Basic 3-Axis CAM in Delcam PowerMILL

Brian Ringley – Woods Bagot

PE4997-L This course will examine the fundamentals of a basic 3-axis setup using Delcam PowerMILL. We will start by learning how to set up a part within stock, and then we will use area clearance toolpaths to rough the part. We will then discuss various finishing strategies, moving from a semi-finish to a true finish, and we will follow up with detailing toolpaths, such as auto cornering and pencil cuts. Finally, we will use workplanes and interactive stock models to set up a second position, commonly known as a "flip mill," before analyzing our toolpaths for gouging and collisions and then simulating our numerical control programs.

Learning Objectives

At the end of this class, you will be able to:

- Learn how to position a part within stock for a 3-axis CAM setup
- Learn how to set up a CNC programming strategy moving from rough to semi-finish to finish to detailing
- Learn how to use workplanes and interactive stock models to set up a second position for a "flip mill"
- Learn how to analyze and simulate NC programs to better ensure quality and avoid gouging and collisions

About the Speaker

Brian Ringley is the design technology platform specialist at Woods Bagot, where he specializes in project-specific design technology integration, design computation methodology, and Building Information Modeling (BIM). Prior to working at Woods Bagot, he was the Fuse Lab technology coordinator for the Department of Architectural Technology at the New York City College of Technology (CUNY), and he was the digital fabrication coordinator for the College of Design, Architecture, Art, and Planning at the University of Cincinnati. He has worked in the architectural offices of Kohn Pederson Fox Associates, Dellekamp Arquitectos, and R&Sie(n), and he has taught fabrication seminars and design studios at the University of Cincinnati, the City University of New York, and Pratt Institute. He has also led design-build workshops at Rensselaer Polytechnic Institute and plusFARM, and he has sat on juries as an invited critic at Columbia Graduate School of Architecture, Planning, and Preservation and Cornell University College of Architecture, Art, and Planning. He is a regular contributor to Designalyze.com and AEC-APPS.com.

email: brian.ringley@woodsbagot.com

Twitter: @brianringley

Index

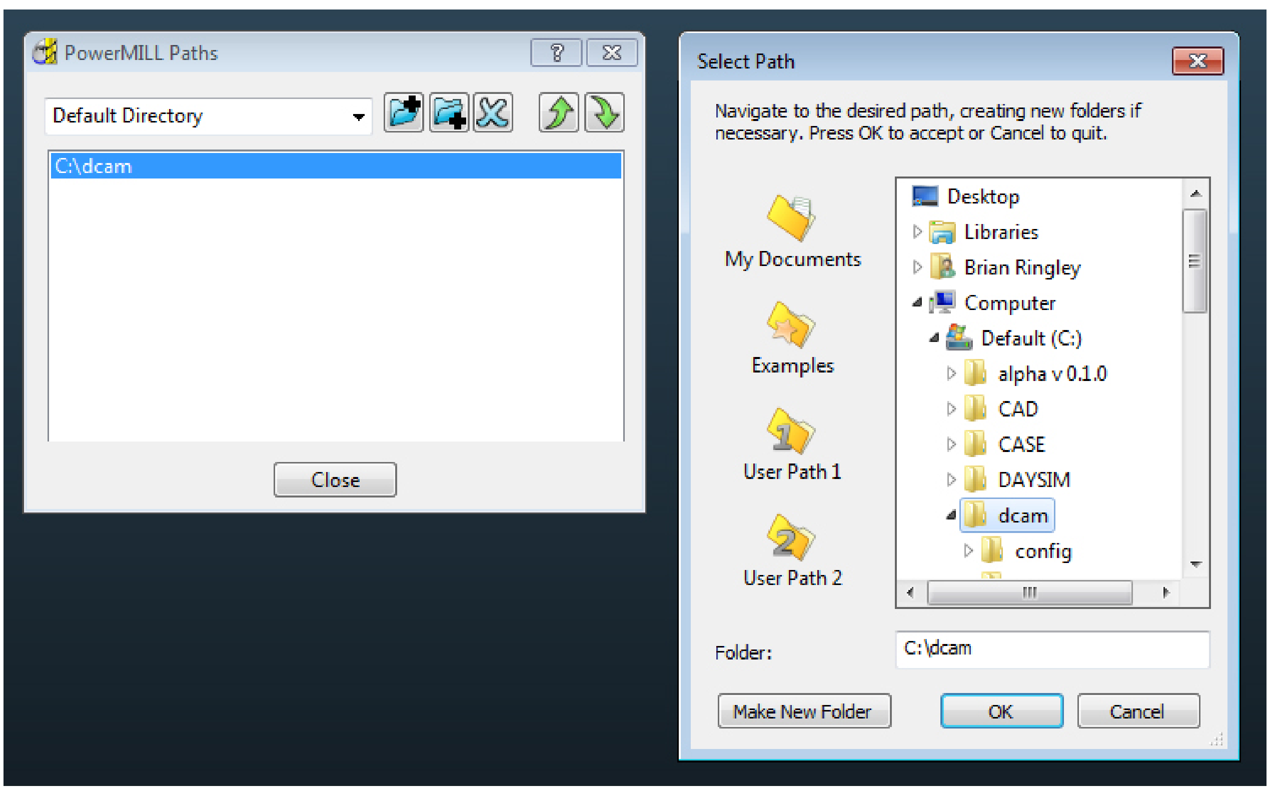
Positioning a Part within Stock

Importing a CAD Model

Setting a Default Directory

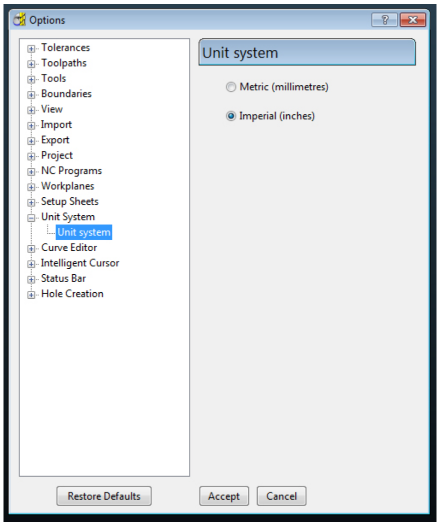
After first launching PowerMILL it's helpful to set a default directory by going to **Tools > Customize Paths**.

1. In the **PowerMILL Paths** window, select **Default Directory** from the drop-down menu. Add a file path by selecting the **Add path to top of list** button and browsing to the desired directory in the **Select Path** window. A good first path first path to add is **C:\dcam\pmill4**. Once the path is selected, hit **OK** to close the **Select Path** window.
2. Repeat the process with the same path for the **File Dialog Button 1** option in the drop-down menu.
3. For **File Dialog Button 2**, I recommend C:\Program Files\Delcam in case there's a need to access system files. Other paths can be set as needed. When finished, select **Close** to close the **PowerMILL Paths** window.



Setting Units

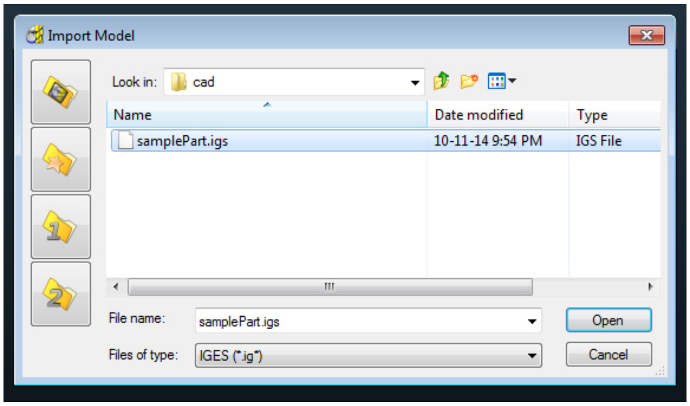
To set the desired units go to **Tools > Options** and then expand **Unit System** in the **Options** window. Select **Unit System** and then select the desired radio button, either **Metric (millimeters)** or **Imperial (inches)**.



Importing the Part

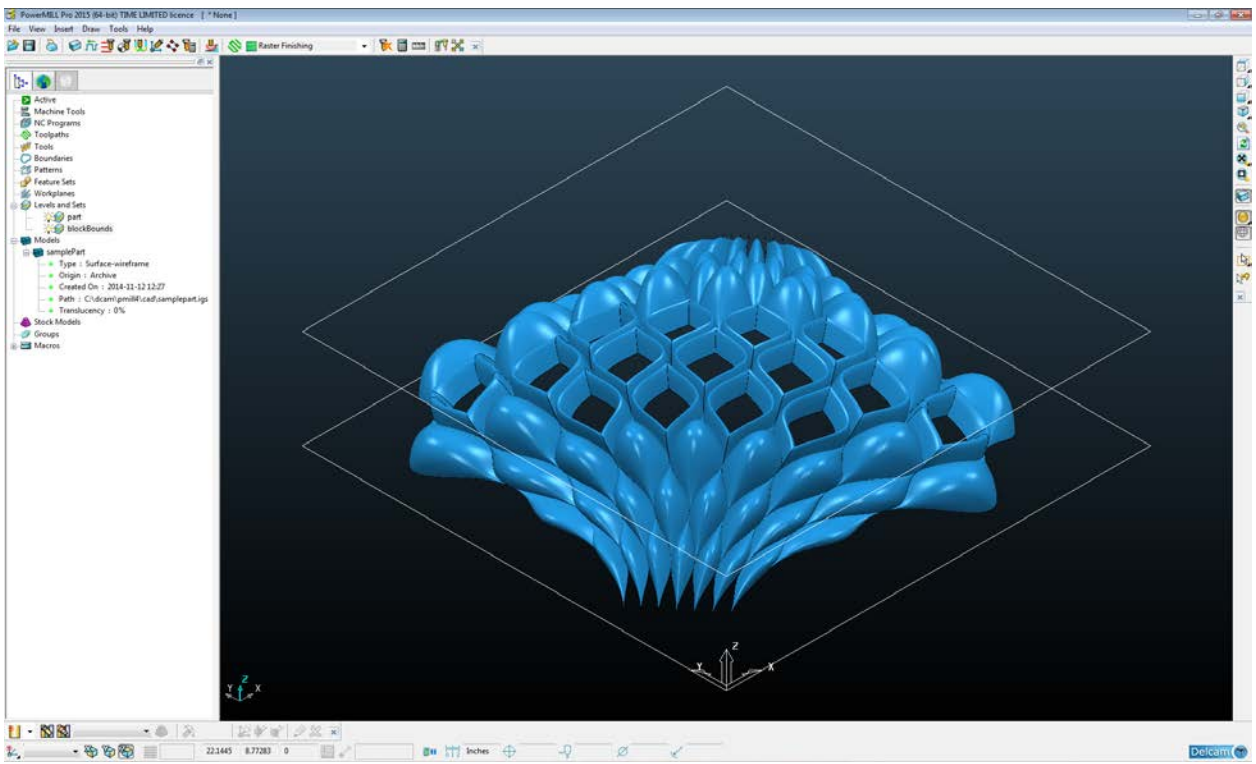
To import a CAD model of the part, go to **File > Import Model…** or right-click on Models in the **PowerMILL Explorer** and select **Import Model…** from the drop-down menu. In the **Import Model** window select **File Dialog Button 1** to automatically jump to the previously specified directory. In this example I will navigate to **C:\dcam\pmill4\cad** and set the **Files of type:** drop-down menu to **IGES (*.ig*)** or **All Files (*)** prior to selecting **samplePart.igs** by double-clicking the file name or selecting **Open**.

Note: PowerMILL is capable of importing many file types depending upon your licensed exchange file types within Delcam Exchange.



