

MFG321728

Understanding CAM Programming in Fusion 360

Paul Clauss
Autodesk, Inc.

Learning Objectives

- Understand error and warning messages and how to resolve them
- Efficiently diagnose problems by isolating variables
- Optimize performance using best practices during CAM programming
- Confidently approach and troubleshoot difficult problems

Description

CAM programming can be difficult—there are so many parameters and variables in both the software and at the machine! Luckily, Fusion 360 software includes some useful tools that can help approach these difficulties when creating and post processing CAM programs.

This class will explain how to optimize performance and efficiency, decipher warnings and errors, diagnose root causes, and resolve common problems by understanding the helpful tools available in the Fusion 360 MFG workspace. Toolpath generation and containment will be covered, along with many useful tips and tricks to shorten programming time and get quality parts completed and out the door.

Speaker

I am the Fusion 360 MFG and HSM CAM Knowledge Domain Expert and Product Support Specialist at Autodesk. Before this I worked in a high-volume microchip manufacturing facility and helped develop a pressure sensitive bicycle saddle used to verify proper fit. I taught myself Fusion 360 during my time in college and have enjoyed expanding on that knowledge at Autodesk and propagating it throughout the community.

In my free time, I enjoy getting outdoors and can usually be found on the mountain bike trails surrounding Portland, Oregon (or at the brewery nearest the trailhead).



Table of Contents

Fusion 360 CAM Programming Concepts and Key Parameters (3)

Fusion 360 MFG Capabilities (3)

Types of Toolpath Strategies (Milling) (3)

2D Milling (4)

3D Milling (4)

Toolpath Containment (4)

Contour & Boundary Selection UI (11)

Tolerance & Smoothing Parameters (12)

Troubleshooting in the Fusion 360 MFG Workspace (14)

Troubleshooting Process (14)

Troubleshooting Tools (14)

Warnings & Errors (15)

.Failed NC Output (17)

Compare & Edit (18)

Derived Operations (21)

Integrated CAD (24)

Defeatured Models and Simple Edits (24)

Creating Machining Boundaries (28)

Imported Data (31)

Surface Quality (33)

Additional Resources (40)

Fusion 360 CAM Programming Concepts and Key Parameters

There are some key concepts to be familiar with in order to efficiently solve problems in the Manufacturing workspace of Fusion 360. Programmers must be aware of the manufacturing capabilities of Fusion 360, the differences in toolpath strategies, how to contain toolpaths, how to select contours and boundaries, and have an understanding of the tolerance and smoothing parameters to guarantee high quality NC output and machined results.

Fusion 360 MFG Capabilities

As of the date of this presentation (November 2019), Fusion 360 supports the following types of manufacturing processes:

- 2D (2.5-axis) & 3D (3-axis) Milling
- 4-axis wrapped milling and continuous 4-axis surfacing with the Rotary strategy (feature flag)
- 5-axis positional and simultaneous milling (5x and 3+2)
- Water/Laser/Plasma Jet Cutting
- Turning & Mill-Turning
- Probing & Surface Inspection
- Additive Manufacturing – 3D Printing of Metals and Plastics

More information on all of these options is available on the Fusion 360 website and in the learning materials linked below:

- [Fusion 360 MFG: Adoption Portal](#)
- [Fusion 360 MFG: Product Documentation](#)

Types of Toolpath Strategies (Milling)

While Fusion 360 does support all the manufacturing processes listed above, this class will focus on 2.5 (2D) and 3D Milling operations to simplify basic concepts that can be applied when troubleshooting problems with any type of toolpath.

A great resource for understanding 2D and 3D Milling in Fusion 360 is the [CAM Deep Dive video](#), from Fusion 360 expert Marti Deans. The Deep Dive video uses the “2D Milling – Patterns” and “3D Milling – Containing Toolpaths” example files available in the “CAM Samples” folder of the Data Panel.

2D Milling

2D toolpath strategies in Fusion 360 allow simultaneous motion on two axes (X and Y), meaning they generate cutting passes on a single Z-plane (with respect to the machining setup in which the Z-axis is parallel to the axial tool axis at the machine).

The 2D strategies are not model-aware – they only “see” the selected geometry (faces or edge/sketch chains). Because these toolpaths do not see the entire 3D model, care needs to be taken to ensure they do not gouge with model geometry after generation.

More information on the 2D Milling options available in Fusion 360 can be found at the link below:

- [Learning: 2D Machining Strategies](#)

3D Milling

The 3D toolpaths available in Fusion 360 support simultaneous three-axis motion on the X, Y, and Z axes. This allows the creation of three-dimensional surfacing programs on parts with complex geometry.

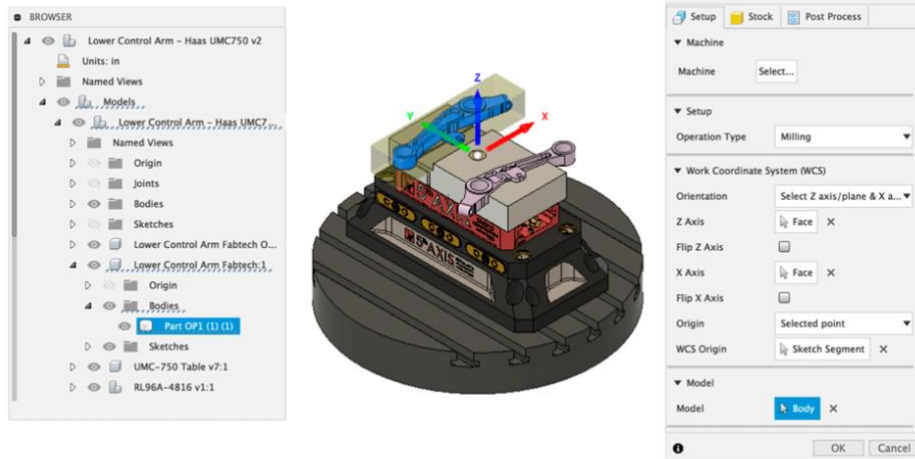
3D machining strategies within Fusion 360 are model aware. These toolpath strategies see the entire 3D model (selected as the model in the CAM setup) and work by projecting a boundary along the Z-axis to limit X-and-Y-axis motion to a particular area on the model.

- [Learning: 3D Machining Strategies](#)

Toolpath Containment

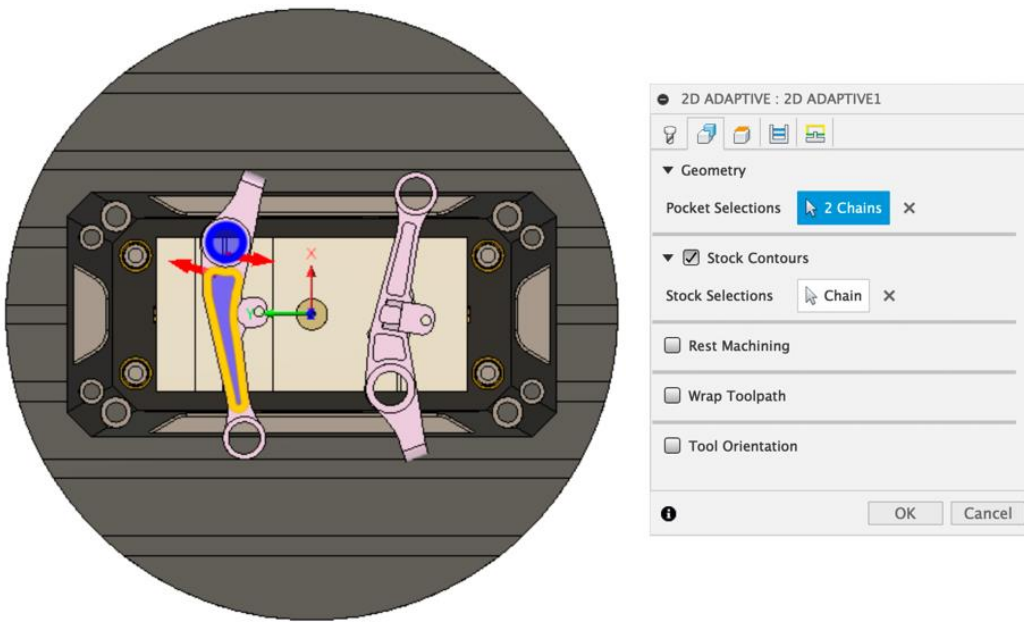
When creating machining operations, it is likely the area being cut will need to be limited. For 2D and 3D milling toolpaths, toolpaths can be limited on the XY-plane (the area being cut) and along the Z-axis (the top and bottom depths of the cut).

The first step in toolpath containment is selecting the model body or component that will be machined in the CAM setup. In the screenshot below, we can see that the “PartOP1 (1)(1)” body in the “Lower Control Arm Fabtech:1” component is selected in the OP-1 Fabtech Normal Cutting Demo” CAM setup. This is shown in the Setup dialog and the browser with the blue highlights shown below.

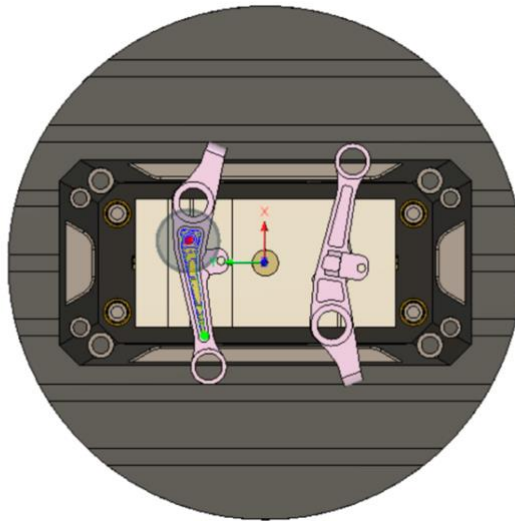


After defining the parts for machining in the CAM setup, toolpaths can be applied. Because 2D toolpaths are not model aware, any geometry (edge, face, or sketch) can be selected. This means 2D Toolpaths can override model selections in the CAM setup with no action required beyond selecting the contour. The model definitions do, however, define the model bodies that are “seen” by the model aware 3D strategies.

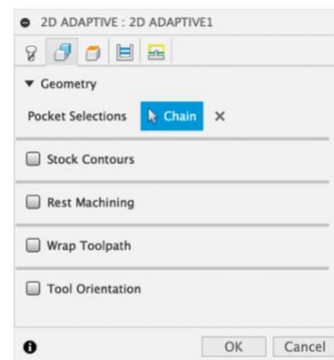
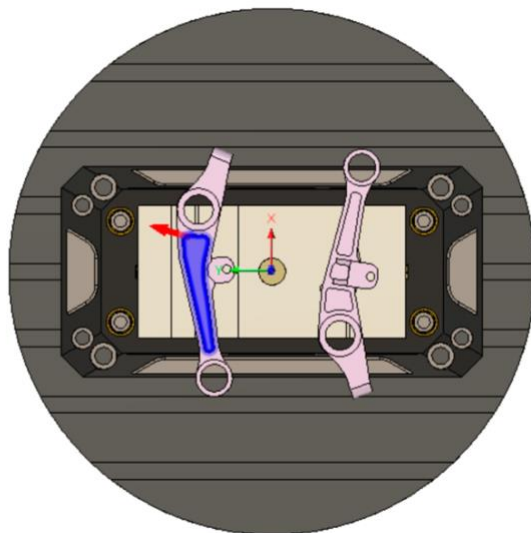
To limit toolpath motion to a particular area on the XY-plane, the 2D milling strategies within Fusion 360 use Pocket/Contour Selections and Stock Contour Selections. Pocket/Contour Selections define areas in which the tool can move, and Stock Contour Selections define areas that need have material that should be removed by the toolpath.



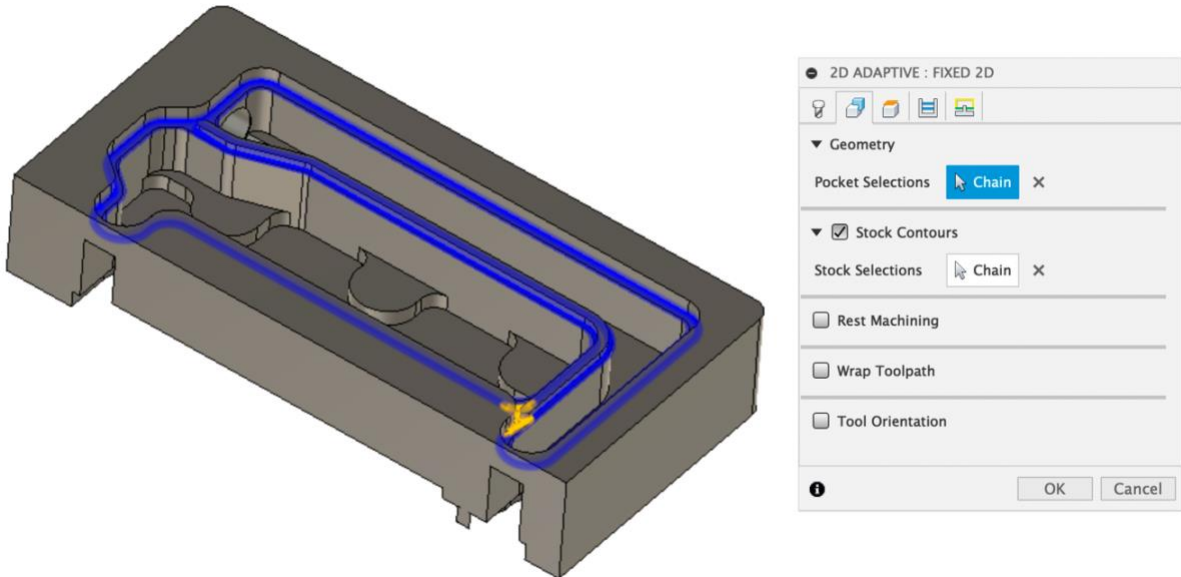
In the 2D Adaptive toolpath above, the faces in highlighted in blue are selected as Pocket Selections. But only the larger bottom slot is selected as a Stock Contour. Because one of the Pocket Selections (the top-hole edge) does not include Stock Contours (as defined in the screenshot above), it will be ignored in the resulting toolpath as shown below.



