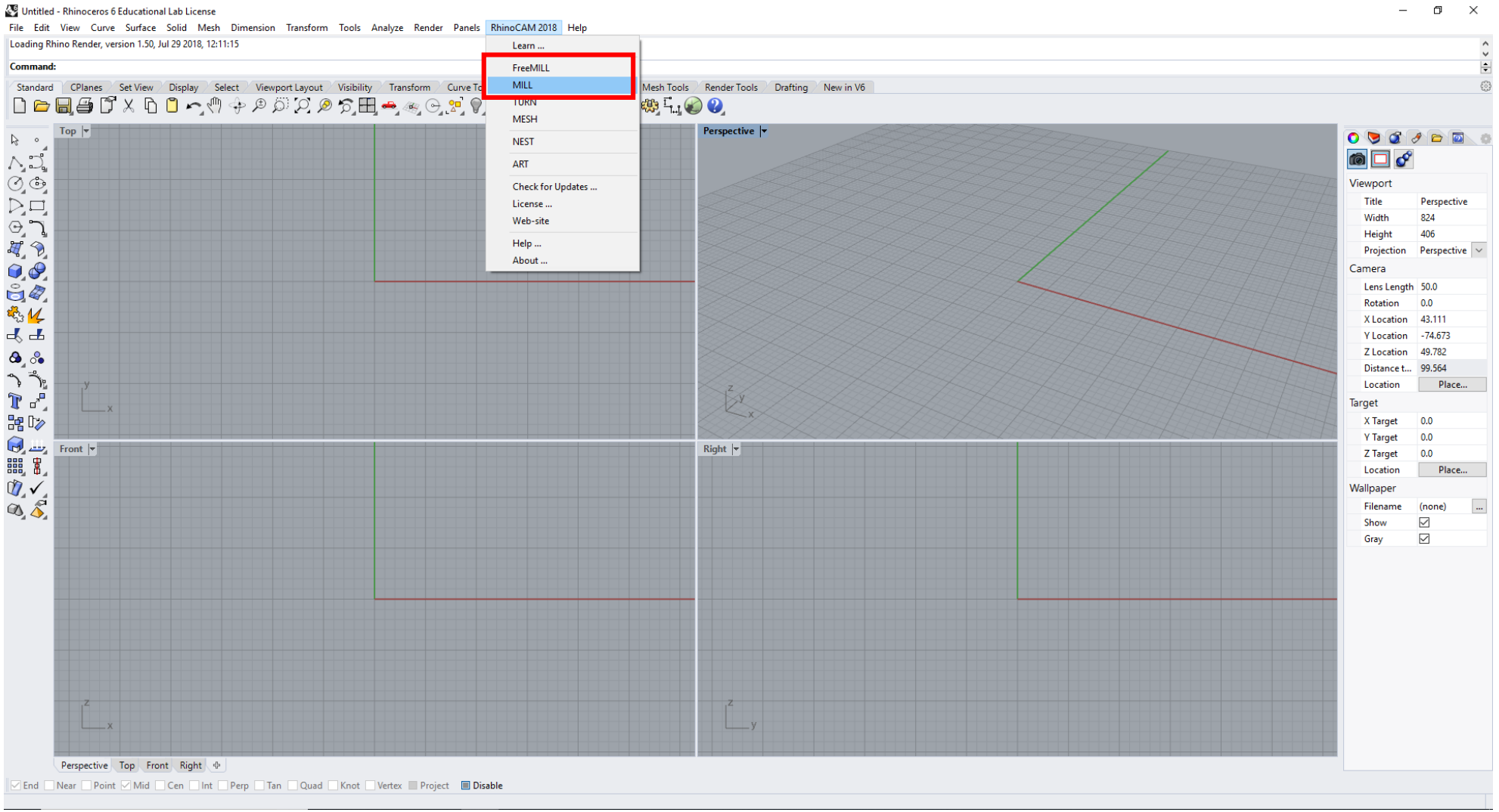
CNC
(Computer Numerical Control)
using RhinoCAM

Introduction

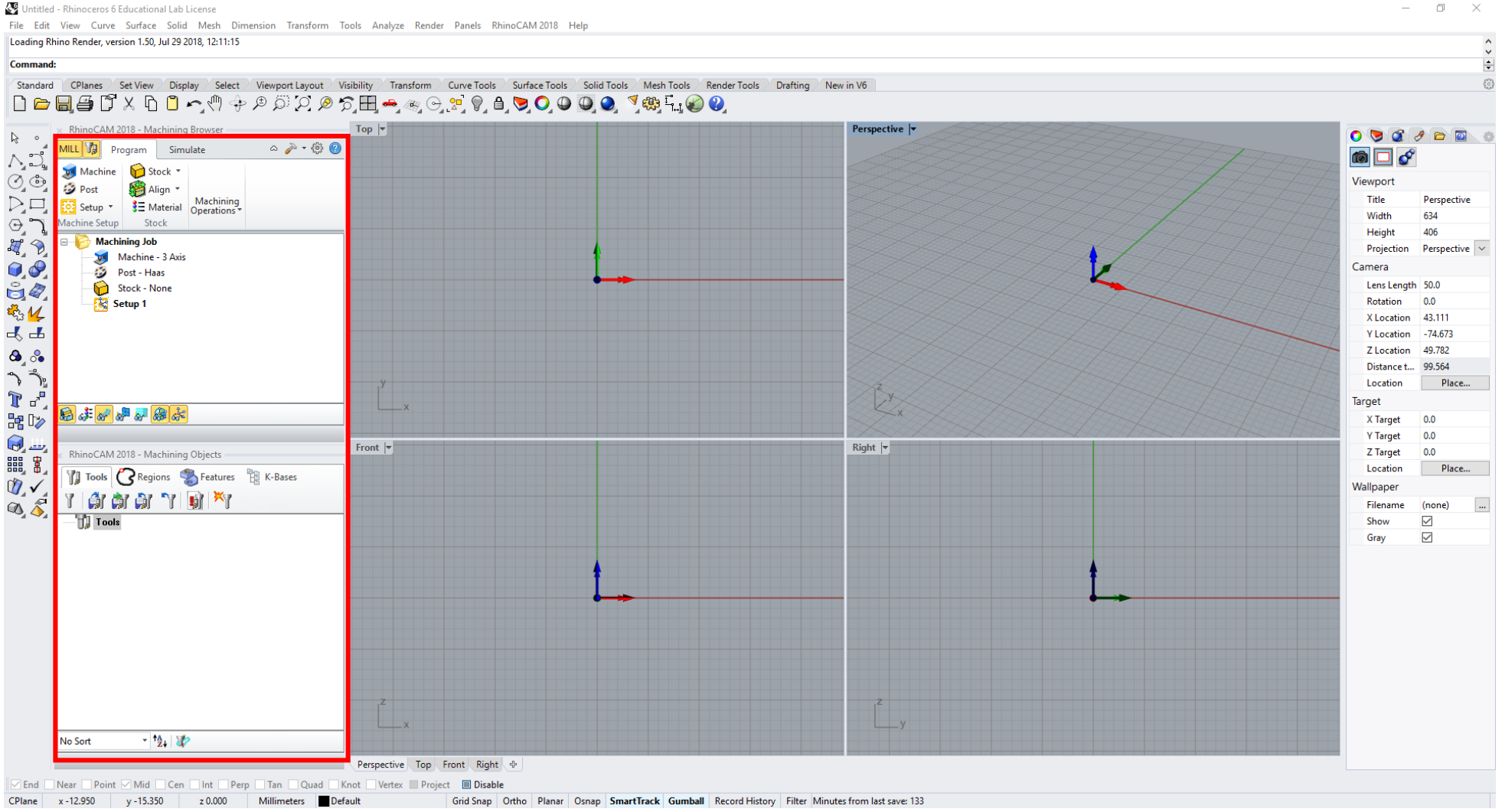
Niloufar Emami

Digital space: RhinoCAM

RhinoCAM will automatically load when you open Rhino
To load “milling” operation, you can select it from the menu