After the model imports successfully there will be a model in the **Models** drawer of the **PowerMILL Explorer** window. Also, the geometry will maintain layer sorting from the original CAD file in the **Levels and Sets** drawer, although the levels will need to be renamed by right-clicking each level name and selecting **Rename** from the cursor menu. To better see the part:

1. Select the **Iso 1** and **Plain Shade** buttons from the toolbar to the right of the screen.
2. Make sure that Wireframe button, also on the right, is selected so that curve objects display.
3. To better see the wireframe curves, select them and right-click over top of them. Then select **Color…** from the drop-down menu to select a custom color.

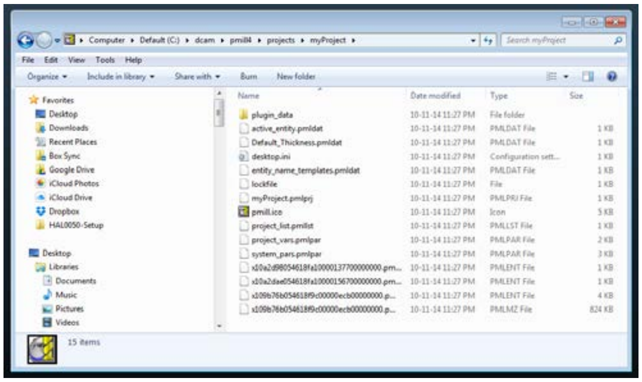


Defining the Block

Saving the File

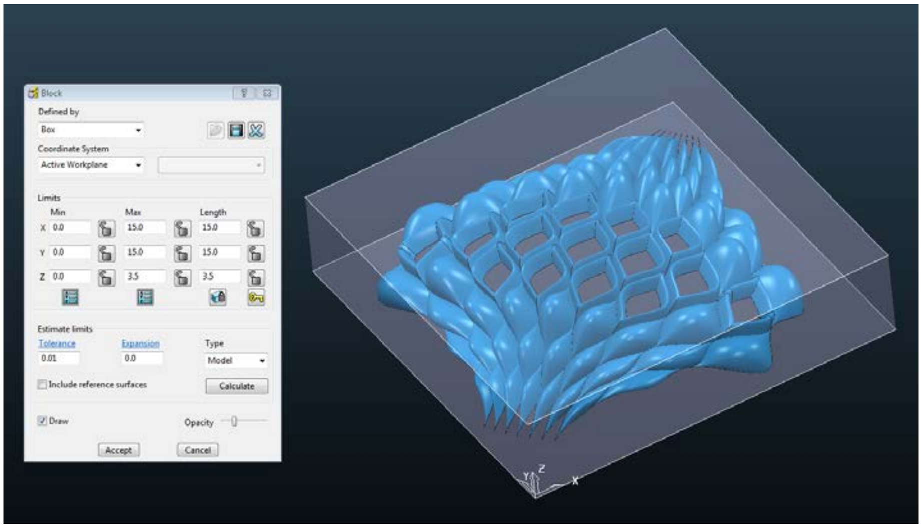
It's a good time to save the file – to do this go to **File > Save Project As…** and save the project to **C:\dcam\pmill4\projects** as **myProject01** (this file is available as a course resource).

*Note: A PowerMILL project file is actually a file directory, so remember to archive the file as a *.zip or *.rar file if exchanging via the email or web. One advantage of this is that it's difficult to inadvertently save over a file.*



Sizing the Block

Select the **Block** icon from the toolbar at the top of the screen to open the **Block** form. We will create a simple **Box** type block from the **Active Workplane**. Because the extent of the block is modeled on the **blockBounds** level of the imported part, we can simply press the **Calculate** button to automatically size our block. Otherwise dimensions can be entered manually into the **Limits** fields.



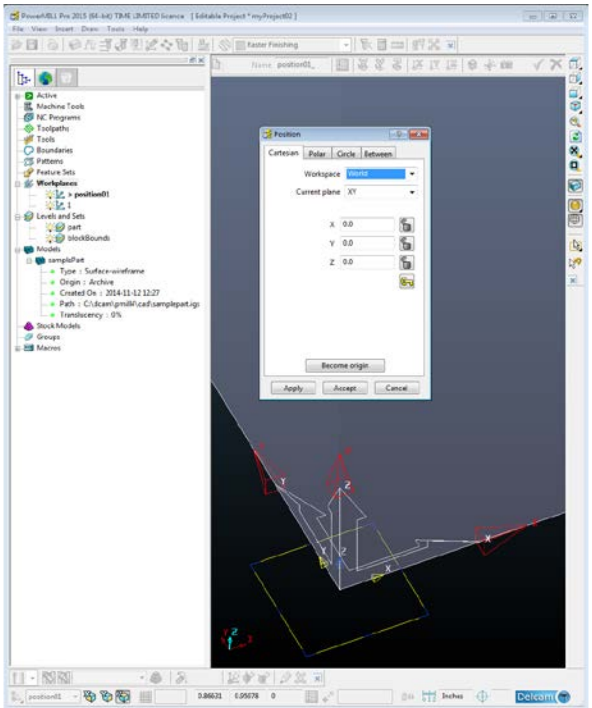
Block Visibility

Block Opacity can be adjusted on a slider within the Block form. Block visibility can be toggled using the Draw checkbox within the Block form or by selecting the Block visibility icon in the right toolbar.

Defining a Workplane

In this first example we are using the default global workplane, an XY plane with Z up at the origin 0,0,0. However, we can explicitly define a workplane by right-clicking on **Workplanes** in **PowerMILL Explorer** and selecting **Create Workplane…**

1. A workplane creation toolbar will appear at the top of the screen. Name the workplane **position01** and then raise the **Position** form to ensure that the workplane is in the **World** workspace on the XY plane at coordinate position 0,0,0.
2. Select **Apply** to exit the form, then accept the changes by clicking on the **Green Checkmark** icon.
3. The new workplane will appear in the **Workplanes** drawer in **PowerMILL Explorer**. Double-click **position01** so that it turns bold and a carrot appears to the left of its name. This means that the workplane is now the active workplane for downstream operations.
4. The progress file **myProject02** is available as a course resource.

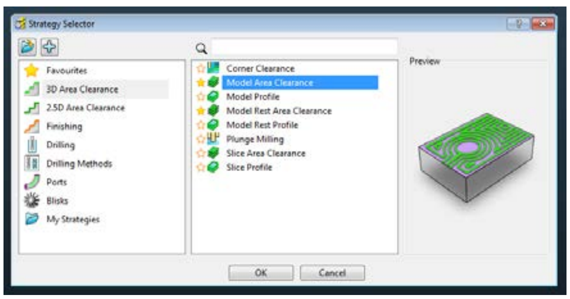


CNC Programming Strategy

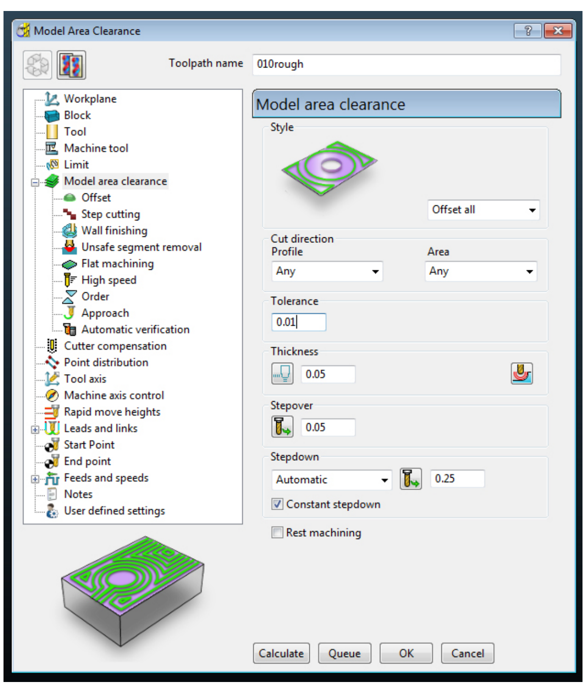
Roughing

Area Clearance Toolpath

Select the green **Toolpath Strategy** icon in the top toolbar to launch the **Strategy Selector** window. In the left-hand list select **3D Area Clearance**, then select the **Model Area Clearance** toolpath form the list on the right and hit **OK**.



The **Model Area Clearance** toolpath form will appear – in the default section set **Style** to **Offset all** and the **Profile** and **Area Cut direction** both to **Any**. This will ensure that the toolpath is offsetting from both the block and the part geometry and is constricted to neither a climb nor a conventional cutting direction. Name the toolpath **010rough**.



Area Clearance - Set Workplane and Block

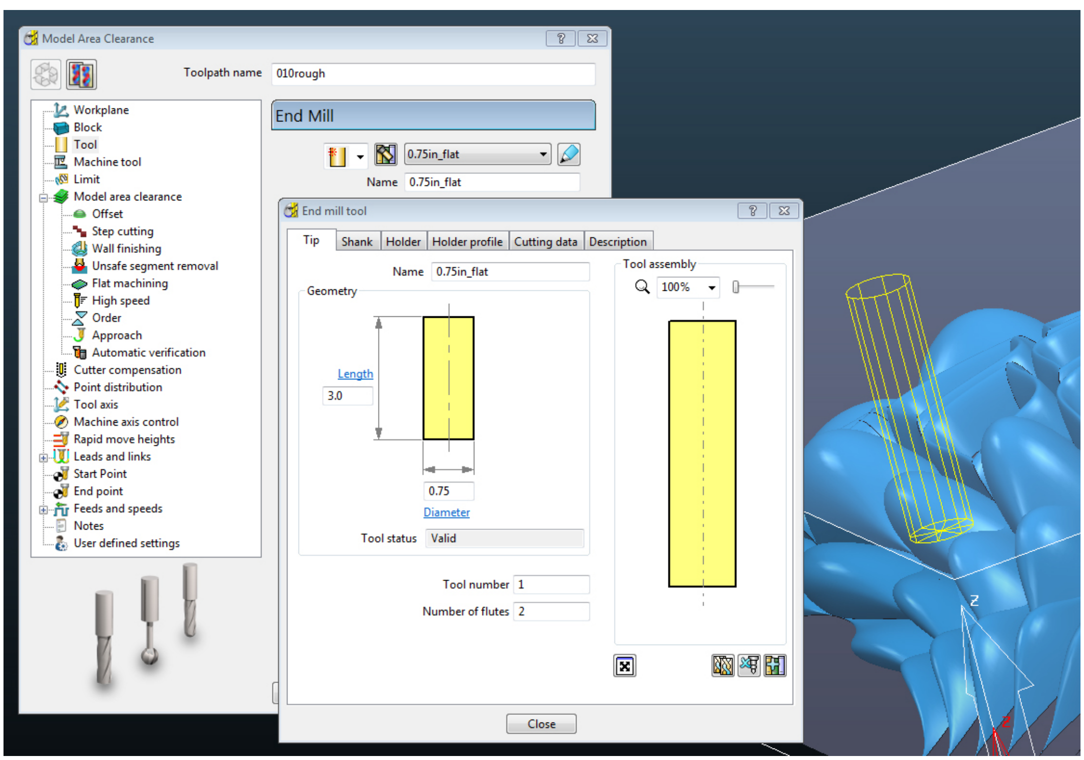
Select **Workplane** in the left menu of the **Model Area Clearance** toolpath form – the **position01** workplane should already be set since it was set active prior to entering the form.

Now select **Block**. This, too, should bring in the block information that was already set. However, we will adjust the bottom of the block as a means of preventing the roughing operation from cutting all the way to Z 0 (for example, we may be relying on a continuous material base for vacuum holding). Set the **Z Min Limit** to **0.25**.

Area Clearance - Set Tool

Select **Tool** from the tree on the left of the form. We will create a cutter for the rough from scratch. Select the **end mill** tool type and name the tool **0.75in_flat**. Then select the **Pencil** button to raise the **End mill tool** form where we can define the tool dimensions. In the **Tip** tab:

1. Set **Length** to **3.0**.
2. Set **Diameter** to **0.75**.
3. Set **Tool number** to **1**.
4. Set **Number of flutes** to **2**.



