

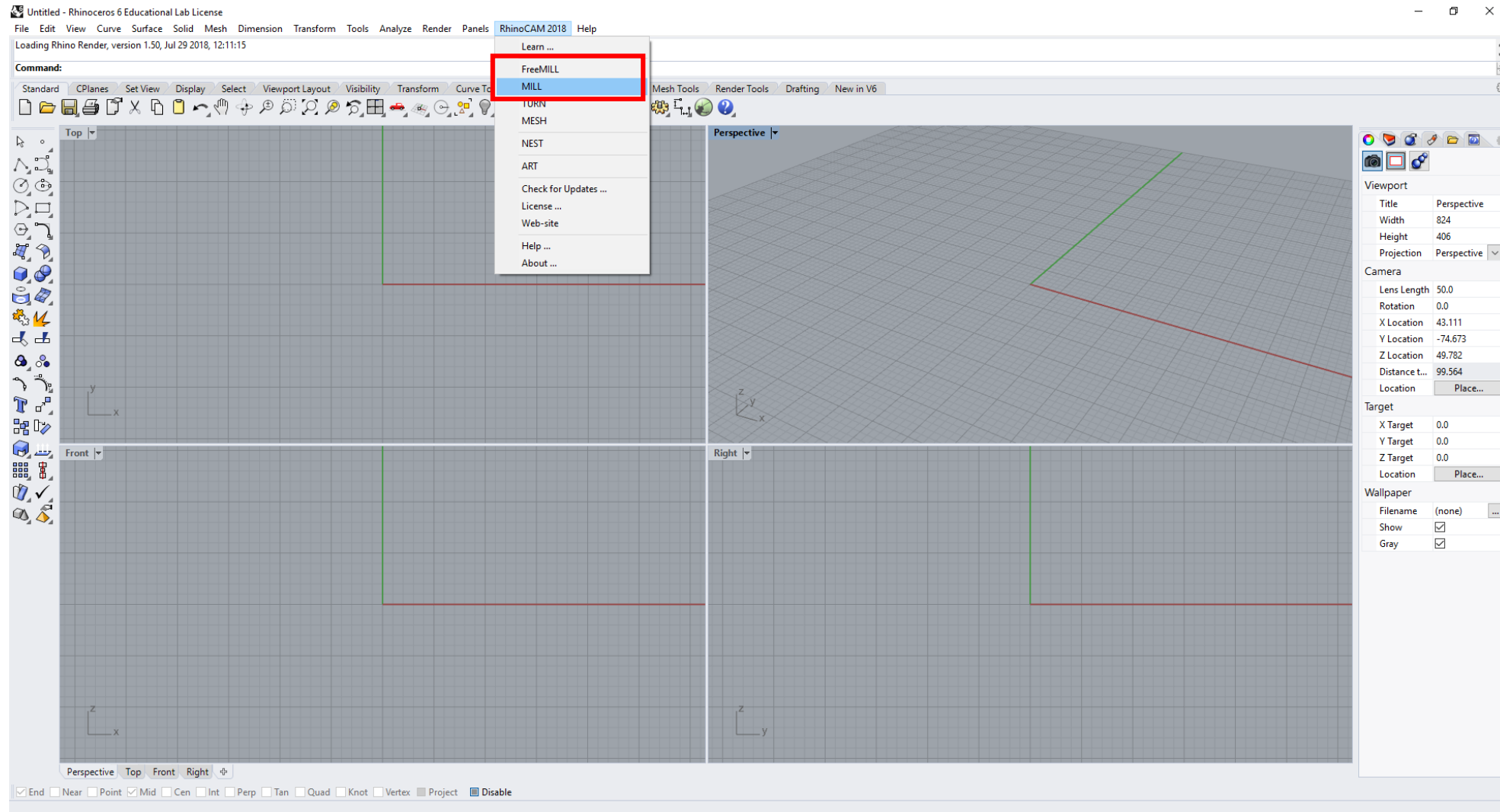
CNC (Computer Numerical Control) using RhinoCAM