Mill operation loaded
we’ll start to go over operatins from top left



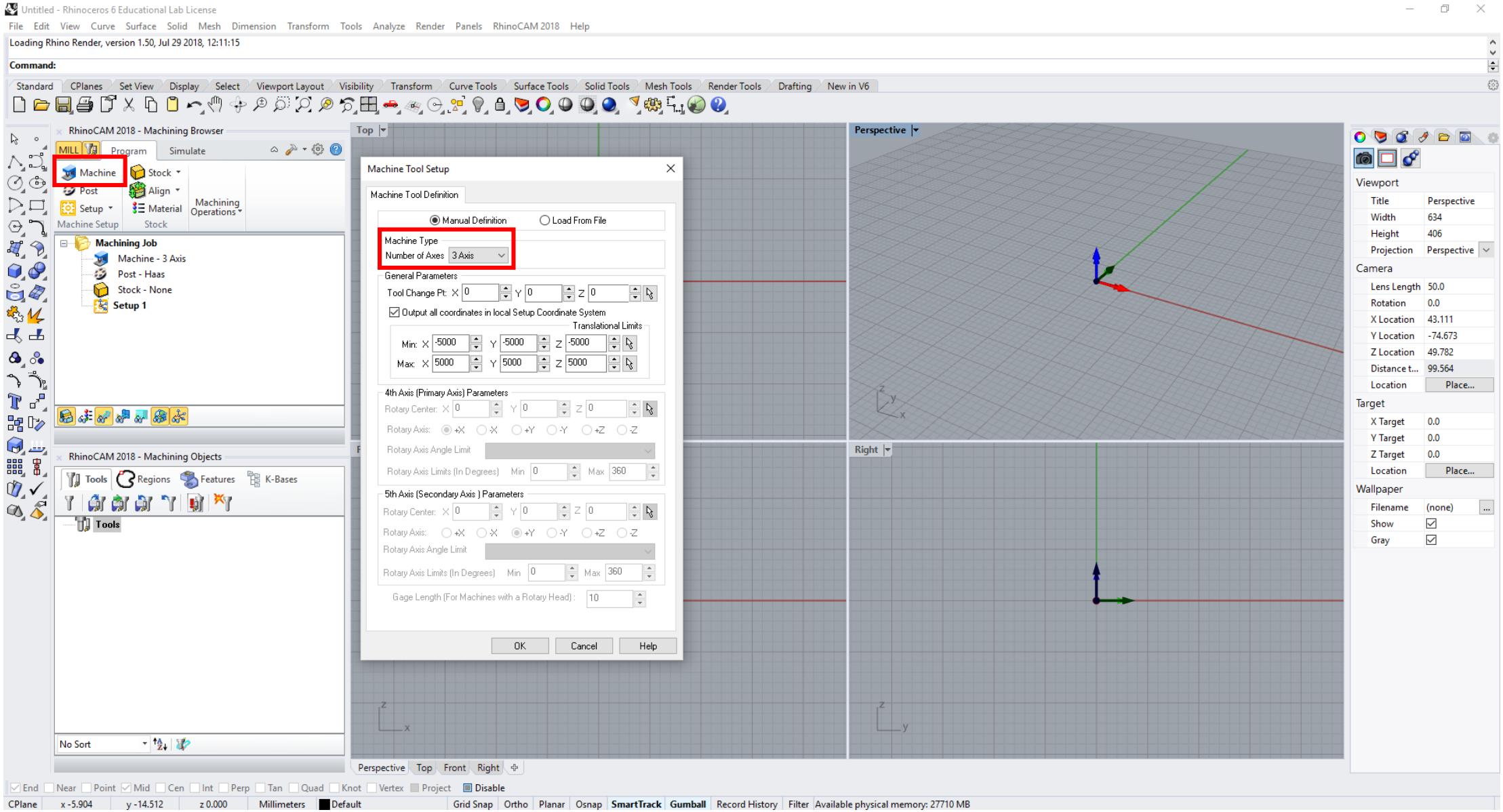
Step 00: Load the Part Model

“Part” refers to the geometry that represents the final manufactured product. You can create parts within Rhinoceros or import geometry created in another CAD system.
Select File / Open from the Main Menu bar, or click the Open icon from the Standard bar.

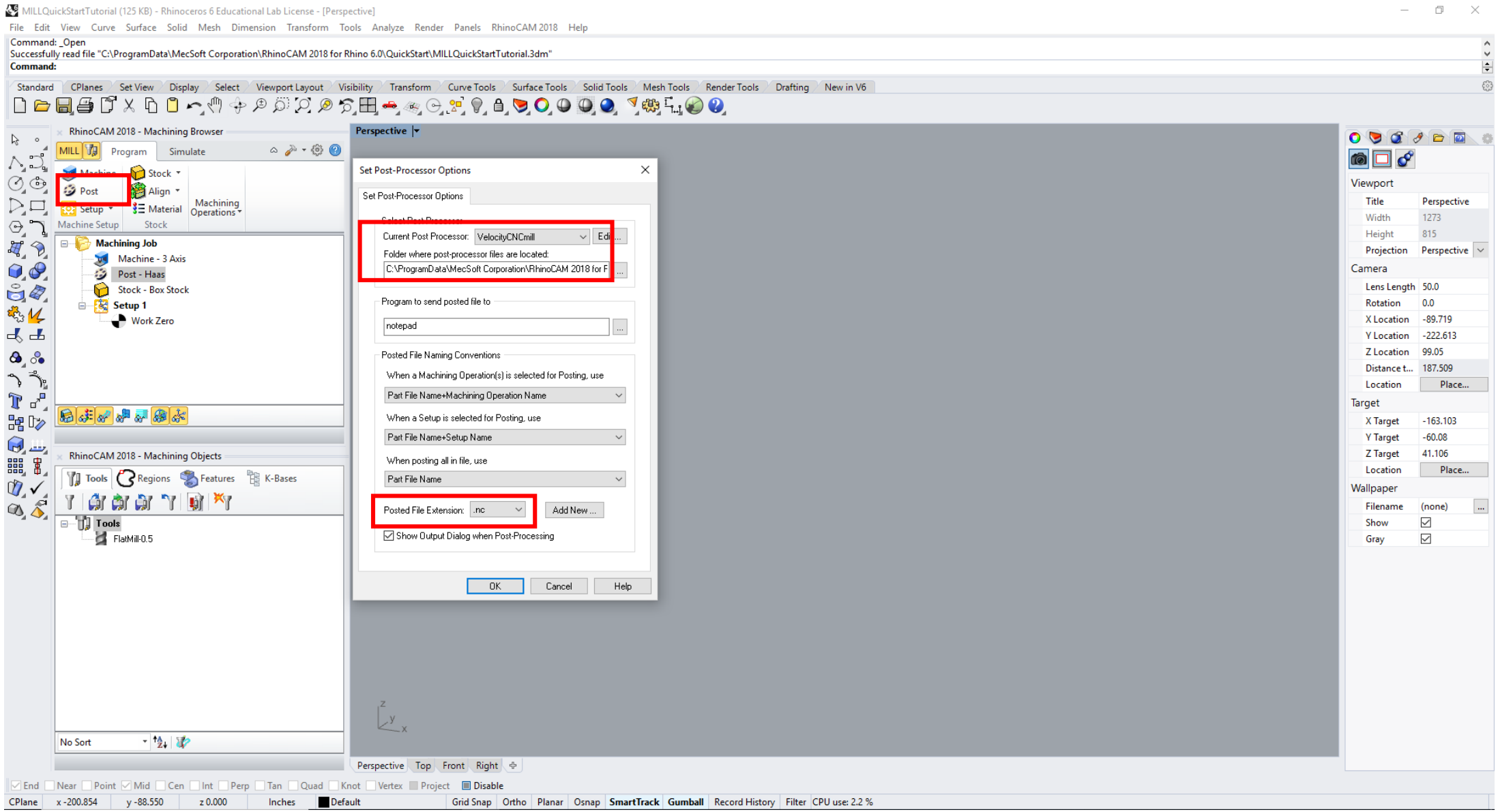
Step 01: set the Machine and post processor

- We will be using a 3 axis machine (it only moves in the x, y, and z axis)
- The post processor that we use in the shop is named “Velocity CNC Mill”

