V2 Series - Desk*Proto* Index Tutorial

V2 Series - DeskProto Index 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 index (3+2) tool paths
  - Section 7: Defining the A90B0 Roughing Tool Operations
  - Section 8: Defining the A90B0 Finishing Tool Operations
  - Section 9: Defining the A0B0 Roughing Tool Operations
  - Section 10: Defining the A0B0 Finishing Tool Operations
  - Section 11: Defining the A0B90 Roughing and Finishing Tool Operations
  - Section 12: Defining the A0B180 Roughing and Finishing Tool Operations
  - Section 13: Defining the A0B270 Roughing and Finishing Tool Operations
  - Section 14: 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: Index milling process on Pocket NC V2-10
- Conclusion

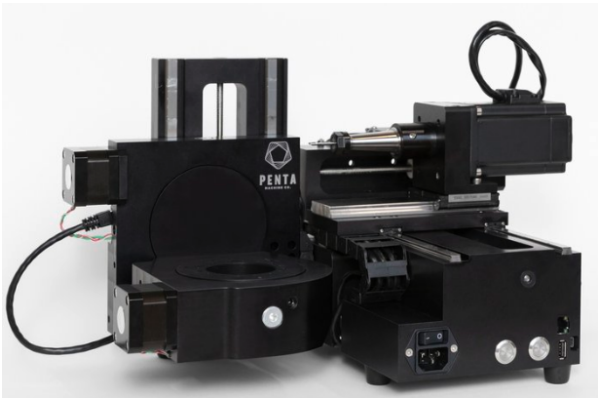
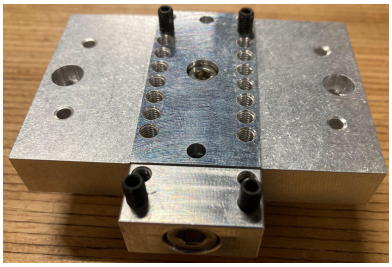
Introduction

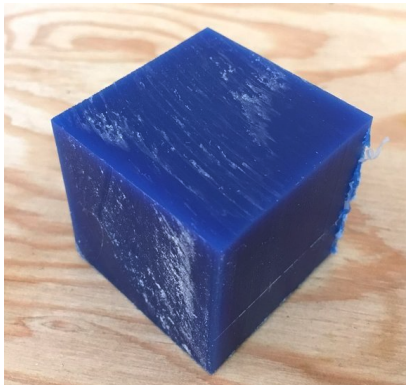
The purpose of this tutorial is to demonstrate how to program index (3+2) toolpaths in Desk*Proto*, and how to set up and cut a 5 axis part on your Pocket NC V2-10 or V2-50. **This tutorial uses the Kinetic Control user interface released on 11/1/2021**. The part and tutorial are geared toward beginners. More experienced users might also find value from the tips provided throughout this tutorial. The tutorial walks through how to correctly set up indexed 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*.

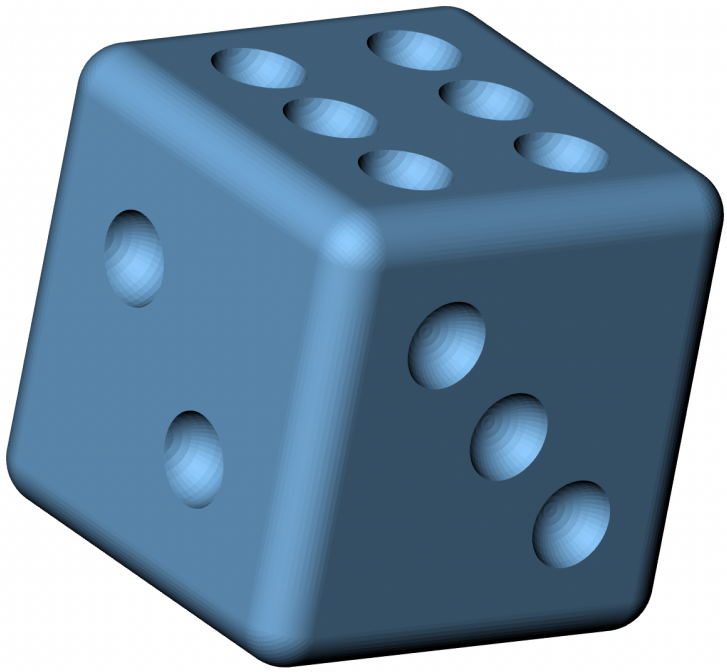
Supplies Needed

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 die courtesy of Andrew Serra. For this tutorial, we will use the *dice.stl*.



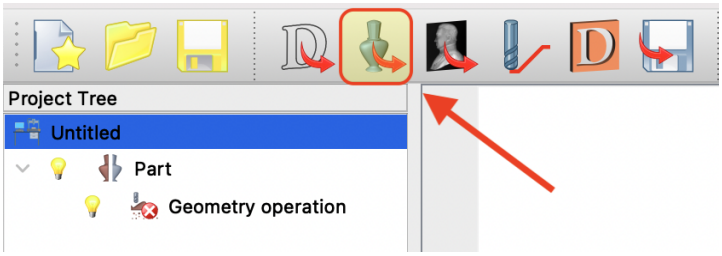
Section 2: Loading a geometry file in DeskProto

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

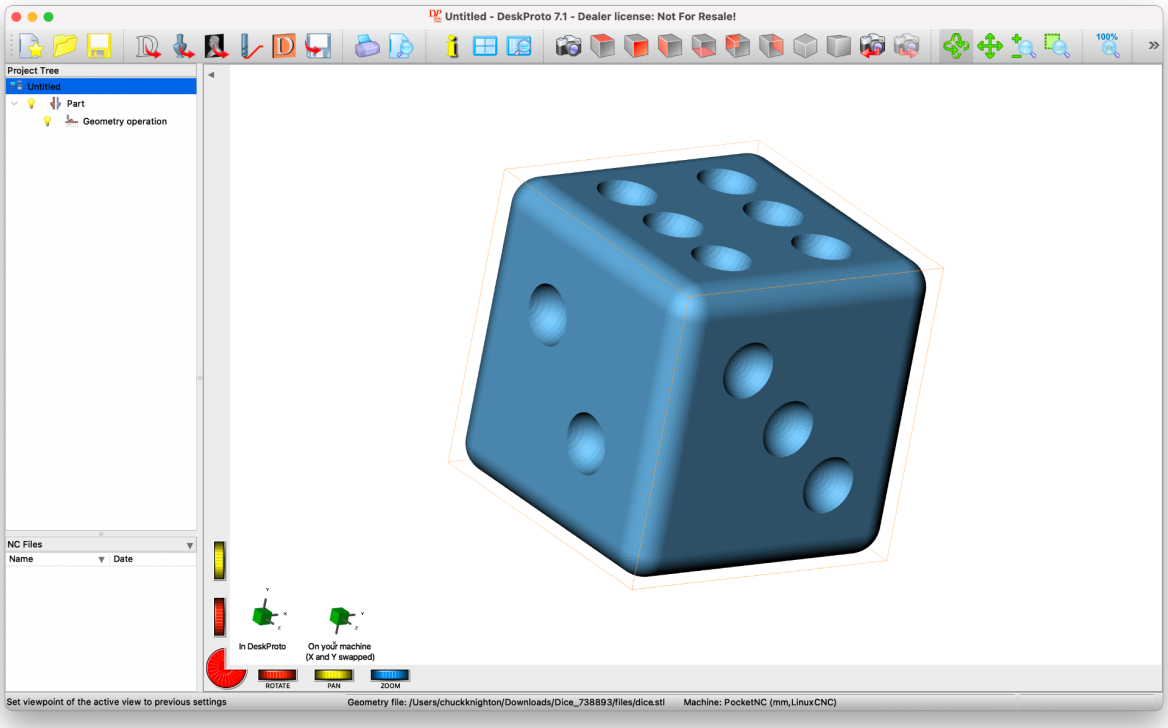
Desk*Proto* has a **Start new project** wizard option. For the purpose of this tutorial, the setup will be performed manually. Click the red X button to close out the following window.



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

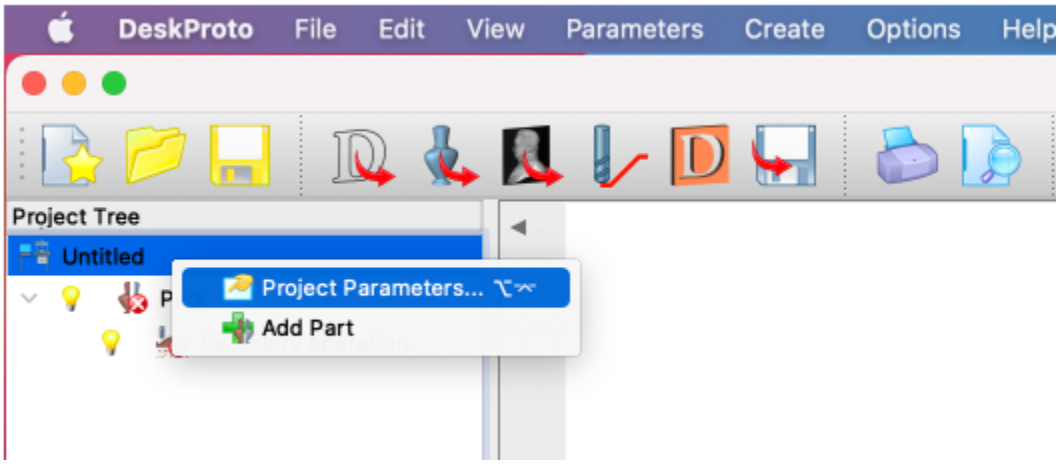


⚠️ View was rotated to display dice part as shown.

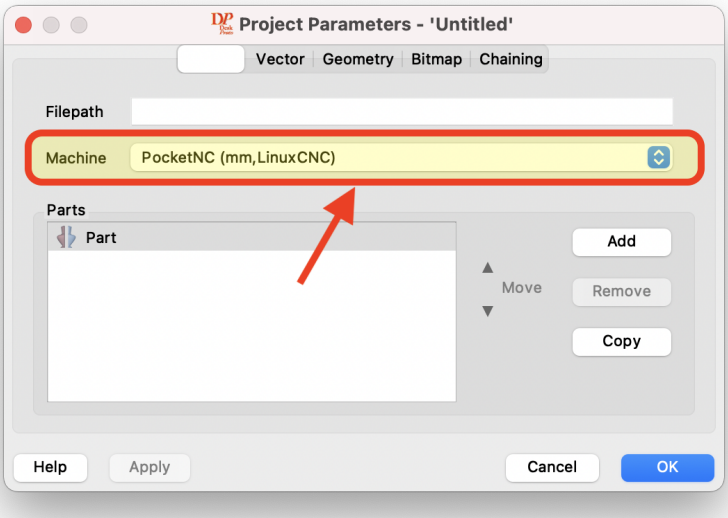


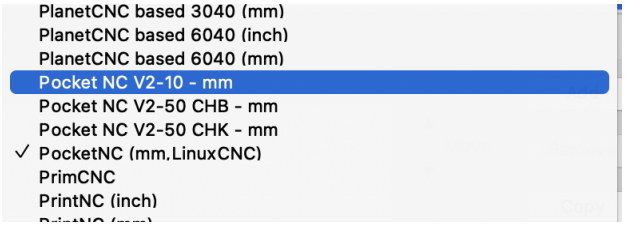
Section 3: Configure Desk*Proto* to use a Pocket NC

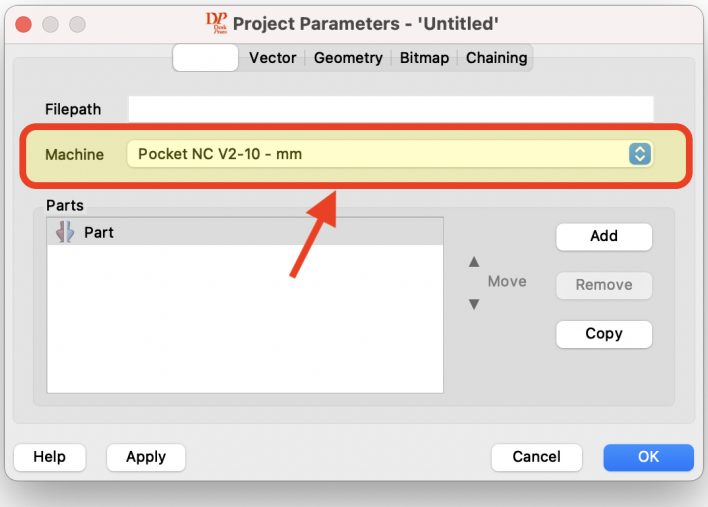
Edit the **Project Parameters**.



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).





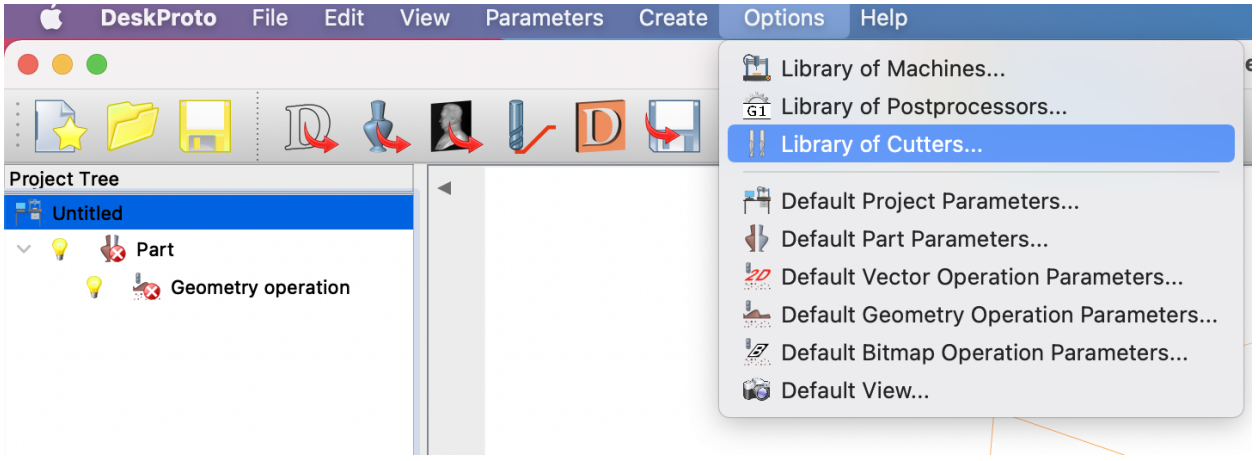


Click the **OK** button and return to the Desk*Proto* main screen.

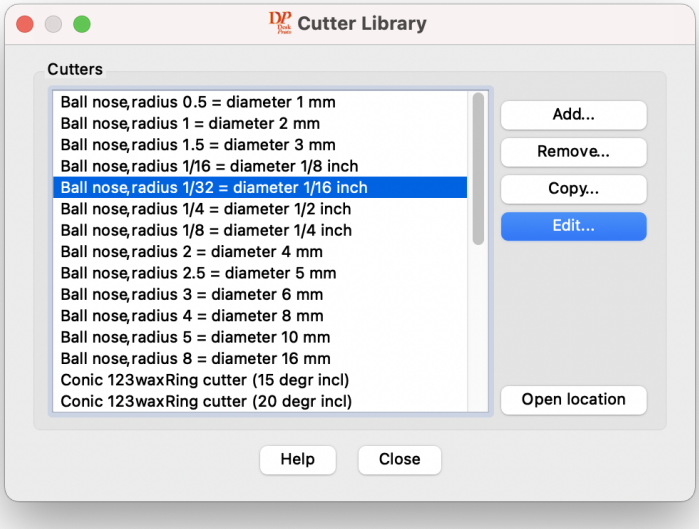
Section 4: Configuring your tool settings in Desk*Proto*

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.

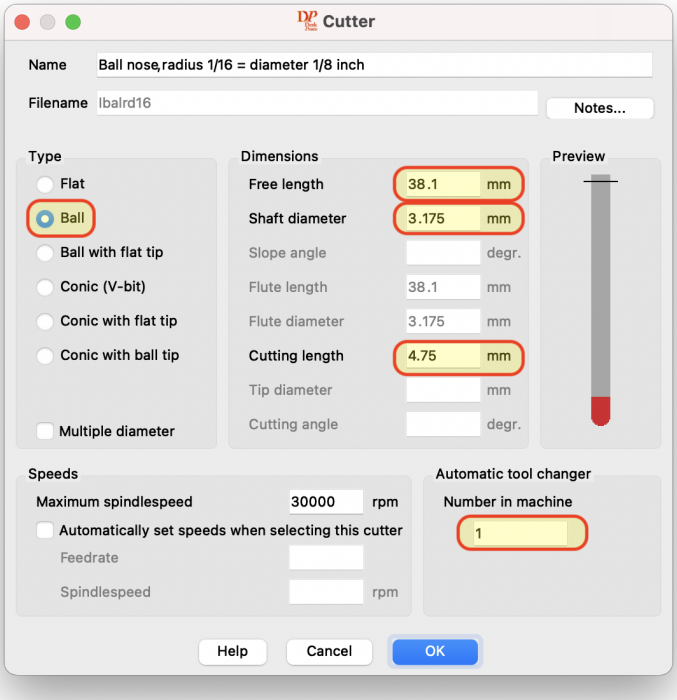
Click on the **Options** tab and select ***Library of Cutters…***.



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



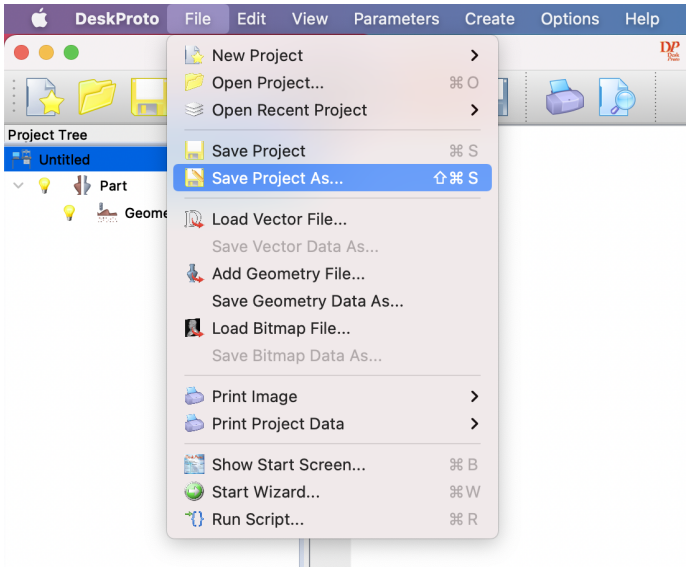
Program the tool parameters as shown below. Make sure that the ***Number in machine*** is set to **1** under the **Automatic tool changer**.



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 ***Dice.dpj***.



Section 6: Setup Desk*Proto* to calculate index (3+2) tool paths

The dice part has five machinable surfaces (right, front, left, back, and top). For the purpose of this tutorial, the dice will remain attached to the stock. It is only necessary to machine to a depth that extends just past the “chamfer” along the edge of the dice for each side.

The five machinable surfaces will each have their own roughing and finishing tool operations. The roughing tool operations remove the majority of the stock around the part while the finishing tool operations remove stock above the part to the desired surface finish.

For the purpose of this tutorial, the five machinable surfaces will be referred to as the rotation angles of the Pocket NC’s rotary axis. For instance, the top surface (with the 6 dice pattern) will be referred to as A90B0. To mill the top of the part, the A axis needs to be rotated 90° so that the tool is perpendicular to this surface.

Below is a summary of the five parts that will be created.

1. Top (dice number 6) - A90B0
2. Right (dice number 3) - A0B0
3. Front (dice number 2) - A0B90
4. Left (dice number 4) - A0B180
5. Back (dice number 5) - A0B270

The bottom surface with dice number 1 will remain attached to the stock.

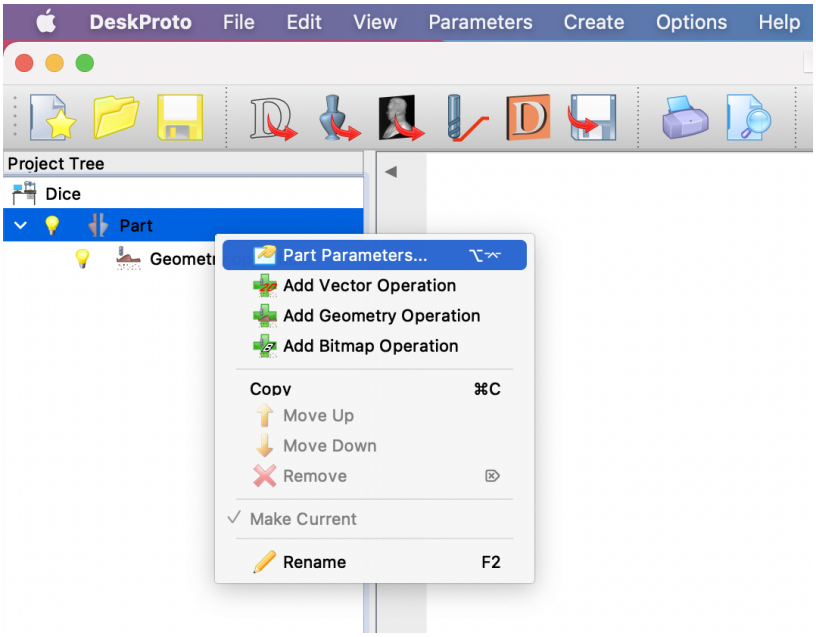
⚠️ Five separate g-code files will be created, one program per side. If a vise that could center the stock at the Pocket NC V2’s center of rotation were used, then only one g-code program would be needed.

This tutorial is an overview of how to use the Pocket NC V2 along with the Pocket NC Custom Vise that is included with the machine. The Pocket NC Custom Vise does not center the stock along the Pocket NC V2’s center of rotation. To overcome this situation, Penta Machine Company has embedded TCPC (Tool Center Point Control) into its simulator. TCPC conveniently allows work offset touch offs to locate the stock for the Kinetic Control user interface. Kinetic Control interprets the TCPC commands and performs the translation necessary to keep the tool traveling correctly over the stock as the rotary axes move.

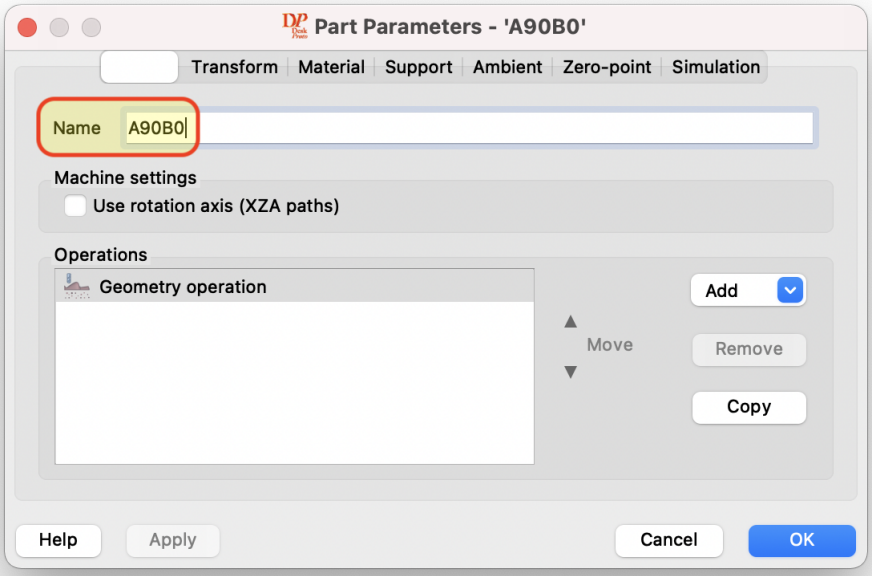
The TCPC conversion process in the Penta simulator requires selecting the correct code to convert to TCPC. While this can be accomplished on one file containing all the tool operations to complete the part, the method becomes much more complex for this simple tutorial.

You are encouraged to investigate Desk*Proto*’s **Chaining** function found under the **Project Parameters**. Chaining allows control over the sequence of the tool operations. For instance, all the roughing tool operations could be put in one chain, performed first, and all the finishing tool operations in another chain and performed last. This would be valuable if the finishing tool were different from the roughing tool, reducing the number of tool changes.

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

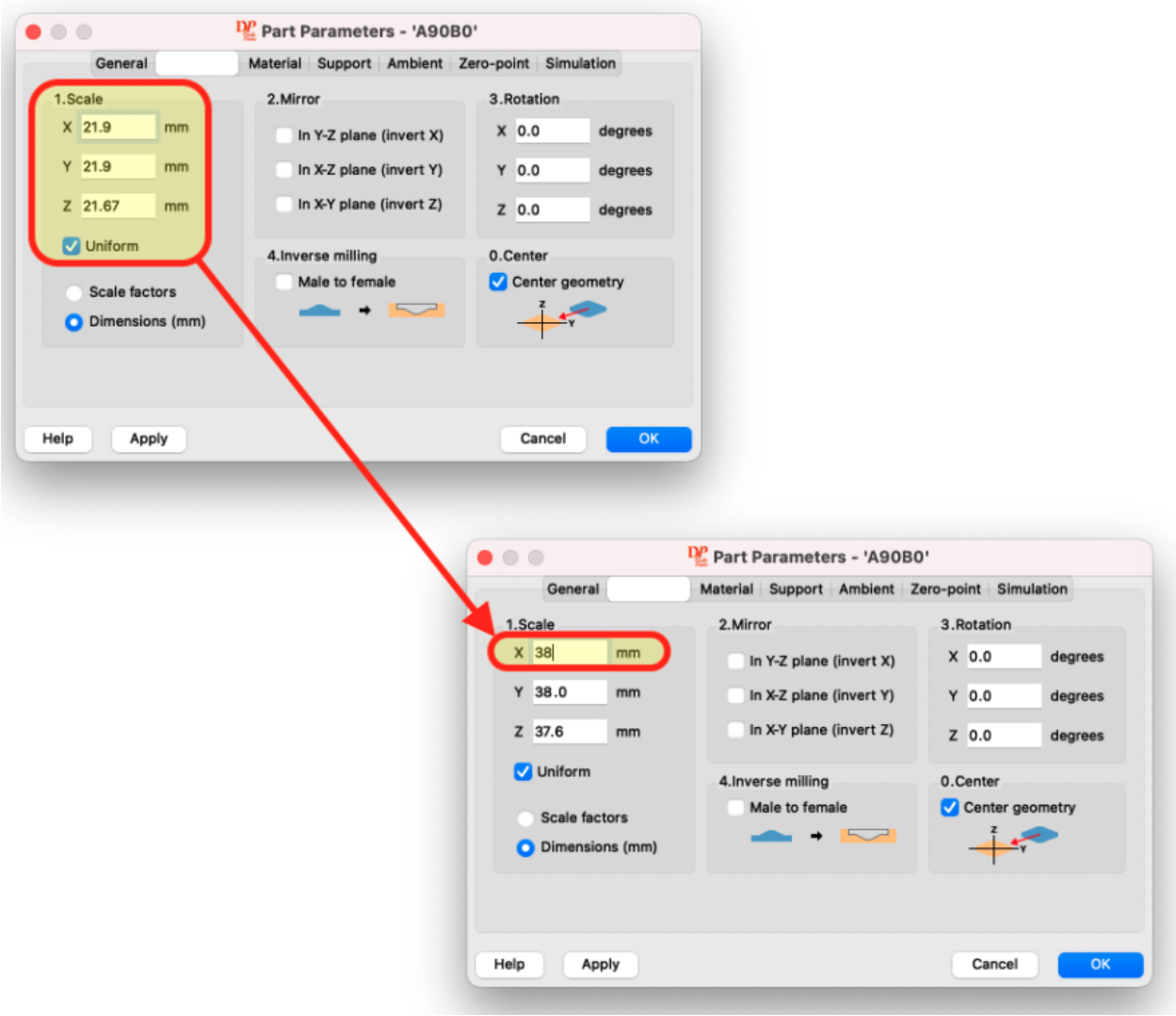


Rename the **Part** to **A90B0**.