Introduction

Niloufar Emami

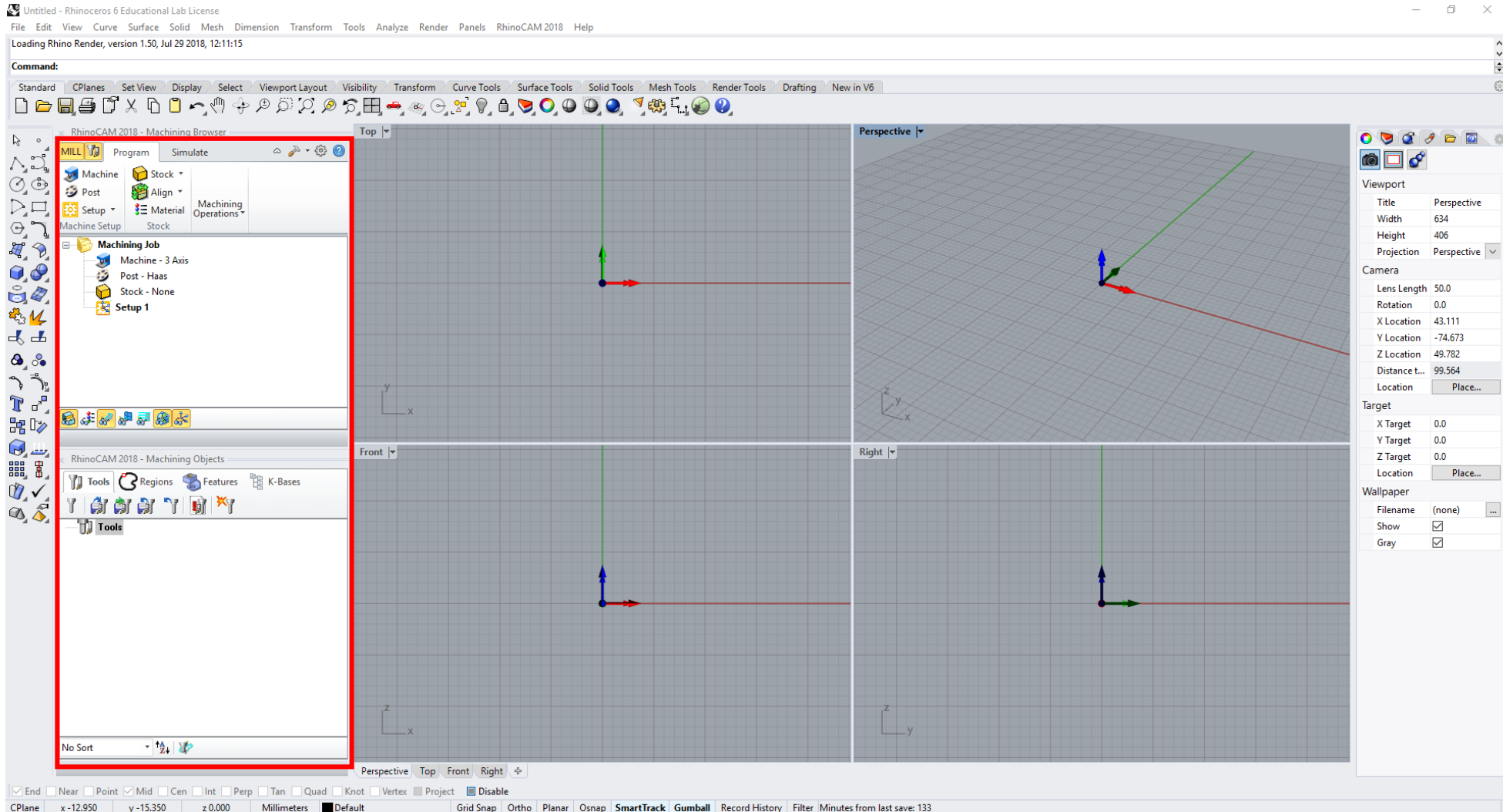
Digital space:
RhinoCAM

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 operations 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

Untitled - Rhinoceros 6 Educational Lab License

File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Panels RhinoCAM 2018 Help

Loading Rhino Render, version 1.50, Jul 29 2018, 12:11:15

Command:

Standard CPlanes Set View Display Select Viewport Layout Visibility Transform Curve Tools Surface Tools Solid Tools Mesh Tools Render Tools Drafting New in V6