Machine: 3 Axis must be selected



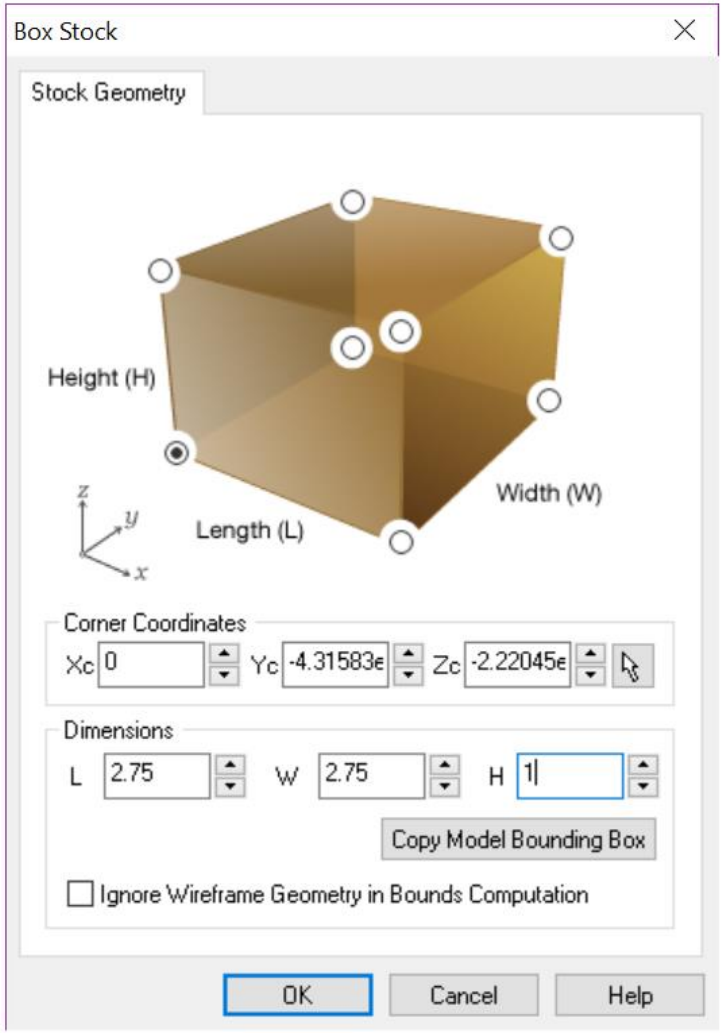
Post: Velocity CNC Mill



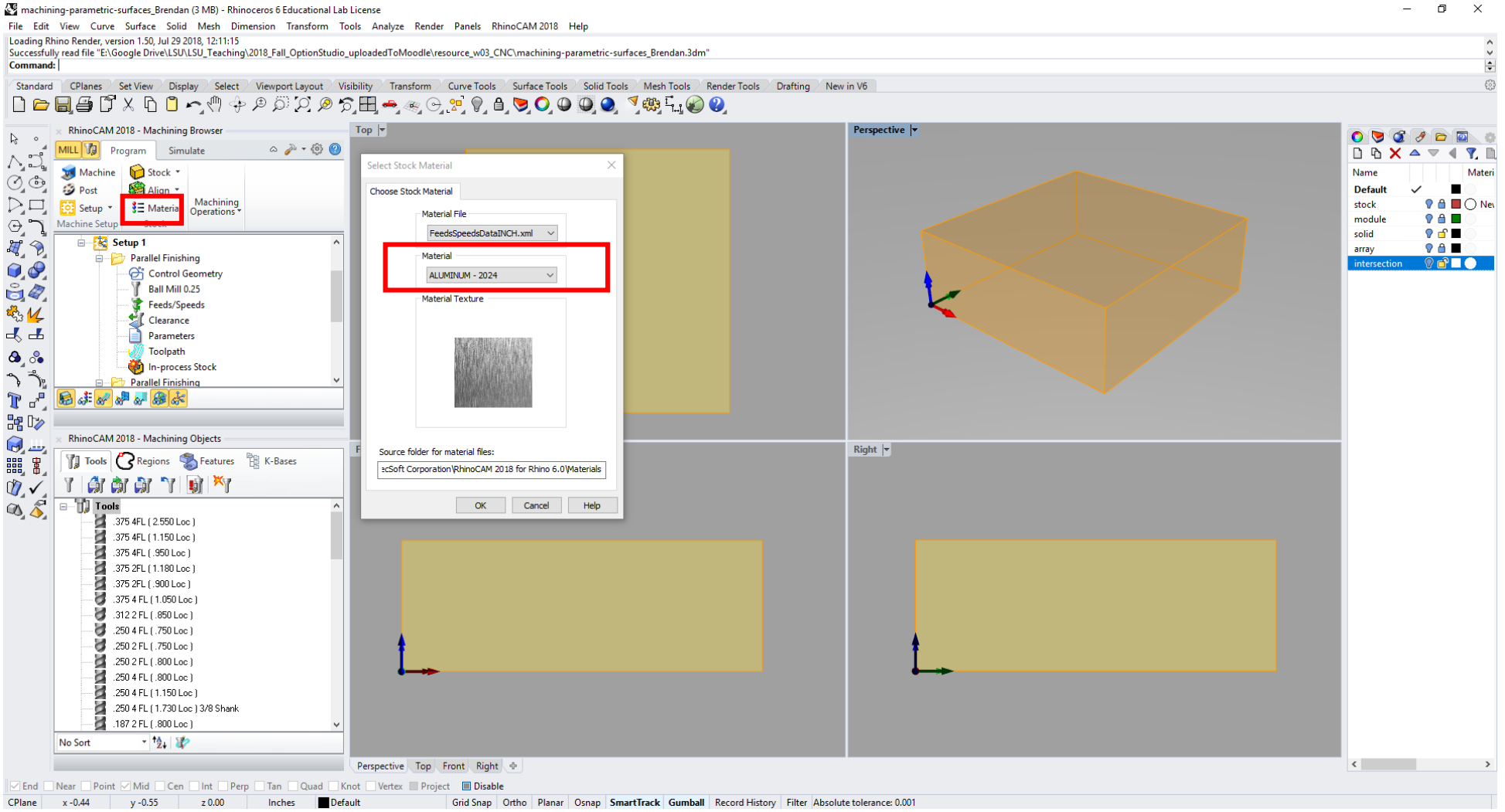
Step 02: define the machine setup (including stock geometry, material, and work zero)

- You will be cutting your part from a stock geometry. We need to define the dimensions of the stock geometry.
- You can route different materials, and the feed rate and speed of the machine will be set accordingly. Therefore, you need to set up the material.
- The Work Zero is a specific point you will define in the Cartesian coordinate system that you can reference in both the digital space in RhinoCAM, and in the physical space on the stock that you intend to cut your shape out of. The CNC mill will reference this point for every movement it makes and it is also how we justify and align our digital drawing within our physical stock. The Work Zero can be any point you set it to be. As long as you can identify it on both the work piece and the drawing.

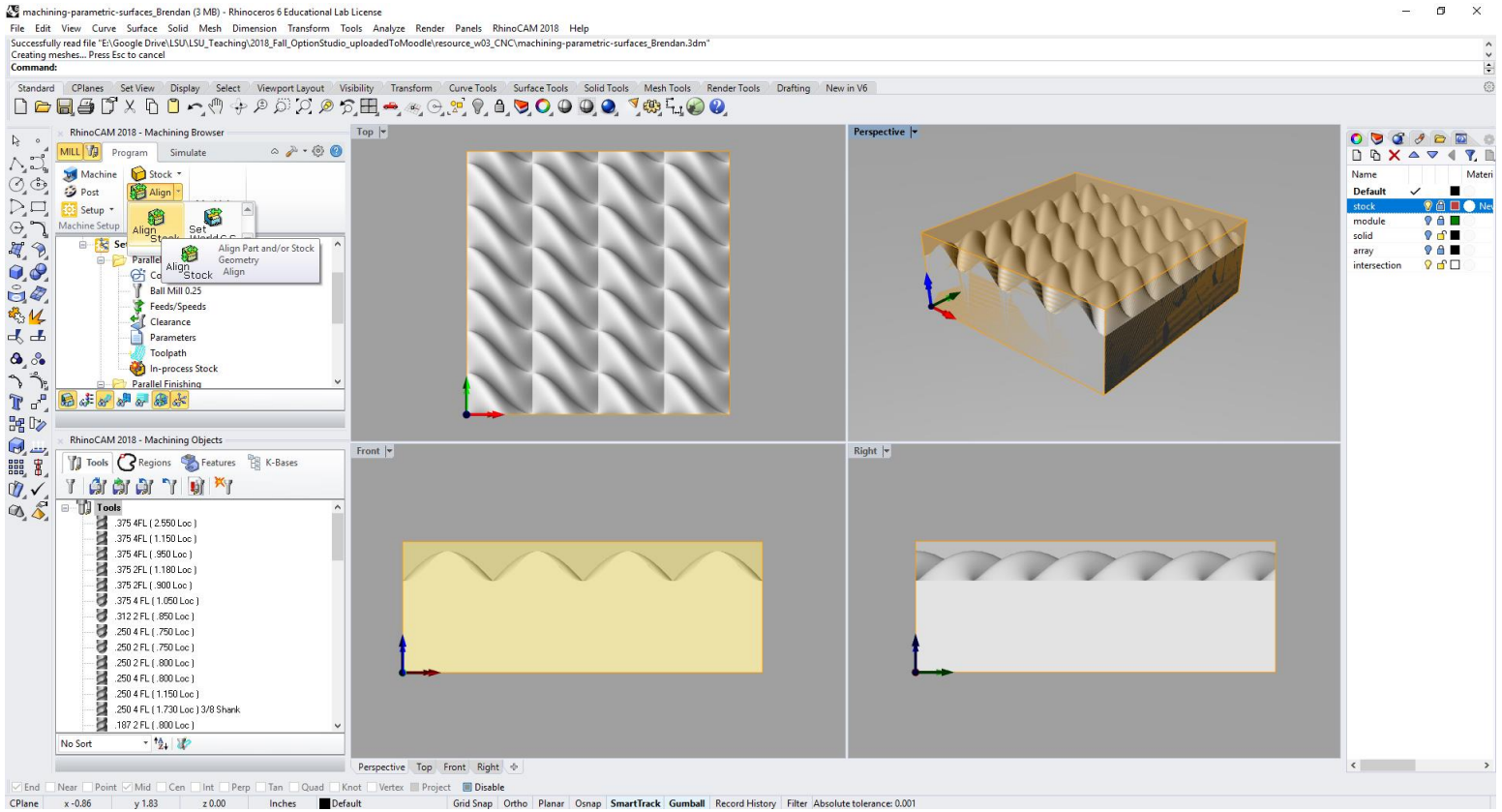
Stock geometry: L=2.75, W=2.75, H =1
Note that the stock dimensions you enter are measured from the corner of the bounding box selected in this dialog. You can click on different corners.

