V2 Series - Desk*Proto* Rotary Tutorial

V2 Series - DeskProto Rotary Tutorial
- Introduction
- Supplies Needed
  - Section 1: Selecting a geometry file (.stl file)
  - Section 2: Loading a geometry file in DeskProto
  - Section 3: Configure DeskProto to use a Pocket NC
  - Section 4: Configuring your tool settings in DeskProto
  - Section 5: Save your DeskProto project
  - Section 6: Setup DeskProto to calculate rotary tool paths
  - Section 7: Scale the part to fit within the boundaries of the stock
  - Section 8: Orient the rotary tool operations around the Pocket NC Y axis
  - Section 9: Define the stock material dimensions
  - Section 10: Set the work coordinate system origin
  - Section 11: Defining the Roughing Rotary Tool Operation
  - Section 12: Defining the Finishing Rotary Tool Operation
  - Section 13: Mounting the stock to the Pocket NC V2 and measuring the G55 work offsets.
  - Section 14: Modifying the G-Code using the Penta Machine Company Simulator to utilize TCPC.
  - Section 15: Run the G-Code Program on the Pocket NC
  - Section 16: Rotary milling process on Pocket NC V2-10
- Conclusion

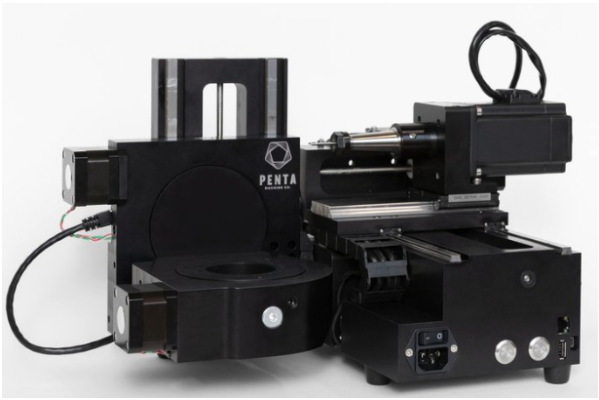
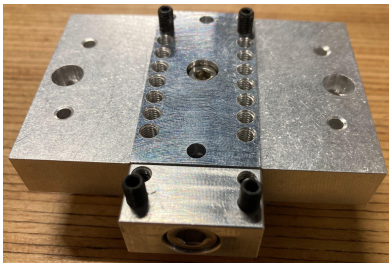
Introduction

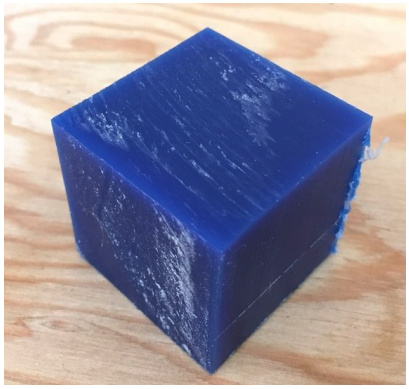
The purpose of this tutorial is to demonstrate how to program rotary toolpaths in Desk*Proto*, and how to set up and cut a 4 axis part on your Pocket NC V2-10 or V2-50. **This tutorial uses the Kinetic Control user interface released on 11/1/2021**. The part and tutorial are geared toward beginners. More experienced users might also find value from the tips provided throughout this tutorial. The tutorial walks through how to correctly set up a rotary roughing and finishing toolpaths. The finish quality of the final part can be improved by reducing the distance between toolpaths (stepover) and the step size along toothpath settings (smoothing). The tutorial

optimizes the part being milled to demonstrate functionality while reducing the amount of time to cut the part.

> ⚠️ This tutorial requires that you have the Multi-Axis Edition of Desk*Proto*.

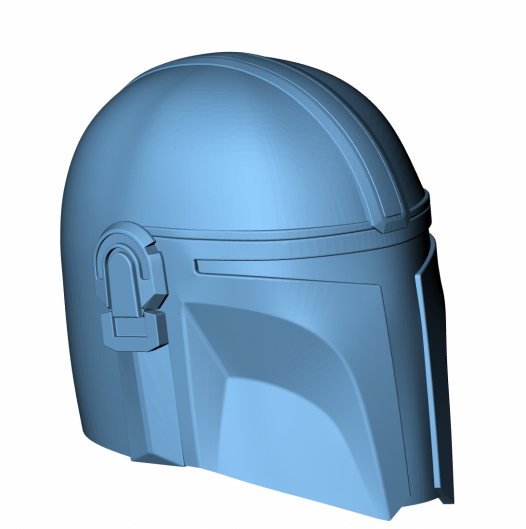
Supplies Needed

Pocket NC (V2-10, V2-50CHB, or V2-50CHK)	
V2-10 Extended Toolholder with ⅛ inch collet (If using a Pocket NC V2-10)	
Harvey Tool 1/8th Ball Long Reach, Stub Flute - part number: 96108	© Harvey Tool Company, LLC. D1 | 0.125(1/8) D2 | 0.125 L1 | 2.5 L2 | 0.187 L3 | 1.25
Pocket NC Custom Vise (plus adapter)	

Wax Stock - (52mmx52mmx52mm)	

Section 1: Selecting a geometry file (*.stl file*)

An *stl* file was selected from the Ultimaker Thingiverse website. Click here to download a copy of a Mandalorian Helmet courtesy of DamaskProps. For this tutorial, we will use the *MandoV2FullBucket.stl* and *MandoV2Visor.stl* files.



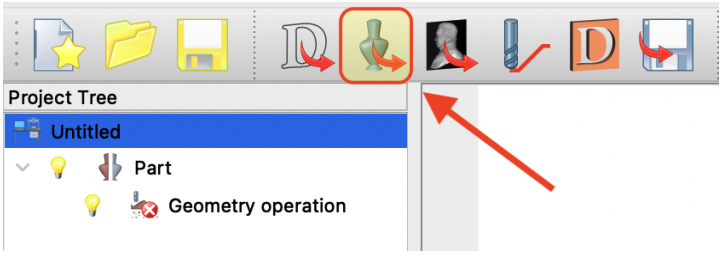
Section 2: Loading a geometry file in DeskProto

Start Desk*Proto*. This tutorial requires the **Multi-Axis Edition**.

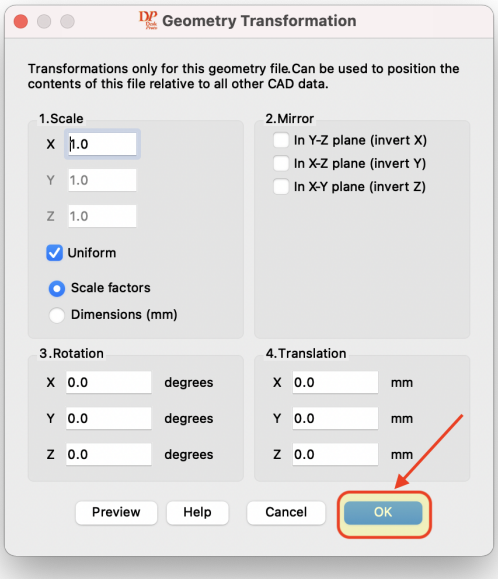
Desk*Proto* has a **Start new project** wizard option. For the purpose of this tutorial, the setup will be performed manually. Click the red X button to close out the following window.



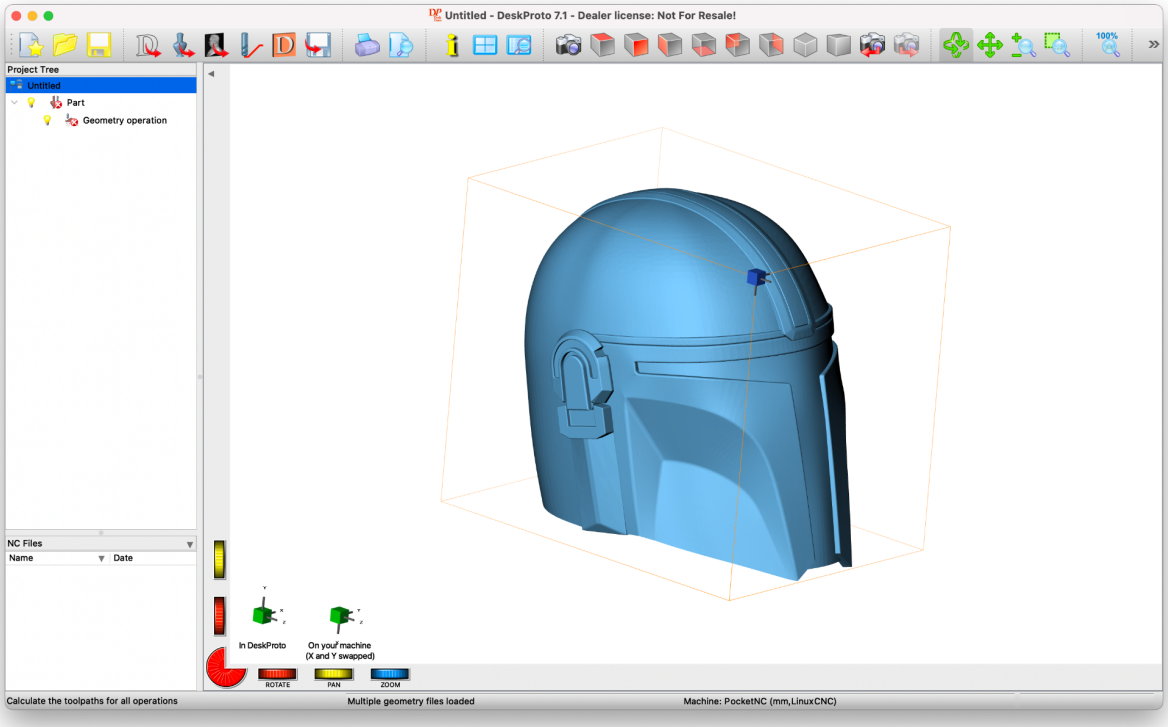
Click on the Load Geometry File tab on the Desk*Proto* tool pane. Follow the file open prompts and load the *MandoV2FullBucket.stl* and *MandoV2Visor.stl* files.



When the *MandoV2Visor.stl* file is loaded, the following window will display. No transformations are needed so click the **OK** button to finish.

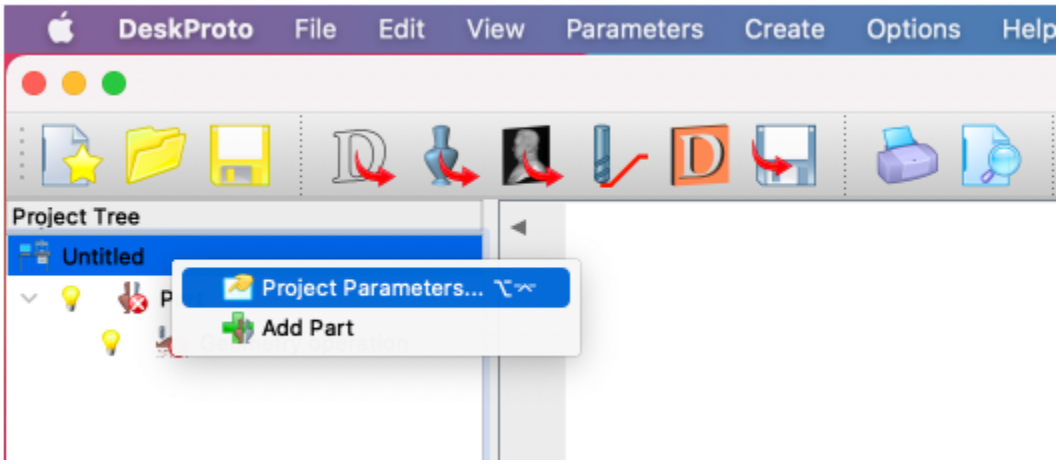


⚠️ View was rotated to display helmet part as shown.

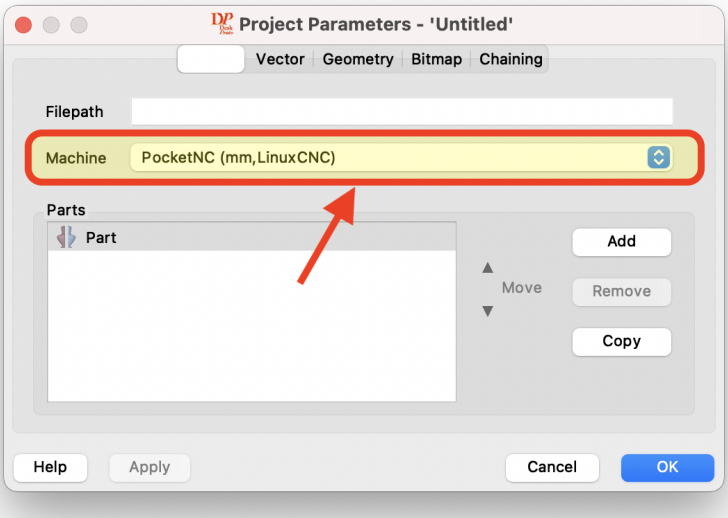


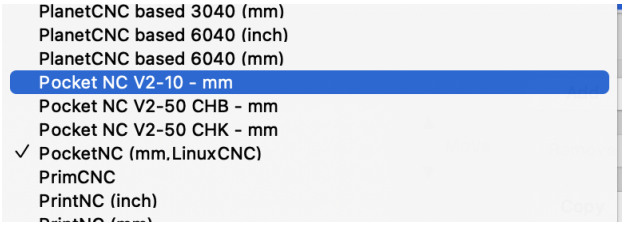
Section 3: Configure Desk*Proto* to use a Pocket NC

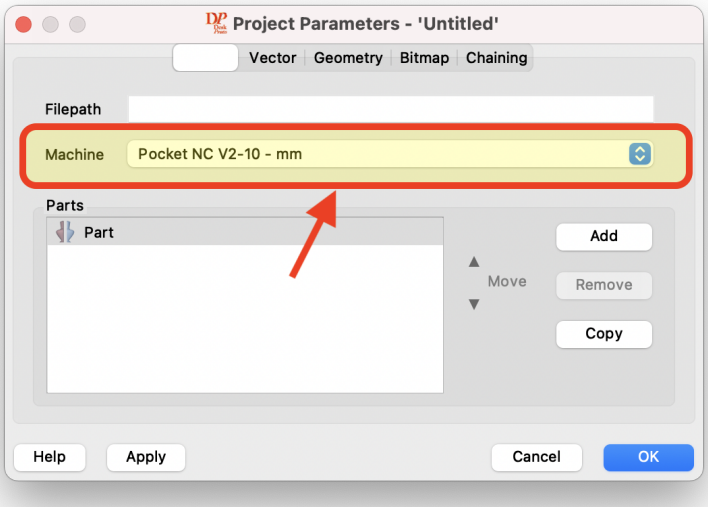
Edit the **Project Parameters**.



Select the ***Machine*** drop down menu and depending on your Pocket NC model, select the ***Pocket NC V2-10 - mm, Pocket NC V2-50 CHB - mm, or Pocket NC V2-50 CHK - mm*** from the **Machines** list. (For the purpose of this tutorial, the ***Pocket NC V2-10 - mm*** machine definition will be used).





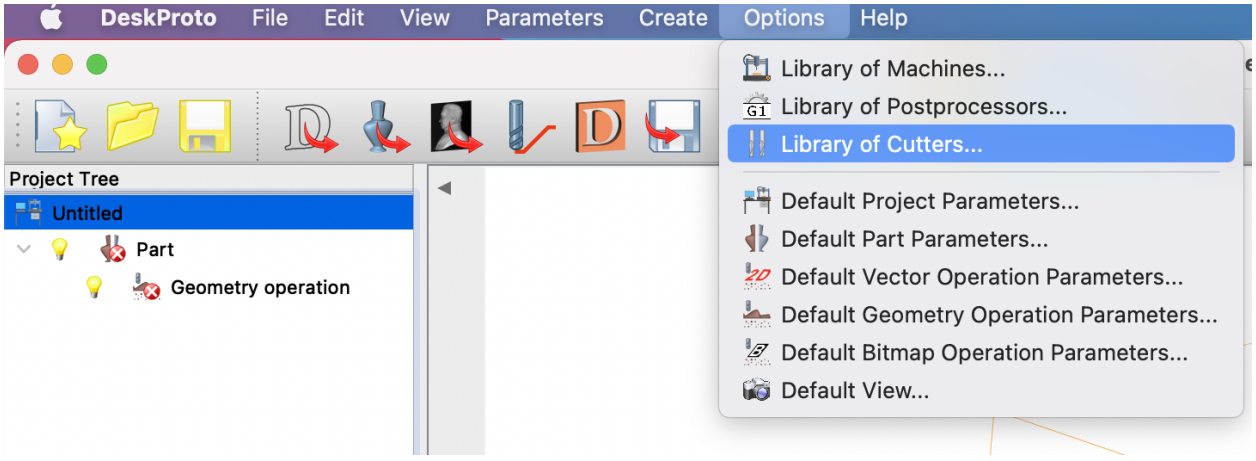


Click the **OK** button and return to the Desk*Proto* main screen.

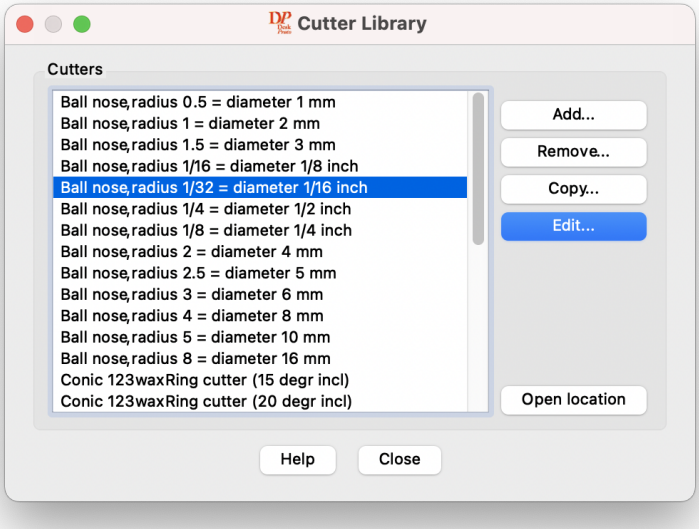
Section 4: Configuring your tool settings in Desk*Proto*

This tutorial uses a 1/8 ball nose cutter. Desk*Proto* has a library of cutters (tool library) where these tools can be set up.

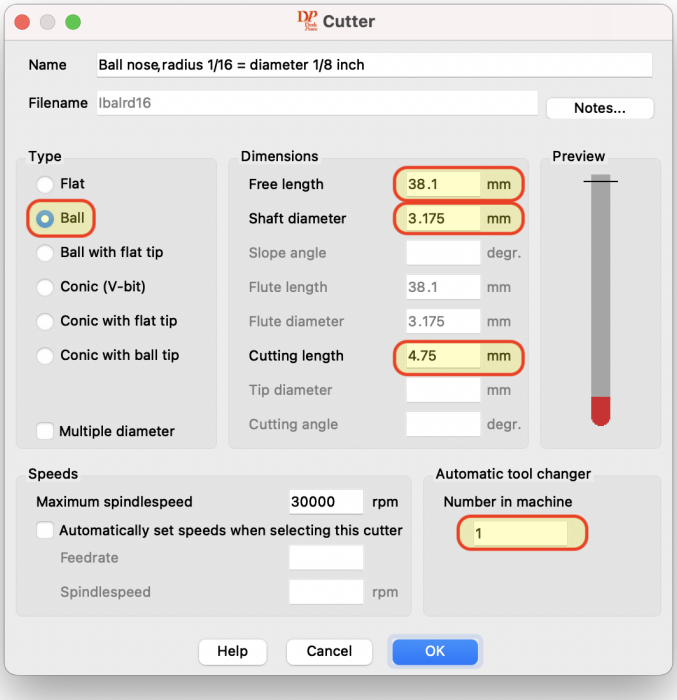
Click on the **Options** tab and select ***Library of Cutters…***.



Select the ***Ball nose,radius 1/16 = diameter 1/8 inch*** tool from the **Cutters** list and then select the **Edit** button.