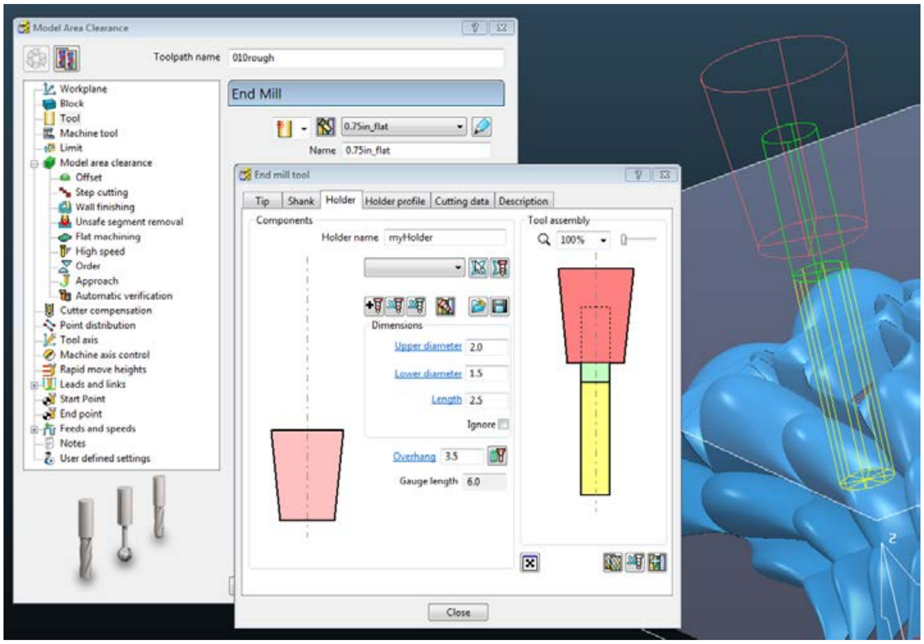
In the **Shank** tab select the **Add Shank Component** button:

1. Set **Upper** and **Lower** diameters both to **0.75** to match the cutting edge diameter.
2. Set **Length** to **2.0** (for a total tool length of 5.0).

In the **Holder** tab select the **Add Holder Component** button:

1. Set **Holder** name to **myHolder**.
2. Set **Upper diameter** to **2.0**.
3. Set **Lower diameter** to **1.5**.
4. Set **Length** to 2.5.
5. Set **Overhang** to 3.5 so that the full cutting length is extended outside of the holder while maintaining 1.5 inches of the shank within the holder for good grip.

Note: Hyperlinked dimensional inputs allow the user to measure parameters directly from the model.



In the **Cutting data** tab set **Coolant** to **None**. We could also set cutter speeds and feeds here so the tool has default values but we will do that in the toolpath form instead. **Close** out the tool form.

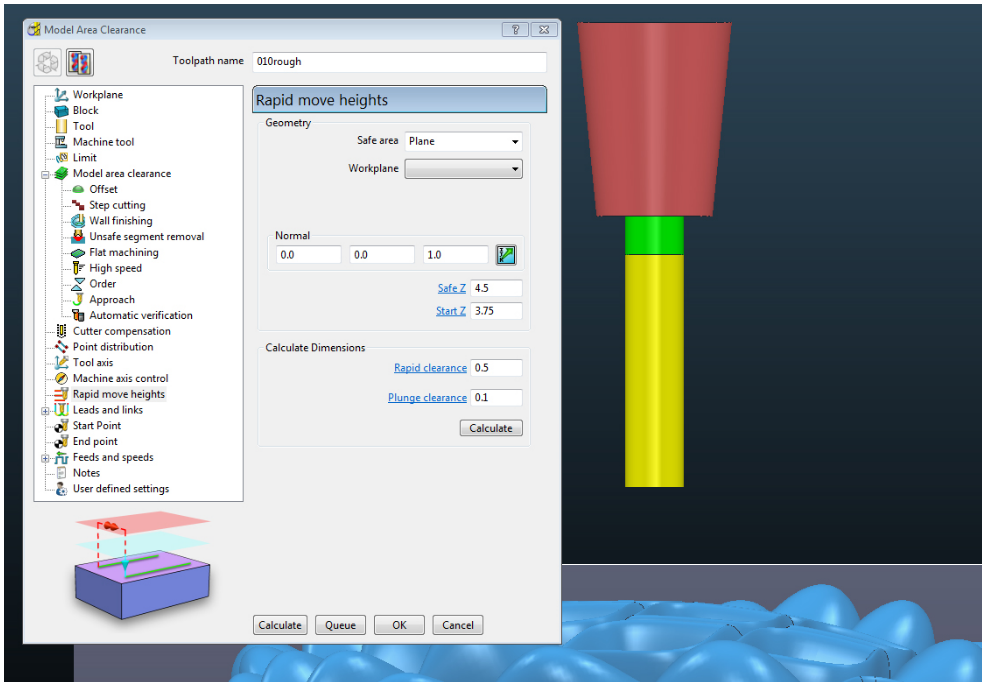
Return to **Model area clearance** – now that we know the diameter of our tool we can determine our toolpath parameters. Set **Stepover** and **Stepdown** both to **0.3**.

Note: The Hand icon signified that the user has manually overridden toolpath form values.

Area Clearance – Rapid Move Heights and Leads and Links

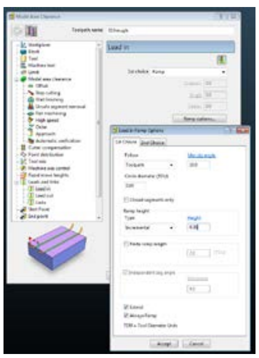
Go to **Rapid move heights** where we can set the **Safe Z**, the height at which the tool moves rapidly over the part and block, to **4.5** (1 inch above the block) and the **Start Z**, the height at which the tool begins its descent into the block, to **3.75** (0.25 inches above the block).

We can easily visualize the Safe Z by selecting the Front View icon for an orthogonal view and then right-clicking the tool and selecting Shaded to better visualize the tool.



Next go to **Leads and links > Lead in** where we will set a **Ramp** entry as our **1st choice** entry strategy. Select the **Ramp options…** button:

1. Set **Follow** to **Toolpath** to make the entry XY coordinates match the corresponding cutting XY coordinates – a good way to make sure the tool doesn't travel somewhere unexpected during entry.
2. For **Height** I use the following rule of thumb to make sure the tool always begins entry prior to hitting stock: Stepdown + Thickness + Tolerance = Height. For us this would be 0.3 + 0.05 + 0.01 = **0.36**.
3. Check **Always Ramp** if you always want this lead-in strategy.



Area Clearance – Feeds and Speeds

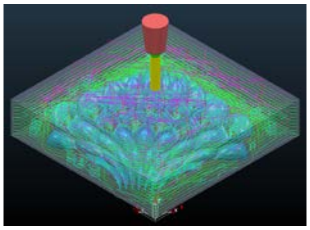
Go to **Feeds and speeds** where we will set our feed rates and spindle speeds:

1. Set **Spindle speed** to **12000 rpm**.
2. Set **Cutting feed rate** to **150 in/min**.
3. Set **Plunging feed rate** to **75 in/min**.
4. Set **Coolant** to **None**.

The progress file **myProject03** is available as a course resource.

Note: Chipload calculations are a good way to get initial speeds/feeds, but it is more of an art than a science (in my opinion) and depends on outside factors like your machine's capabilities.

Hit the **Calculate** button at the bottom of the toolpath form to finish, then hit **Close** to close out the form.



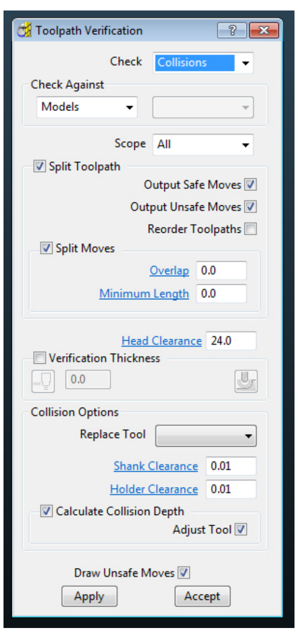
Toolpath Simulation

Safety Status

The **Safety Status** is the colored checkmark to the left of any toolpath name. All Safety Status icons need to be green to guarantee, limited by the accuracy of your programming, that there are neither collisions nor gouging resulting from your tool or tool holder.

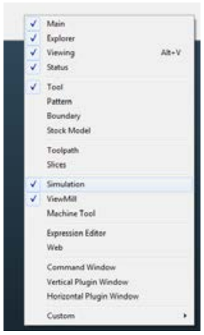
To check the status, activate the toolpath in question and select the **Toolpath Verification** icon to open the corresponding form. It's recommended to add a **Shank** and **Holder Clearance** value that is at least as large as the tolerance used to calculate the toolpath, so I will use a value of **0.01** for both.

Check Against Models for both **Collisions** (holder in contact with stock) and **Gouges** (cutter violating surface model) by selecting Apply. If no problems are found the Safety Status turns green. Otherwise you will want to select Draw Unsafe Moves in the Toolpath Verification Form to further investigate where the problem lay.



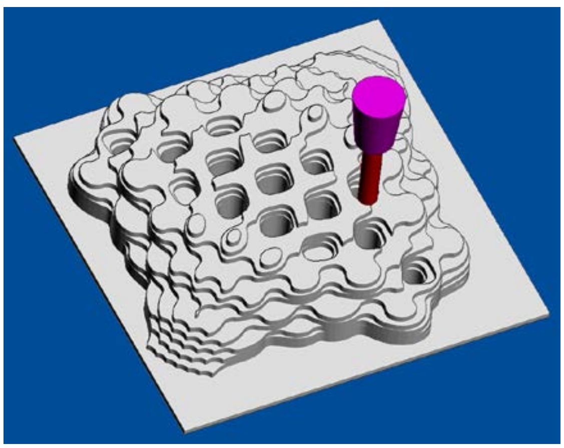
Simulation

To get all of the toolbars you need for toolpath simulation, right click in the upper toolbar area and select **Simulation** and **ViewMill**.



Prior to starting the simulation, make sure to set the **Z Min Limit** back to **0** for an accurate block representation and adjust your view to a good vantage point. There are dynamic simulations but high quality shaded simulations are static.

In the **ViewMill** toolbar, toggle **ViewMill On** and select the **Plain Shaded Image** icon. In the **Simulation** toolbar, select toolpath **010rough** from the drop-down list and hit the **Play** button to simulate the rough. Back in the **ViewMill** toolbar, select the **No Image** icon to return to the model view, or select **Exit ViewMill** to start the simulation all over again.

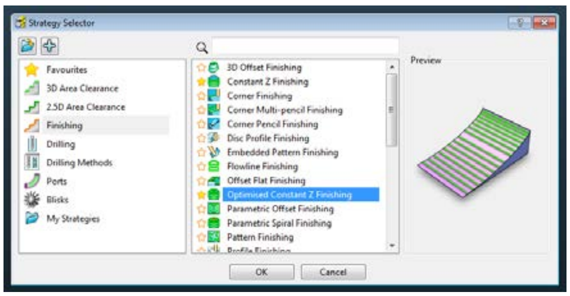


The progress file **myProject04** is available as a course resource.

Finishing

Finishing Toolpath

Select the green **Toolpath Strategy** icon in the top toolbar to launch the **Strategy Selector** window. In the left-hand list select **Finishing**, then select the **Optimized Constant Z Finishing** toolpath form the list on the right and hit **OK**.



