MFG225409-L

# Advanced Toolpath Editing in PowerMill 2019

**Christopher Marion** - Speaker
Autodesk

**Todd Steele** – Lab Assistant
Autodesk

## Learning Objectives

- Go through some of PowerMill's basic editing options as well as some of the more advanced toopath options that even a 3-axis programmer can take advantage of
- Learn how to run a full machine simulator and detect collisions or near collisions
- Learn how to find potential problematic areas during the simulation and correct them before sending the file to the machine
- Gain a basic overview of PowerMill software's different types of tool axis options for simultaneous programming
- Learn how to manipulate specific regions of a toolpath and override the behavior of the machine tool

## Description

This class will venture into some of basic editing tools that are offered with PowerMill.

We will them apply these to some of the advanced editing tools, like the Dynamic Machine Control and Editing Within a Region. We will demonstrate these editing tools live.

## Speaker(s)

Christopher Marion is Product Specialist with the Advanced Manufacturing group of products that are offered by Autodesk. He is based out of the Windsor, Ontario, Canada office. He offers support, training and technical consultation in PowerMill, PowerShape, FeatureCAM & Fusion Production across North America.

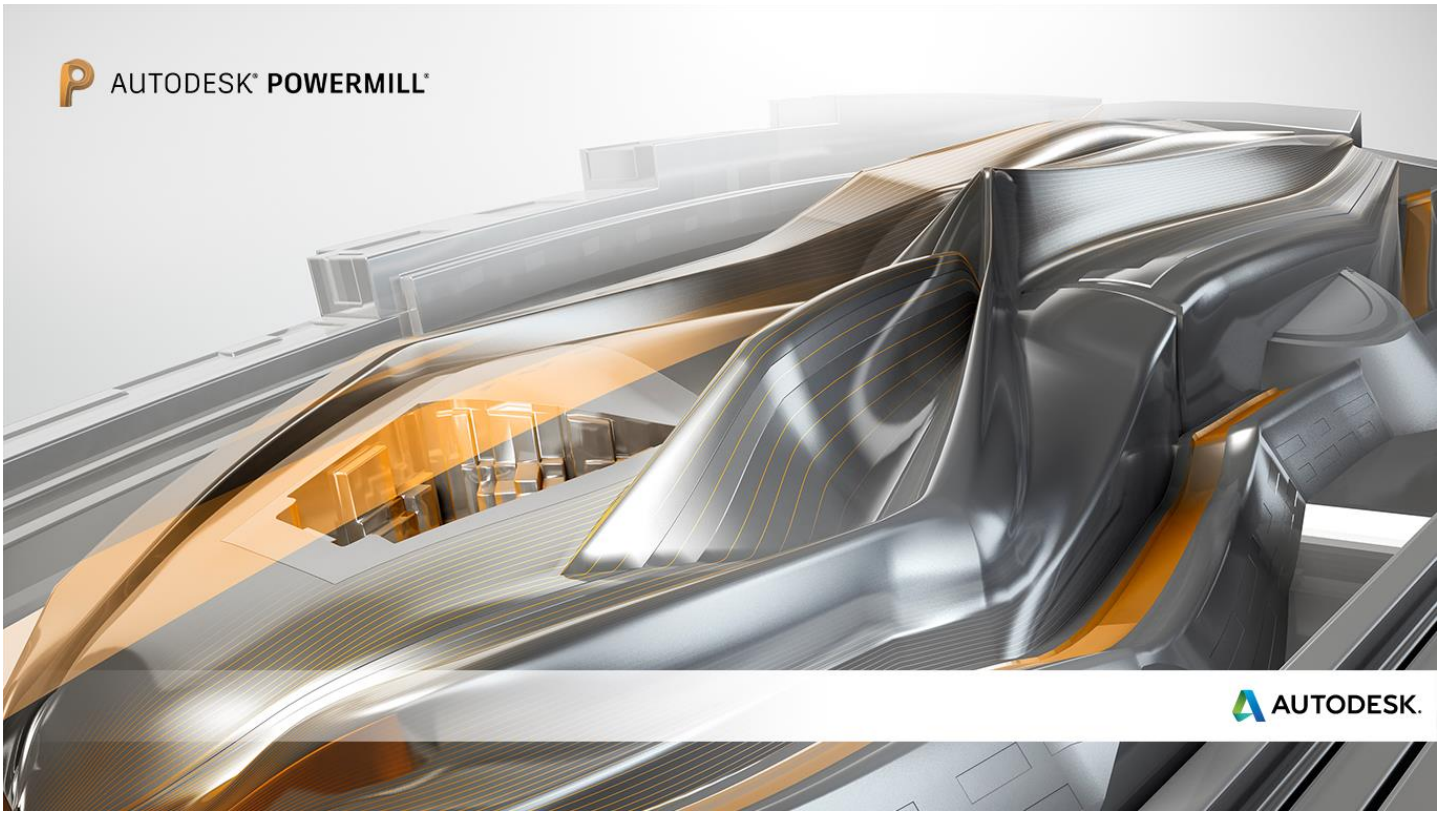
His 20+ years of direct industry related experience in subtractive manufacturing has given him the ability to convey his knowledge to others during conversation about these products. Before coming to Autodesk, Christopher worked as a CAM programmer, a Designer (plastic injection molds and fixturing) and as a continuous improver in process planning and development.

In Christopher's free time, he enjoys spending time with his wife, son, daughter and dog Jasper. He loves to spend time on the ice coaching and playing hockey (He's Canadian, what did you expect?) as well as root for his favorite inept sports teams (Detroit Tigers, Lions and Red Wings).

AUTODESK. UNIVERSITY

# Table of Contents

# PowerMill Introduction



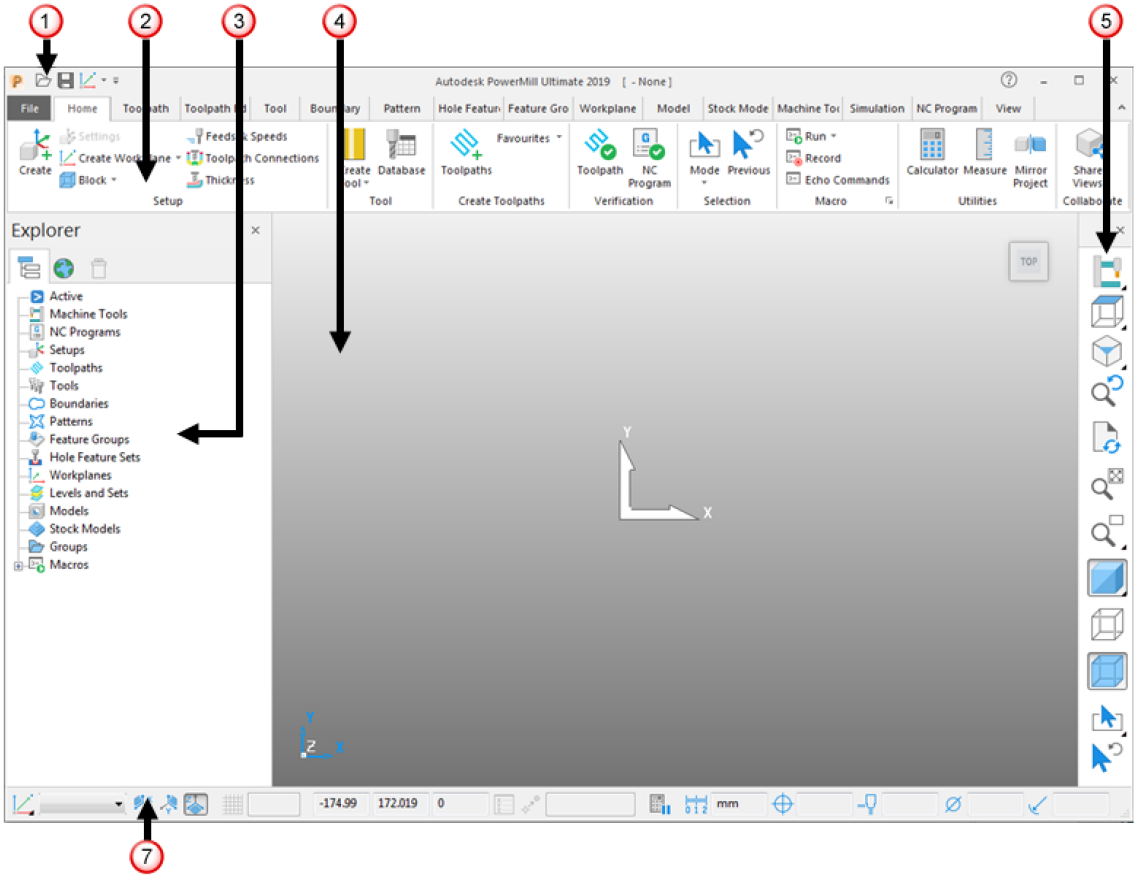

PowerMill is a stand-alone CAM system that quickly produces additive toolpaths, inspection probe paths, and gouge-free NC toolpaths from CAD models using 2.5D machining, 3-axis machining, or multi-axis simultaneous machining. These toolpaths can then be checked for collisions against other models (for example, clamps) and the tool holder before being output to cutter location and tape files.

PowerMill has a wide variety of direct interfaces. It supports IGES, VDA, and STL data formats, which enables you to import data from any CAD system that supports these neutral formats.

Using PowerMill's simulation, you can load entire machine tools to verify toolpaths and visualise machine and cutting tool actions from different perspectives.

# Screen Layout

The following window is displayed when starting up PowerMill




1 **Quick Access Toolbar** - Enables you to quickly access frequently used ribbon commands, such as Save, without having to navigate the ribbon.

2 **Ribbon** - A collection of tabs and groups that organise PowerMill functionality.

3 **Explorer** - Provides control over all PowerMill entities.

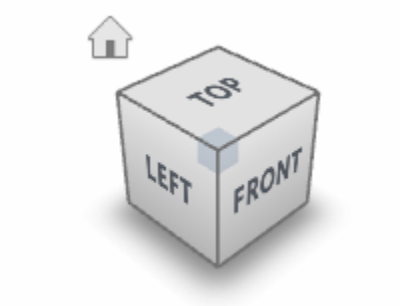
4 **Graphics Window** - The working area of the screen.

5 **View toolbar** - Provides quick access to common views and shading options in PowerMill. The full range of viewing options are available on the view tab.

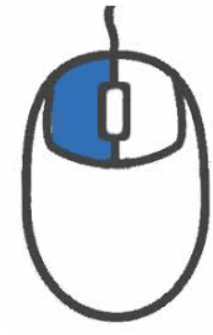
6 **ViewCube** - Enables you to interactively orientate the contents of the graphics window.

7 **Status and Information toolbar** - Enables you to create and activate workplanes, display various preset and user-defined settings. If you hover the cursor over a button, help is displayed. The help can be, for example, a brief description of the item beneath the cursor, or information about the calculation that is in progress

# Mouse Manipulation



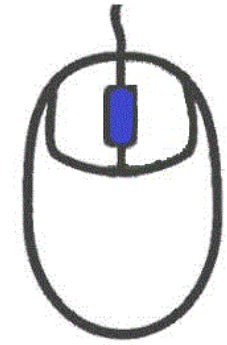

The View Cube - used to manipulate the view orientation, either dynamically, or by using the fixed views.



**Left Mouse Button** - Clicking the left mouse button on an entity in the graphics window allows you to select it.

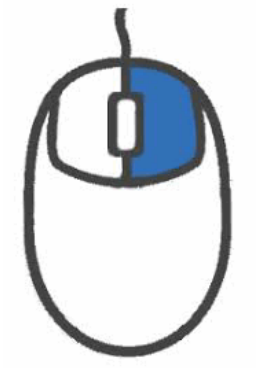
Holding down the left mouse button and dragging it to the left and upwards or to the right and downwards, allows for selection of entities inside of a box.

Double clicking on an item in the Explorer allows the user to quickly activate it.



**Center Mouse Button** - Clicking and holding down the middle mouse button down and moving the mouse around the graphics area allows you to rotate the model.

Scrolling the mouse wheel (if equipped) back and forth does allow for zooming towards the model or zooming away from the model too.



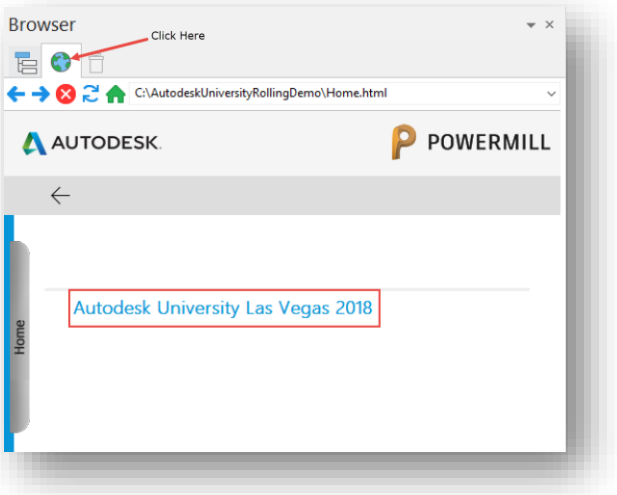
**Right Mouse Button** - Right clicking on an entity will open up menu's.

Right clicking on an item in the Explorer, will produce a different menu and right clicking on the model in the Graphics Window.

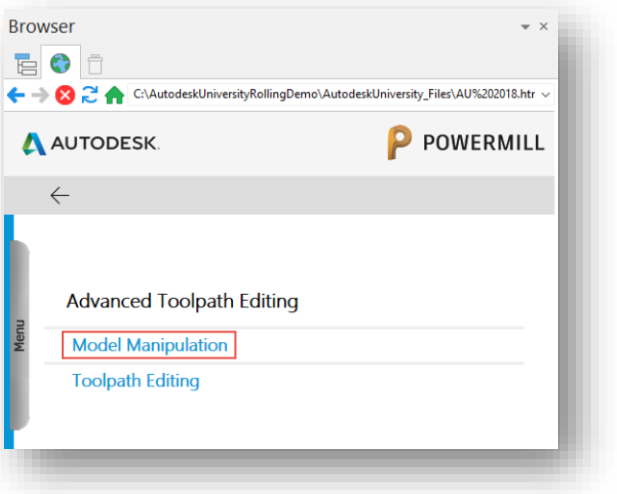
# Interactive Manipulation Examples

Let's get familiar with PowerMill's mouse and interface controls.

Go to the browser the is located at the top of the Explorer on the left-hand side of the UI. Clicking on the browser tab will display the home page of where we will be working during this lab.




Then click on the Autodesk University Las Vegas 2018 link to go to the next page. From here, click on the Model Manipulation link to get to the interactive examples on basic mouse functions needed to progress further in this course.



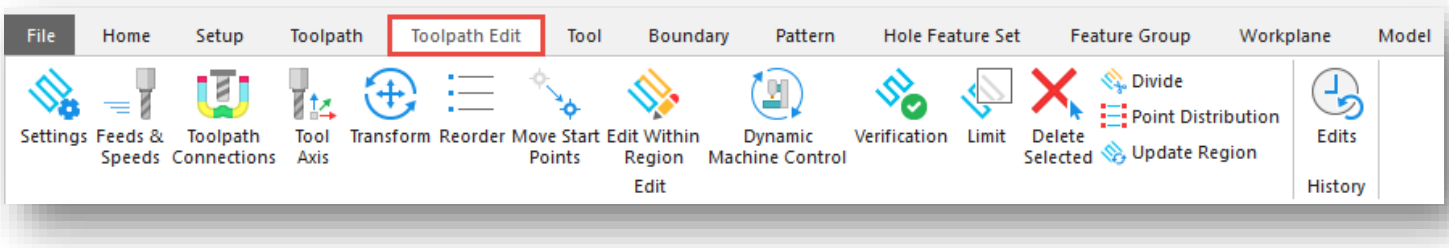

These examples should only take approximately 10 minutes. Please read through everything carefully

# Toolpath Editing

A huge advantage PowerMill has over its competitors, is that a lot of changes that the programmer needs to make to a calculated toolpath can be done afterwards.  PowerMill allows quick Modification to **non-cutting** segments items like speeds and feeds, change entry and exit moves (leads and links) as well as intuitive toolpath editing functionality which enables you to easily reorder or remove segments of a toolpath, and to modify the properties of toolpath segments such as start and end points.

Toolpath editing allows for quick tidy up unwanted toolpath segments, which leads to more efficient toolpaths. Additionally, toolpath editing is used to improve the safety of your toolpaths by avoiding collisions or alleviating problems on specific machine tools. For example, editing the tool axis across a segment to avoid a collision with the machine spindle or table.

You can access many of the functions you need to edit toolpaths from the **Toolpath Edit** tab on the ribbon.

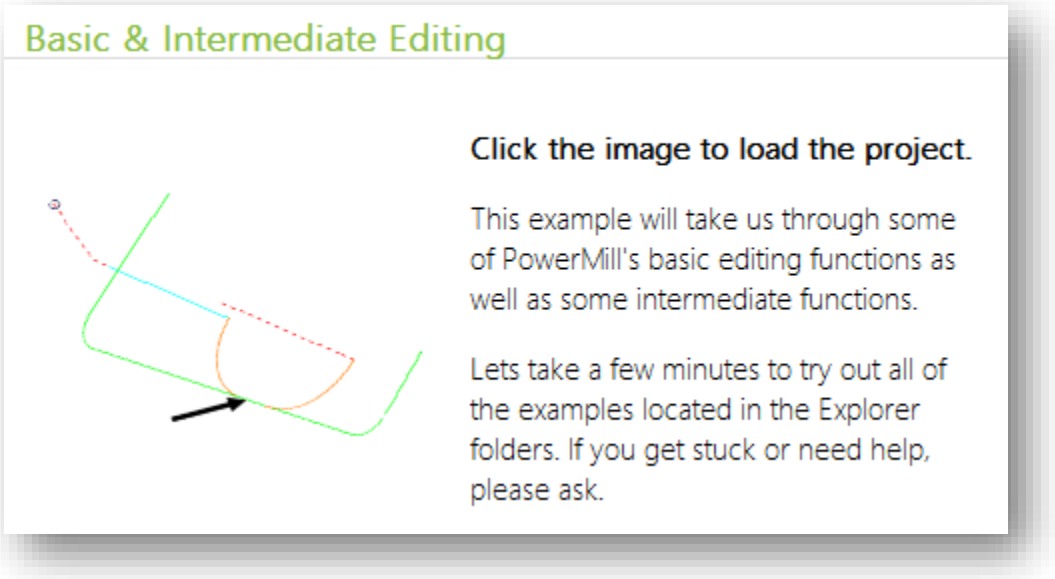


# Basic & Intermediate Toolpath Editing

Let's get familiar with some of PowerMill's basic and intermediate toolpath editing options.