Scale the part to fit within the boundaries of the stock. While you are still editing the **Part Parameters…**, select the **Transform** tab.

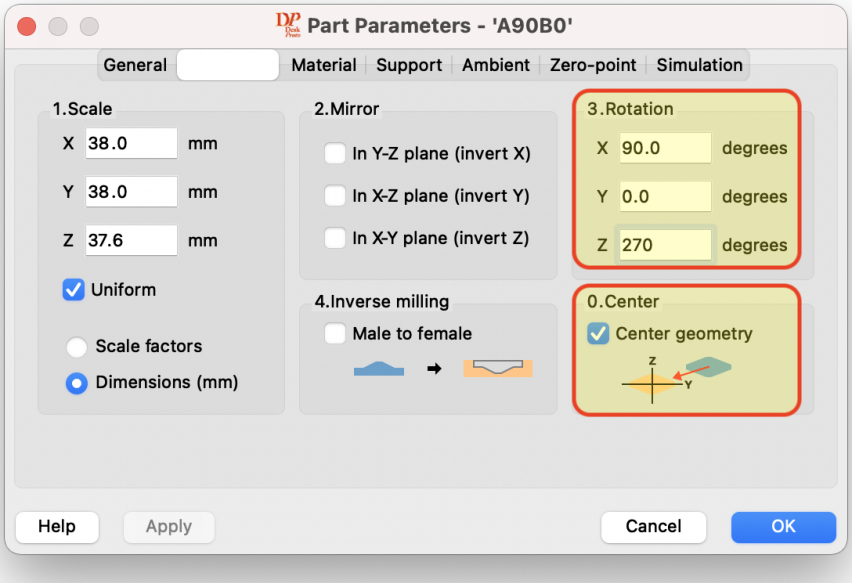
Under **Scale**, select **Dimensions** to review the size of the original part. Make sure the **Uniform** checkbox remains checked. In the case of the dice part, the X and Y dimension of the part are the larger values. To maximize the size of the finished part, the X value can safely be adjusted to 38 mm. Click the **Apply** button to save the setting.



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.

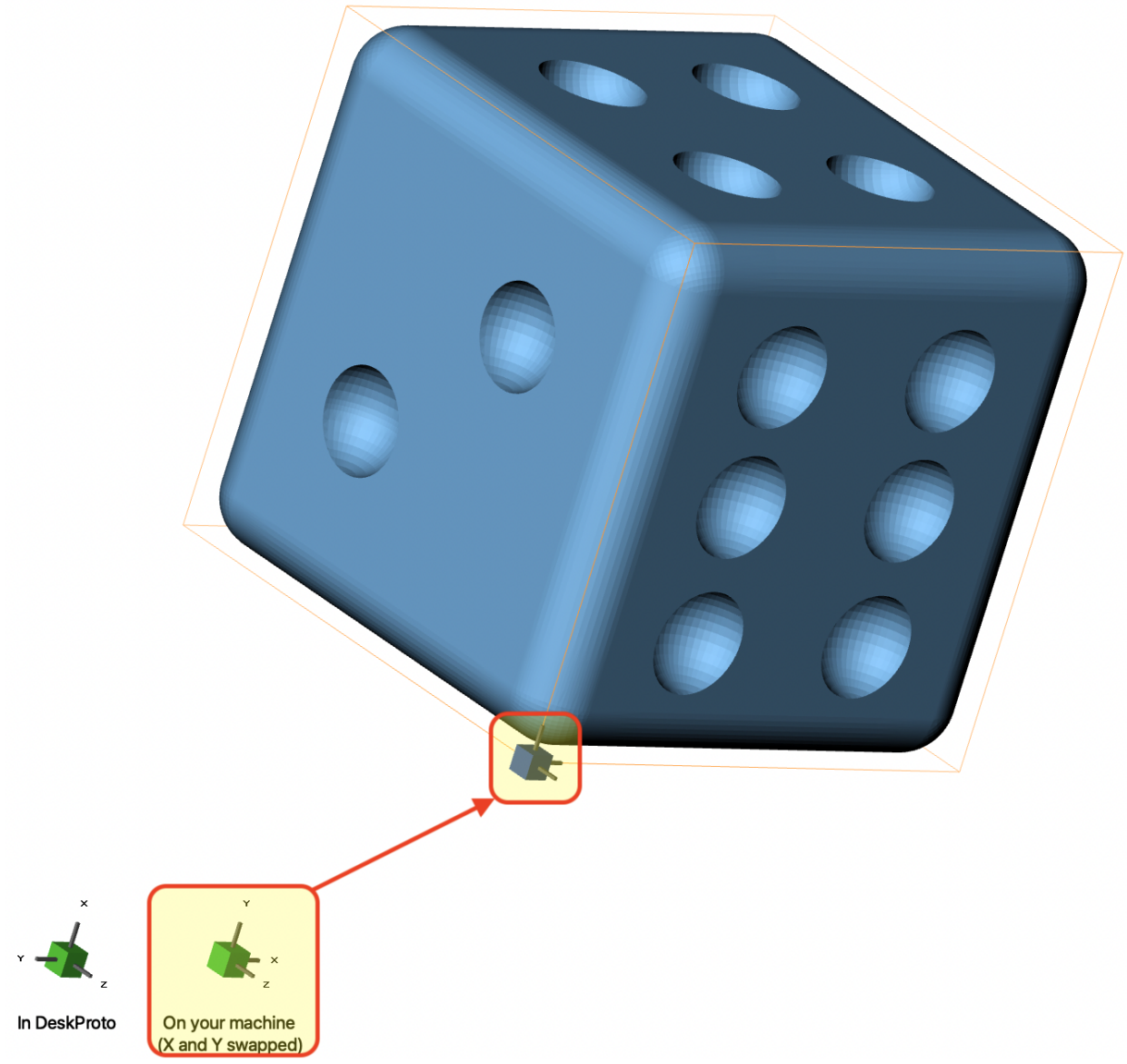
⚠️ Selecting **Center Geometry** configures Desk*Proto* to calculate toolpaths around the exact center of the part.

The next step is to orient the work coordinate system so that the Z axis for the top surface is in line with the tool . This can be accomplished by adjusting the **Rotation** values under the **Part Parameters Transform** section as shown.



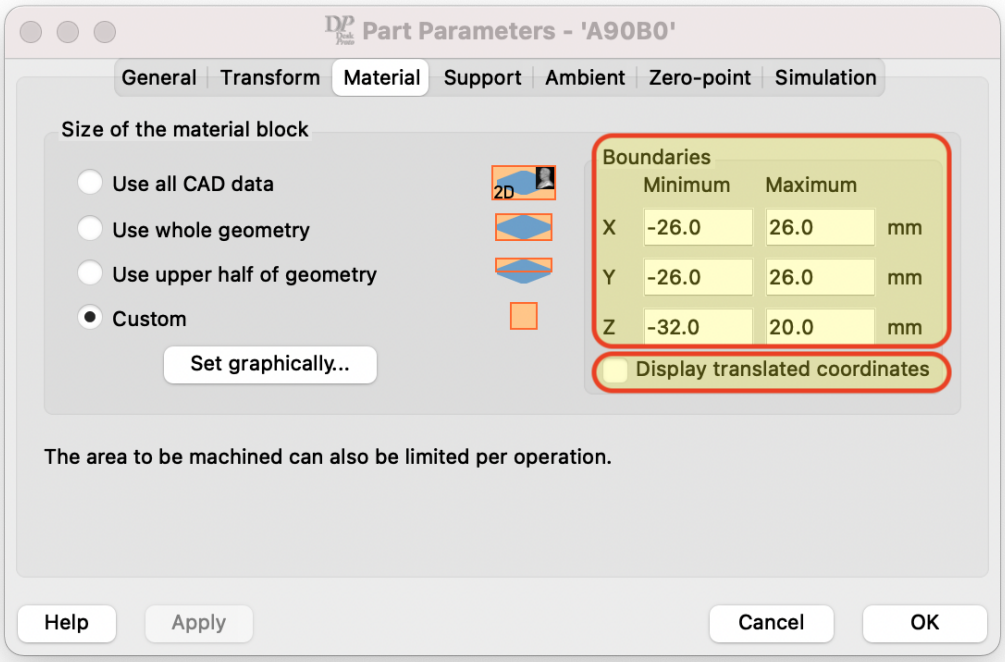
For this tutorial, change the ***Rotation*** to ***X=90***, ***Y=0***, ***Z=270***. Click the **Apply** button.

Notice that the area to be machined rotates around the Pocket NC Y axis and the tool approaches the part from the correct orientation.



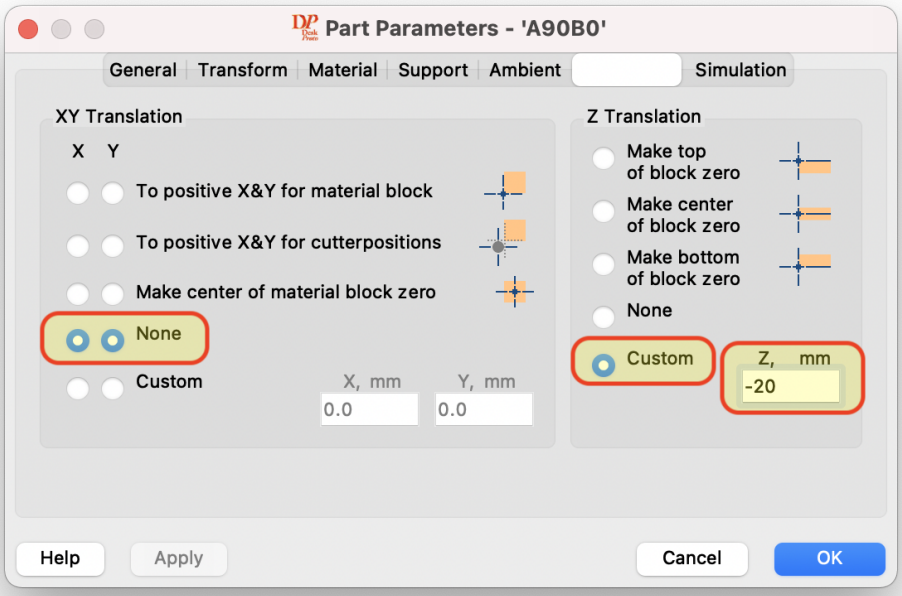
The stock material used for this tutorial is approximately 52 mm per side. Select the **Material** tab under **Part Parameters**. Under **Size of the material block** choose ***Custom***. Under the **Boundaries** section, deselect the ***Display translated coordinates*** and adjust the **Boundaries** values as shown. The ***Z* 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.

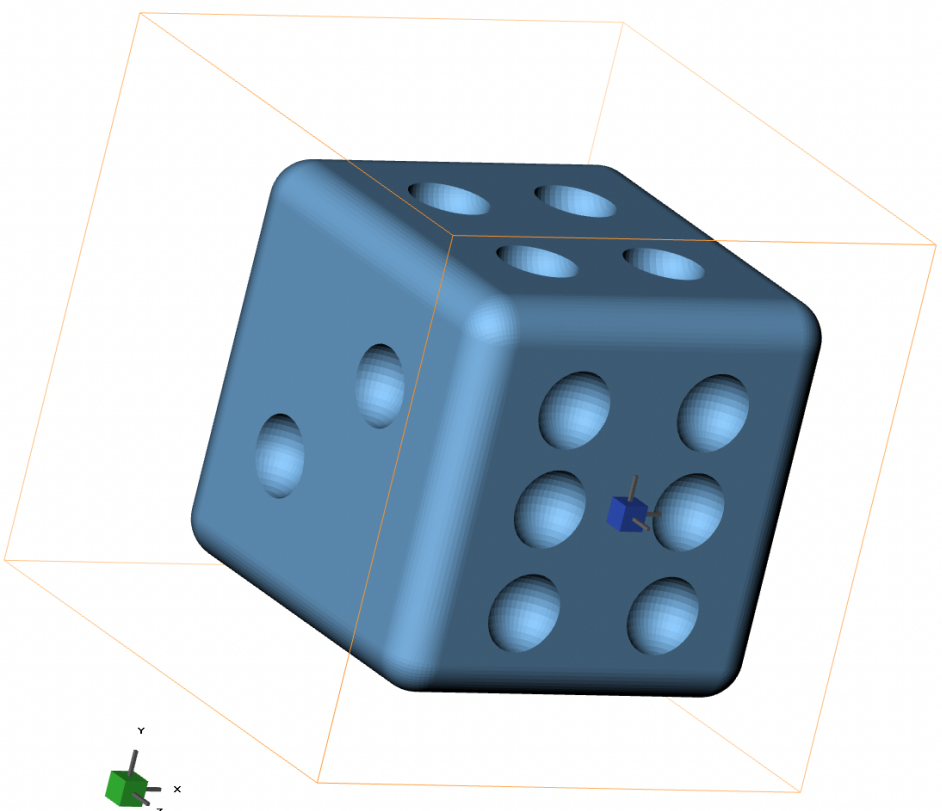
⚠️ The ***X***, ***Y***, and ***Z* Boundary** values relate to the Desk*Proto* CAM coordinates.

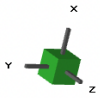


The final step is to select where on the stock to place the work coordinate system zero point. In this tutorial, since we have chosen to mount the stock with the Pocket NC Custom Vise, the most convenient location is the top-center of the stock. This choice of zero-point will become obvious in the next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

Select the **Zero-point** tab and set the **Z Translation** to ***Custom*** and set the ***Z*** value to -20 mm, and the **X Translation** and **Y Translation** to ***None***.





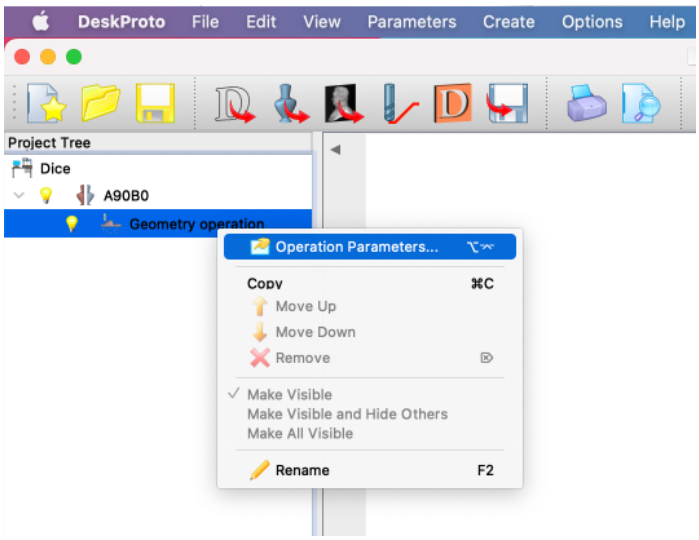


In DeskProto

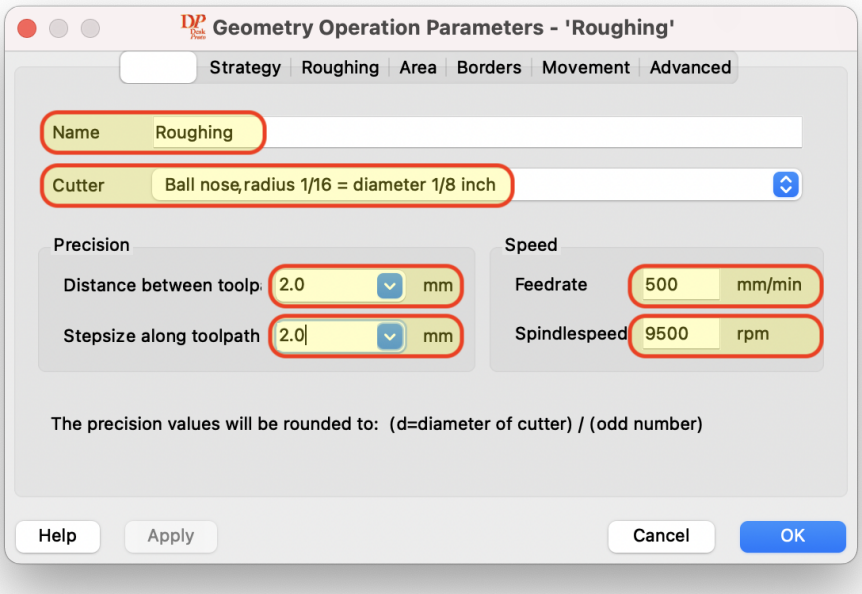
On your machine
(X and Y swapped)

Section 7: Defining the A90B0 Roughing Tool Operations

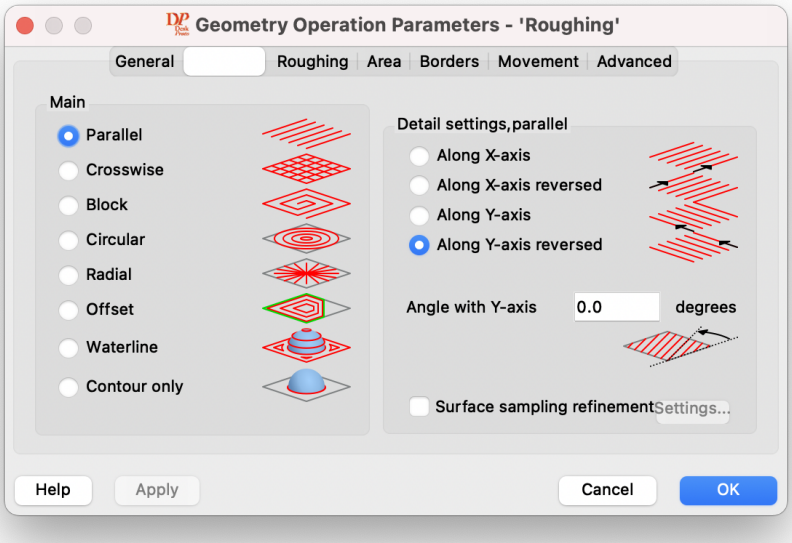
Edit the **Geometry Operation Parameters**.



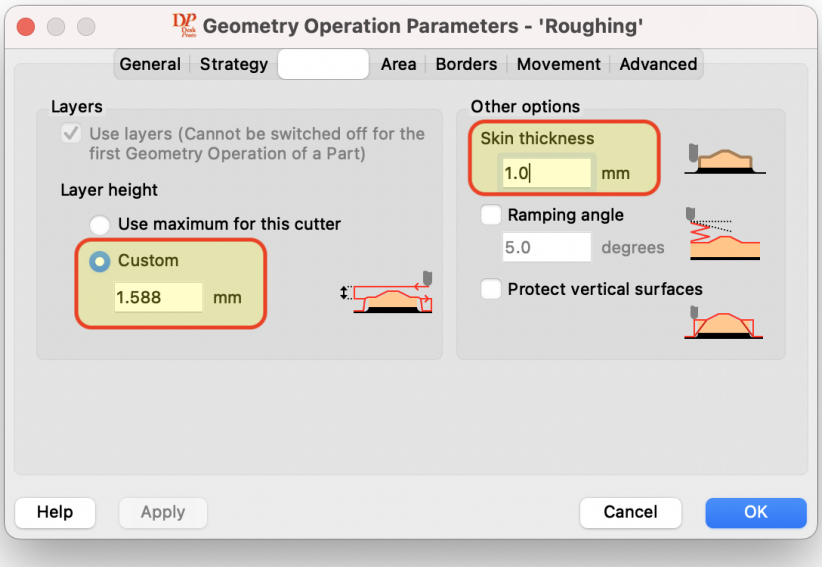
Rename the operation ***Roughing***. Select the ***Cutter*** (tool) to be at ***Ball nose, radius 1/16 = diameter 1/8 inch***. Under the **Precision** section, select both the ***Distance between toolpath*** and ***Stepsize along toolpath*** to be **2.0 mm**. These are slightly conservative values for 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.



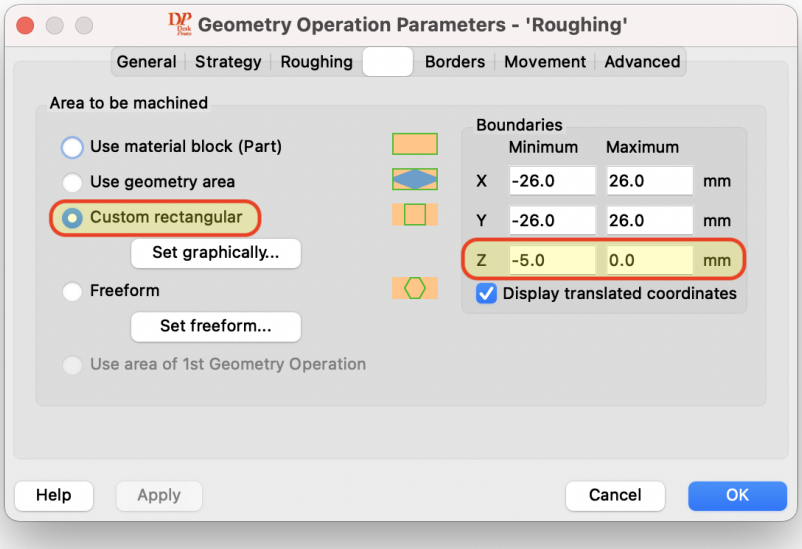
Select the ***Strategy*** tab and select ***Parallel*** tool path ***Along Y-axis reversed***. This will take a series of cuts beginning at the top of the stock and moving down the Y-axis and ending at the bottom of the part.

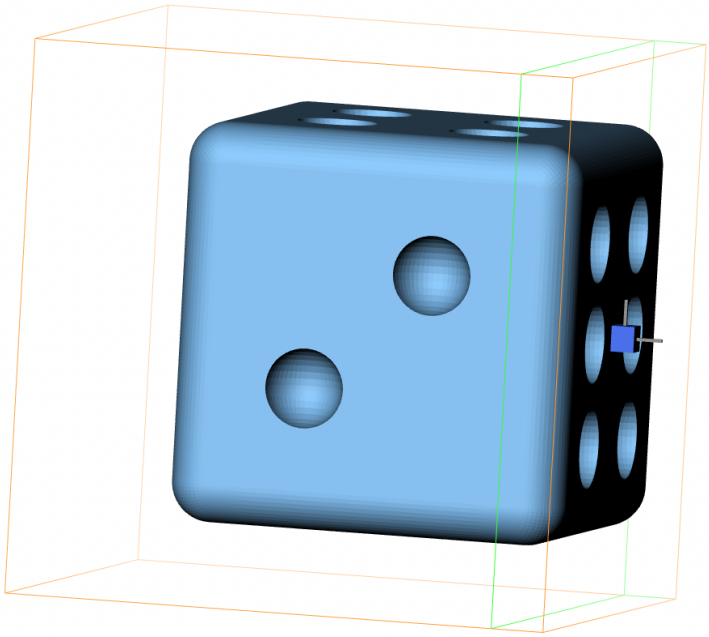


Select the ***Roughing*** tab and select ***Custom Layer Height***. In this case, a 1/8 inch ball nose tool is being used on wax so a depth of cut of ***1.588 mm*** was chosen. A ***Skin Thickness*** of 1 mm was chosen to avoid tool trails showing after completing the finishing operation.

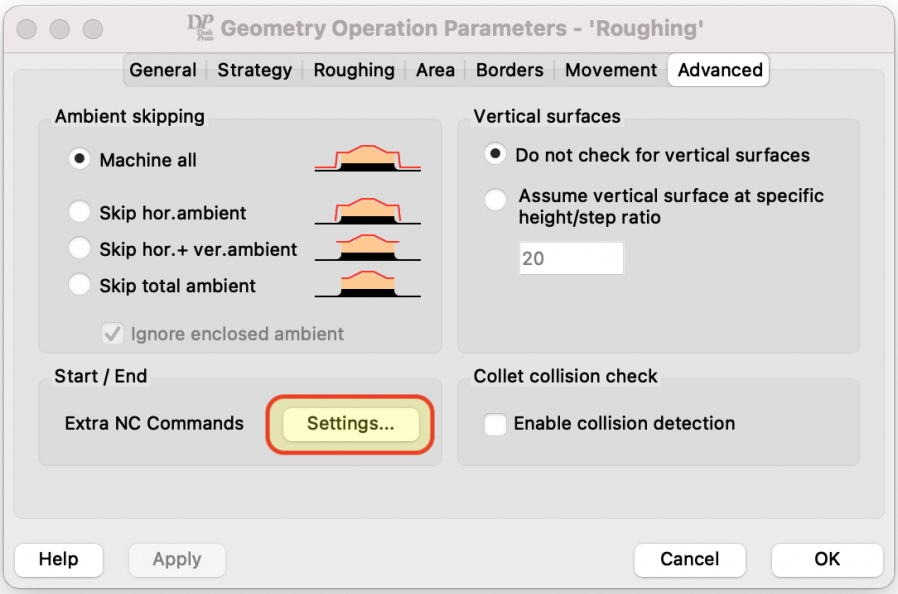


Select the ***Area*** tab and select the **Area to be machined** to be ***Custom***. Adjust the **Boundaries *Z minimum*** value to -5 mm. This defines the tool to cut just below the chamfer on the top side of the dice.





The final step in setting up the roughing operation for the top of the part is to command the rotary axes to their correct rotation angles. This is performed in the **Advanced** tab under the **Start/End** section. Click the ***Extra NC Commands*** settings button to configure the rotary movement.



⚠️ It is important to know where the tool is positioned during any axis move, particularly rotary axes moves, to prevent damage to the tool or the Pocket NC V2.

Desk*Proto* conveniently allows extra NC commands at the beginning and end of a tool operation to safely position the tool and to avoid collisions.