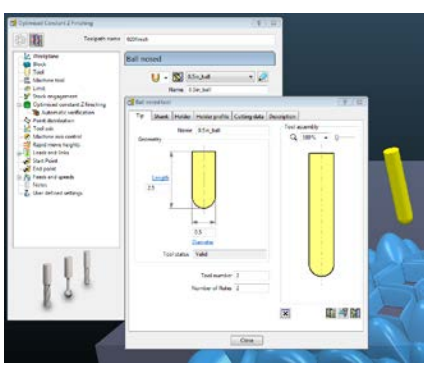
The **Optimized Constant Z Finishing** toolpath form will appear.

1. Name the toolpath **010rough**.
2. Verify that the **Workplane** is set to the active workplane **position01**.

Optimized Constant Z - Set Tool

Select **Tool** and set the tool type to **Ball nosed** and name it **0.5in_ball**. Then select the **Pencil** button to raise the **Ball nosed tool** form where we can define the tool dimensions. In the **Tip** tab:

1. Set **Length** to **2.5**.
2. Set **Diameter** to **0.5**.
3. Set **Tool number** to **2**.
4. Set **Number of flutes** to **2**.

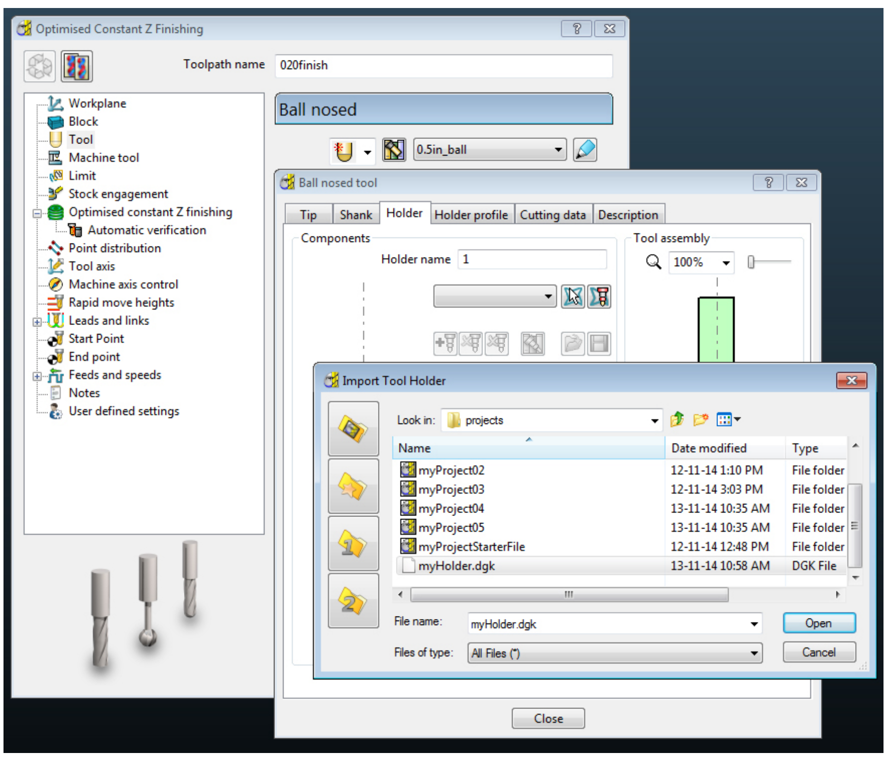


In the **Shank** tab select the **Add Shank Component** button:

3. Set **Upper** and **Lower** diameters both to **0.5** to match the cutting edge diameter.
4. Set **Length** to **1.5** (for a total tool length of 4.0).

Now we've already made a tool holder so instead of making it all over again from scratch, let's reference it from our previous tool:

1. **Close** the **Ball nosed tool** form, then select the **0.75in_flat tool** from the drop-down list within the toolpath form.
2. Select the **Pencil** icon to edit and go to the **Holder** tab.
3. Select the **Save Tool Holder** icon and name it **myHolder.dgk**.
4. Close the **0.75in_flat** tool form and go back to **0.5in_ball** tool form.
5. Go to the **Holder** tab and select the **Load Tool Holder** icon to load **myHolder.dgk**.
6. Set the **Overhang** to **2.875**, leaving 1.25 inches of the shank firmly within the collet.
7. **Close** out the tool form.



Optimized Constant Z – Adjust Block

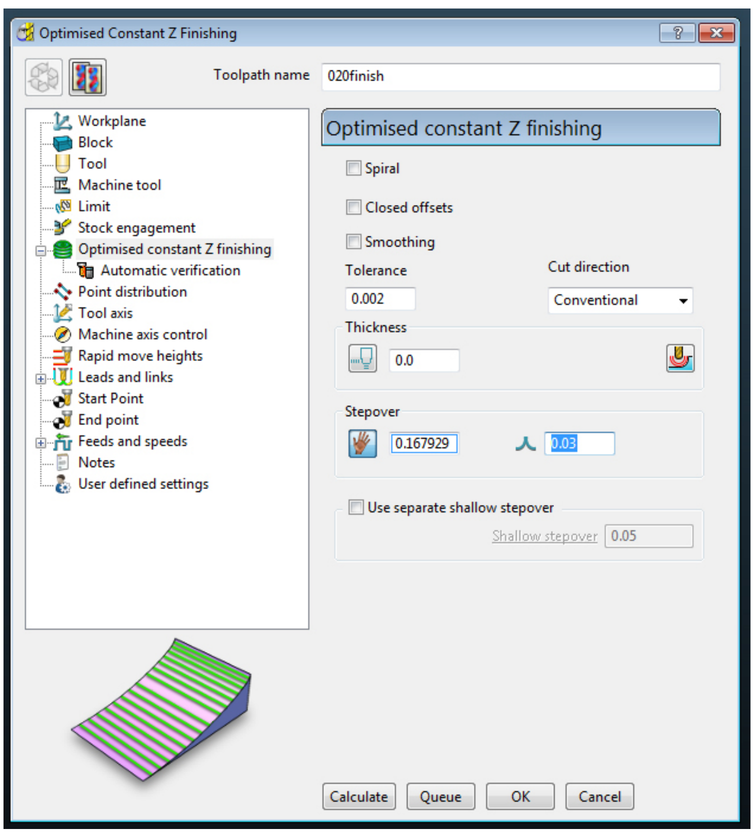
Now that we know the diameter of our ball mill, we can adjust the block accordingly. For a complete finish of a three-dimensional part with a ball mill there is often a need for the tool to dip below the base of the model by the radius of the tool, necessitating a spoil board or fixture for part elevation.

Go to **Block** and set the **Z Min Limit** to **-0.125** to allow the tool to travel this extra depth should it be necessary to finish the part.

Optimized Constant Z Toolpath Parameters

Go to **Optimized constant Z finishing**:

1. Constrain **Cut direction** to **Conventional**.
2. Thickness should be 0.0.
3. Notice that the Stepover can be controlled by the resultant cusp height. Set the Cusp to 0.3 and observe the corresponding change in the Stepover field.



Optimized Constant Z – Rapid Move Heights and Leads and Links

Go to **Leads and links > Lead in**:

1. Set **Ramp** entry as the **1st choice** entry strategy and select the **Ramp options…** button.
2. Set **Follow** to **Toolpath**.
3. Reduce **Height** only slightly to **0.28** as we are directly following a rough and there could still be large steps of stock for the tool to run through.
4. Accept the changes to the **Lead In Ramp Options** form then go to **Leads and links > Links**. Because we have cleared away much of the stock in the prior operation we no longer need to use the Safe height for our rapid moves.
    a. Set **Short** to **On surface** to keep the tool from lifting when linking moves between toolpaths are under the **Short/Long threshold**.
    b. Set **Long** to **Skim** to keep the tool moving at a relative distance from the part rather than always retracting to the Safe height.

Optimized Constant Z – Feeds and Speeds

Go to **Feeds and speeds** where we will set our feed rates and spindle speeds:

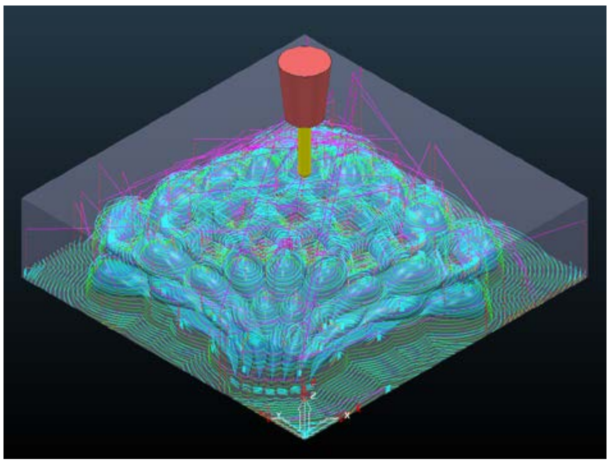
1. Set **Spindle speed** to **18000 rpm**.
2. Set **Cutting feed rate** to **200 in/min**.
3. Set **Plunging feed rate** to **125 in/min**.
4. Lower the **Skim feed rate** to **500 in/min.**
5. Make sure that **Coolant** is set to **None**.

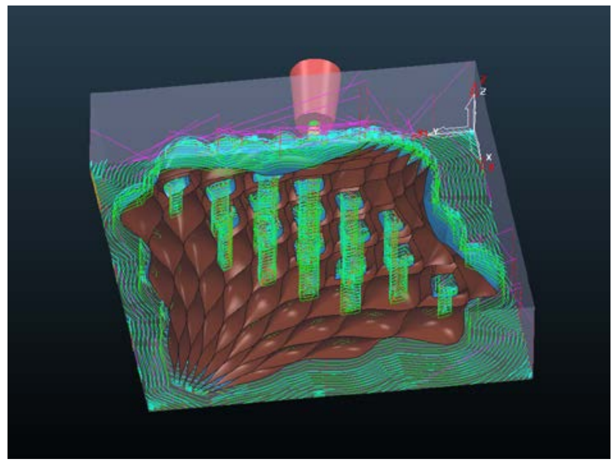
Hit the **Calculate** button at the bottom of the toolpath form to finish, then hit **Close** to close out the form.

Adjusting Toolpaths

Recycling and Cloning

Looking at our toolpath result you'll notice certain efficiencies, such as excessive air moves, superfluous cutting outside of the part, excessive ramping, and the tool dropping too deeply into part pockets. Let's recalculate with different settings and also try using a boundary.





To edit a toolpath, make it active in **PowerMILL Explorer** and then right-click on the toolpath title. Then select **Settings** to launch the toolpath form. To edit a form, you can either **Recycle** or **Clone** the toolpath using the two icons in the upper left hand of the form.