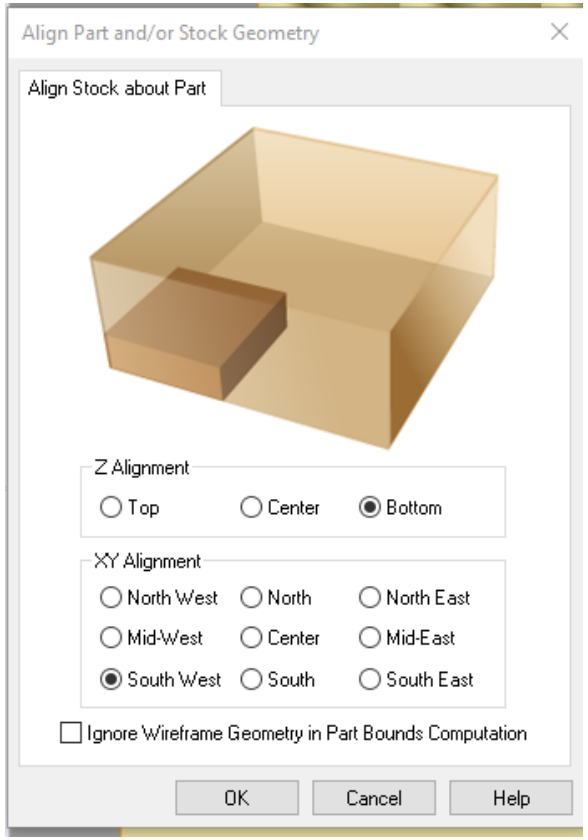


Material: For milling the Richlite Black Diamond(here), choose “Aluminum 2024”



Align part and stock material





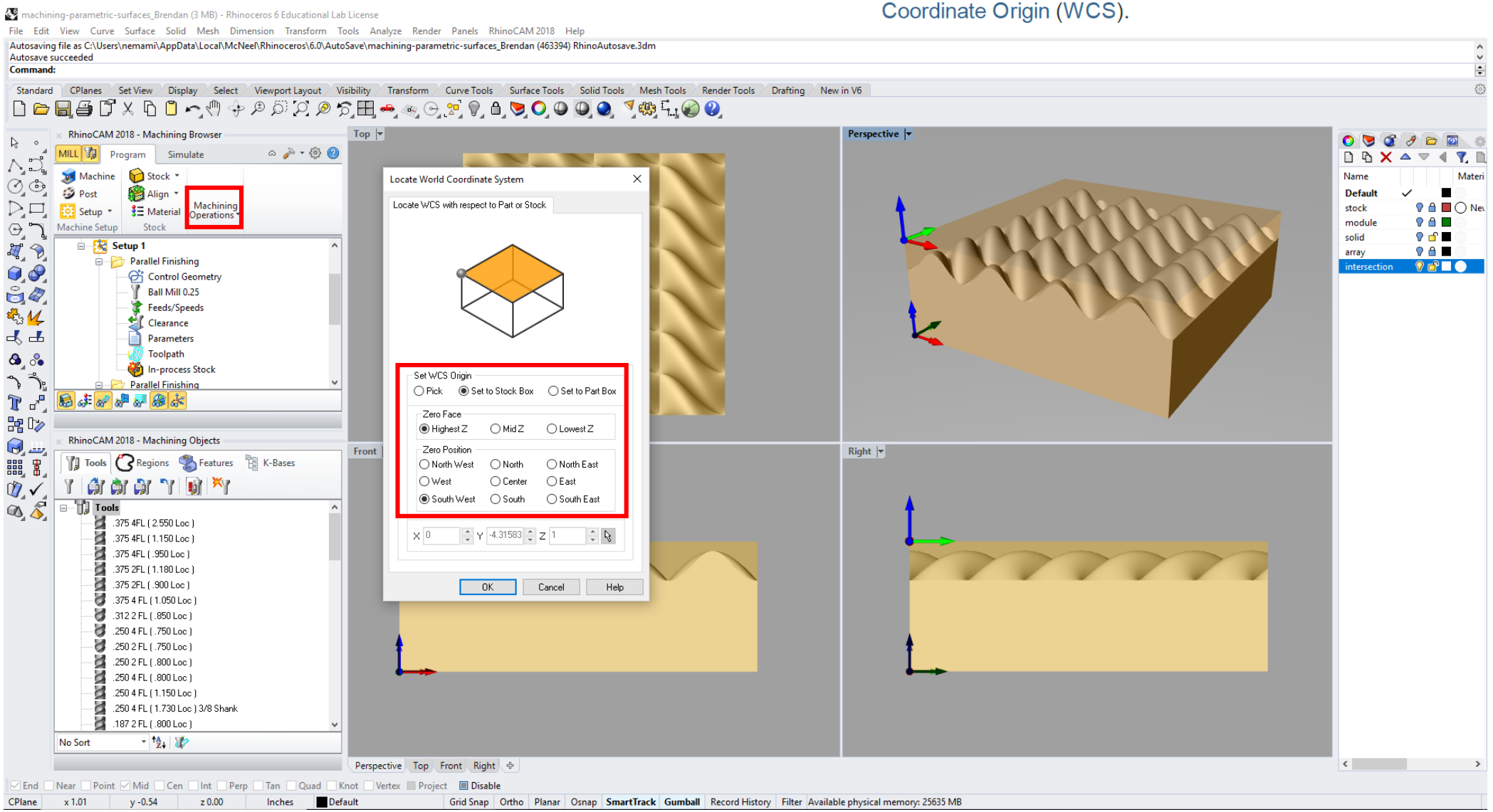
- Let's say you are cutting a small part out of a bigger stock material. You need to align it to the stock so that you know which portion of the stock will be milled.
- In this example, the stock and part size match (2.7" by 2.7" by 1")

Step 03: work zero

- In the previous steps, you aligned the part to the stock material
- Now you need to define the work zero
- The Work Zero defines the zero point with respect to which all toolpath points are interpreted by the controller. This would normally be the same as the tool touch off point on the actual work-piece on your machine. So care should be taken to make sure that this Work Zero point defined in

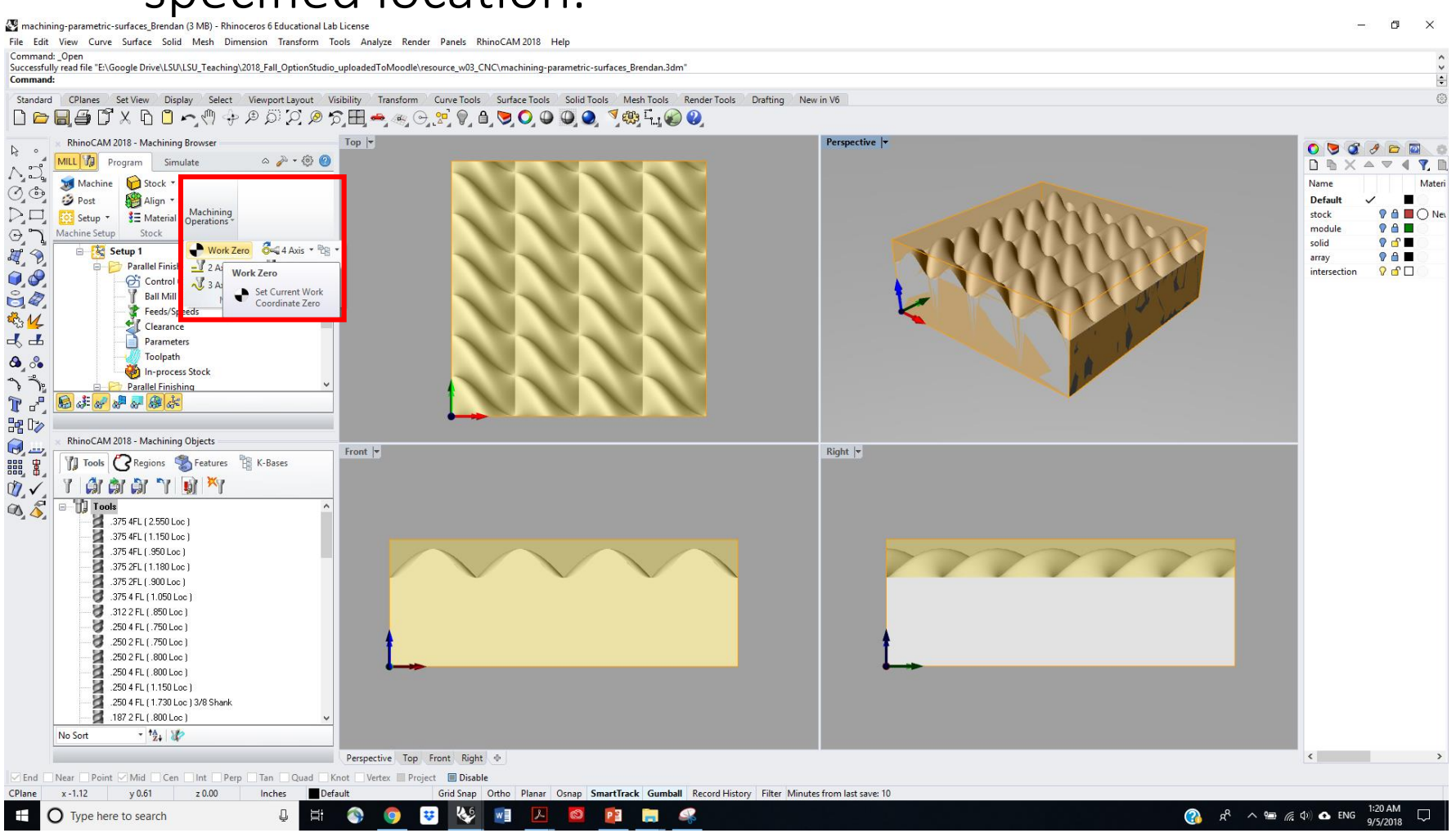
Click on Align and select “set world CS” then pick “set to stock material” and “SouthWest” on top or “highest z”