Go back to the browser and navigate to the **Advanced Toolpath Editing** page.

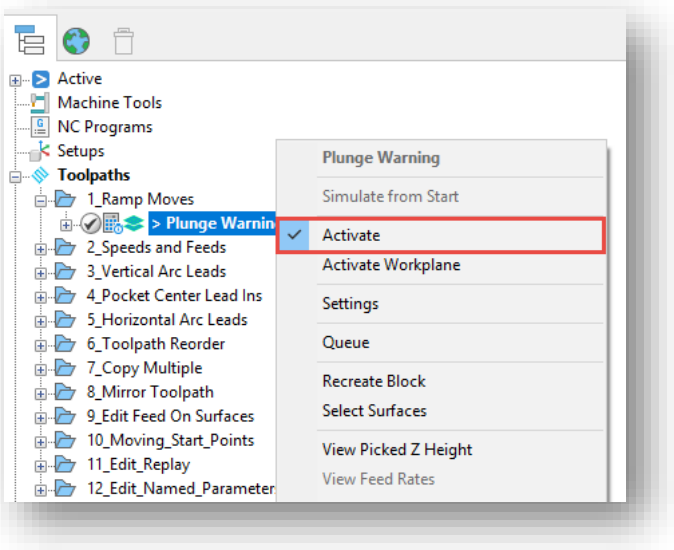
Click on the Toolpath editing link (shown above), then click on the image in the **Basic & Intermediate Editing** section to load in the project.



# Ramp Moves

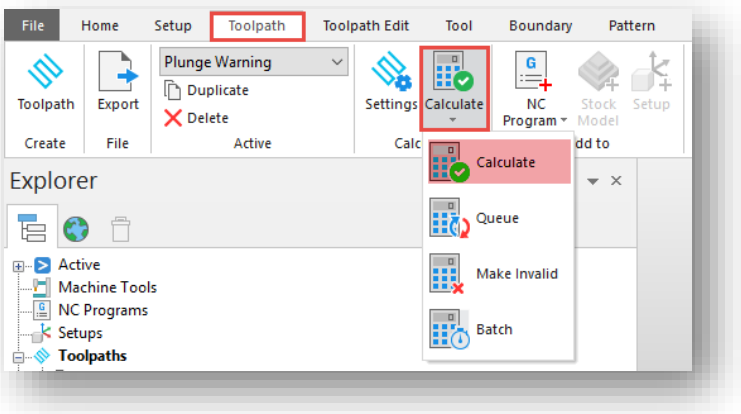
PowerMill has the intelligence to warn the user about movements into the stock that need some attention during roughing operations. In this first example, expand the Toolpaths branch that is located in the Explorer and expand the **1_Ramp Moves folder**. In this folder is the uncalculated toolpath **Plunge Warning.**

We need to activate this toolpath before we can calculate it. Right click on the toolpath to open up the menu of options available on this toolpath. Select **Activate** from the menu. You will notice that the toolpath in the folder has now been bolded.

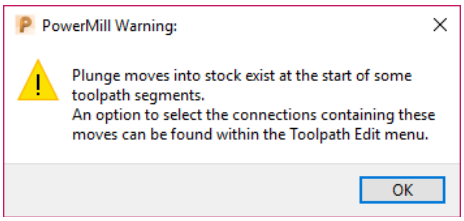



Now:

1. Go to the **Toolpath** tab, located on the ribbon at the top of the UI
2. In the **Calculate** group, select the calculator icon to drop down a list of choices
3. Click on the **Calculate** icon



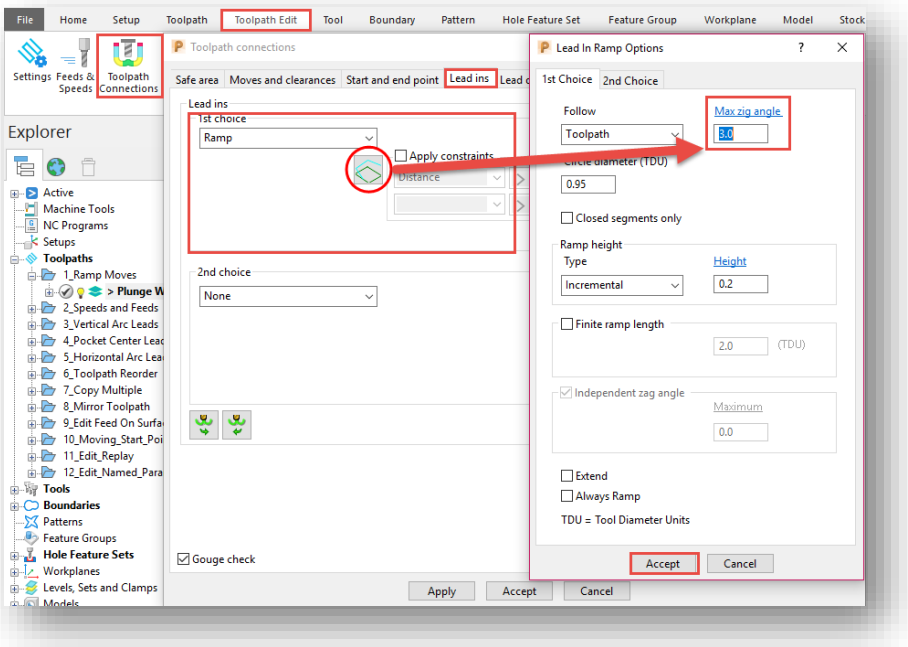
After the toolpath calculates, the following warning will display.




Reading the warning, PowerMill is letting the user know that there are areas within the roughing toolpath that are potentially plunging into stock. Depending on the material that is being machined, the toolpath should be edited to adjust to these movements. Press **OK** to proceed.

Since PowerMill's cutting segments and non-cutting segments are independent from each other, we can simply add ramp moves after the calculation.

1. Click on the **Toolpath Edit** tab
2. Click on the **Toolpath Connections** icon to open the Toolpath connections form
3. Go to the **Lead ins** tab
4. In the **1st choice** pull down menu, select the **ramp** option
5. Now click on the **Ramp options icon**
6. In the **Max zig angle** field, place a value of 3
7. Press the **Accept** button
8. Press the **Apply** button, followed by **Accept**




The non-cutting segments update quick to allow the user to move onto programming their part instead of waiting for the entire toolpath to recalculate itself.

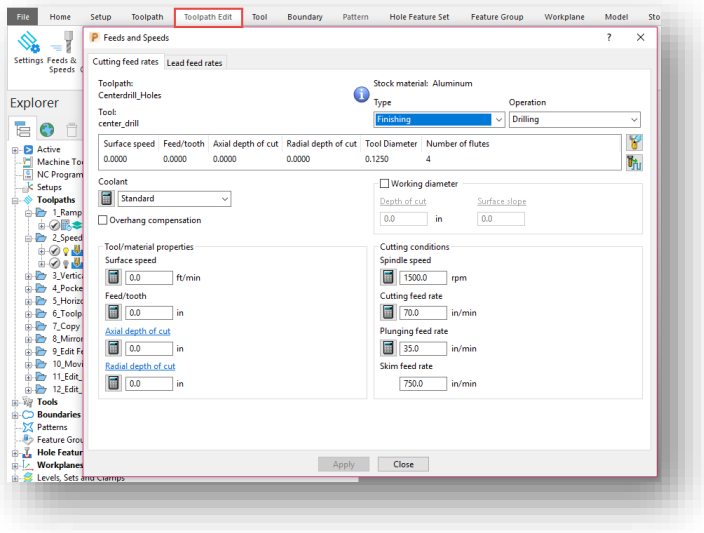
# Speeds and Feeds

Simply changing the speeds and feeds of a particular toolpath can be a daunting task for some CAM systems. With PowerMill, this is relatively simple process.

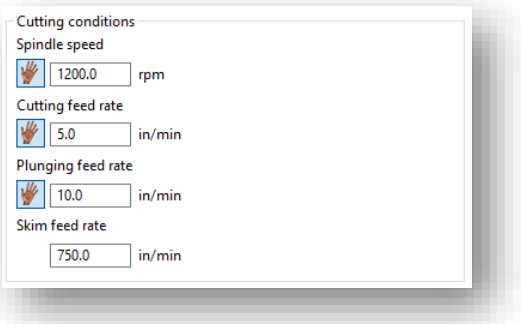
In the folder **2_Speeds and Feeds** there are (2) toolpaths that have the default speeds and feeds values of 1500 RPM and 70 IPM.  Since these are drilling toolpaths, these feeds could potentially cause damage to the tool.

To fix this:

1. Active toolpath **Centerdrill_Holes**
2. Go to the **Toolpath Edit** tab
3. Click on the **Feeds & Speeds** icon to open up the Feeds and Speeds form




4. In the **Spindle speed** field, enter **1200**
5. In the **Cutting feed rate** field, enter **5**
6. In the **Plunging feed rate** field, enter **10**
7. Press **Apply**



Again, this was a very simple process to change the speeds and feeds for the active toolpath without having to recalculate the toolpath.

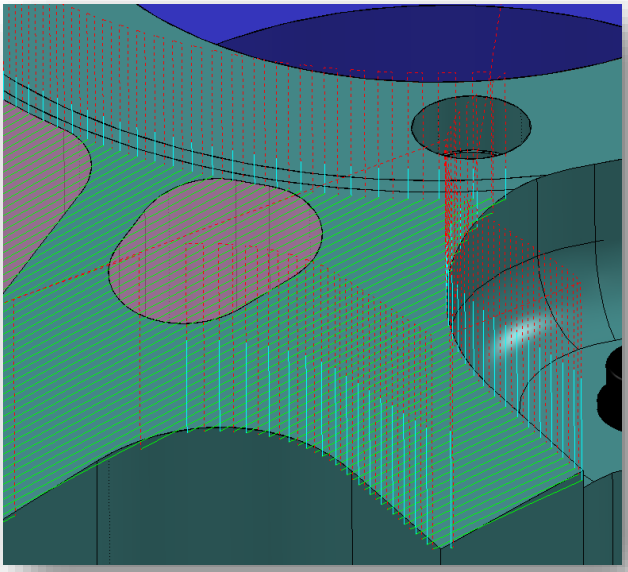
There is another toolpath in the same folder.  Activate **Peckdrill_Holes** and enter in the following values:

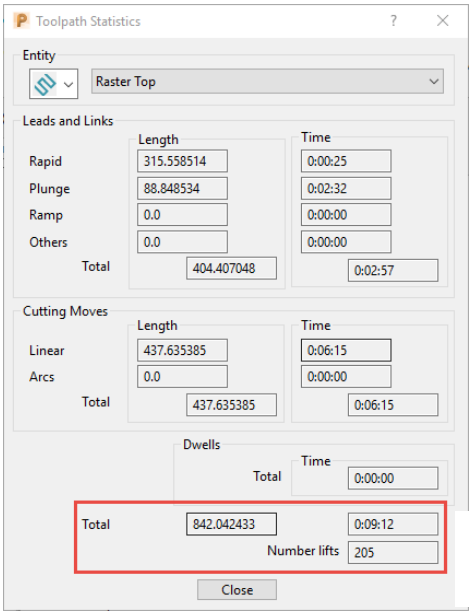
- 650 RPM
- 4 IPM cutting
- 20 IPM plunge
- 1200 IPM skim

## Leads & Links

Entry moves and exit moves are easily driven and modified after the calculation of the toolpath. Allowing the ability to make these changes freely can save lots of time since there is no waiting for calculation times. This also allows drastic time savings because of improved efficiencies.

Look at the image below, this is an image of **Raster Top** that is located in folder **3_Vertical Arc Leads**






If we activate the toolpath in the folder and then right click to show the menu, we can check the estimated run time of this toolpath if we select the **Statistics** option.

The dialog shows towards the bottom that there are 205 lift moves and the run time is just over 9 minutes

Let's go back into the **Toolpath Connections** in the **Toolpath Edit** tab.

1. Go to the **Lead ins** tab
2. In the **1st choice** option, select **Vertical arc** and enter in the following values:

**Angle - 90**
**Radius - 0.125**

3. Press **Apply**

Notice that the lead in moves (orange segments) updated instantly. Now place the same values in the **leads outs** tab.

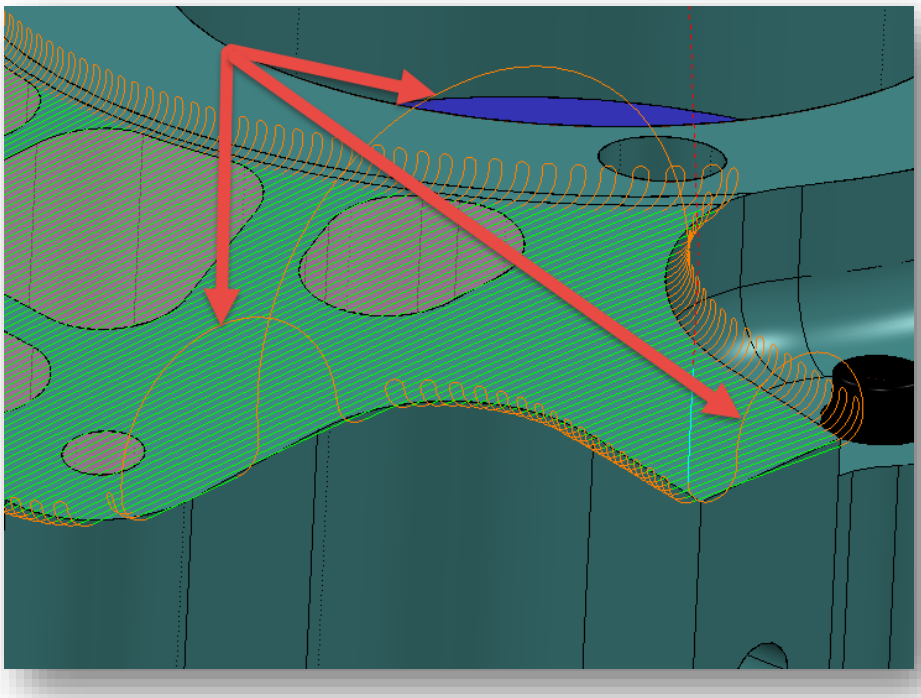
The movement into each segment and the movement out of each segment looks much smoother. However, we are still left with retract moves that are unnecessary.

Go into **Toolpath Connections** and go to the **Links** page. Change the **1st choice** option to **Skim** and uncheck the **Apply constraints** checkbox. Then press the **Apply** button.

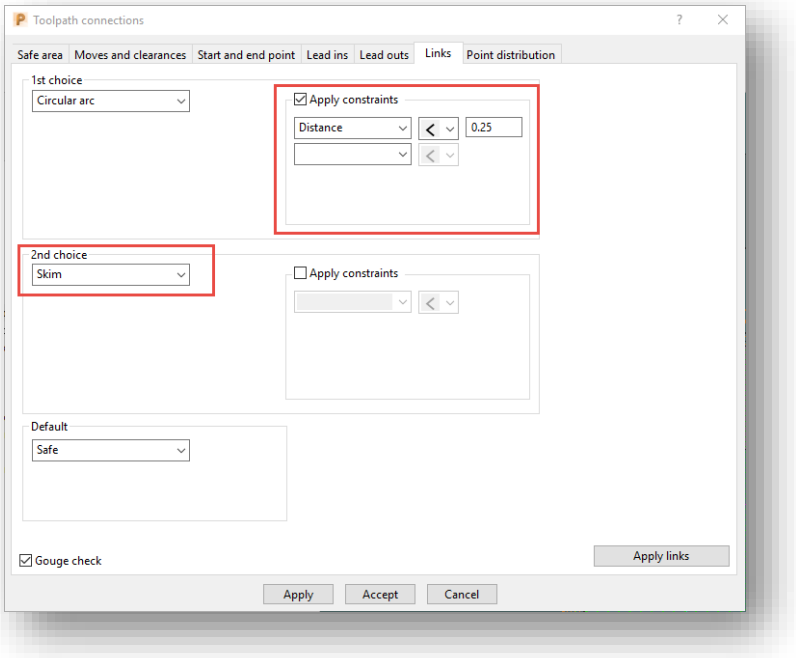
Look how the retract moves have changed.

Still not very efficient. Change the 1st choice option to **On surface**. Try using **Circular arc**. Observe the differences.

Notice the large arc moves.

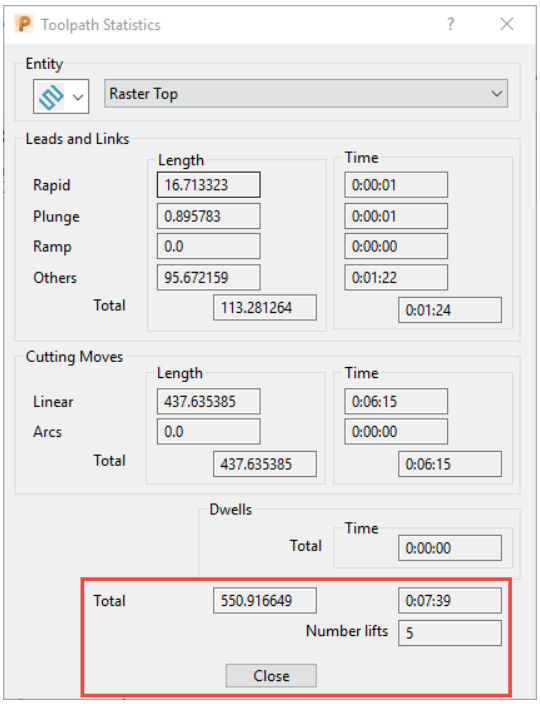


While these arc moves may not pose any issues on the machine tool.  Let fix this to utilize the rapid travel feedrates.