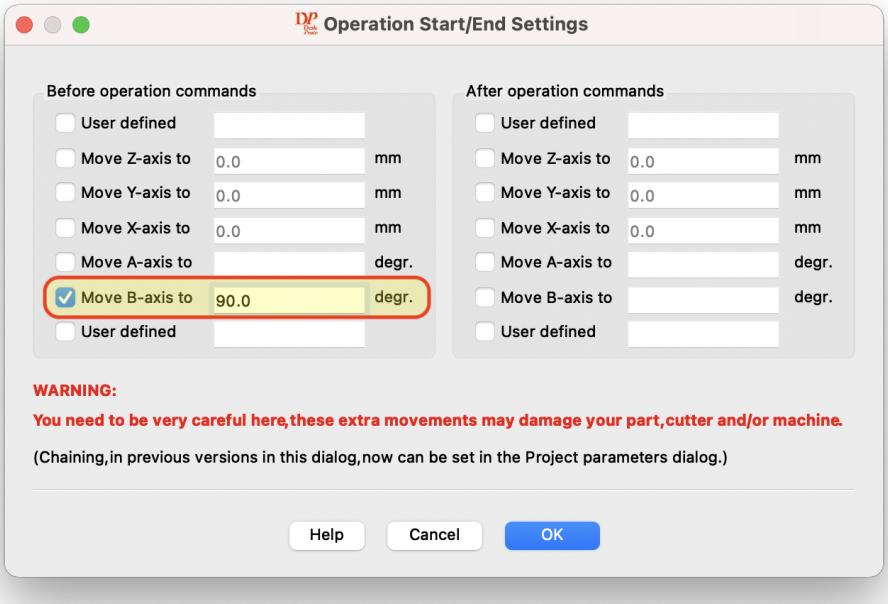
In the **Operation Start/End Settings** shown on the following page, the different axes can be commanded to move to specific locations before and after a tool operation.

Since work offsets will be used to locate the stock on the machine, the tool operation code will be using the work offsets (G55 in the case of this tutorial). This means that the **Operation Start/End Settings** commands will be relative to the work offset settings. Moves using work offsets are hard to predict and can even result in limit move errors for the different axes.

The safest tool position for the Pocket NC V2 is when the Z axis remains in its home position. The Pocket NC V2 is commanded to the home position at the beginning of each program. The settings in this tutorial use the home position as the safe position for the tool and keep the tool from moving away from the Z home position until the rotary axes have completed their initial moves.

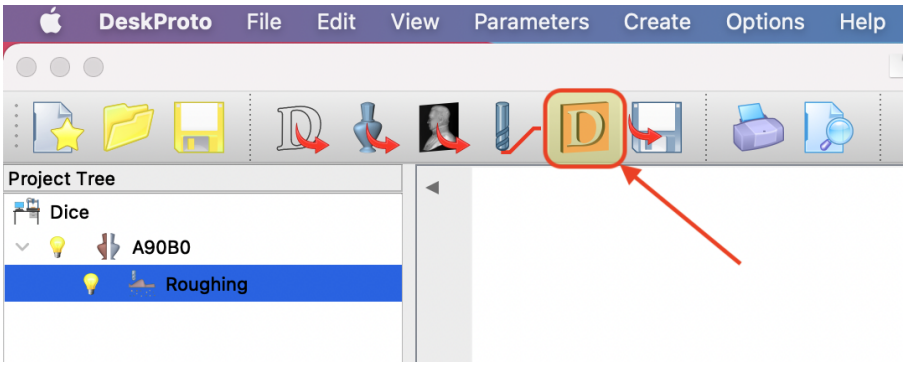
If work offsets were not being used and the stock was centered on the machine's center of rotation, then using the **Operation Start/End Settings** can be safely used to position the tool at a point of your choice.

⚠️ Adjusting the ***A-axis*** moves the **B rotary axis** of the Pocket NC.

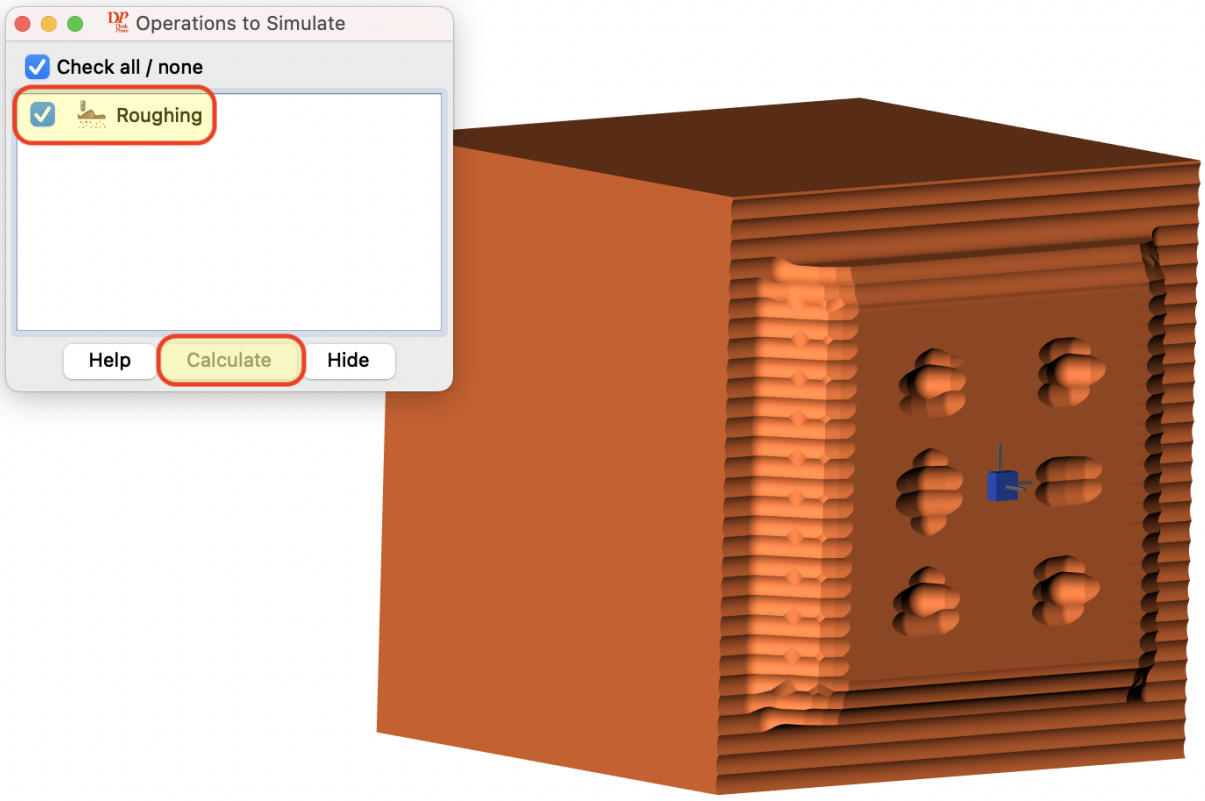


Notice that before the **A90B0** roughing operation begins, the A axis is commanded to rotate 90°. By default the B axis begins at 0°.

Desk*Proto* can simulate the roughing tool operation. Click on the **Desk*Proto* Simulation** icon.



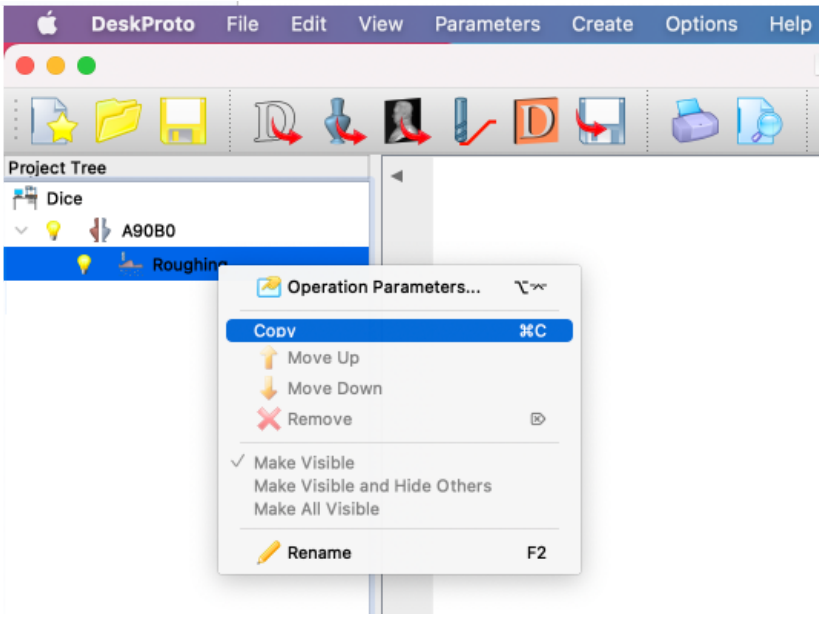
Check the ***Roughing*** operation and click the ***Calculate*** button.



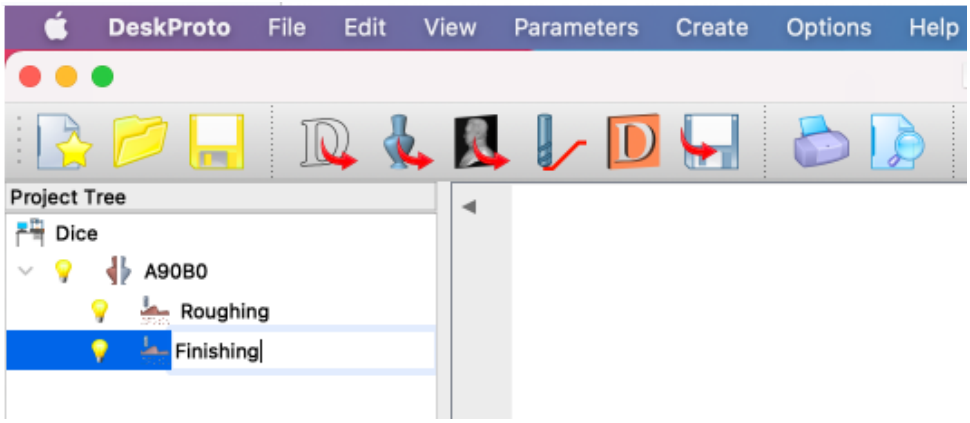
Notice that the roughing cut is only used to quickly remove the stock material. A finishing operation will be added to refine the finished surface.

Section 8: Defining the A90B0 Finishing Tool Operations

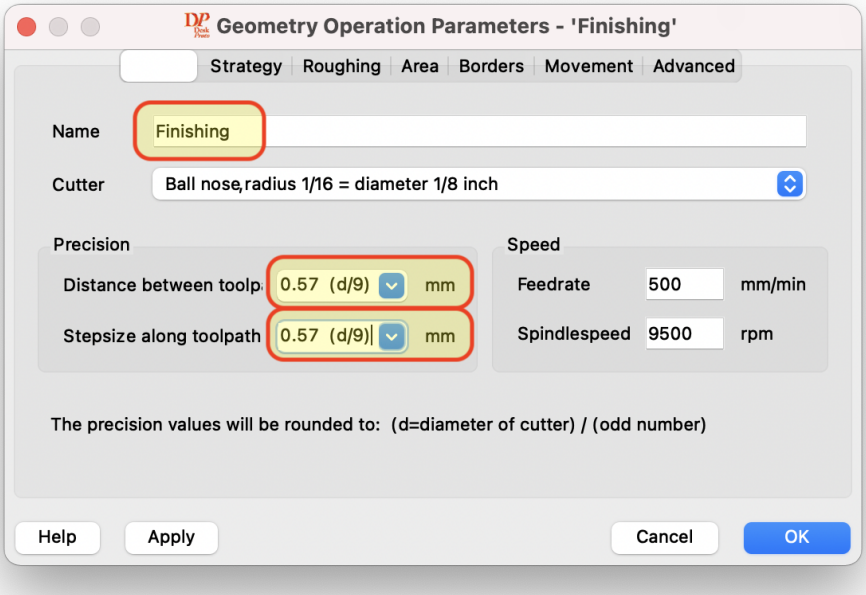
Copy the Roughing Operation.



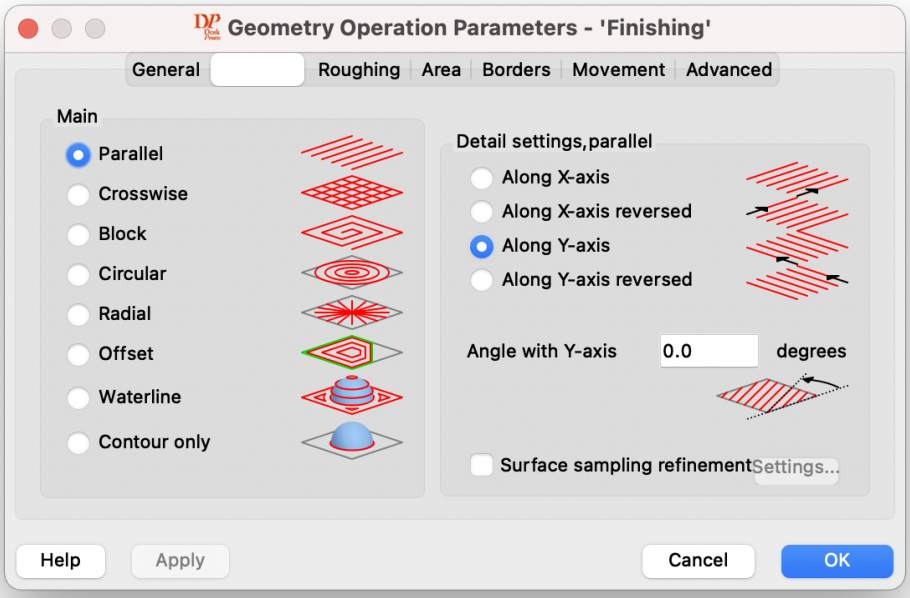
Rename the operation **Finishing**.



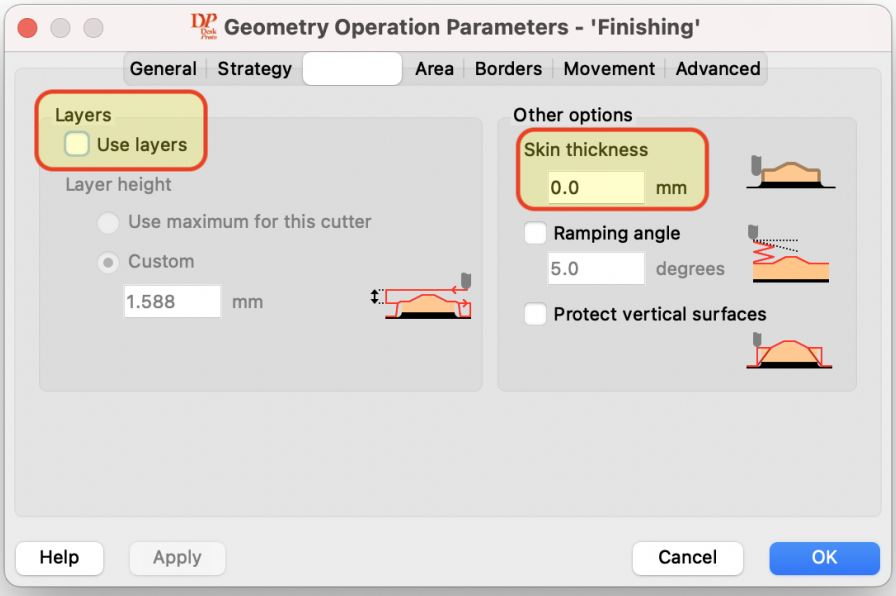
Edit the **Finishing** operation. Under the **Precision** section, select both the ***Distance between toolpath*** and ***Stepsize along toolpath*** to be **0.57 mm** or **(d/9)**.



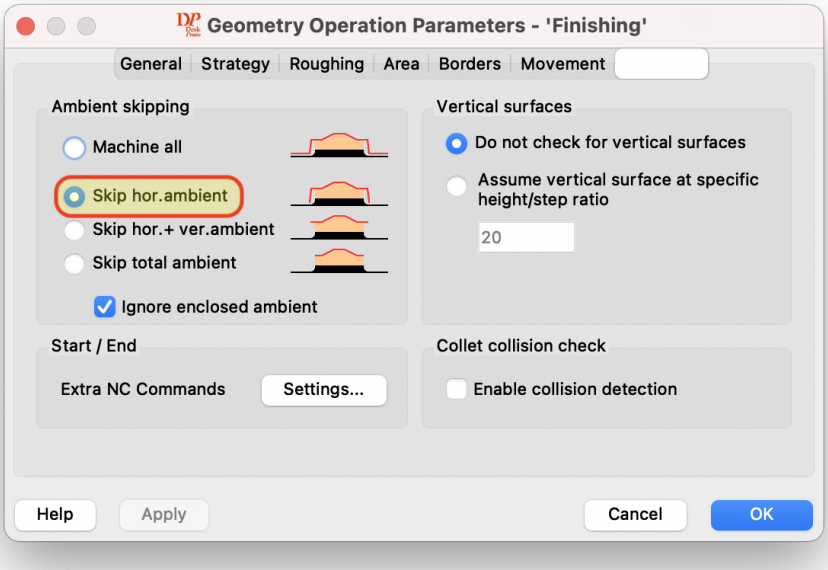
Select the ***Strategy*** tab and select ***Parallel*** tool path ***Along Y-axis***. This will take a series of cuts beginning at the bottom of the stock and moving up the Y-axis and ending at the top of the part.



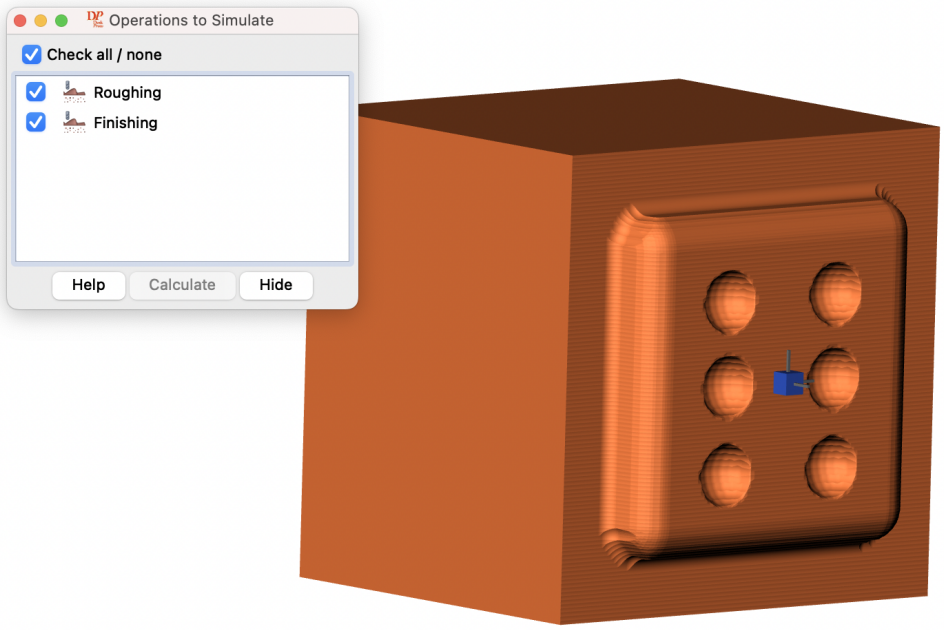
Select the **Roughing** tab. Under **Layers**, uncheck the ***Use Layers*** box. Reduce the ***Skin Thickness*** to 0.0 mm under the **Other Options** section.



Select the **Advanced** tab. Under **Ambient skipping**, select ***Skip hor.ambient***.



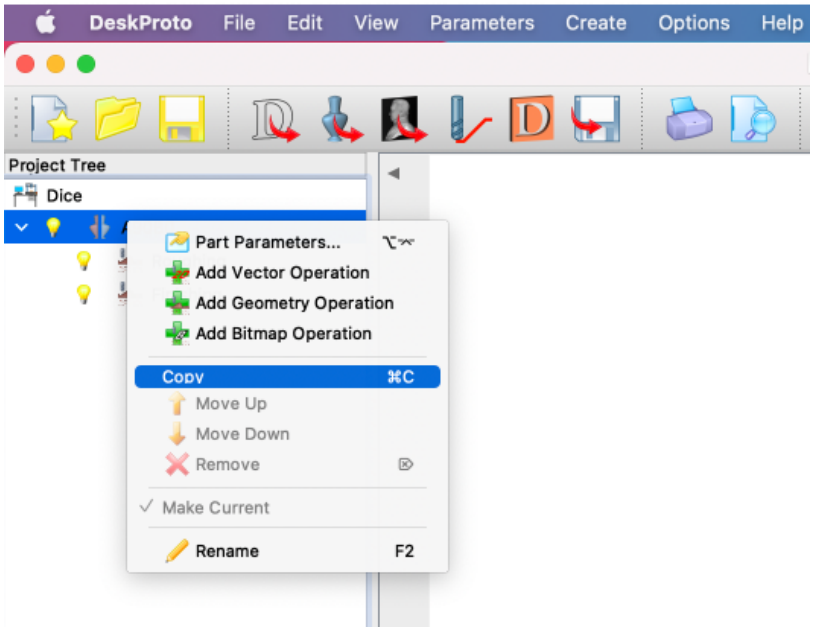
Select the Desk*Proto* simulator and view both the **Roughing** and **Finishing** operations.



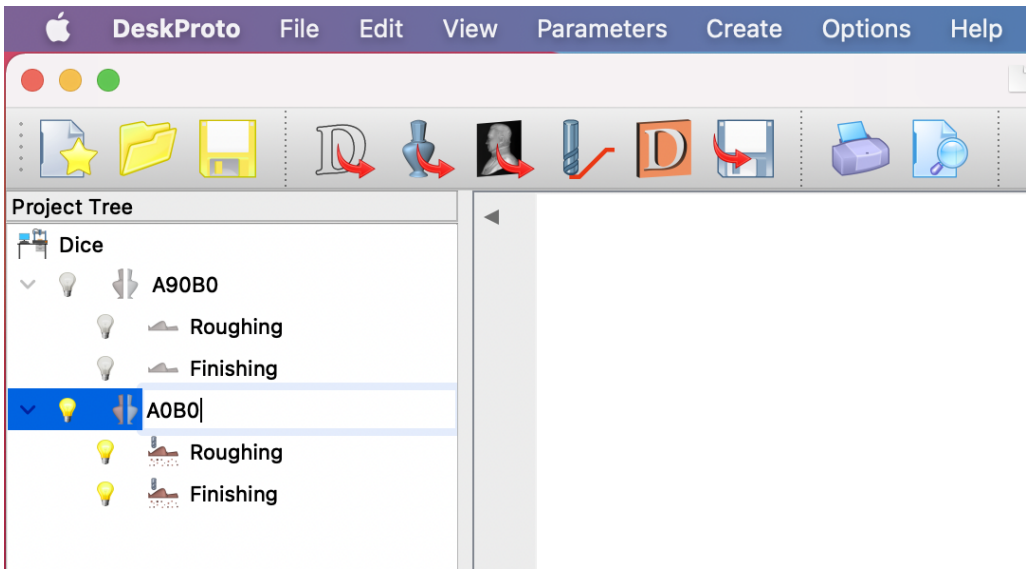
Section 9: Defining the A0B0 Roughing Tool Operations

The remaining tool operations can be derived from the A90B0 tool operation. Each side requires that the work coordinate system be oriented such that the tool approaches perpendicular to the surface being cut.

Begin by copying the **A90B0 Part Parameters**.

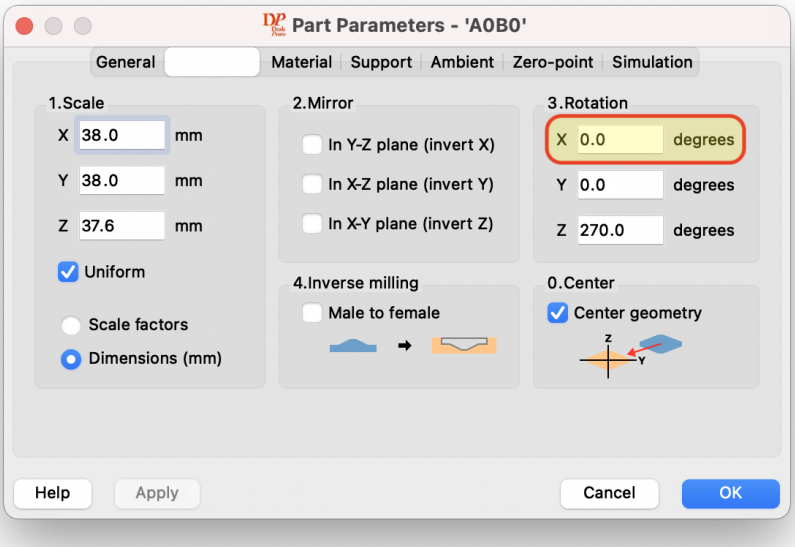


Rename the part **A0B0**.

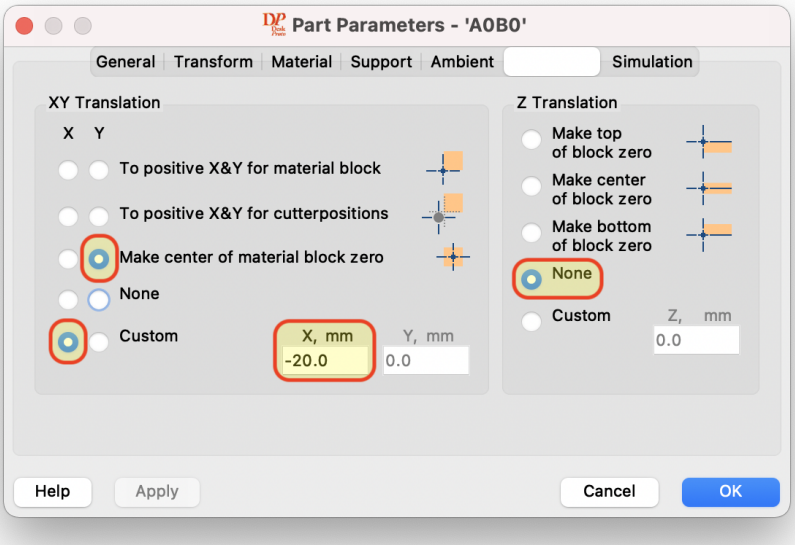


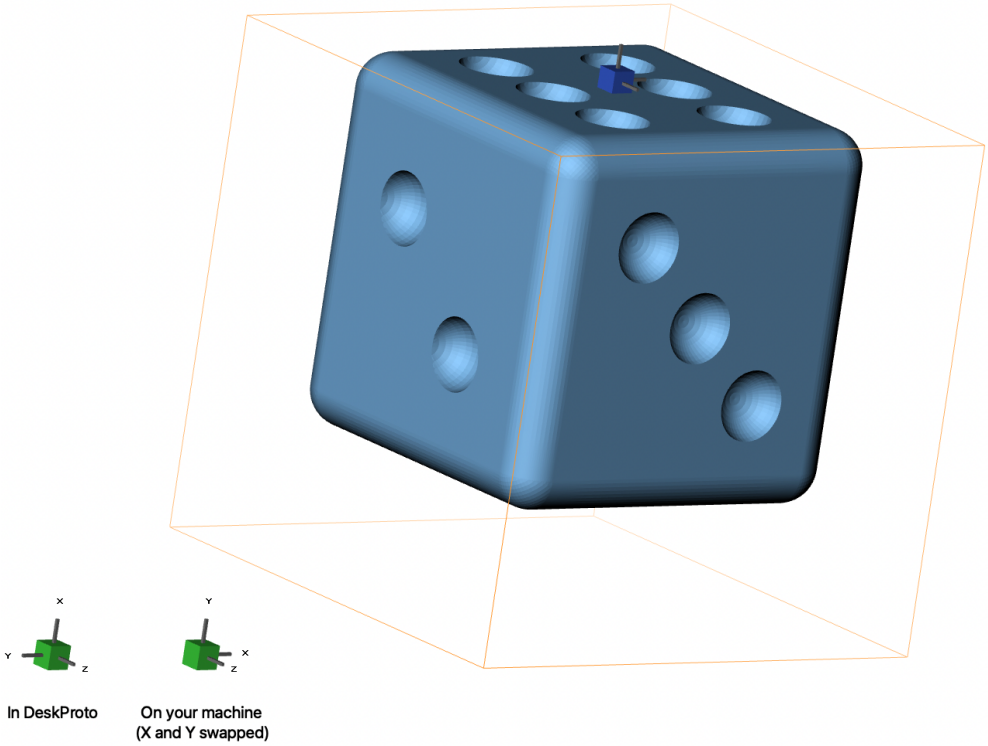
The work coordinate system must be transformed so that the Z axis is perpendicular to the dice side 3. It is important that the work coordinate system remains in the same location on the stock (only the orientation of the work coordinate system should change depending on what side is being machined).

