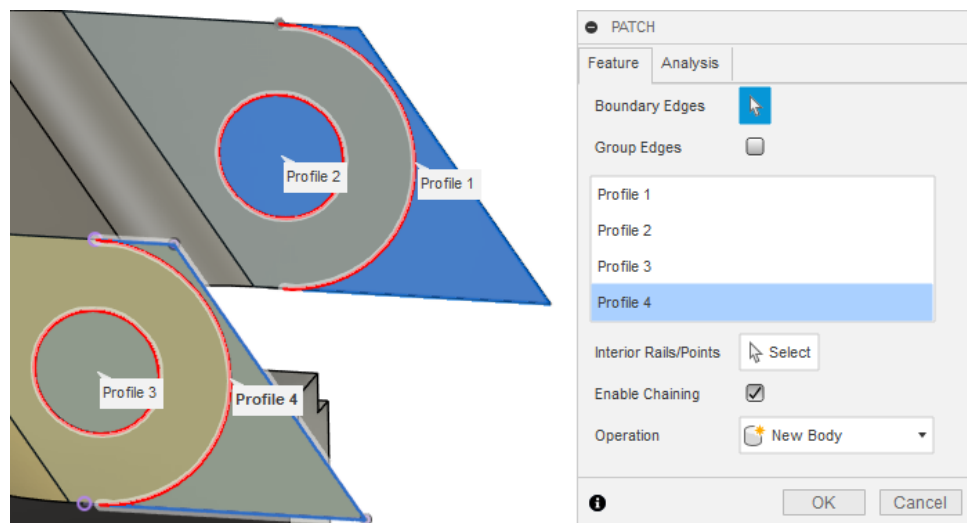
13. Project the 3 lines you created on the first sketch, then click OK.



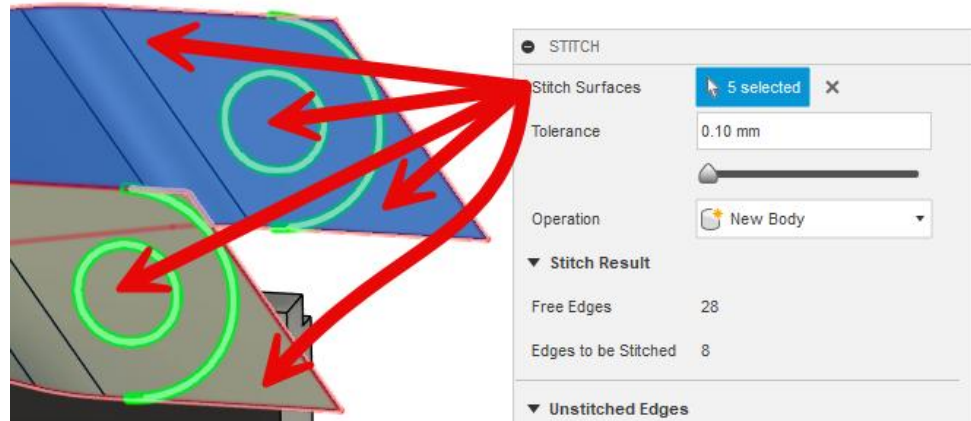
14. Finish the sketch.