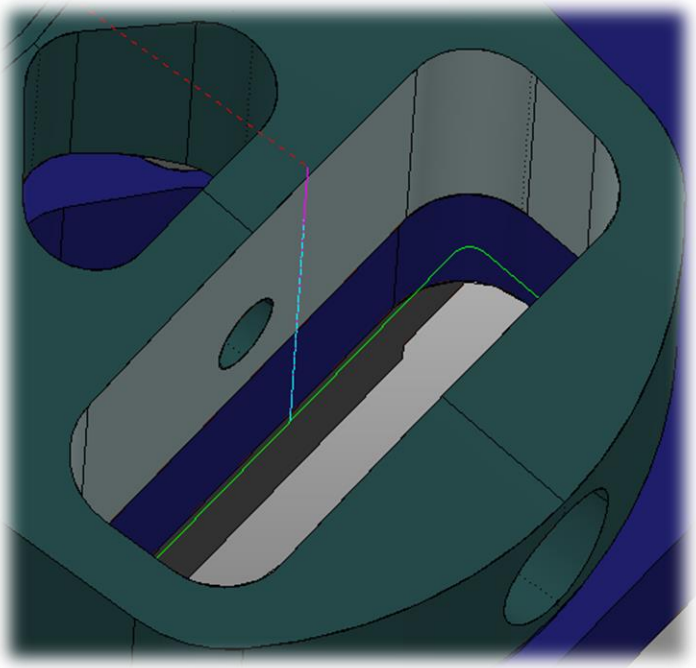

1. Go back into **Toolpath Connections**
2. Go to the **Links** tab
3. Check the **Apply constraints** back on
4. Leave **Distance** as the default option
5. Chance the value to **0.25**
6. Chance the **2nd choice** option to **Skim**
7. Press **Apply**

Now look at the statistics on the toolpath (Right click on the toolpath, select Statistics)




Notice the number of lifts and the runtime.

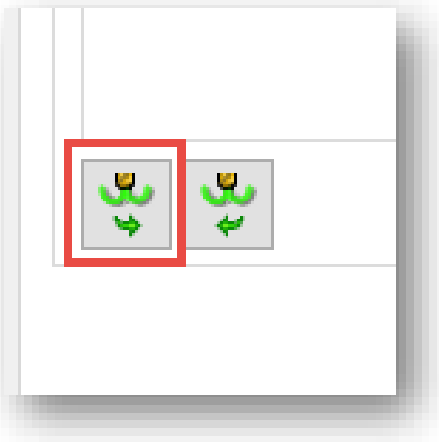
Let's look at a different toolpath.  Active toolpath **Swarf Pocket** in folder **4_Pocket Center Leads**



Notice that the toolpath starts on the wall of this vertical pocket.

Activate the toolpath and go to **Toolpath Connections**

1. Go to the **Lead ins** tab
2. In 1st choice, select **pocket centre**
3. Press **Apply**
4. Make the lead out the same.  Instead of going to the lead outs page, press **the Make lead outs the same as lead ins button**

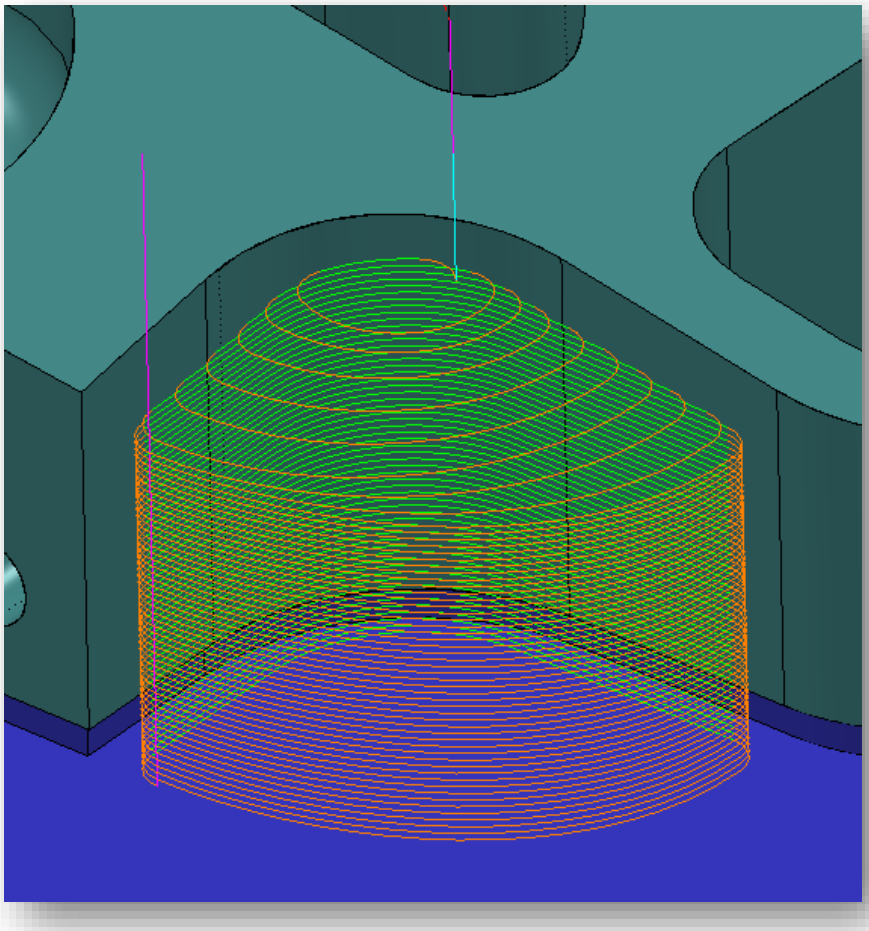


The lead out page will adopt all the settings that were placed into the lead ins page.  This saves time manually placing this information in if you want them to be the same.

Open up folder **5_Horizontal Arc Leads** and activate toolpath **Swarf Corner No Leads**

1. Apply horizontal arc moves to the leads ins and lead outs
2. Apply appropriate link moves

Try to mimic the image below



# Reorder

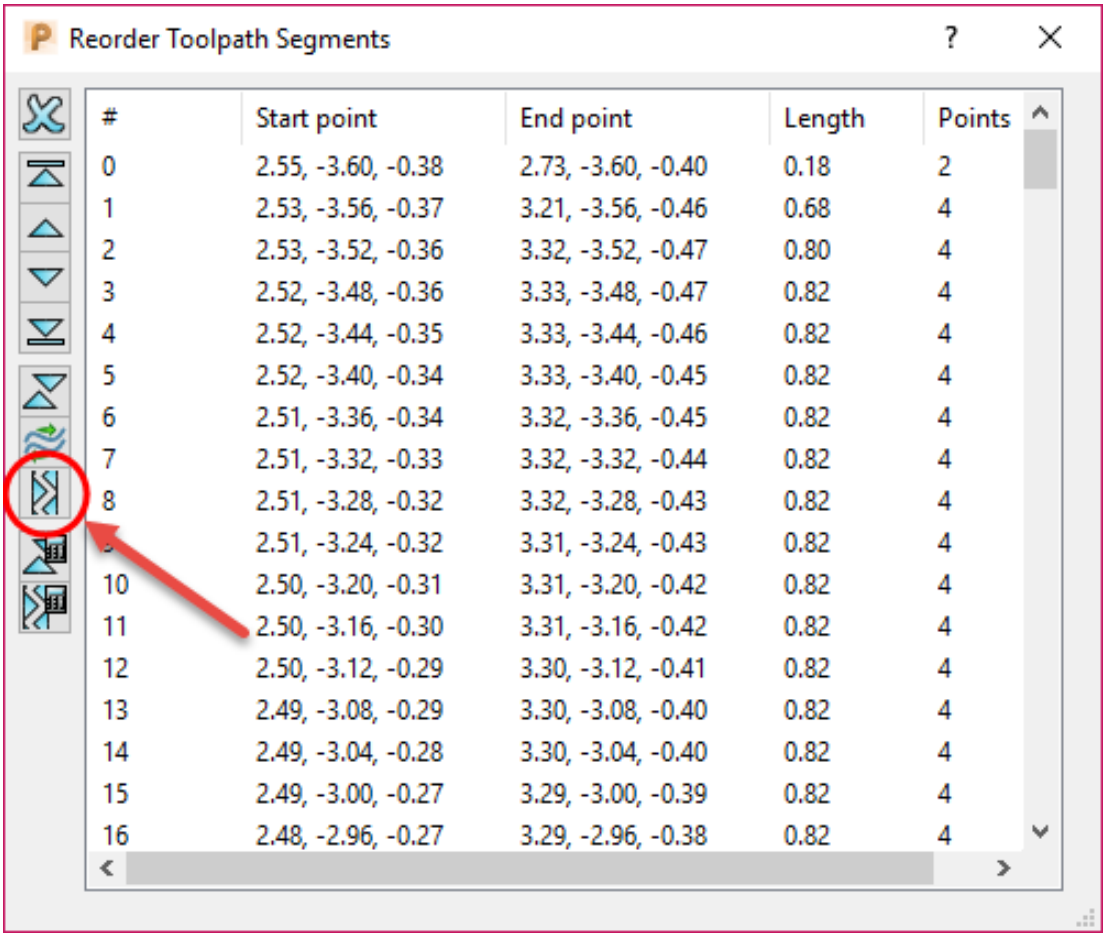
The reorder function allows the user to take selected segments or all segments associated with the toolpath and change the placement as to where these segments occur or change the segment direction.

Activate toolpath **Reorder-Alternate Directions** that is in folder **6_Toolpath Reorder**

1. Add vertical arc lead in and lead outs of 90 degrees and 0.125 radius
2. Change the links to circular links with a constraint value of 0.25

Notice that the toolpath is still very inefficient. This is because the program did not have the direction set to both (climb and conventional directions) before calculating the toolpath.

This can be modified by going to the **Toolpath Edit** tab and selecting the **Reorder** icon. This will open up a form with all of the cutting segments listed. Select the alternate directions icon on the left hand side of the box.



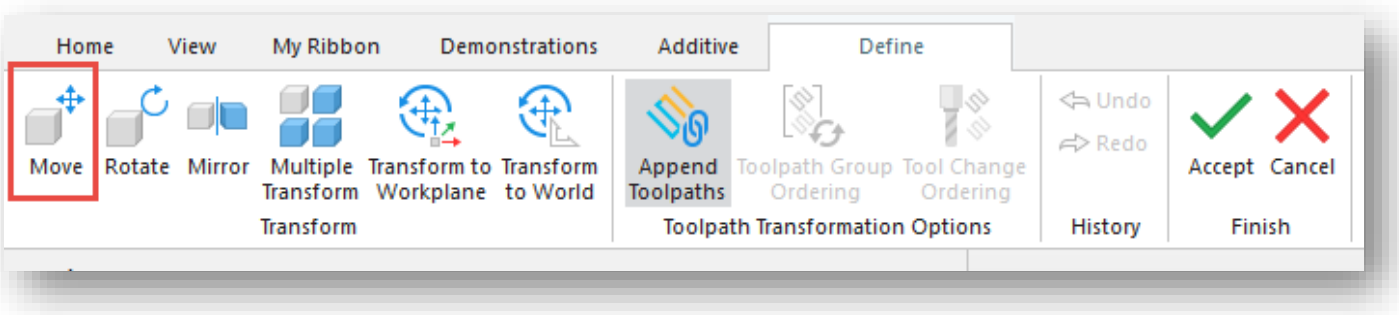

Without recalculating the toolpath, we can changed the cut direction from climb only, to both directions.

# Toolpath Transformations

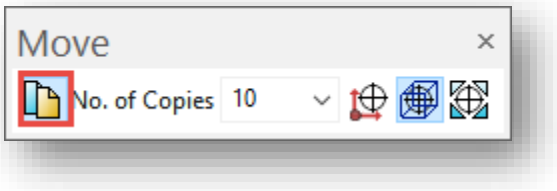
Transforming options like copying, moving, rotating and mirroring can all be applied to toolpaths.

Activate the toolpath **Multiple Passes** that is inside the folder **Copy Multiple**. Then go to the **Toolpath Edit** tab and select the **Transform** icon.

Select the **Move** icon



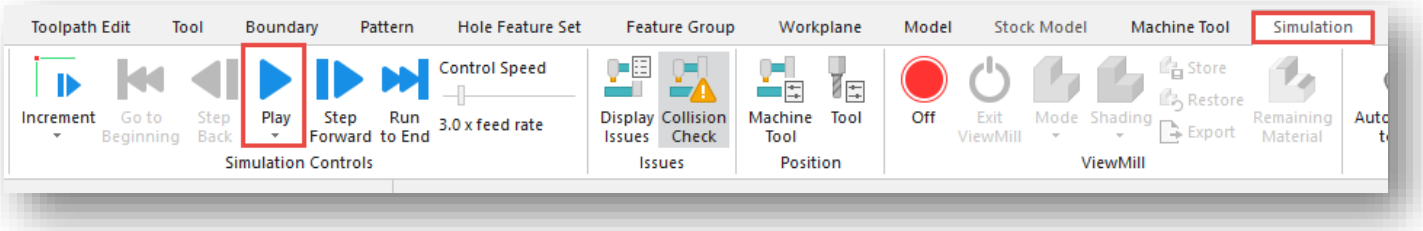

Then, in the floating toolbar, select the copy option and place a value of 10 in the field.



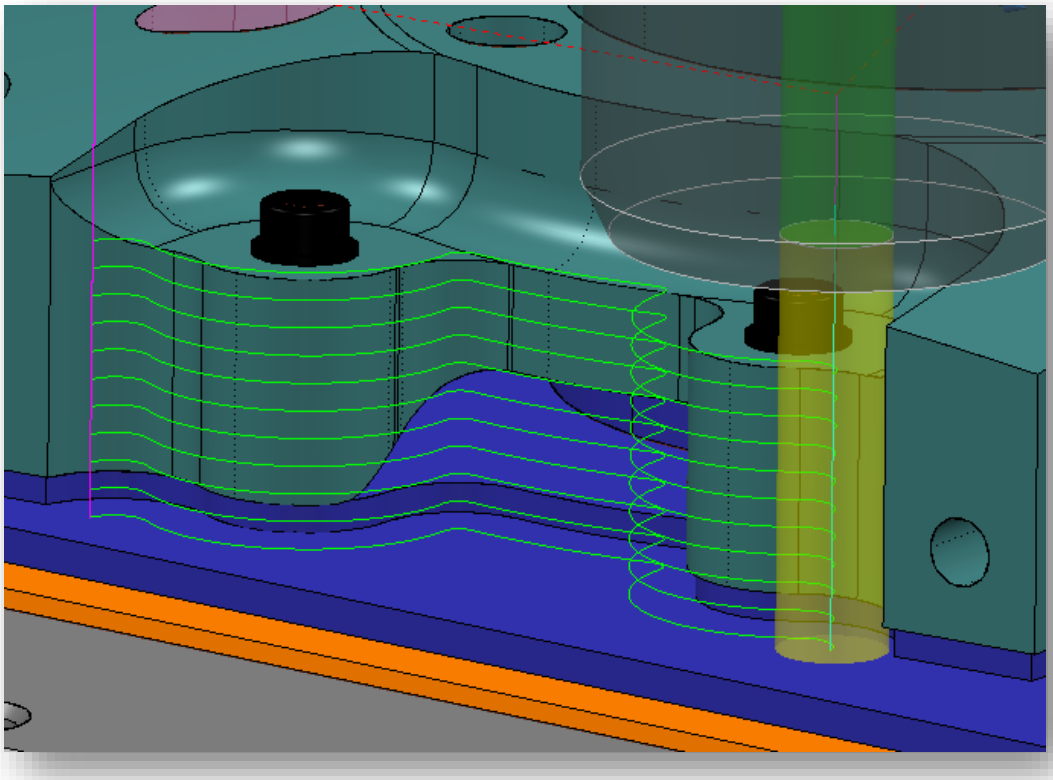

At the bottom of the interface in the green field (in the status bar), place a value of 0 0 0.125 and hit enter.

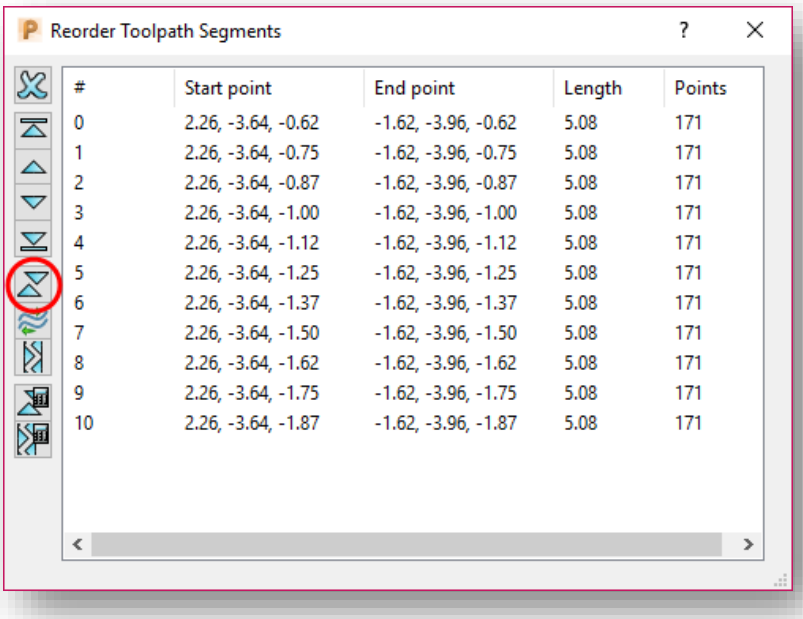
Close the toolbar and press the green check mark on the ribbon to accept the change. PowerMill will keep the original toolpath and create a copy **Multiple Passes_1**

Activate this toolpath and right click on it. Select the very first item **Simulate from Start.** The Simulation tab will activate automatically. Press the Play button.