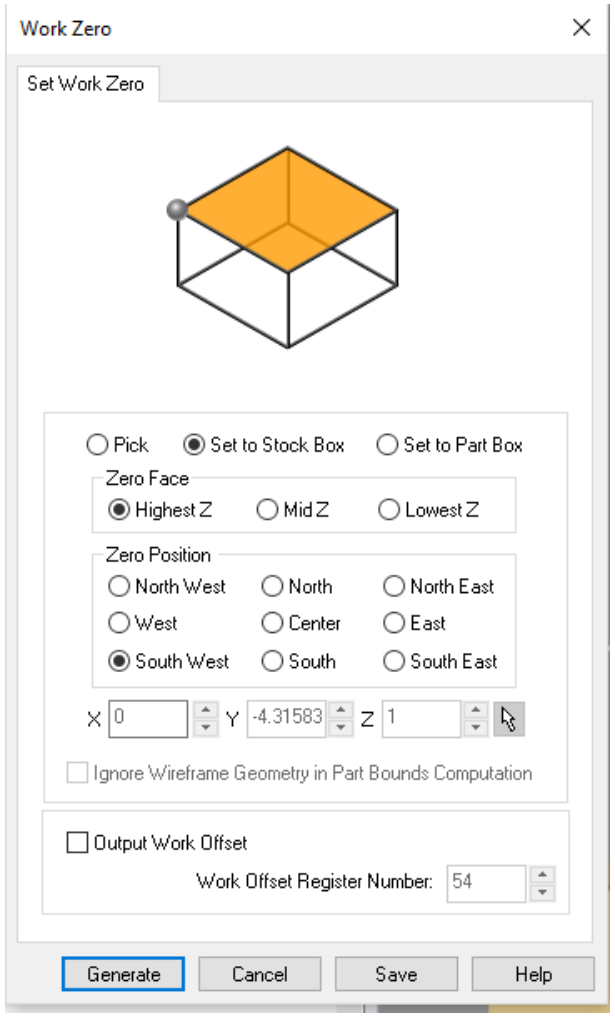
2. Then select Set to Stock Box.
3. Then set Zero Face to Highest Z and Zero Position to South West corner. This sets the machine home to the top of the stock material and the southwest corner of the stock geometry.
4. Pick Generate and the part and stock geometry are now transformed to the World Coordinate Origin (WCS).



If you do this, it will move your part and stock material (tied together) to negative z, in a way that the highest point of your stock material is located on a plane with z coordinates of 0.

Alternatively you can use Work Zero to set the work coordinate origin. Instead of moving the part and stock to the WCS (Work Coordinate system) origin, this moves the machine coordinate system origin to the specified location.





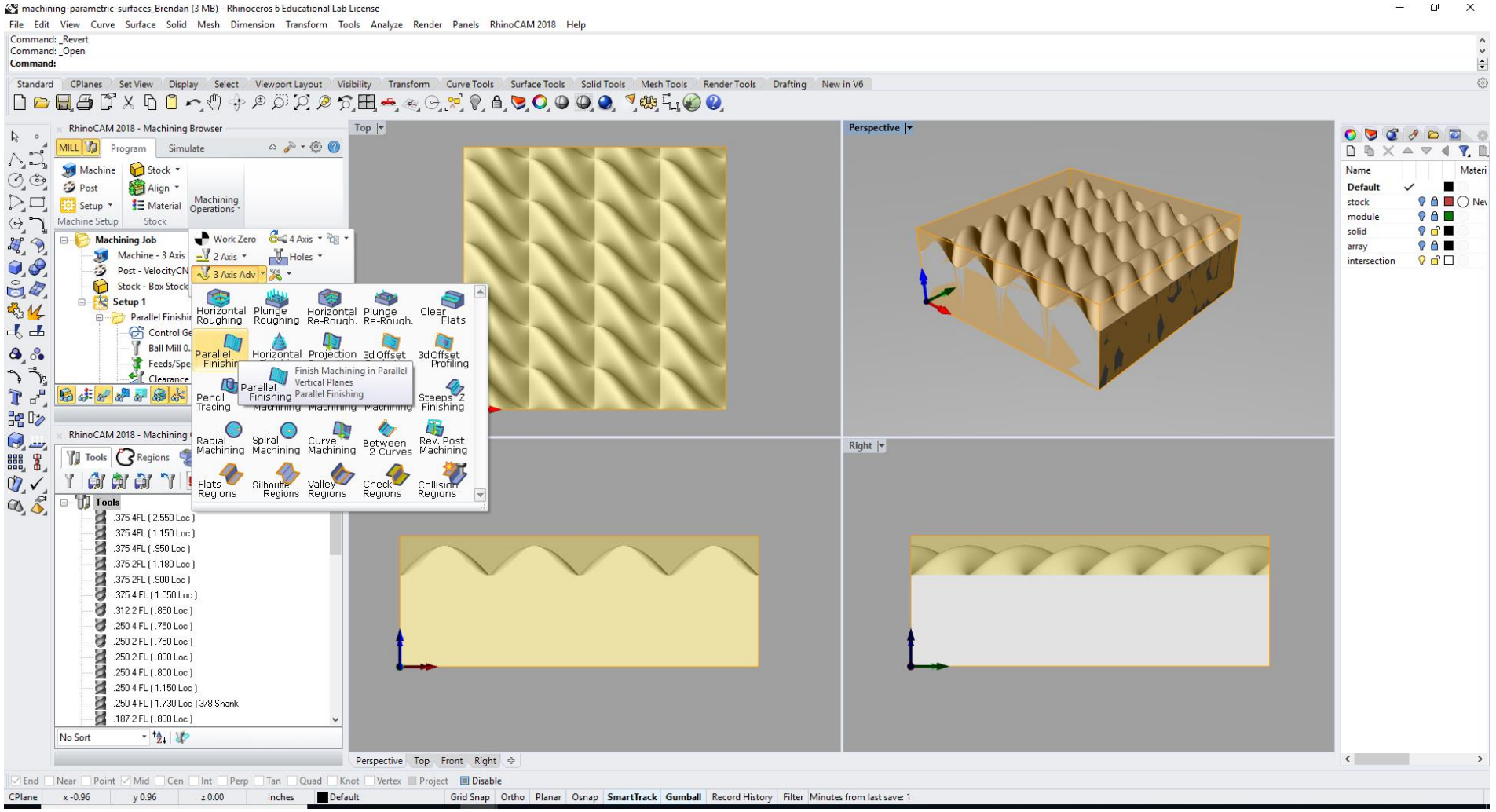
This sets the machine home to the top of the stock material and the southwest corner of the stock geometry.

Step 04: set the toolpath

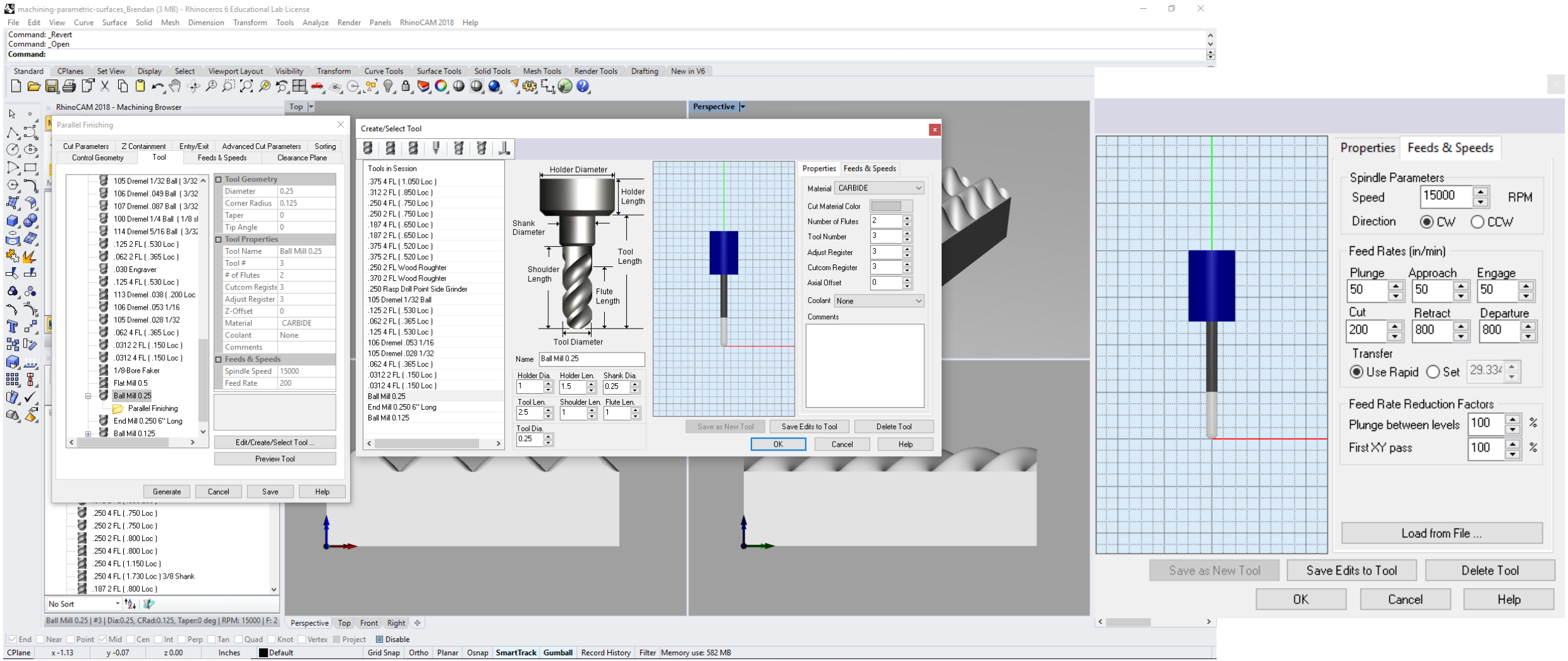
- Now you can set up the toolpath
- We will be doing a rough cut first, and then a finishing operation
- Because of the scale of the part, the rough cut that we do first will be selected from “parallel finishing” in the menu.