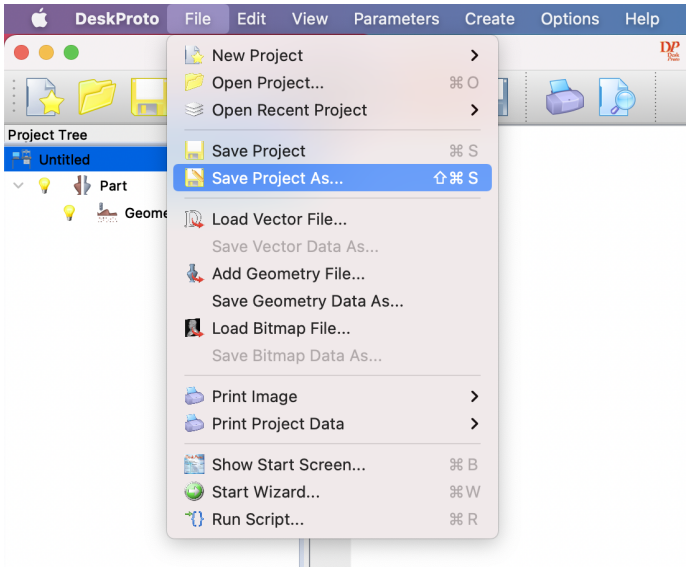
Program the tool parameters as shown below. Make sure that the ***Number in machine*** is set to **1** under the **Automatic tool changer**.



Click the **OK** button until back at the **Cutter Library** screen and then click the **Close** button.

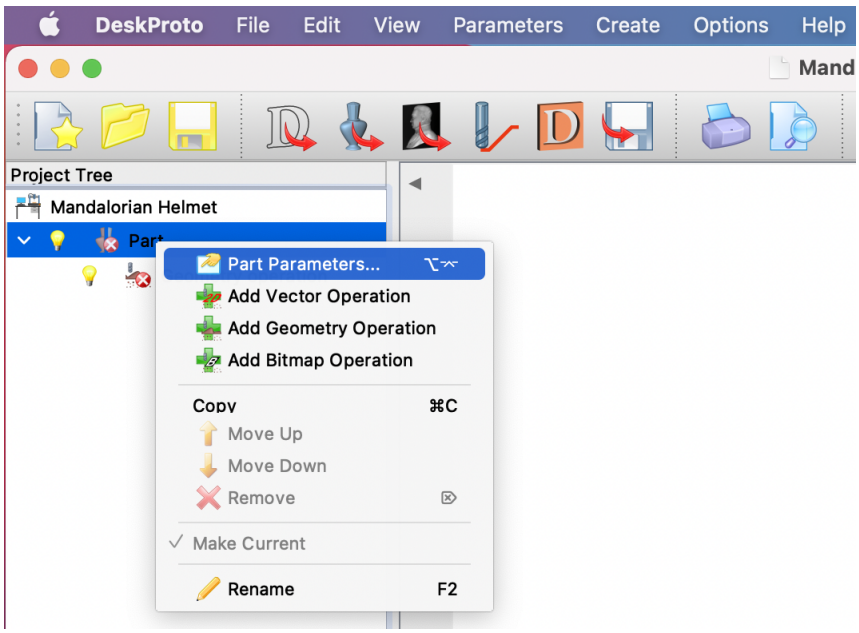
Section 5: Save your Desk*Proto* project

Now is a good time to save your Desk*Proto* project. Select the **File** tab and choose the ***Save Project As…*** We are going to save this project as ***Mandalorian Helmet.dpj***.

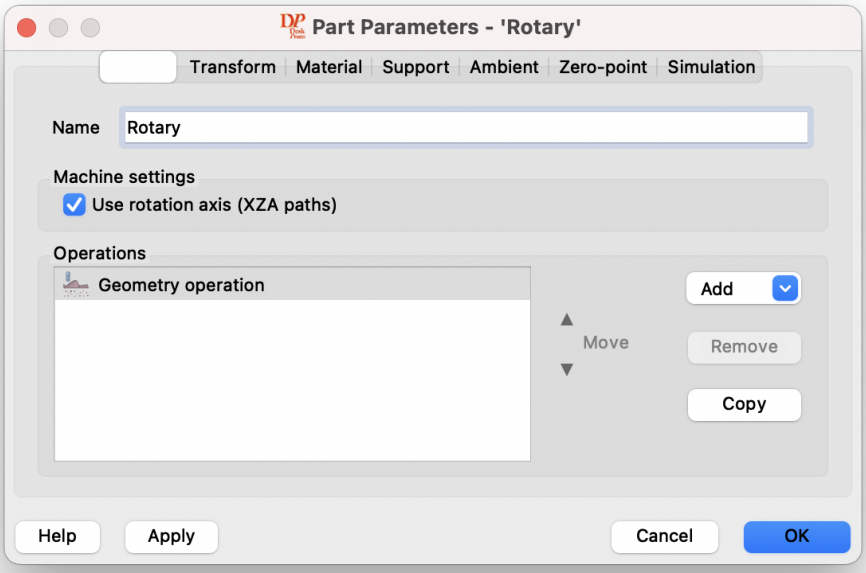


Section 6: Setup Desk*Proto* to calculate rotary tool paths

Right click on **Part** under the **Project Tree** and select **Part Parameters…**

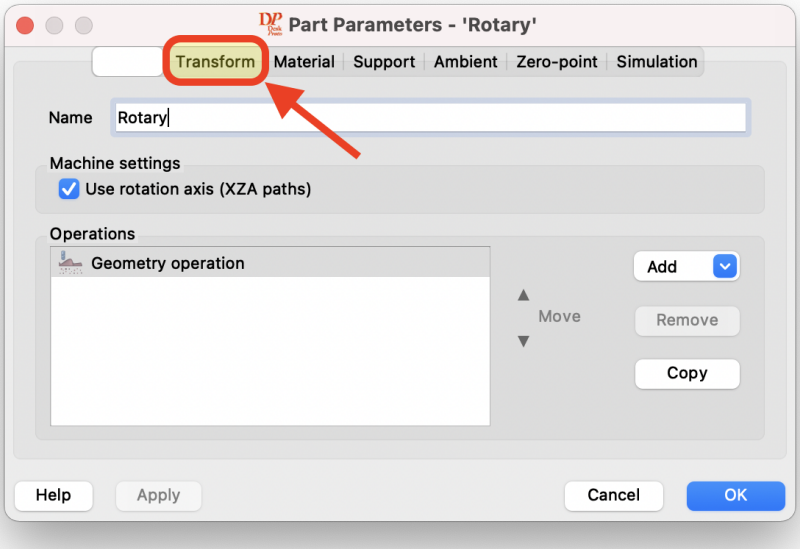


Rename the **Part** to **Rotary**. Select ***Use rotation axis (XZA paths)***. Desk*Proto* is now configured to calculate rotary toolpaths.

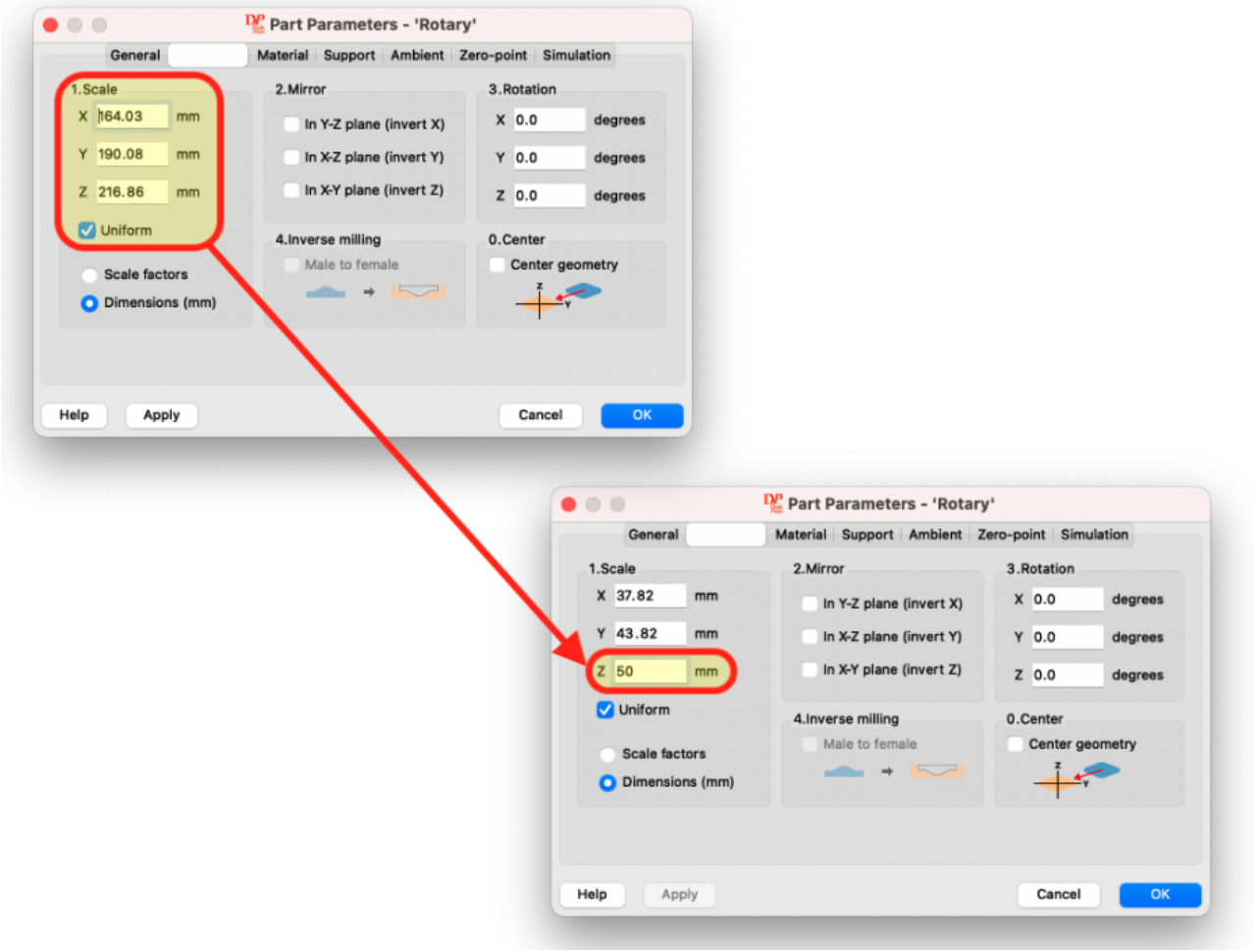


Section 7: Scale the part to fit within the boundaries of the stock

While you are still editing the **Part Parameters…**, select the **Transform** tab.



Under **Scale**, select **Dimensions** to review the size of the original part. Make sure the **Uniform** checkbox remains checked. In the case of the Mandalorian Helmet part, the Z dimension of the part is the larger value. To maximize the size of the finished part, the Z value can safely be adjusted to 50 mm, which is 2 mm smaller than the stock dimension. Click the **Apply** button to save the setting.

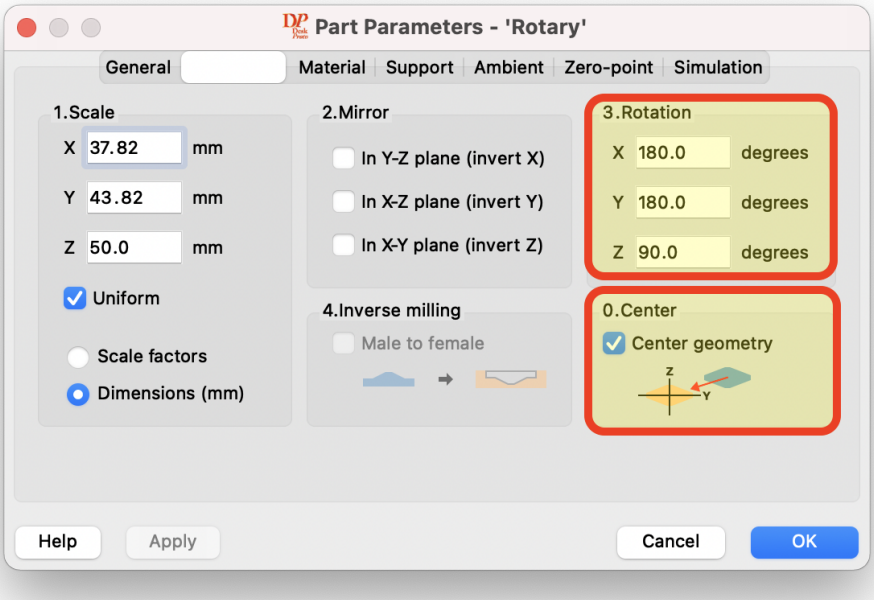


Section 8: Orient the rotary tool operations around the Pocket NC Y axis

While you are still editing the **Transform** tab, make sure to select the checkbox **Center geometry**. For the purpose of this tutorial, the Pocket NC custom vise will be used to secure the stock to the Pocket NC.

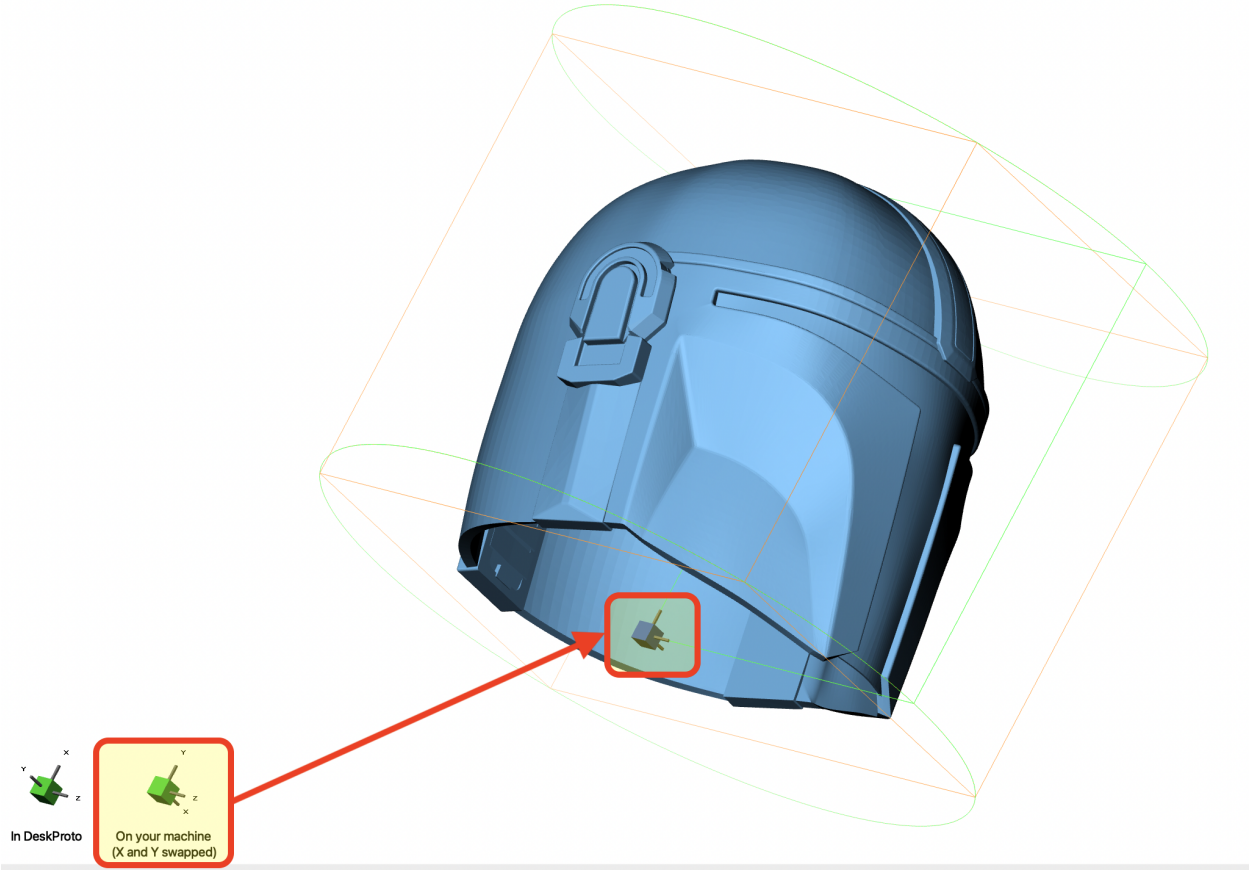
> ⚠️ Selecting **Center Geometry** configures Desk*Proto* to calculate toolpaths around the exact center of the part.

The next step is to orient the tool (Z axis for the Pocket NC) to be perpendicular to the axis of rotation (Y axis for the Pocket NC). This can be accomplished by adjusting the **Rotation** values under the **Part Parameters Transform** section as shown.



For this tutorial, change the ***Rotation*** to ***X=180***, ***Y=180***, ***Z=90***. Click the **Apply** button. Notice that the area to be machined rotates around the Pocket NC Y axis and the tool approaches the part from the correct orientation.

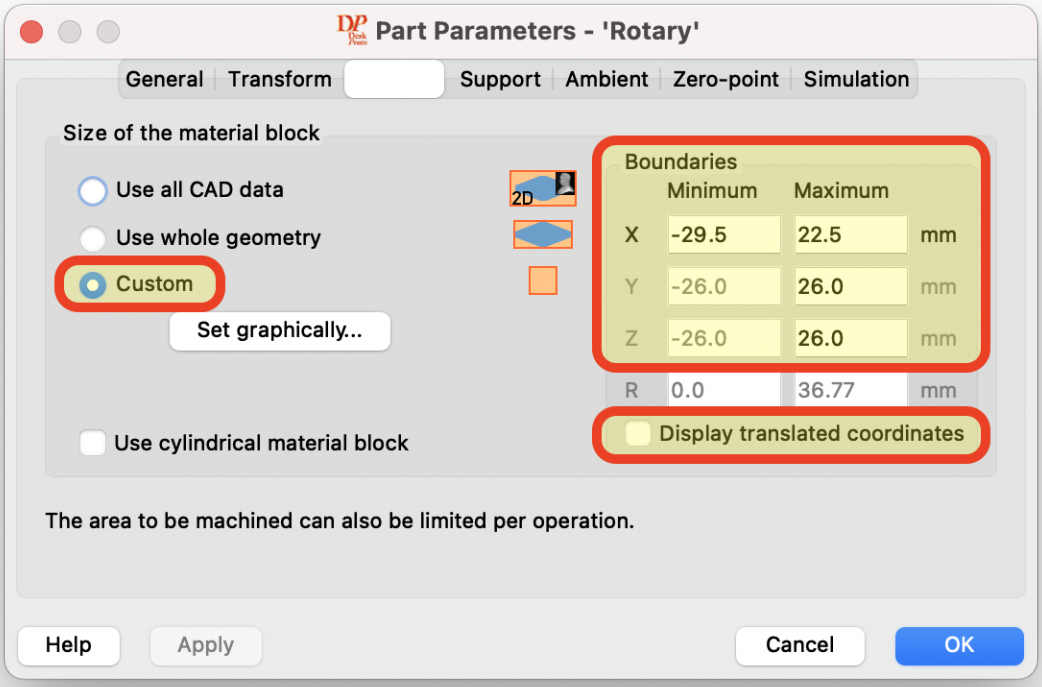
AFTER ROTATION VALUES APPLIED



Section 9: Define the stock material dimensions

The stock material used for this tutorial is approximately 52 mm per side. Select the **Material** tab under **Part Parameters**. Under **Size of the material block** choose ***Custom***. Under the **Boundaries** section, deselect the ***Display translated coordinates*** and adjust the **Boundaries** values as shown. The ***X* minimum** and **maximum** values were selected to position the part close to the top of the stock, allowing for the maximum clearance of the tool holder when machining at the bottom of the part.

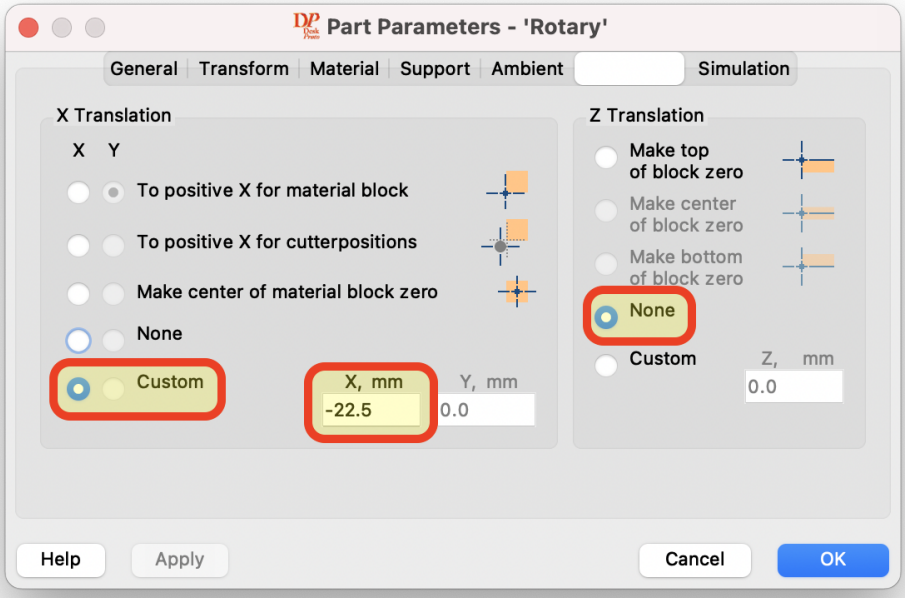
⚠️ The ***X***, ***Y***, and ***Z* Boundary** values relate to the Desk*Proto* CAM coordinates.