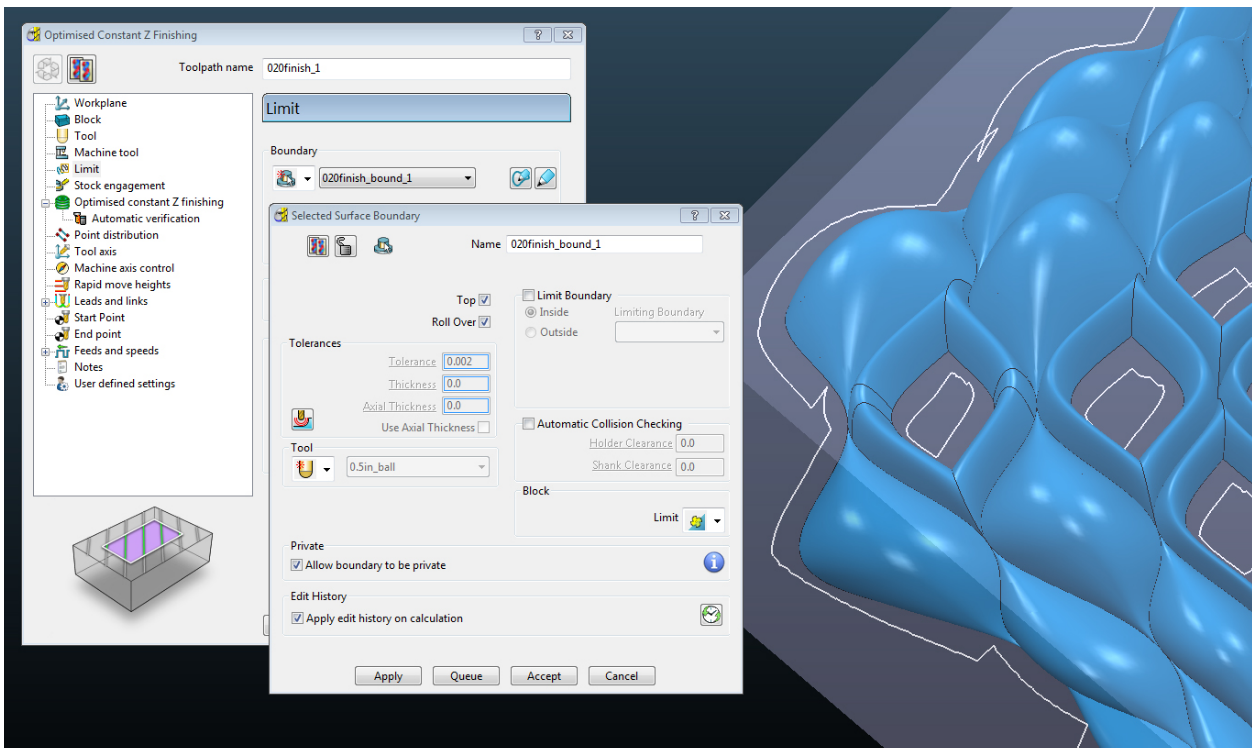
Let's clone it so we can compare the two toolpath results. In cloned toolpath **020finish_1**, go to **Optimized constant Z finishing** and check **Closed offsets** and **Smoothing**.

Go to **Lead In** and set **Max zig angle** to **14.0** and lower the **Height** to **0.15**. Uncheck **Always Ramp**.

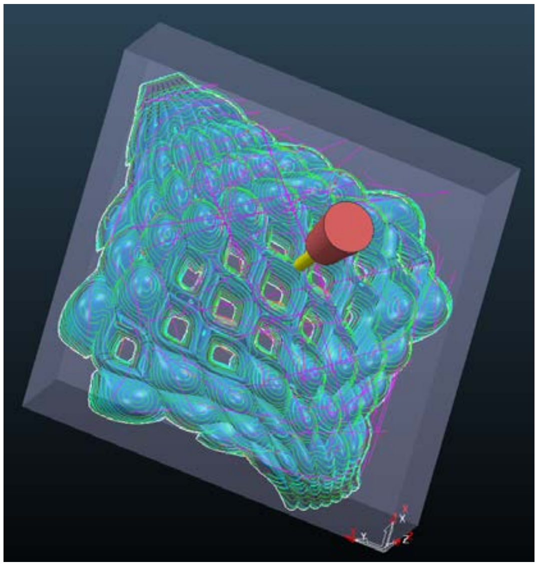
Limiting with Boundaries

Go to **Limit** and:

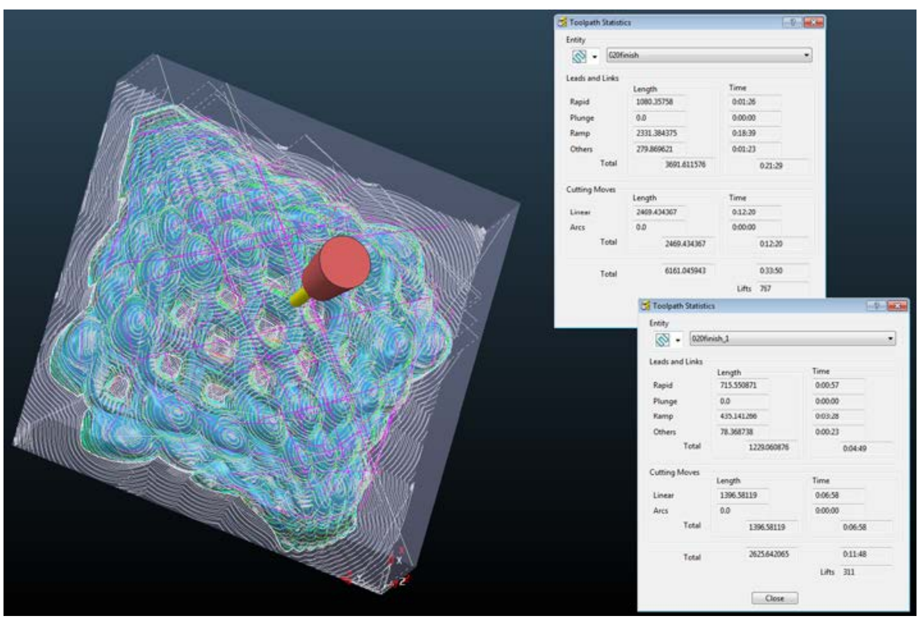
1. Select **Boundary** type **Selected Surfaces Boundary**.
2. Name it **020finish_bound** and check **Roll Over**.
3. **Tolerance** and **Thickness** should match your toolpath settings, **0.002** and **0.0**, respectively.
4. The tool, too, should match, and should be set to **0.5in_ball**.
5. Lastly, select the surfaces to which the toolpath will be bound by right-clicking on the Level **part** and selecting **Select All**.
6. Hit **Apply** to create the boundary.



Hit **Apply** within the toolpath form to recalculate the toolpath – the result should look much cleaner.



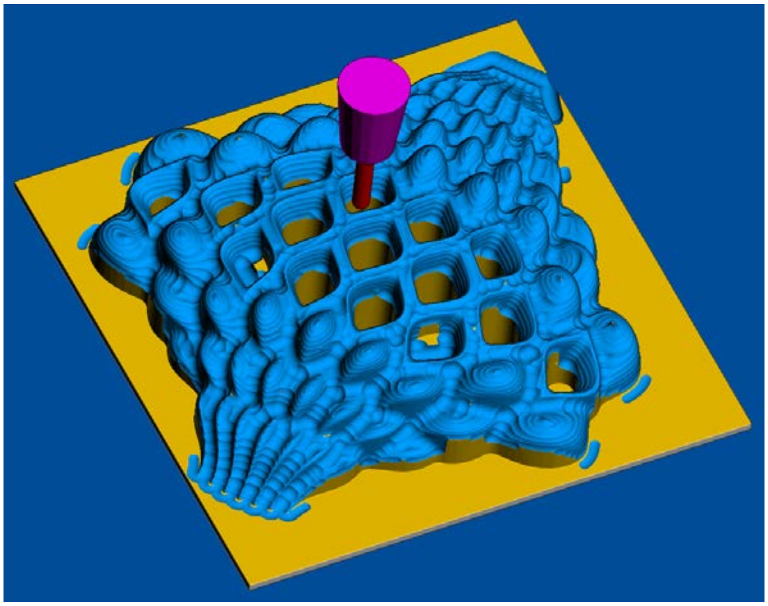
Compare results by toggling the **Light Bulb** (visibility) icon next to the prior toolpath name. You can also compare estimated machine time for each toolpath by right-clicking on the toolpath and selecting **Statistics**. We have almost half as many lifts and are saving over 20 minutes!



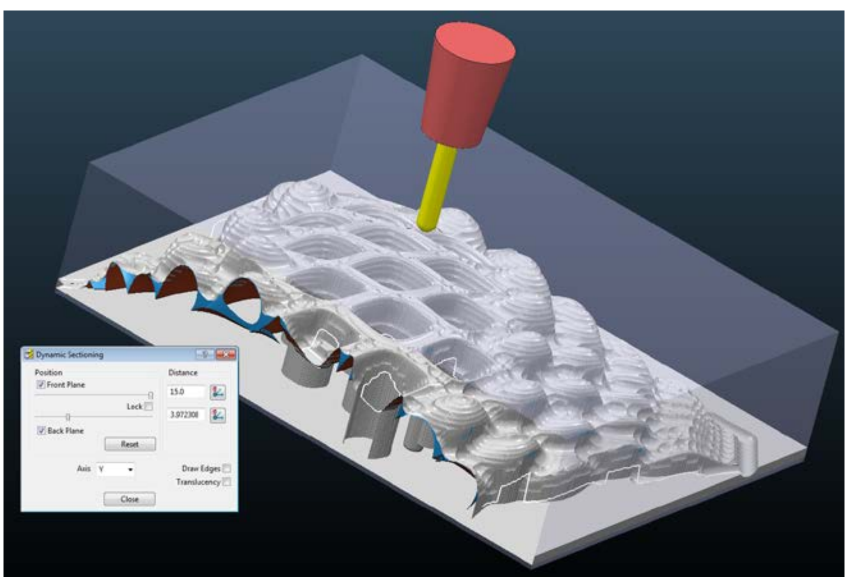
Note: A typical finish toolpath would have a much finer stepover value but in the interest of saving calculation and simulation time we are going with a larger value for this demonstration.

Stock Visualization and Analysis

Simulate **020finish_1** in **ViewMill** using the **Shade Cuts by Toolpath** option which uses coloration to distinguish which areas of the machined result were affected by which toolpaths.



You can get a better idea of remaining stock on the model using **Dynamic Sectioning**, available in the **View** menu. Constrain the **Axis** to **Y** and check the **Front** and/or **Back Planes** to take live section cuts through the part. If done while using the **ViewMill Dynamic Image** option you can rotate the model to closely inspect it.



The progress file **myProject05** is available as a course resource. **myProject06** uses a finer stepover and more accurately finishes the part – we will proceed with this latter file.

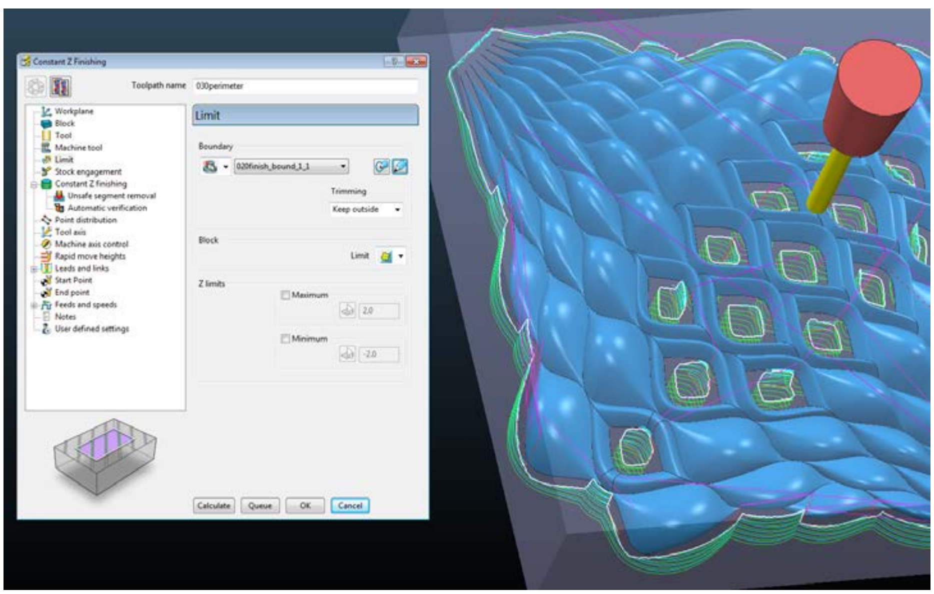
Detailing

Refer to **myProject07** in the course resources to review a detailing strategy consisting of the following toolpaths:

1. Perimeter vertical wall finishing (**Constant Z** toolpath)
2. Inside corner detailing (**Auto Corner** toolpath)
3. Inside corner center line cut (**Pencil** toolpath)

Constant Z Perimeter Finishing

In the case of our part, any naked edge above Z 0 will be considered a vertical wall, projected in plan from a 3 axis approach. Therefore, in order to finish the outer perimeter of the part as well as in the interior holes, we can utilize a **Constant Z Finishing** toolpath constrained by a **Selected Surface** boundary. The key difference to this approach is to modify the boundary such that **Trimming** is set to **Keep outside**, rather than the **Keep inside** option we used by default for the **Optimized Constant Z Finishing** toolpath.