15. From the Create panel on the ribbon, select the Patch command.

16. Select the 2 profiles from each of the two sketches, then click OK.



17. From the Modify panel on the ribbon, select the Stitch command



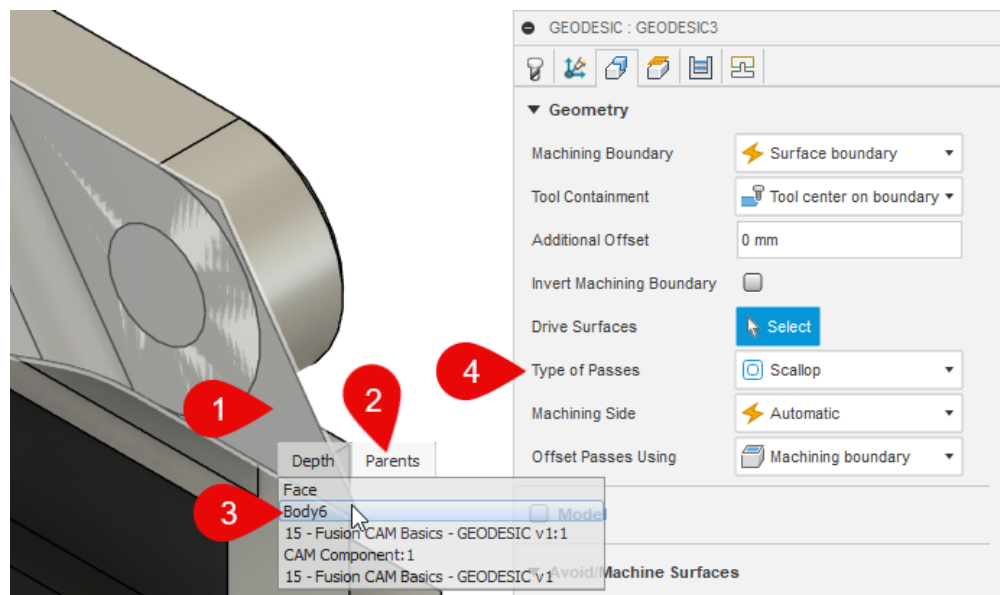
18. Select the 5 surface bodies and click OK.

19. Turn the visibility of the Main Component back on.

20. Switch back to the Manufacturing workspace.

Geodesic

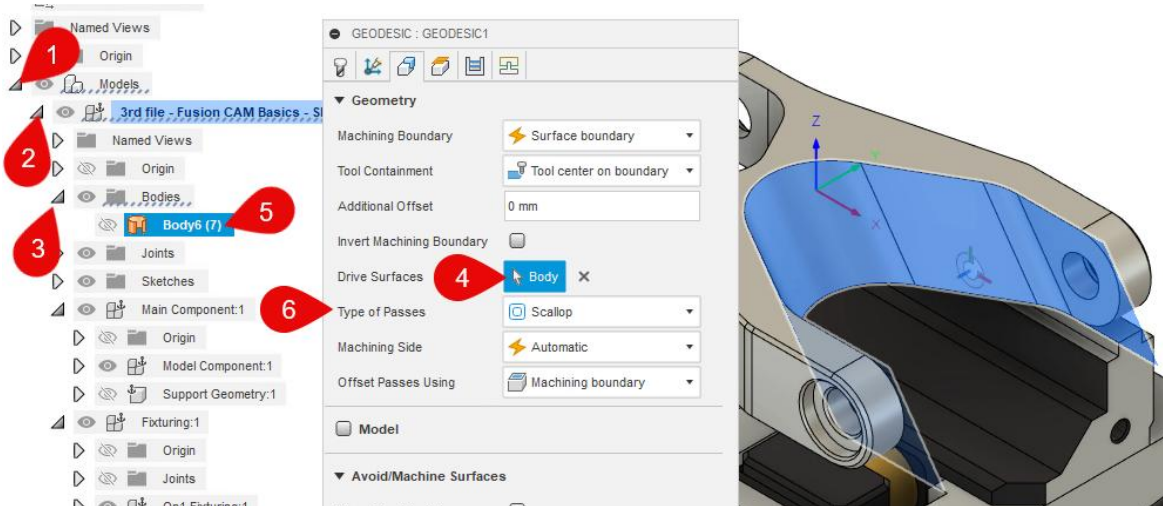
1. From the **3D** panel drop-down menu on the **Milling** ribbon, select **Geodesic**.
2. Keep the same tool and switch to the **Geometry** tab.
3. Left-click and hold on a face of the surface body you modelled, switch to the Parents tab on the Heads-Up dialog which appears. Then select the Body in the list as input for the Drive Surfaces field.