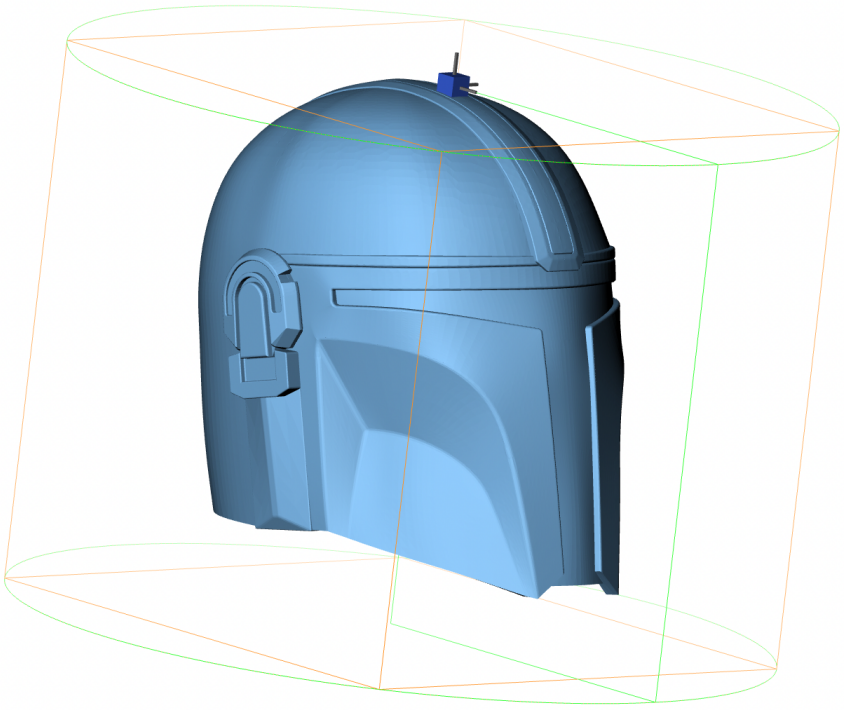


Section 10: Set the work coordinate system origin

The final step is to select where on the stock to place the work coordinate system zero point. In this tutorial, since we have chosen to mount the stock with the Pocket NC Custom Vise, the most convenient location is the top-center of the stock. This choice of zero-point will become obvious in the next section.

Select the **Zero-point** tab and set the **X Translation** to ***Custom*** and set the ***X*** value to -22.5 mm, and the **Z Translation** to ***None***. The **Y Translation** cannot be edited.







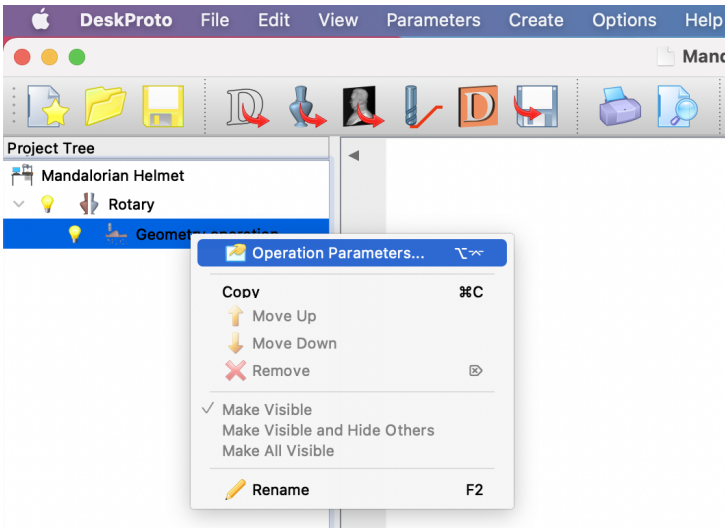
In DeskProto



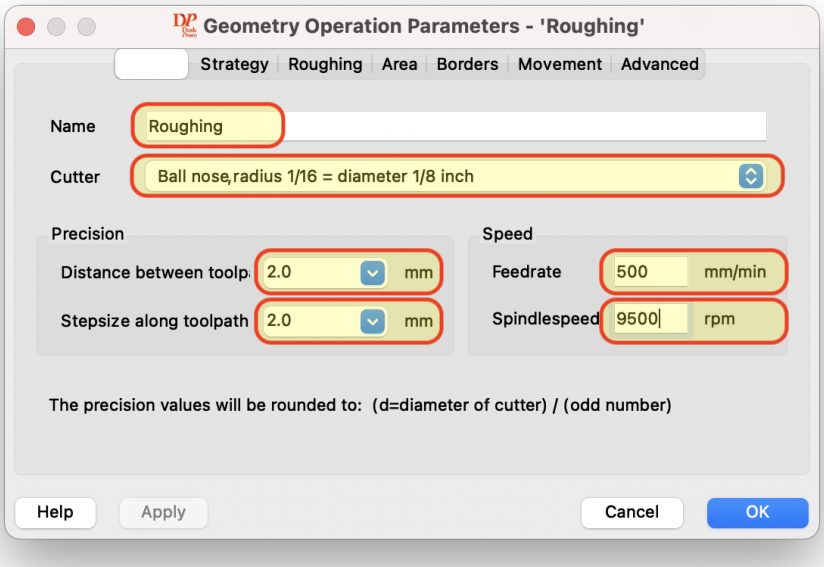
On your machine
(X and Y swapped)

Section 11: Defining the Roughing Rotary Tool Operation

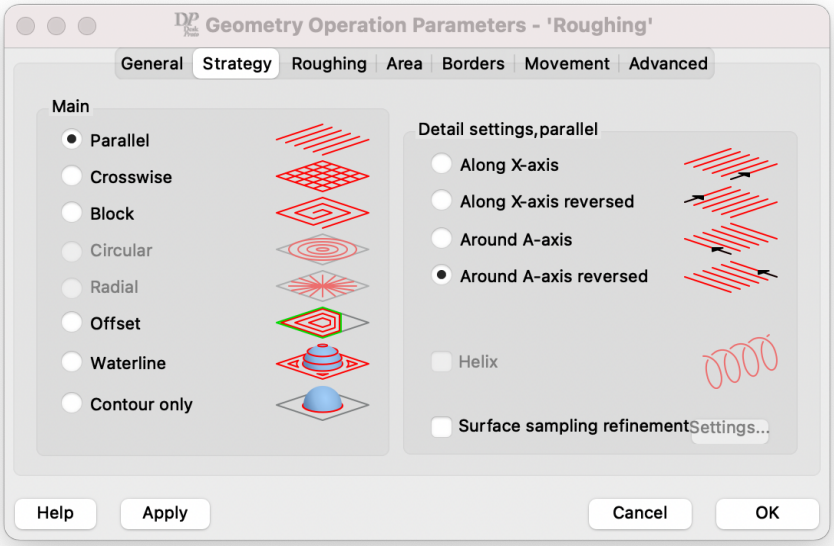
Edit the **Geometry Operation Parameters**.



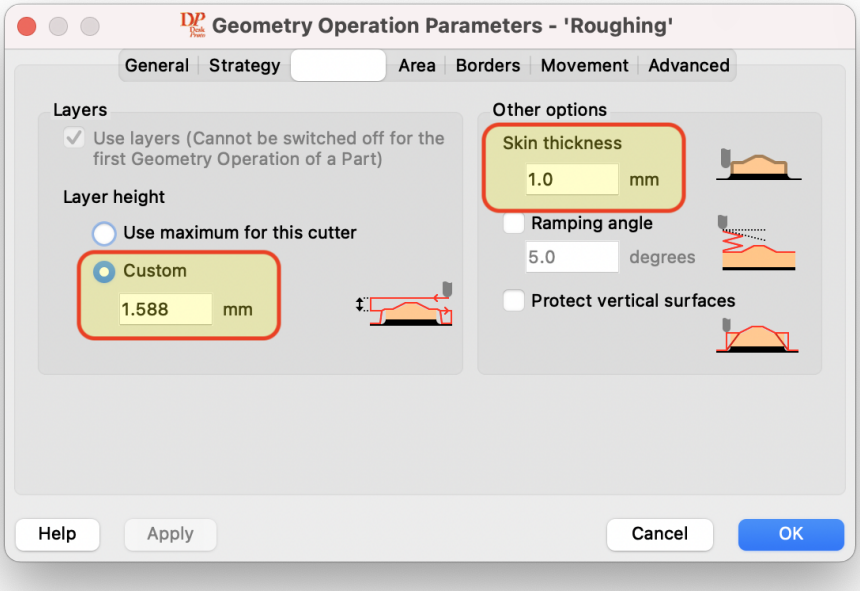
Rename the operation ***Roughing***. Select the ***Cutter*** (tool) to be at ***Ball nose, radius 1/16 = diameter 1/8 inch***. Under the **Precision** section, select both the ***Distance between toolpath*** and ***Stepsize along toolpath*** to be **2.0 mm**. These are slightly conservative values for the selected tool when machining wax. Keep the default ***Feedrate*** at 500 mm/min. A ***Spindlespeed*** of ***9500 rpm*** works well with wax stock. Click the ***Apply*** button to save the settings.



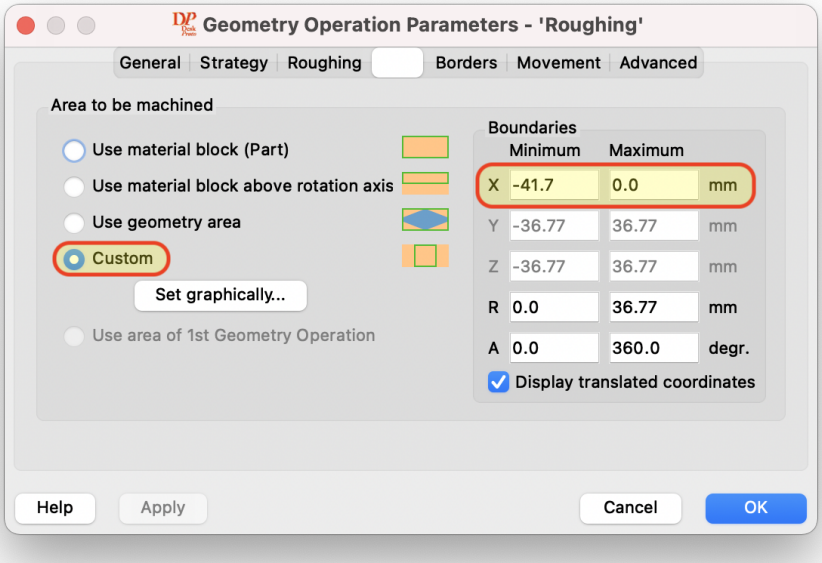
Select the ***Strategy*** tab and select ***Parallel*** tool path ***Around A-axis reversed***. This will take a series of cuts beginning at the top of the stock and moving around the B-axis and ending at the bottom of the part.



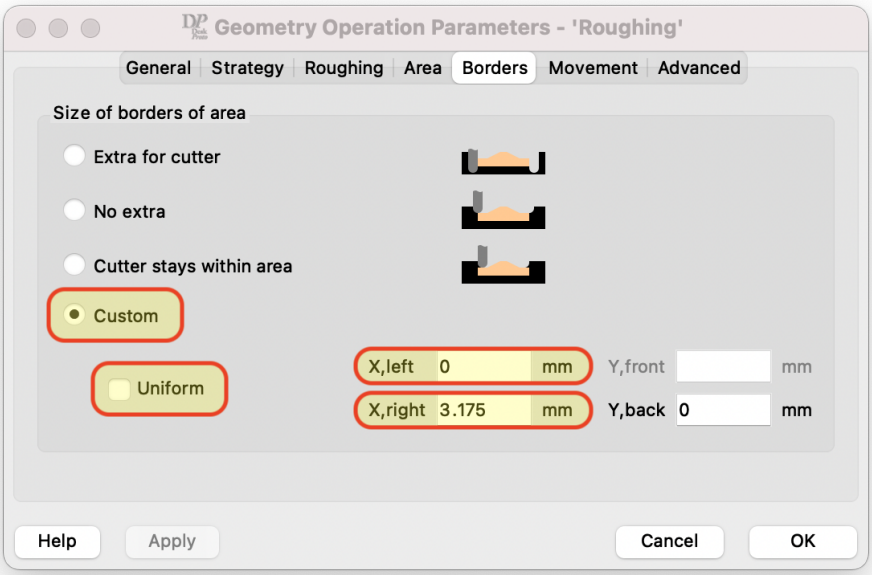
Select the ***Roughing*** tab and select ***Custom Layer Height***. In this case, a 1/8 inch ball nose tool is being used on wax so a depth of cut of ***1.588 mm*** was chosen. Since the roughing tool is a ball nose cutter, a ***Skin Thickness*** of 1 mm was chosen to avoid tool trails showing after completing the finishing operation.



Select the ***Area*** tab and select the **Area to be machined** to be ***Custom***. Adjust the **Boundaries *X minimum*** value to -41.7 mm. This defines the tool to cut the entire part.

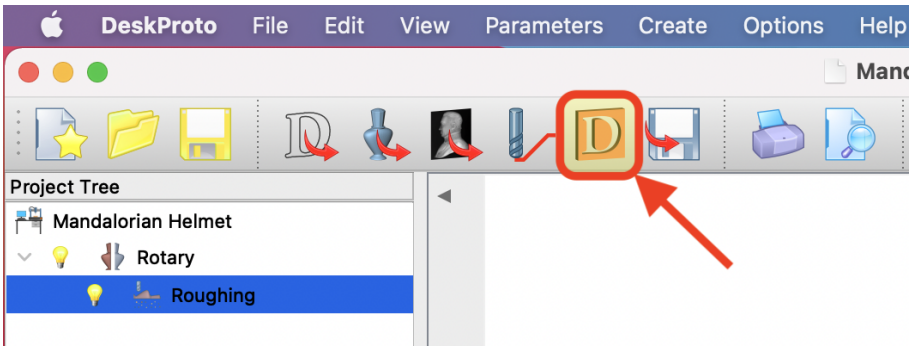


Select the ***Borders*** tab and select **Size of borders of area** to be ***Custom***. Adjust the ***X,left*** value to 0 mm and ***X,right*** value to 3.175 mm.

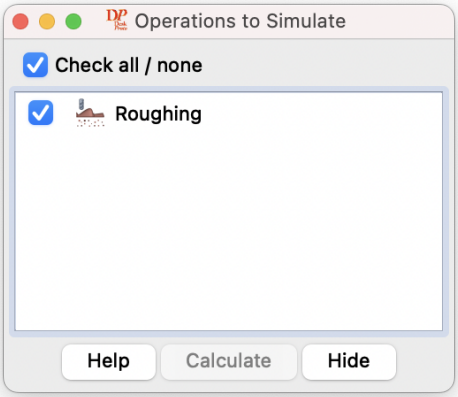


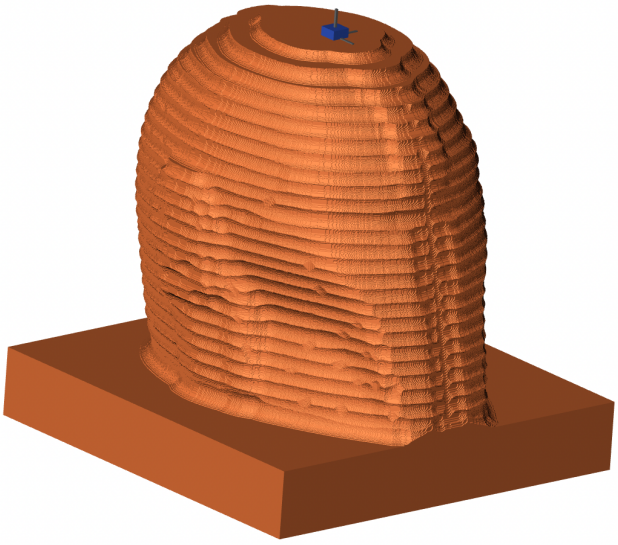
The Mandalorian part does not come with a flat base. For the purpose of this tutorial, the Mandalorian helmet will remain attached to the stock.

Click the Desk*Proto* simulation tab.



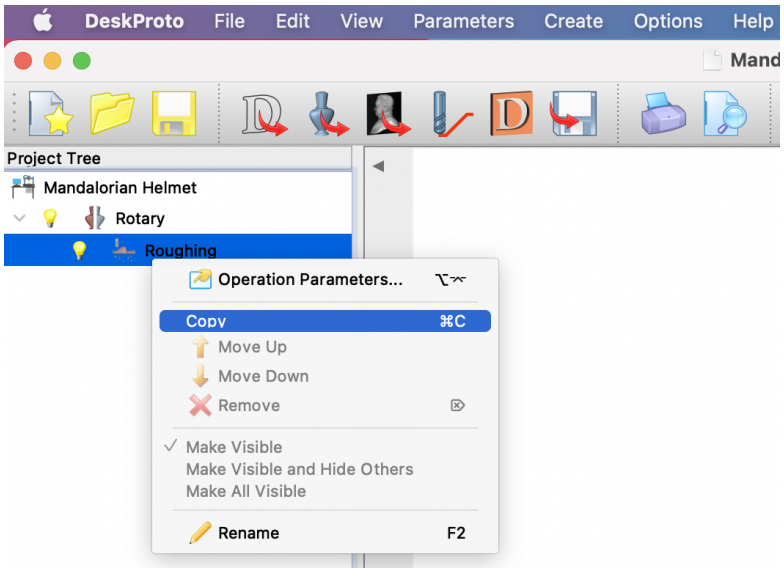
Check the **Roughing** operation and click the **Calculate** button.



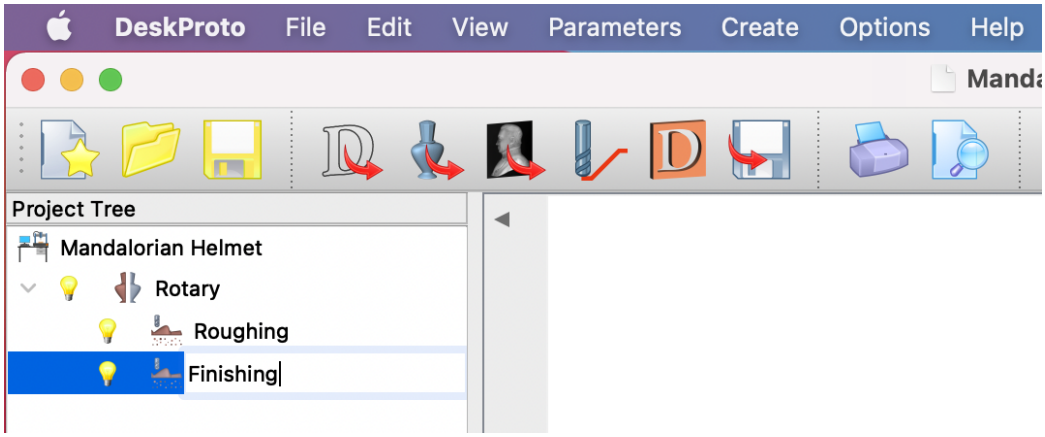


Section 12: Defining the Finishing Rotary Tool Operation

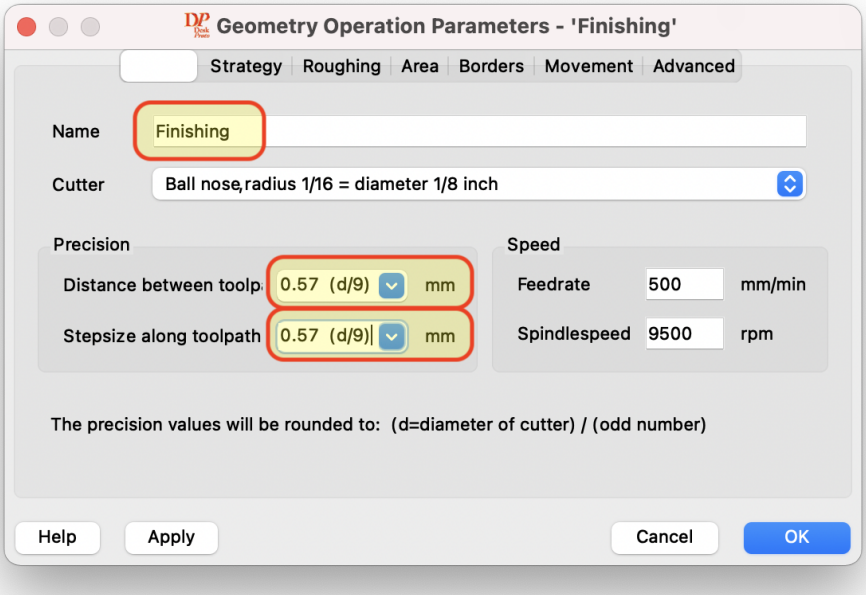
Copy the Roughing Operation.