Edit the **A0B0 Part Parameters** and select the **Transform** tab. Change the ***Rotation X*** value to 0 degrees.

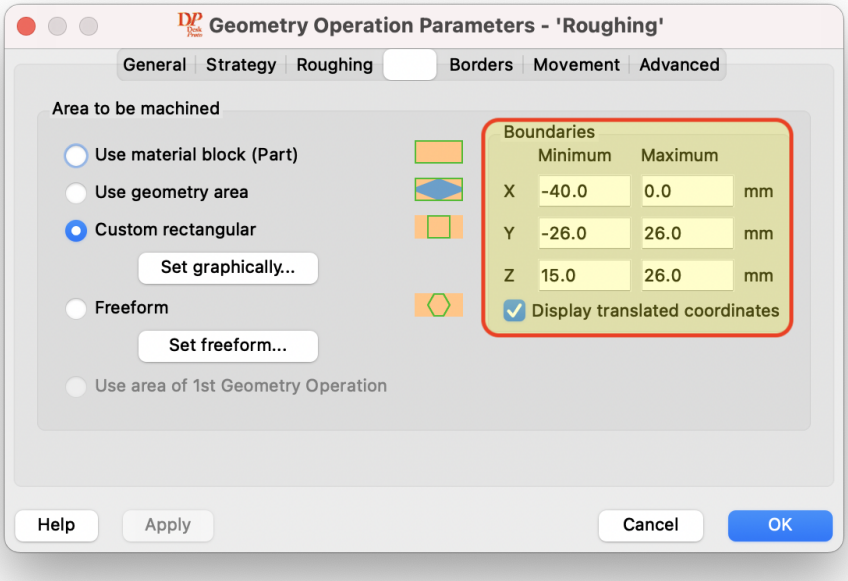


Select the **Zero-point** tab and set the **X Translation** to ***Custom*** and set the ***X*** value to -20 mm, the **Y Translation** to ***Make center of material block zero***, and **Z Translation** to ***None***.

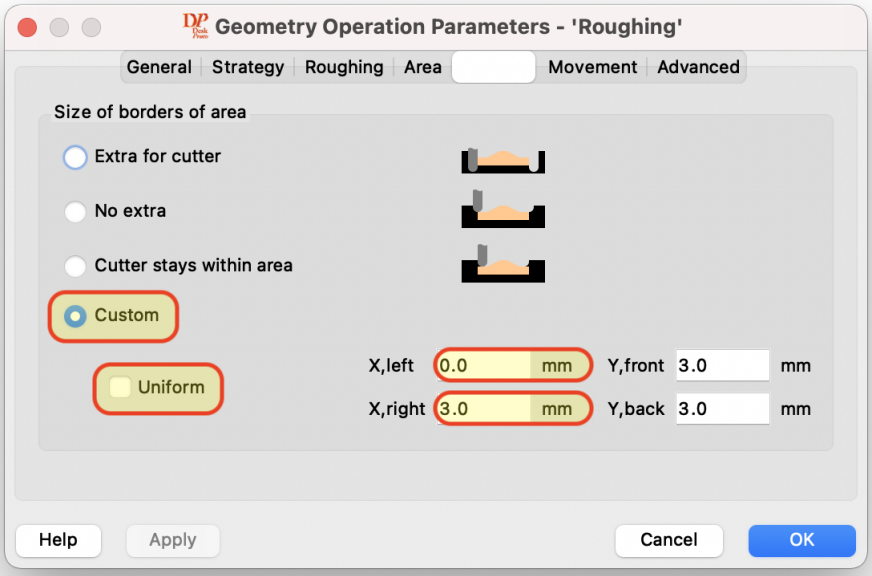




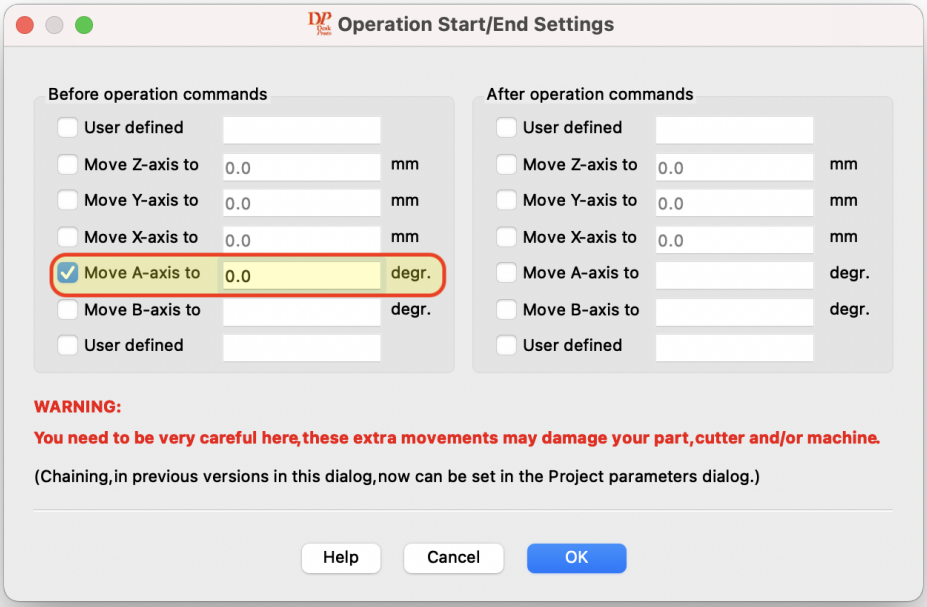
Edit the **A0B0 Roughing Operation Parameters** and select the **Area** tab. Set the values for the ***Boundaries*** as shown below.



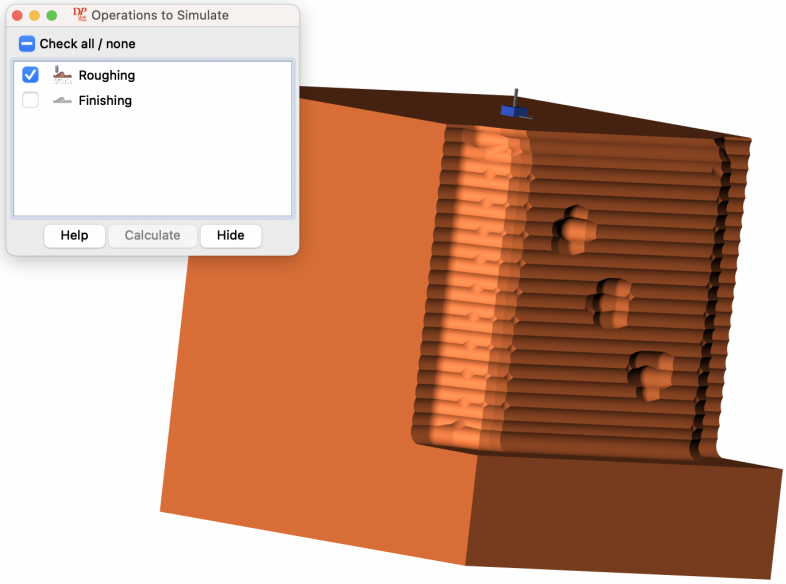
Select the **Borders** tab and choose ***Custom*** for the **Size of borders of area**. Deselect the ***Uniform*** checkbox and adjust the values as shown below. Adjusting ***X,left*** to -3.0 mm keeps the tool a safe distance from the vise.



Select the **Advanced** tab and click the ***Extra NC Commands*** settings button to configure the rotary movement as shown below.

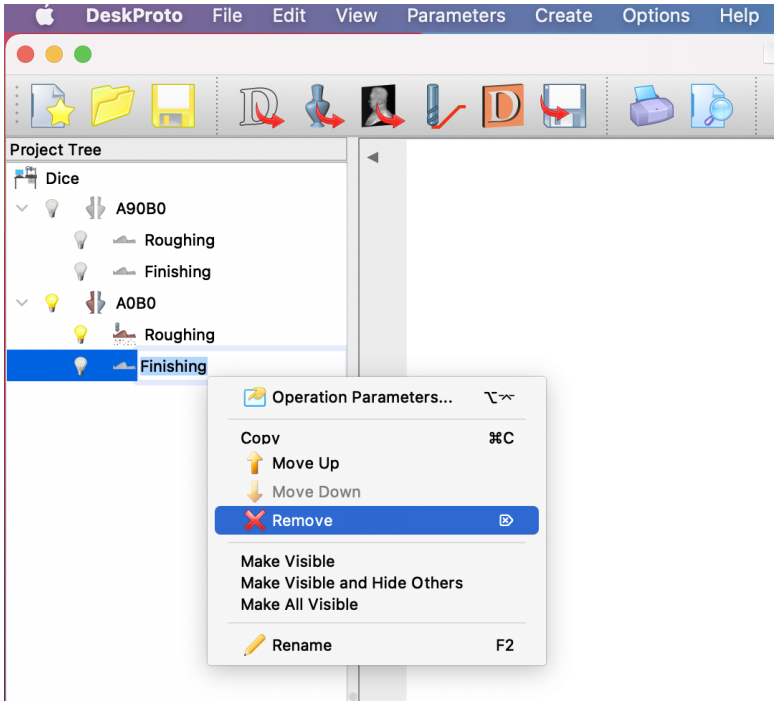


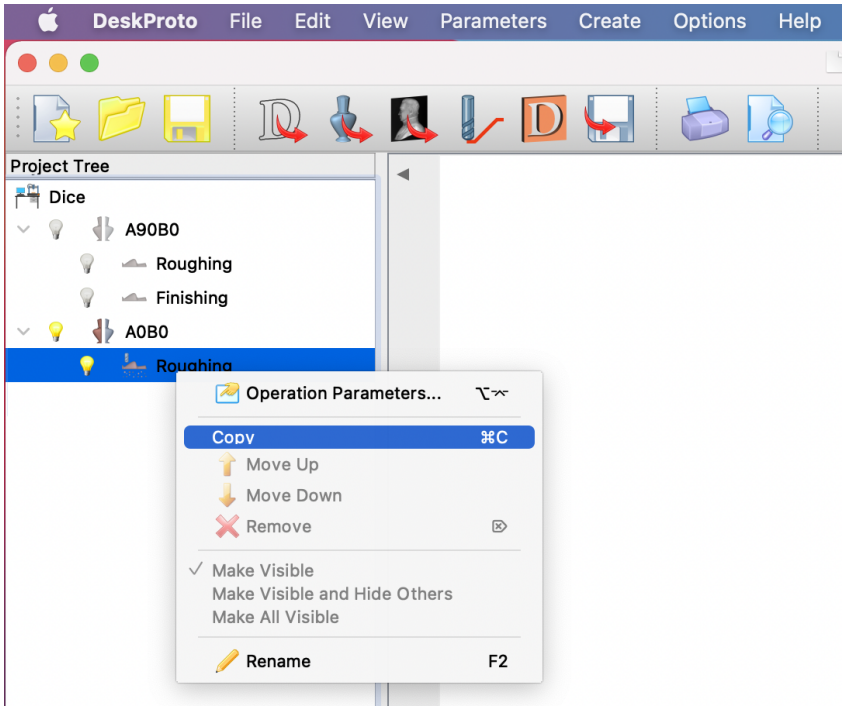
Use the Desk*Proto* simulator to view the material being cut.



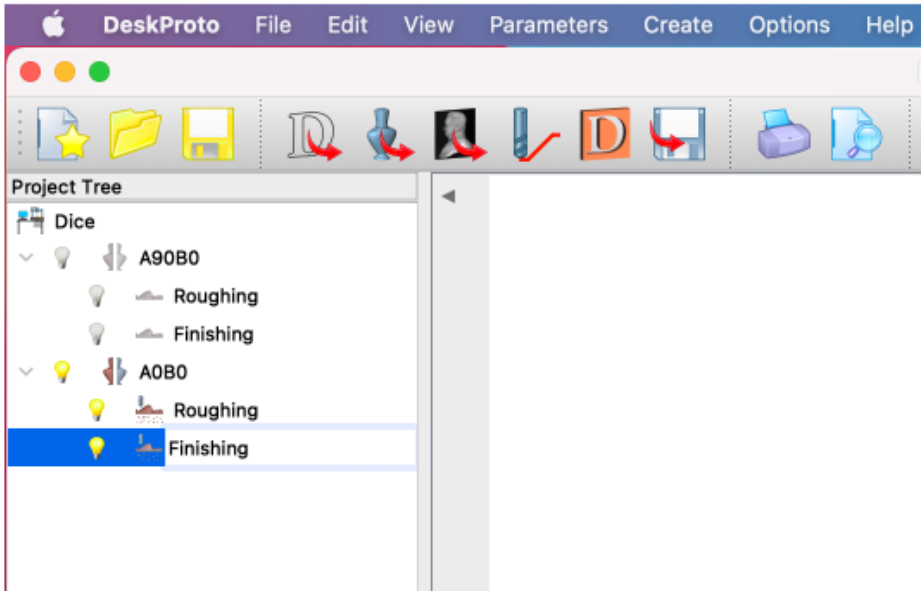
Section 10: Defining the A0B0 Finishing Tool Operations

The most efficient way to configure the A0B0 finishing tool operations is to **Remove** the Finishing operation that was copied from the A90B0 part setup and copy the A0B0 roughing operation.



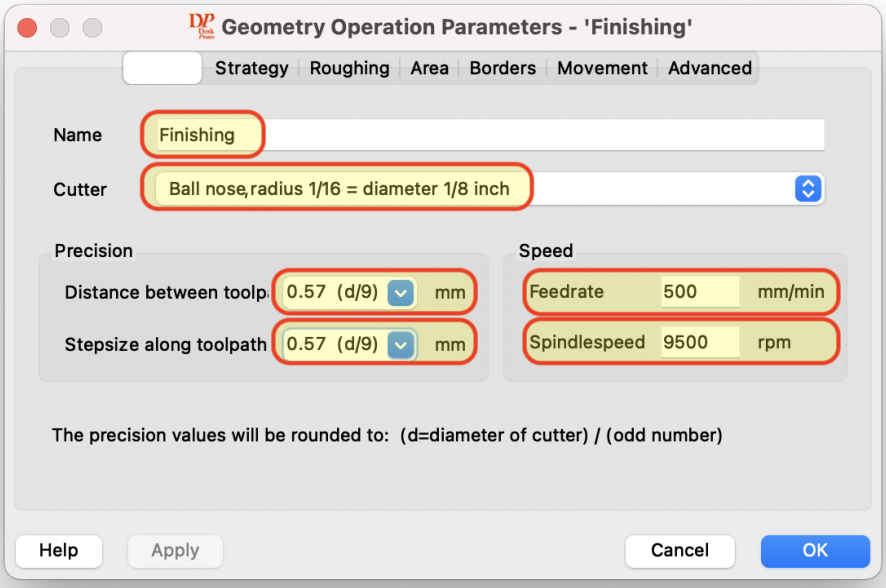


Rename the new tool operation Finishing.

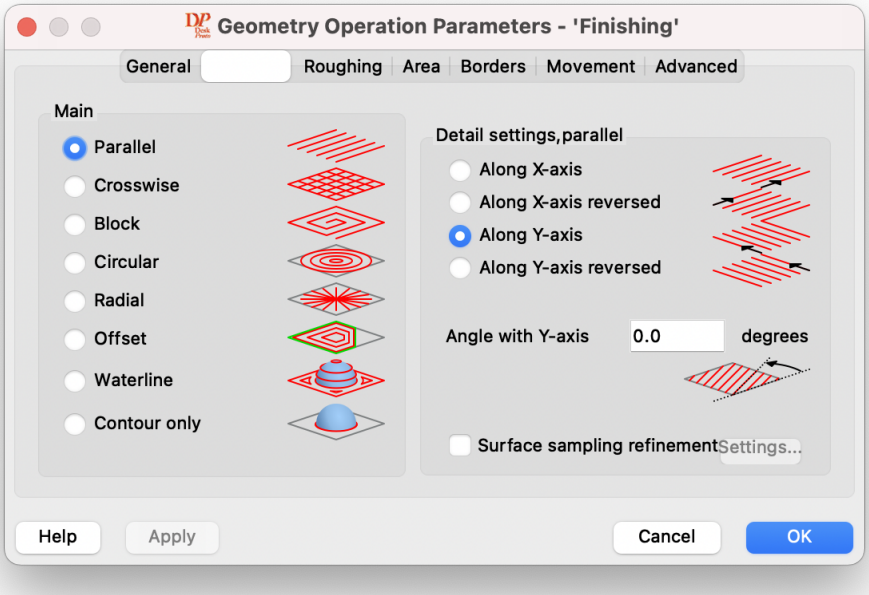


Configure the A0B0 finishing tool operation by configuring the following tabs accordingly.

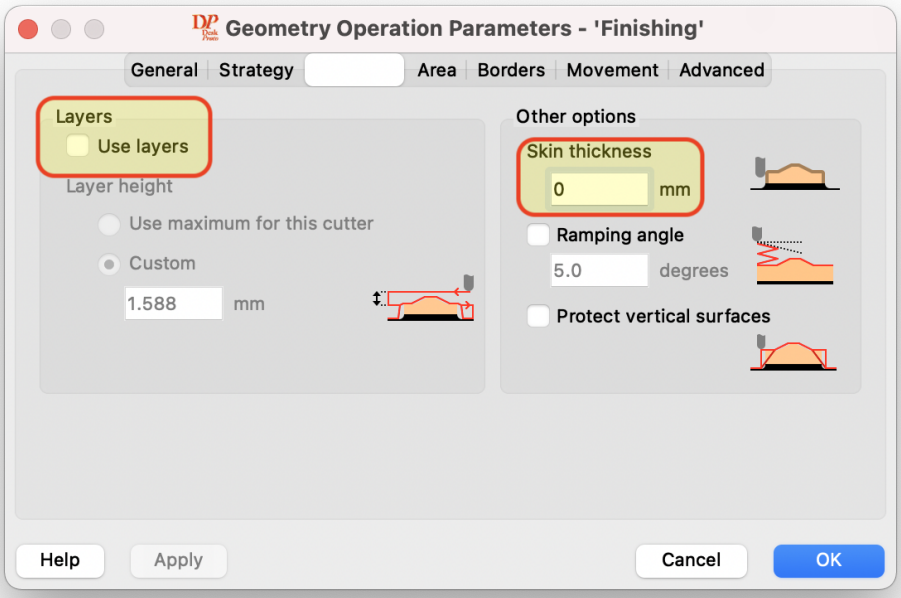
General Tab



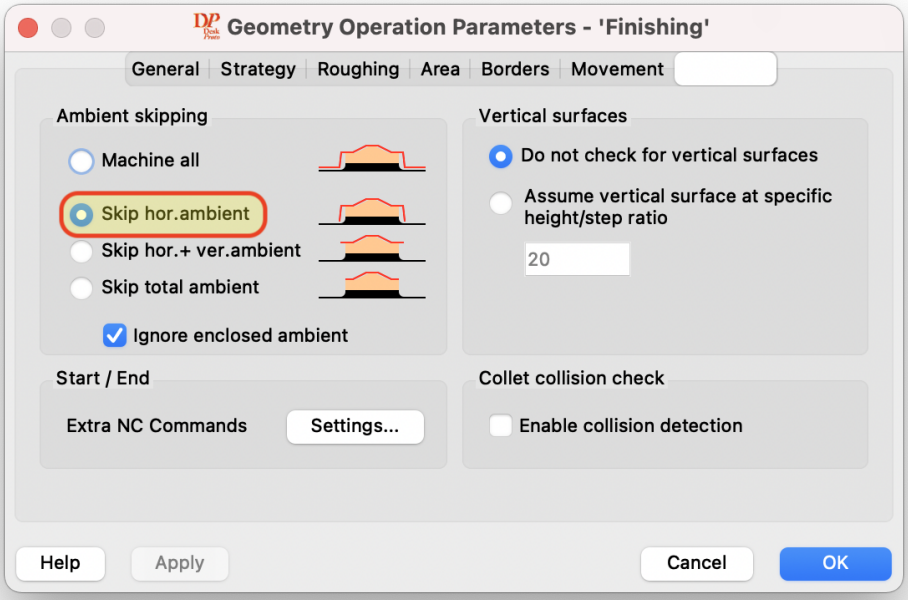
Strategy Tab



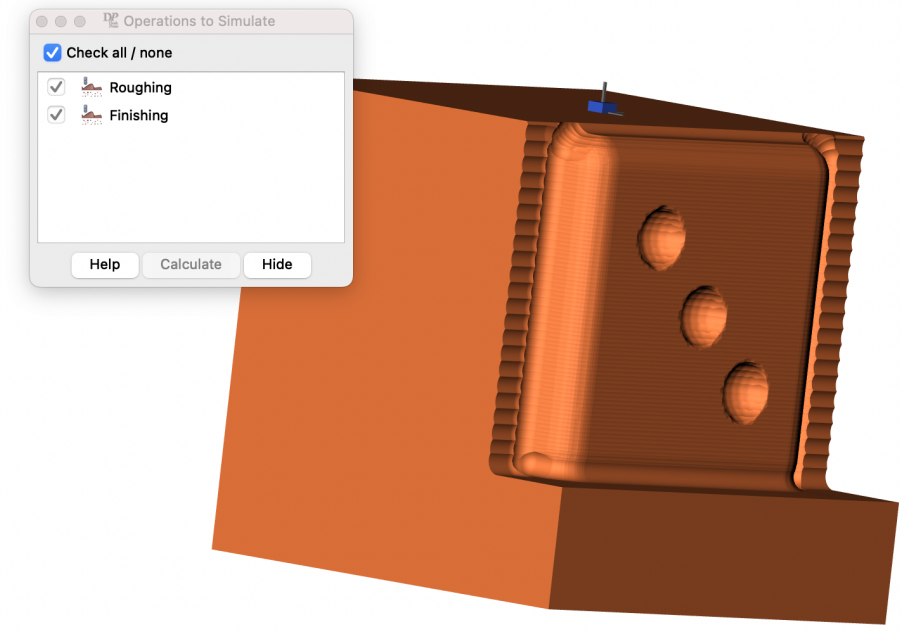
Roughing Tab



Advanced Tab



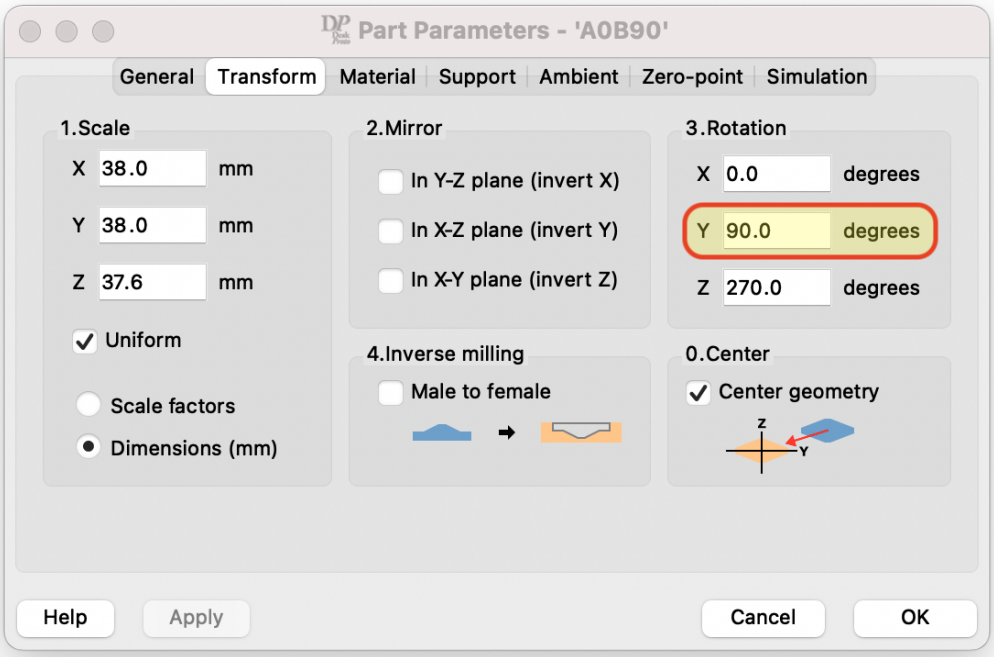
Select the Desk*Proto* simulator and view both the **Roughing** and **Finishing** operations.



Section 11: Defining the A0B90 Roughing and Finishing Tool Operations

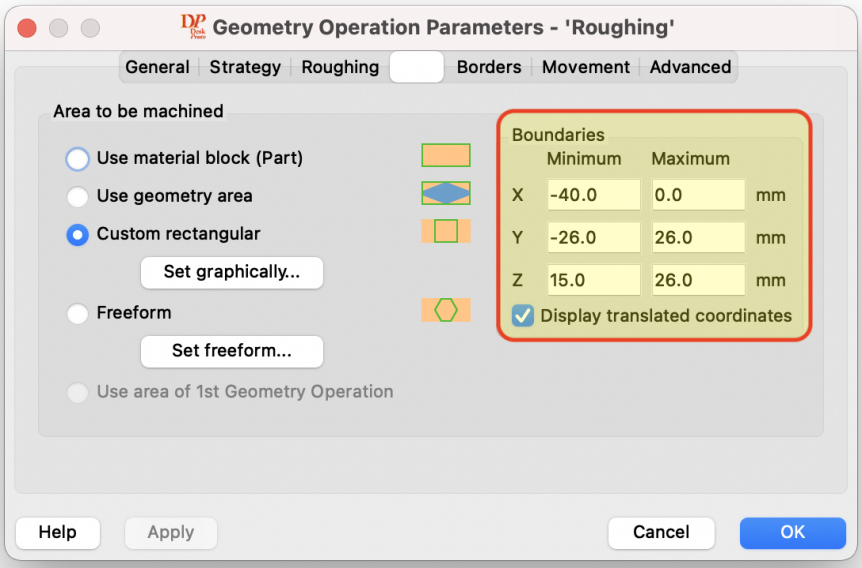
The A0B90 tool operations can be derived from the A0B0 tool operation. Begin by copying the **A0B0 Part Parameters** and naming the new part parameter **A0B90**.