Notice how the toolpath, starts and the bottom and cuts upwards.





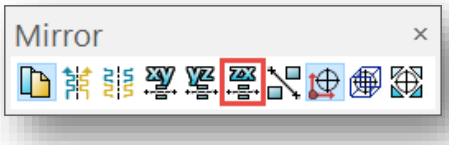

Let's fix this:

1. Go to the **Toolpath Edit** tab
2. Select the **Reorder** icon
3. Click on the **Reverse order** icon on the left side
4. Rerun the simulation

Let's try a different type of transformation.  Open folder **8_Mirror Toolpath** and activate toolpath **Multiple Passes Copied.**

This part is symmetrical about the X axis.  We will mirror this toolpath to the other side of the part to save some time.

Go to the **Toolpath Edit** tab and select the **Transform** icon.  Select the **Mirror** icon.  When the floating toolbar pops up, select the **ZX** button.  You see the preview of the new toolpath in purple.




Run the simulation.

# Editing Feedrate on Selected Surfaces

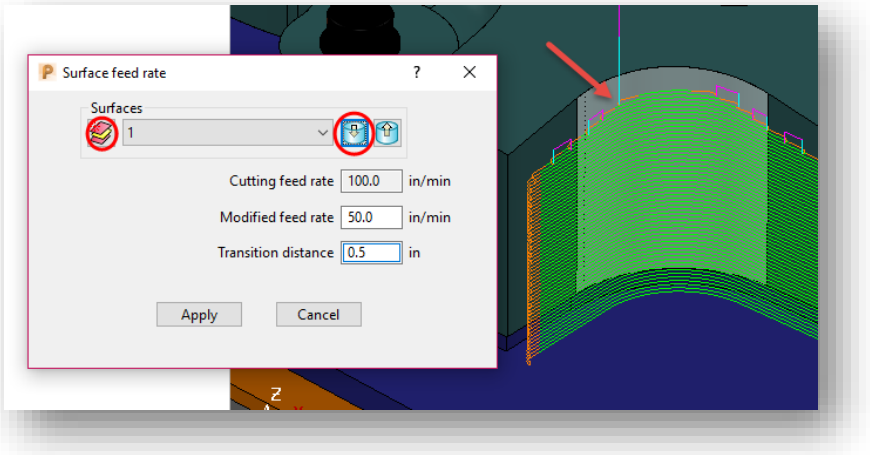
In some cases, we may want to control how the tool cuts at certain locations in the toolpath. We can select certain surfaces and change the feedrates independently from the rest of the programmed toolpath.

Open folder **9_Edit Feed On Surfaces** and activate toolpath **Swarf Corner With Leads**

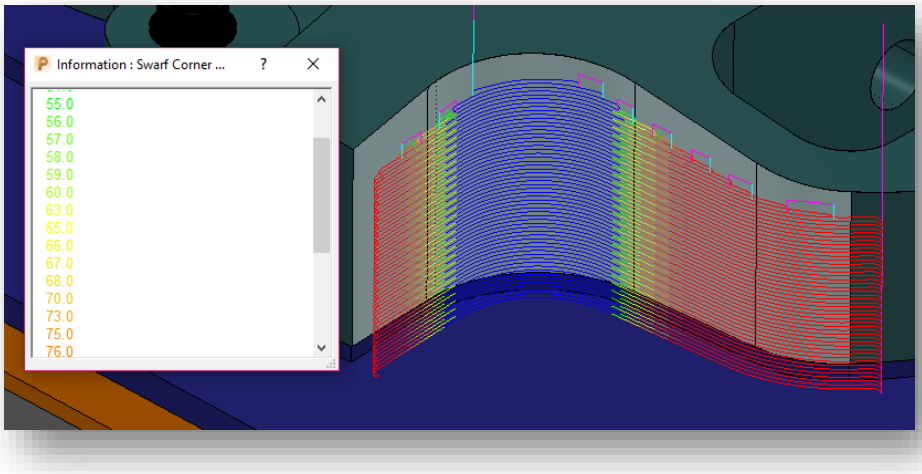
Right click on the toolpath in the explorer and search for the sub menu Edit. Another menu will fly out. Select the option **Update Feed Rate On Chosen Surfaces…**

A pop-up dialog box will appear. In the graphics window, select the large interior fillet where the toolpath is cutting and then press the icon on the dialog box to create a set (left). Then select the acquire icon (right) to add this surface to the set. Leave the cutting parameters the same.

Press **Apply** and **Cancel**




To view the change. Go to the **Toolpath tab** and select the **Feeds** within the Draw group.



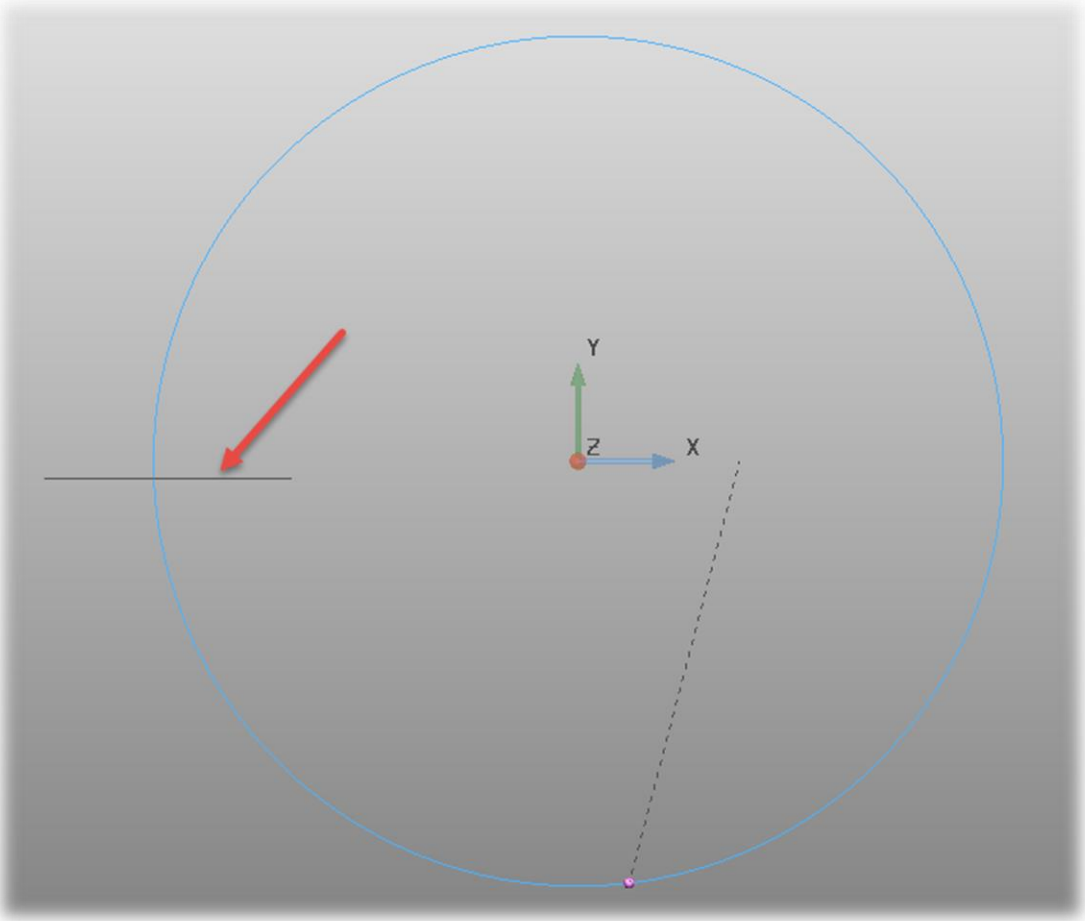
## Moving start Points

PowerMill allows the movement of start and end points on a closed toolpath.

Open folder **10_Moving_Start_Points** and activate toolpath **Moving Start Points.**

Go the **Toolpath Edit** tab in the ribbon and apply some appropriate leads and links to avoid the toolpath lifting and plunging on the wall of the circular pocket. Once finished, select the **Move Start Points** icon.

The ribbon interface will alter to give only the options available in this editing tool, there are two options for moving start points. The default option is to move all the start points at one time. Draw a line to where the new start points should take place.

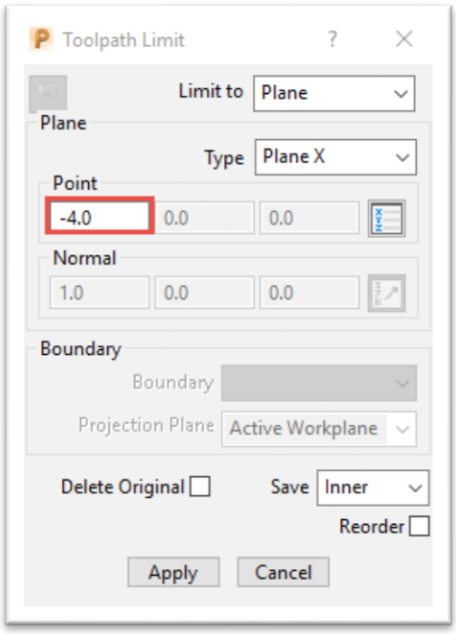


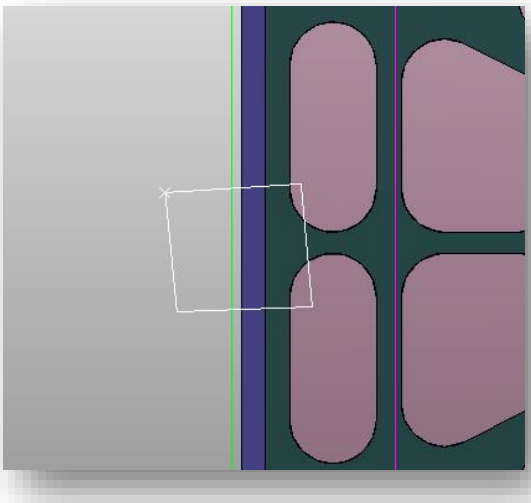

Try using the Single option. Drag the control points individually. Notice how the leads and links adjust automatically.

# Edit History Replay

Some applied edits will replay themselves if the toolpath needs to be recalculated for items that cannot be edited outside the calculation (ie, thickness, tolerance, stepovers……)

Open folder **11_Edit Reply** in the Explorer and activate toolpath **Toolpath_Limiting.** Let's make some changes to the toolpath.

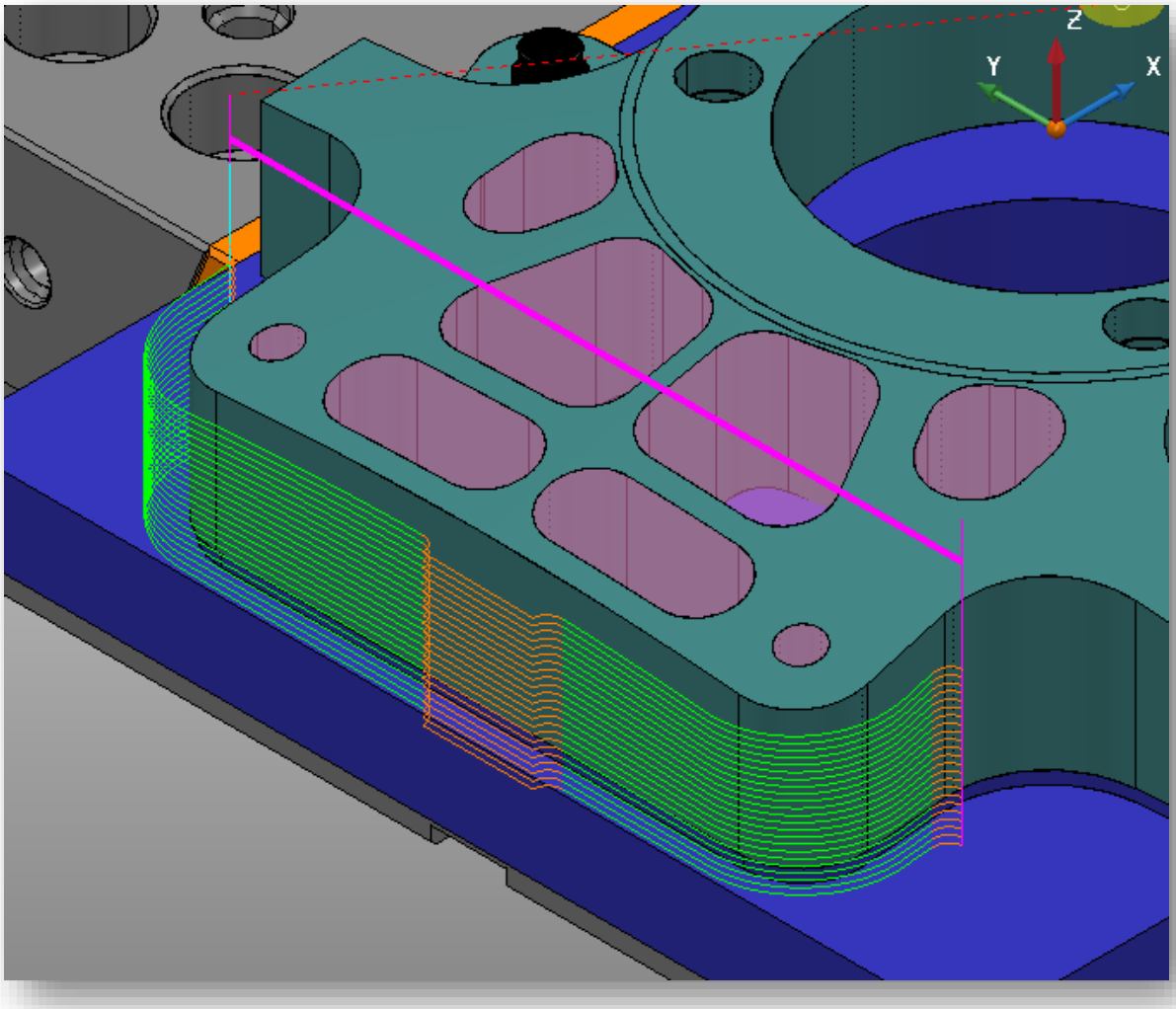


1. Go to the **Toolpath Edit**, and select the **Reorder** option
2. Click the **Alternate direction** icon to make toolpath cut in a climb cut fashion
3. Click on the **Limit** icon in the **Toolpath Edit** tab
4. Go to a top view
5. Leave the Limit to option to **Plane** and Type to **Plane X**
6. Enter **-4** in the open field
7. Press **Apply**
8. Leaving the Toolpath Limit form open, change the **Limit to** option to **Polygon**
9. Create a freeform polygon in the center of the toolpath
10. Change the **Save** option to **Outer** and press **Apply**
11. Press Cancel

12. Right click on toolpath **Toolpath_Limiting_1_1** and press **Select Surfaces**
13. Right click on the toolpath again and select **settings**
14. Press the **recycle** icon to open the parameters of the toolpath
15. Change the **Thickness** to **0.02** and hit **Calculate**

Notice that the toolpath re-generated itself to the new thickness value that was applied and then it replayed the edits back to it that we just applied earlier.



# Edit Named Parameters

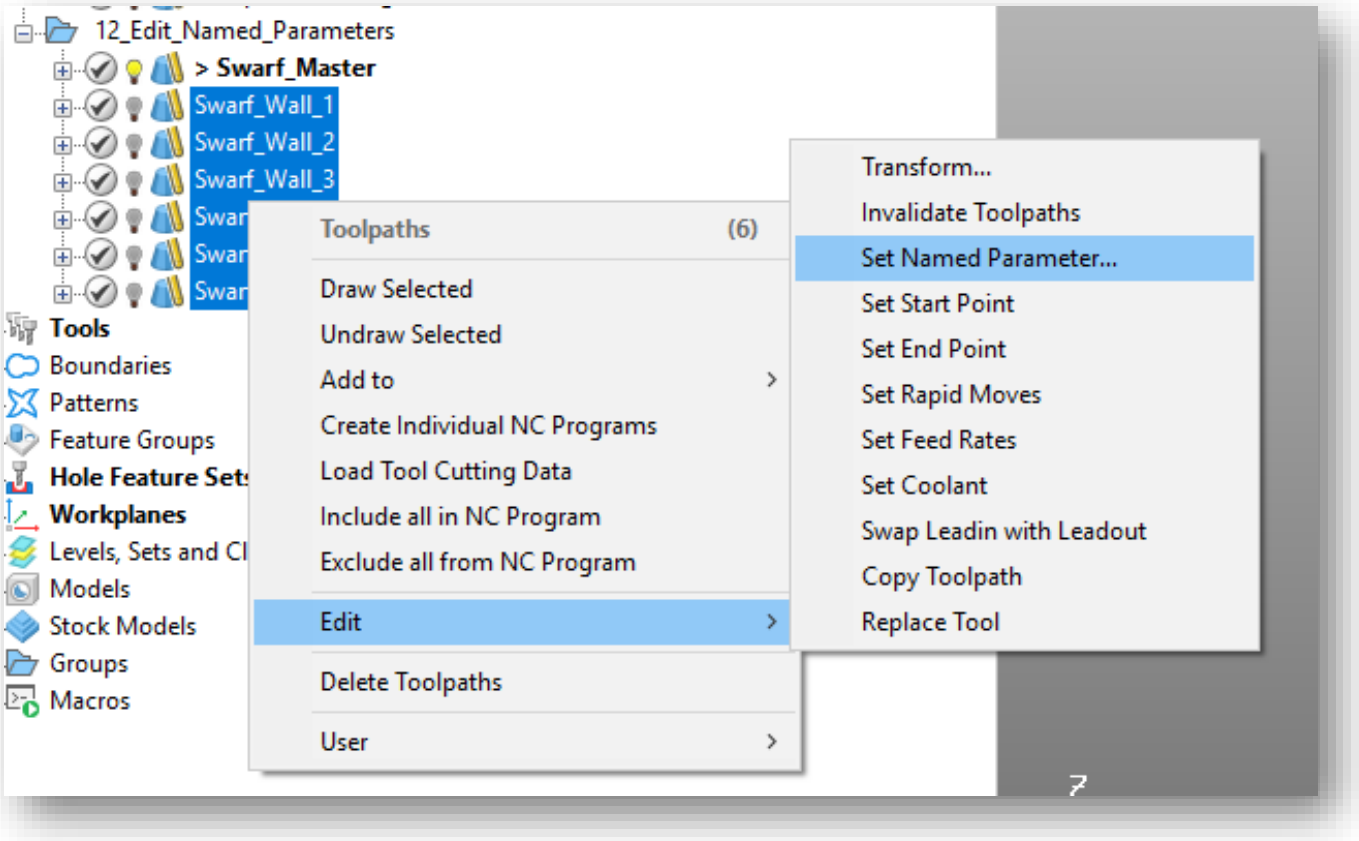
There are times where you may want to change a certain parameter in multiple toolpaths.