Setting the first toolpath:

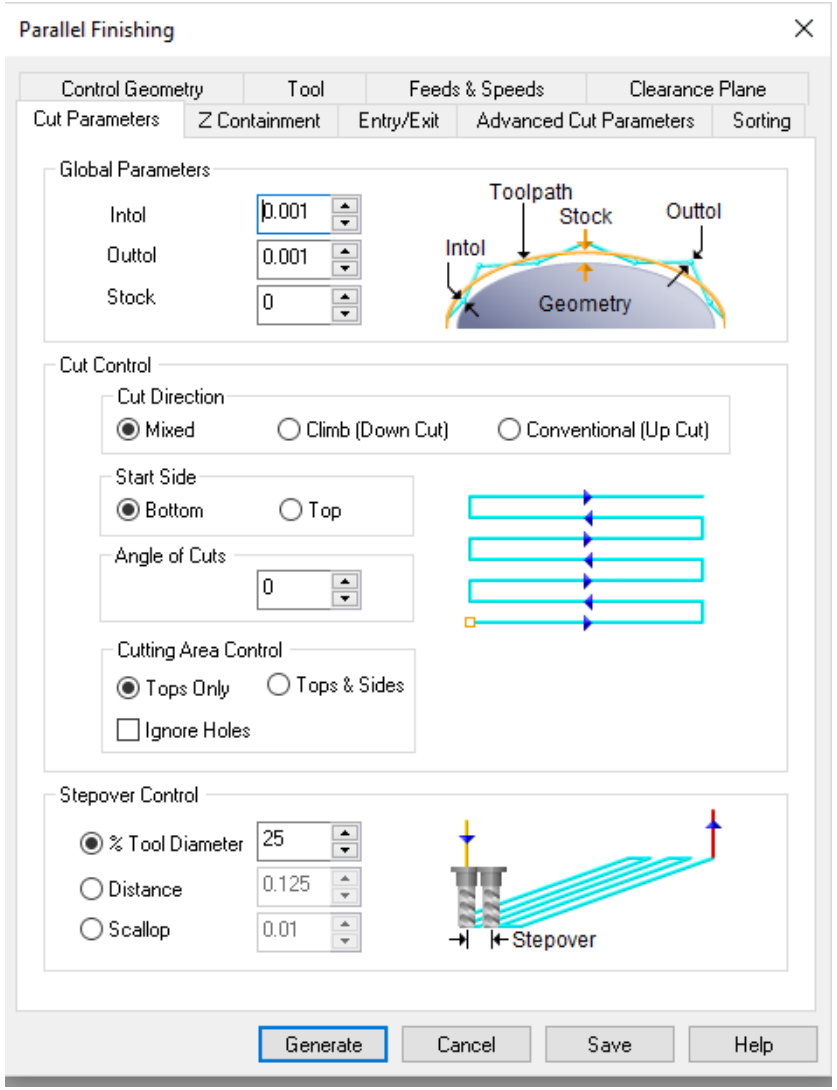
Machine operations-> 3-axis Adv -> Parallel finishing



In the TOOL parameters, scroll down the screen and select Ball mill ¼”