4. Change the Type of Passes parameter from Scallop to Blend.

5. Switch to the Passes tab and set the Maximum Stepover to 1mm. Click OK.

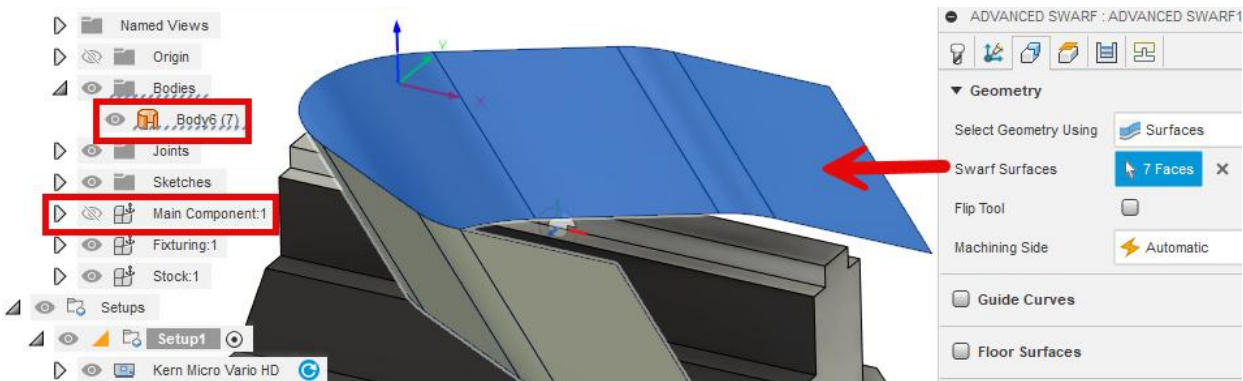
6. From the **3D** panel drop-down menu on the **Milling** ribbon, select **Geodesic**.
7. Keep the same tool and switch to the **Geometry** tab.
8. Expand the Models browser out until you see the first Bodies folder, expand that to see the surface body created to simplify the original model geometry.



9. In the **Avoid/Machine Surfaces** group, set the **Clearance** value for the **Drive** row to 0.2mm. This will leave stock on the part for the upcoming Swarf toolpath.
10. Change the **Type of Passes** parameter from **Scallop** to **Blend**.
11. Switch to the **Passes** tab and set the **Maximum Stepover** to 1mm. Click **OK**.

Advanced Swarf

1. Launch the **Advanced Swarf** command from the **Multi-Axis** panel drop-down on the ribbon.
2. On the **Tool** tab, select Tool #21 from the document library.
3. Turn on visibility of the surface body we selected in the previous section.
4. Move onto the **Geometry** tab. To make face selection easier, turn the visibility of the “Main Component” off in the browser. Then satisfy the **Swarf Surfaces** input by selecting the 7 faces which make up the surface body you modelled earlier.



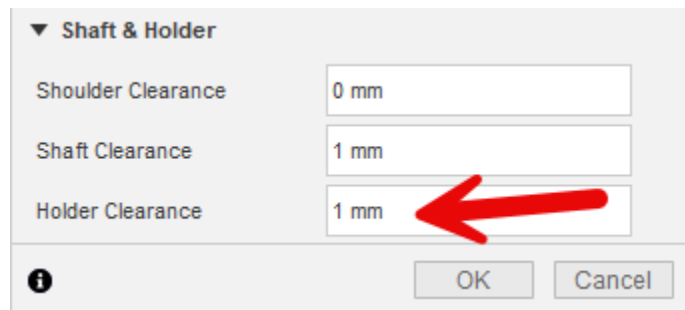
5. Click **OK** to generate the toolpath
6. Turn the “Main Component” visibility back on.

7. Turn off the visibility of your surface body, you don't need it anymore.

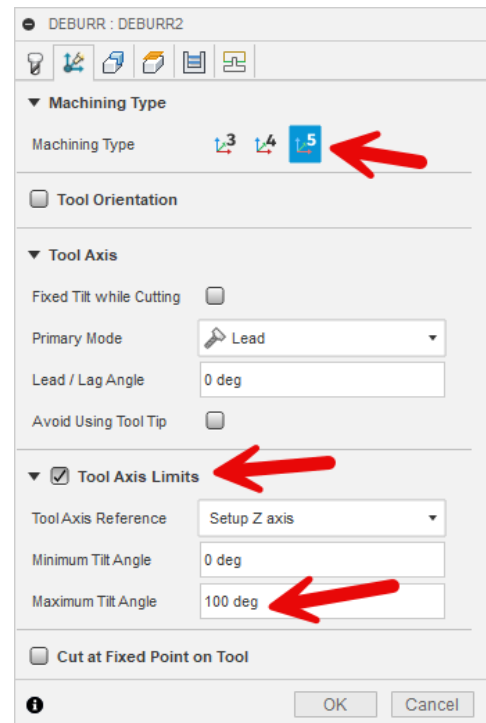
Deburr

The final step of programming the first operation of this part is to deburr all the sharp edges. Fortunately Fusion has an excellent weapon in its arsenal for this, Deburr.