Go to folder **12_Edit_Named_Parameter.**

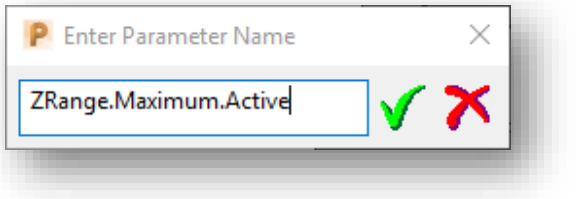
There are seven toolpaths in this folder. The toolpaths labelled **Swarf_Wall _1** thru **Swarf_Wall_6** all have a Z maximum height value of **-1.5**. This value was a mistaken left in during the calculation.

Toolpath **Swarf_Master** has the appropriate value of 0, which allows the toolpath to machine from the top of the walls down to the bottom. We want to apply this value to the others in this folder.

1. Activate toolpath **Swarf_Master**
2. Now, select toolpath **Swarf_Wall_1** thru **Swarf_Wall_6** by holding down the CTRL button when selecting them.
3. Once selected, right click on any one of the toolpaths and go to **Edit**, then **Set Named Parameter**



4.In the Enter Parameter Name field, type in **ZRange.Maximum.Active** and press the green checkmark.




5.These toolpaths will be set back to an invalidated state (uncalculated)
6. Right click on the **Toolpaths** branch
7. Click on **Batch Process**

All of the toolpaths in the **Explorer** re-calculate themselves

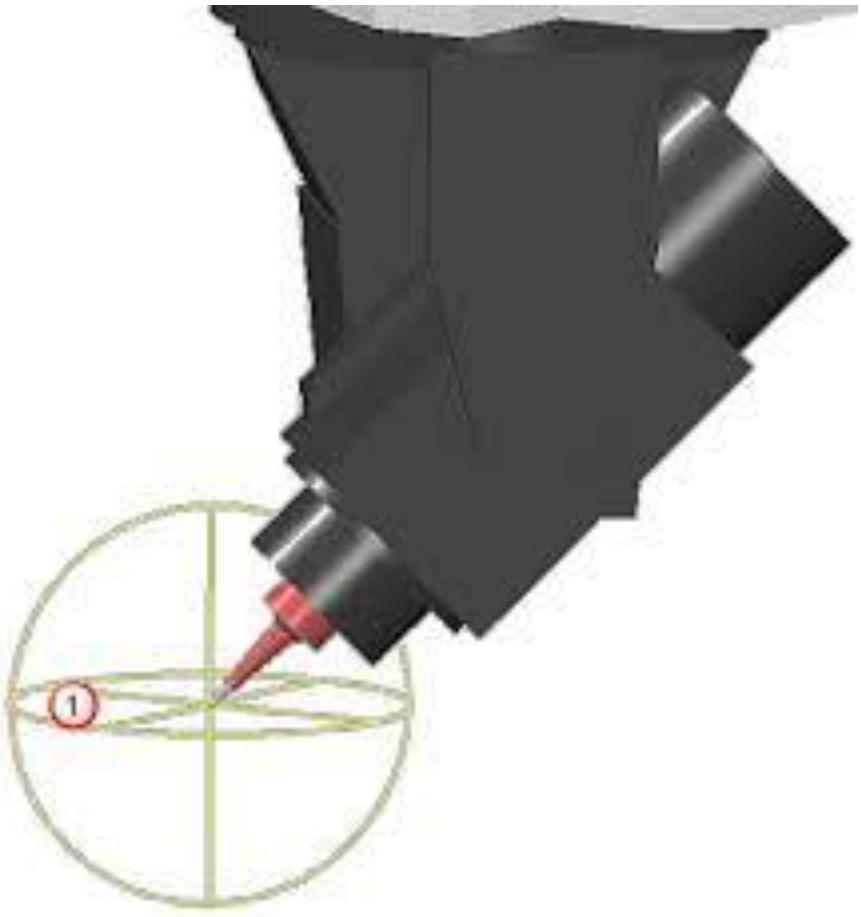
Notice now that each one of those toolpaths now cuts from the top of the wall down to the bottom of the wall like the Swarf_Master does.

This concludes the portion of this first exercise.

# Dynamic Machine Control

When programming parts with 4th and 5th axes movements, a large benefit in programming is being able to use a simulation file to check the behavior of the toolpath in the machining environment before outputting code and finding faults before it is too late.

Another large benefit is being able to use the machine simulation file (MTD) to correct these faults either before or after the toolpath has been calculated.



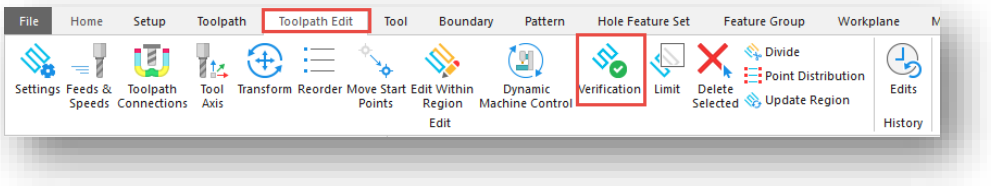
# Collision Checking

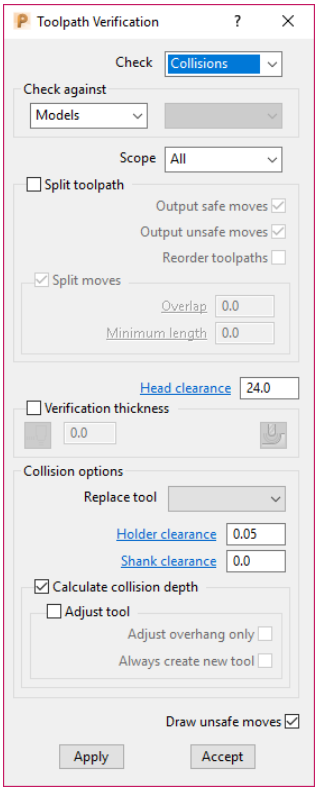
During the toolpath calculation, generally PowerMill gouge checks against the model with the cutting portion of the tool assembly. After the calculation, a colored status light is displayed. These status lights give information about the safety of the toolpath. A white check mark, means that the toolpath is gouge free.

It is up to the user to take this safety to the next level. They have the option to check for collisions against the tool assembly after the toolpath calculation or during the toolpath calculation.

Go up to the **Browser** and select on the image in the **Dynamic Machine Control** section. Expand the **Toolpaths** branch and expand the **Collision Checking** folder & activate toolpath **Finish Locks.**

The toolpath indicates that is has been calculated as gouge free. Let's make sure that the whole tool assembly is safe before it gets sent out to the machine.




1. Go to the **Toolpath Edit** tab and select the **Verification**
2. Make sure the **Toolpath Verification** form looks like the image to the left and hit **Apply**
3. The simulation should come back with a message that states, "No collisions were found"
4. Press **OK**
5. Press **Accept** to close the form

Notice that the status light has changed from white to blue. This indicates that the entire tool assembly is safe to within 0.05 to the model.

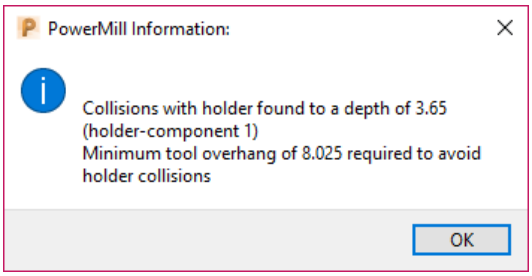
# Colliding Segments

Toolpath segments can be selected and deleted to suit the program needs.

Open folder **Colliding Segments** and activate toolpath **Finish_Pocket_Delete**

Go to the **Toolpath edit** tab and select the **Verification** icon. Leave the parameters and press **Apply.**

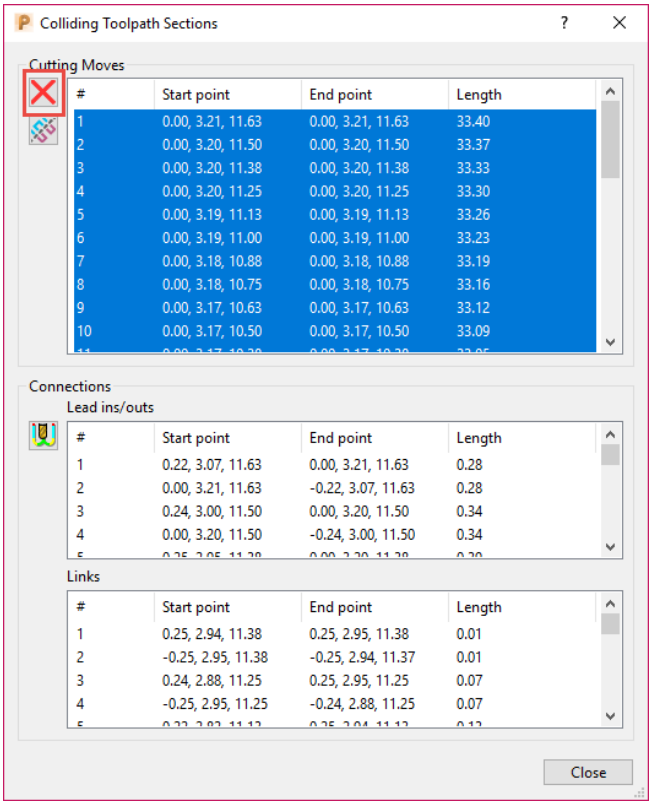
The following warning is issued…..




Press **OK**

Notice that the segments that are unsafe are highlighted in red.

Un-shade the model by pressing the **F3** key or pressing the  icon on View toolbar




1. Go to the **Home** tab and select the **Colliding Sections** icon in the **Verification** group.

2. Once the dialog box opens, select the first line (1), hold the shift button and select the last line (30). Then press the red X in the top left corner.

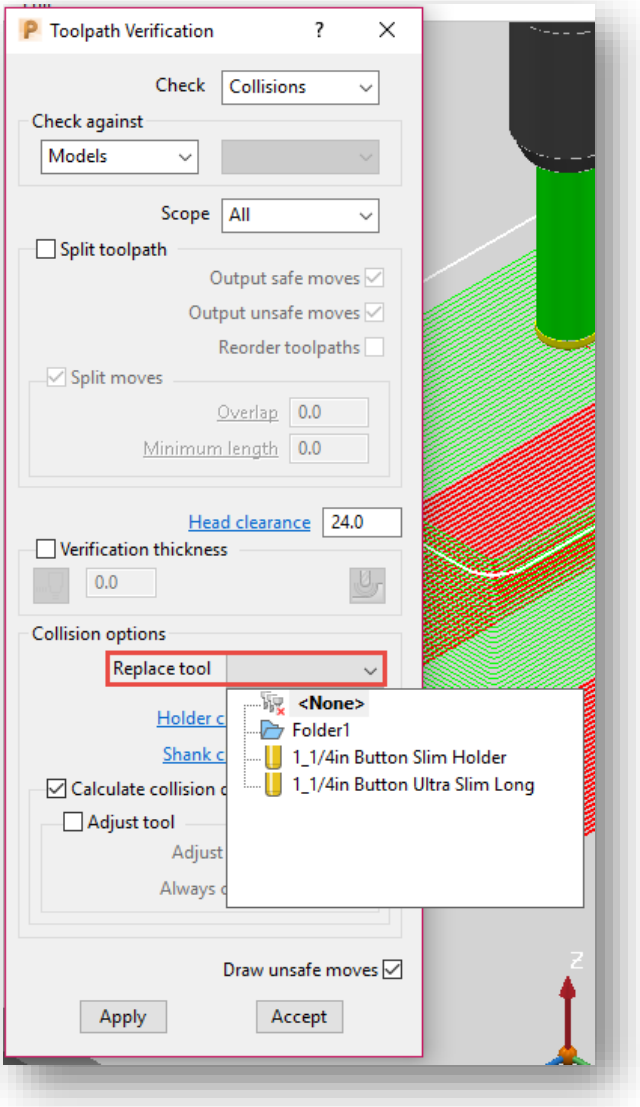
3. Selecting the X will delete the colliding segments of the toolpath and set the safety status to blue.

**Alternate method:** Re-run the verification on the other toolpath in this folder. Instead of selecting the X, select the icon right below. This will split the toolpath in a safe toolpath and an unsafe toolpath.

# Replace Tool

Open folder **Replace Tool** and activate toolpath **Finish_Pocket_Replace**

Run the same verification as we did in the last two examples. Once the PowerMill box warns the user about the collision, press **OK.**




1. Go to the Replace tool menu.

2. Notice that there are (2) tools that can be selected. Select tool **1 1/4in Button Slim Holder**.

3. Press **Apply**

PowerMill will now replace the original tool with this new tool and update the collision status. Since there are still collisions involved, repeat the process, but select tool **1 1/4in Button Ultra Slim Long.**

4. Press **Apply**

PowerMill should now return a message that there were no collisions found and the status light will change to blue.

5. Press **OK**

6. Press **Accept** to close out of the verification page.

# DMC – Swap Configuration

This editing tool can be used to easily modify a 3 axis, 3+1 axis or 3+2 axis toolpath to avoid potential near collisions or actual collisions using a replica of the machine tool (MTD file).

Open folder **DMC – Fix** and activate toolpath **Raster_Flats_Safe.** Run the verification with the same parameters. PowerMill indicates a collision at a depth of **0.12, minimum tool over hang is 3.12**

Open the **Tools** branch in the **Explorer.** The tool that is associated with this toolpath is already active.

Right click on **1in Button HeatShrink Long** and select Settings. Go to the holder tab and adjust the overhang to **3.125** and press **Close**

Notice the status light on that toolpath changed from red to blue, indicating that it is now a safe toolpath.

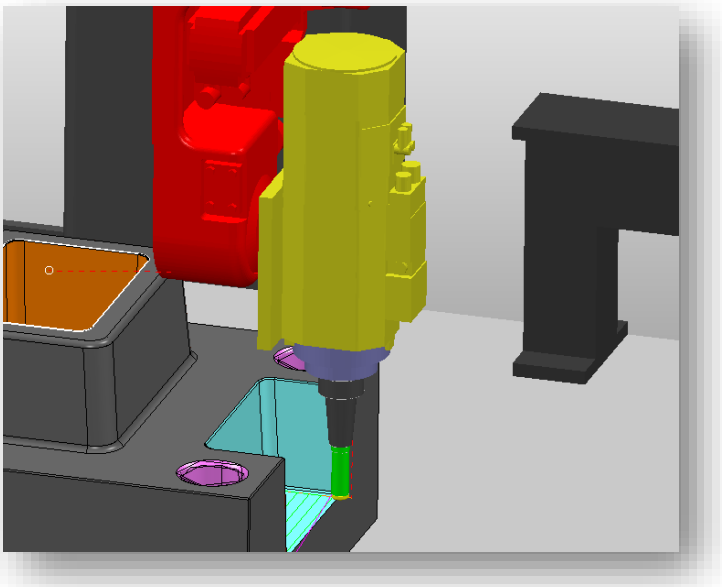
Normally we would be satisfied with sending this off to the machine, however since we have a machine simulation file (MTD), we can run a full machine simulation to verify this.

Right click on the toolpath and select **Simulate from Start**

The machine file will attach itself to the tool holder and the **Simulation** tab will open. Press the play button.

The simulation plays through without any issues.

Activate toolpath **Raster_Flats_Unsafe** and run the verification. This time there were no collisions because we already fixed the overhang on the previous toolpath.

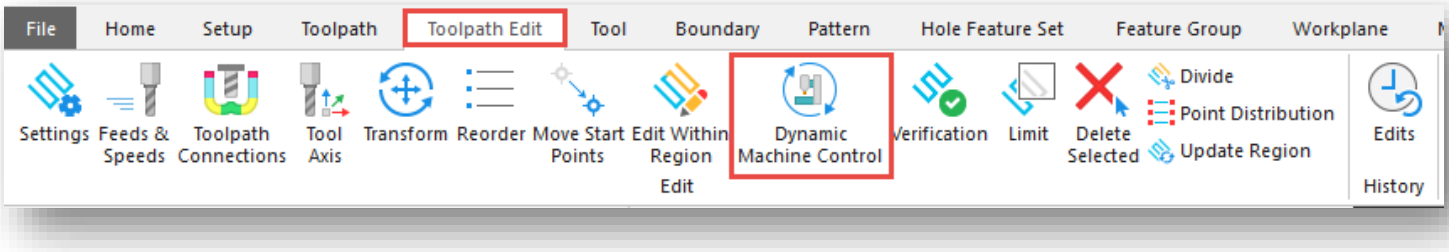


Run the machine simulation.

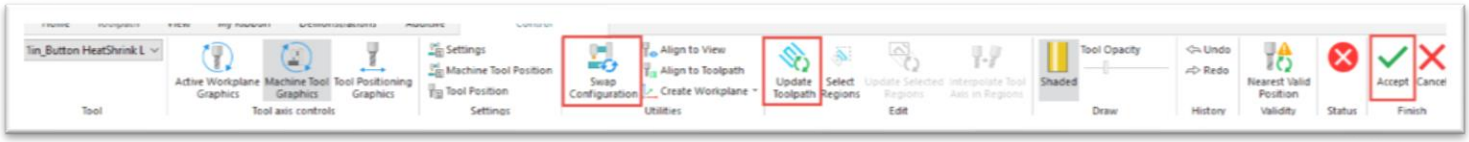
The simulation found a collision with the machine tool and stops to show the user.