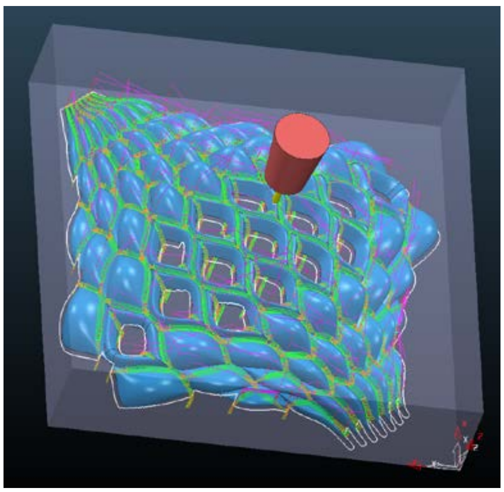
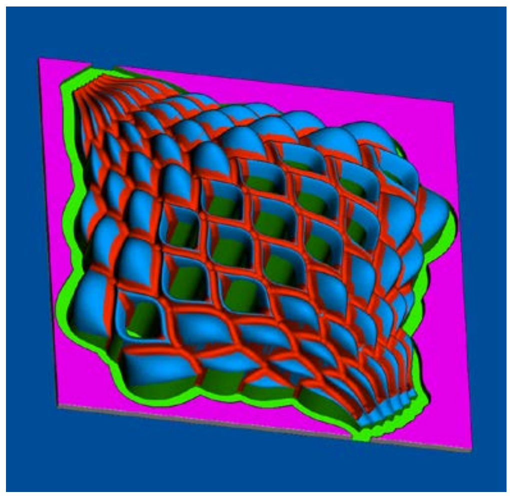


Auto Corner and Pencil Finishing

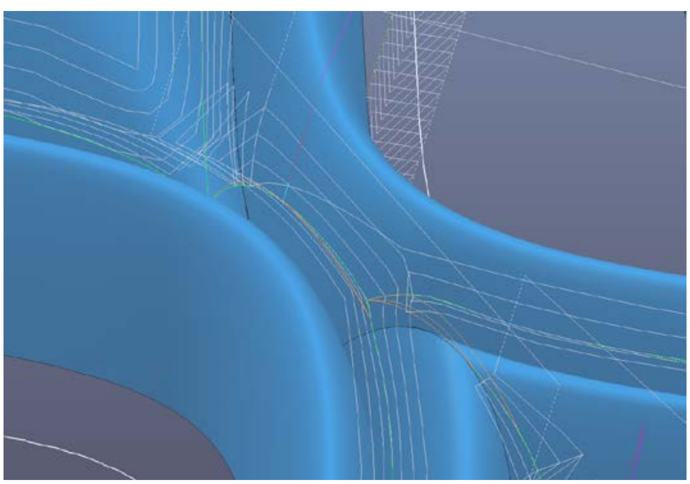
Our part has many curvature discontinuous concavities – this type of inside corner is not particularly well suited to the profile of a ball mill, nor easily detectible as minimum radius curvature due to the discretization of the model, but if we're willing to accept an interior fillet to the form we can still approach relative verisimilitude to the original surface model.

The **Auto Corner Finishing** toolpath limits the cutting of what is typically a smaller diameter tool to only those areas of the part which the prior toolpath's larger cutter could not reach without the need for specialized boundaries.

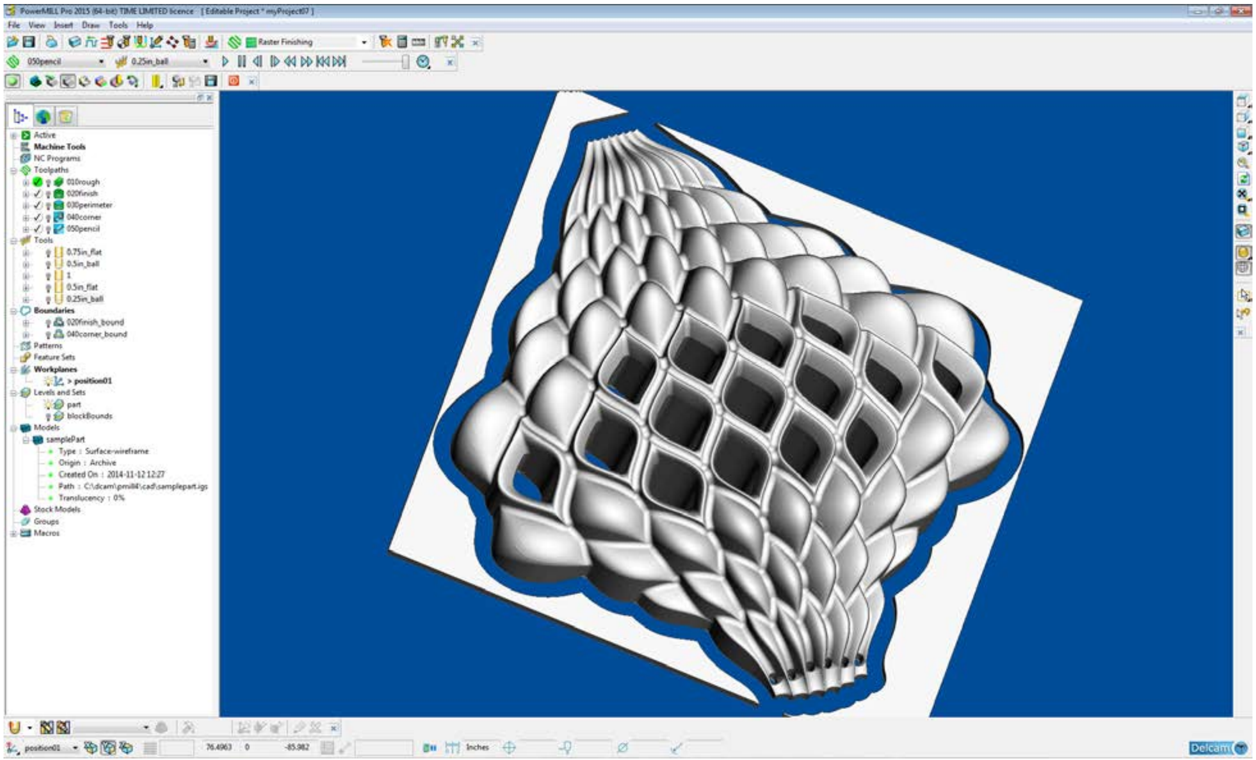
This is perhaps most easily visible using the **Shade Cuts by Toolpath** option in **ViewMill**.

The addition of the Pencil toolpath as the final toolpath in our detailing strategy is arguably unnecessary as the auto corner typically includes a pencil path, but depending upon the geometric complexity of the part there are moments where it may be necessary to include it. By viewing both toolpaths simultaneously we can see some of these moments.



And the result!



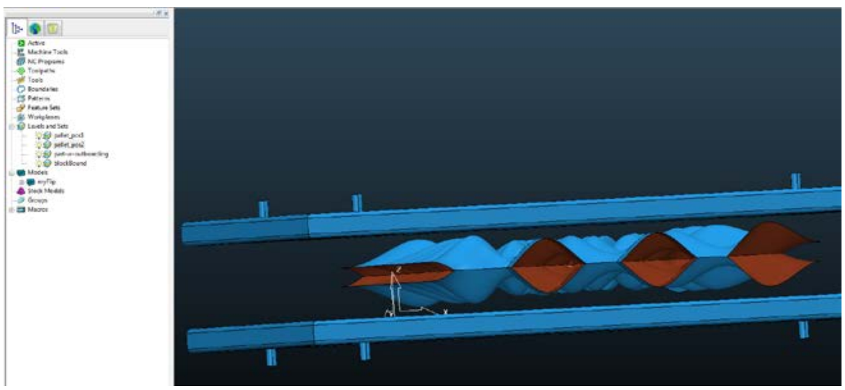
Additional Position Example: The "Flip" Mill

Set Up

Importing the Sample Model

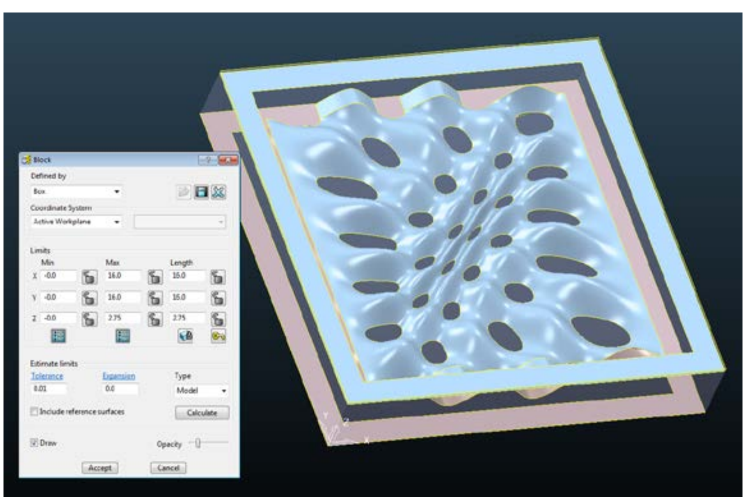
Import the sample model myFlip.igs from **C:\dcam\pmill4\cad.** It will import with 4 levels – rename them in the following order:

1. **pallet_pos1**
2. **pallet_pos2**
3. **part-w-outboarding**
4. **blockBound**



Sizing the Block to Selected Geometry

Turn off visibility for the **pallet_pos1** and **pallet_pos2** levels and open the **Block** form. Then window-pick the remaining geometry and then select **Calculate** to size the block.



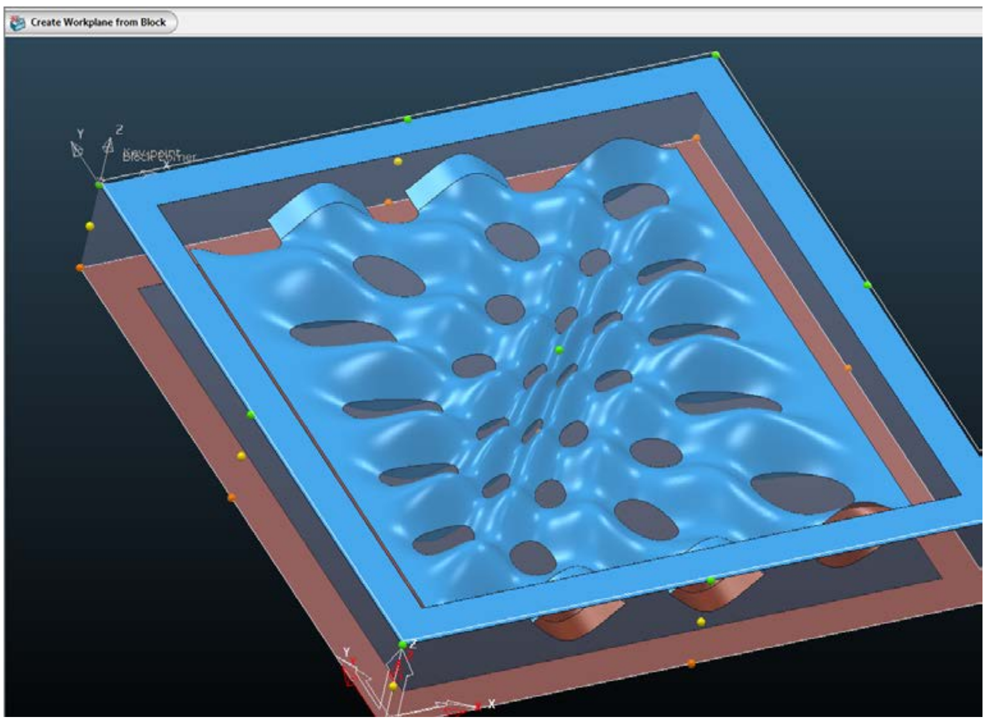
Setting Workplanes from the Block

To set the position 1 workplane, right-click on **Workplanes** in **PowerMILL Explorer** and select **Create Workplane…**, then hit **Accept** immediately to place a workplane at the default origin. Name this workplane **pos1**.