Rotate the work coordinate system 90° by adjusting the **A0B90 Part Parameters Transform** rotation value for ***Y*** to 90°.

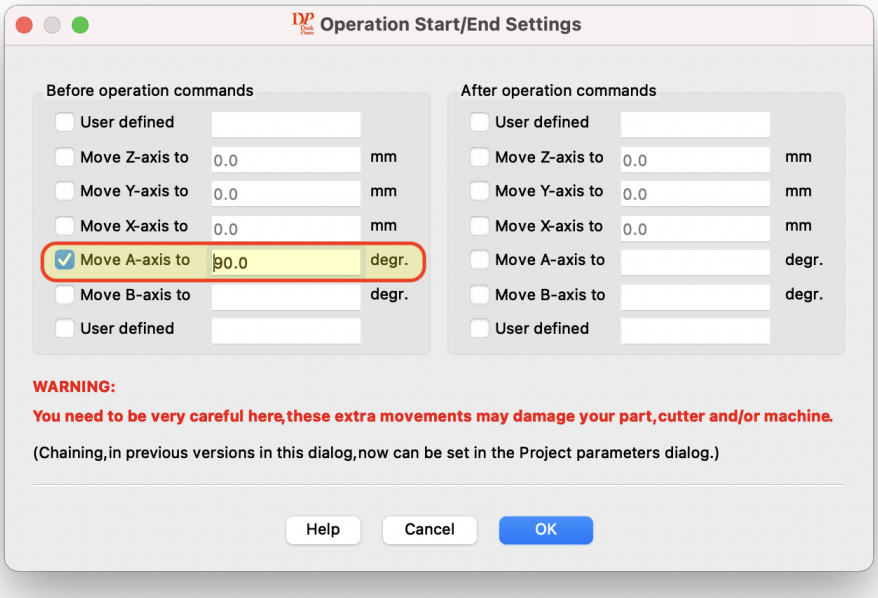


Configure the A0B90 roughing and finishing tool operations by configuring the following tabs accordingly. (Roughing is shown but the same changes are made to finishing operations.)

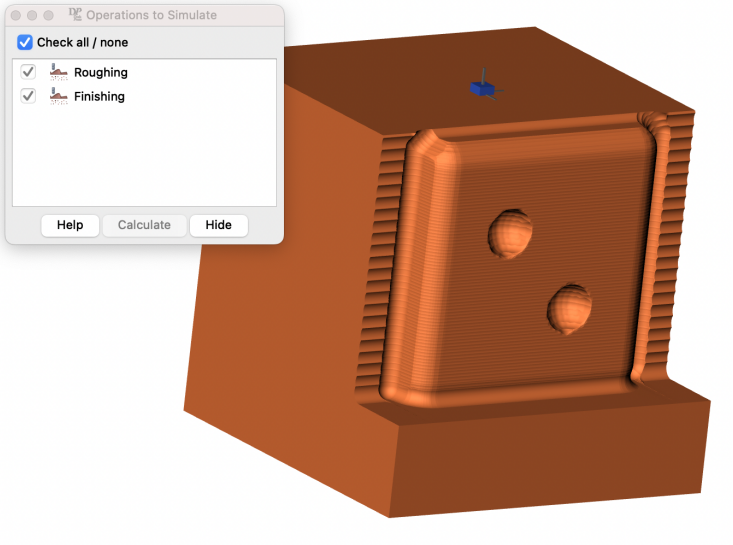
Area Tab



Advanced Settings Tab



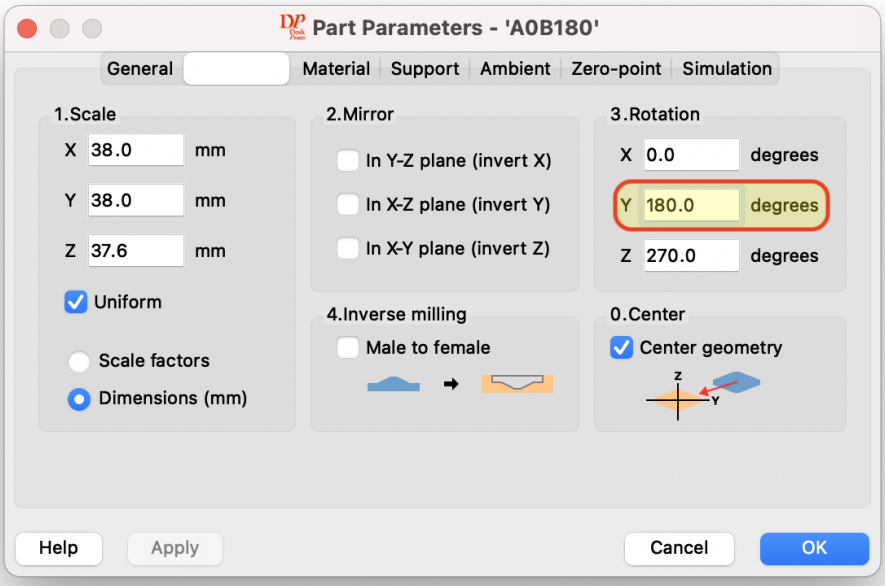
Select the Desk*Proto* simulator and view both the **Roughing** and **Finishing** operations.



Section 12: Defining the A0B180 Roughing and Finishing Tool Operations

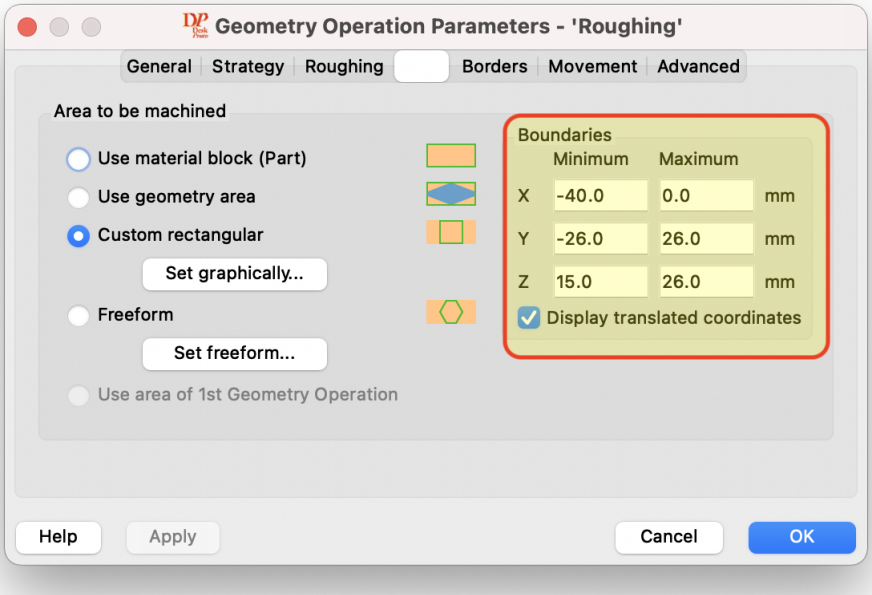
The A0B180 tool operation can be derived from the A0B90 tool operation. Begin by copying the **A0B90 Part Parameters** and naming the new part parameter **A0B180**.

Rotate the work coordinate system 90° by adjusting the **A0B180 Part Parameters Transform** rotation value for ***Y*** to 180°.

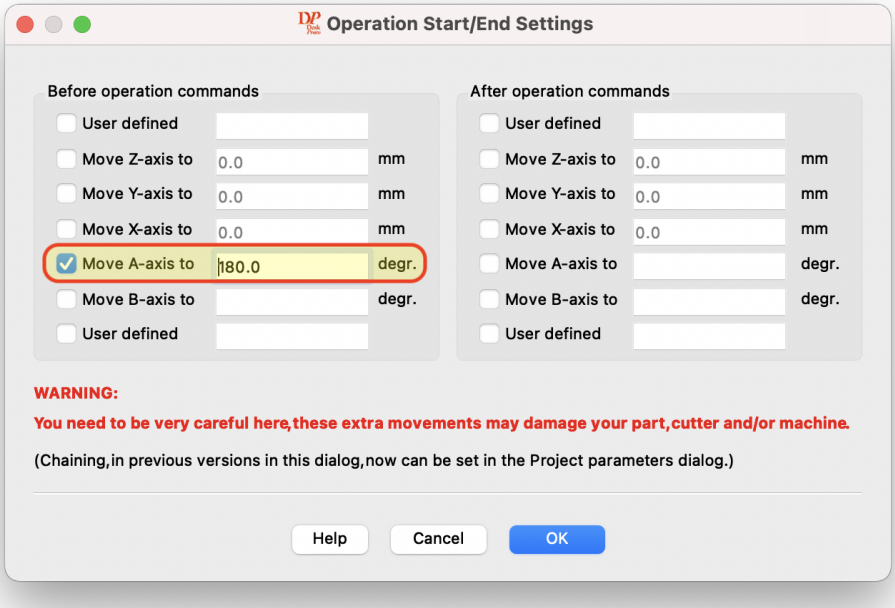


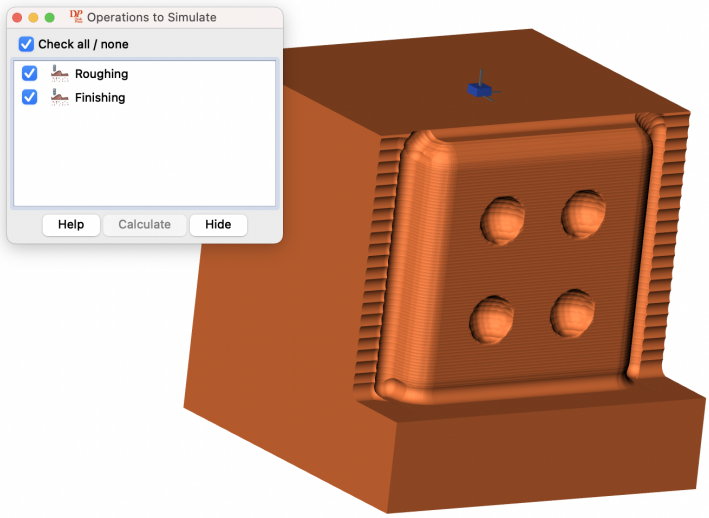
Configure the A0B180 roughing and finishing tool operations by configuring the following tabs accordingly. (Roughing is shown but the same changes are made to finishing operations.)

Area Tab



Advanced Settings Tab

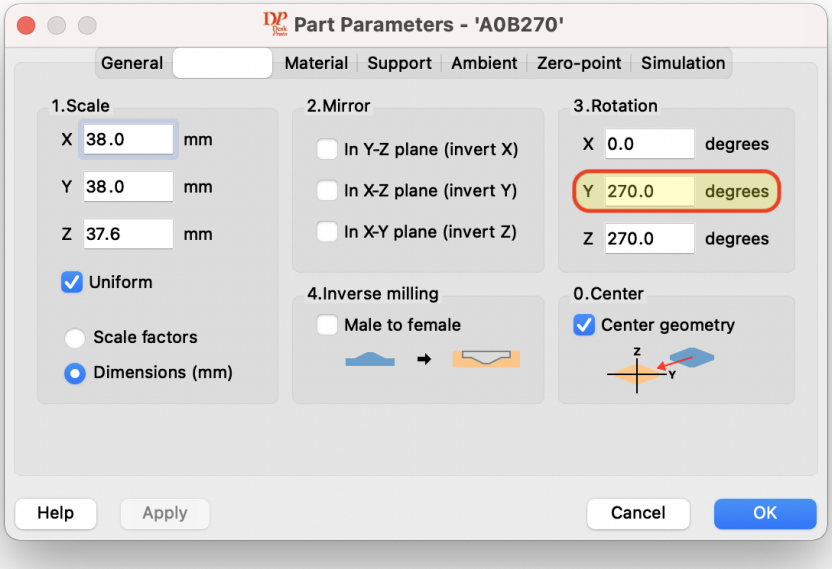




Section 13: Defining the A0B270 Roughing and Finishing Tool Operations

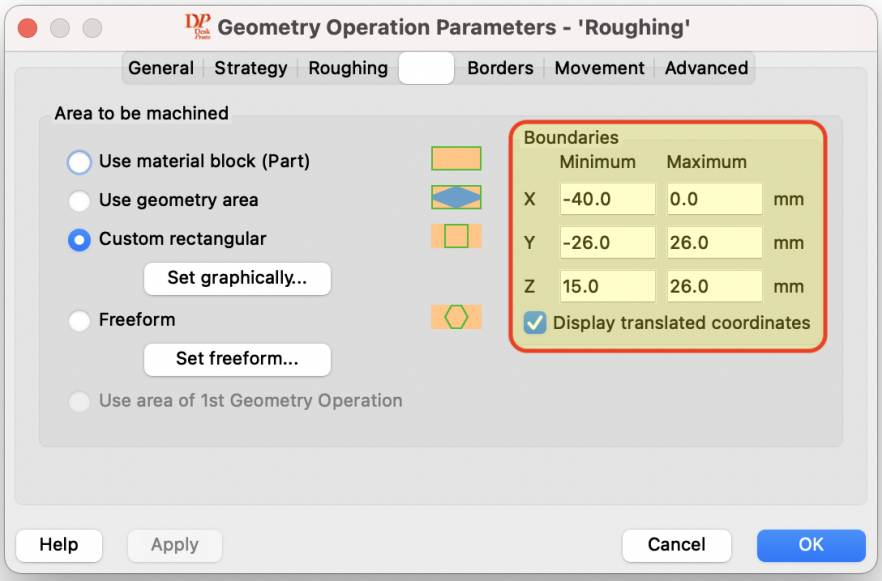
The A0B270 tool operation can be derived from the A0B180 tool operation. Begin by copying the **A0B180 Part Parameters** and naming the new part parameter **A0B270**.

Rotate the work coordinate system 90° by adjusting the **A0B270 Part Parameters Transform** rotation value for ***Y*** to 270°.

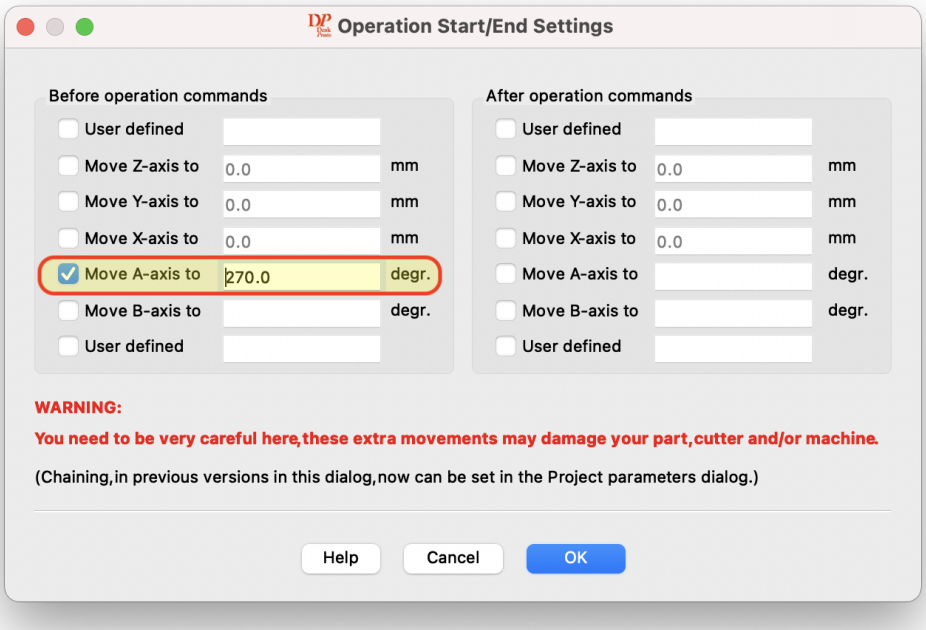


Configure the A0B180 roughing and finishing tool operations by configuring the following tabs accordingly. (Roughing is shown but the same changes are made to finishing operations.)

Area Tab



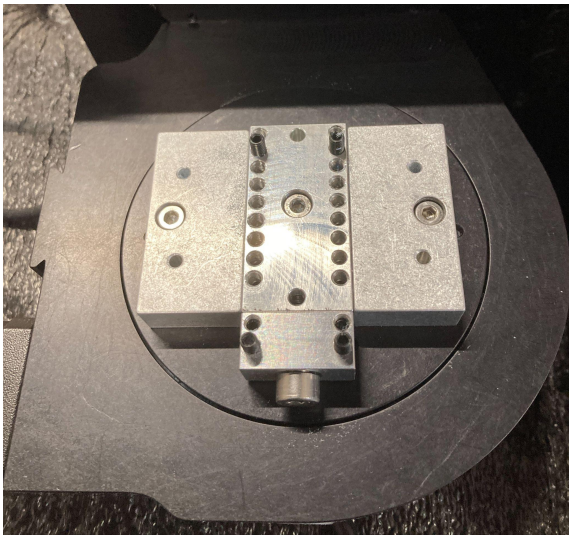
Advanced Settings Tab



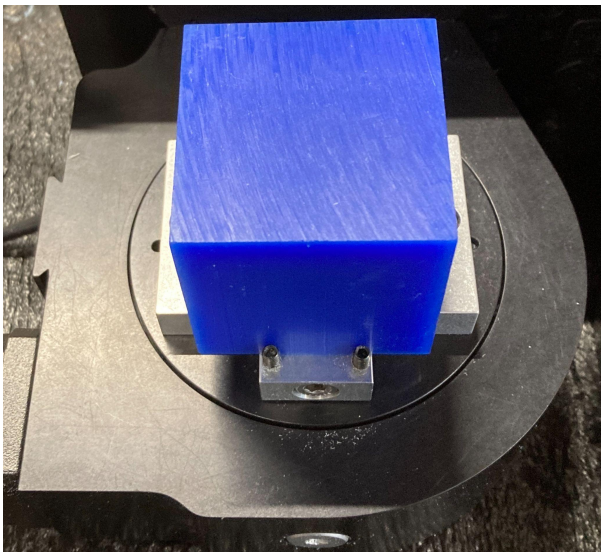
Section 14: Mounting the stock to the Pocket NC V2 and measuring the G55 work offsets.

The dice part will be cut from a 52 mm 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.

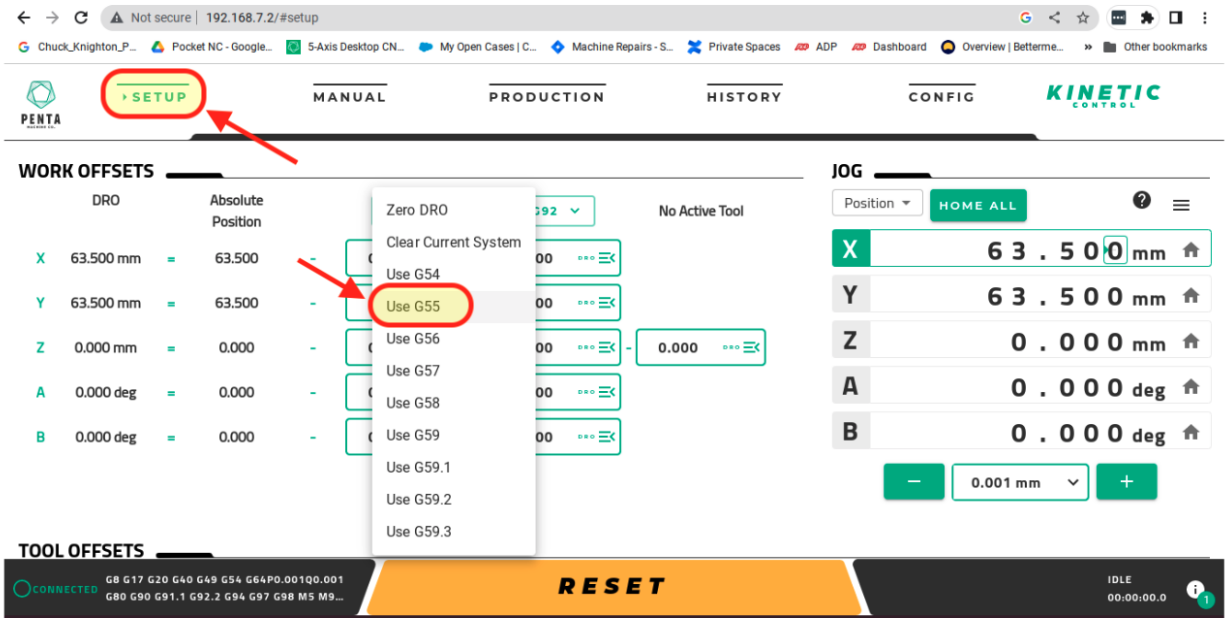


Next, mount the cube stock on the vise.

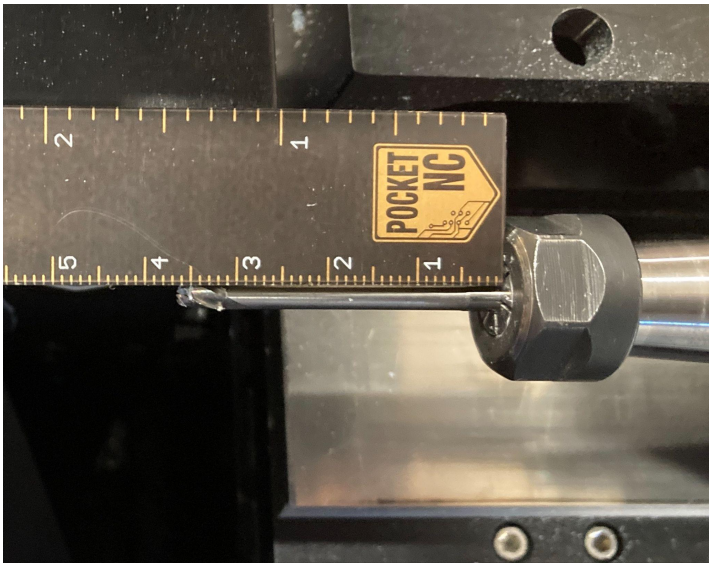


The following steps measure the work offsets that will be used by the Kinetic Control interface to locate the stock's work coordinate origin in relation to the machine's center of rotation.

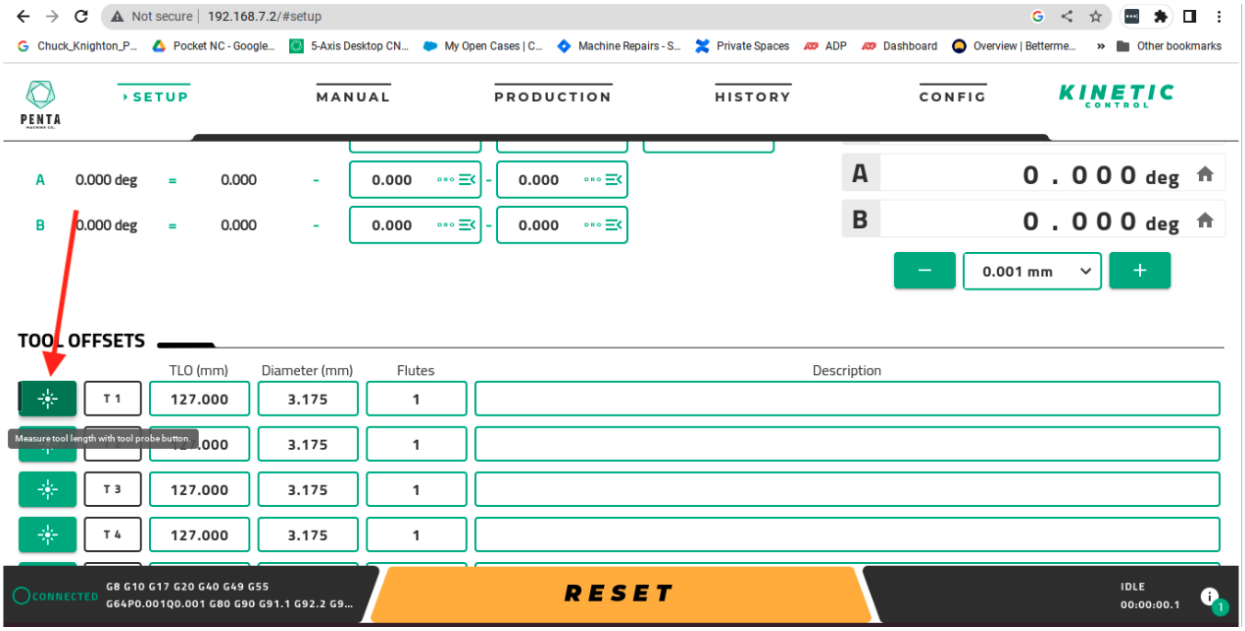
Desk*Proto* uses the G55 work offsets by default. To set up Kinetic Control, select the **G54** pulldown menu and select the ***G55*** under the **WORK OFFSETS** section.