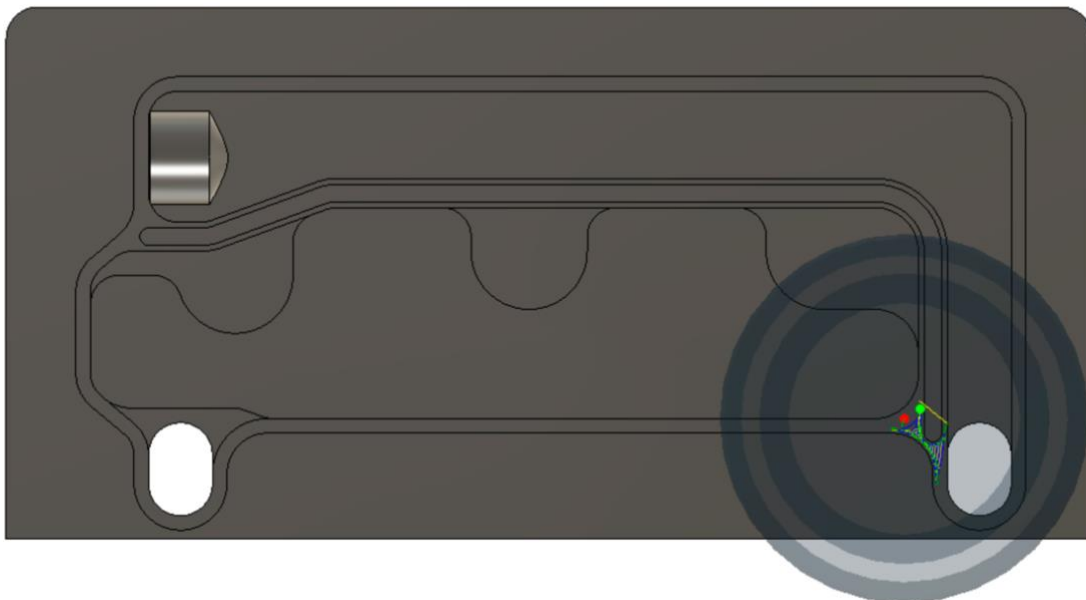
If no stock contours are selected, Fusion 360 defaults to assume that any Pocket/Contour Selections contain stock to be machined. The selections shown in the example above do not represent a realistic use case – if machining of only the bottom pocket selection was desired, the selections below would give identical results with more efficient and straightforward manner, as no stock contours or additional pocket selections need to be defined. The example shows that Stock Contours can be used to specify cutting areas *within* a pocket selection.



The example below is a more realistic use of Stock Contours – the Stock Contours (shown in yellow) tell the toolpath to generate cutting passes in only a specific area within the Pocket Selection. This could be a modification to an existing part.

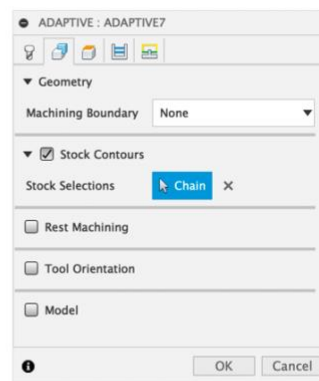
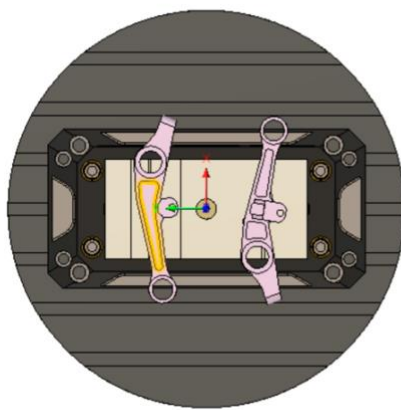


The resulting toolpath is shown below. The entire pocket selection except the area overlapping the stock contour selection is ignored by cutting passes. In this case, rest machining may be another option to get the desired results depending on how the setup stock is defined.

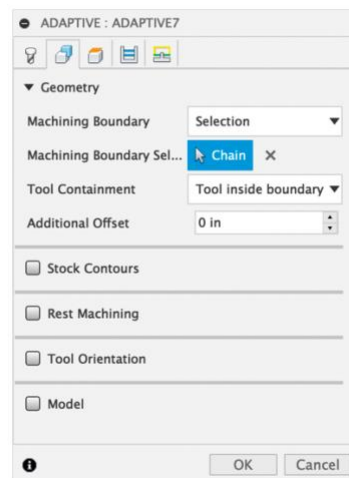
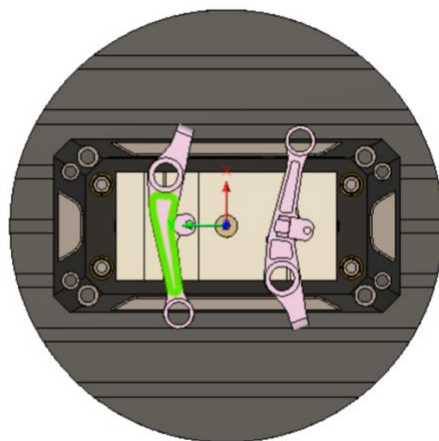


3D Milling toolpaths use Stock Contours in the same way as 2D strategies but use Machining Boundaries rather than Pocket Selections. Because 3D operations see the entire part, machining boundaries limit the toolpath to a particular area rather than applying a toolpath to the Pocket selection like the 2D options.

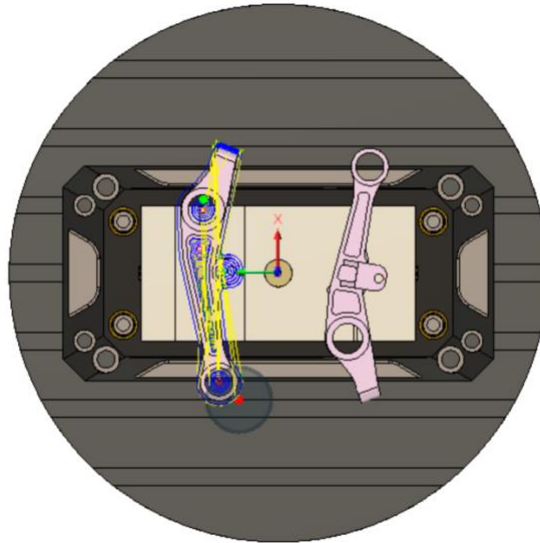
Machining boundaries in 3D milling operations can be ignored (no machining boundary needs to be set) or set to “tool inside boundary,” “tool center on boundary,” or “tool outside boundary.”



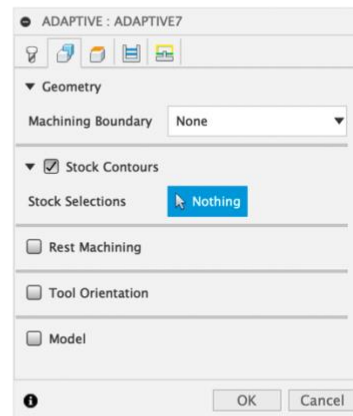
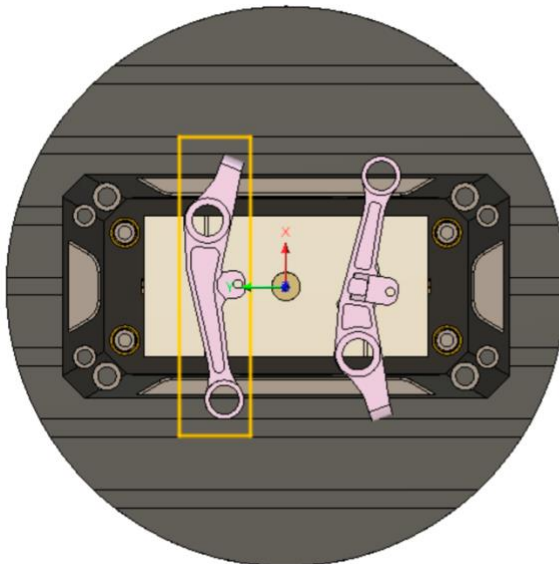
In this example, no machining boundary is selected and Stock Contour selections are used to define the cutting area. The same end result could be achieved by not defining Stock Contours and selecting the same edge as the machining boundary with a “tool inside boundary” condition, as shown below. The difference is that, with the machining boundary, the tool is allowed outside the boundary during non-cutting moves.



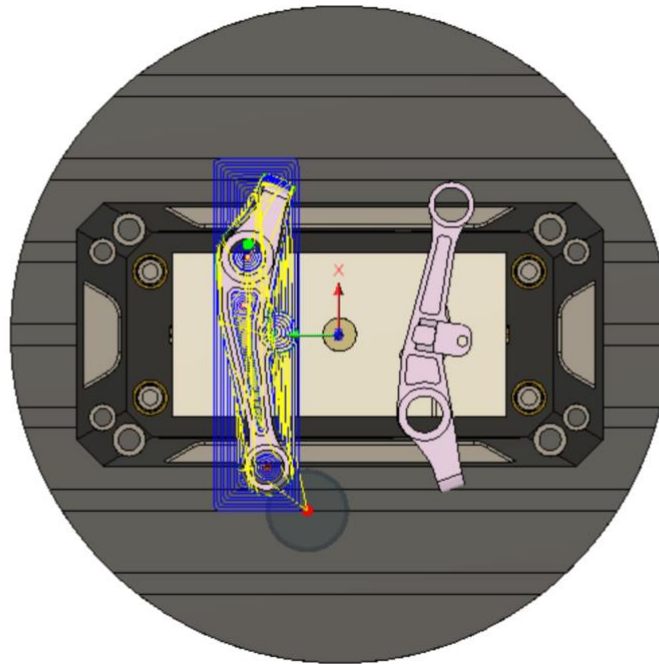
If no machining boundary or stock contour is selected in a 3D Adaptive toolpath, the result will be a toolpath that machines to the part surfaces but does not include passes to clear stock outside the body selected for machining. The result below has no boundary or stock selections.



If the Stock Contour tab is enabled but no selection is made, the toolpath will default to clear all the stock defined in the CAM setup, as shown in the two images below. The yellow rectangle appears when the stock contours tab is enabled (with the check mark) but no selection is made.



The resulting toolpath will clear stock out to the yellow rectangle, which is the perimeter of the stock defined in the CAM setup. It will also machine the part surfaces and internal pockets, as no machining boundary was defined.



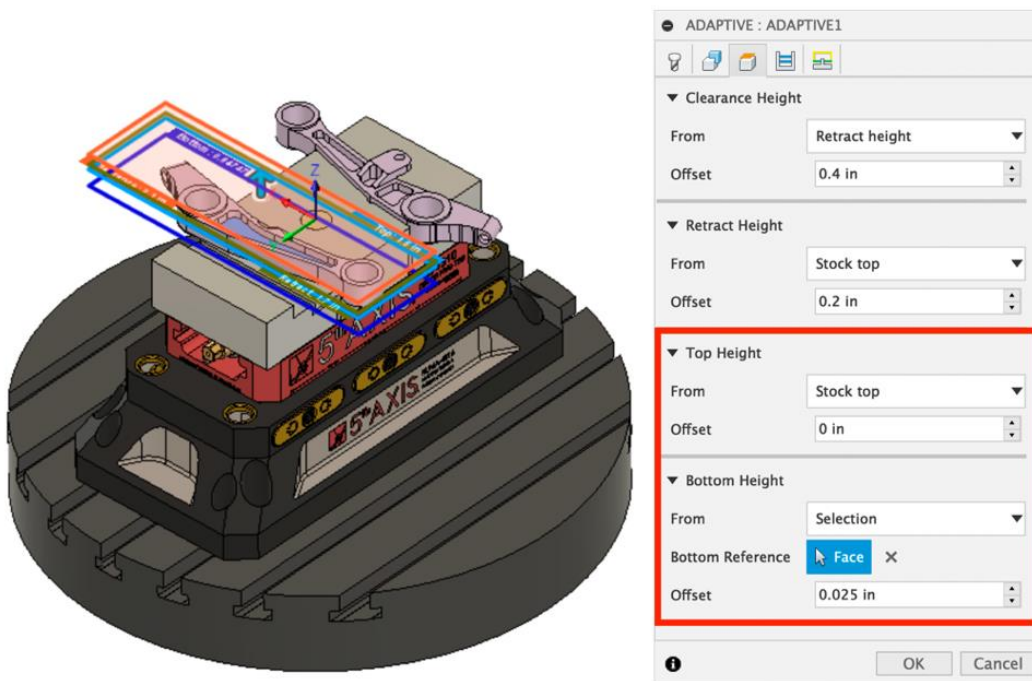
While Pocket/Contour Selections (2D), Machining Boundaries (3D), and Stock Contours (2D and 3D) are the main methods to limit toolpath motion on the XY-plane, there are additional containment options available. These are described briefly below.

- “Slope” containment limits motion by applying an allowable range of surface angles to the toolpath. This option is available in all 3D finishing toolpaths in the “Geometry” tab.
- “Model” selections allow bodies or surfaces to be recognized by 3d toolpaths without having been added into the CAM Setup, which could cause interference in other toolpaths. This allows construction geometry to be used to drive toolpaths.
- Avoid/touch (or “Check surfaces” in HSMWorks/Inventor CAM) defines surfaces to be machined or avoided during machining. This is a good option for multi-axis toolpaths as it avoids the need for boundaries to be projected along the Z-axis and allows containment of

multi-axis toolpaths with face selections. Construction or fixture bodies can be avoided using “Avoid Surfaces” in the 3D strategies.

- Automatic containment is used by Pencil and Horizontal toolpaths. Pencil looks for internal fillets within a range defined by the tool size plus/minus the “Overthickness” parameter and Horizontal machines surfaces parallel to the setup Z-axis.
- Rest Machining limits toolpaths to remove only remaining stock based on a number of stock definition options. More information is available in [the Rest Machining Learning Tutorial](#).