This toolpath will need to be modified

1. Go to the **Toolpath Edit** tab and select the **Dynamic Machine Control** icon




2. Press the **Swap Configuration** icon (this flips the machine tool to the alternate solution)
3. Press the **Update Toolpath** icon to accept the change
4. Press the green checkmark to finish editing the toolpath



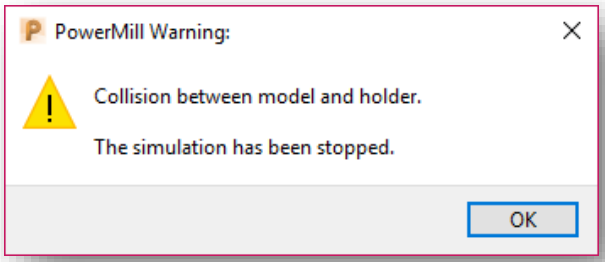

5. Rerun the simulation.

Everything looks good.

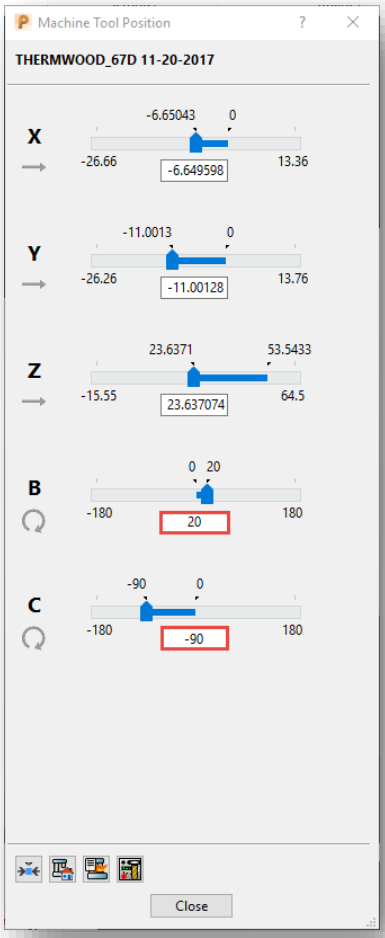
**Here is a video of this example…. [https://autode.sk/2CL5aM2](https://autode.sk/2CL5aM2)**

# DMC – Change Tool Axis in One Region

Let's look at another example. Activate toolpath **Raster_Wall_One_Solution** and simulate. The simulation should stop once PowerMill detects the collision between the toolholder and the model.




Press **OK** to continue




Let's correct the issue by using the Dynamic Machine Control.

1. Go to the **Toolpath Edit** tab and select the **Dynamic Machine Control** icon.
2. Click on the **Machine Tool Position** option.
3. Enter **20** in the B axis field
4. Enter **-90** in the C axis field
5. Press **Close**
6. Press the **Update Toolpath** icon
7. Press the **green** checkmark
8. Rerun the simulation

**Watch this video of this example [https://autode.sk/2Cj9zoC](https://autode.sk/2Cj9zoC)**

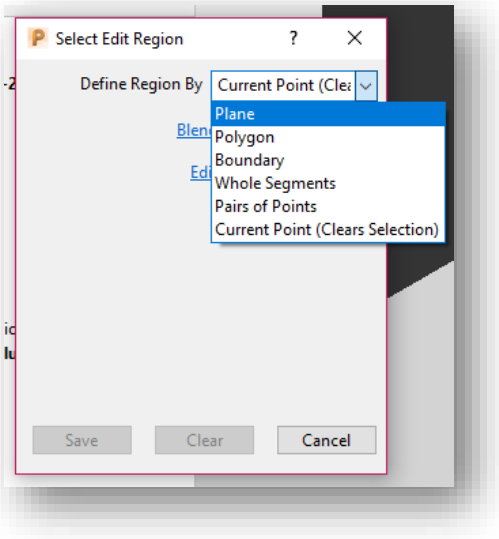
# DMC – Change Tool Axis In Multiple Regions

The Dynamic Machine Control now also allows us to select different regions in the toolpath to make edits.

Activate toolpath **Corner_Multiple_Solutions** in folder **DMC – Fix.** Running a simulation will surely cause a collision to the toolholder. But since this toolpath wraps around both walls, a single solution is not possible.

Go to the **Toolpath Edit** tab and select the **Dynamic Machine Control** icon

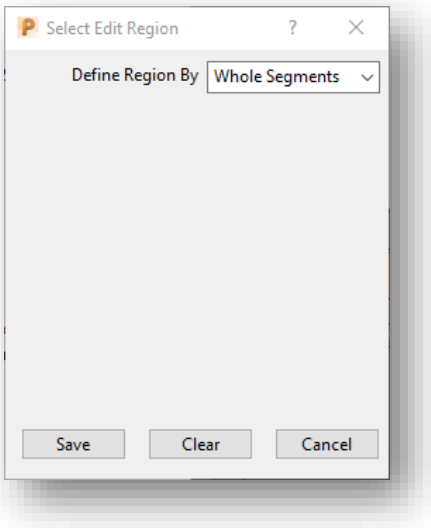
Press the **Select Regions** icon opens the following dialog box, then:




1. In the **Define Region By** menu, select **Plane**
2. Make sure Plane X is enabled
3. Leave the point at 0
4. Press the Save button

The segments that we are editing will turn white

5. Grab the tool and snap it to one of the white segments.
6. Press the **Machine Tool Position** icon on the ribbon and type in the following values. **C -45** & **B -30**
7. Press the Update **Selected Regions icon** to set these values

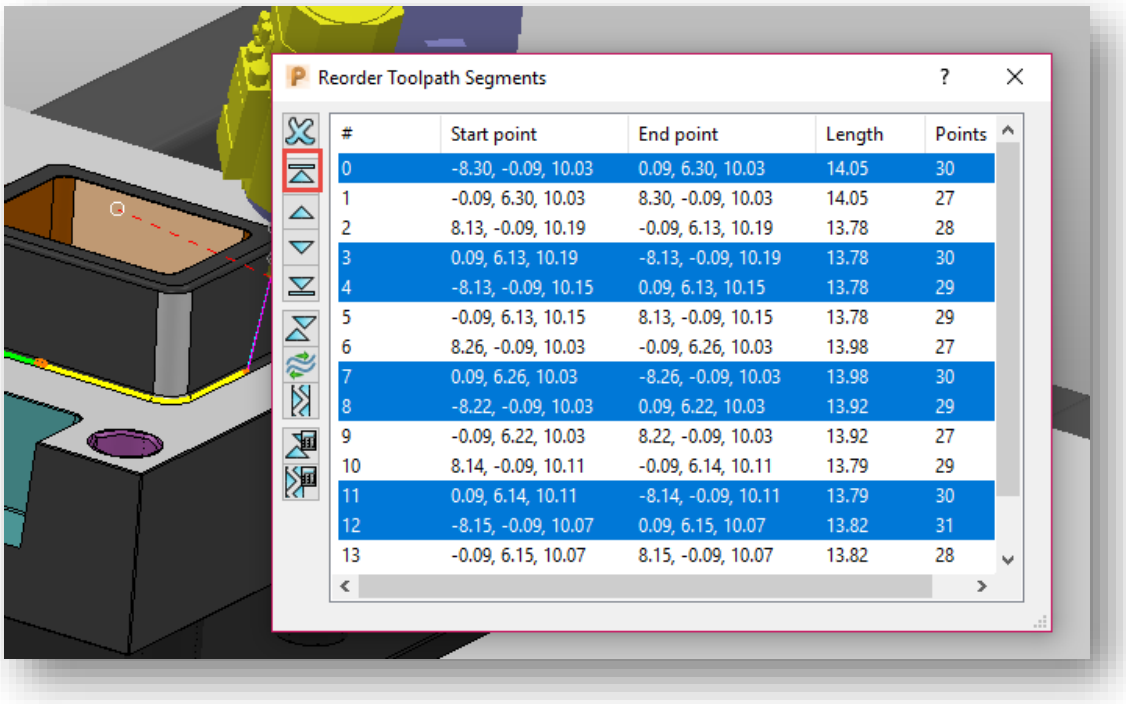



8. In the **Select Edit Region** box, change the **Define Region By** option to **Whole Segments**
9. Box select the other segments that have not been edited yet and Press the Save button
10. Drag the tool to the opposite side and enter the values in the position form **C -135** & **B -30**
11. Press the **Update Selected Regions** icon
12. Close the forms if any are open, then press the green checkmark
13. Simulate the toolpath

You will notice how the machine jumps from one angle to the next angle as it cuts from one segment to the next. This is because the ordering of the toolpath has not been affected.

Let's fix this by going to the **Toolpath Edit** tab and selecting the **Reorder** icon

Manually select the first half of the toolpath by selecting with a box




Press the **Move to start** icon

Close the form

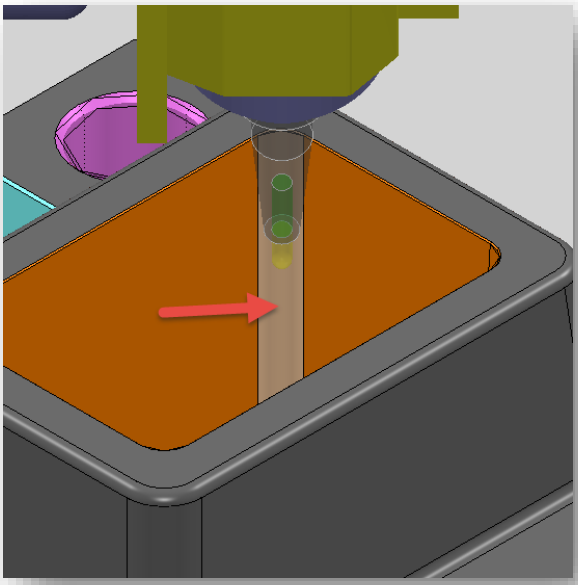
Perform a gouge check and collision check

Simulate the toolpath

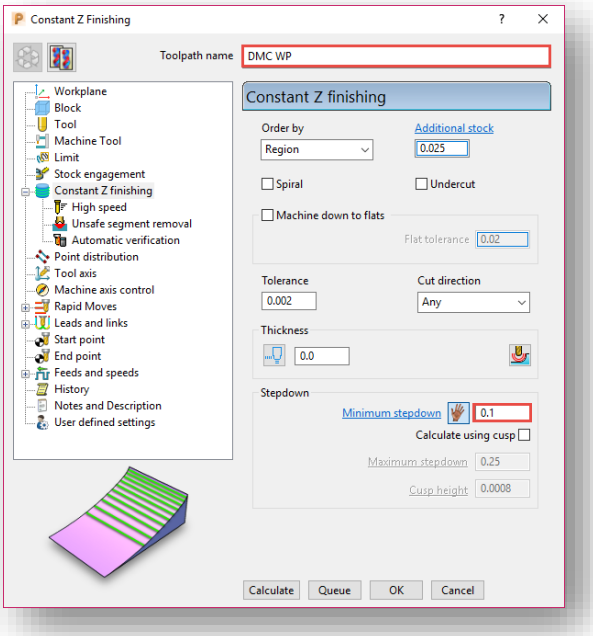
**Here is a video demonstration of the example [https://autode.sk/2O02b3v](https://autode.sk/2O02b3v)**

# DMC – Creating Workplanes

The DMC also allows the user to preplan their next toolpath by creating the tip with the machine tool and then applying it to a new toolpath.



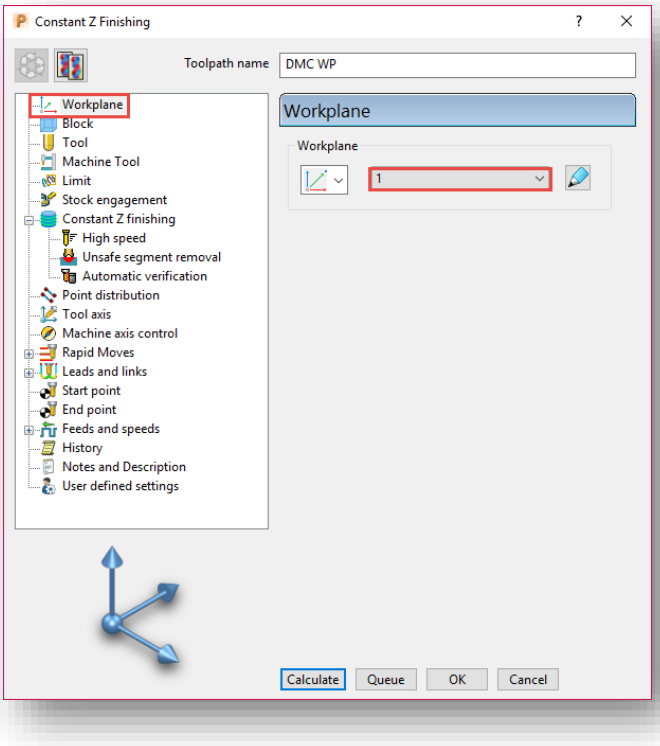
1. Activate tool **1/2in Ball Long**
2. Activate folder **DMC - Create**
3. Go to the **Toolpath Edit** tab and select the **Dynamic Machine Control** icon.
4. Grab the tool and drag it to the following corner in the large orange pocket (lighter surfaces)
5. Select the Machine Tool Position icon
6. Place values of **C 45** & **B 20** (Depending on the corner selected, these values may differ)
7. In the **Utilities** group, select the **Create Workplane** menu and select **at Tool tip**



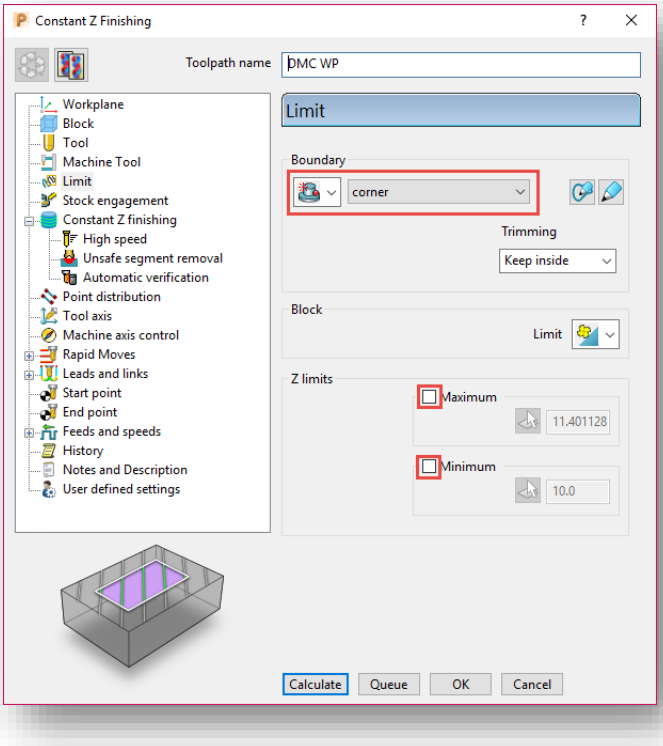

8. Press the green checkmark

A new workplane will be created on the model at the tool location. Now,

9. Go to the **Toolpath** tab and select the **Toolpath** icon in the **Create** group
10. Select **Constant Z Finishing** in the **Finishing** page
11. Change the toolpath name to **DMC WP**
12. Change the Stepdown to **0.1**




13. Go to the **Workplane** page and select workplane **1** from the dropdown menu




14. Then go the **Limit** page

15. Change the boundary to **corner** and uncheck both Maximum and Minimum Z limits

16. Press the **Calculate** button

17. Update the leads and links moves

18. Simulate the toolpath

**Watch the video demonstration here**

**https://autode.sk/2D2zbHh**

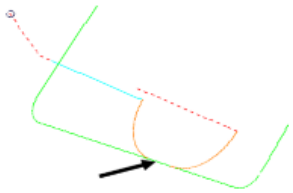
This concludes the portion on the second exercise.

# Editing Within A Region

AUTODESK. POWERMILL

Autodesk® PowerMill® - Toolpath Editing

## Basic & Intermediate Editing

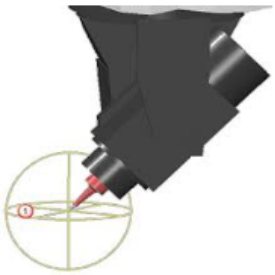


**Click the image to load the project.**

This example will take us through some of PowerMill's basic editing functions as well as some intermediate functions.

Lets take a few minutes to try out all of the examples located in the Explorer folders. If you get stuck or need help, please ask.

## Dynamic Machine Control



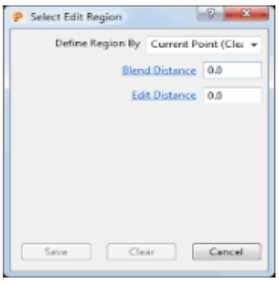
**Click the image to load the project.**

The View Cube can be used to manipulate the view orientation, either dynamically, or by using the fixed views.

Please take a few minutes to try out the view cube functions by clicking on the different control points using the left mouse button.

**When finished, go onto the next example below**

## Edit Within Region




**Click the image to load the project.**

The View Cube can be used to manipulate the view orientation, either dynamically, or by using the fixed views.

Please take a few minutes to try out the view cube functions by clicking on the different control points using the left mouse button.