RhinoCAM 2018 - Machining Browser

MILL Program Simulate

Machine

Stock

Post

Setup

Machine Setup

Stock

Machining Job

- Machine - 3 Axis
- Post - Haas
- Stock - None
- Setup 1

RhinoCAM 2018 - Machining Objects

Tools Regions Features K-Bases

Tools

No Sort

Machine Tool Setup

Machine Tool Definition

☒ Manual Definition ☐ Load From File

Machine Type

Number of Axes 3 Axis

General Parameters

Tool Change Pt: X 0 Y 0 Z 0

☒ Output all coordinates in local Setup Coordinate System

Translational Limits

Min: X -5000 Y -5000 Z -5000

Max: X 5000 Y 5000 Z 5000

4th Axis (Primary Axis) Parameters

Rotary Center: X 0 Y 0 Z 0

Rotary Axis: ☒ +X ☐ -X ☐ +Y ☐ -Y ☐ +Z ☐ -Z

Rotary Axis Angle Limit

Rotary Axis Limits (In Degrees) Min 0 Max 360

5th Axis (Secondary Axis) Parameters

Rotary Center: X 0 Y 0 Z 0

Rotary Axis: ☐ +X ☐ -X ☒ +Y ☐ -Y ☐ +Z ☐ -Z

Rotary Axis Angle Limit

Rotary Axis Limits (In Degrees) Min 0 Max 360

Gage Length (For Machines with a Rotary Head): 10

OK Cancel Help

Perspective

Top

Right

Viewports

Title	Perspective
Width	634
Height	406
Projection	Perspective

Camera

Lens Length 50.0

Rotation 0.0

X Location 43.111

Y Location -74.673

Z Location 49.782

Distance t... 99.564

Location Place...

Target

X Target 0.0

Y Target 0.0

Z Target 0.0

Location Place...

Wallpaper

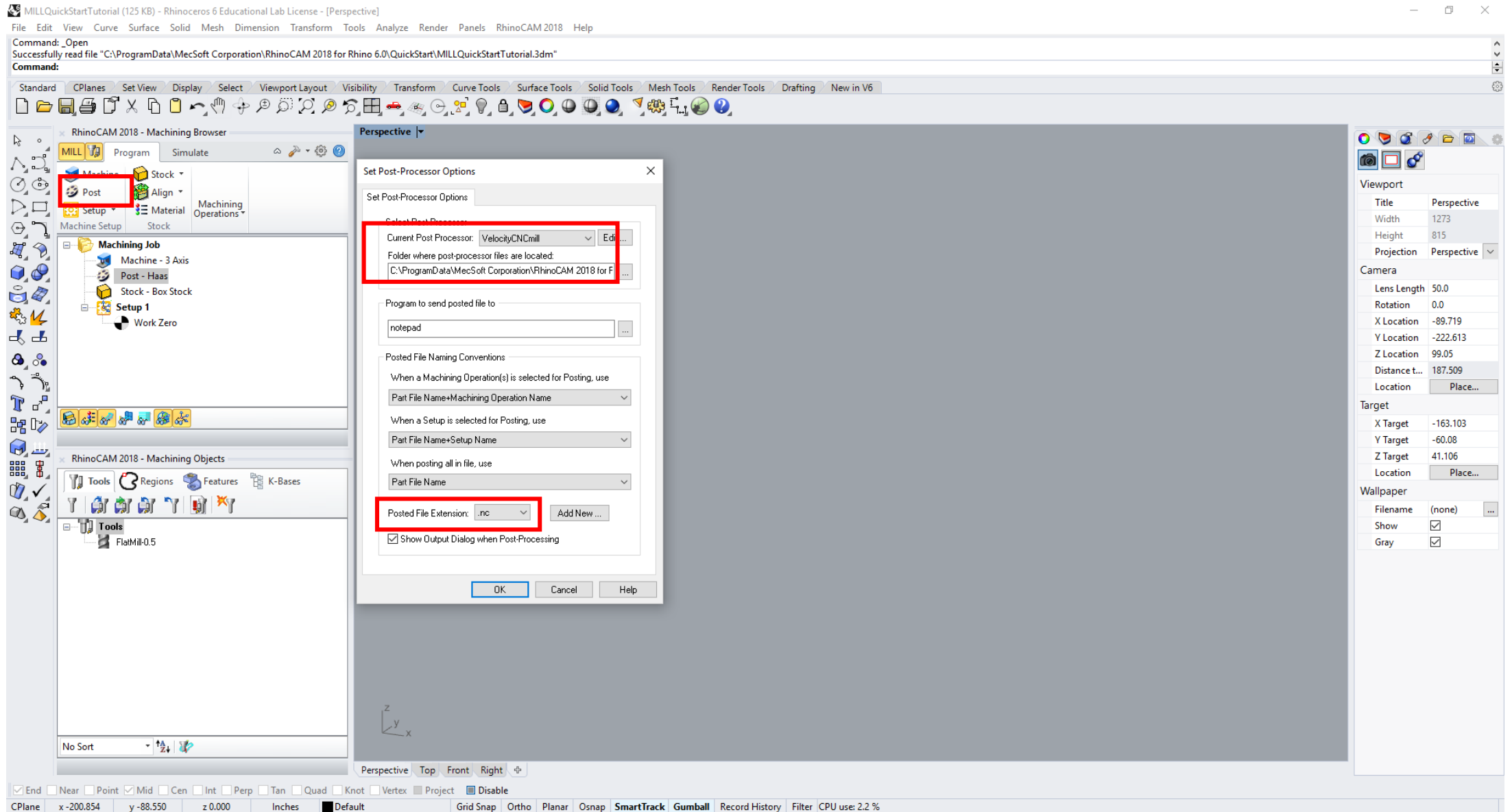
Filename	(none)
Show	<input checked="" type="checkbox"/>
Gray	<input checked="" type="checkbox"/>