Limiting toolpaths along the **Z-axis** is done in the Heights tab for all milling toolpaths in the Manufacture workspace of Fusion 360. Cutting passes will be generated between the Top and Bottom height definitions.



Contour & Boundary Selection UI

In order to apply machining boundaries, stock contours, or contour/pocket selections to CAM strategies, the appropriate model or sketch geometry needs to be selected.

When selecting model edges or sketch contours, Fusion 360 defaults to select closed loops. There may be times where Fusion 360 will not close the loop using the desired path or situations in which an open contour is desired.

Edge or sketch selections can be edited using the Contour Editor. The Contour Editor is accessed by hovering the mouse cursor over a selection until the selection turns red and then clicking on the selection again.

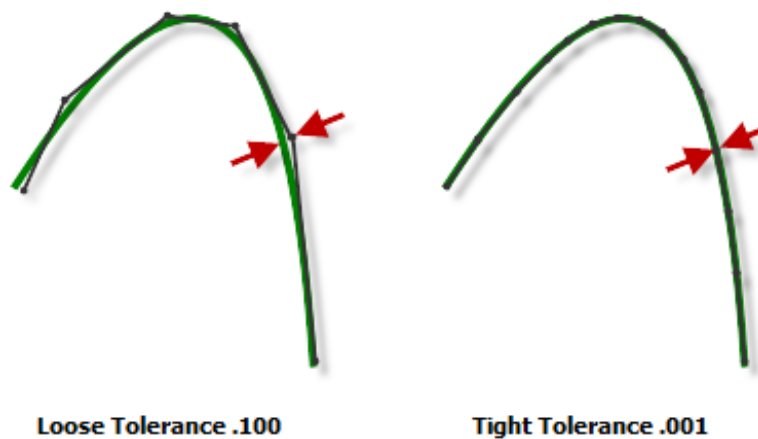
More information on using the Contour Editor and screenshots illustrating the process are available in the article linked below:

- [How to edit contours in Fusion 360 CAM](#)

Tolerance & Smoothing Parameters

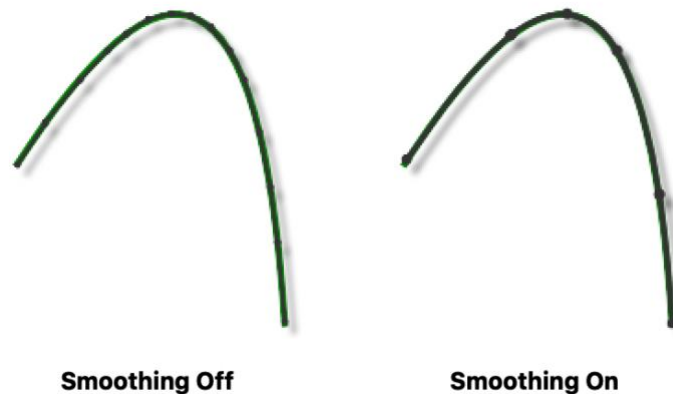
The Tolerance parameter controls the accuracy of the mesh model that is created to drive toolpath generation. The b-rep geometry in the Fusion file is “meshed” and sent to the CAM kernel.

The “mesh” of the part created during toolpath generation defines the linear segments of the toolpath. The allowable variance between the true model geometry and the mesh as a chord distance is defined by the toolpath Tolerance parameter.

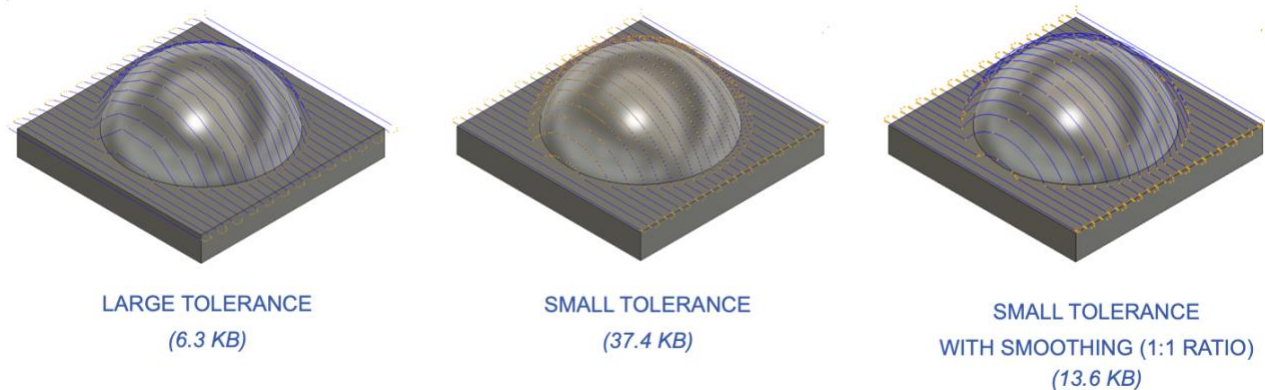


A tight tolerance will result in a more accurate toolpath but in larger NC output due to the increased number of linear segments – each segment is a line of code. Toolpath generation will also take longer. A loose tolerance will generate quickly and give smaller output sizes but may not follow model geometry as accurately.

While the Tolerance controls the accuracy of linear segments of the toolpath, the Smoothing parameter fits arc moves to toolpath within specified tolerance.



An arc may take many linear segments to match but could only take one circular move to fit. Using this concept, the large toolpath size that comes with a tight (small) tolerance can be managed by applying Smoothing.



A 1:1 ratio is recommended for the best balance between output size, accuracy, and generation time. The total allowable variance allowed between the toolpath and model geometry is the sum of the tolerance and smoothing parameters.

The resources below explain the Tolerance and Smoothing parameters in great detail:

- [Optimizing Toolpath Generation in the Fusion 360 MFG Workspace \(Autodesk Knowledge Network Article\)](#)
- [Understanding Smoothing \(Forum Thread\)](#)
- [Autodesk HSM: Toolpath Tolerances with Expert Elite Rob Lockwood \(recorded webinar\)](#)
- [Autodesk Visits Haas Automation \(Haas Video\)](#)

Troubleshooting in the Fusion 360 Manufacture Workspace

CNC machining and CAM programming are complex subjects and it is easy to create toolpaths that will give less than optimal results, warnings, errors, or unexpected behavior. Understanding software capabilities, containment, selections, and key parameters like tolerancing and smoothing will help avoid problems, but programming can still be difficult.

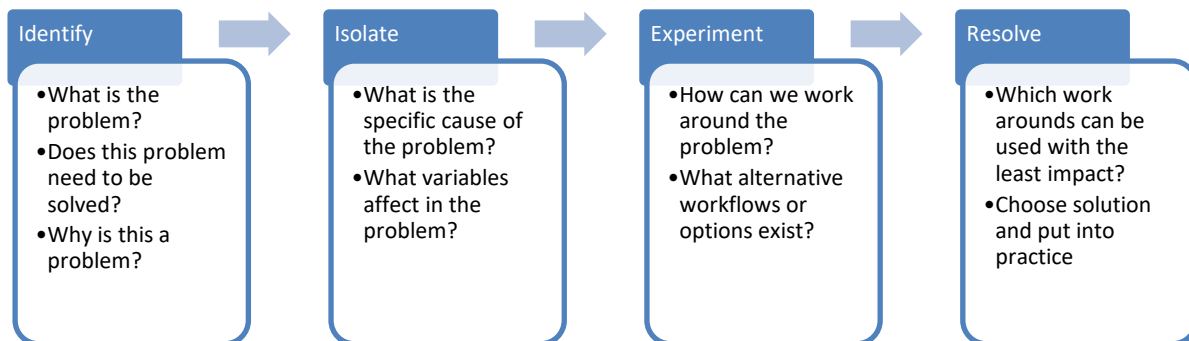
The wide range of parameters available during CAM programming in Fusion 360 can make it difficult to pinpoint the exact cause of these problems. Luckily, Fusion 360 includes some tools that are very useful when troubleshooting.

This section will discuss the troubleshooting process and explain how to use the tools in Fusion 360 to ensure efficient solutions and optimal results in the software and at the machine.

Troubleshooting Process

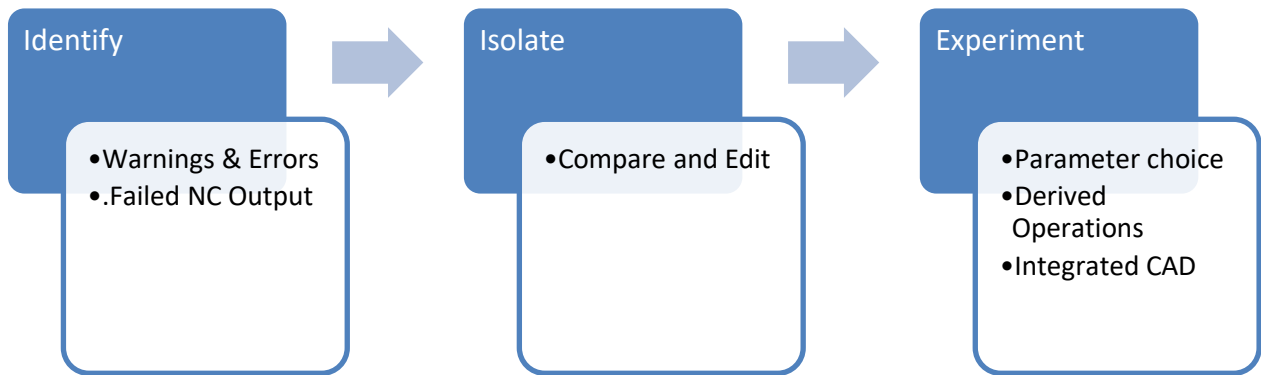
Troubleshooting requires a different approach than that of i CAM programming. Instead of selecting and applying parameters, problem solving requires an organized method to isolate variables.

When addressing unexpected behavior or poor results in CAM programs (and generally in any CAD work), using the process below will help.



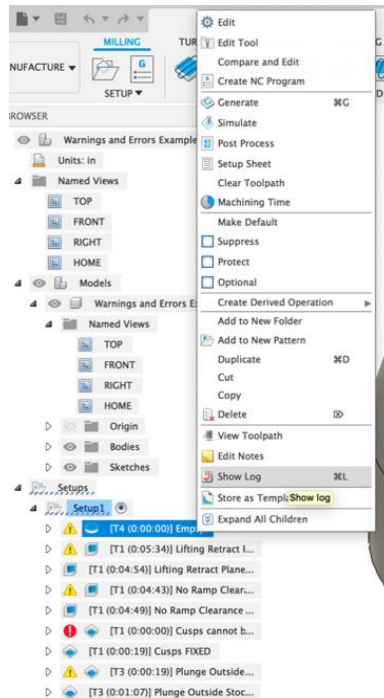
Troubleshooting Tools

Fusion 360 includes tools to help programmers through the steps outlined above and efficiently solve problems when they do occur, lowering stress and ensuring productivity.



Warning & Errors

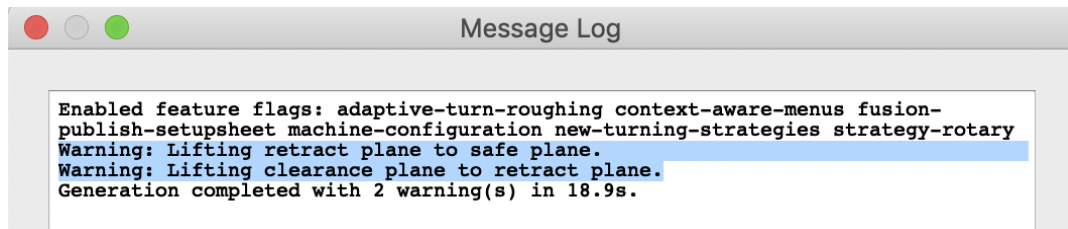
Toolpath warning and error messages are intended to identify why a toolpath was not generated as expected. Messaging is accessed by clicking on the flag in the MFG Browser or by right-clicking the toolpath and selecting “Show Log.”



Warnings are indicated with a yellow flag next to the toolpath in the MFG Browser. When a warning is shown, the toolpath may or may not generate.



Warnings often indicate changes that were made automatically to a toolpath that was allowed to generate but required edits to the parameters to do so. Examples of this often include changes to retract and clearance plane heights.



The toolpath log above indicates that the toolpath was allowed to generate, but the retract and clearance planes were lifted to a safe plane.

To give some detail, this means that the retract plane was moved upwards to a safe plane. The height of the safe plane is dependent on the Safe Distance parameter (default of 2mm above top height), the Ramp Clearance Height, axial stock to leave, and the entry and retract positions in the toolpath.

To resolve this warning, the ramp clearance height or axial stock to leave could be reduced, or the retract and clearance heights could simply be lifted until they met the safe plane.

Toolpath warnings in Fusion 360 will generally address changes to heights, ramping moves, or entry positions. They may state that a toolpath is empty (no toolpath was generated), note that certain passes were not generated, or warning about tool parameters or feeds and speeds. Because warnings and errors are known results, a good first step is performing an internet search for the text in the message if it is unclear.

There is a wide variety of warning messages that can be triggered in Fusion 360 and articles on troubleshooting some common issues are linked below:

- [Troubleshooting Empty Toolpath and No Passes to Link warnings in Fusion 360/HSM CAM](#)
- [“Warning: Lifting retract plane to safe plane” message in Fusion 360 and HSM CAM](#)
- [“Warning: One or more passes were discarded due to linking constraints” in Fusion 360/HSM CAM](#)

Toolpath errors are noted by red exclamation point flags in the CAM Browser:



Errors show that a toolpath could not be generated given the current parameters. Errors often address missing selections caused by model edits or changes to the CAM setup, irresolvable parameters (which will be mentioned in the error message text), or unexpected errors during the toolpath generation process.

The most difficult errors to resolve are the dreaded “Internal CAM Kernel Error” or “CAM Kernel Died Unexpectedly” errors. These errors are best approached by using the Compare and Edit tool to isolate variables, which will be discussed later.