**When finished, go onto the next example below**

# Tool Axis to Lean

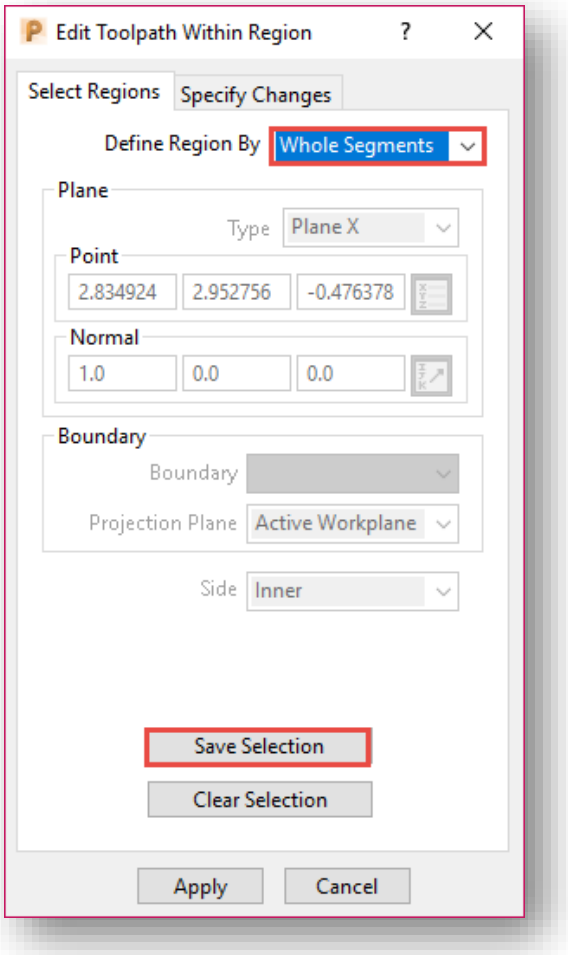
Using the Edit Within Region tool allows the ability to edit the tool axis of selected (partial) segments of a toolpath while preserving the toolpath contact positions.

Go up to the **Browser** and select on the image in the **Edit Within Region** section. Expand the **Toolpaths** branch and expand the **Apply Lean** folder. Activate toolpath **Corner_Finishing_Lean.**

Change the leads and links to appropriate moves then simulate it. The toolpath will stop simulating because of a collision between the holder and the model.

At this point, we could re-calculate the toolpath with an appropriate tool axis movement to utilize some live five capabilities. However, we can take advantage of a powerful editing tool that PowerMill provides to the user.

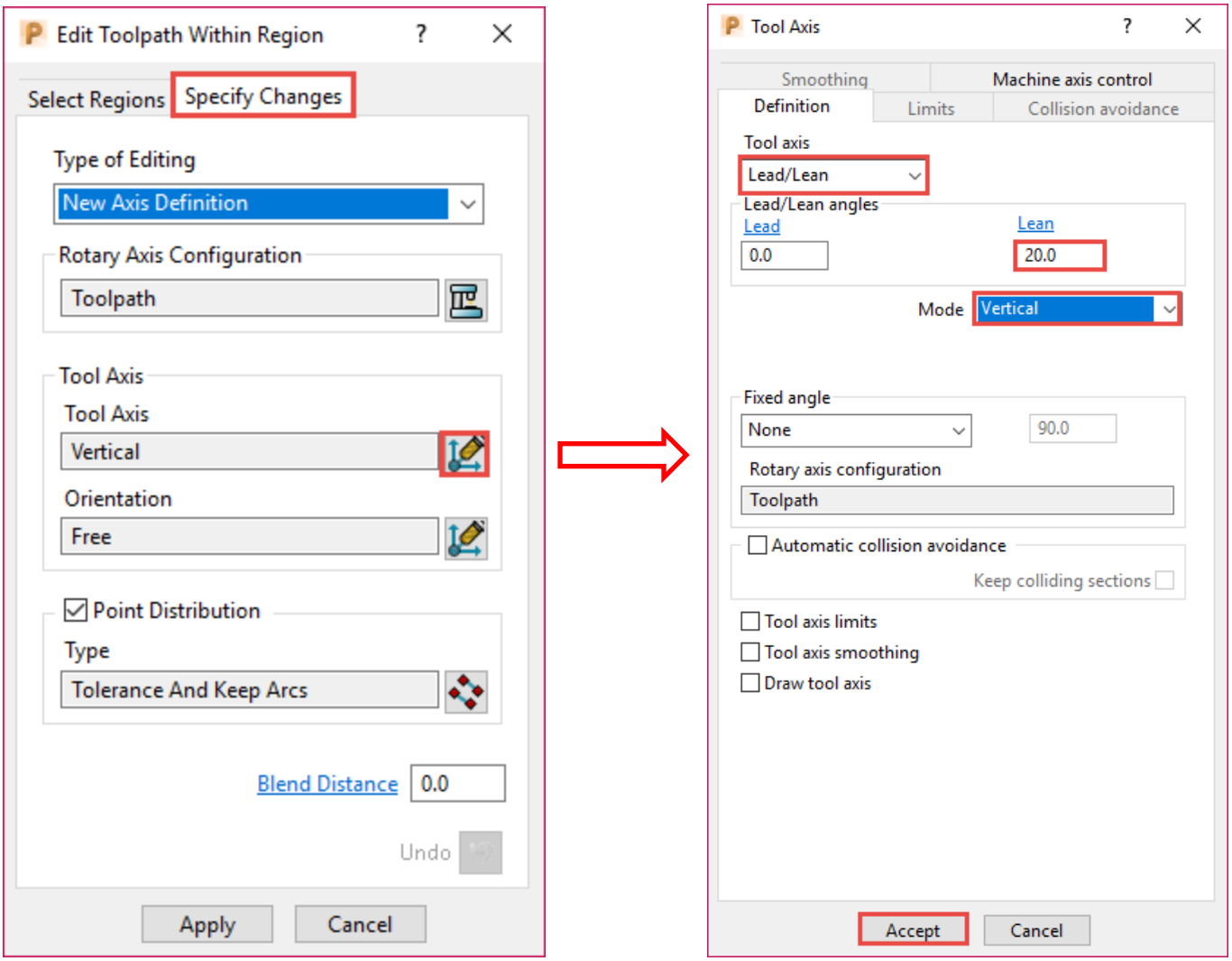
Go to the **Toolpath Edit** tab and select the **Edit Within Region** icon.




1. In the **Define Region By** select **Whole Segments** in the dropdown menu
2. Left mouse drag a selection box across the entire toolpath
3. Select Save **Selection**

Click on the **Specify Changes** tab, then:

1. Click on the Icon next to Tool Axis box
2. Change the Tool axis from Vertical to **Lead/Lean**
3. Enter a **20** value in the **Lean** field
4. Change the Mode option from Contact Normal to **Vertical**
5. Press Accept




6. Then press the **Apply** button
7. Then re-simulate the toolpath

**Watch the video demonstation here…….[https://autode.sk/2SimKvG](https://autode.sk/2SimKvG)**

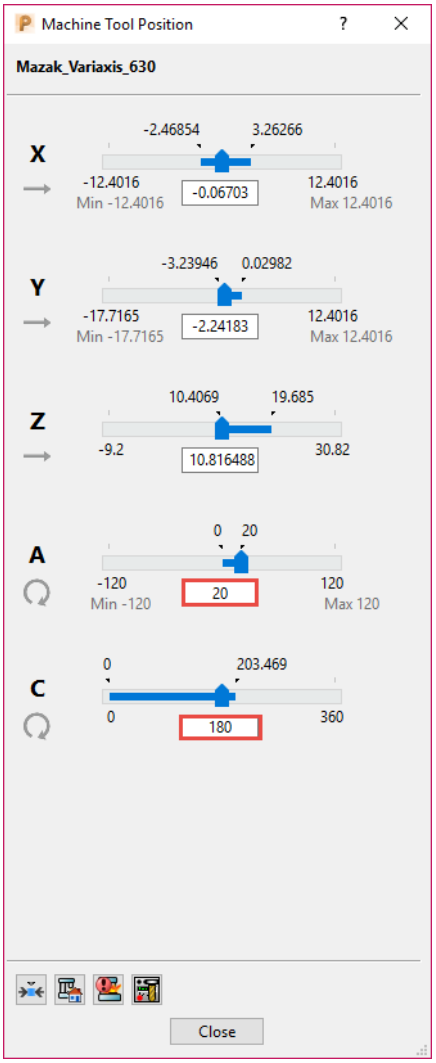
# Tool Axis to Fixed Direction

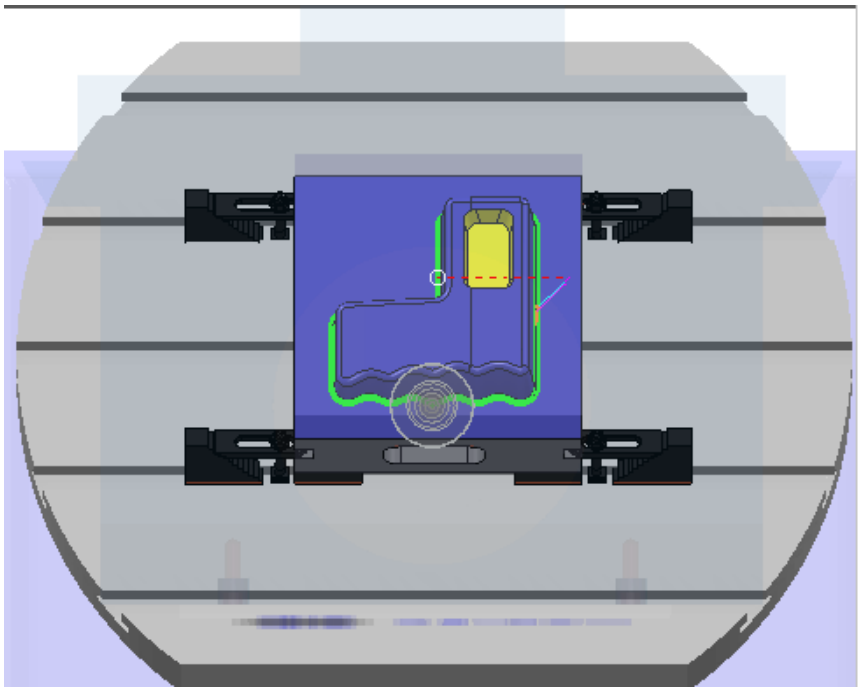
During the simulation, you may have noticed that there was quite a bit of movement in the rotary axis when the toolpath cut on the side of the model that has the wavy cut outs. Since we applied a lean to the entire toolpath, the tool axis by default stays perpendicular to the toolpath segment.

Let's use the same tool, but this time we will only edit the portion of this toolpath to eliminate the unnecessary table movement.

Open up folder **Fixed Direction** and activate toolpath **Corner_Finishing_Lean_Fixed_Direction**

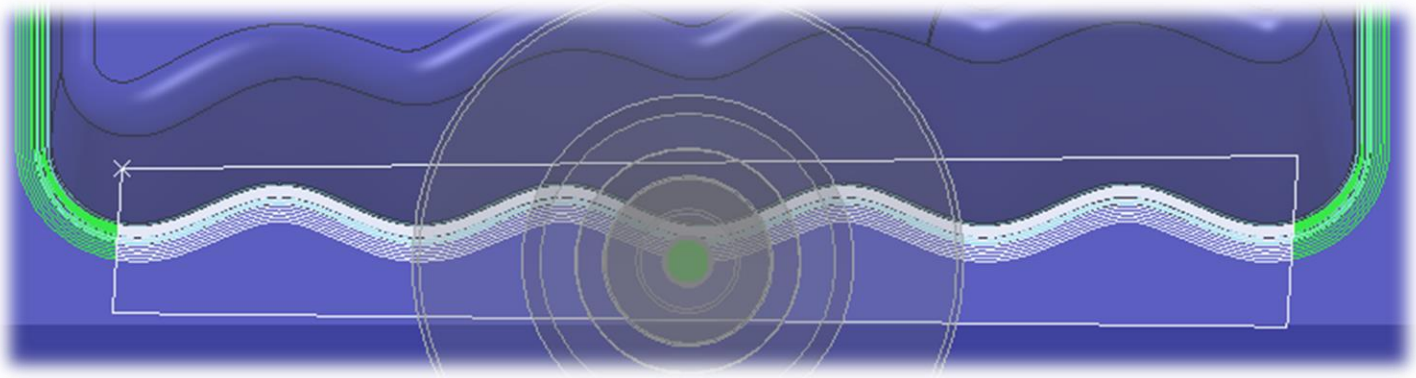
Right click on the toolpath and simulate from start. This time pause the simualtion in the area we are trying to fix, but try to pause it close to the A20 C180 mark (turn on the Machine Tool position to view the angles). Once paused, type in these values into the A & C axis fields.



Go to the **Toolpath Edit** tab and select the **Edit Within Region** icon. Change the **Define Region By** option to **Polygon.**

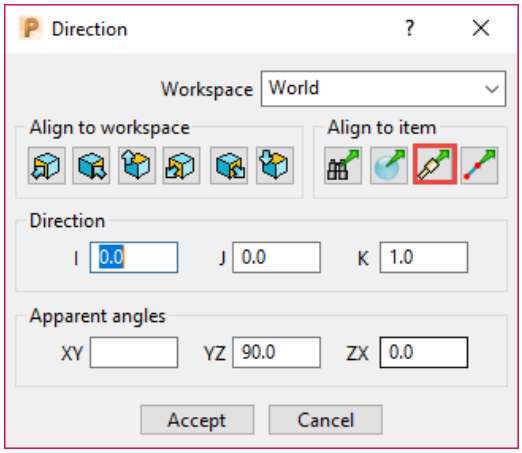
Select a box around the portion we are going to modify and press **Save Selection**



Go to the **Specify Changes** tab and select the icon next to the Tool Axis.

Change the Tool Axis to **Fixed Direction.**

Select the icon in the Direction section to input the new vector.


Click on the **Align with tool** icon to change the new vector to how we have the tool aligned currently in the simulation.

Press **Accept** twice to get back to the **Specify Changes** area

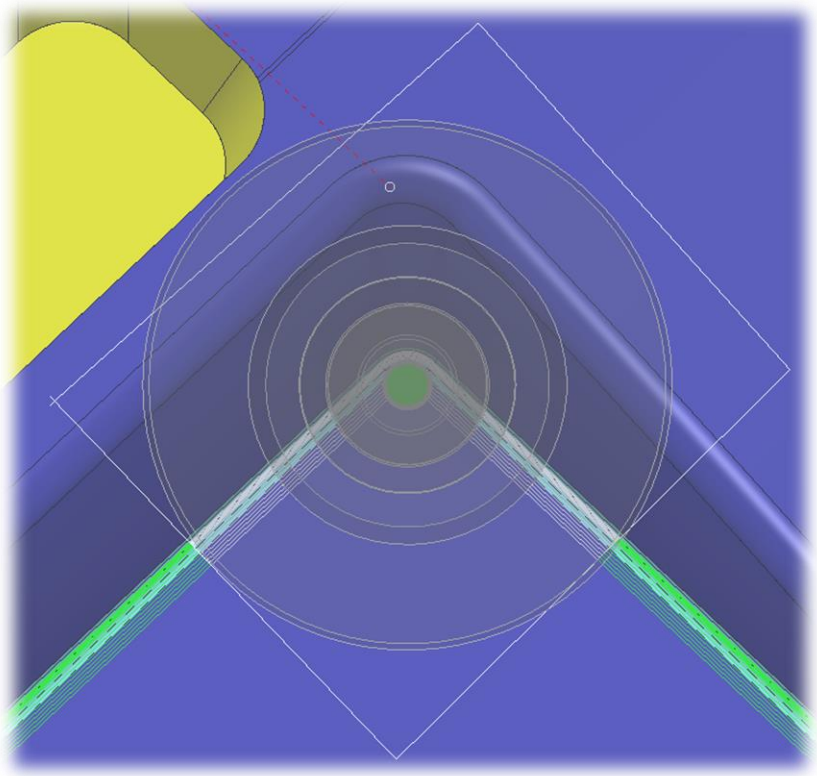
Enter 0.125 in the **Blend Distance** field

Press Apply, but do not hit cancel. Leaving the tab open allows the option to undo the changes. Just simply move the box out of the way by dragging it.

Run the simulation on the toolpath.  The tool axis it that region now stays fixed to the new vector we applied. However, the interior corner in the opposing side of the part is causing a large violent change of direction.

Pause the simulation near **A20 B315**.  Place these value in the Machine Tool Position form

Go back to the Select Regions tab and keep the selection on **Polygon**.  Create a polygon in the corner and press **Save Selection.**



Go to:

1. **Specify Changes**
2. Click on the **Tool Axis** icon
3. Click on **Direction Input** icon
4. Click on the **Align to tool** icon
5. Press **Accept** twice
6. Place a **0.5** value in the **Blend Distance** field
7. Press **Apply**

Do not close the Edit Within Region tool, but simply drag it out of the way.

Run the simulation

Run a gouge check

Run a collision check

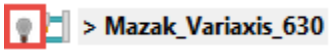
**Watch the video demonstation here…….https://autode.sk/2O7jy2m**

# Axis Interpolation

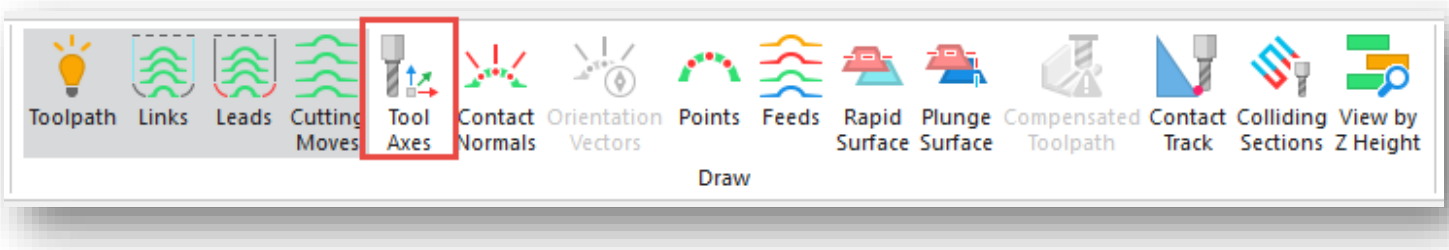
In the previous example, we changed the behavior of the tool axis by creating a new vector from the placement of the tool. Let's us the same editing tool in this next example, but choose an alternitive method instead.

Open folder **Axis Interpolation** and active toolpath **Corner_Finishing_Axis Interpolation.** This toolpath already has the lean tool axis applied to it, but the rotary table swings in the back of machine during the cycle.