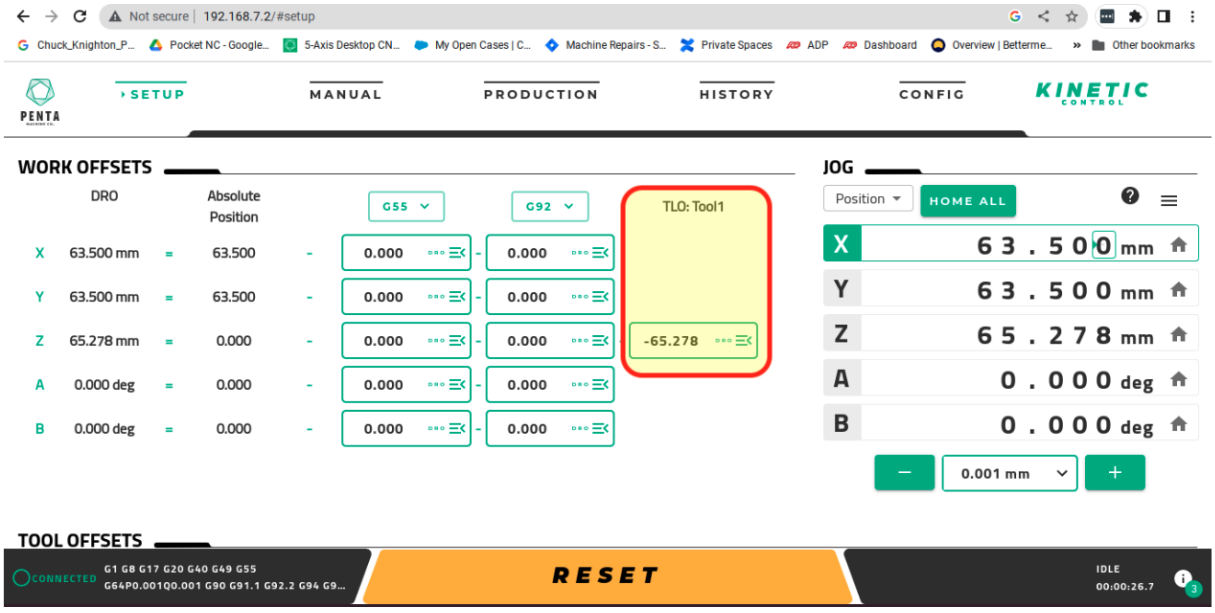
To properly measure the work offsets, the tool length offset for Tool 1 must be measured. Load the Harvey Tool 1/8th Ball Long Reach, Stub Flute - part number: 96108 in the tool holder for your V2. For the purpose of this tutorial, a Pocket NC V2-10 was being used. The picture below shows the tool loaded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



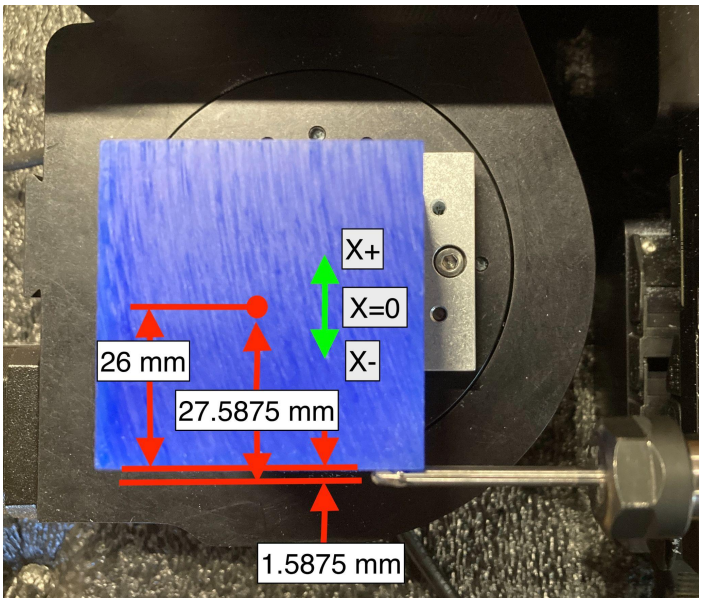
Next measure the Tool Length Offset using Kinetic Control. Navigate to the **SETUP** tab on Kinetic Control. Click the green button next to Tool 1 to initiate a tool length offset measurement. Once the Tool Length Offset for Tool 1 is completed. Click the white T1 button to make Tool 1 the active tool.



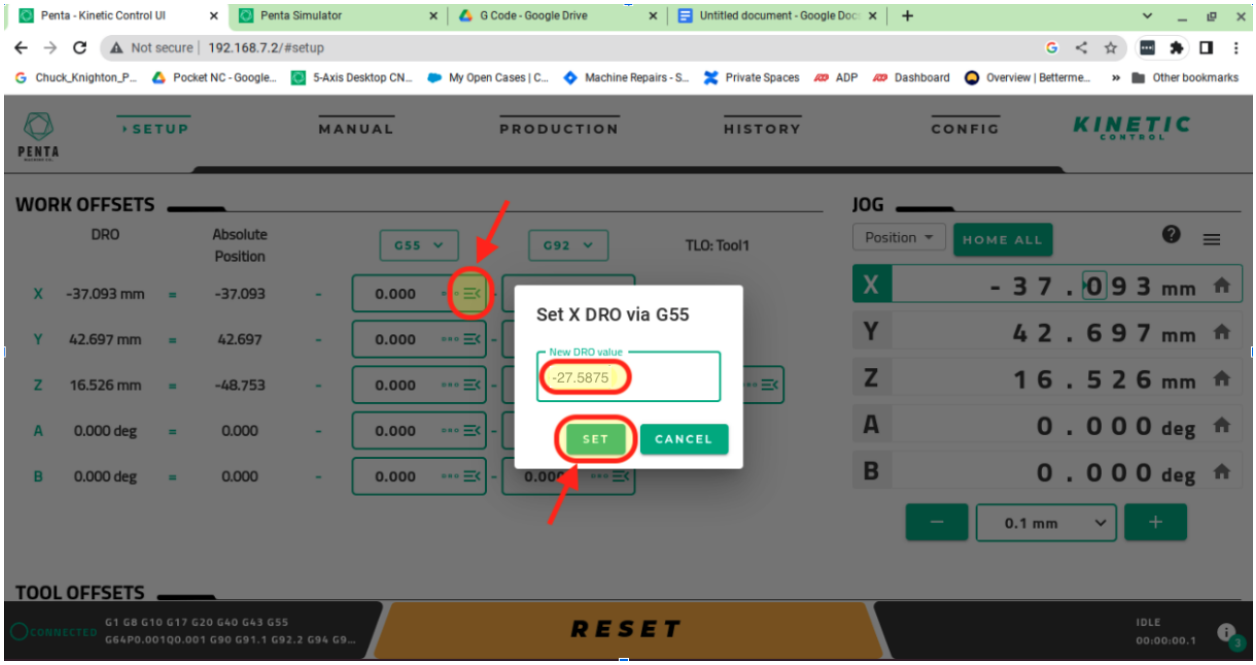
The **SETUP** screen should show Tool 1 as active and the tool offset value should be approximately -65 mm if using a Pocket NC V2-10 (see below).



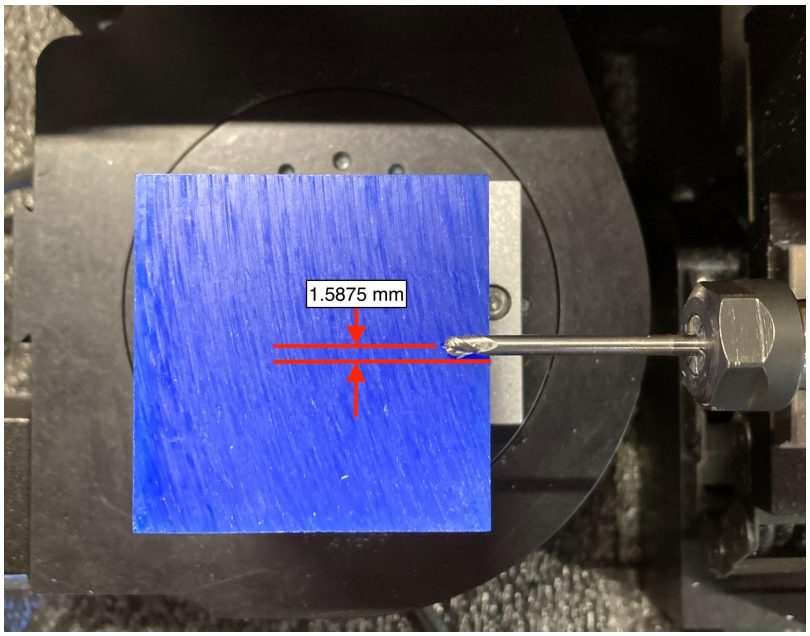
Perform an X touch off by jogging the tool so that it just touches the front face of the stock. The tool center is located one half of the tool diameter (1.5875 mm) plus one half of the stock's side dimension (26 mm) to the negative from X=0 on the stock. The total distance to X=0 on the stock is -27.5875.



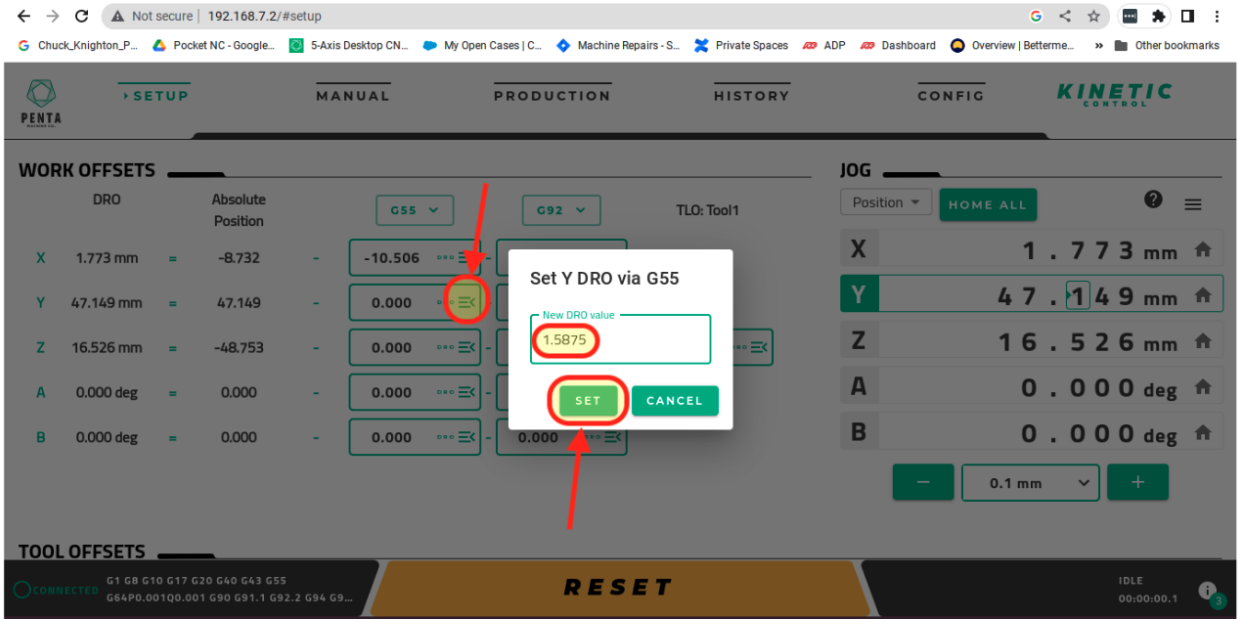
Click the three bars icon in the X work offset box. Enter -27.5875 for the **New DRO value** and click the **SET** button.



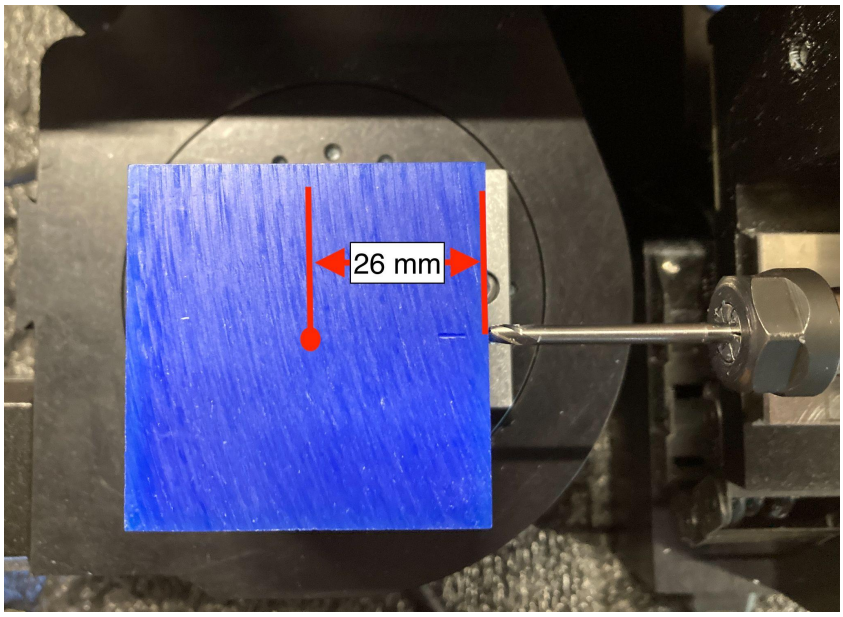
Perform a Y touch off by jogging the tool so that it just touches the top face of the stock. The tool center is located one half of the tool diameter (1.5875 mm) to the positive from Y=0 on the stock. The total distance to Y=0 on the stock is +1.5875 mm.



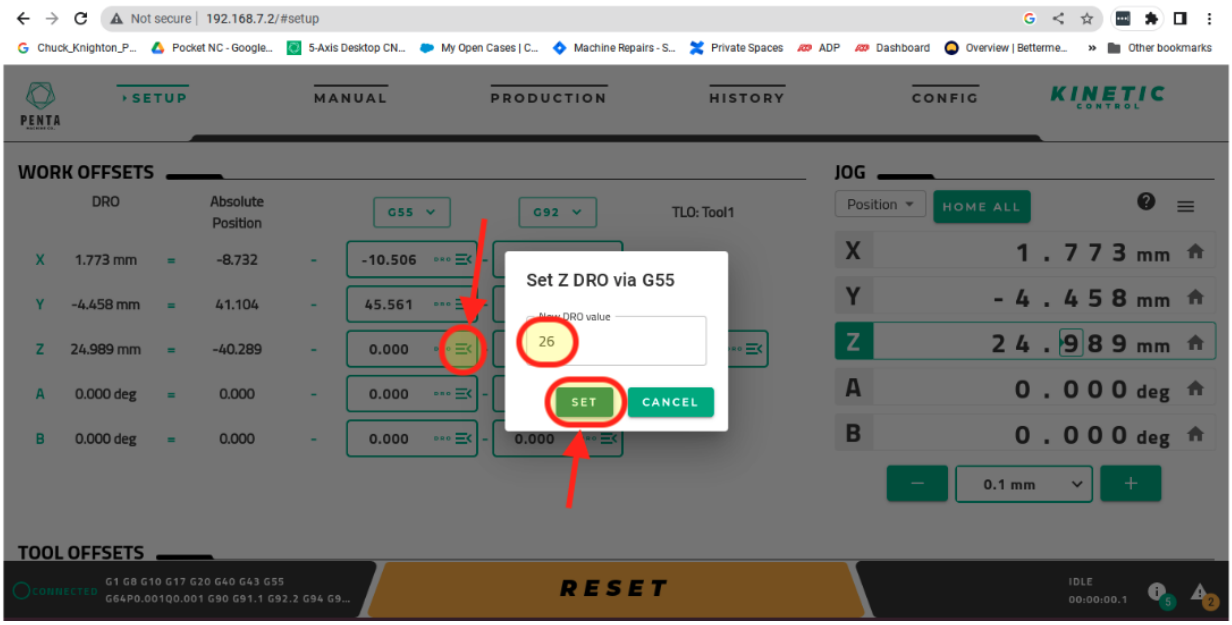
Click the three bars icon in the Y work offset box. Enter +1.5875 for the **New DRO value** and click the **SET** button.



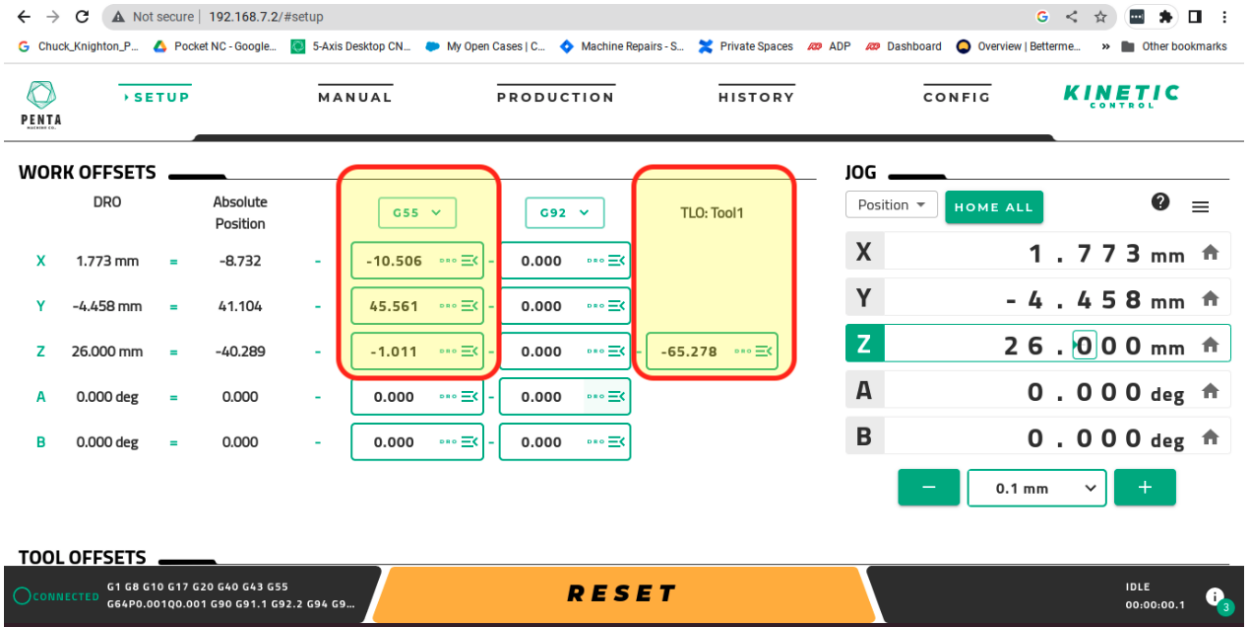
Perform a Z touch off by jogging the tool so that it just touches the right face of the stock. The tool center is located one half of the stock's side dimension (26 mm) to the negative from Z=0 on the stock. The total distance to Z=0 on the stock is +26 mm.



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.



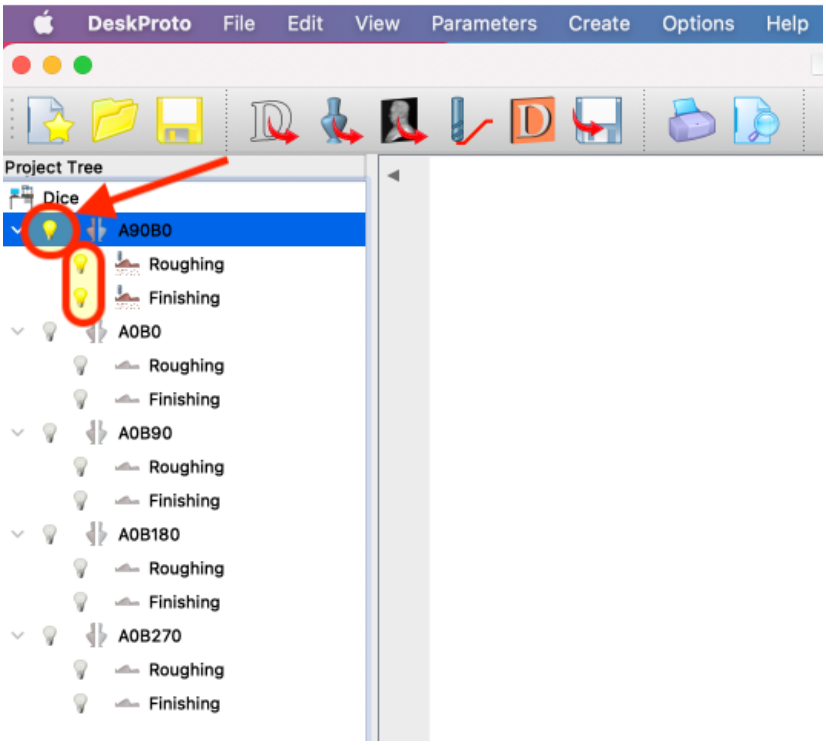
The G55 values and the Tool Length Offset value will be used in the next section to evaluate the G-Code created by Desk*Proto*.

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

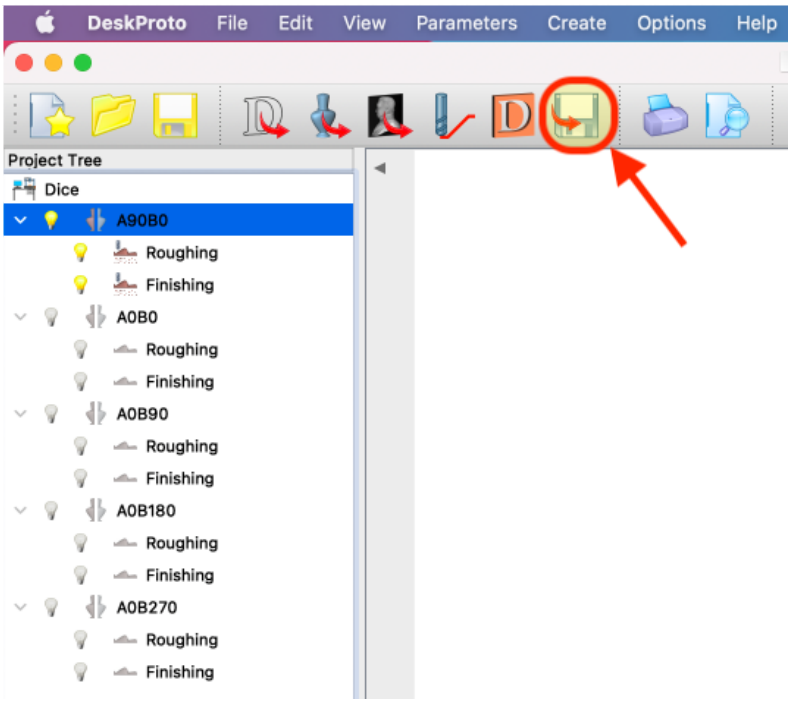
These following steps are necessary when using the Pocket NC custom vise. The stock material will not be centered around the machine center of rotation. Kinetic Control uses TCPC (Tool Center Point Control) to translate the stock work center coordinates to the machine center of rotation.

Five programs will be used to create the dice part. One g-code program will be created for the top tool operations (dice number 6) and another four g-code programs for the remaining four sides.

Highlight the A90B0 tool operation.



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

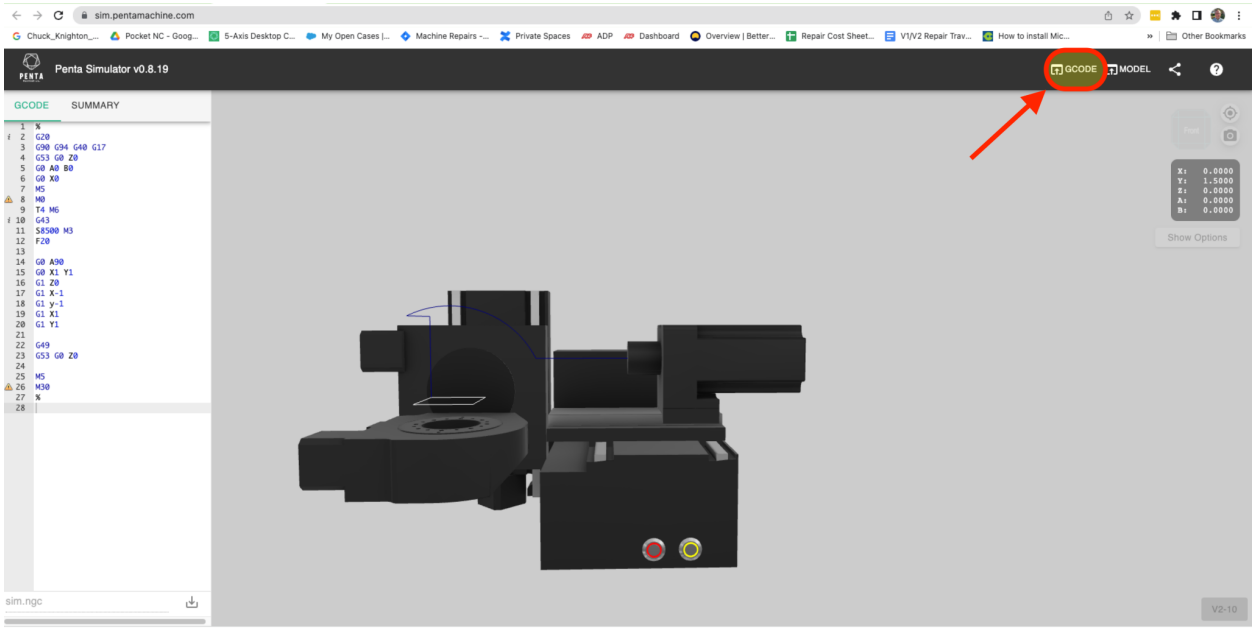


Desk*Proto* will only post process the highlighted tool operations. Name the file to indicate that it is the g-code for cutting the top of the dice part (example - Dice - A90B0.ngc).

Continue the process for the A0B0, A0B90, A0B180, and A0B270 tool operations.

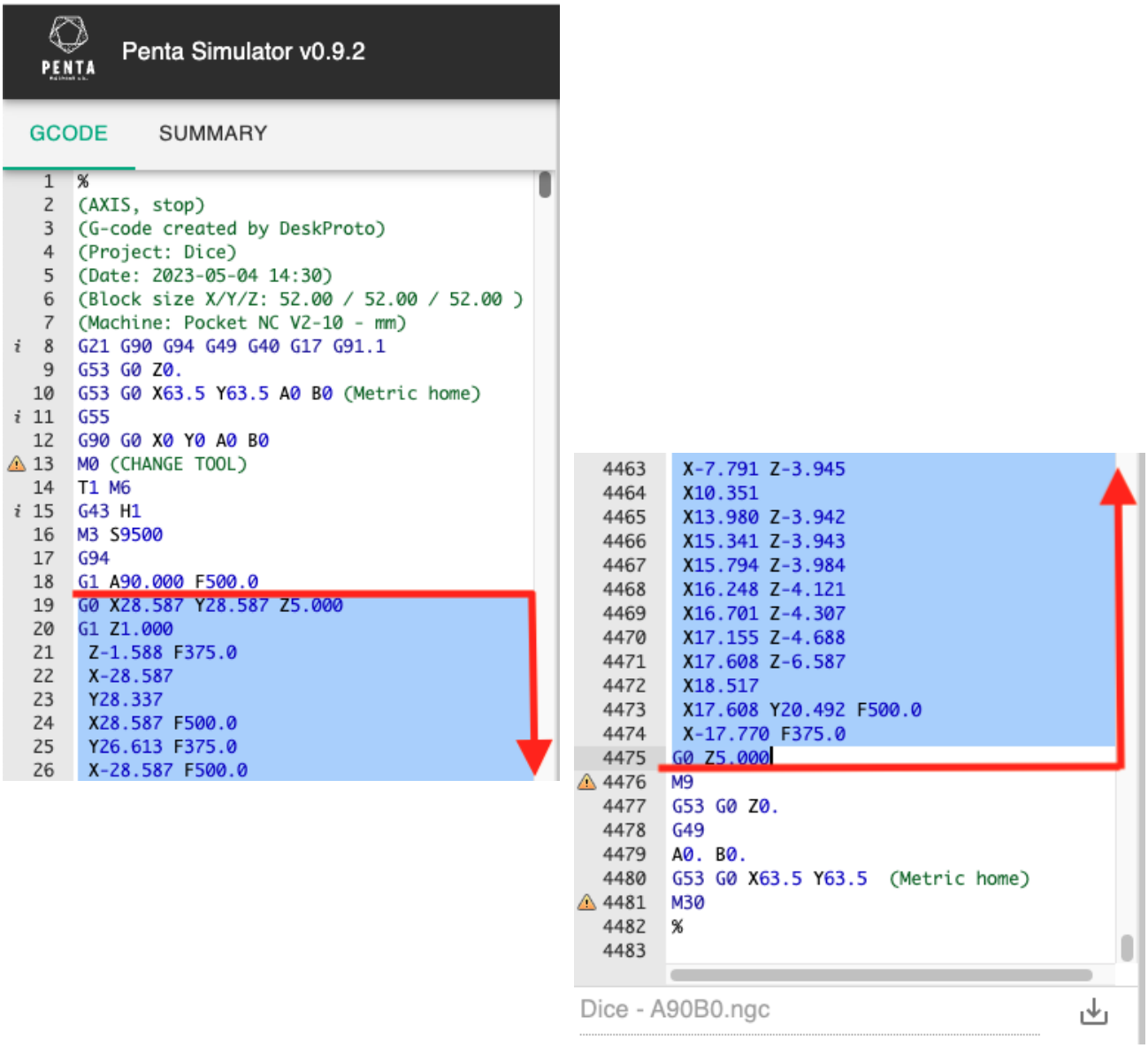
The next step is to load the program g-code into the Penta Machine simulator, which can be accessed by clicking here.