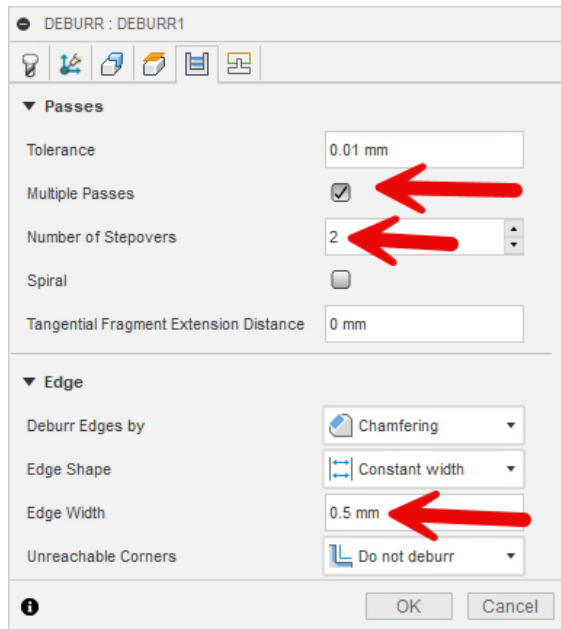
1. Under the **3D** panel drop-down, select **Deburr**.
2. Select the Lollipop mill from the Document library, Tool #1.
3. There are some tight clearances required to access all the edges effectively, so at the bottom of the **Tool** tab, in the **Shaft & Holder** group, change the **Holder Clearance** value from 5mm to 1mm.



4. Moving to the **Multi-Axis** tab, change the **Machining Type** to 5-axis.
5. Due to the position of the component on the table, we need to restrict the rotary motion a bit so we don't get a collision between the table and the head. Enable the **Tool Axis Limits** group, then set the **Maximum Tilt Angle** value to 100 degrees.
6. On the **Geometry** tab, the 3rd setting down is the **Minimum Edge Angle**, set that to 50 degrees. This will stop Deburr from applying passes to the modelled chamfers you surfaced with Flow earlier.



7. Finally, switch to the **Passes** tab:
 - a. Enable **Multiple Passes** and set the **Number of Stepovers** to 2.
 - b. Change the **Edge Width** value to 0.5mm
8. Click **OK** to generate.



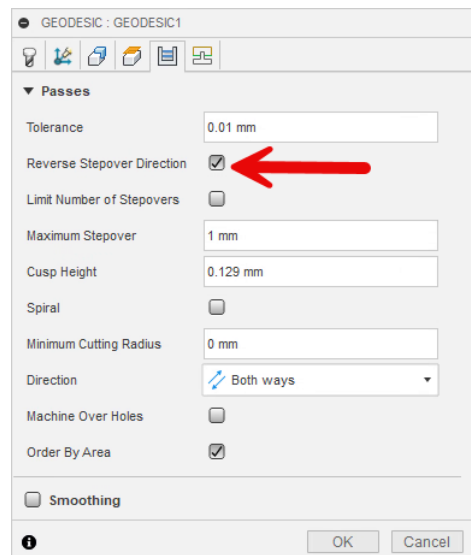
Machine simulate and verify

Now you can select the Setup, right-click and select Simulate with Machine.

Play around with the various settings and have some fun. If you want the simulation to run faster, disable the Stock simulation from the menu on the right of your screen.

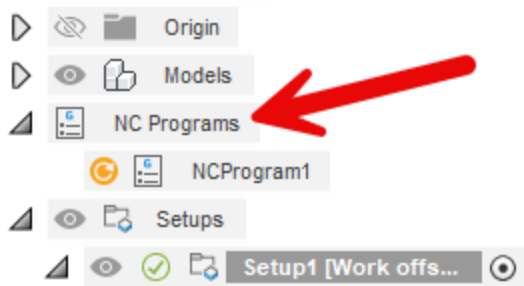
You will notice some collisions within the Geodesic toolpath. To fix them:

1. Right-click on Geodesic and **Edit** it
2. On the **Passes** tab, enable **Reverse Stepover Direction**.
3. Click **OK** to generate.
4. Re-simulate to see the collisions are no longer there.