.Failed NC Output

Sometimes a toolpath will generate and simulate correctly in Fusion 360 but will show a “.failed” file extension on the NC output file when post processed. This occurs when the post processor being used is unable to translate the toolpath information from Fusion 360 into g-code for the machine.

This happens due the machine and controller capabilities defined in the post processor. The .failed NC file can be opened in any text editor and contains details on why the program could not be posted. Some common examples are included in the Autodesk Knowledge Network article linked below:

- [Error messaging in g-code programs posted by Fusion 360 or HSM CAM](#)

One way to perform initial troubleshooting when receiving a unique error message is to search in the post processor itself for the exact text of the error message. The logic in the post configuration will show what conditions trigger the error message.

A good example of this is the "Program number is out of range" message. If the CPS post processor file is opened in a text editor and the exact text of the error message is searched for, the section of code below is found:

```
if (!(programId >= 1) && (programId <= 9999)) {
    error(localize("Program number is out of range."));
    return;
}
```

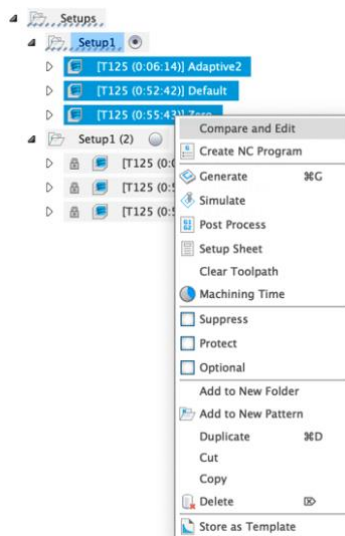
This line shows that, if the program number is outside of the allowable range of 1-9999, the "Program number is out of range" message will be shown. This same troubleshooting concept can be applied to any error message generated by a post processor.

So, to get this program posted, the program number (in the Post Process dialog) would need to be reduced or the post could be edited to allow five-digit program numbers by changing "9999" to "99999."

Compare & Edit

The Compare and Edit tool can be used to isolate variables in the MFG workspace of Fusion 360. Using Compare and Edit, the exact parameter differences between any number of toolpaths can be easily identified.

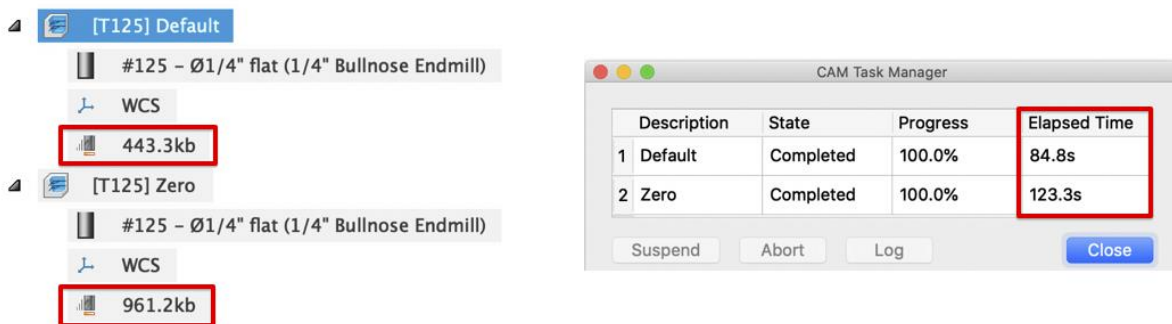
The Compare and Edit dialog is accessed by right-clicking selected toolpaths in the MFG browser. Multiple operations can be selected by holding the Shift or CTRL(PC)/Command(Mac) keys. As many operations as are selected can be edited simultaneously through the Compare and Edit dialog.



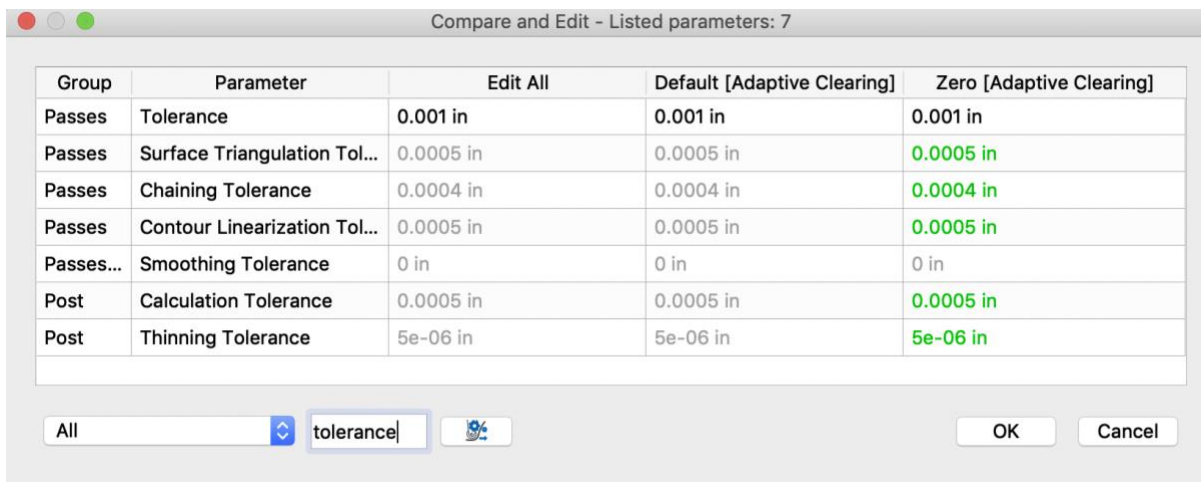
In some cases, a single parameter can have a large impact on a toolpath. If an "Internal CAM Kernel Error" is occurring, a new toolpath using the same strategy can be created with the default parameters. This toolpath can then be selected with the original problem toolpath in

the Compare and Edit dialog. The “different” toolpath parameters filter can be applied, and the new toolpath parameters can be edited, one by one, until the “Internal CAM Kernel Error” occurs in the new toolpath. The last parameter that was changed to match the original toolpath is the cause of the error.

A simple example of this process is using Compare and Edit to observe the effect of a single parameter change on toolpath generation time and toolpath size. In the screenshots below, the “Zero” toolpath is nearly twice the size of the “Default” operation and takes over 30 seconds longer to generate. These toolpaths are both 3D Adaptive roughing operations using the same tool and applied to the same area on the same model.

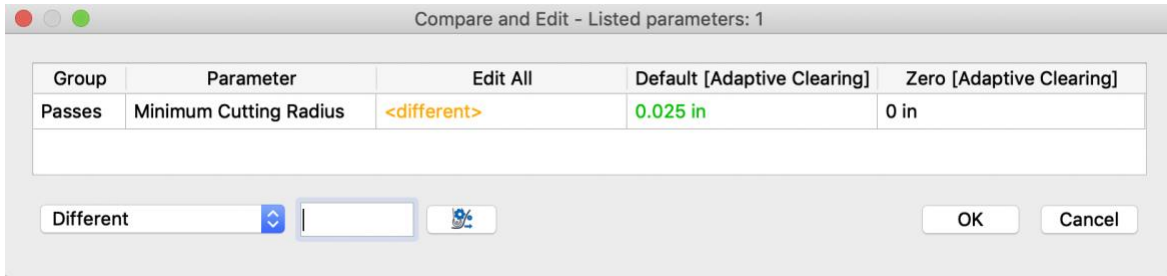


The obvious solution here, based on the generation time and toolpath size, is to check the tolerance and smoothing settings for operation. This can be quickly done by searching the Compare and Edit dialog for “tolerance.”

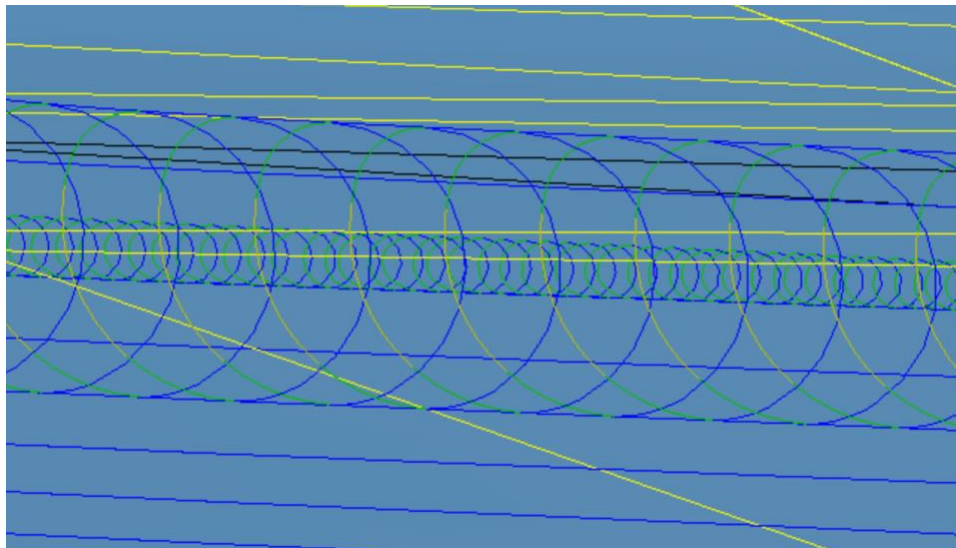


Using the Compare and Edit dialog, all the tolerance and smoothing values can be quickly checked for each operation. This verifies that these parameters are the same between the two toolpaths and confirms that the differences in generation time and NC output size is due to a different parameter.

To investigate which parameters are different between the two toolpaths, the filter in the bottom left hand corner of the Compare and Edit dialog can be used to show only variables with different values.



While a toolpath with a zero Minimum Cutting Radius (MCR) will generate, it will attempt to generate passes anywhere the selected tool can reach, even with tiny arc moves. Applying an MCR value prevents sharp corners and minimizes chatter but may prevent material from being removed in certain areas. The effect of the zero MCR can be seen in the toolpath previews if the toolpath is viewed very closely. In the screenshot below, the larger circular moves are from the operation using the default MCR (0.025") and the small arc moves are in the operation with a zero MCR value.

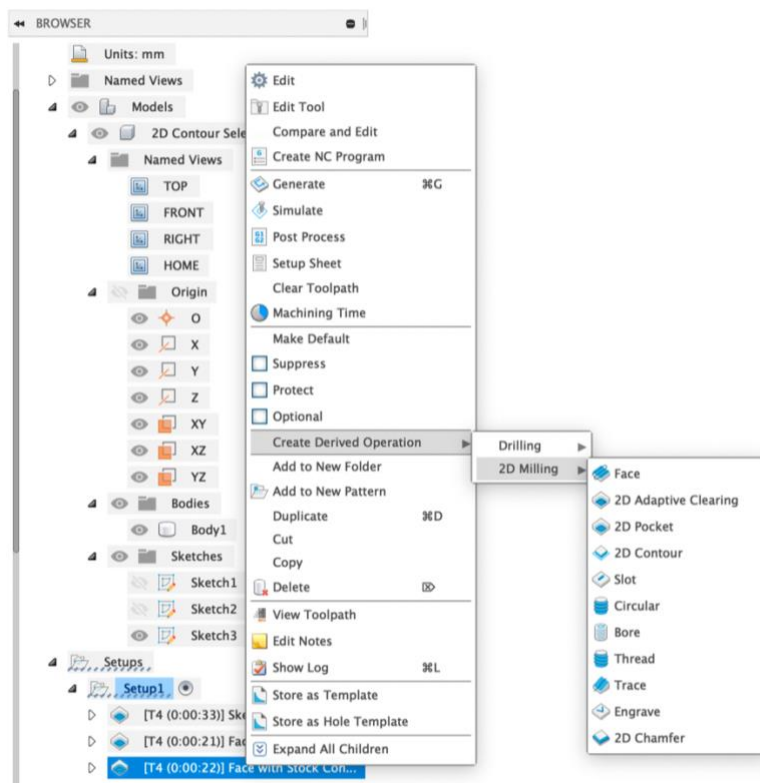


In this roughing toolpath, it is not important that the tool reaches every small area – this roughing operation will be followed by a smaller tool. In this case, the default MCR radius of 1/10th of the tool diameter will save about 40 seconds of toolpath generation time and half the size of the program. While this can be seen plainly when viewing the two toolpaths, it was not readily apparent which parameter was the cause before isolating variables via Compare and Edit.

Derived Operations

Derived operations allow alternative toolpaths strategies to be created quickly while maintaining the same parameters as an existing toolpath. This allows different types of machining operations to be efficiently created and observed during the programming process.

Derived operations are created by right-clicking any existing operation in the MFG browser and selecting “Create Derived Operation” from the menu:

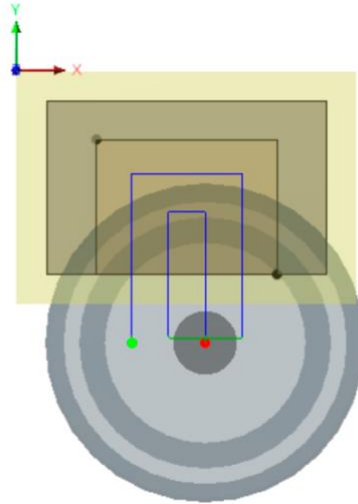


Fusion 360 can create a derived operation of the same type from any operation. For example, any 2D milling toolpath can be derived into any other 2D strategy. A 3D milling operation can be derived into any alternative 3D strategy, and a Turning toolpath can be derived into any alternative Turning strategy.

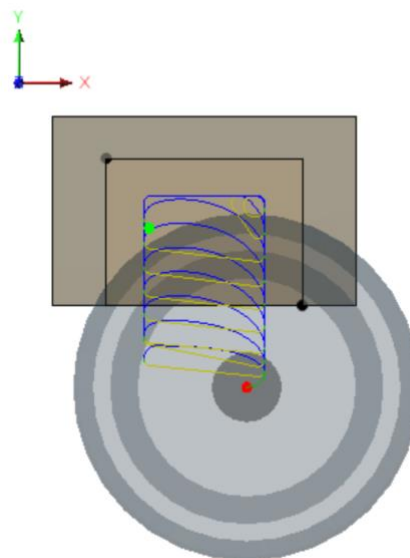
When a derived operation is created, all of the toolpath parameters that can be transferred over from the source operation will be copied into the derived operation. Any parameters that are not present in the source operation will use the default values.