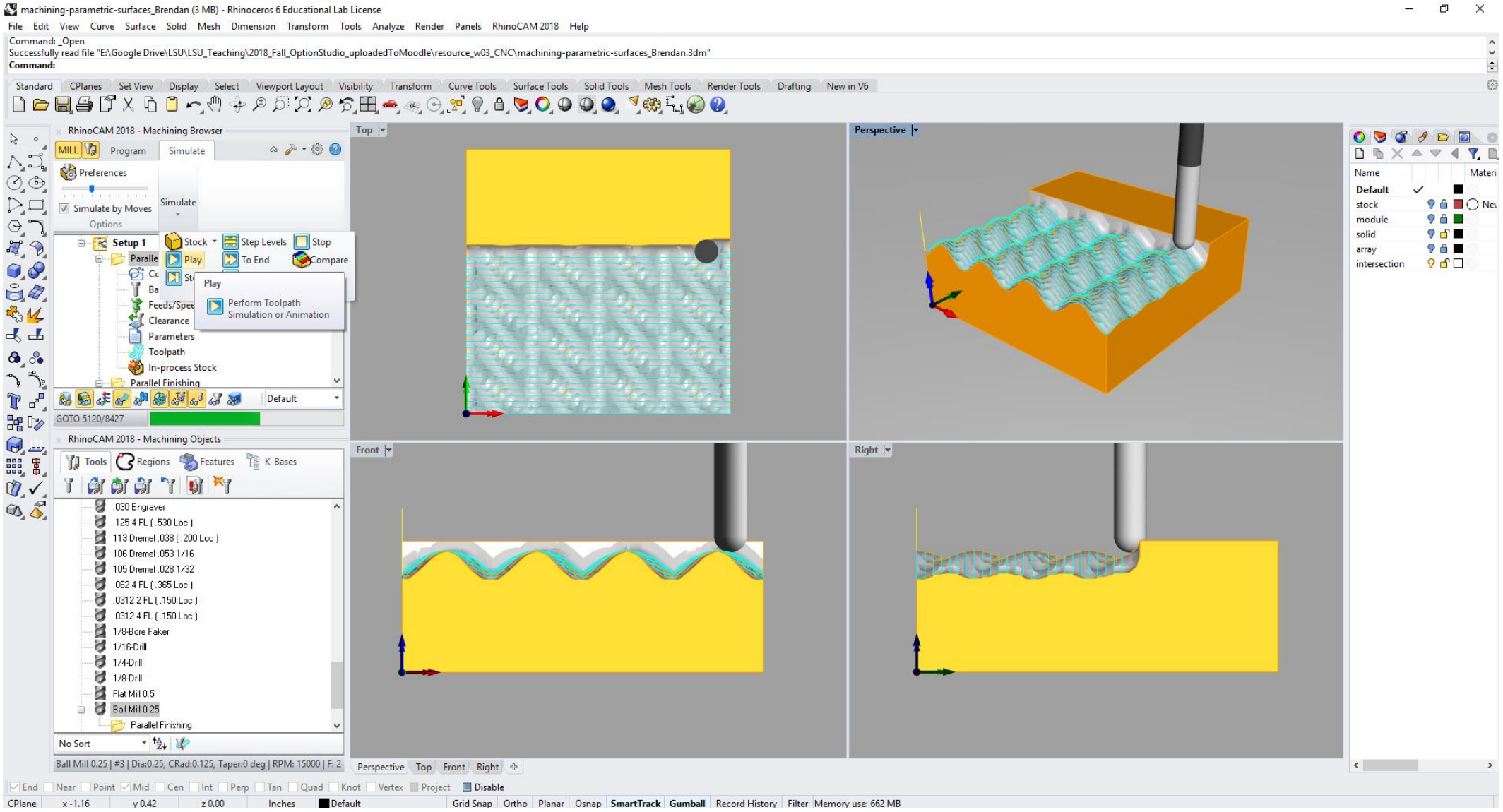


In the CUT parameters, set stepover control to 25% of tool diameter



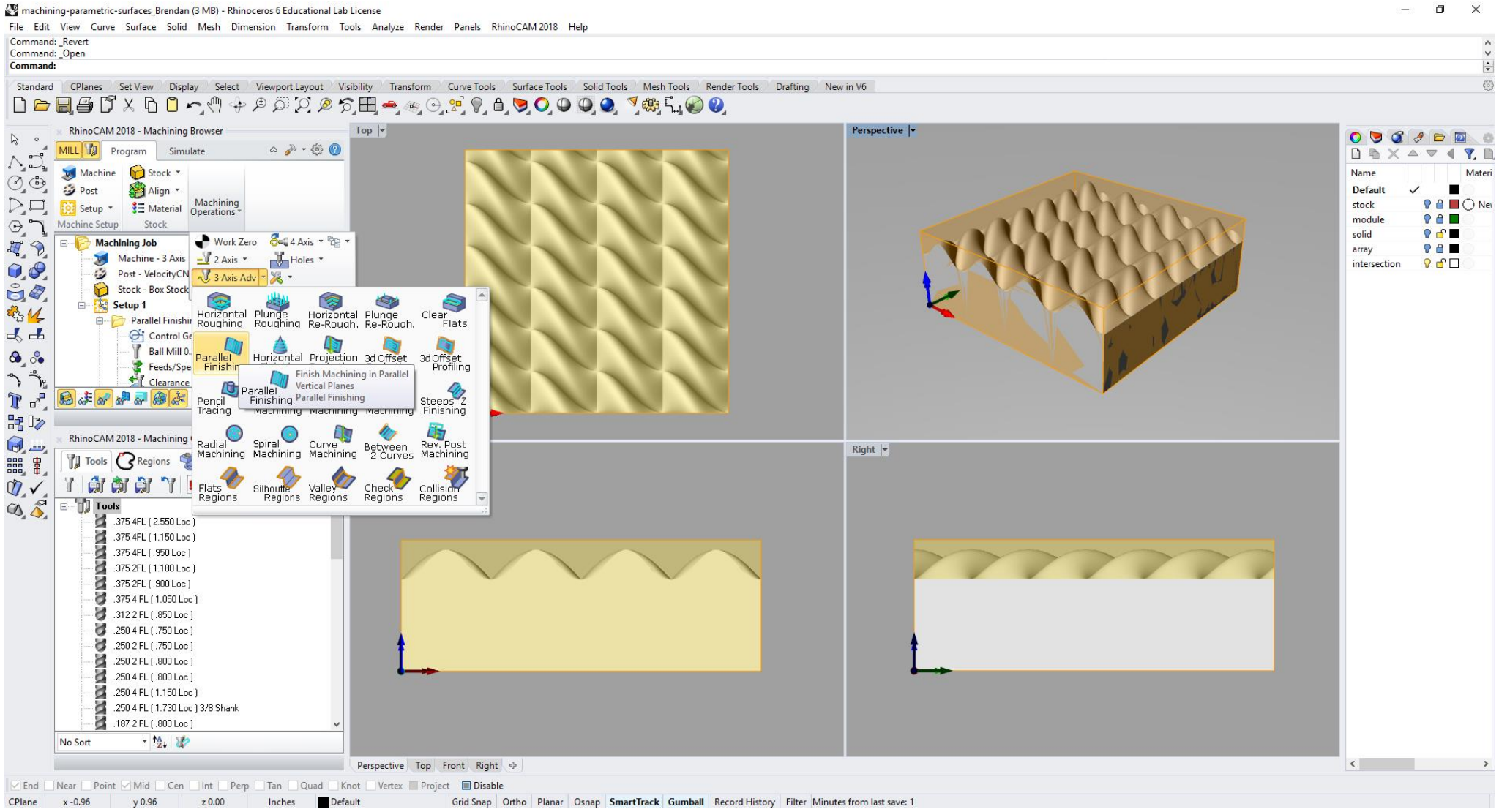
hit generate toolpath

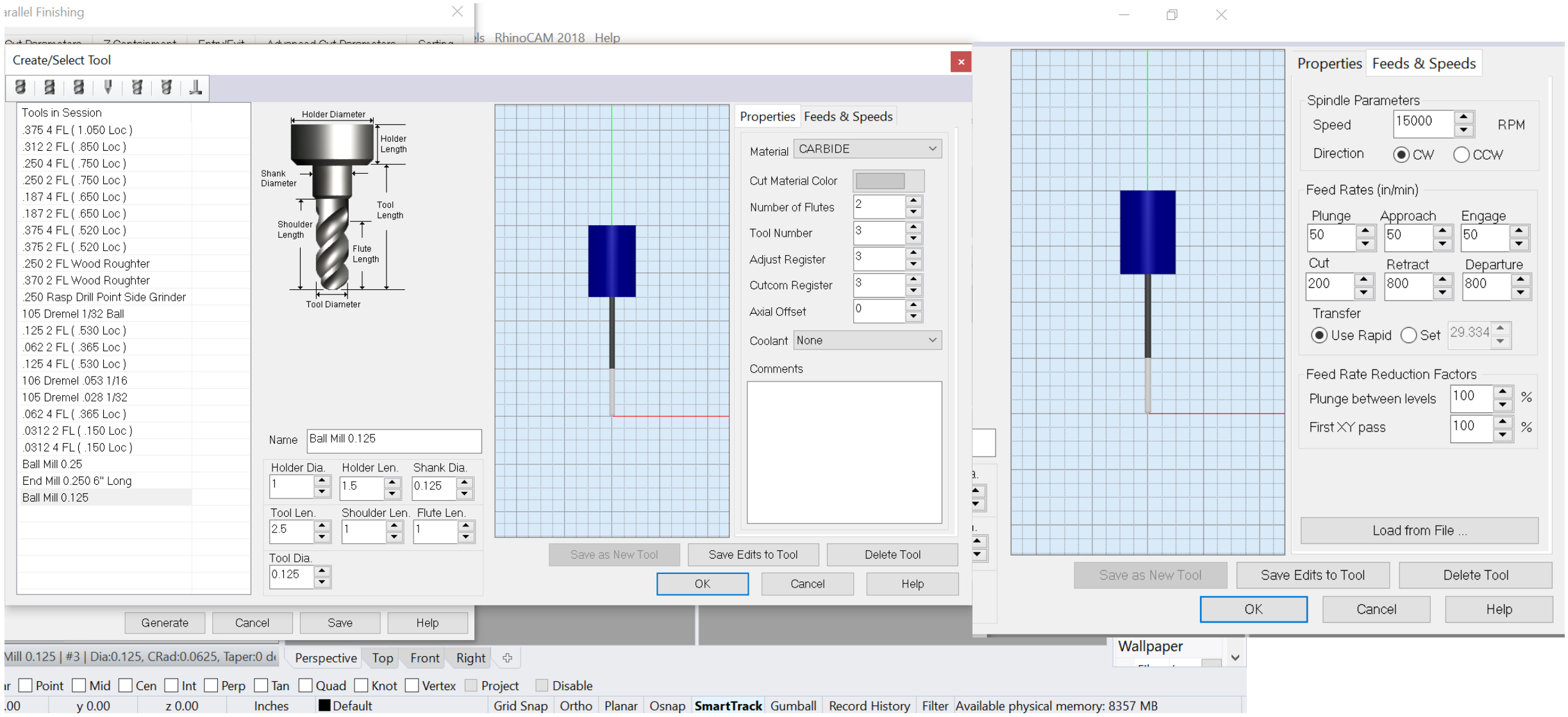
Go to simulate tab, and hit “play”



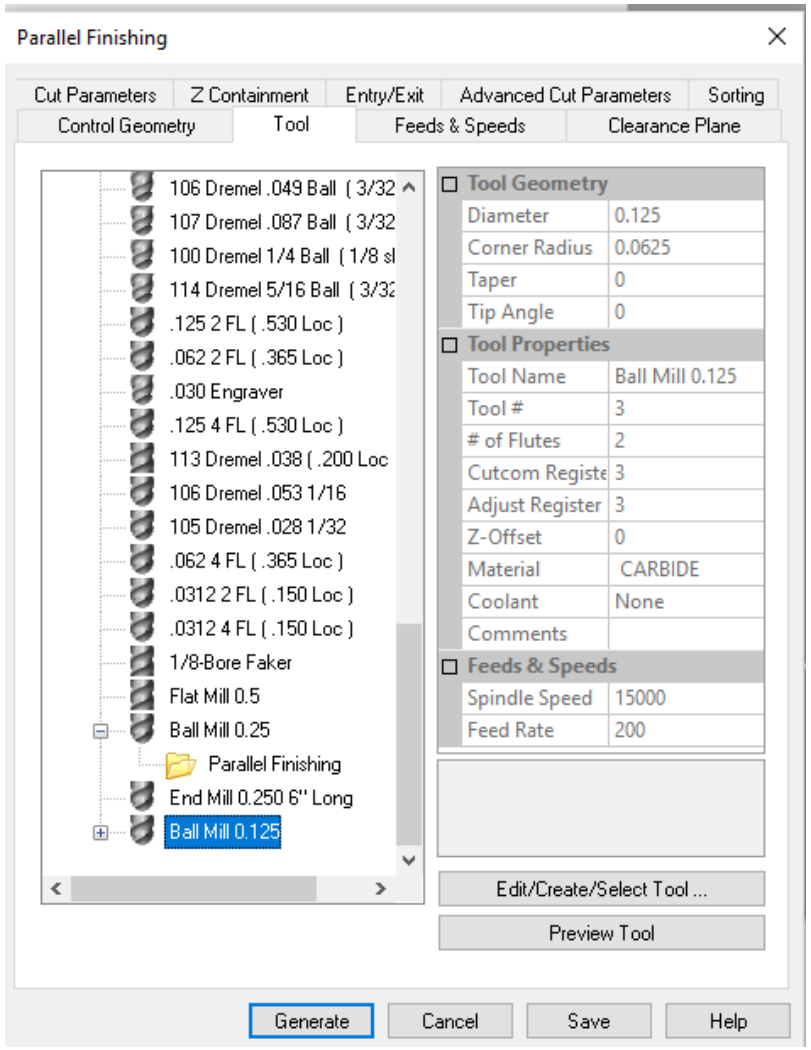
Now setting the second toolpath:

Machine operations-> 3-axis Adv -> Parallel finishing

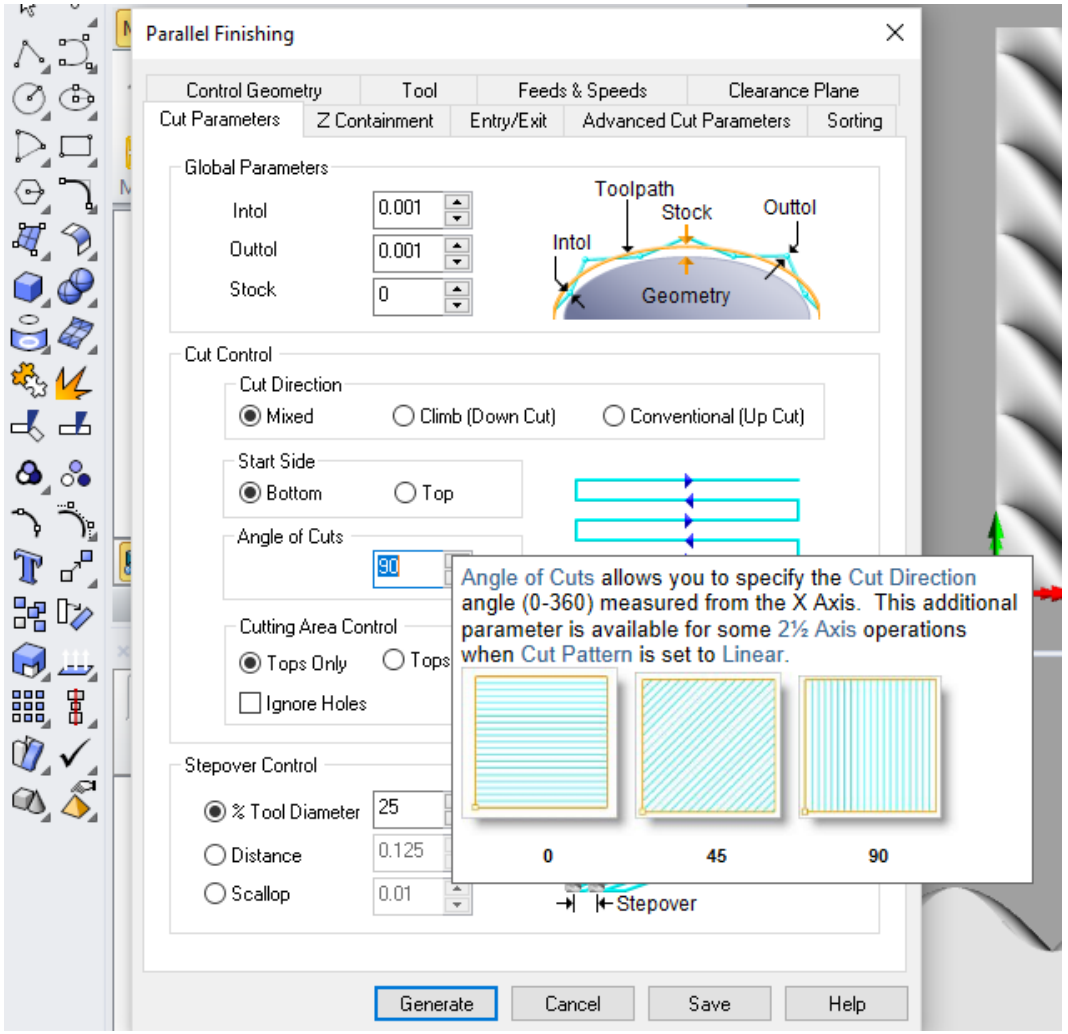



Create/Select Tool
Tools in Session
.375 4 FL (1.050 Loc)
.312 2 FL (.850 Loc)
.250 4 FL (.750 Loc)
.250 2 FL (.750 Loc)
.187 4 FL (.650 Loc)
.187 2 FL (.650 Loc)
.375 4 FL (.520 Loc)
.375 2 FL (.520 Loc)
.250 2 FL Wood Roughter
.370 2 FL Wood Roughter
.250 Rasp Drill Point Side Grinder
105 Dremel 1/32 Ball
.125 2 FL (.530 Loc)
.062 2 FL (.365 Loc)
.125 4 FL (.530 Loc)
106 Dremel .053 1/16
105 Dremel .028 1/32
.062 4 FL (.365 Loc)
.0312 2 FL (.150 Loc)
.0312 4 FL (.150 Loc)
Ball Mill 0.25
End Mill 0.250 6" Long
Ball Mill 0.125
Holder Diameter
Holder Length
Shank Diameter
Tool Length
Shoulder Length
Flute Length
Tool Diameter
Name
Ball Mill 0.125
Holder Dia.
Holder Len.
Shank Dia.
Tool Len.
Shoulder Len.
Flute Len.
Tool Dia.
Properties
Feeds & Speeds
Material
CARBIDE
Cut Material Color
Number of Flutes
Tool Number
Adjust Register
Cutcom Register
Axial Offset
Coolant
None
Comments
Save as New Tool
Save Edits to Tool
Delete Tool
OK
Cancel
Help
Generate
Cancel
Save
Help
Properties
Feeds & Speeds
Spindle Parameters
Speed
15000
RPM
Direction
CW
CCW
Feed Rates (in/min)
Plunge
Approach
Engage
Cut
Retract
Departure
Transfer
Use Rapid
Set
Feed Rate Reduction Factors
Plunge between levels
First XY pass
Load from File ...
Save as New Tool
Save Edits to Tool
Delete Tool
OK
Cancel
Help
Perspective
Top
Front
Right
Wallpaper
Point
Mid
Cen
Int
Perp
Tan
Quad
Knot
Vertex
Project
Disable
y 0.00
z 0.00
Inches
Default
Grid Snap
Ortho
Planar
Osnap
SmartTrack
Gumball
Record History
Filter
Available physical memory: 8357 MB

In the TOOL parameters, scroll down the screen and select Ball mill 0.125” (1/8”)

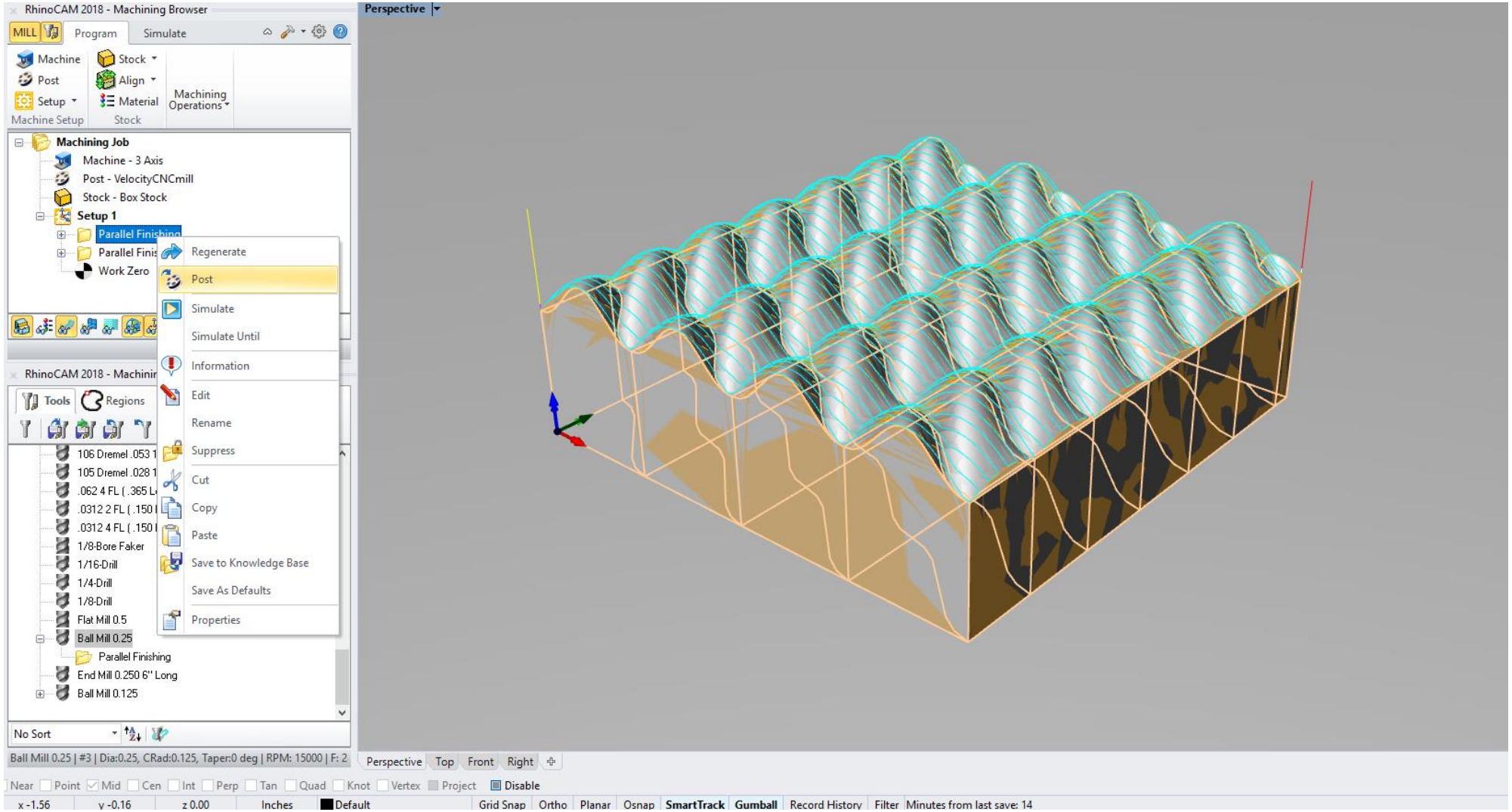


Set the angle of cut to 90 degrees. This second parallel finish is perpendicular to the first toolpath



hit generate toolpath

Step 05: generate G-code



- Once satisfied with the toolpaths, right click on each toolpath and select “post.” This will create a .nc file
- Note: if it did not generate the file, go back to your post and make sure that “velocityCNC” is selected and .nc file extension is set.

File is ready

- Put your .nc files on a USB. We will load them to the computer that is controlling the CNC machine

Physical space:
Running the CNC machine

CNC (Computer Numerical Control)

- **Introductory video**
- https://www.youtube.com/watch?v=QTi7dnwYTVw
- The router moves in the x, y, and z axis
- You can mount a collar and then bits to the router
- There is a spoil board on which you fix your material. You may cut through the spoil boards when you cut through the material.
- **Changing bits**
- https://www.youtube.com/watch?v=hCCEvbU_BXQ