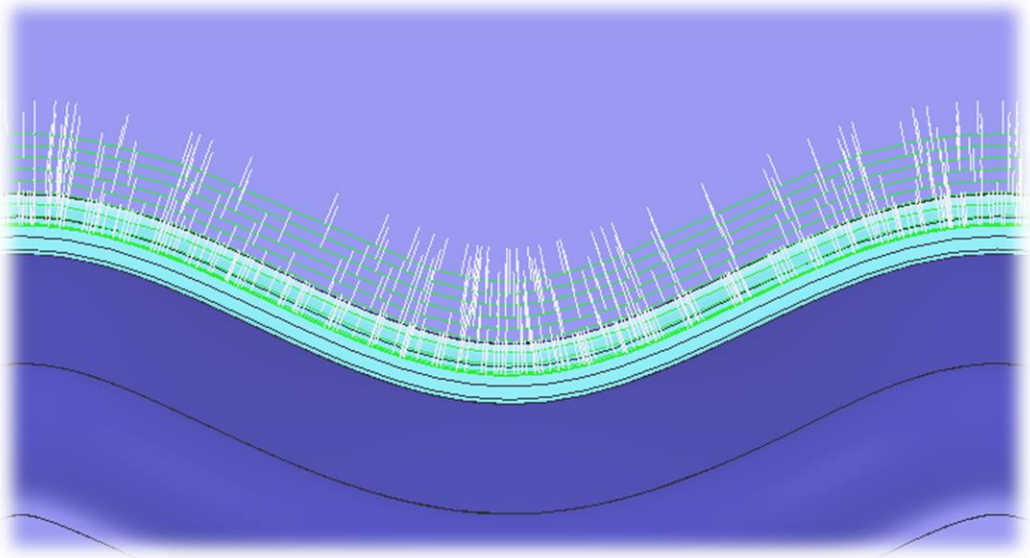
Go to a top view (CTRL + 5) or select the option from the Viewcube

Turn off the MTD and zoom into the part 


We can draw the tool axes at each point in the toolpath by going to the **Toolpath** tab and selecting the **Tool Axes** icon in the **Draw** group.




Zooming into the same area as the previous example, we can clearly see how the tool would behave at each point in the toolpath in the wavy section of the model.

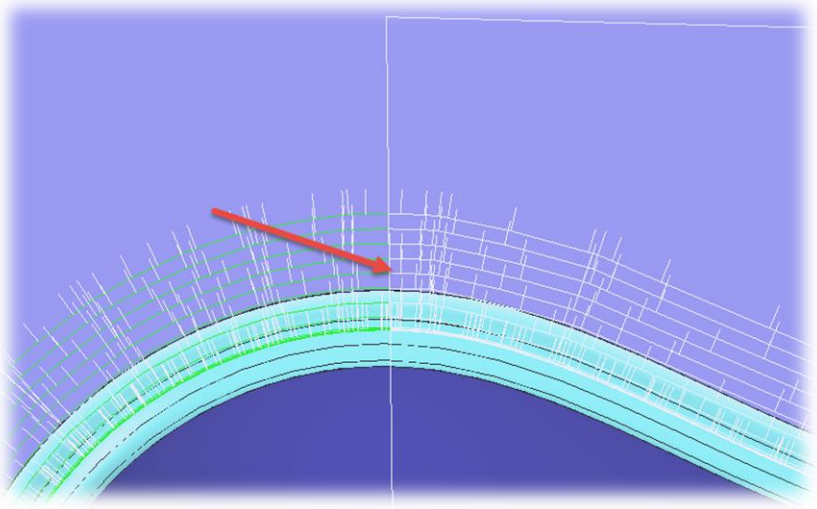


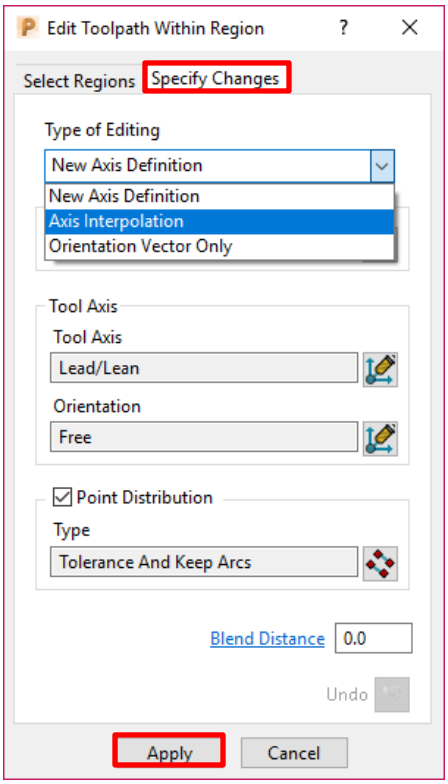
We will use an option in PowerMill to smooth these tool axis combs within a area that we will define.

Go to the **Toolpath Edit** tab and select the **Edit Within Region** icon

Use the **Polygon** method to select a box around the wavy section in the back of the model

**Tip:** When creating the polygon, look for axes that are vertical in both the lefthand and righthand sides of the bounding box. This will make all of the tool axes in between straighten towards these on the outside.






1. Press the **Save Selection** button
2. Go the **Specify Changes** tab
3. In the **Type of Editing** dropdown, select **Axis Interpolation**
4. Press **Apply**

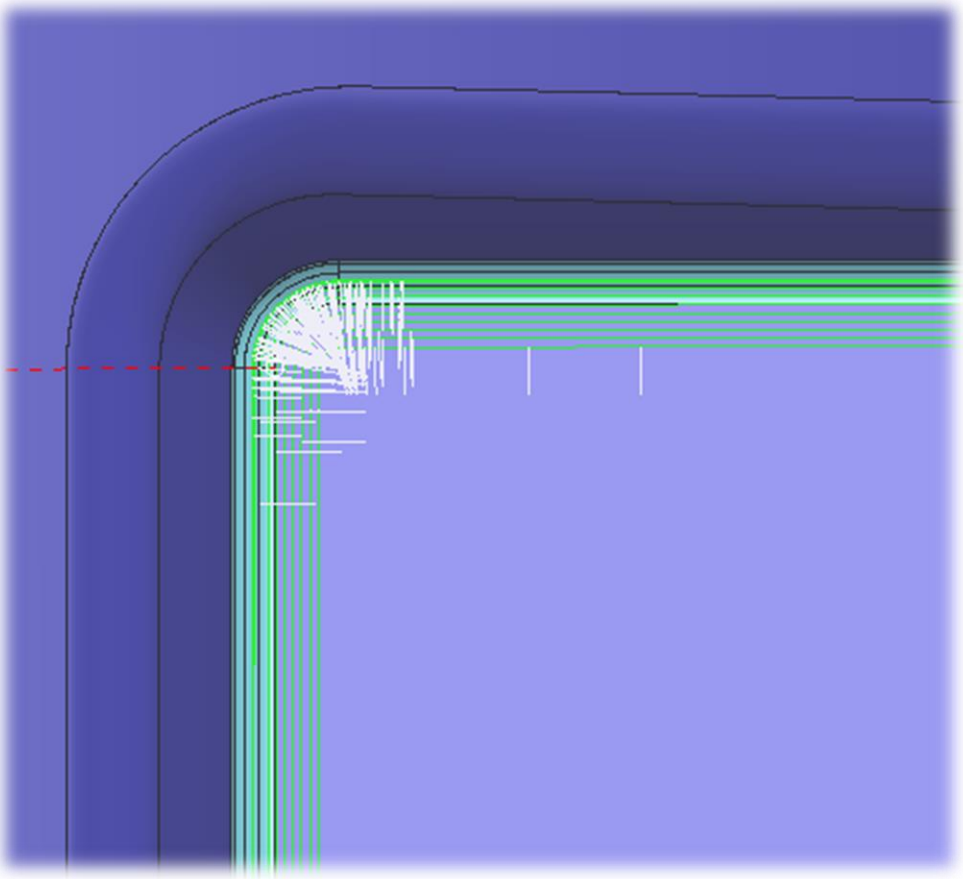
Do not close the **Edit Within Region** tool, just drag it out of the way for now.

Turn the MTD file back on and run a simulation  **> Mazak_Variaxis_630**

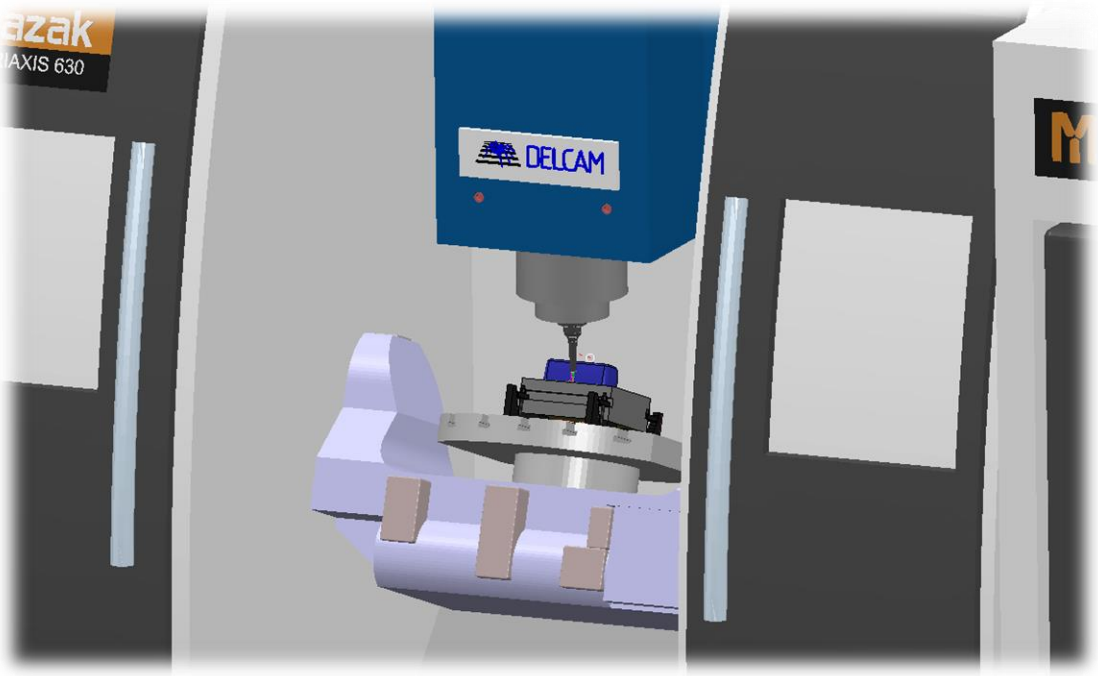
Notice how the tool axis behave in the wavy portion of the model

Press **Cancel** to close the editing tool.

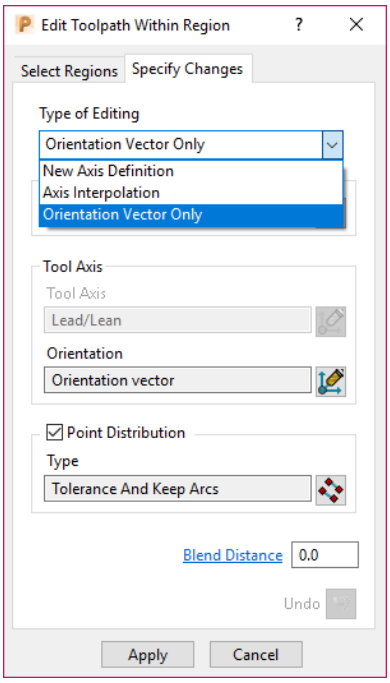
Now follow these steps from above and do the same operation in the interior corner on the opposing side of the part.



Another minor issue that we can correct is the orientation of the table during the simualtion.  You may or may not have noticed, but the table tips towards the back of the machine.  The angle isn't that great, but it may be difficult for the operation to see what is happening.

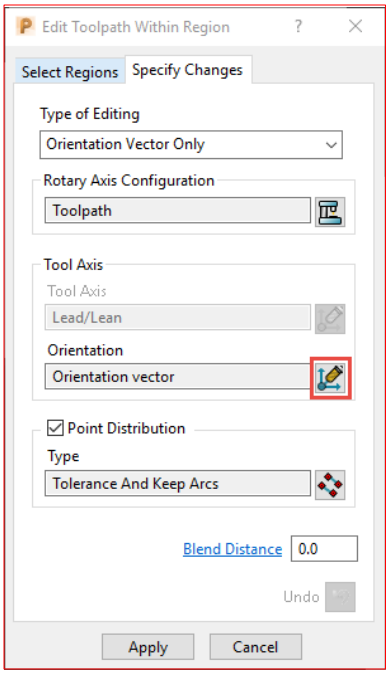


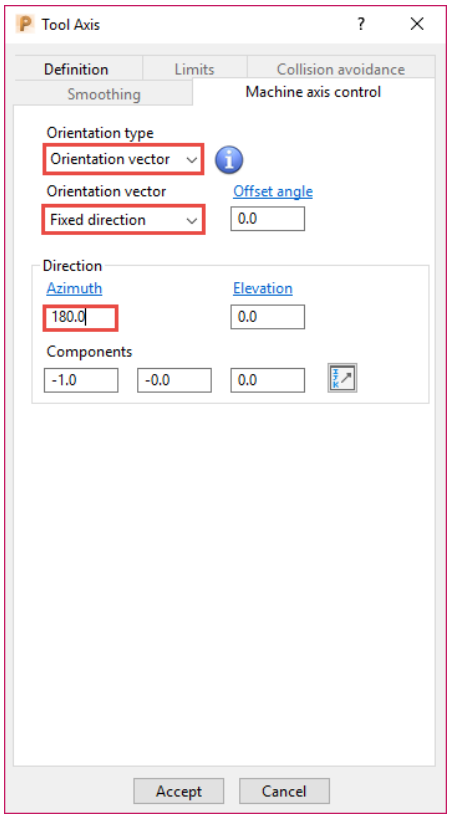

Let's modify this so the table tips towards the front of the machine.




1. Go to the **Toolpath Edit** tab, and select the **Edit Within Region** icon
2. Change the **Define Region By** to **Whole Segments**
3. Box select the entire toolpath and **Save Selection**
4. Go to the **Specify Changes** tab and change the **Type of Editing** to **Orientation Vector Only**

Click on the icon next to Orientation vector



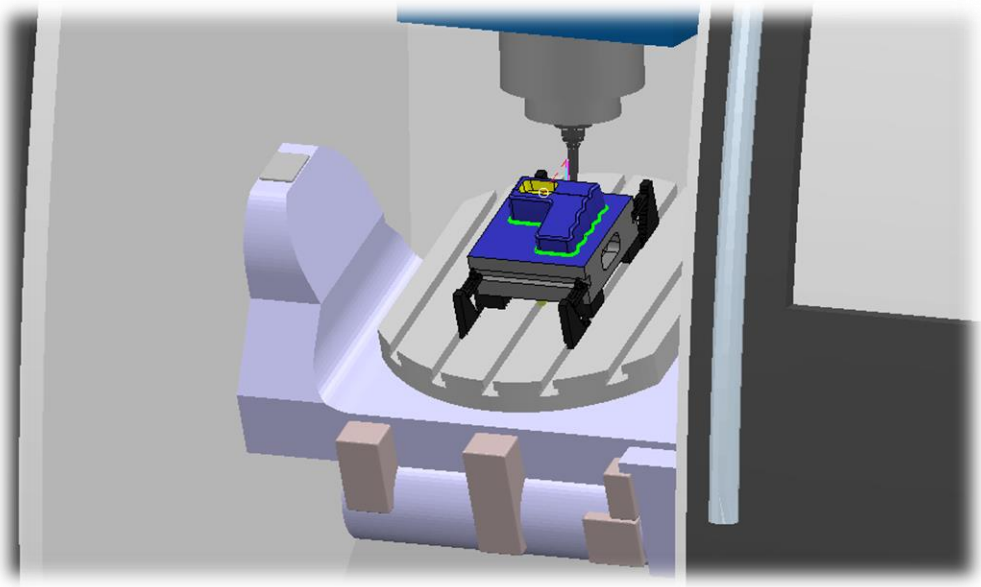




Fill the Machine axis control form as shown in the image to the left.

Press **Accept**

Press **Apply** but do not close the editing tool, drag it out of the way for now

Run the simulation and view the results. The table should tip towards to operator.



**Watch a video demonstration of this example here…….https://autode.sk/2SiArdZ**

# Extra Time?

Feeling good about what you've learned here?

Open up folder **Challenges**

Try and use some of the options we went through today, and fix the following toolpaths.

**Yellow Pocket:**

1. Apply Appropriate Leads and Links
2. Re-order toolpath to cut any direction
3. Make sure the status light is blue
4. Move start/end points to a safe location (not in the corner)

**Raster Wall:**

1. Remove the collsion(s)
2. Change the tool axis to C-45 & A25
3. Swap the table towards the operator
4. Make sure the status light is blue

*Tip: Look at the tools available & use the DMC*

**Autodesk Scribe:**

1. Change the tool axis to a fixed direction
2. Table toward the operator
3. Reorder the toolpath to cut from theAutodesk logo towards the letter K
4. Make (4) multiple passes as 0.01, cutting from the top down
5. Be sure the status light is blue

*Tip: Use a combination of machine simulation and Edit Within Region to establish the fixed direction. Using the reordering function to bypass the cutting behavior (use the move to end option).*

# Thank You!

I'd like to thank you all for attending my lab here at Autodesk University 2018. Hopefully you gained some knowledge and understanding of how a great product like PowerMill provides amazing toolpath editing options for its users.

Even if you are a beginner or an well versed programmer, you can see for yourself how much control the software allows with little ease.

If you'd like to find out more about what PowerMill can provide for you or your company, go visit https://www.autodesk.com/products/powermill/overview. You can download a free 30 day trial there and review some great information listed there.

Here is another great link on the complete catalogue of manufacturing software Autodesk provides https://www.autodesk.com/solutions/manufacturing

Lastly, if you'd like some free introductory training, visit https://learn.manufacturing.autodesk.com