A simple example of this involves entry strategies for 2D or 3D Pocket operations. In some cases, it is desirable for a roughing operation to enter a pocket that is open on one side from

that side. If the Pocket selection is used, the toolpath can be forced to plunge outside the stock, but it is very difficult to force Pocket to avoid full tool engagement when it enters from the side. Pocket strategies will give a result like the screenshot below:

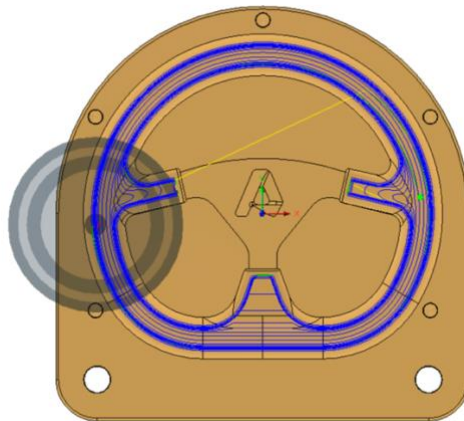


The Pocket operation enters from the side but does not account for tool engagement and therefore has inconsistent (and heavy) tool loading during cutting moves. Adaptive roughing operations do take tool engagement into consideration and may be a better option if side cutting and entry from outside the stock is desired. By creating a derived Adaptive operation from the original Pocket toolpath, a toolpath with consistent tool loading and side entry can be quickly created even if it was not the original strategy applied to the particular geometry. The derived Adaptive result from the original Pocket toolpath is shown below:

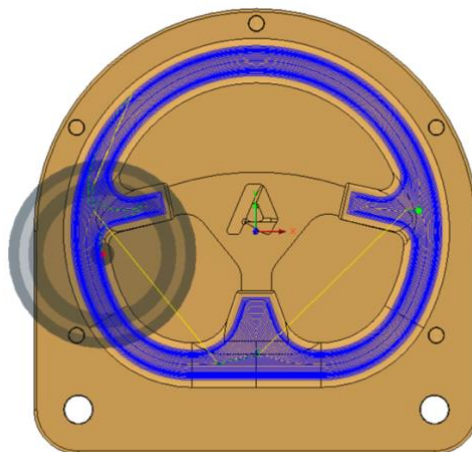


This roughing operation uses derived operations to improve tool entry and loading throughout the toolpath. Derived operations can also be used to quickly visualize different 3D Finishing strategies, as all parameters will be carried over – no geometry or tools will need to be selected and no non-default parameters will need to be re-entered.

For example, the 3D Contour toolpath shown below may do a good job on the steeper areas of the mold shown below but it will leave cusps in the shallower areas with the ball-end mill.



This could be resolved by duplicating the 3D Contour operation, applying slope containment, and reducing the stepdown for the shallower areas. This will not, however, completely resolve the cusps at very shallow areas like the bottom center of the toolpath area. It is easier and quicker to create a derived operation from the 3D Contour that will use a strategy with a constant *stepover* rather than *stepdown*. Scallop is a good option and deriving the existing 3D Contour into the Scallop strategy gives the result below. This consistent stepover will give better results at the machine and is quickly and easily generated with Derived operations, even if the initial 3D Contour strategy does not give optimal results.



Integrated CAD

Fusion 360 differentiates itself from most other CAM software by integrating manufacturing options into the same tool used for design. Using Fusion 360, designers can work with manufacturers in a single file throughout the entire design and manufacture process. This eliminates the conversion problems and repeated work prevalent in traditional CAM-specific packages and allows manufacturing plans to quickly adapt to design changes.

Integrated CAD/CAM has other advantages as well. Beyond ensuring design changes can be efficiently absorbed by the manufacturing plans, the ability to create sketches, surfaces, or construction bodies allows for increased flexibility when working towards specific results.

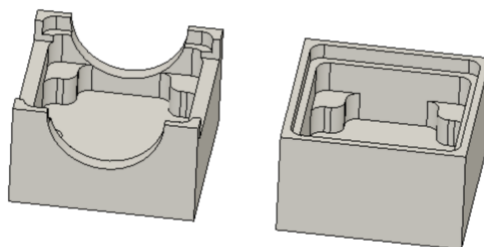
The limitations of using CAD to tweak CAM results is dependent on modeling experience – the more easily geometry can be created as a means to a specific MFG end, the more powerful this integration becomes. Expert multi-axis programmers often create construction surfaces to drive toolpaths. Using integrated CAD to force CAM results is not a “workaround,” it is an important tool.

The basic uses of integrated CAD while performing CAM programming include defeating models and using sketch or surface geometry as alternative boundary or machined geometry. More advanced applications include fixing/improving imported data or low-quality surfaces.

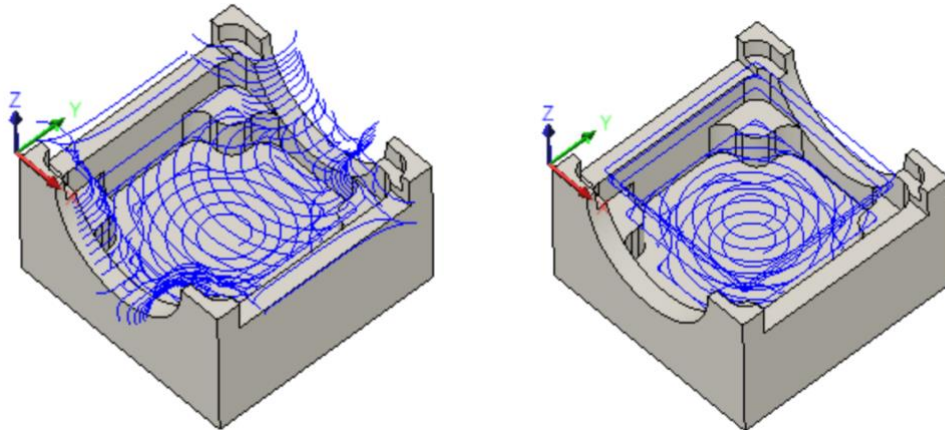
Defeated Models and Simple Edits

Defeating or simplifying models is a simple but powerful tool. These techniques can be used to include or remove features that should be avoided during roughing operations or create stock for finish machining on cast parts.

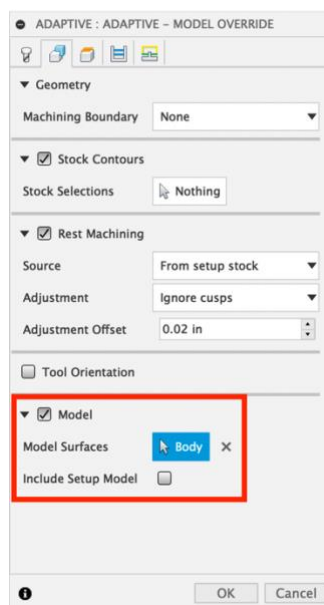
Simplified models can be created most easily during the initial design process, if a design for manufacture approach is taken. In the example below, the body on the right was created during the design process, before the side cutouts were added as model features.



Using the model to the right will make creating roughing operations much easier and ensure clean results on the curved walls at the corners of the part. The Adaptive Roughing operations shown below illustrate this clearly.

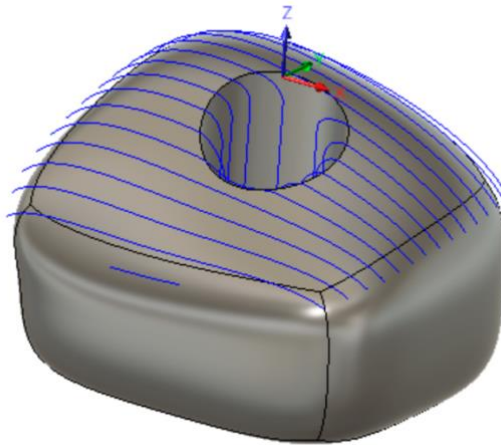


The toolpath on the left uses the final part geometry and the 3D Adaptive default parameters. The operation on the right uses the simplified geometry shown on the previous page and the 3D Adaptive default parameters. This operation will give cleaner results by avoiding the rollover and allowing the features that were removed from the final geometry to be avoided during roughing. To accomplish this, the defeatured body is selected in the “Model” field within the “Geometry” tab. The Model selection is a model *override* – because “include setup model” is not enabled, the 3D operation will be aware of the defeatured model rather than the finalized model selected in the parent setup.



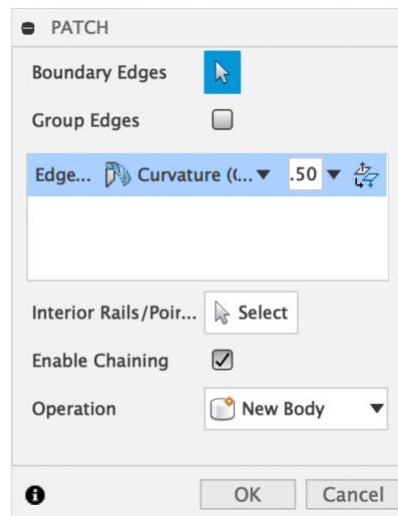
In 2D operations, geometry from the defeatured model can simply be selected in place of geometry from the final modeled part. This is because 2D operations are not model aware – they only see the selected geometry.

Another example of this could occur when attempting to avoid holes to be machined after surfacing operations or later in the manufacturing process. The example below gives a simple example – the tool should not drop down into hole.

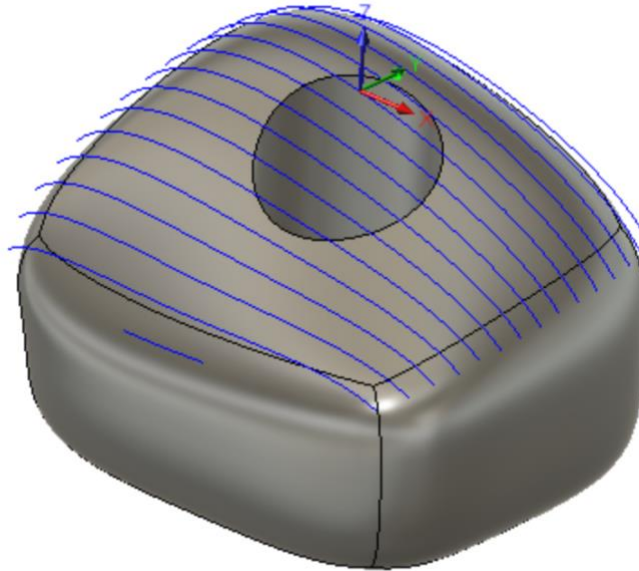


To avoid dropping into the hole during surfacing, the body could be copied and the hole feature could be removed from the copy. The new body would then be selected in model override.

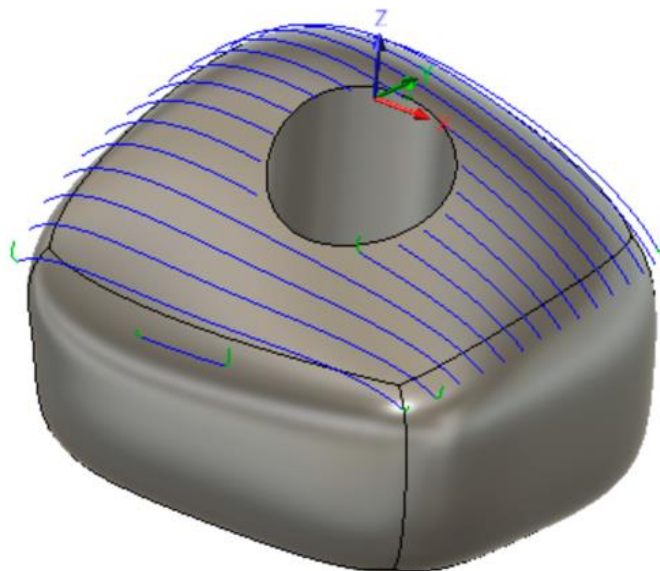
Alternatively, a patch surface could be created in the Design workspace (Surface tab) to cover the hole. The Patch can be created with curvature (G2) boundaries, ensuring the complex surface is continuous.



The “Model” parameter in the CAM workspace can then be enabled with “Include Setup Model” enabled. This results in the toolpath below. The patch is hidden and the toolpath no longer dives into the hole geometry and surfaces the part continuously.



While “Avoid/Touch Surfaces” could be used to avoid dropping into the hole (by selecting the hole face as a surface to avoid), it is necessary to create the patch surface to force the toolpath to machine over the hole. Avoid/Touch Surfaces gives the result to the left below when the hole face is selected.

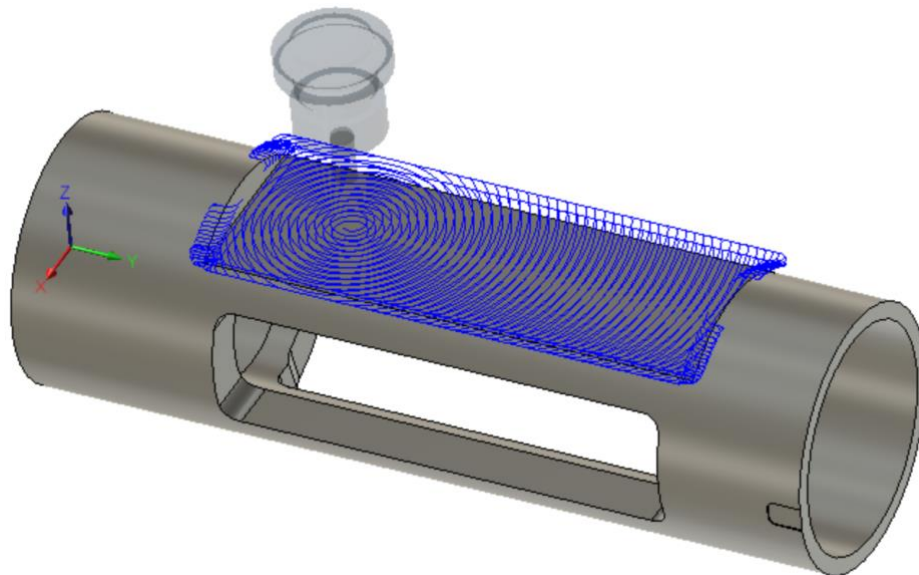


The examples above show how surfaces can be used with the Model or Avoid/Touch Surfaces options in the Fusion 360 Manufacturing workspace. Surfaces and sketch geometry can also be used for all other geometry selections – machining boundaries, stock contours, or as height references.