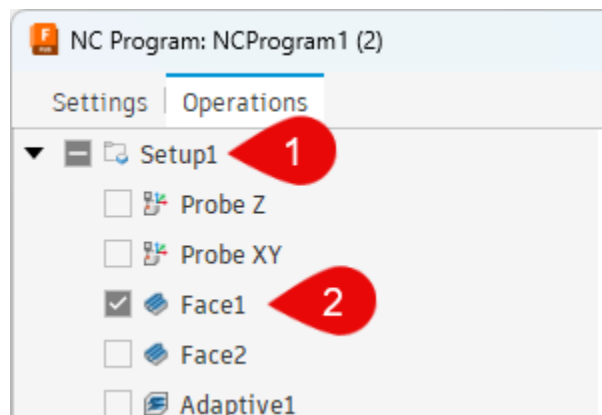


Post processing

1. Right click on the setup and select **Create NC Program**.
2. Enable the **Open NC file in editor** checkbox and leave everything else as default.
3. Then click **Post...** the file will open in Visual Studio Code where you will be able to read through the resulting code.
4. Return to Fusion
5. Expand the NC Programs folder and right-click on the NC Program you just created then select **Duplicate**.

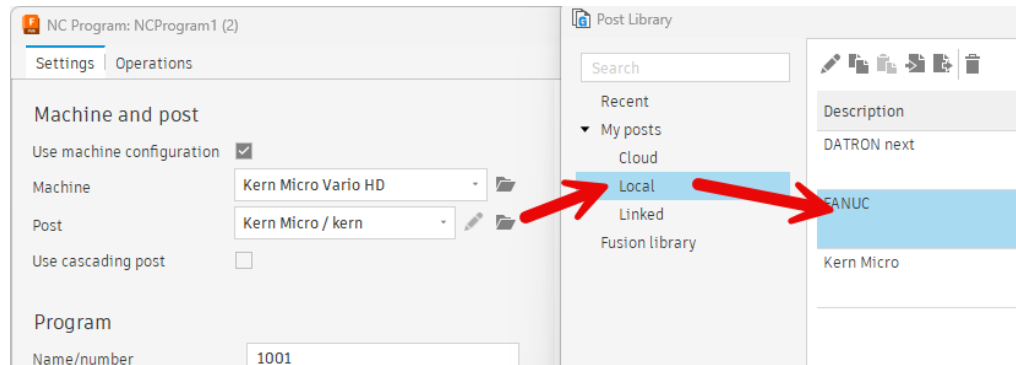


6. Right-click and Edit the duplicate NC Program
7. Switch to the Operations tab
8. Deselect the Setup checkbox, then enable the first Face toolpath



9. Switch back to the **Settings** tab, then click on the folder icon to the right of the **Post** setting.

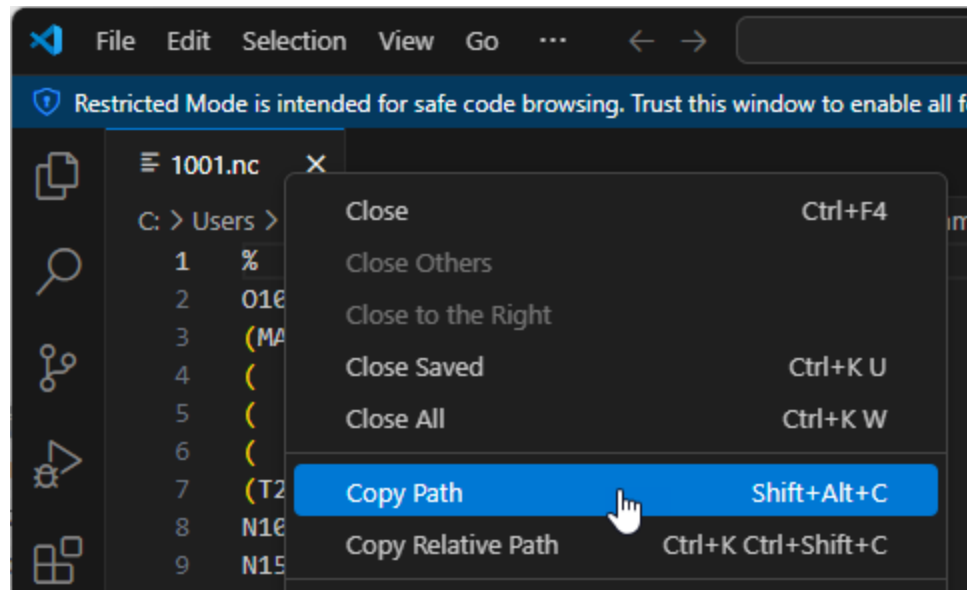
10. Select the Fanuc post from the Local library