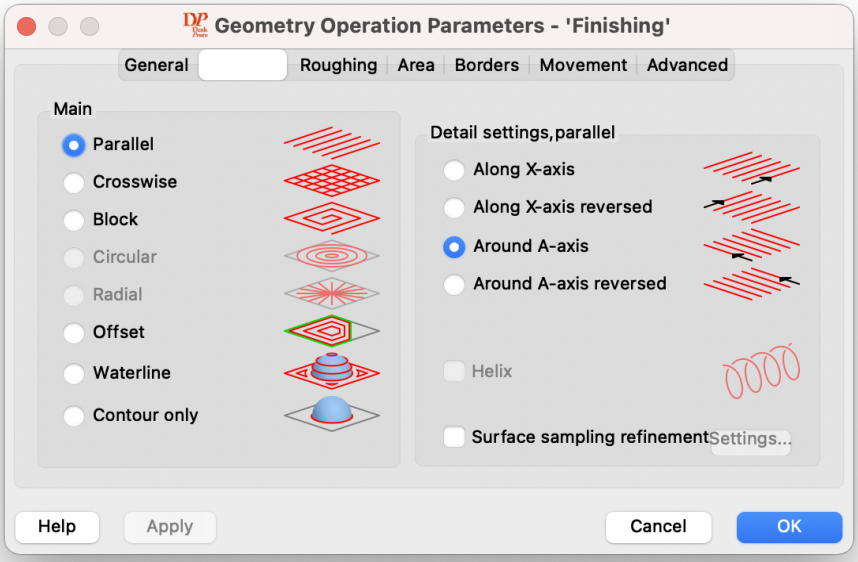
Rename the operation **Finishing**.



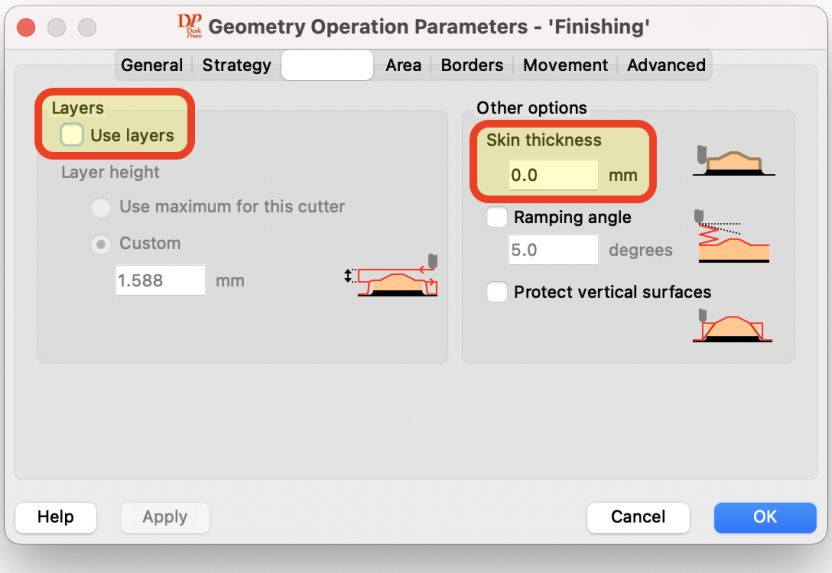
Edit the **Finishing** operation. Under the **Precision** section, select both the ***Distance between toolpath*** and ***Stepsize along toolpath*** to be **0.57 mm** or **(d/9)**.



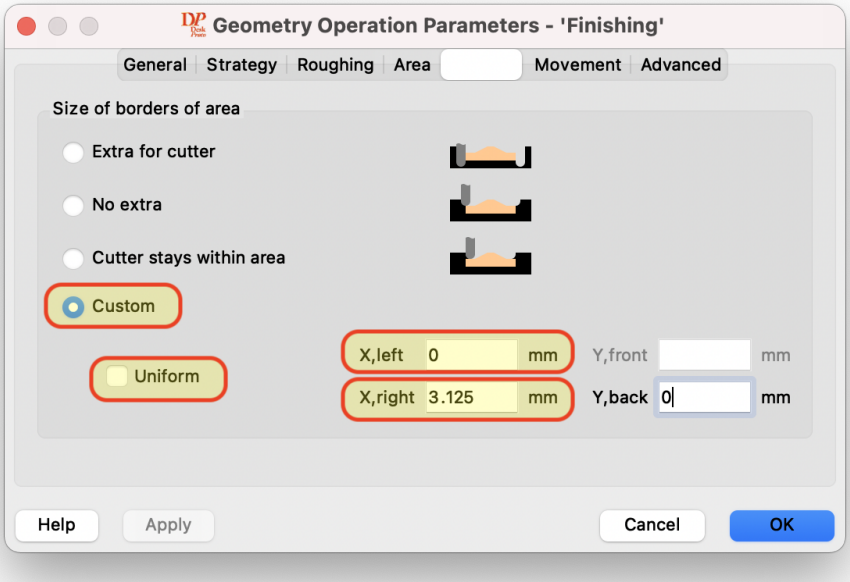
Select the ***Strategy*** tab and select ***Parallel*** tool path ***Around A-axis***. This will take a series cuts beginning at the bottom of the stock and moving around the B-axis and ending at the top of the part. Selecting this strategy eliminates the finishing tool from rubbing on top of the helmet since a skin was applied to the roughing operation.



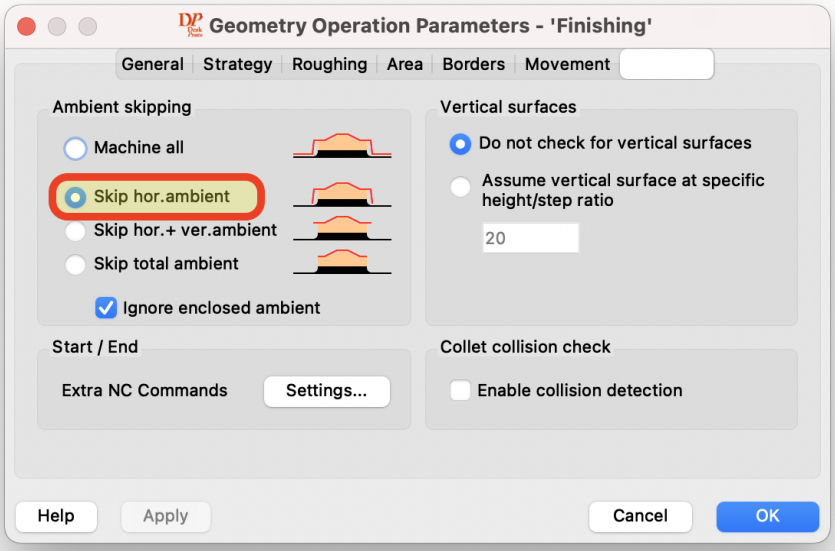
Under **Layers**, uncheck the ***Use Layers*** box. Reduce the ***Skin Thickness*** to 0.0 mm under the **Other Options** section.



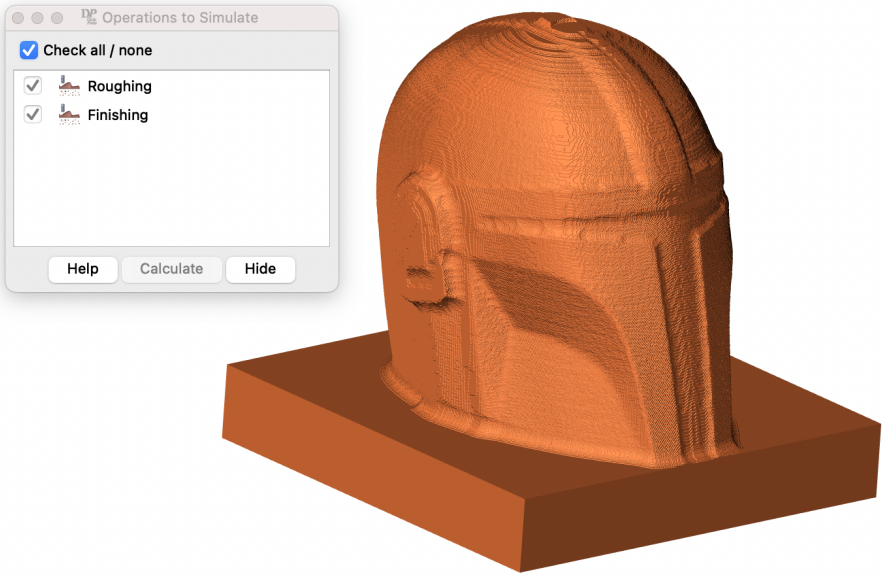
Under the **Border** tab, choose ***Custom*** under the **Size of borders of area** section. Uncheck the ***Uniform*** checkbox. Adjust the ***X,left*** to 0 mm and ***X,right*** to 3.125 mm.



Under the **Advanced** tab, choose ***Skip hor.ambient*** in the **Ambient skipping** section.



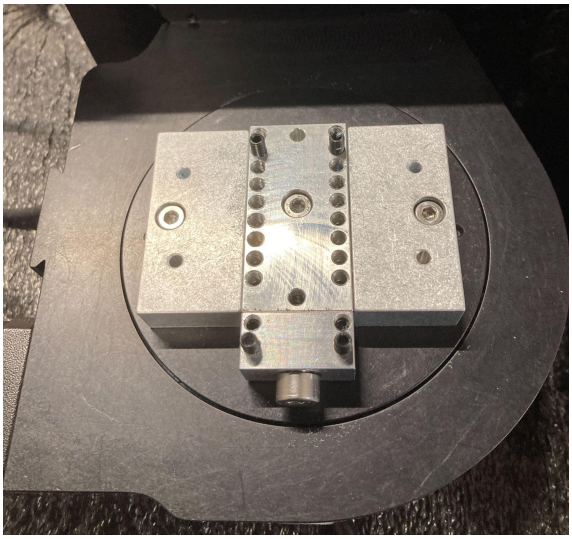
Select the Desk*Proto* simulator and view both the **Roughing** and **Finishing** operations.



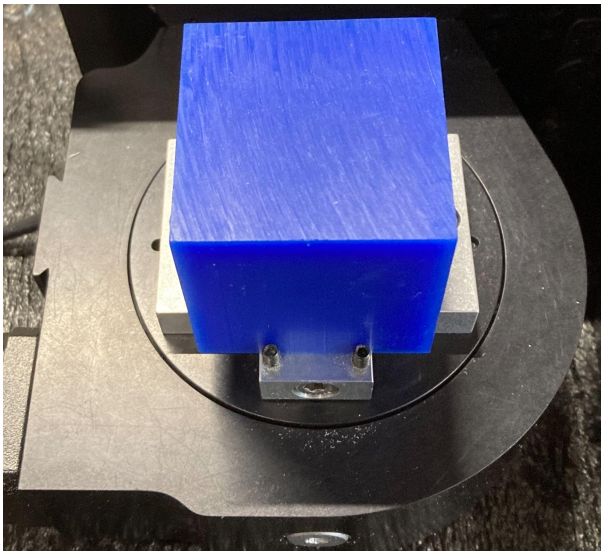
Section 13: Mounting the stock to the Pocket NC V2 and measuring the G55 work offsets.

The Mandalorian Helmet will be cut from a 2 inch cube wax stock. For the purpose of this tutorial, the Pocket NC custom vise that comes with the Pocket NC V2 will be used to mount the stock.

Mount the Pocket NC custom vise plus adapter to the Pocket NC V2 as shown below.

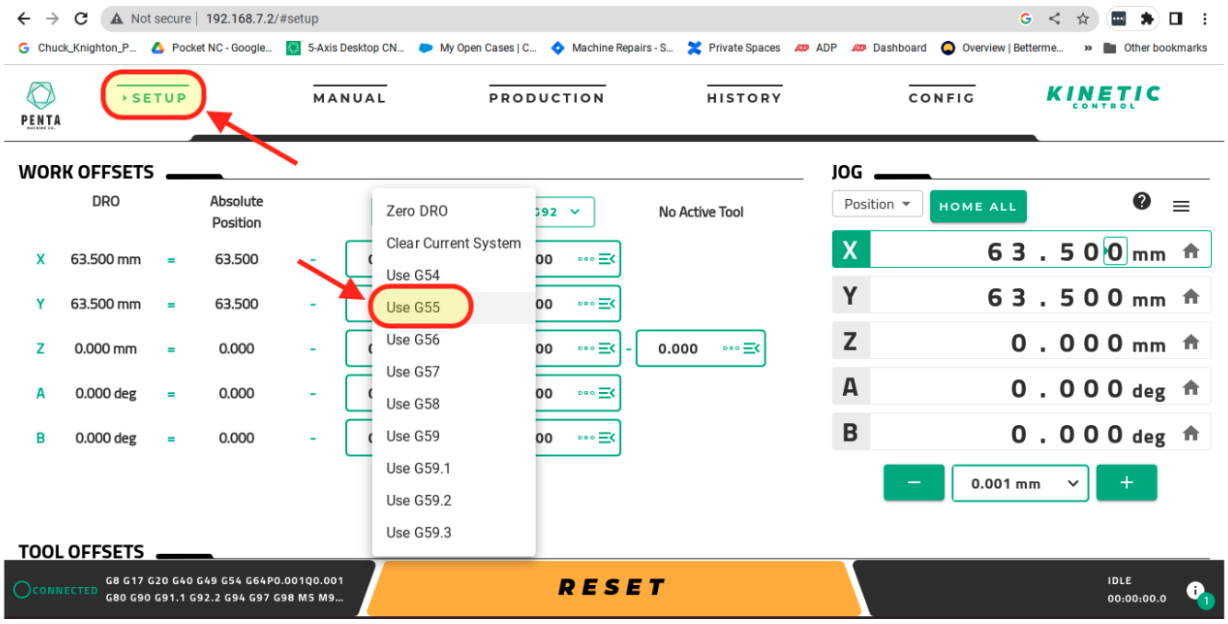


Next, mount the cube stock on the vise.

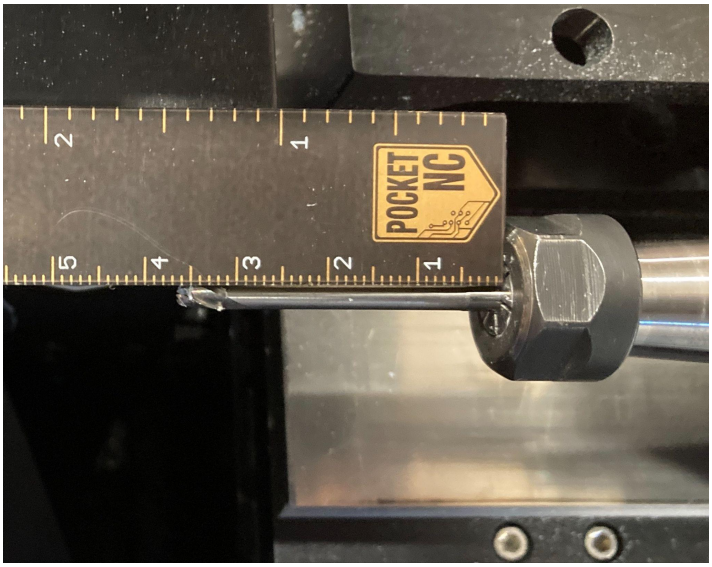


The following steps measure the work offsets that will be used by the Kinetic Control interface to locate the stock's work coordinate origin in relation to the machine's center of rotation.

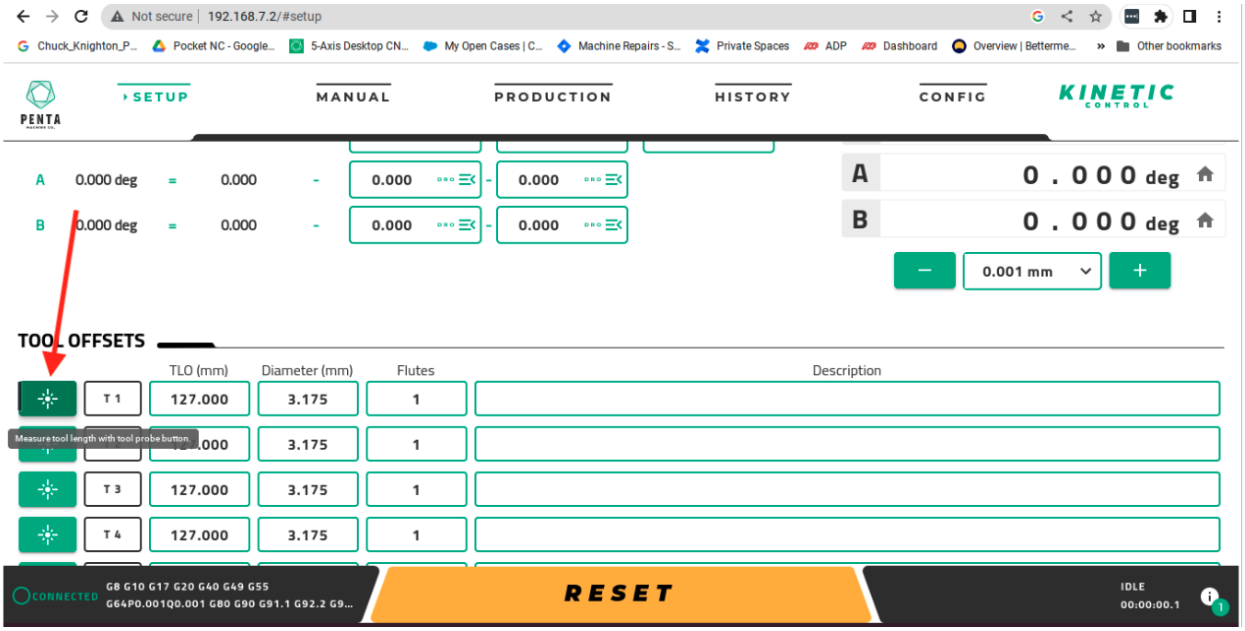
Desk*Proto* uses the G55 work offsets by default. To set up Kinetic Control, select the **G54** pulldown menu and select the ***G55*** under the **WORK OFFSETS** section.



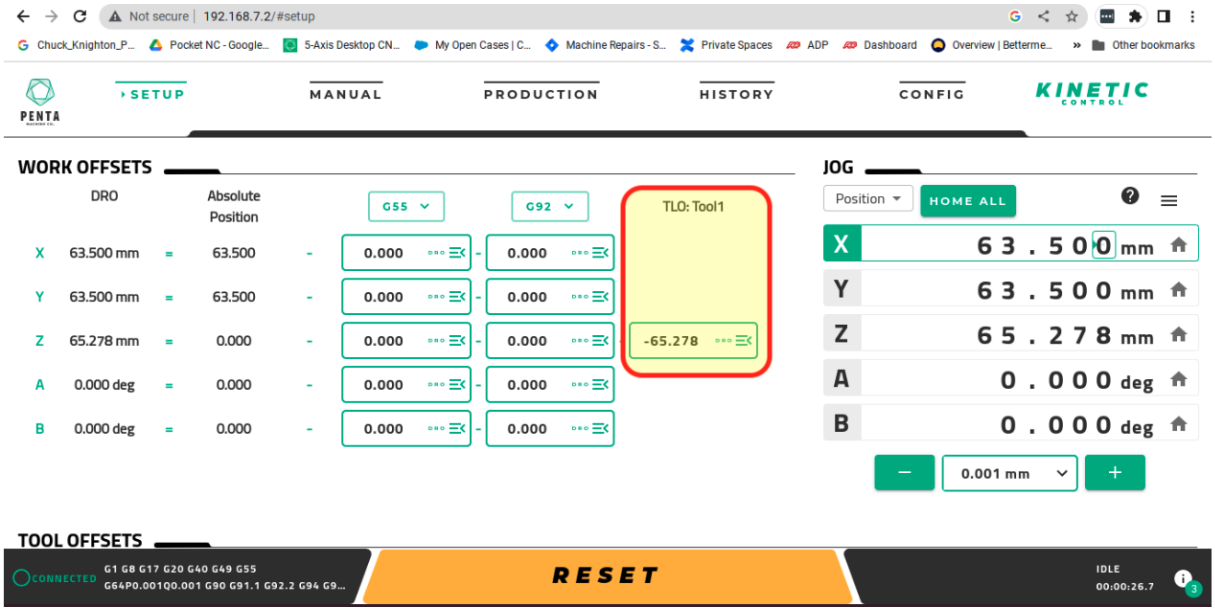
To properly measure the work offsets, the tool length offset for Tool 1 must be measured. Load the Harvey Tool 1/8th Ball Long Reach, Stub Flute - part number: 96108 in the tool holder for your V2. For the purpose of this tutorial, a Pocket NC V2-10 was being used. The picture below shows the tool loaded in the V2-10 extended tool holder. Adjust the tool so that the tool stickout is approximately 36 mm.