A common example of a situation that requires sketch or surface geometry as a machining boundary or stock contour is when certain edges of the geometry to be machined have different edge conditions (open or closed) than the others.

Creating Machining Boundaries

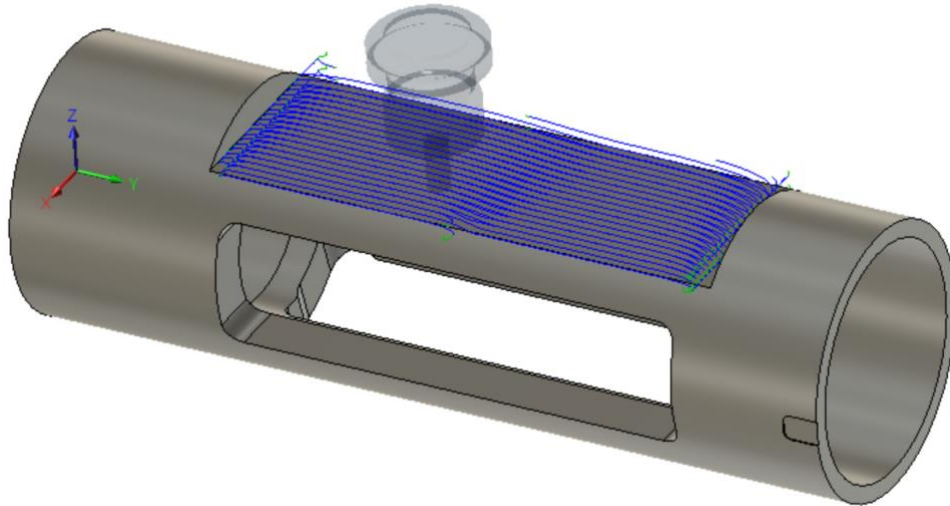
In the part below, a flat face needs to be cut on a cylindrical face. With the 3D Adaptive toolpath, it is desirable to enter the operation from the side of the face, allowing the tool to side cut. This would require the tool to be allowed outside the machining boundary. However, using the “Tool Outside Boundary” option also allows the tool outside the boundary on the other sides of the boundary. This results in unwanted wraparound in the cutting moves as shown below.



To avoid the wraparound, the machining boundary condition can be changed to “tool inside boundary.” This will, however, force entry inside the pocket and will not allow side entry.

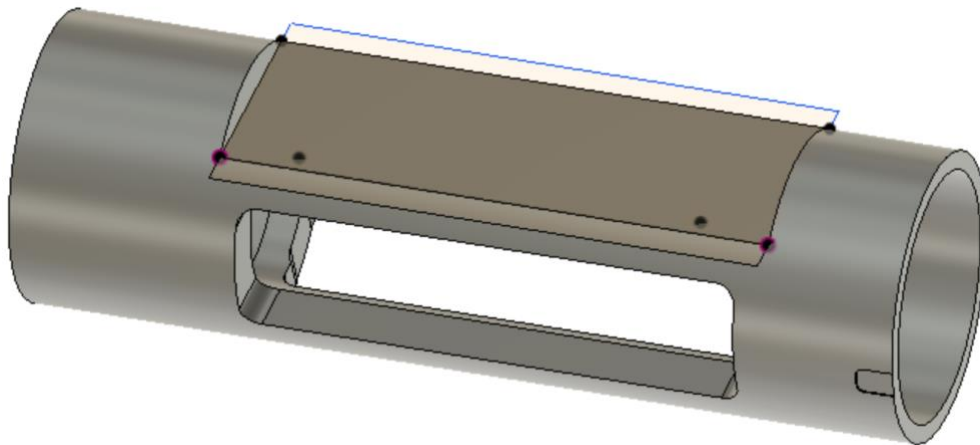
The boundary needs to be extended, but only on the open edges of the pocket. This can be done with a face selection and a 2D Adaptive toolpath. But, if this operation requires multiple stepdowns, they will all do the exact same passes on each cutting plane, as the 2D operations are not aware of the round stock. 2D Adaptive with a selection of the flat face will give the

results below. This forces side entry, which is desired, but will result in air cutting on the initial upper passes because the stock is not as wide the further above the face the cutting occurs.

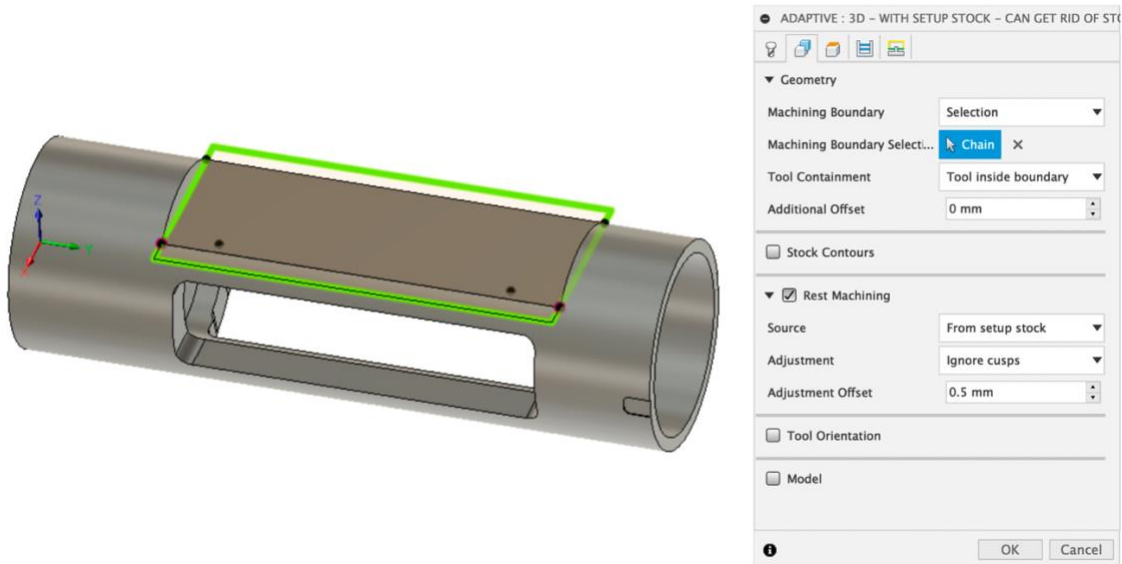


This is an improvement on the wraparound and helical entry in the initial 3D operation but the air cutting on the upper passes should be removed, particularly if a high volume of these parts will be made. To do this, a 3D strategy will need to be used so the shaped stock can be “seen” by the toolpath.

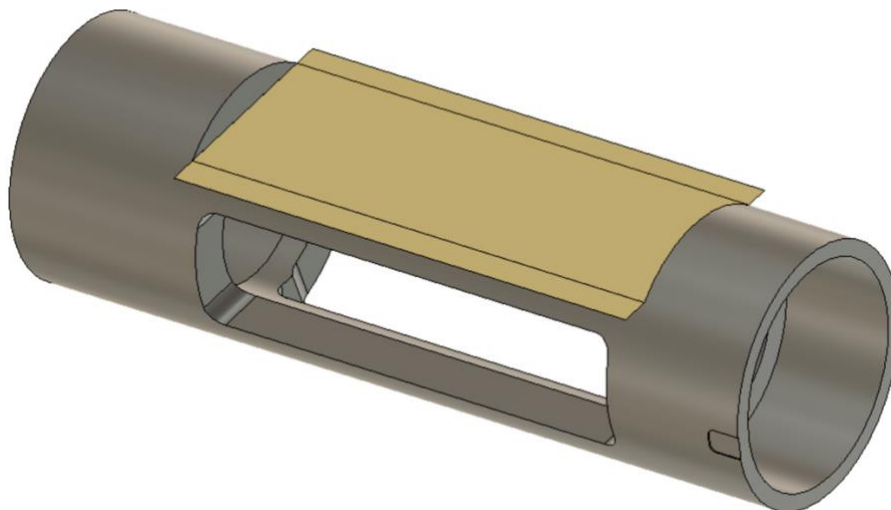
To use a 3D Adaptive toolpath on this geometry, a machining boundary can be sketched. The boundary will allow extra space on the open edges of the face (a little over the tool radius) so a “Tool inside boundary” condition can be used. This will avoid the wraparound on the closed edges of the face.



Applying the sketch above as the machining boundary with the “tool inside boundary” condition allows side entry, avoids wraparound, and allows rest machining from setup stock to be enabled to eliminate the air cutting on the first stepdown.

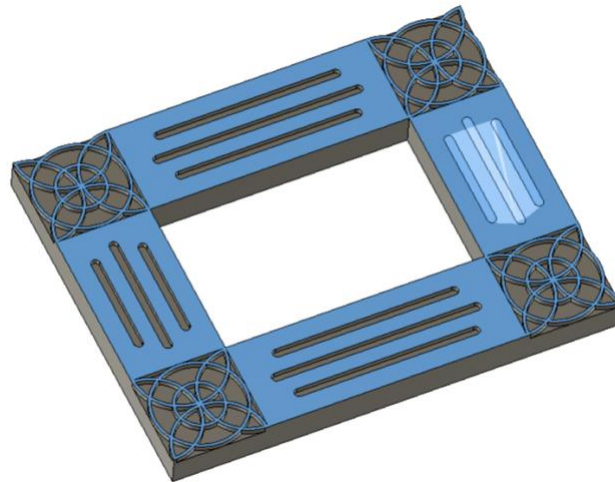


The settings and sketch boundary selection shown above give the toolpath result below, which avoids air cutting and wraparound while allowing side entry. The sketch is easily created on the flat plane. Alternatively, a zero-offset surface could be created (from the flat face) and the open edges could be extended when creating geometry for the machining boundary. In this case, the sketch or surface can be used interchangeably. If the alternative selections were being used to drive the cutting passes of the toolpath in a different manner than the original model allowed, a surface would need to be used instead of the sketch selection.



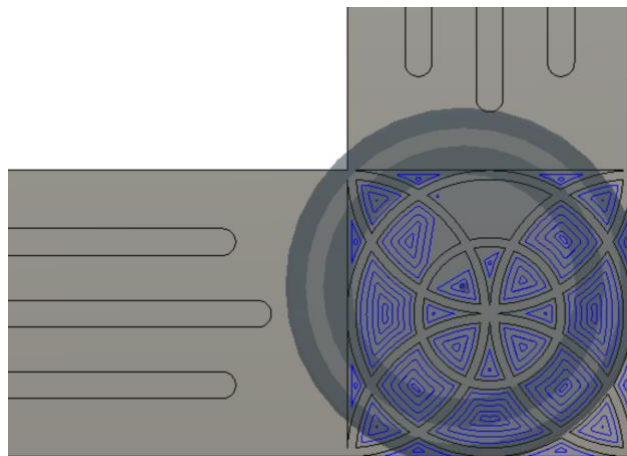
Imported Data

Because Fusion 360 includes both the Design and Manufacturing workspaces, it can also be used to clean up imported files. In some cases, imported data will exhibit graphics faceting, as shown below to the right-hand side.



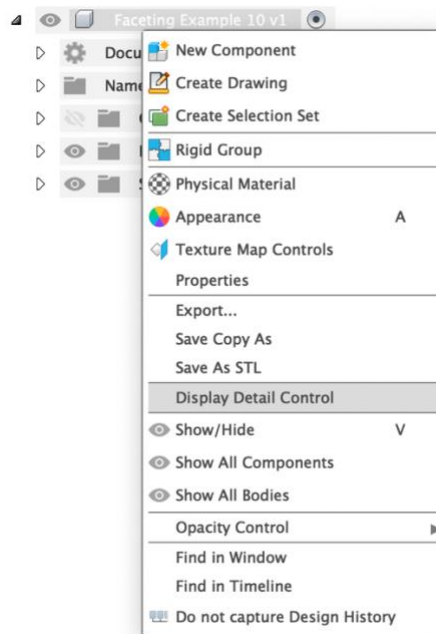
The slots near the heaviest faceting errors are easy enough to work around with a 2D Slot operation. Because the 2D operations only see the selected geometry, selecting one of the clean edges on the top face and manually setting the bottom height allows 2D operations to generate with no errors.

For the patterns in the corners, it is desirable to use a 3D operation to avoid the need to select every internal edge in the pattern in a 2D operation. When creating a 3D Pocket operation on this geometry, the faceting issue is shown in areas it is not visible when viewing the model. This is illustrated in the missing areas of the toolpath below.

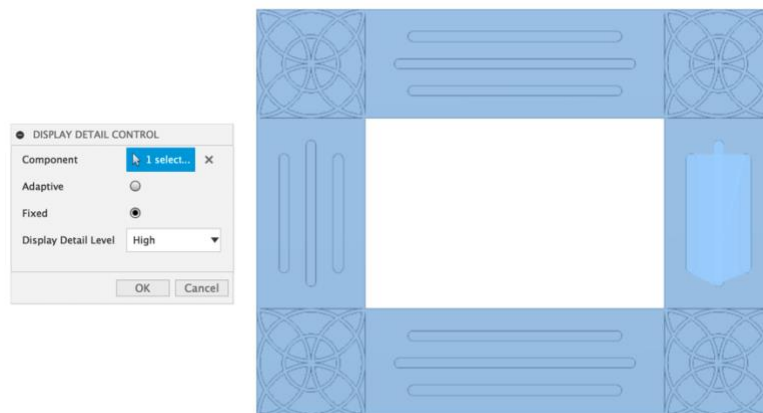


While the model geometry is shown correctly, the toolpath is not machining all the desired geometry in the knot pattern. This is due to the way the imported data is toleranced in both the Design and Manufacture workspaces.