To set the position 2 workplane, we will create a workplane from the block. First we will have to adjust the block so that its top is defined as the top of the outboarding rather than the top of the true block – this is because we want to use a facing operation to ensure that we have a perfectly flat base for the flip.

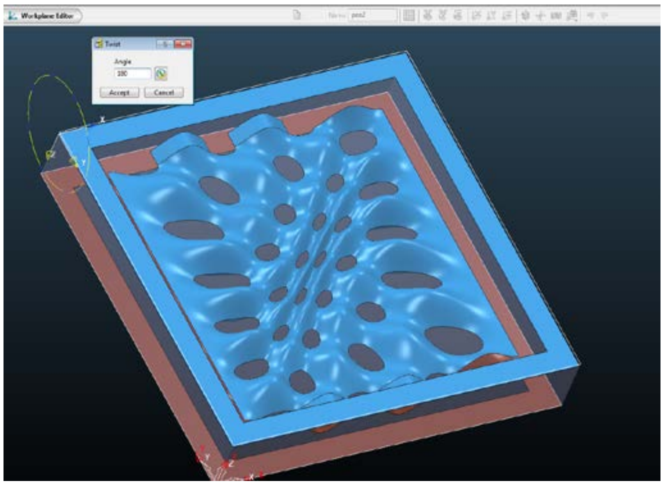
Window-pick all the visible geometry and then deselect the top curve on the **blockBound** level. Open the **Block** form and hit **Calculate**, then **Accept**. Right-click on **Workplanes** in **PowerMILL Explorer** and select **Create and Orient Workplane > Workplane Positioned Using Block…**

In order to set the position 2 workplane so that my part is flips about the X axis of a YZ plane centered on the left face of my block (post-facing), I'll select the top left green sphere.



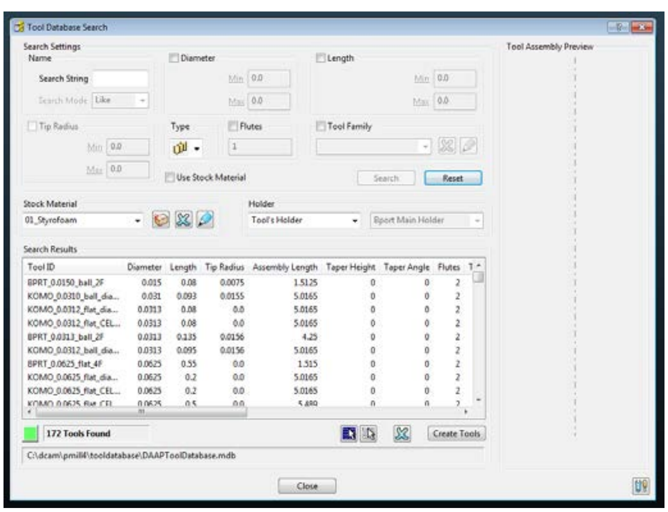
The resultant coordinate system is not yet correctly oriented. Rename the new workplane **pos2** and then right-click on its name and select **Workplane Editor…**

In **Workplane Editor** mode select the **Twist about X** button and enter **180.0** into the **Angle** field, then Accept. Accept the changes in the **Workplane Editor** to return to the model.



Importing Tools from a Database

Raise the **Tool Database Search** form by selecting the corresponding button in the lower left of the screen. No tools will be found initially as we have not set the correct path for our tool database file. Select the **Tool Database Options** button in the bottom right of the form and enter **C:\dcam\pmill4\tooldatabase\DAAPToolDatabase.mdb**, then **Accept**. Now if you hit Search with no filters, 172 tools (the entire library) will appear.



Note: This sample database was generously shared by the University of Cincinnati DAAP Rapid Prototyping Center – special thanks to Nick Germann for the creation of this file.

Now let's limit our search:

1. Enter **KOMO** into the **Search String** to limit the results to tools for the **Komo** router.
2. Constrain the **Type** to **ball nosed tools**.
3. Set **Diameter Min** to **0.25** and **Diameter Max** to **0.5**.
4. Hit **Search**.

Now the search results have been limited appropriately, and we can select the following tools to load into our project:

1. **V2235**
2. **V2265**

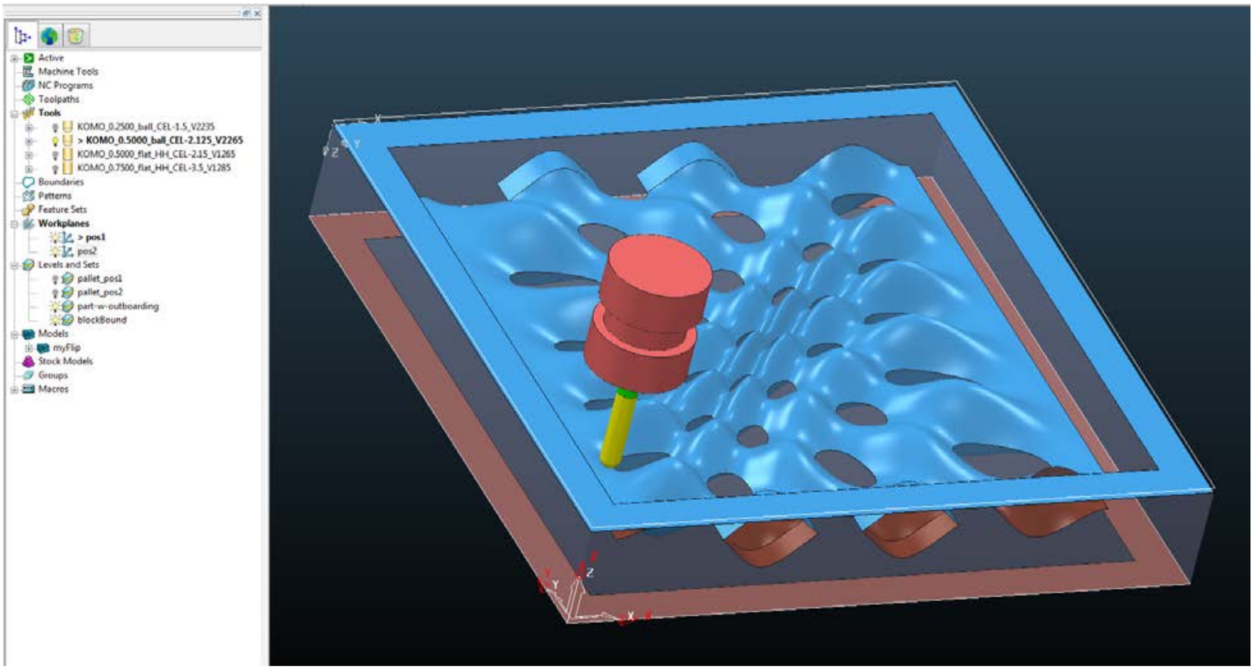
Hit **Create Tools** to load the tools into the project. Now change the search:

1. Change **Type** to **end mills**.
2. Increase **Max Diameter** to **0.75**.
3. Decrease **Min Diameter** to **0.5.**

And load the following end mills:

1. **V1265**
2. **V1285**

Now our tools are loaded and ready for use. The progress file **myFlip01** is available as a course resource.



Flip Mill Position 1 Toolpathing Tips

General Tips

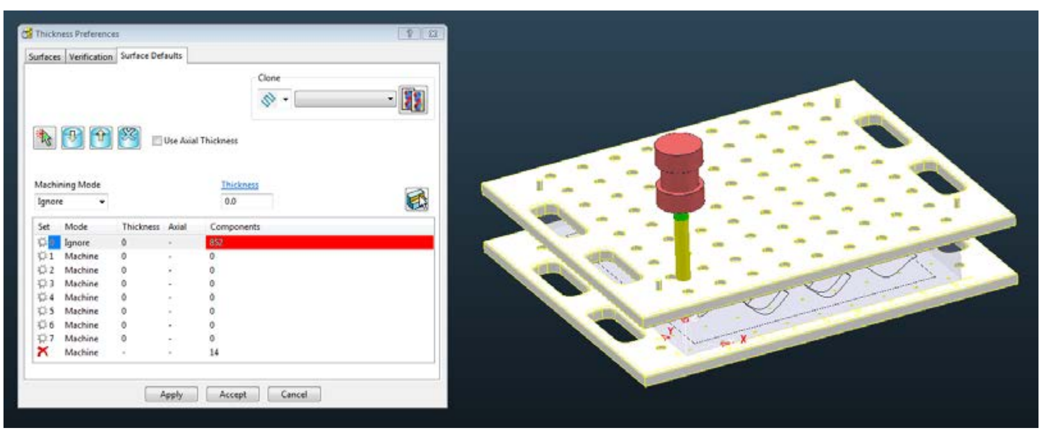
When using multiple workplanes it's very important to always be sure that the correct workplane has been activated prior to generating a new toolpath – this way the toolpath form will import the proper workplane by default.

It's also recommended to sort your **PowerMILL Explorer** assets, such as toolpaths, in separate folders based on the machining position by right-clicking on an **Explorer** item such as **Toolpaths** and selecting **Create Folder**.

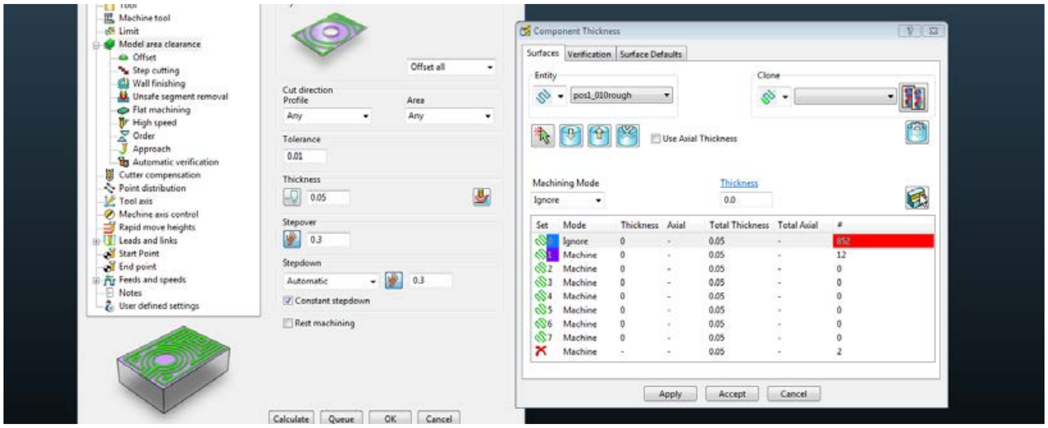
Component Thickness

You'll notice if you try to calculate a toolpath nothing will happen. This is because our pallets, though hidden, are still being calculated and are blocking the tool's ability to reach within the block. You can assign machining behaviors to specific geometry groups by selecting the **Default Thicknesses** icon in the upper toolbar.