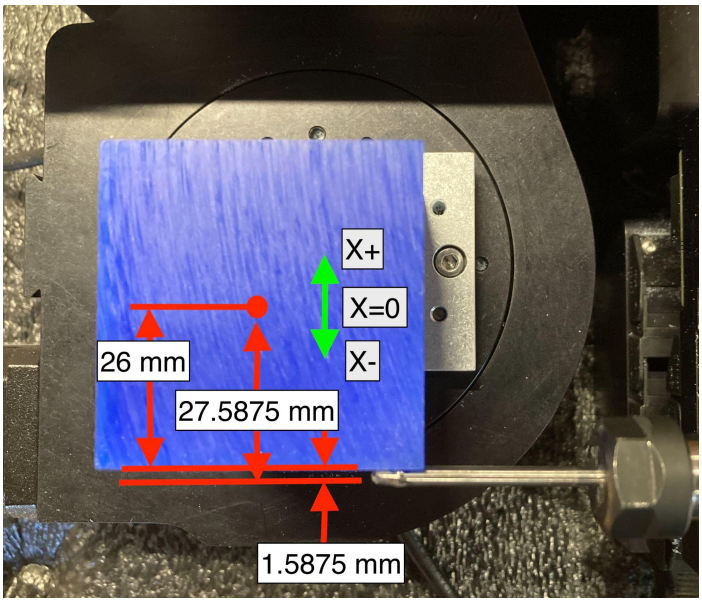
Next measure the Tool Length Offset using Kinetic Control. Navigate to the **SETUP** tab on Kinetic Control. Click the green button next to Tool 1 to initiate a tool length offset measurement. Once the Tool Length Offset for Tool 1 is completed. Click the white T1 button to make Tool 1 the active tool.



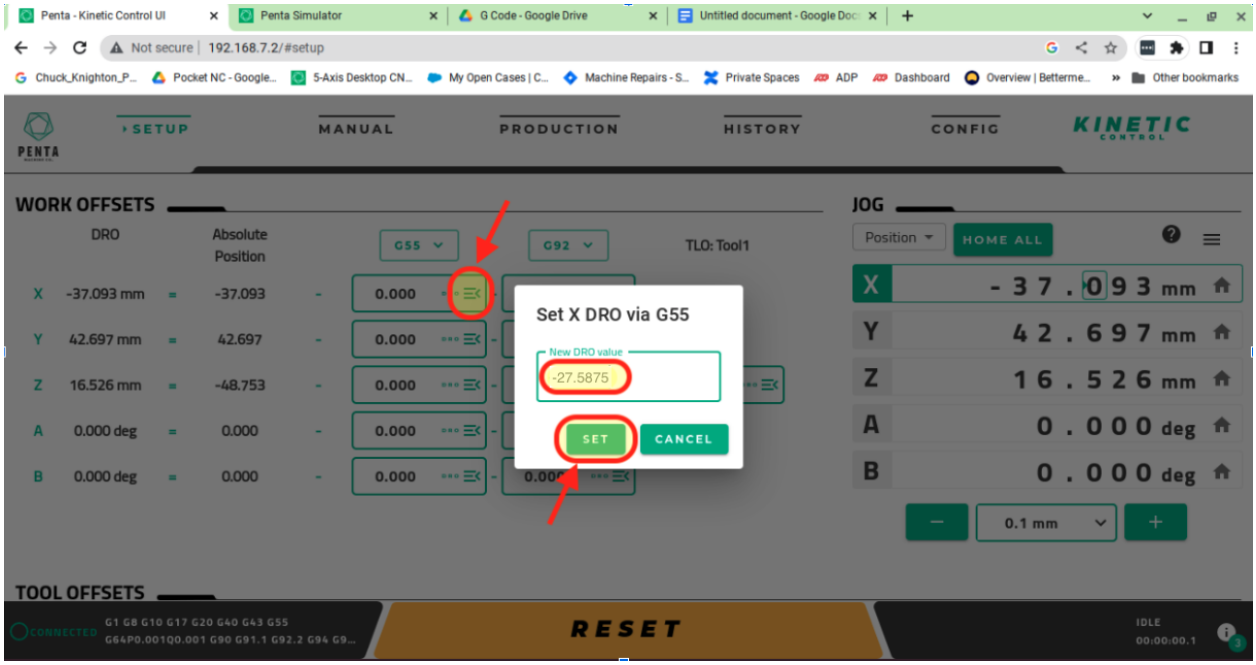
The **SETUP** screen should show Tool 1 as active and the tool offset value should be approximately -65 mm if using a Pocket NC V2-10 (see below).



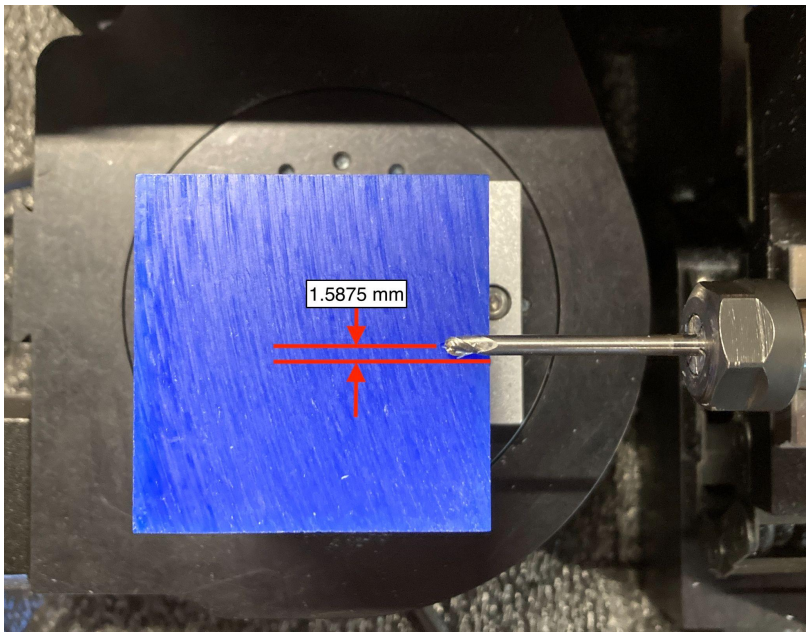
Perform an X touch off by jogging the tool so that it just touches the front face of the stock. The tool center is located one half of the tool diameter (1.5875 mm) plus one half of the stock's side dimension (26 mm) to the negative from X=0 on the stock. The total distance to X=0 on the stock is -27.5875.



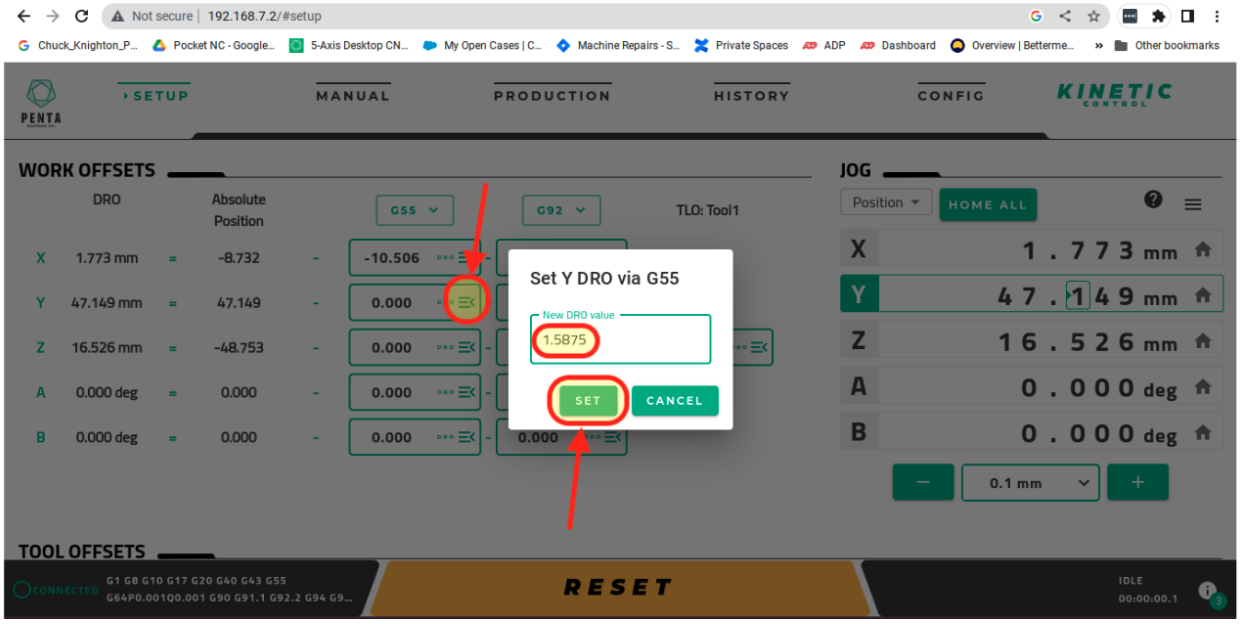
Click the three bars icon in the X work offset box. Enter -27.5875 for the **New DRO value** and click the **SET** button.



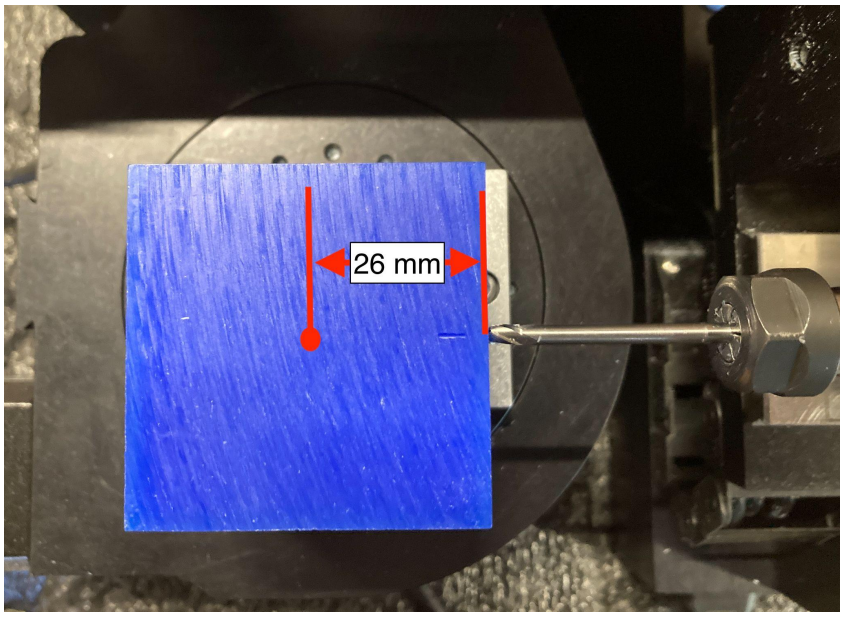
Perform a Y touch off by jogging the tool so that it just touches the top face of the stock. The tool center is located one half of the tool diameter (1.5875 mm) to the positive from Y=0 on the stock. The total distance to Y=0 on the stock is +1.5875 mm.



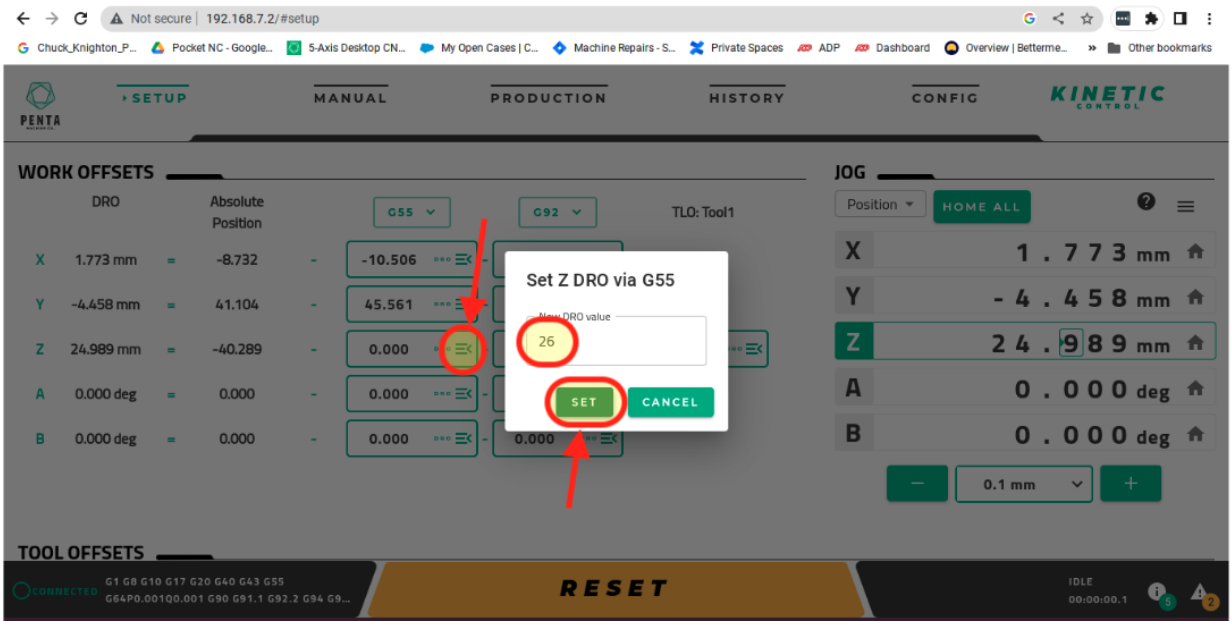
Click the three bars icon in the Y work offset box. Enter +1.5875 for the **New DRO value** and click the **SET** button.



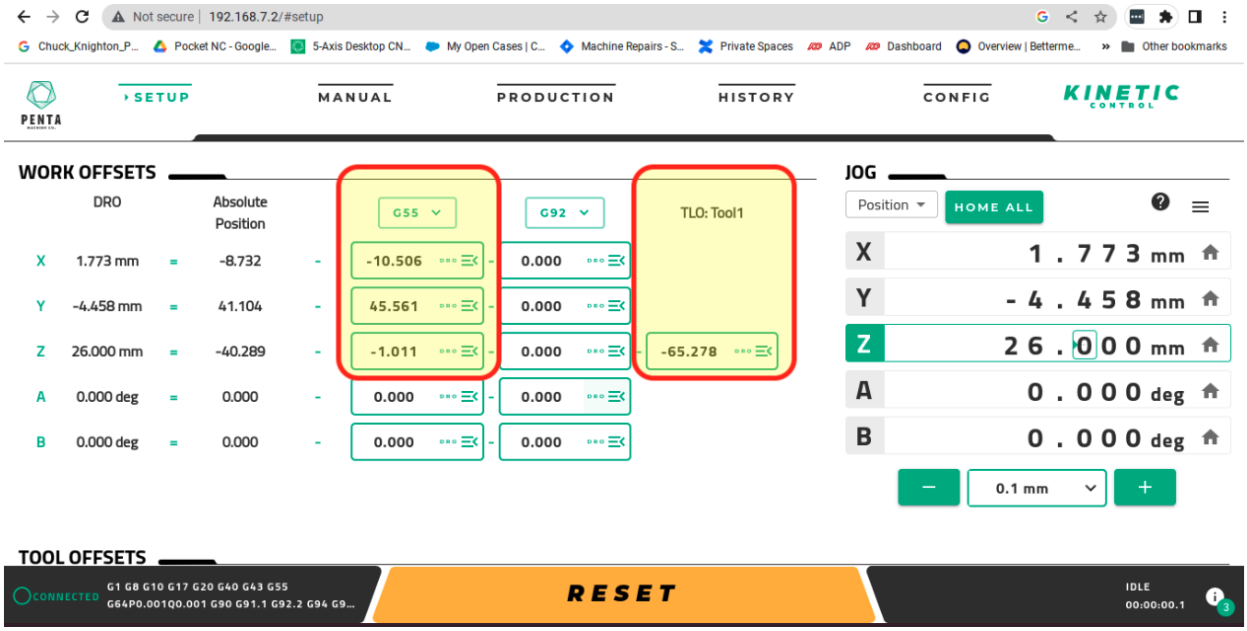
Perform a Z touch off by jogging the tool so that it just touches the right face of the stock. The tool center is located one half of the stock's side dimension (26 mm) to the negative from Z=0 on the stock. The total distance to Z=0 on the stock is +26 mm.



Click the three bars icon in the Z work offset box. Enter +26 in for the **New DRO value** and click the **SET** button.



The G55 **WORK OFFSETS** should be entered for **X**, **Y**, and **Z** and Tool 1 should be selected as the active tool.

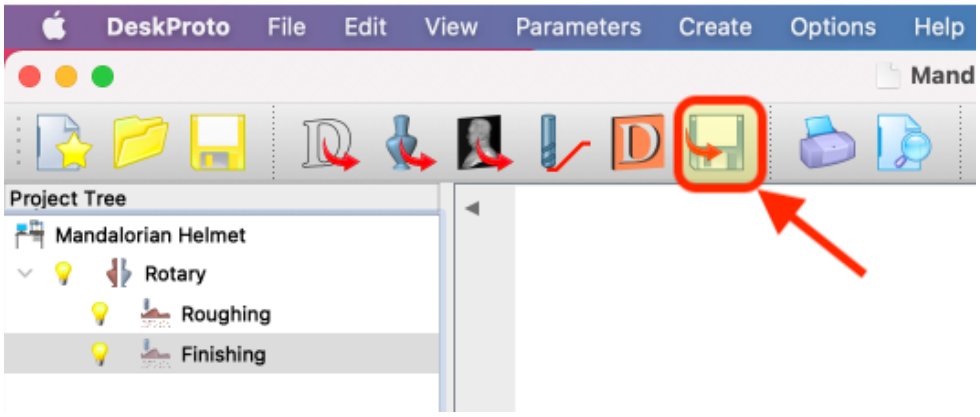


The G55 values and the Tool Length Offset value will be used in the next section to evaluate the G-Code created by Desk*Proto*.

Section 14: Modifying the G-Code using the Penta Machine Company Simulator to utilize TCPC.

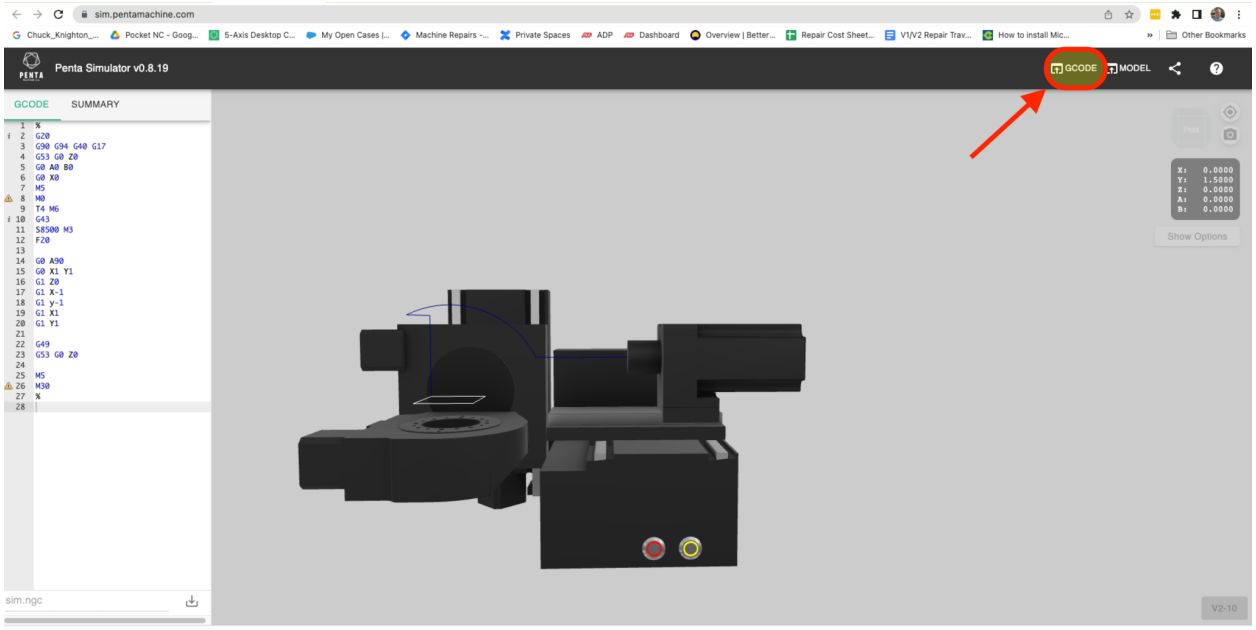
These following steps are necessary when using the Pocket NC custom vise. The stock material will not be centered around the machine center of rotation. Kinetic Control uses TCPC (Tool Center Point Control) to translate the stock work center coordinates to the machine center of rotation.