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

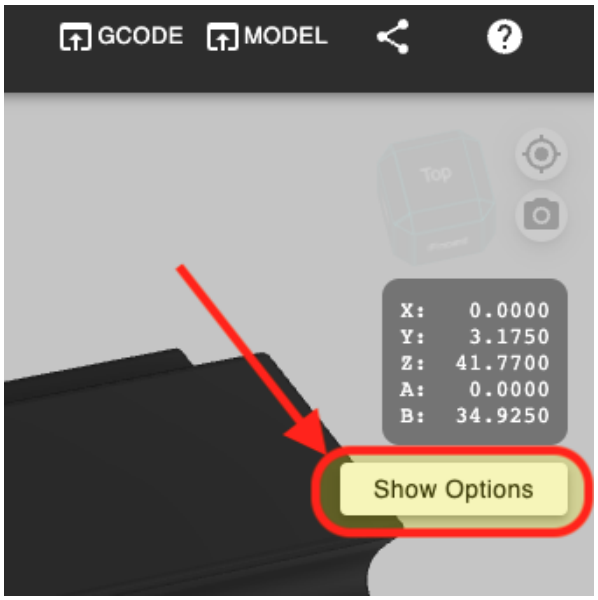


The Penta Machine Company simulator, version 0.9.0 and higher, now can convert g-code to use TCPC.

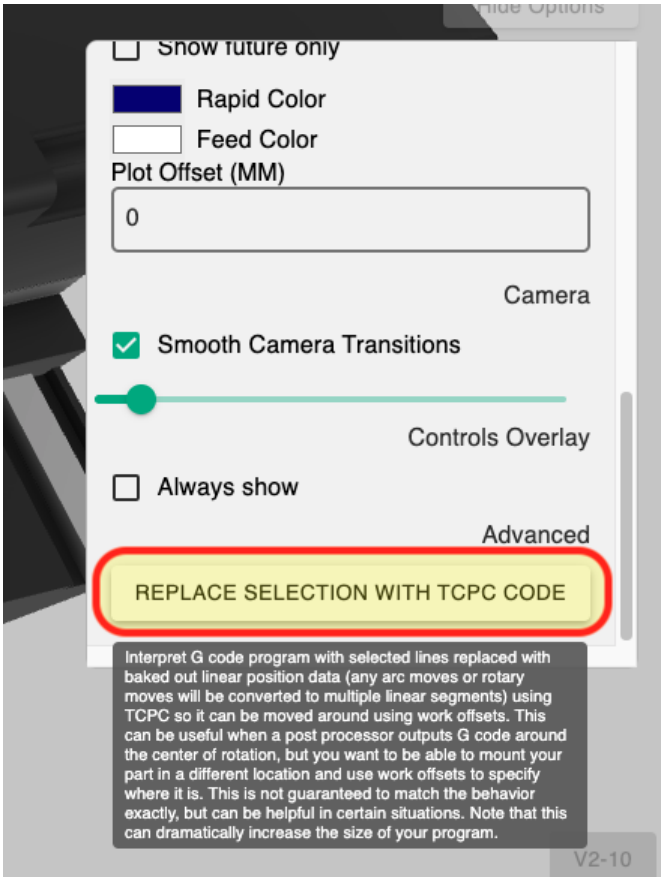
Using this conversion process requires selecting the g-code where TCPC should be applied. This process is easy when the roughing and finishing tools are the same. Below is an example of the beginning and ending code for the top tool operations for the dice part. Select the code after the G1 A90.000 F500.0 rotary command and G53 command (move in machine coordinates). The M9 (turn mist and flood coolant off) is ignored by the simulator.



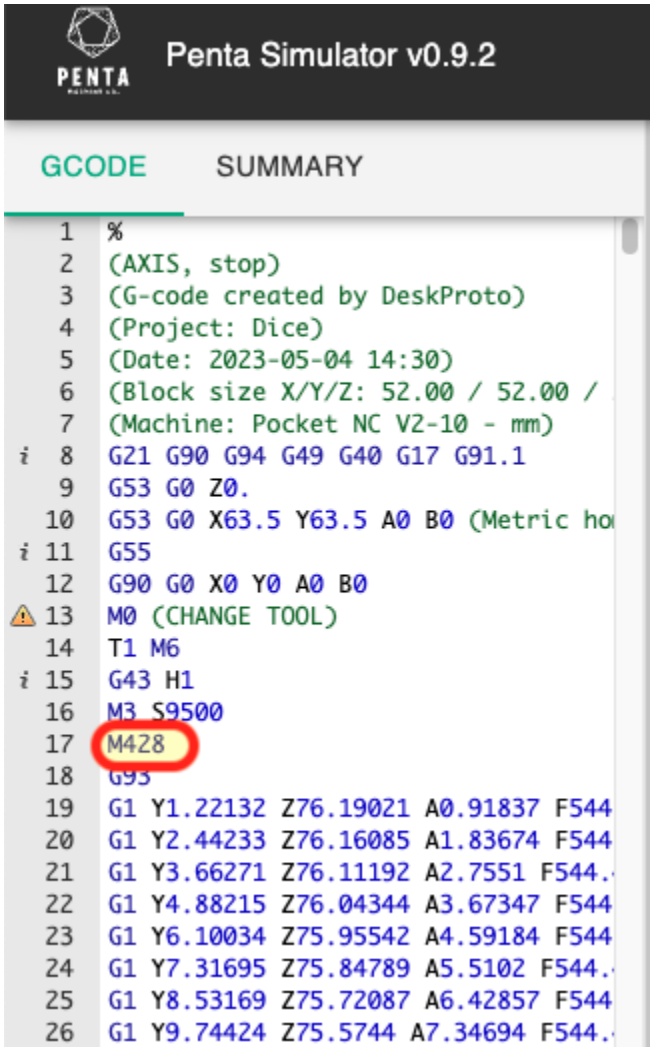
Click the **SHOW OPTIONS** button (located in the upper right corner of the display).

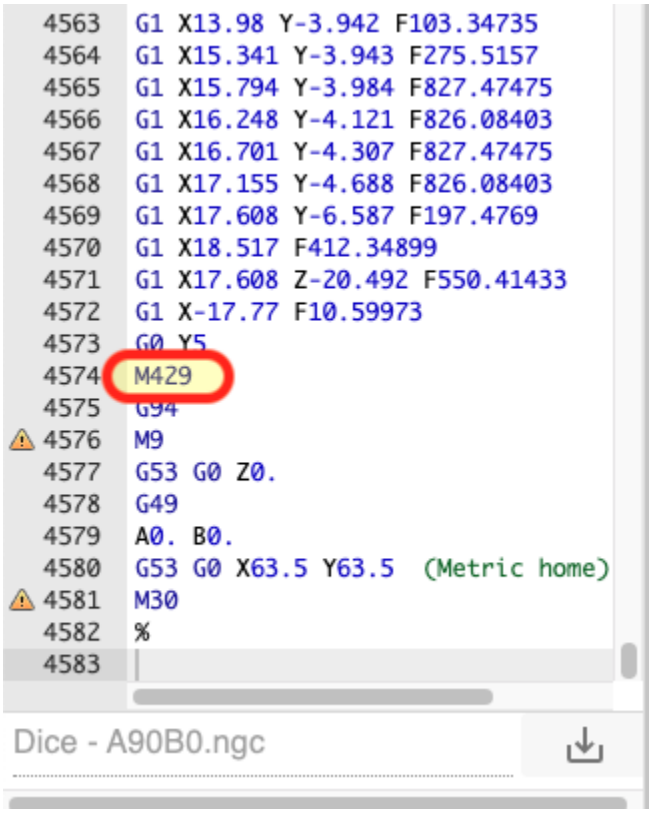


Scroll to the bottom of the **SHOW OPTIONS** and click the ***REPLACE SELECTION WITH TCPC CODE***.

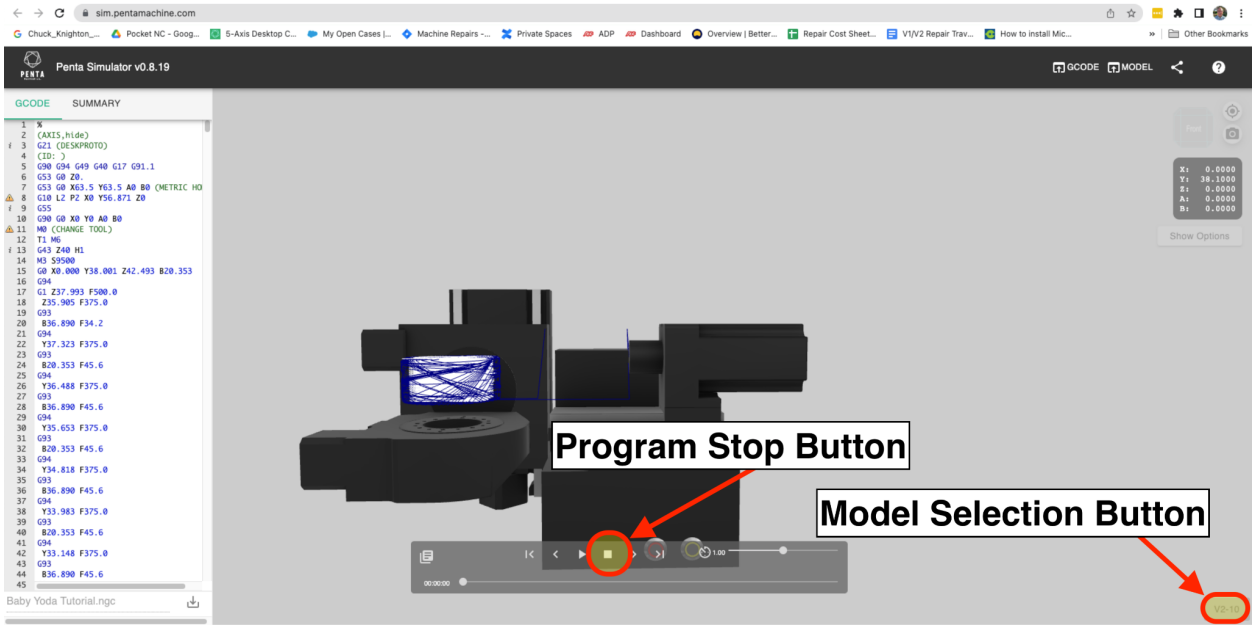


The simulator will now place an M428 (Enable TCPC) at the beginning of the tool operation where TCPC should begin and an M429 (Disable TCPC) at the end of the tool operation where TCPC should end.

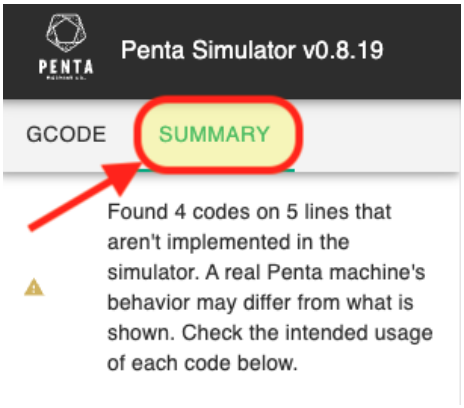




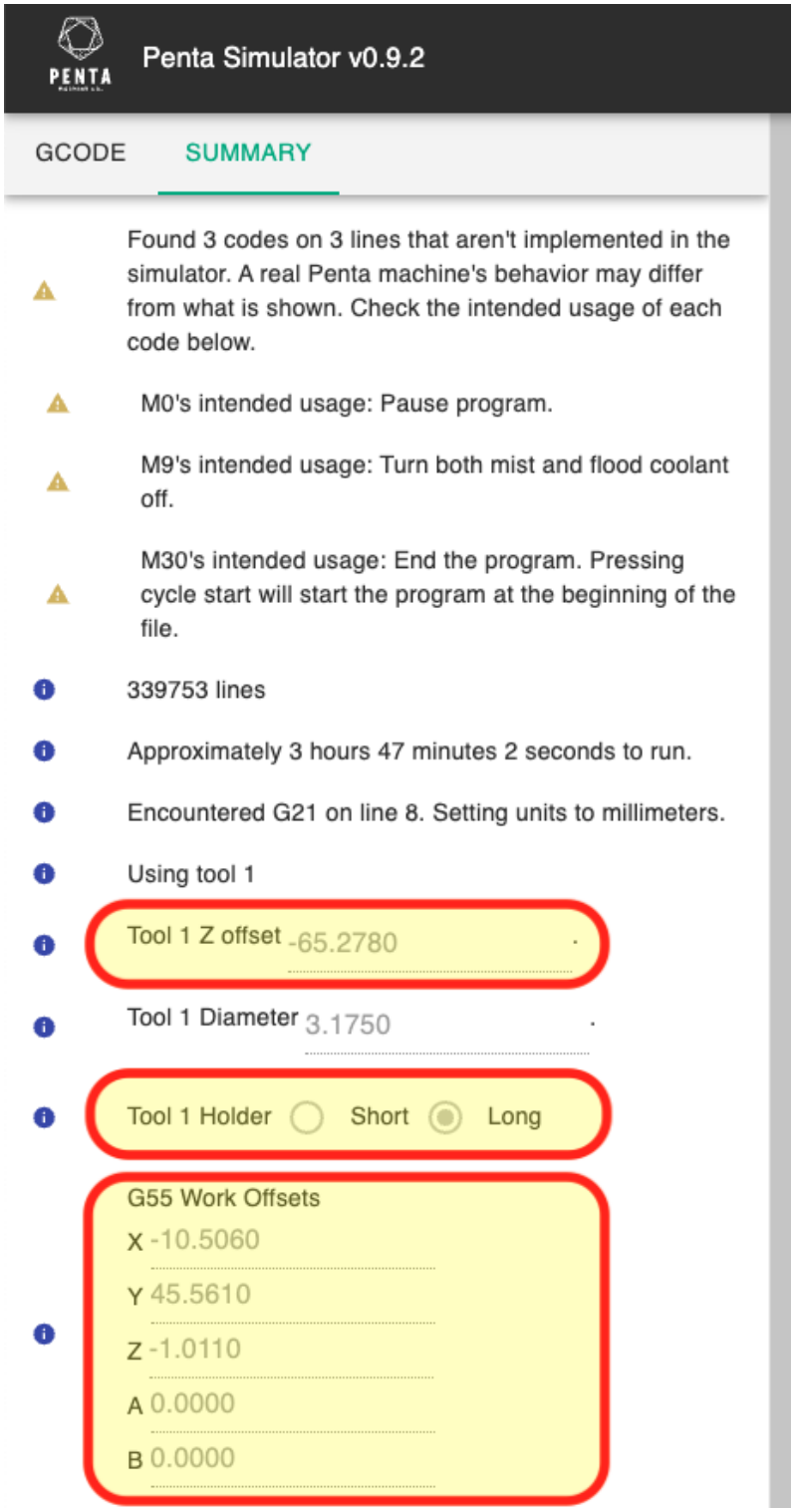
Next select your model of Pocket NC. First click the **Program Stop** button and then click on the **Model** tab in the lower right corner.



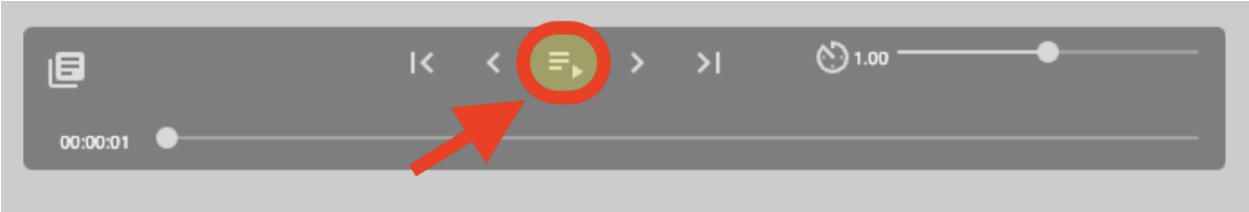
Configure the **Penta Machine** simulator by entering the **Tool Length Offsets** and the **G55 work offsets**. Click on the **SUMMARY** tab.



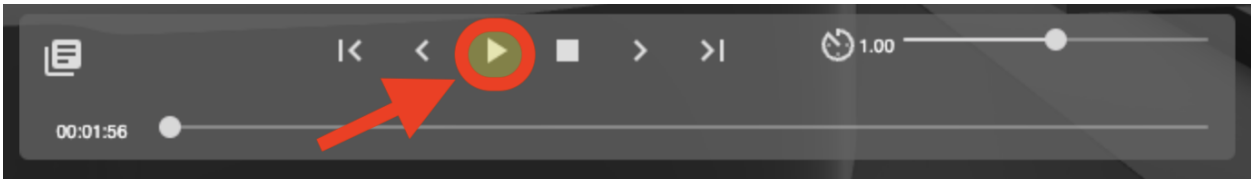
The **Tool Length Offset** and the **G55 Work Offsets** were determined in **Section 13**. By default, the Pocket NC V2-10 machine uses the **Long Tool Holders**.



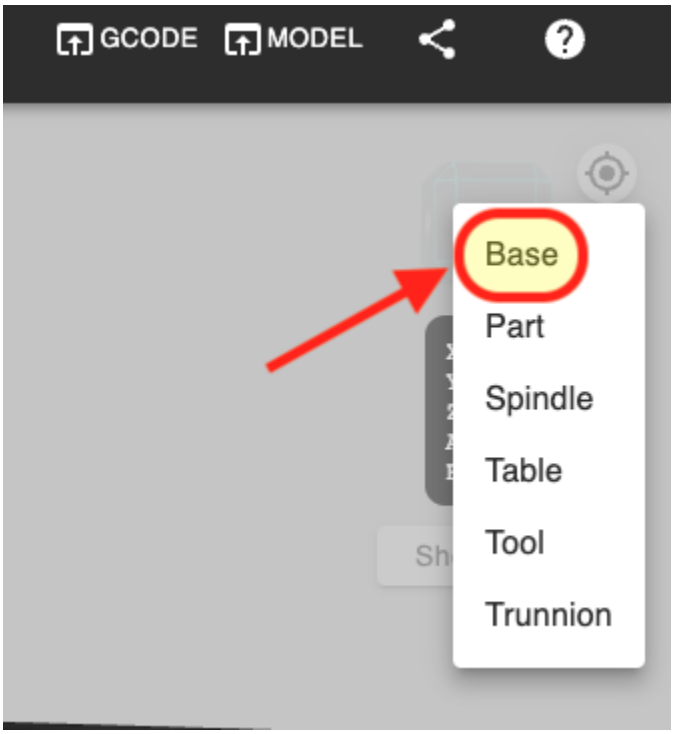
Click the **Program Run** button to accept the values.



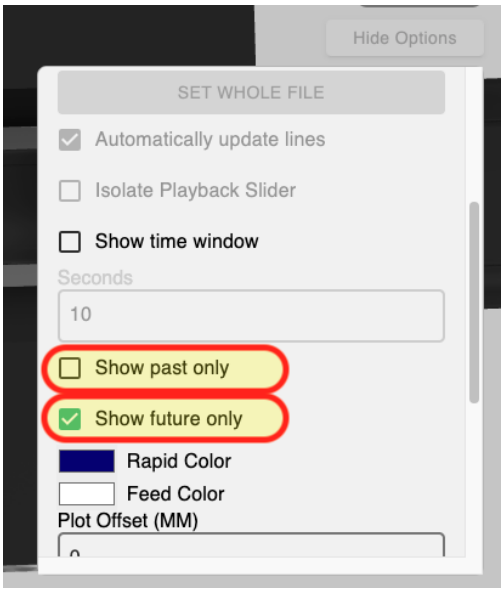
Click the **Play** button to run the program.



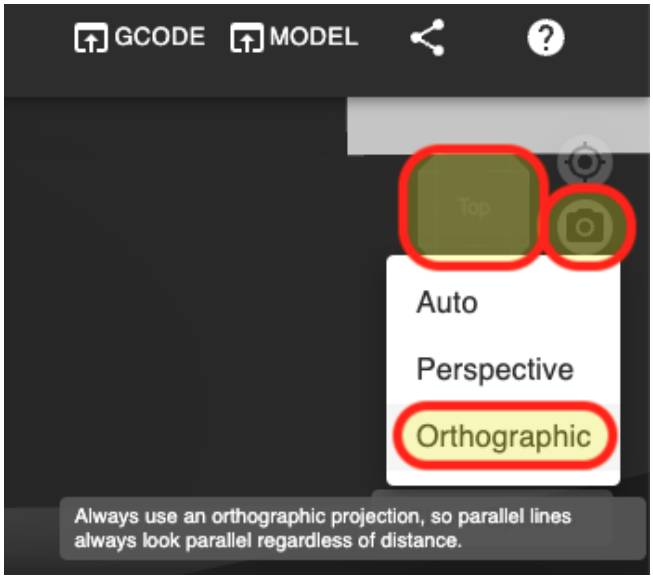
The **Camera Options** button makes it easy to view the desired details. Click on the **Camera Options** button and select the **BASE** option to focus on the part being machined.



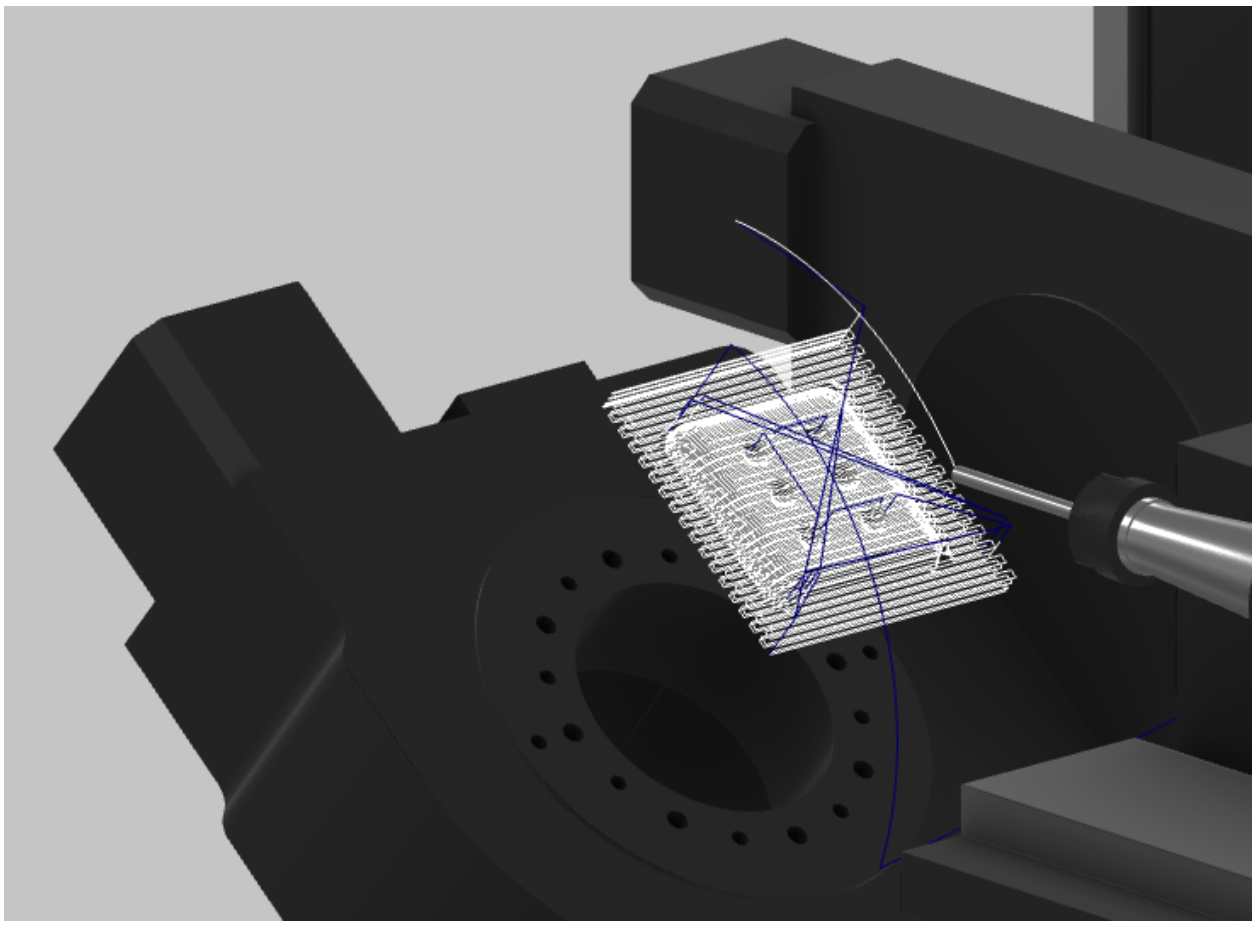
Under **Show Options** select ***Show future only*** and deselect ***Show past only***.