With the **Thickness Preferences** form open to the **Surface Defaults** tab, turn on the visibility of the pallet levels and select the pallet geometry. Set the **Machining Mode** to **Ignore** and then click the **Acquire Selected Components** button in the form. Hit **Accept** and you'll notice the pallet geometry changes color to indicate that it will be ignored by subsequent machining operations. Turn pallet visibility back off and recalculate the toolpath



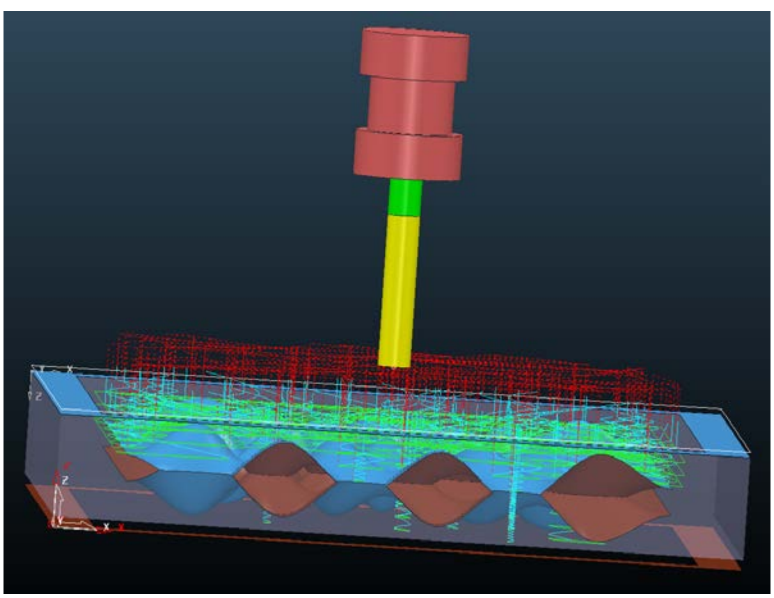
When calculating your first toolpath, you will need to select the **Component Thickness** button within the toolpath form and select the **Copy Thickness Data from Default** button in the **Surfaces** tab. This change should propagate to ensuing toolpaths.



Position 1 Roughing Tips

While some would only Rough only to the part line, I would suggest roughing as much as possible form the first position (so long as there are not tool reach issues) and then rough what remains on the flip, while still maintaining a block **Z Min Limit** of **0.25**.

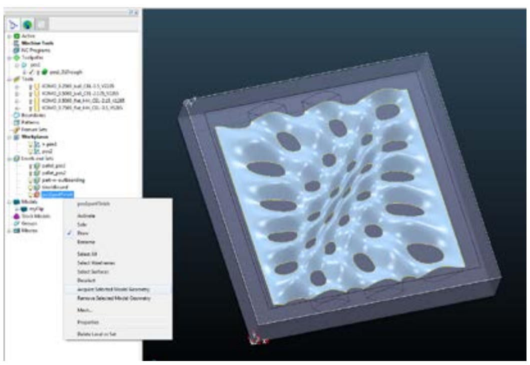
Lower the block **Z Max Limit** to prevent unnecessary roughing of the outboarding’s top face as we will get this with a facing operation later.

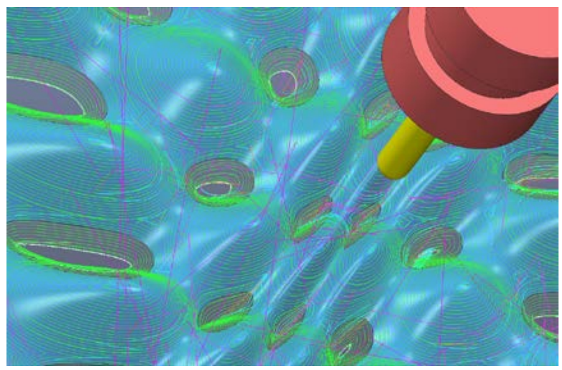


Position 1 Finishing Tips

When created a **Selected Surfaces** boundary for the finish, we only want to select those surfaces that are approachable from the current position. Anytime geometry needs to be further organized apart from levels, you can create a **Set** by right-clicking **Levels and Sets** in **PowerMILL Explorer** and selecting **Create Set**.

Create a set and call it **pos1partFinish**. Select the upper surface of the part, then right-click on the set name **pos1partFinish** and select **Acquire Selected Model Geometry** to acquire the geometry into the set.

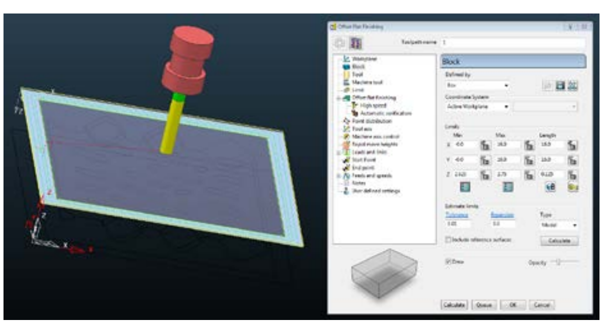




Indexing the Flip

Facing

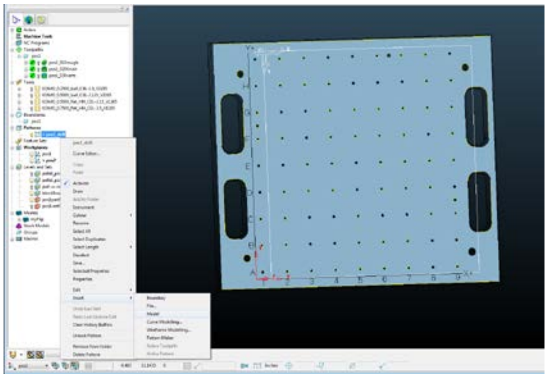
We can face the top of the outboarding (so that position 2 rests on a perfectly flat, machine-calibrated face) using the **Offset Flat Finishing** toolpath and constraining the toolpath by calculating the block with only the top outboarding surface and top block boundary curve selected – this way we avoid the need to create boundaries.



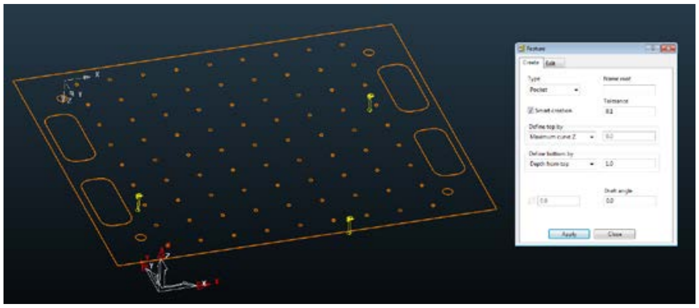
Patterns and Feature Sets

The imported pallets have pin holes with which we can use to index the flip. To do this properly, we will need to drill the corresponding pin holes into the part:

1. Isolate the visibility of the position 2 pallet and create a new pattern by right-clicking **Pattern** and selecting **Create Pattern**. Rename the pattern to **pos1_drill**.
2. Select the top face of the pallet and right-click on the **pos1_drill** pattern, then select **Insert > Model** to duplicate the naked edges of the surface.



3. Create a new feature set by right-clicking on **Feature Sets** in **PowerMILL Explorer** and selecting **Create Feature Set**. Rename the feature set **pos1_drill**.
4. Select the three 0.125 inch radius indexing pin holes in the pattern and right-click the feature set **pos1_drill** again, selecting **Create Features**.
5. In the **Feature** form:
    a. Set **Type** to **Pocket**.
    b. **Define top by Maximum curve Z**.
    c. **Define bottom by Depth from top** and enter **1.0** into the number field.



Now you have a feature set to specify drill location and depth.

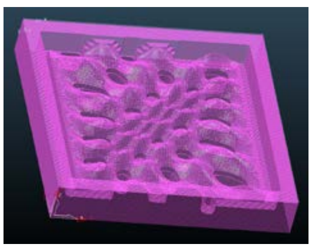
Note: Drilling not covered in this course. Maybe next year!

Flip Mill Position 2 Toolpathing Tips

Stock Models

To only rough what remains in position 2 we will need to save the resulting stock model from position 1:

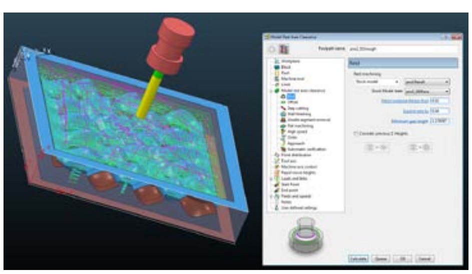
1. Start by right-clicking on **Stock Models** in **PowerMILL Explorer** and selecting **Create Stock Model**, then rename it **pos1Result**.
2. Open the **Block** form and set the dimensions to correctly reflect the actual block size, then right-click on stock model **pos1Result** and select **Apply** > **Block**.
3. Activate each toolpath in their proper order and apply them to the stock by right-clicking on stock model **pos1Result** and selecting **Apply > Active Toolpath Last**.
4. Right-click again and select **Calculate** to generate the stock model.



Roughing from a Stock Model

To calculate the position 2 rough from the stock model, activate the pos2 workplane and create a **Model Rest Area Clearance** toolpath. Select **Rest** within the toolpath form and:

1. Set the **Rest machining** drop-down menu to **Stock model**.
2. Set the stock model drop-down menu to **pos1Result**.
3. Make sure that **Stock Model state** is set to the latest state, the **pos1_040face** toolpath.
4. **Calculate** the toolpath and **Accept**.



*Note: From here on out you can complete the toolpathing normally! The progress file **myFlip02** is available as a course resource.*

Writing an NC Program

Position 1 NC Program

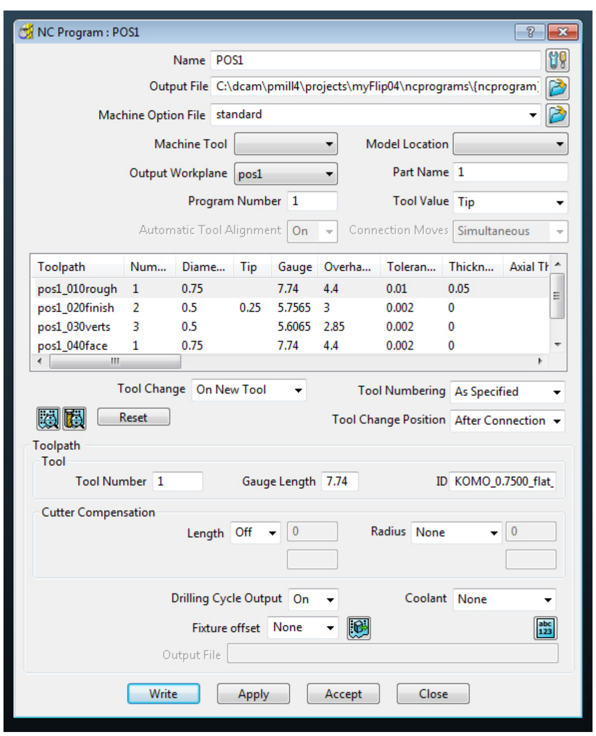
Right-click **NC Programs** in **PowerMILL Explorer** and select **Create NC Program**. In the **NC Program** form:

1. Set **Name** to **POS1**.
2. Maintain the default **Output File** path, which will store your NC programs within their corresponding project folders.
3. **Apply** and **Accept**.

Select all toolpaths within the Toolpath **pos1** folder and drag them into the **POS1** NC Program. Make sure that they are in the right order.

Right-click on the **POS1** NC Program and select **Settings**. Set **Output Workplane** to **pos1**. **Apply** and **Write** the program.

The program will fail to write! You will need a post-processor configured for your machine for the graphical data of PowerMILL to be translated properly into machine code.



The progress file **myFlip03** is available as a course resource.