Post the g-code by clicking on the **Post Processor** icon.



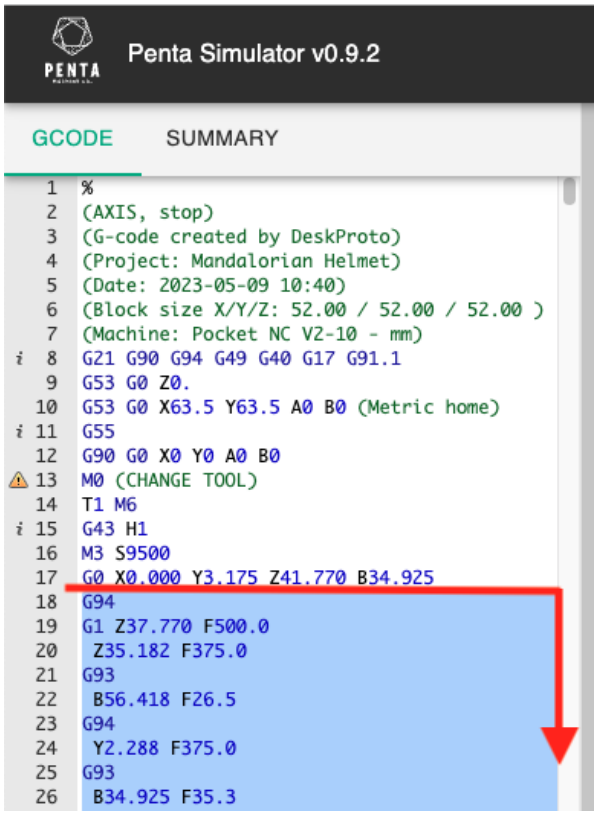
The next step is to load the program g-code into the Penta Machine simulator, which can be accessed by clicking here.

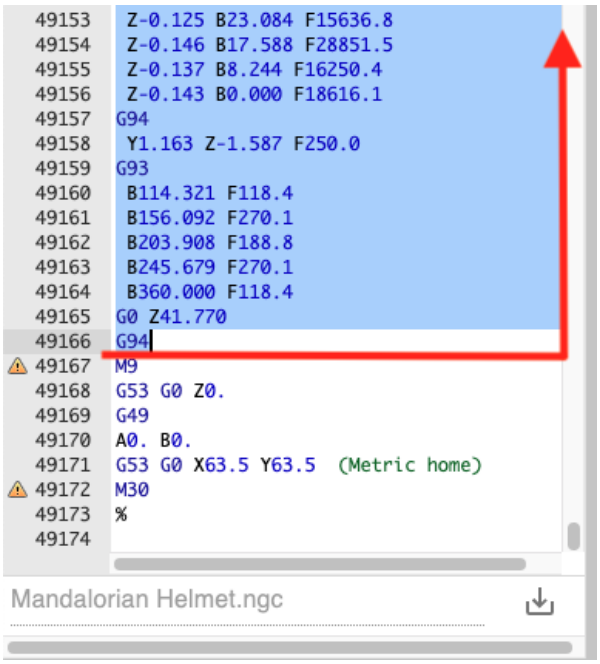
Load the program into the Penta Machine simulator by clicking the **G-Code** tab.



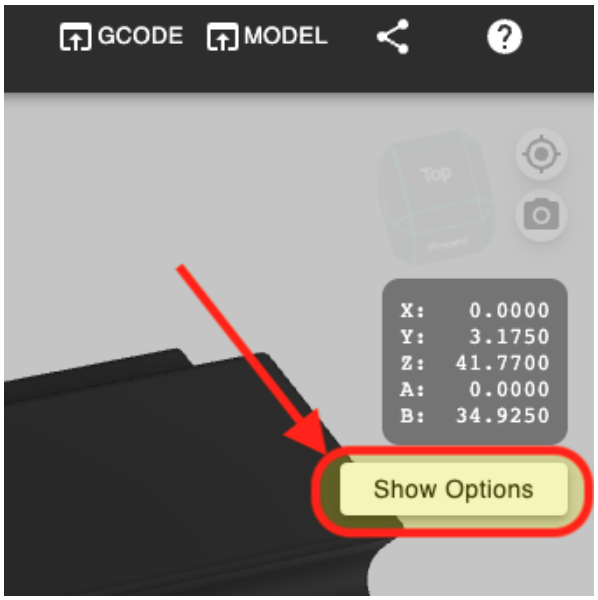
The Penta Machine Company simulator, version 0.9.0 and higher, now can convert g-code to use TCPC.

Using this conversion process requires selecting the g-code where TCPC should be applied. This process is easy when the roughing and finishing tools are the same. Below is an example of the beginning and ending code for the Mandalorian Helmet. Select the code after the M0 (Change Tool) section. Select the code after the G0, rapid move, command and the G53 , move in machine coordinates, command. The M9, turn mist and flood coolant off, is ignored by the simulator.

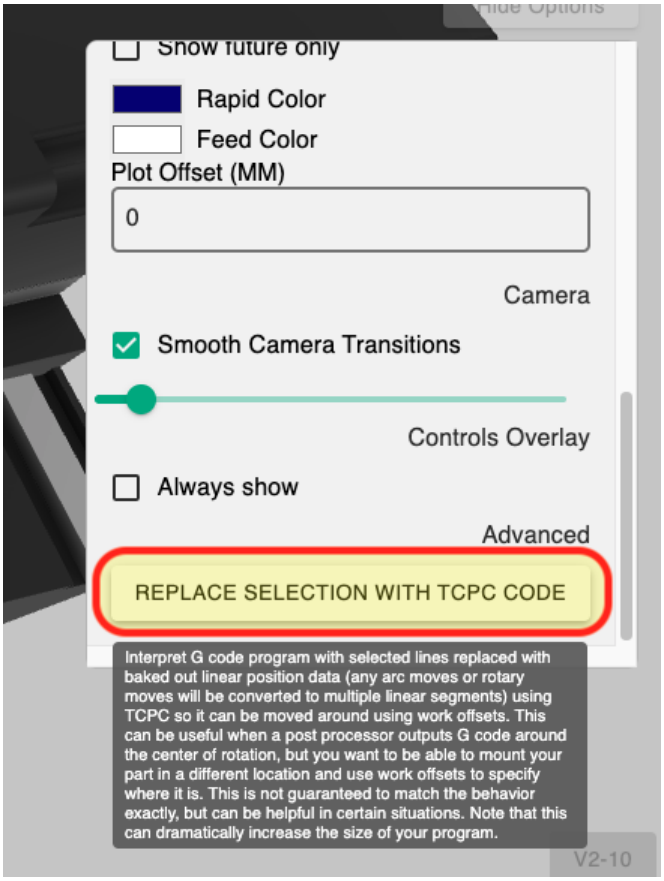




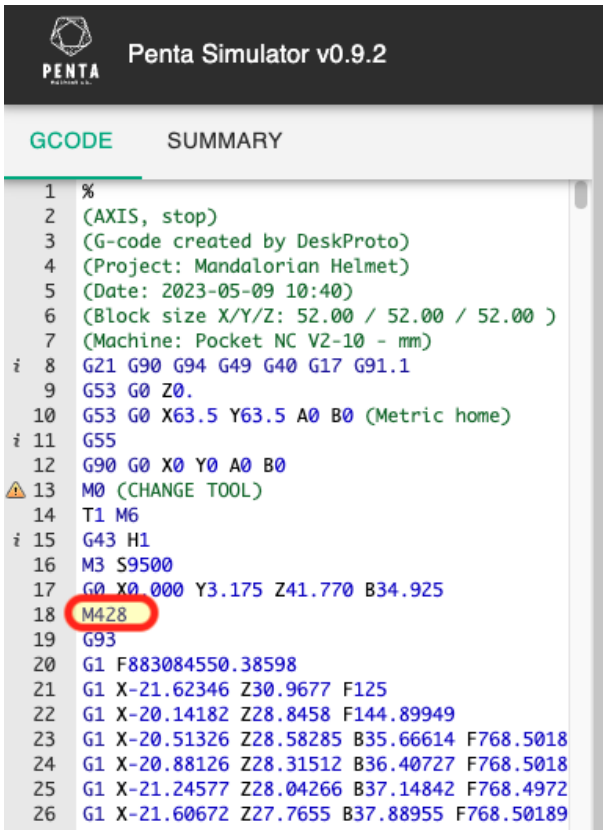
Click the **SHOW OPTIONS** button (located in the upper right corner of the display).

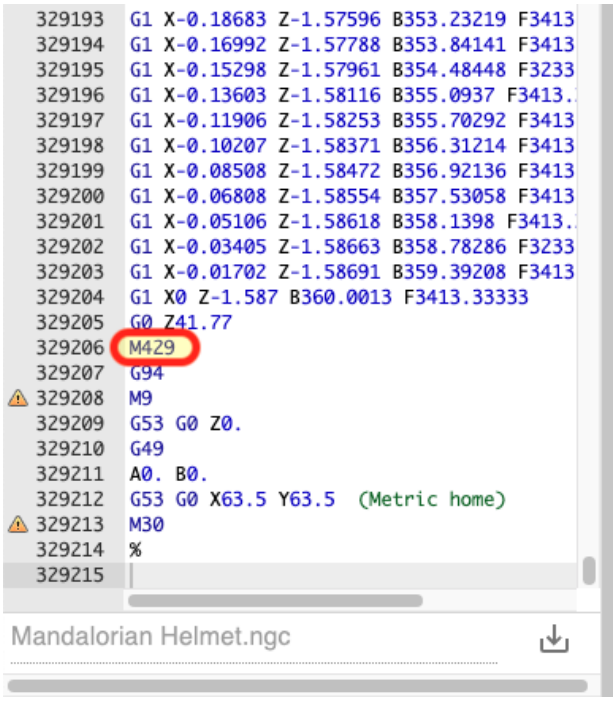


Scroll to the bottom of the **SHOW OPTIONS** and click the ***REPLACE SELECTION WITH TCPC CODE***.

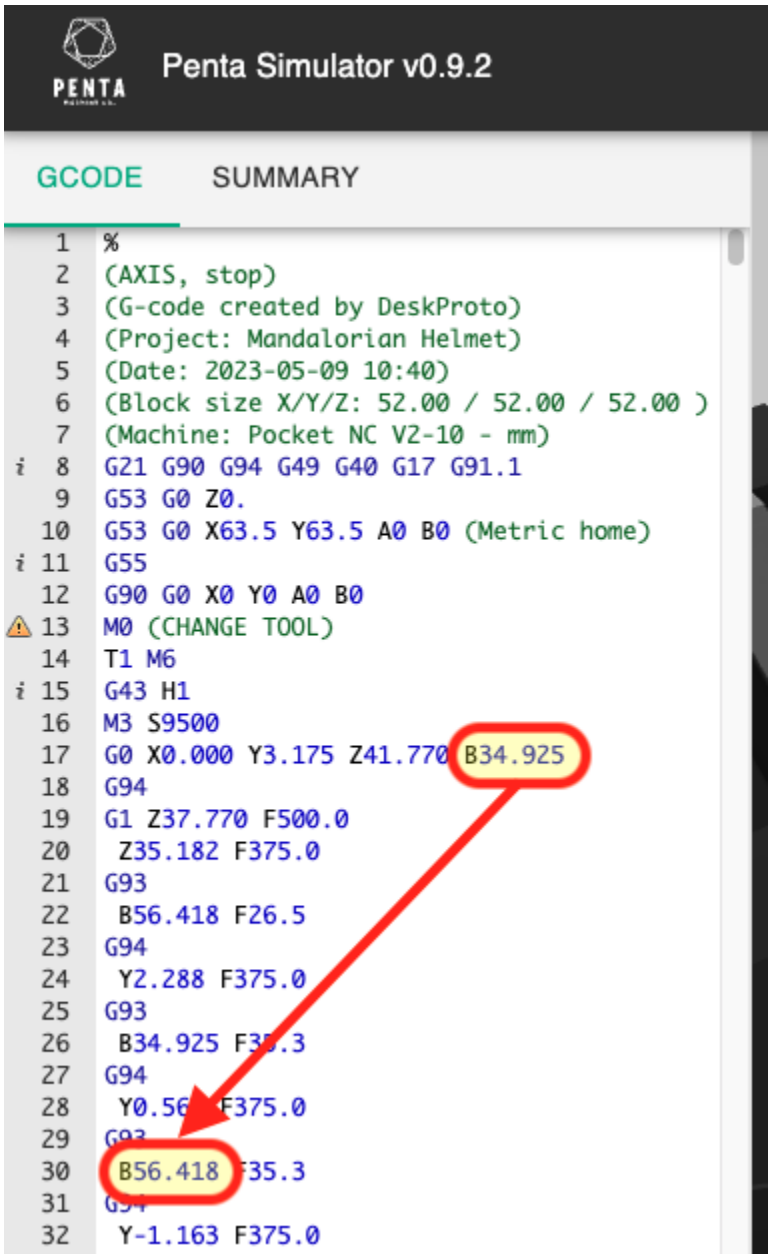


The simulator will now place an M428 (Enable TCPC) at the beginning of the tool operation where TCPC should begin and an M429 (Disable TCPC) at the end of the tool operation where TCPC should end.

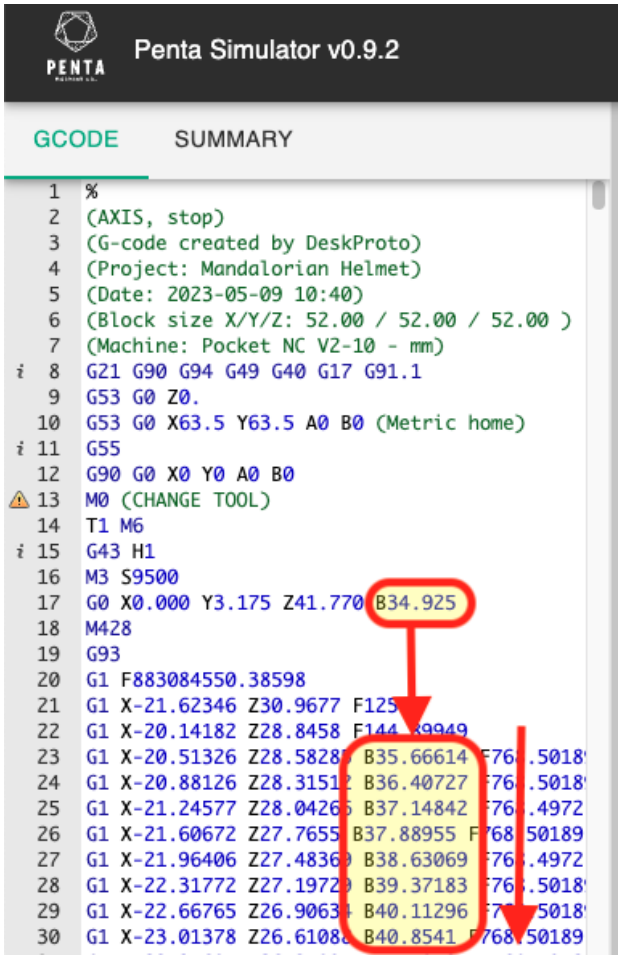


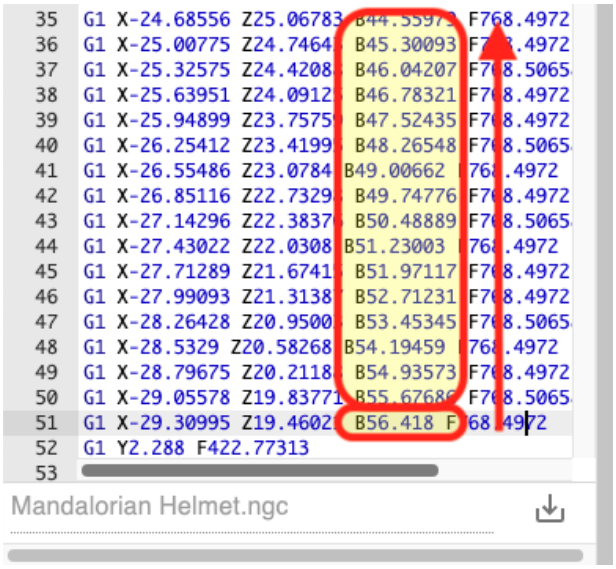


- The TCPC conversion process also interpolates B axis rotation moves to maintain the tool point at the correct Z position as the B axis rotates. Below is the beginning part of the code before the TCPC conversion. Notice that B begins to rotate from 34.925° and move to 56.418°.

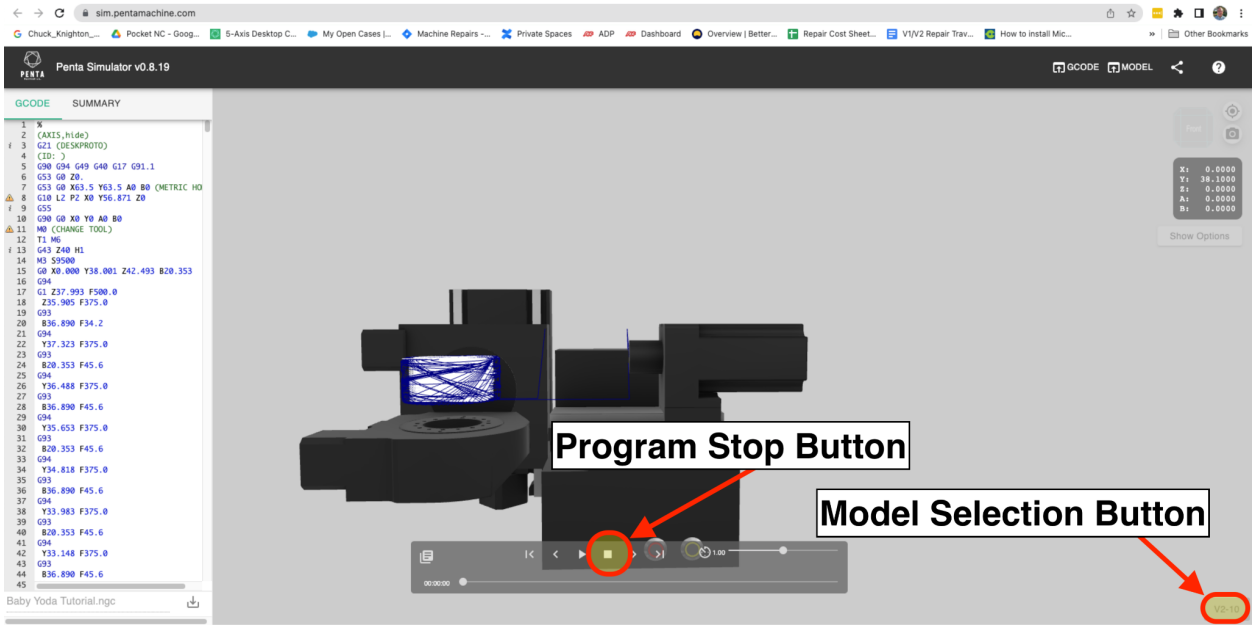


When the code is converted to TCPC, several steps are added to the program to provide a smooth transition from the start of the rotation to the end of the rotation (see below).