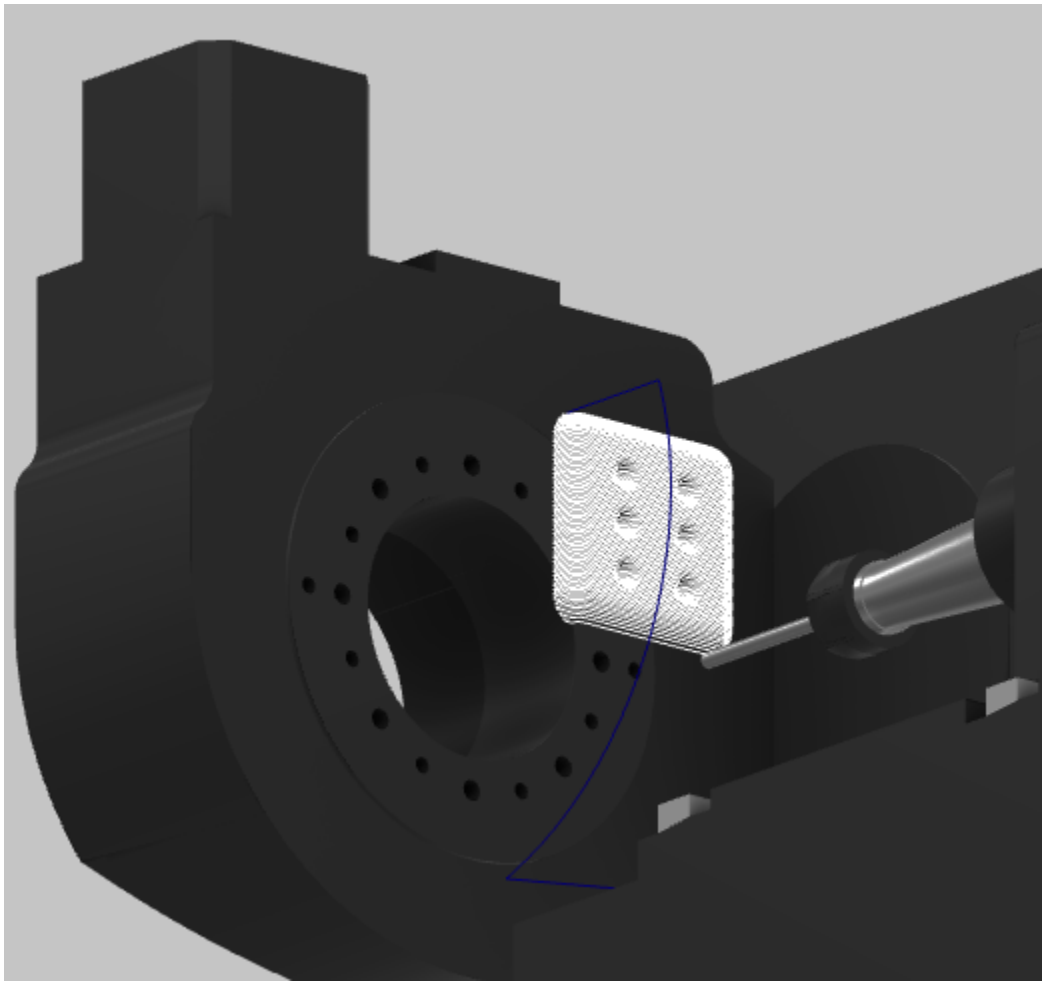
Select the **Top** view of the part. Then select the **Orthographic** view to provide a clearer illustration of the tool paths.



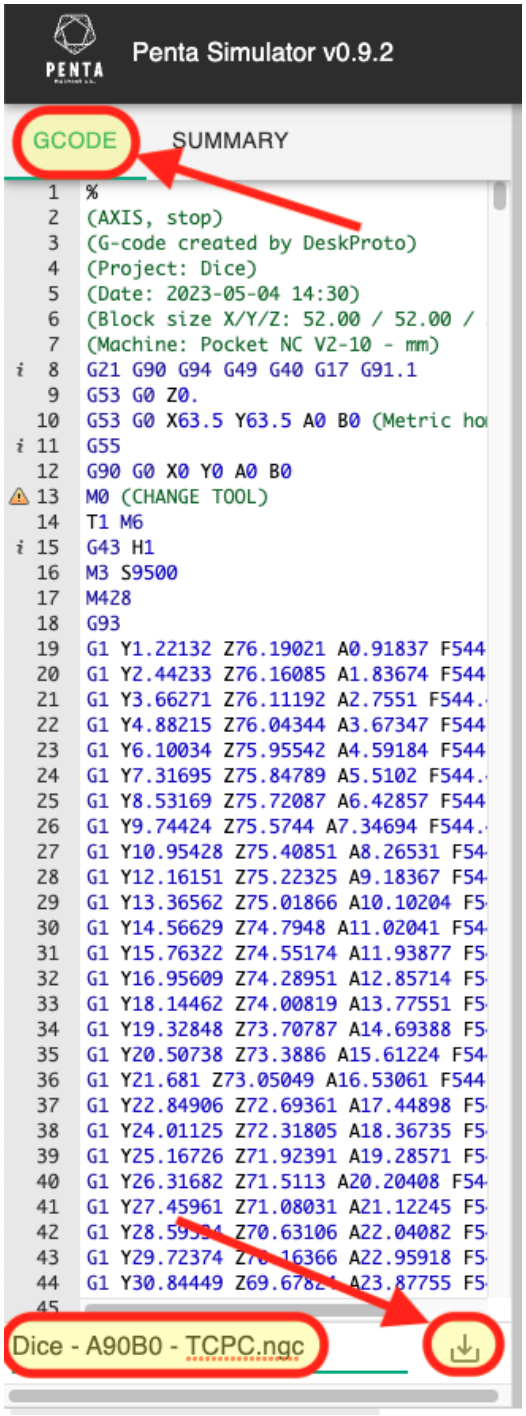
The simulator view should look similar to the image below as the program starts for the top of the dice.



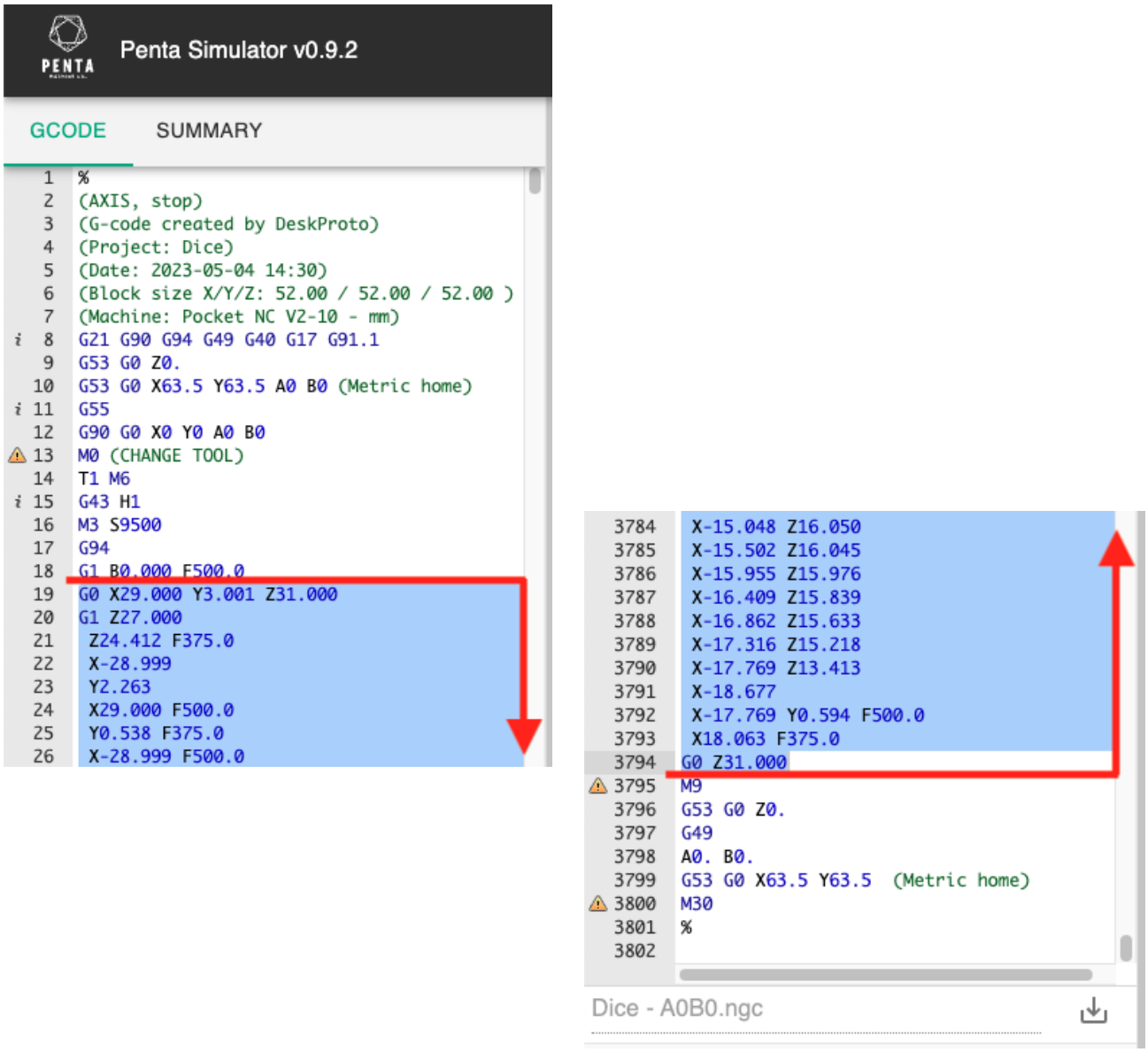
Below shows the simulator as it begins its finishing tool paths for the top of the dice.



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.



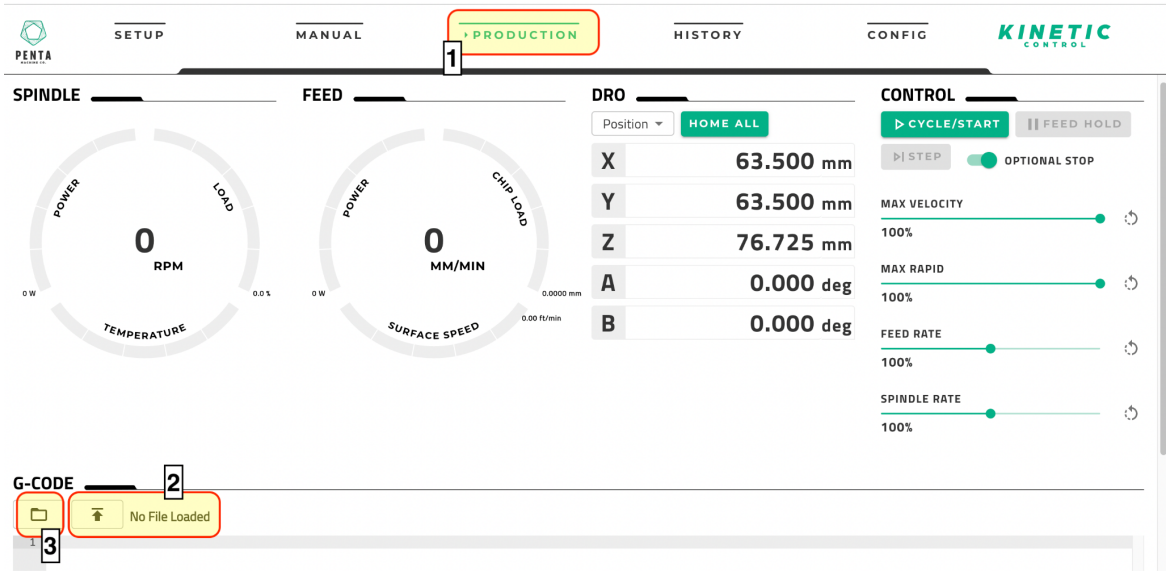
The remaining four sides (g-code programs) of the dice part need to be converted to TCPC. Below is an example of the beginning and ending code for the A0B0 tool operations for the dice part. Select the code after the G1 B0.000 F500.0 rotary command and G53 command (move in machine coordinates). The M9 (turn mist and flood coolant off) is ignored by the simulator.



Follow the above steps until there are five g-code programs, one for each side, converted to TCPC.

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

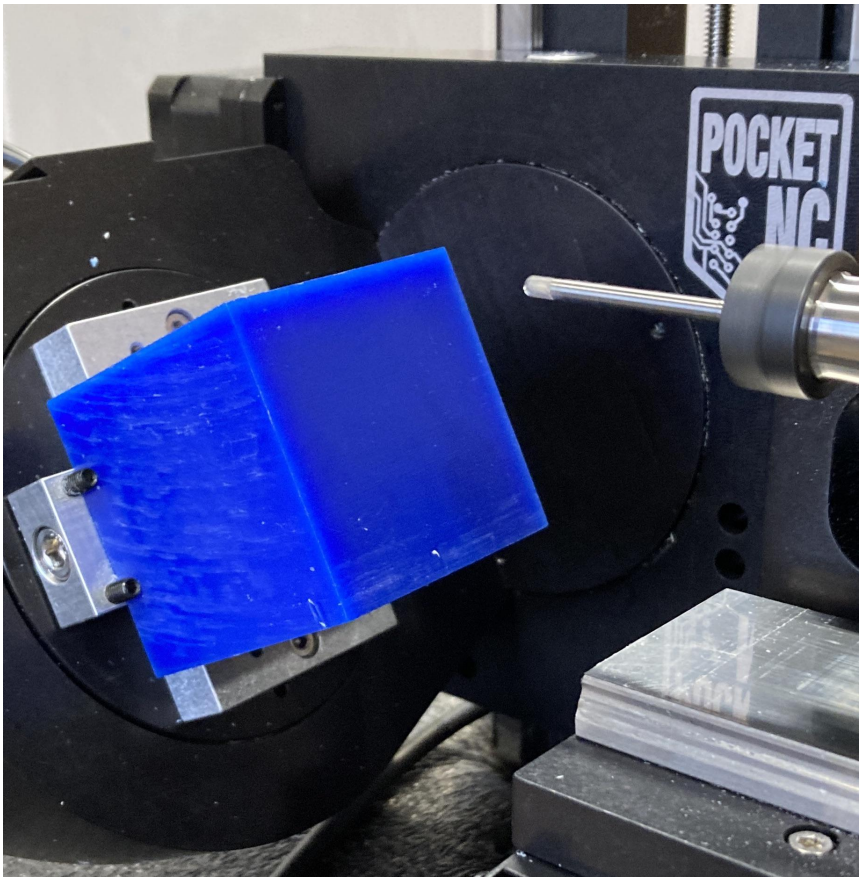
1) Navigate to the **PRODUCTION** tab on Kinetic Control.
2) Select the **File Upload** icon in the lower left hand corner of **Kinetic Control** and choose the g-code file.
3) Finally, select the file to run from the **File Folder** icon.



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.



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.

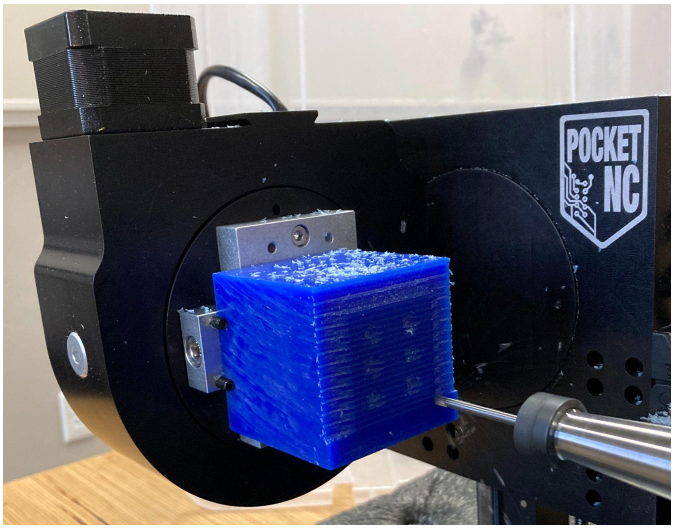
Resume the program by clicking the **CYCLE/START** button on the **PRODUCTION PAGE**.



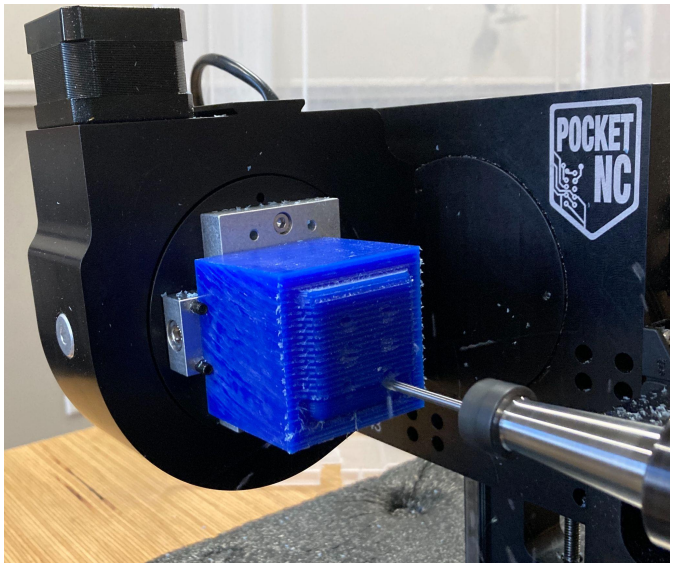
Section 16: Index milling process on Pocket NC V2-10

The A90B0 g-code begins by removing the majority of the stock above the number 6 surface of the dice part, beginning at the top of the stock. Once the roughing cut is completed, the A90B0 g-code then begins finishing the number 6 surface of the dice, beginning at the bottom of the part.

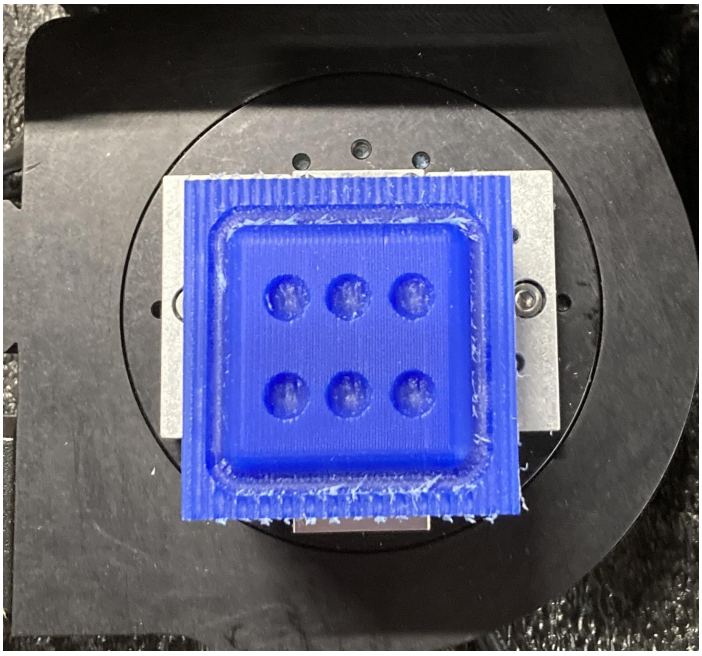
Dice - Side 6 - Roughing



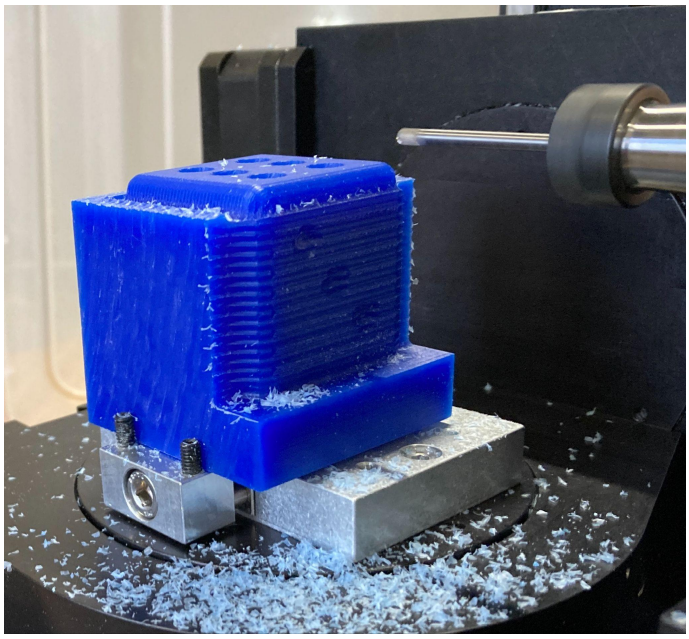
Dice - Number Side 6 - Finishing



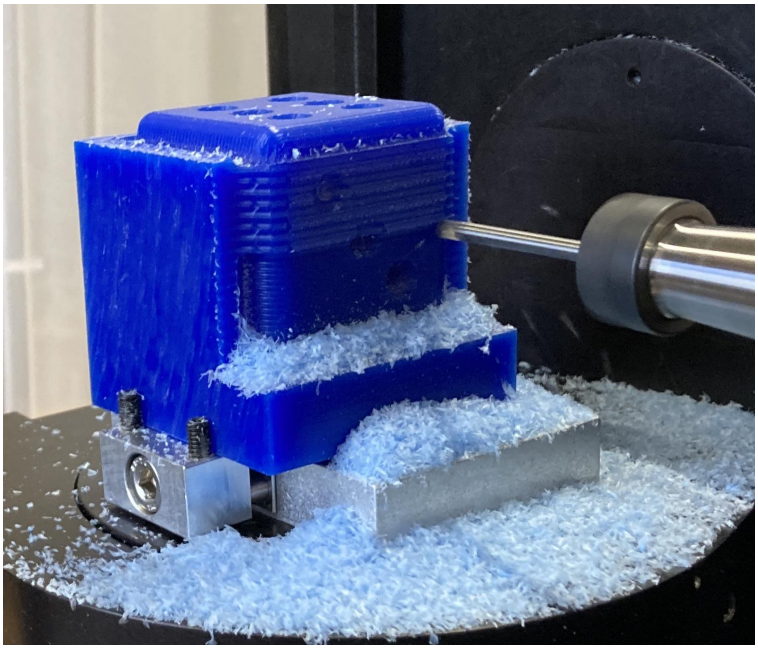
Dice - Number Side 6



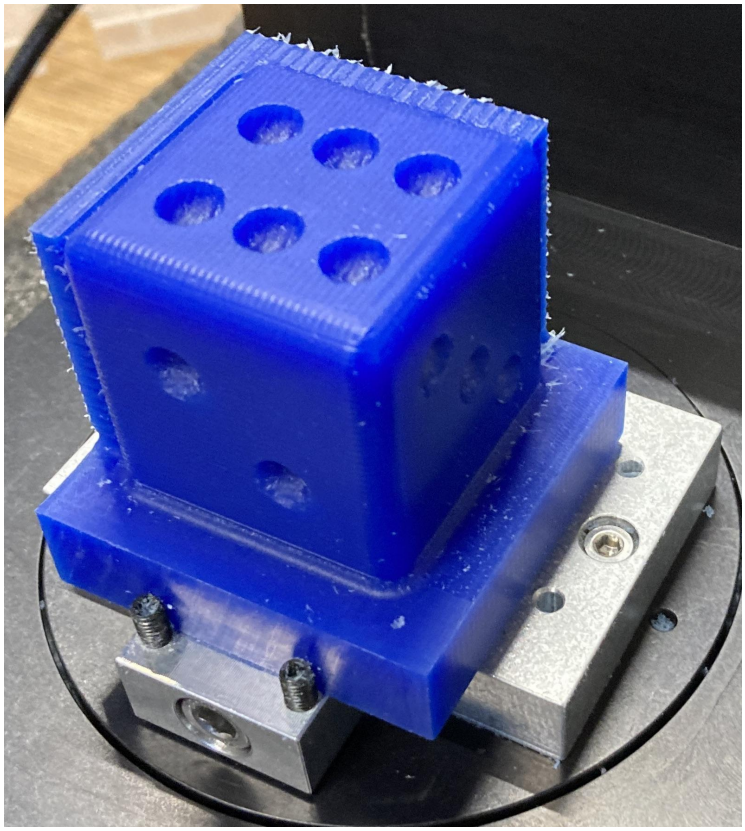
Dice - Number Side 3 - Roughing



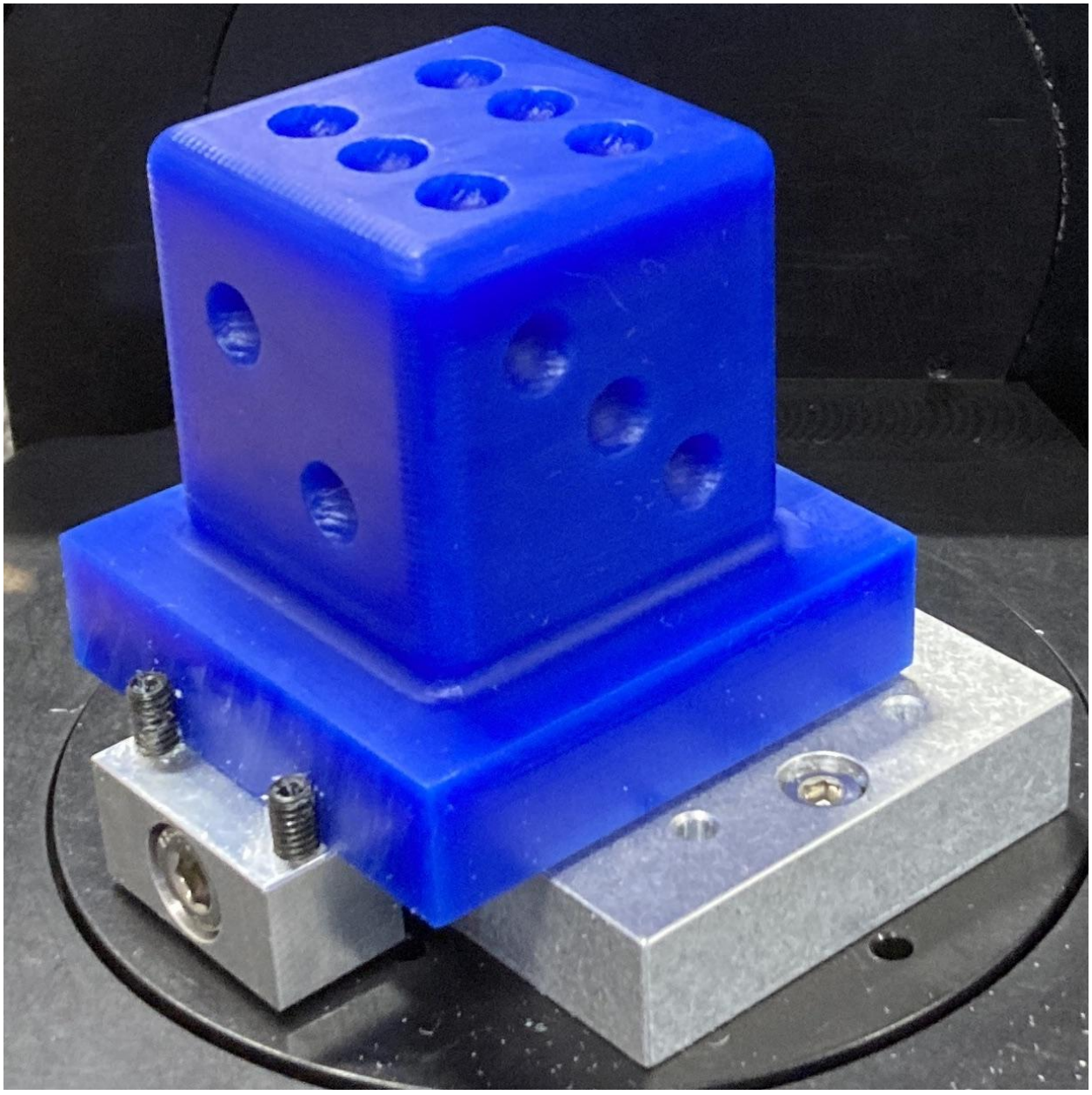
Dice - Number Side 3 - Finishing



Dice - 3 of the 5 sides - Completed



Finished Dice Part



Conclusion

The purpose of this tutorial was to familiarize you with the functionality of the Desk*Proto* CAM software and how to create index (3+2) 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.