In the Design workspace, graphical tolerancing is controlled by the “Detail Display Controls” parameter, accessed by right-clicking any body or component in the Design Browser.

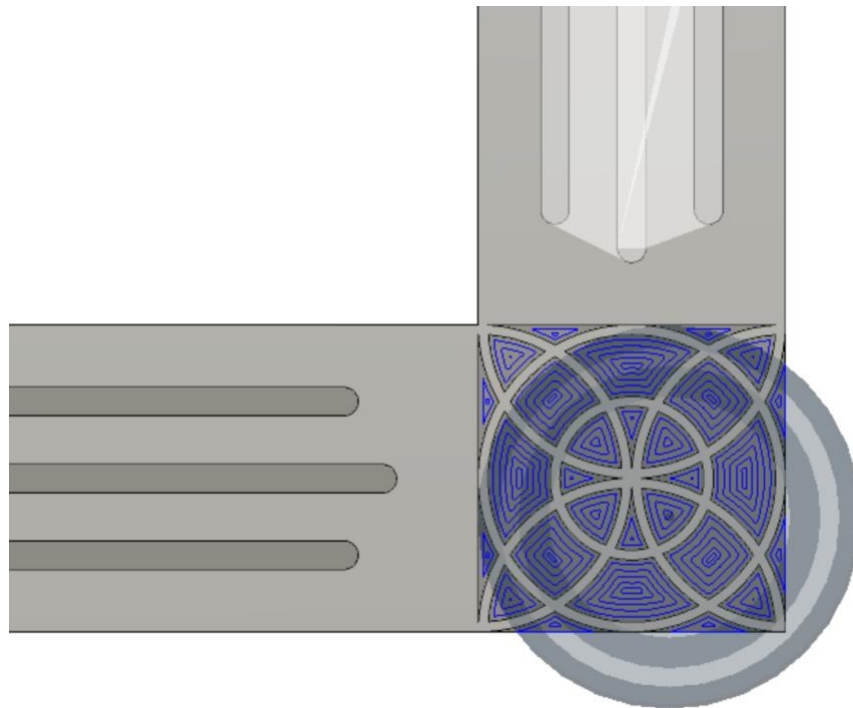


Switching to Fixed > High detail display levels often resolves these visual issues in the Design workspace, but it does not with this example file. It does, however, help identify why this problem is occurring.



The faceting observed in this model is due to the graphical tolerance of Fusion 360. This tolerance works in the same way as the toolpath tolerance discussed earlier in this document, but defines the way the model is shown on the screen instead of the “mesh” model created during toolpath generation. Because this identifies that the issue is due to the model tolerance, the toolpath tolerance is the next logical step in troubleshooting.

If the toolpath tolerance in the 3D Pocket operation is reduced to 0.002” (down from the default value of 0.004”), the operation will generate as expected, even though the faceting is still seen in the model.

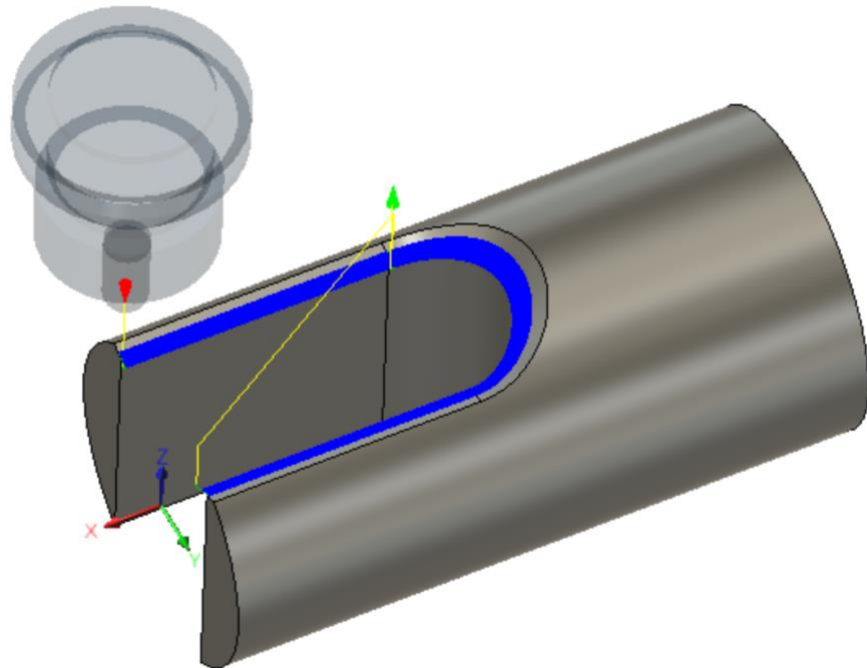


This takes some experimentation, as a tolerance value of 0.003” shows no difference from 0.004”. Reduce the tolerance by a small amount until the toolpath generates correctly, as too low a tolerance will result in very large NC outputs with many linear segments. Reducing the toolpath tolerance below the graphics tolerance allows the toolpath to generate correctly, even though the issue can still be observed graphically in canvas.

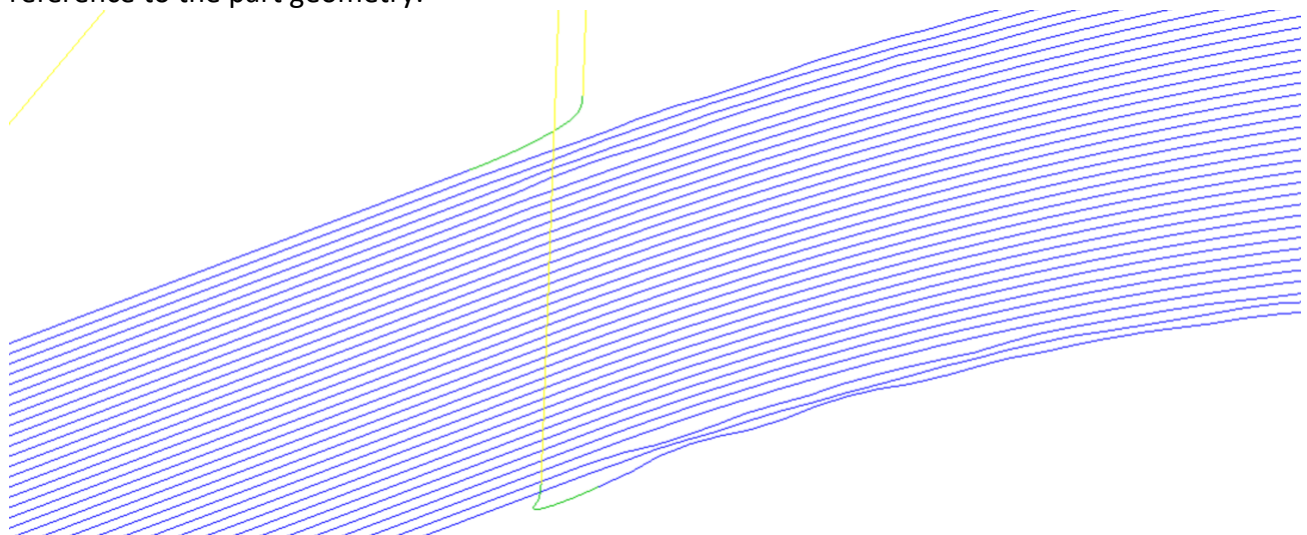
Surface Quality

Fusion 360 also includes the option to create G2 continuous surfaces in the Form/Sculpt workspace, which allows surface quality problems to be addressed. This is particularly important in simultaneous multi-axis operations but can also be used to improve results in 3-axis operations.

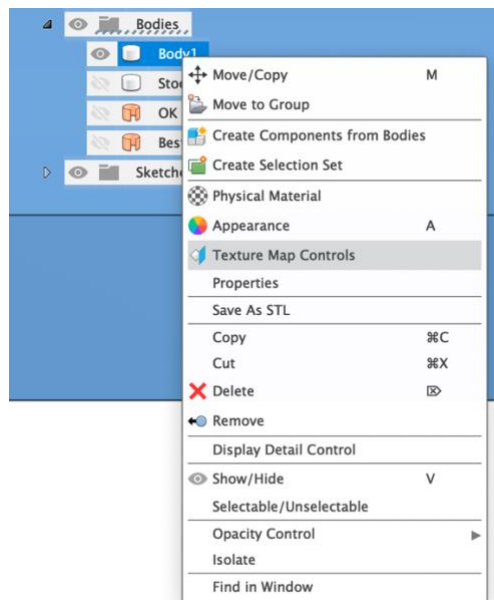
Toolpath problems due to poor model surface qualities are prevalent in all CAM systems, particularly when working with UV following toolpaths like the Flow strategy in Fusion 360. When working with this type of CAM strategy, it is not uncommon to receive results like below.



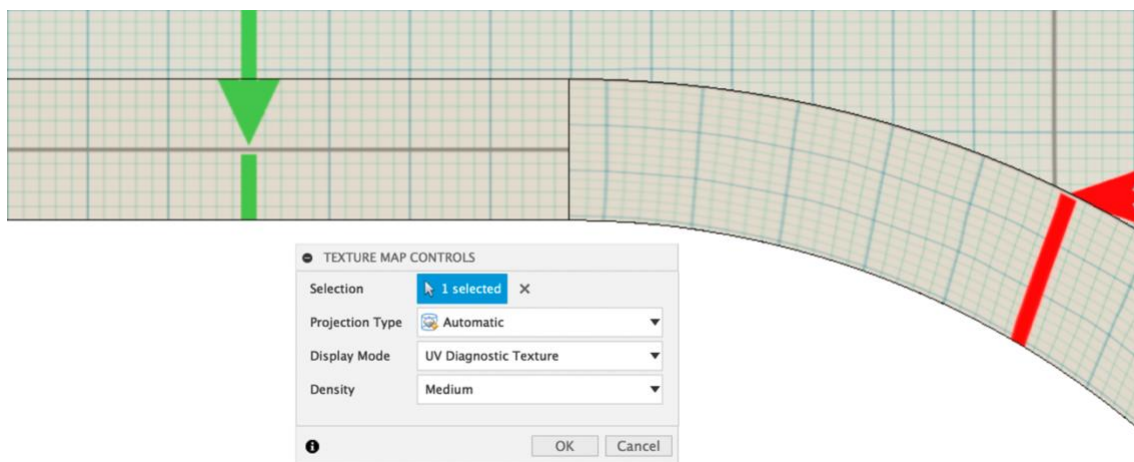
At first glance, this does not appear to be a bad result. However, when the retract is zoomed in on, inconsistencies between passes can be observed. In the screenshot below, the part is hidden so the lines of the toolpath can be clearly observed. The retract moves are shown for reference to the part geometry.



These passes are not obeying consistent stepovers and direction. This suggests that the surface may not be as cleanly modeled as it could be. A way to investigate this is to use the UV Diagnostic tools in Fusion 360, which are accessed by right-clicking any body or component in the Design browser and selecting “Texture Map Controls.”

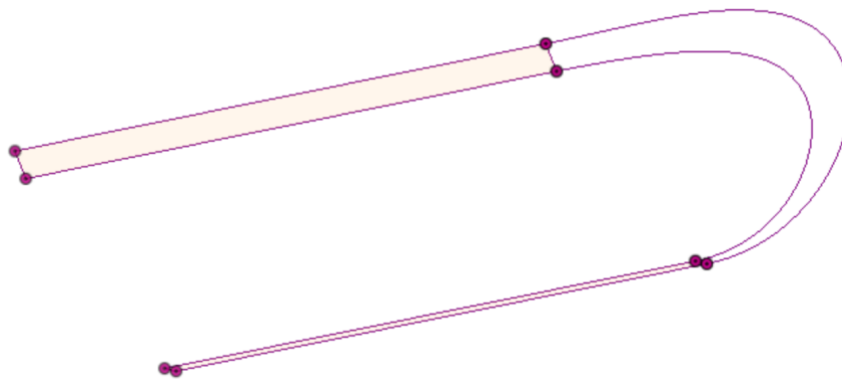


In the Texture Map Controls dialog, the Display Mode can be set to “UV Diagnostic Texture.” This will display the UV isolines for all faces on the selected body or component. In the screenshot below, investigation of the UV Diagnostic Texture shows the isolines do not line up where the straight and curved chamfer faces meet, which is where the retract and inconsistent passes were observed.

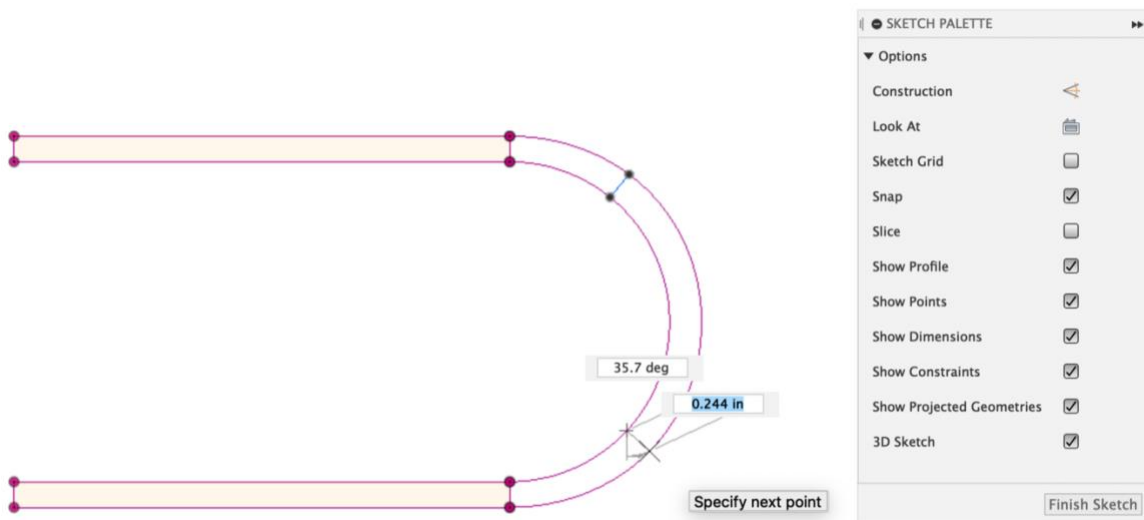


The UV Diagnostics examination shows that the poor toolpath quality is likely due to the isolines not lining up between the two chamfer faces – the Flow strategy follows these isolines. Because curvature continuity is maintained by Sculpted/Formed surfaces or bodies in Fusion 360, a new surface can be created to clean up the isolines and improve this result.