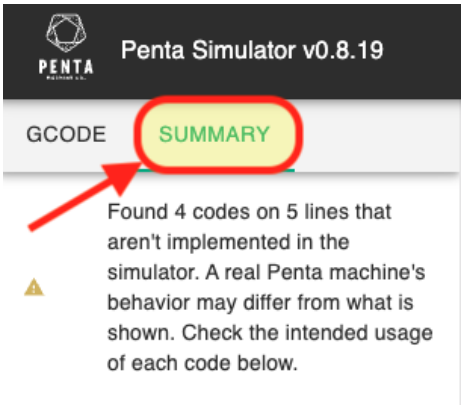




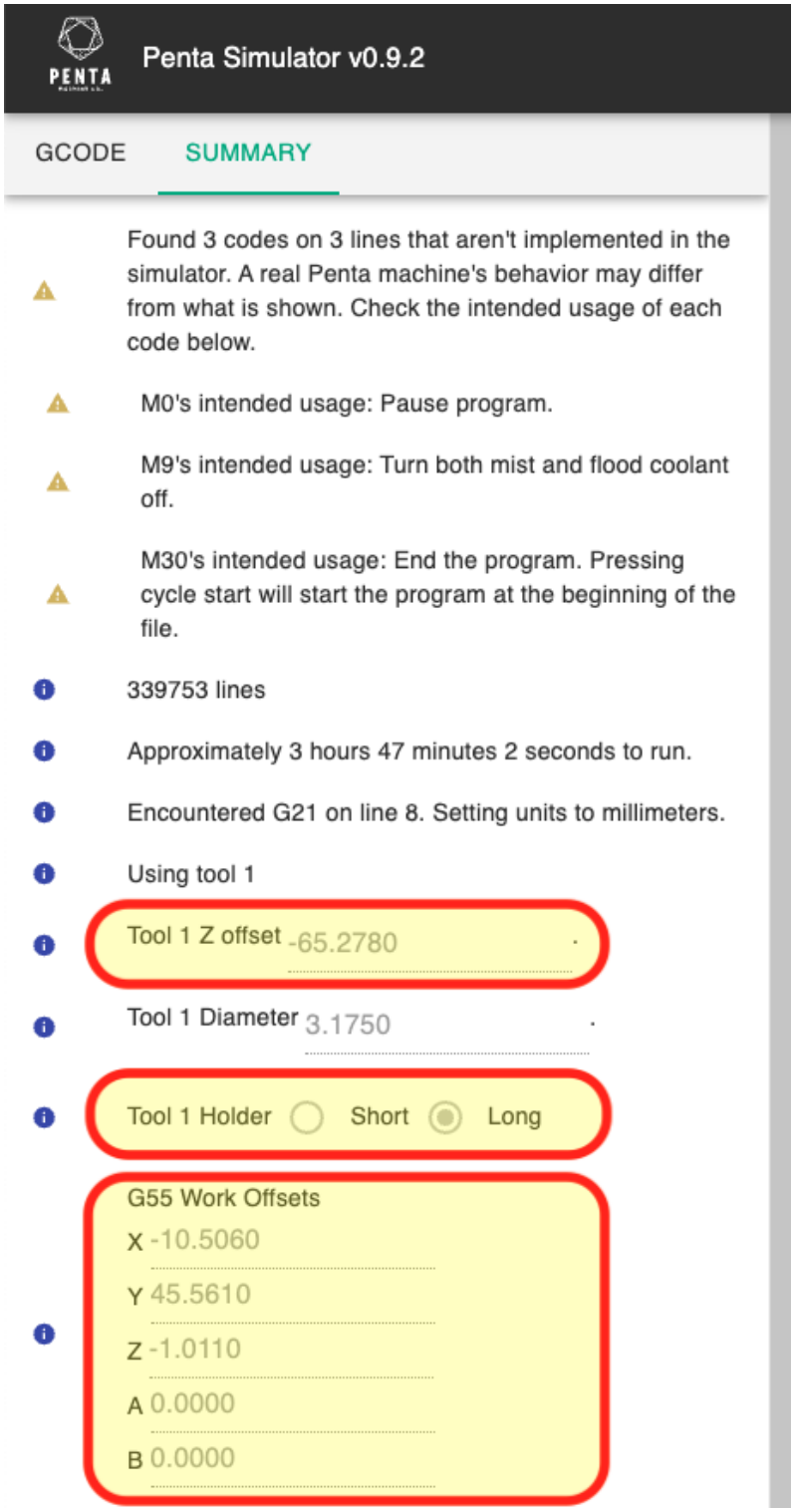
Next select your model of Pocket NC. First click the **Program Stop** button and then click on the **Model** tab in the lower right corner.



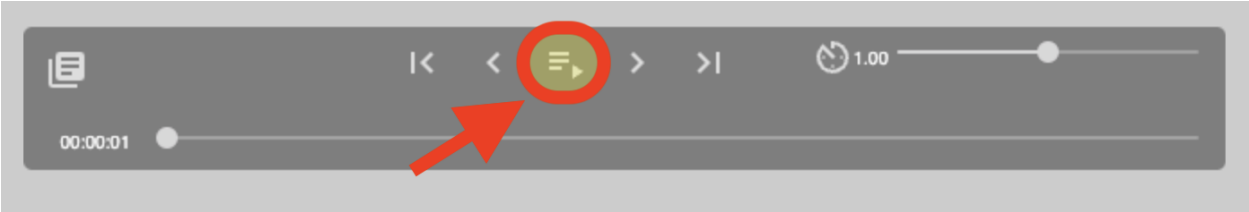
Configure the **Penta Machine** simulator by entering the **Tool Length Offsets** and the **G55 work offsets**. Click on the **SUMMARY** tab.



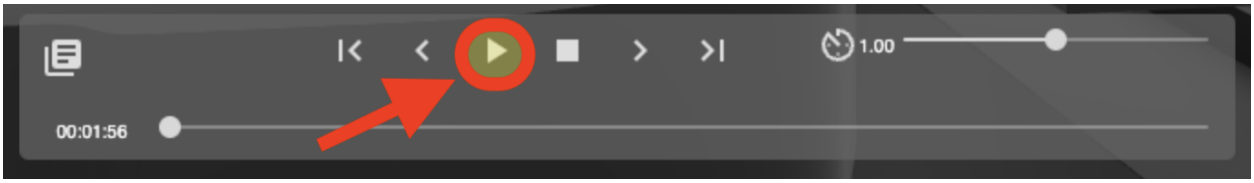
The **Tool Length Offset** and the **G55 Work Offsets** were determined in **Section 13**. By default, the Pocket NC V2-10 machine uses the **Long Tool Holders**.



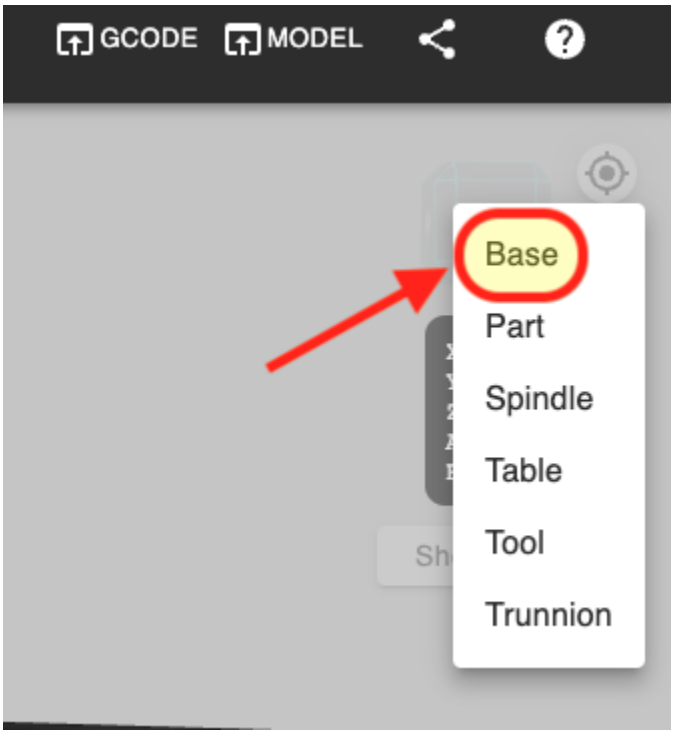
Click the **Program Run** button to accept the values.



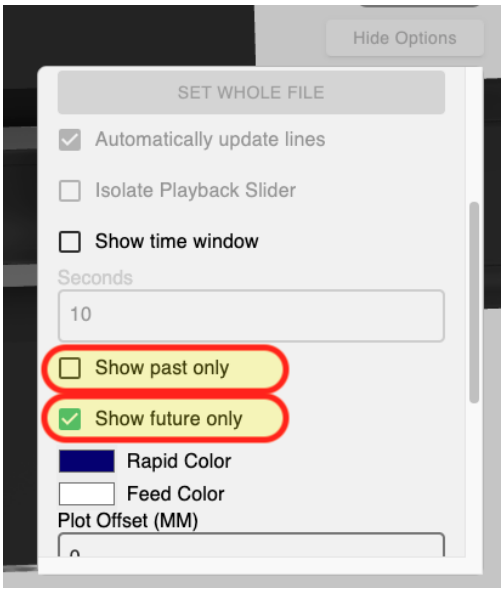
Click the **Play** button to run the program.



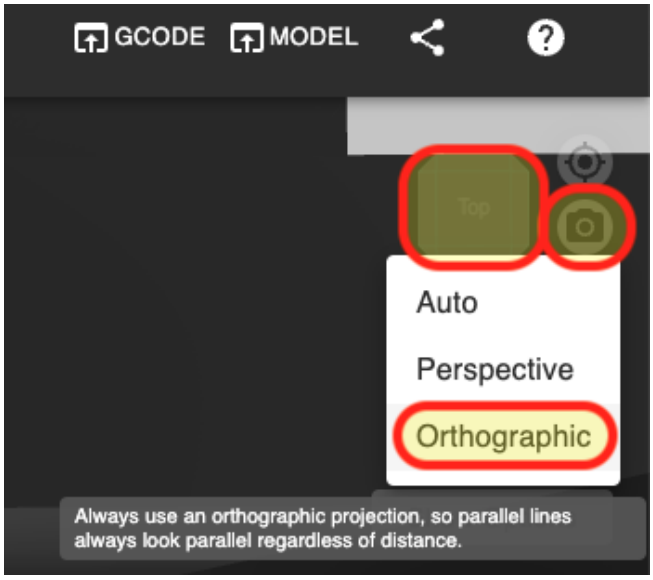
The **Camera Options** button makes it easy to view the desired details. Click on the **Camera Options** button and select the **BASE** option to focus on the part being machined.



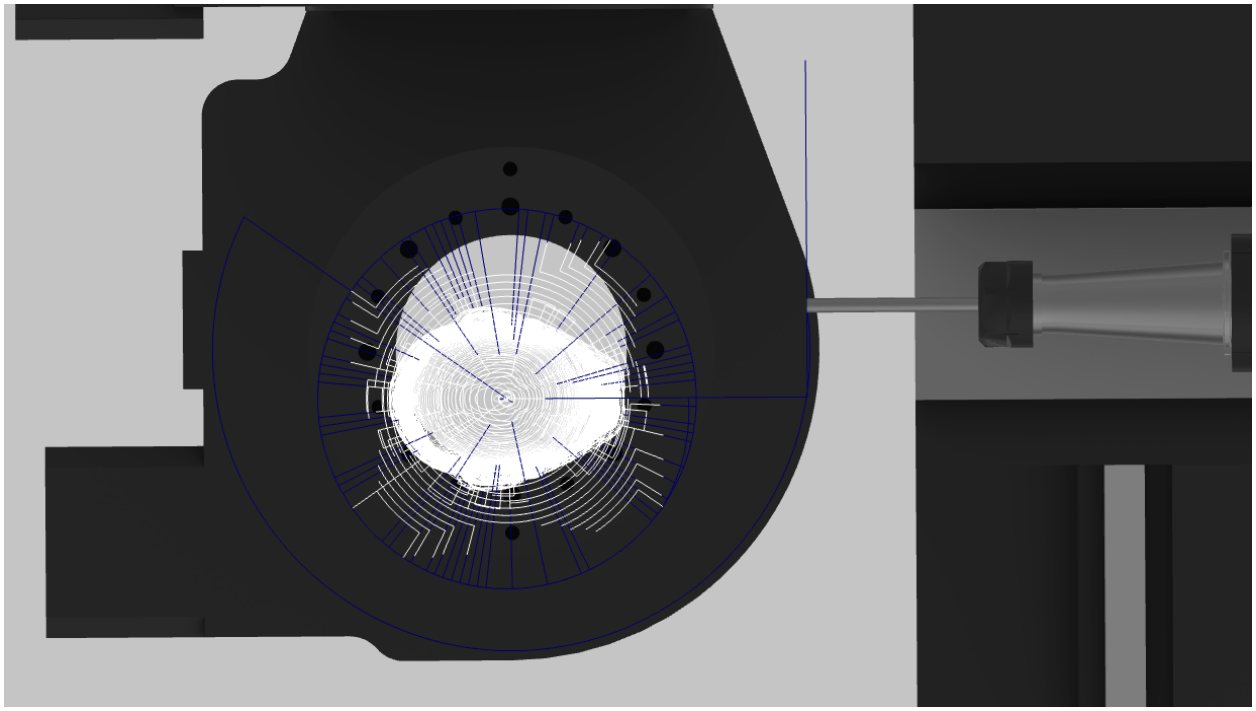
Under **Show Options** select ***Show future only*** and deselect ***Show past only***.



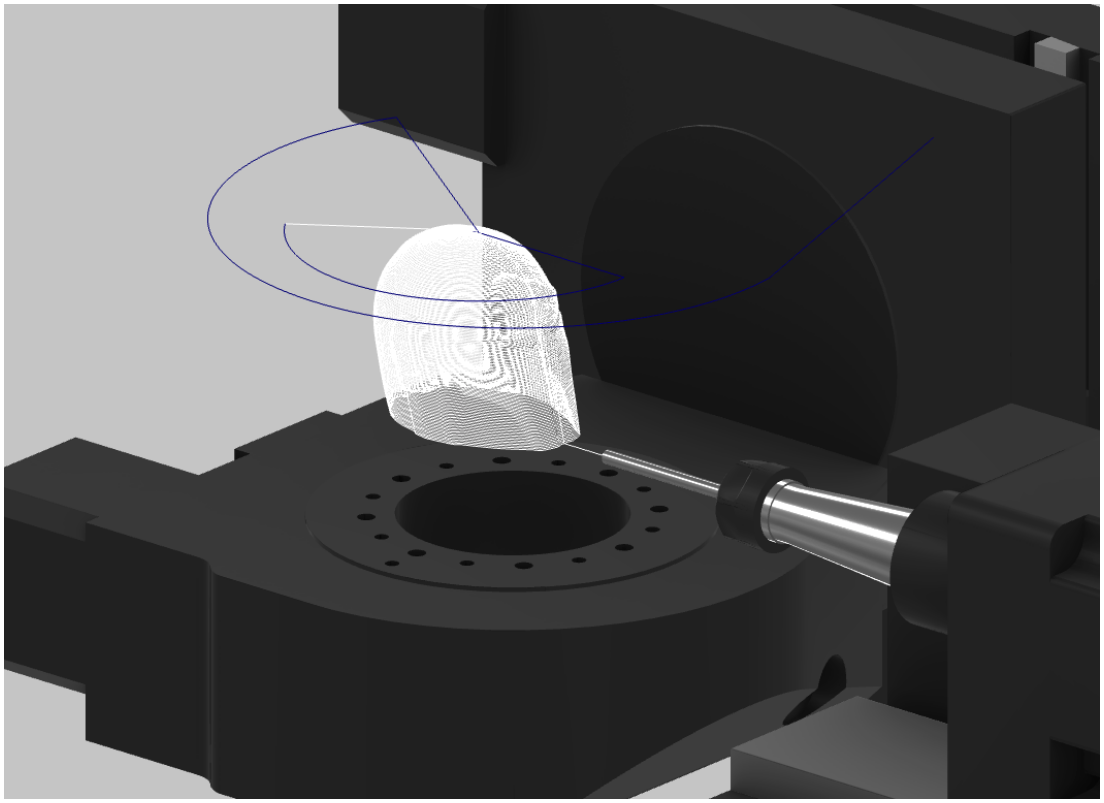
Select the **Top** view of the part. Then select the **Orthographic** view to provide a clearer illustration of the tool paths.



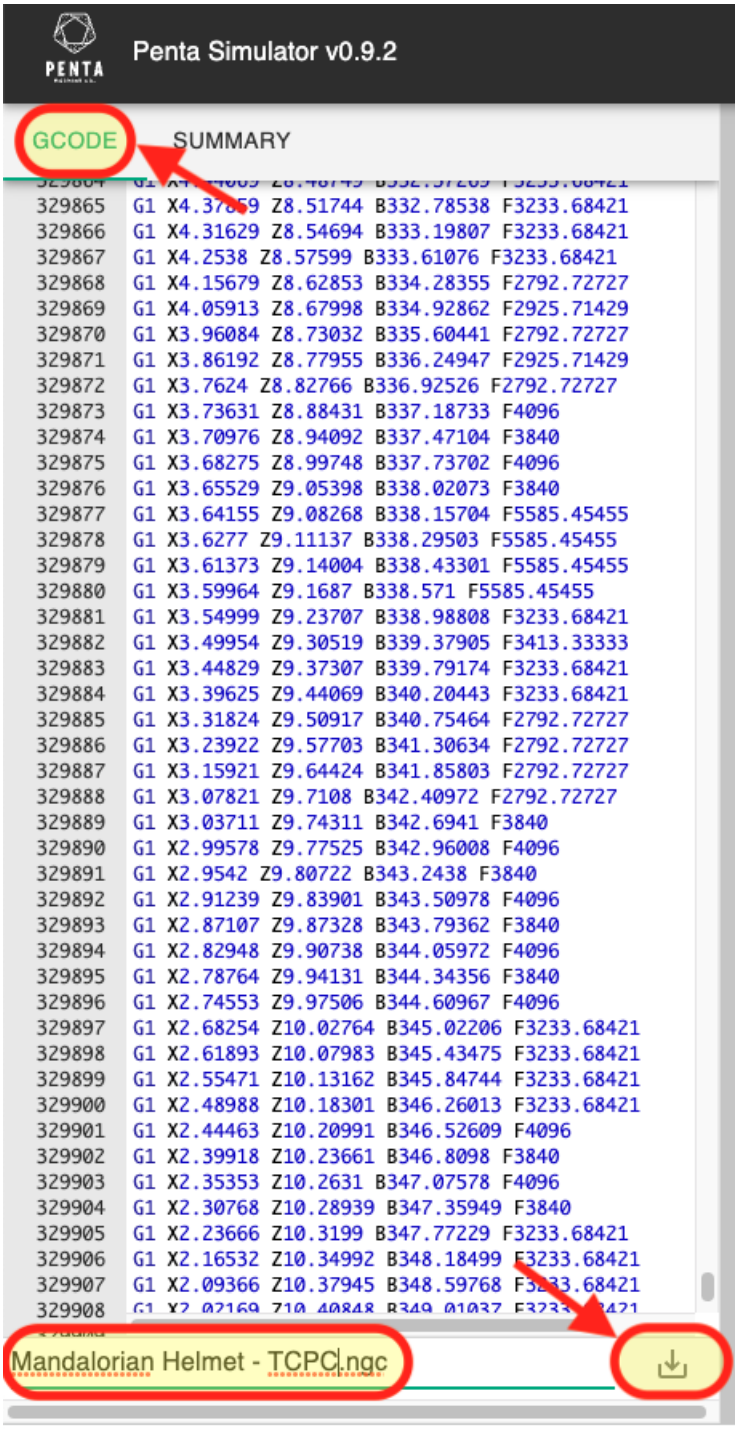
The simulator view should look similar to the image below as the program starts.



The below view of the simulator shows the finishing tool paths.