11. Click **Post**

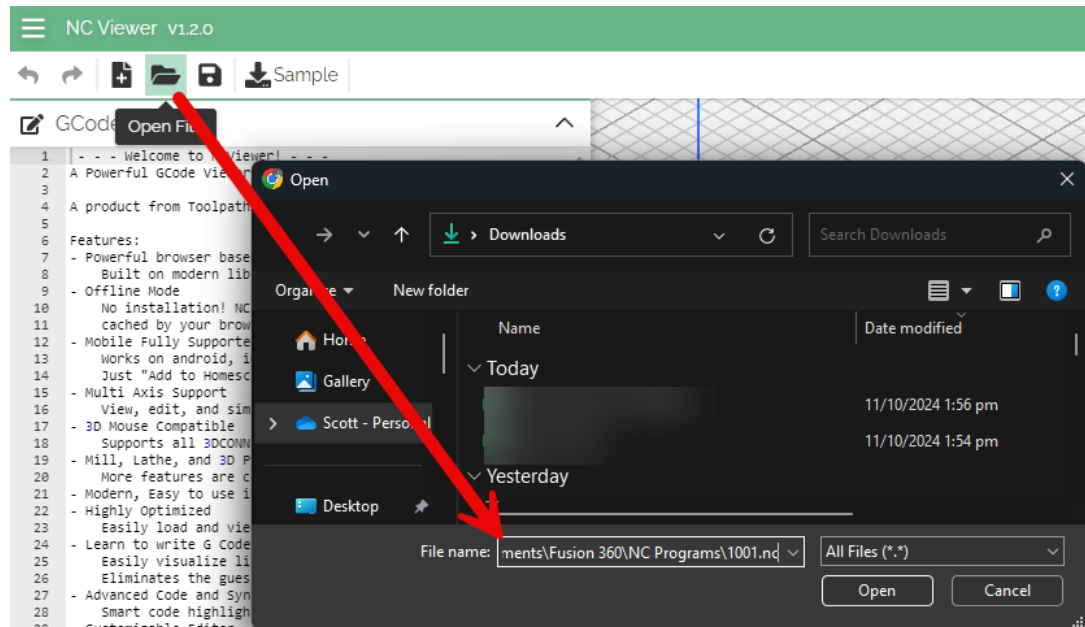
Notice how different the code is compared to post file we generated. That's Heidenhein vs Fanuc.

1. Right-click on the VS Code document tab for the Fanuc file you just posted.
2. Select **Copy Path**



3. Launch ncviewer.com in your browser of choice.

4. Select the **Open File** button in the top left of the page



5. Paste the file path into the **File name** field then click **Open**.
6. Run through the G-code and relate it to the passes you see on screen in the graphics area.

Resources:

[CNC Handbook](#)

[Toolpath YouTube Channel](#)

[Toolpath Webinars](#) << We will run a webinar series on the topic of this class, the recordings will be hosted here.

[Haas & Autodesk Fusion Course Part 1](#)

[Haas & Autodesk Fusion Course Part 2](#)

[Haas & Autodesk Fusion Course Part 3](#)

[Haas & Autodesk Fusion Course Part 4](#)

[Work offset and tool offset animations came from this YouTube channel](#)

[Haas tool offsets video](#) [Aaron Powter's Tool Offset Video](#)

[Haas work offsets video](#)