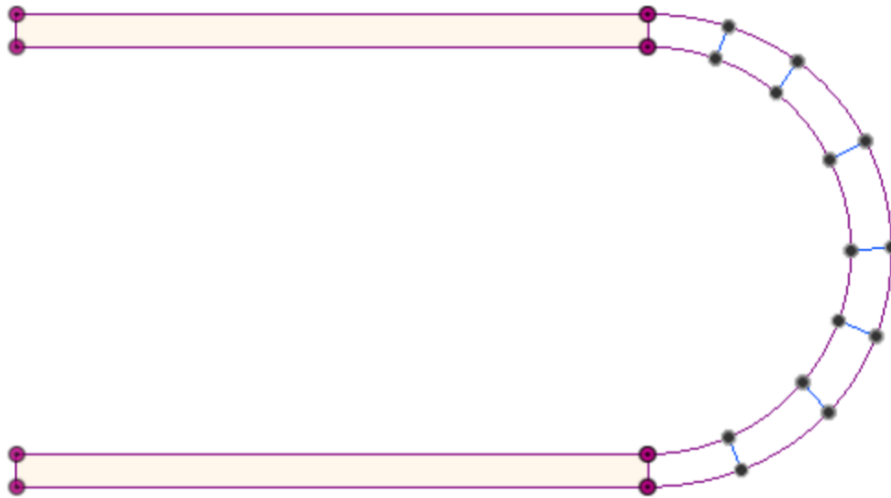
The original surface and its edges will be used as a starting point from which a lofted t-spline/sculpted/form surface can be created. To begin the process, a sketch will be created on any plane and the model edges of the surface will be added to the sketch through the “Include 3D Geometry” option (in the Sketch > Create > Project/Include menu). This results in the sketch shown below.



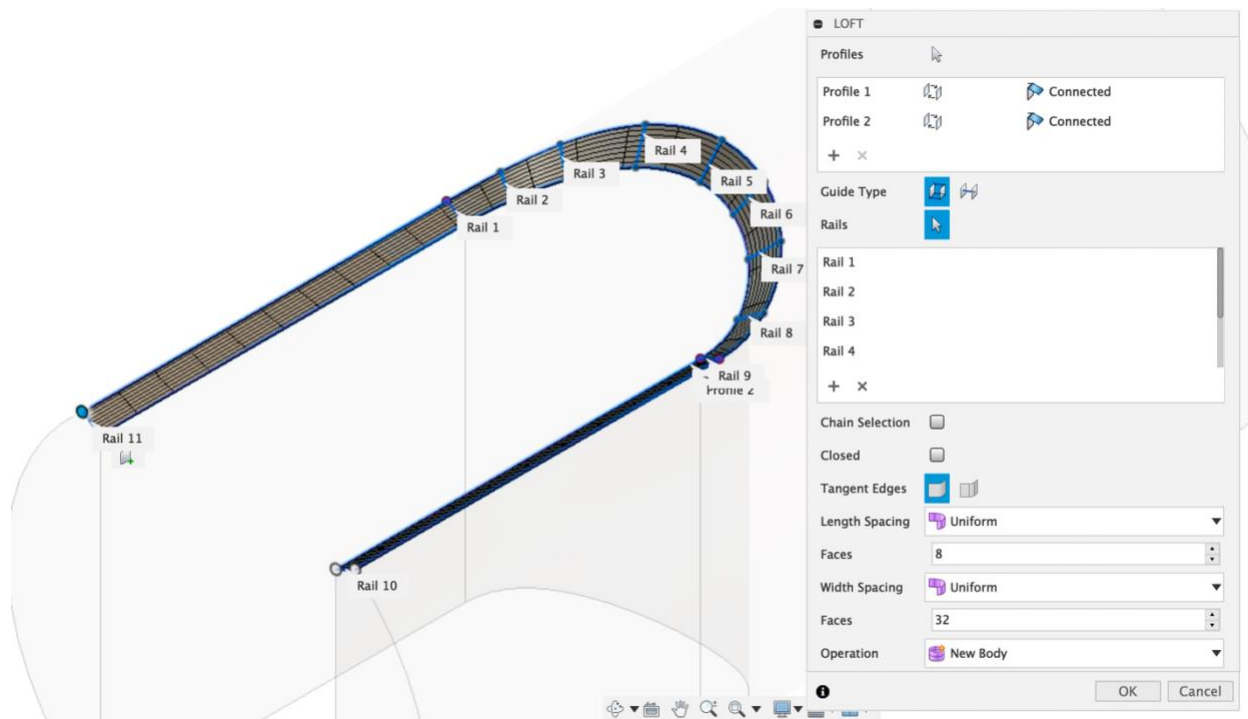
The goal will be to create a loft between the top and bottom edges of the contour. To do this, adding control lines between the two contours will be helpful. If “3D Sketch” is enabled in the Sketch Palette, black x-marks will be shown when a sketch endpoint will be snapped to the 3D geometry created earlier.



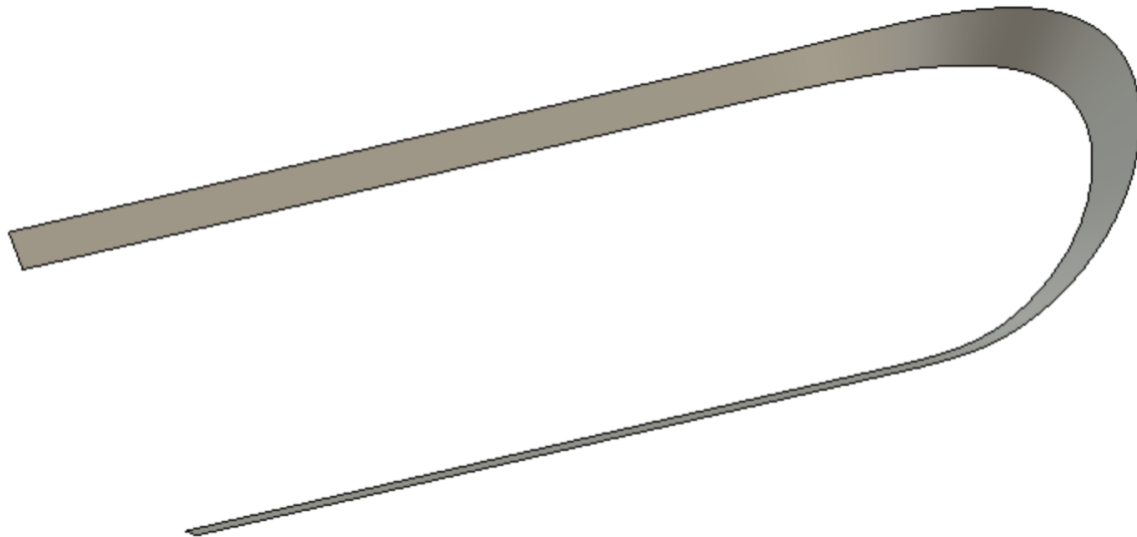
The rule of thumb is to create enough guidelines to accurately form the loft around curves, which can require some experimentation. In this example, the sketch shown below gave good surface results. For more complex examples, it may be beneficial to trace exterior curves with the Sketch > Spline tool to clean up the loft profiles as well.



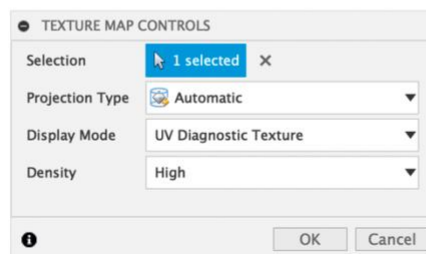
After creating the sketch, the loft can be built. This is done in the Form/Sculpt workspace. The number of faces for the length and width should be experimented with to match the original model geometry accurately.



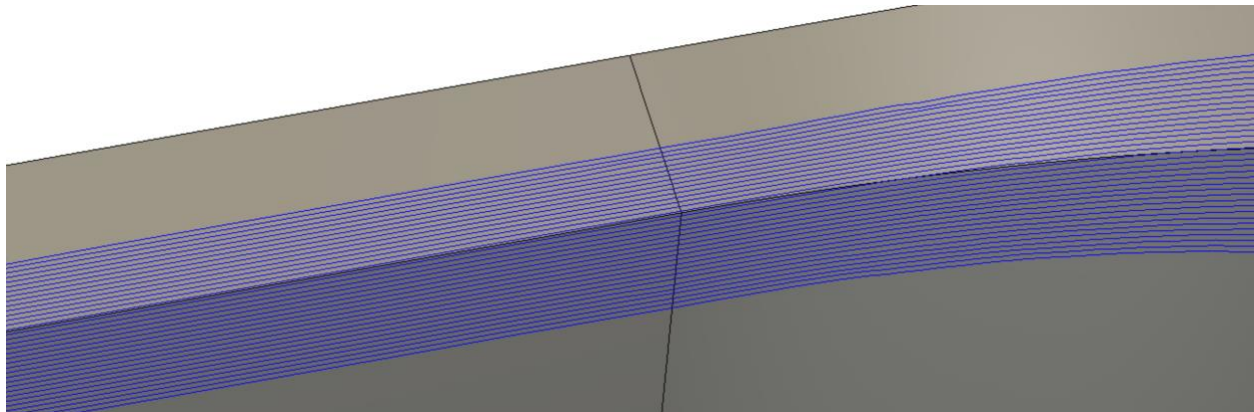
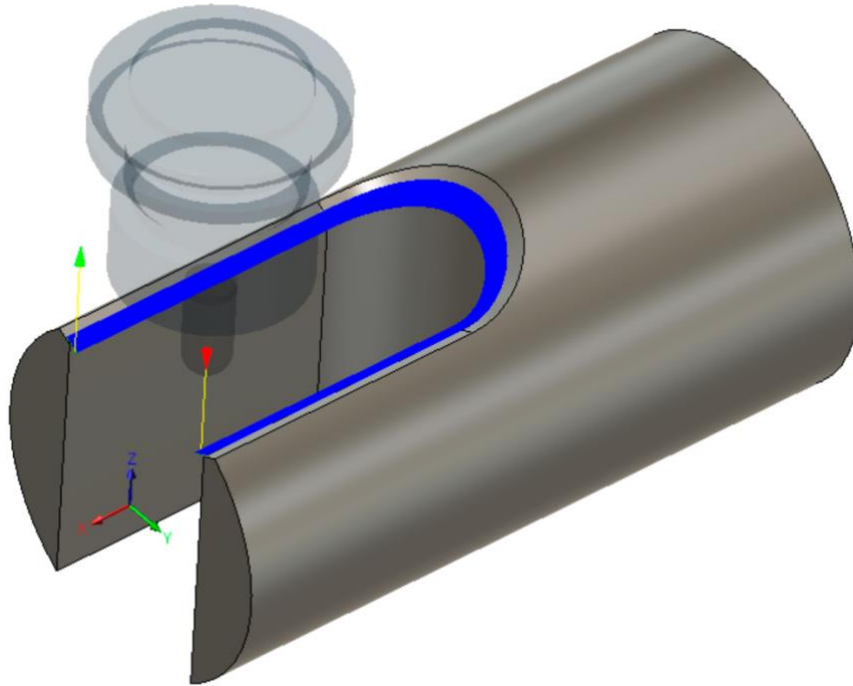
The form can then be finished, resulting in the smooth and continuous surface body shown below.



Observing the UV diagnostics on this surface shows that the UV isolines are much more continuous than the original surfaces. In the screenshot below, the red isoline is continuous and at a consistent distance of the edge of the surface.



The resulting toolpath avoids the unnecessary retract and smooths out machine motion as the cutting passes enter the curved section of the chamfered face. This will give a cleaner result at the machine because static between the surface sections is eliminated. The resulting toolpath and detail on the original problem areas are shown below. The resulting toolpath reduces the inconsistencies enough to give a good result at the machine.



This process does require knowledge of surface modeling concepts like UV isolines as well as basic knowledge of working with t-spline bodies in the Sculpt/Form workspace of Fusion 360. Autodesk Advanced Manufacturing Sales team member Dan Pacific gives a great, in-depth demonstration of these concepts in the Fusion 360 Academy class in his [“Surfaces: The Multi-Axis Toolpath Control You Never Knew You Had” recording](#), available in the Fusion 360 Adoption Portal.

Additional Resources

Fusion 360 and MFG Learning

- [The Autodesk Knowledge Network](#)
- [Fusion 360 Learning Materials – Adoption Portal](#)
- [Fusion 360 Learning Materials – Product Documentation](#)
- [Fusion 360 YouTube Channel](#)
- [Autodesk HSM YouTube Channel](#)
- [Fusion 360 Academy, Portland 2019. Manufacturing Track Classes](#)
- [Autodesk Fundamentals of CNC Machining Handbook Download](#)
- [Post Processor Customization Policy and Resources for Fusion 360, Inventor CAM, and HSMWorks](#)

Get Support!

- Get self-serviced, peer, and support from the Autodesk Team on the [Fusion 360](#) and [HSM CAM](#) Forums!
- Create a Support Case with the Fusion 360 Product Support Team!
 1. Go to manage.autodesk.com
 2. Click the “Contact support” link
 3. Choose what you need help with from the list of support teams
 4. Enter the product you are working with
 5. Describe the problem you are experiencing
 6. The support options available with your subscription will be shown. Select the best option to create the support request!