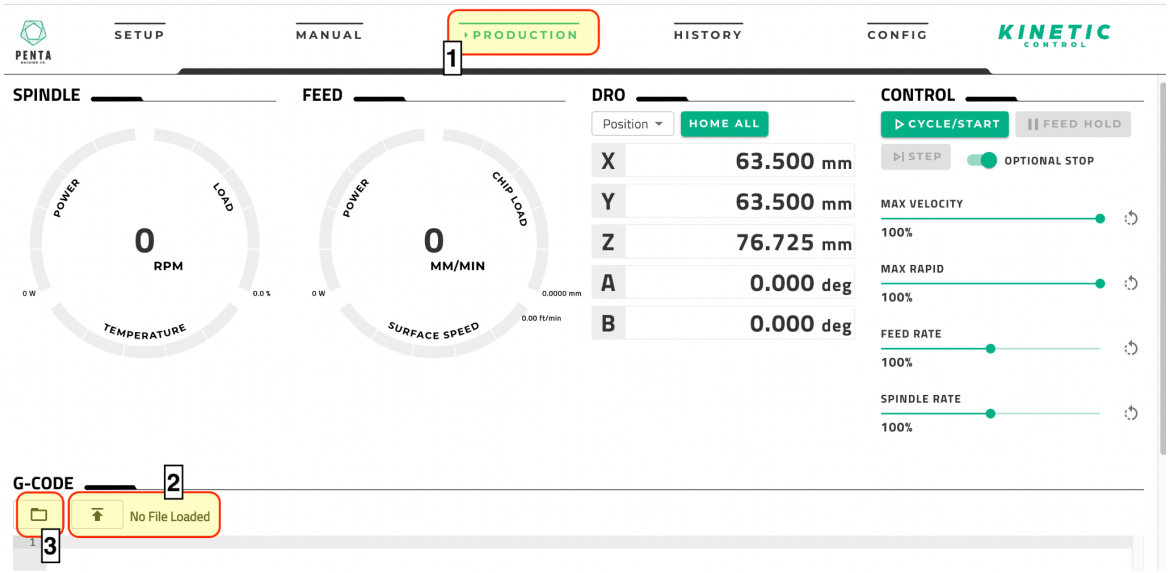


Now is the time to save the modified g-code. Click the simulator's **STOP** button. Click on the **GCODE** tab and locate the file name pane in the bottom left hand corner of the simulator. Modify the g-code name by adding a "- TCPC" to identify it as modified code and click the download icon. The modified g-code will be stored in your computer's DOWNLOADS folder.



Section 15: Run the G-Code Program on the Pocket NC

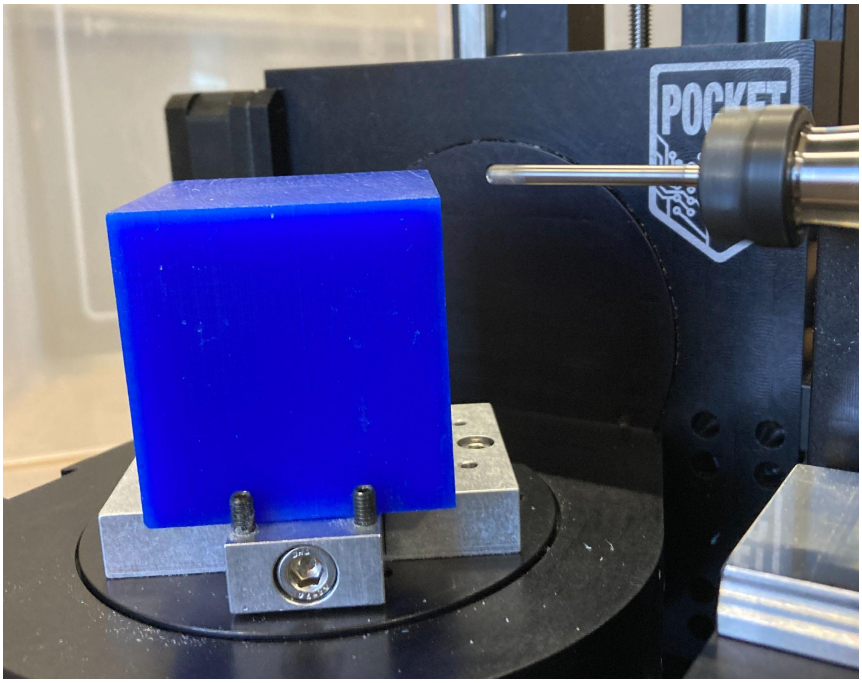
1) Navigate to the **PRODUCTION** tab on Kinetic Control.
2) Select the **File Upload** icon in the lower left hand corner of **Kinetic Control** and choose the g-code file.
3) Finally, select the file to run from the **File Folder** icon.



Once the program is loaded into **Kinetic Control**, click the **Cycle/Start** button on the **PRODUCTION** page. **Kinetic Control** will measure a **Tool Length** offset for Tool 1 if **M6** is enabled.

⚠️ Click the **FEED HOLD** button right after program starts. (If **M6** is enabled, wait until the **Tool Length Offset** function completes first).

Advance the program using the **STEP** button until the tool approaches the stock as shown below. Notice that the tool is above the top surface of the stock. This was accomplished in STEP 11 for defining the BORDERS of the tool movement.



It is important to attend the Pocket NC as it begins an untested program.

> ⚠️ Be prepared to press the red **E-STOP** button if the program behaves in an unplanned way to prevent damage to your stock, your tool, your vise, or the Pocket NC.

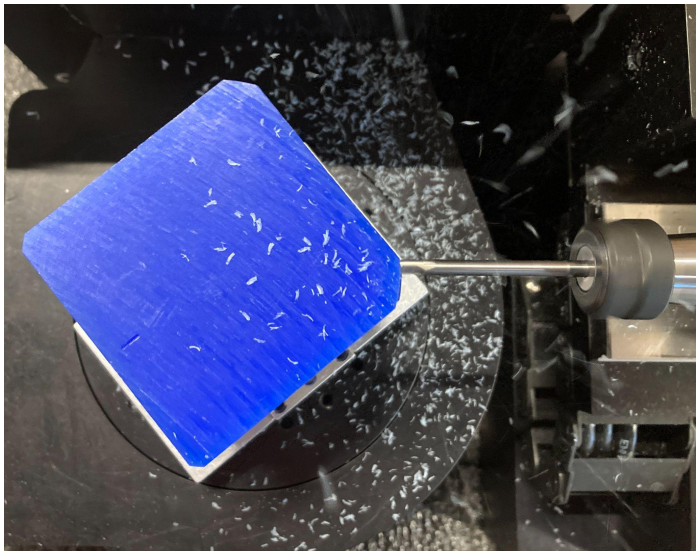
Resume the program by clicking the **CYCLE/START** button on the **PRODUCTION PAGE**.



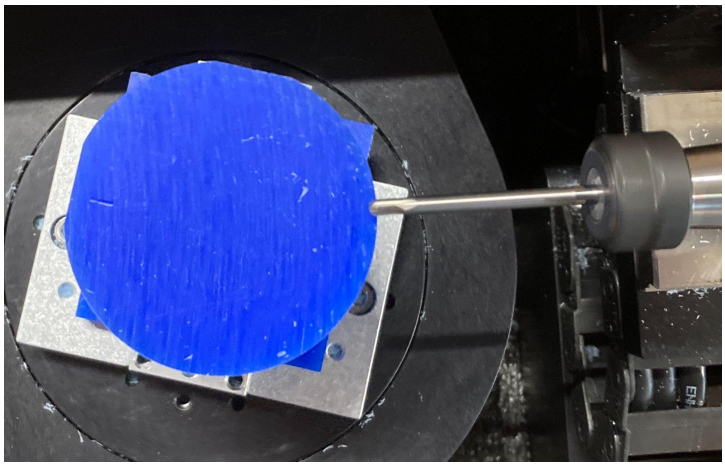
Section 16: Rotary milling process on Pocket NC V2-10

The Desk*Proto* g-code begins by removing the corners of the cube stock. The initial corner removal might not be symmetric since the size of the cube stock was an approximation.

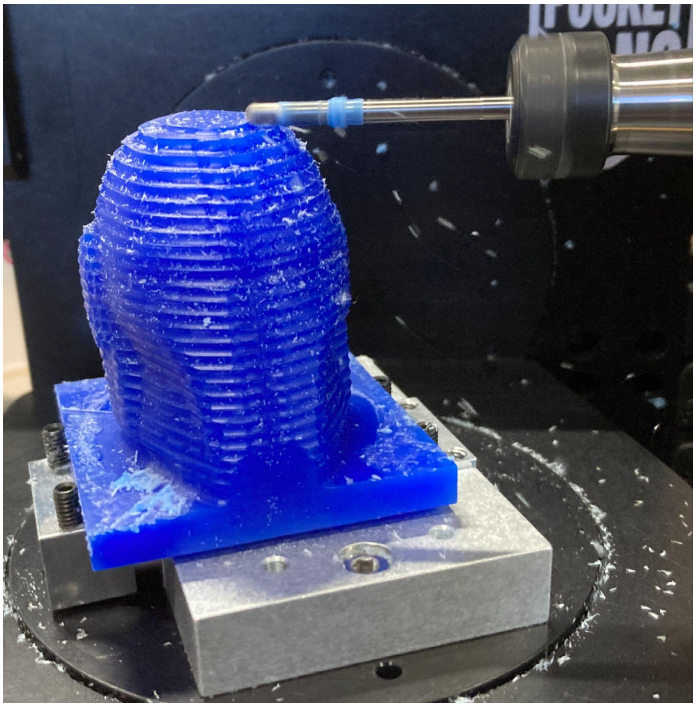
Trimming Corners



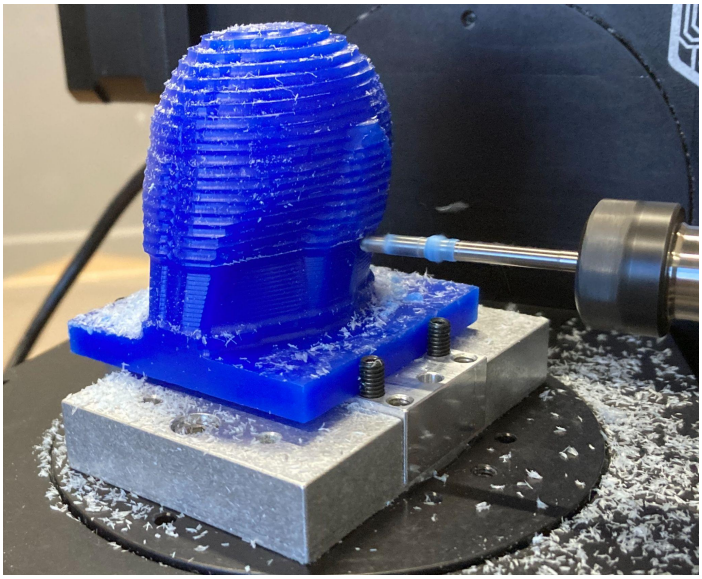
Rounding the Cube Stock



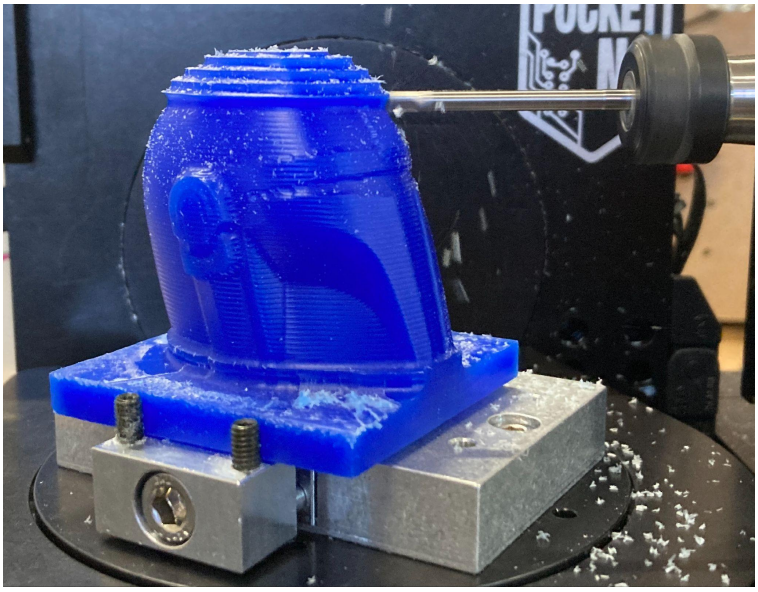
Roughing Cut



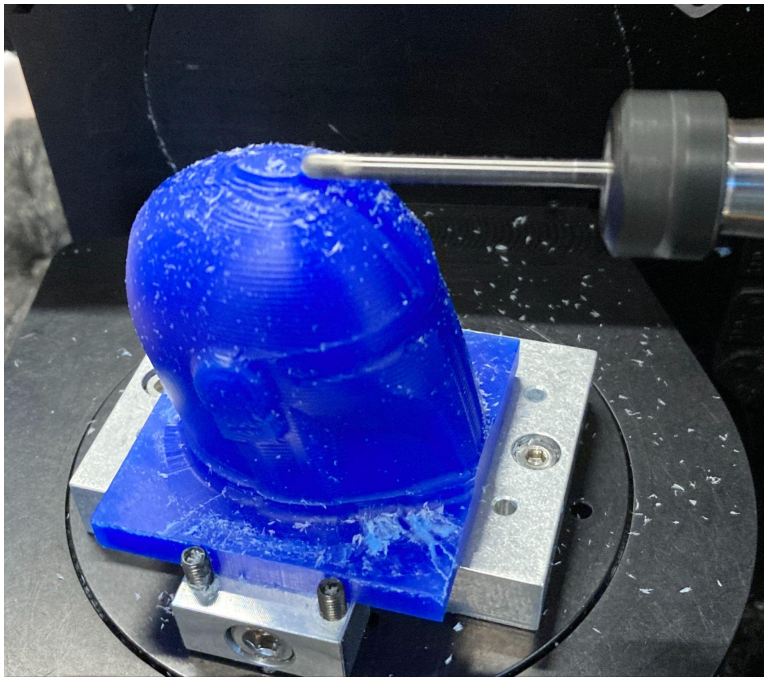
Beginning the Finishing Cut



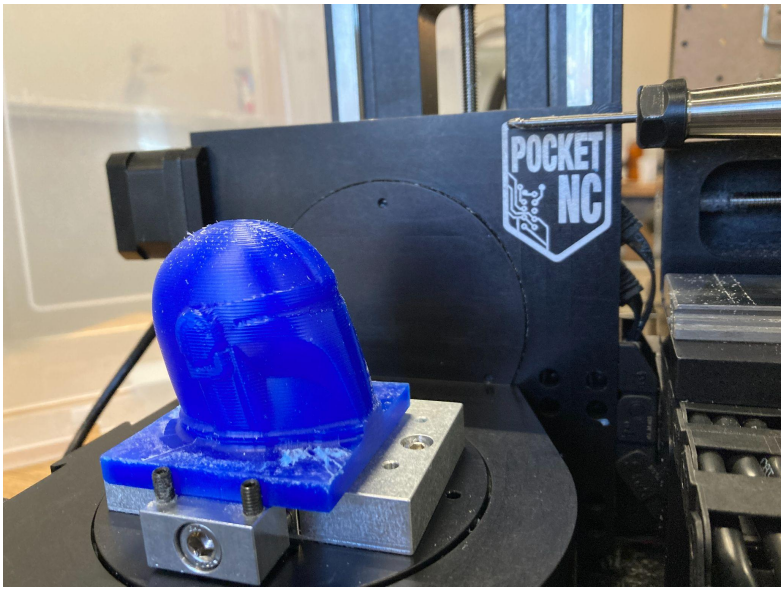
Finishing Cut

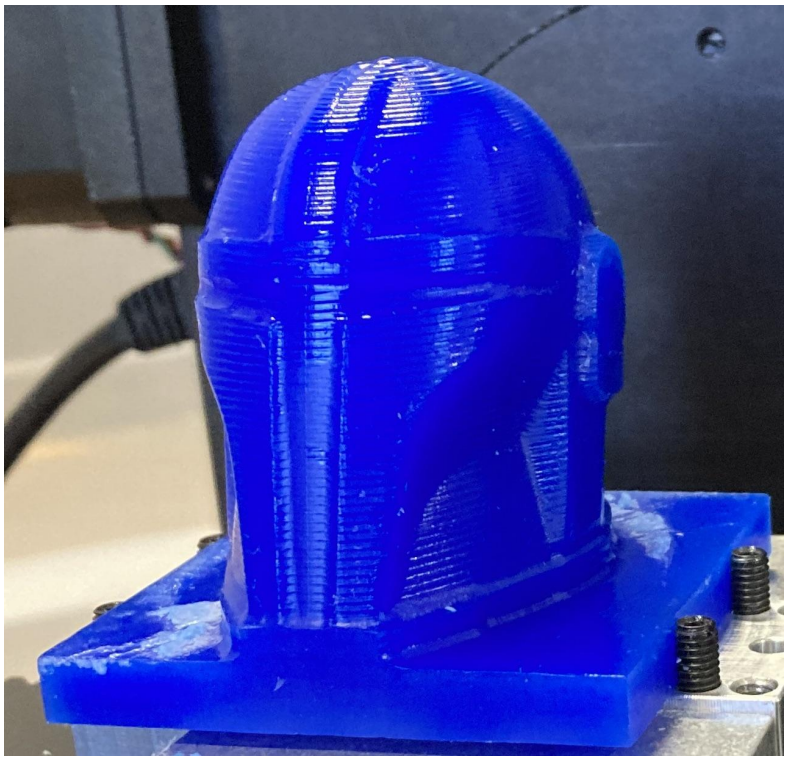


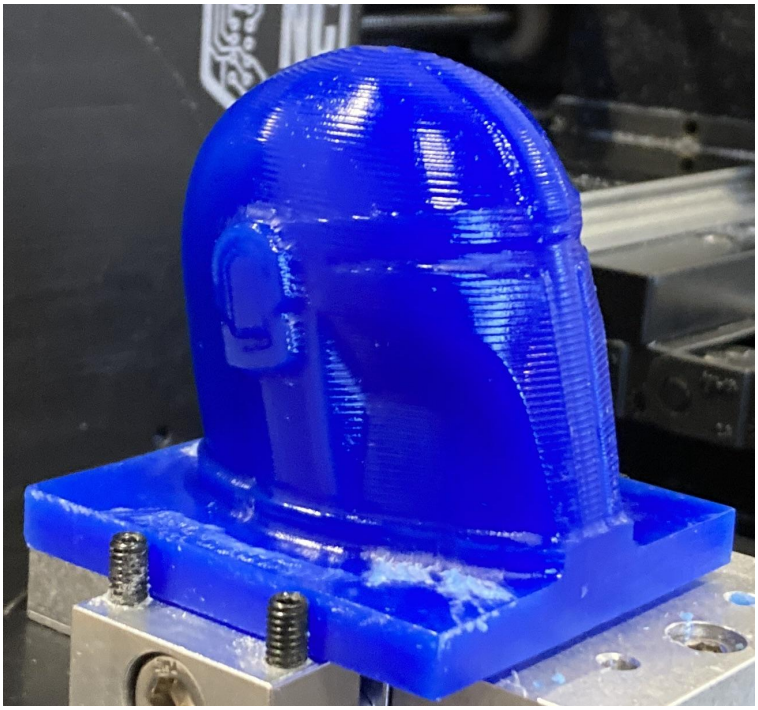
Removing Remaining Skin

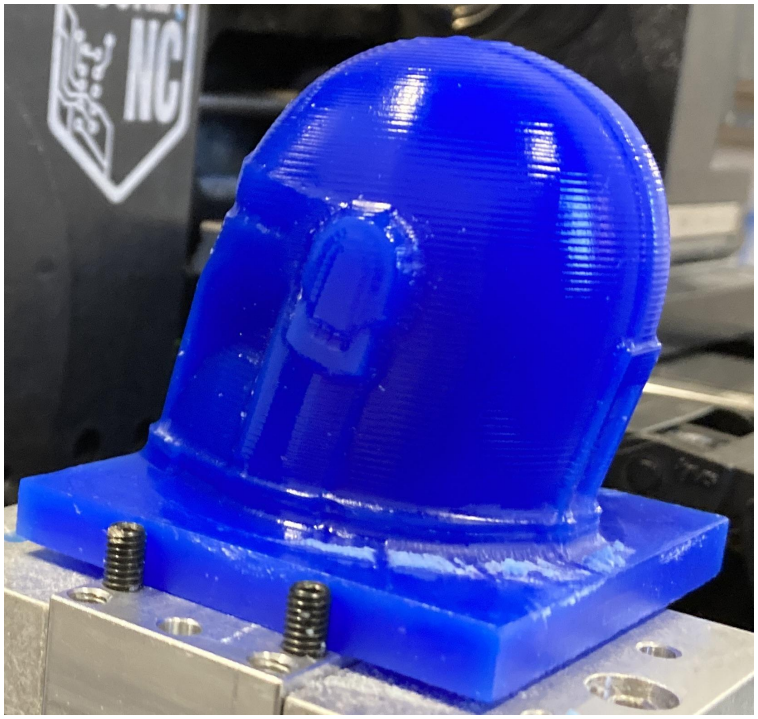


Finished Mandalorian Helmet









Conclusion

The purpose of this tutorial was to familiarize you with the functionality of the Desk*Proto* CAM software and how to create rotary tool operations that work with the Penta Machine Company Pocket NC 5 Axis Milling Machine. A more refined part can be created by reducing the distance between toolpaths (stepover) and the stepsize along toothpath settings (smoothing). This tutorial was created to provide experience with the Desk*Proto* CAM software and the Pocket NC milling machine within a reasonable cutting time