End Near Point Mid Cen Int Perp Tan Quad Knot Vertex Project Disable

CPlane x -5.904 y -14.512 z 0.000 Millimeters Default

Grid Snap Ortho Planar Osnap SmartTrack Gumball Record History Filter Available physical memory: 27710 MB

Post: Velocity CNC Mill

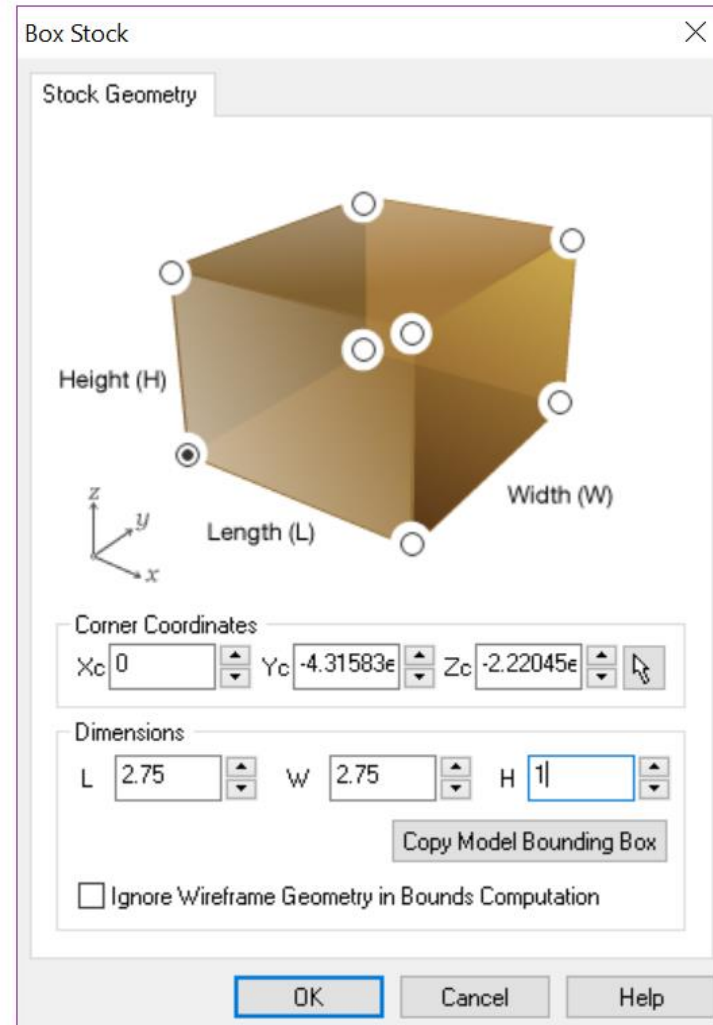


Step 02: define the machine setup (including stock geometry, material, and work zero)

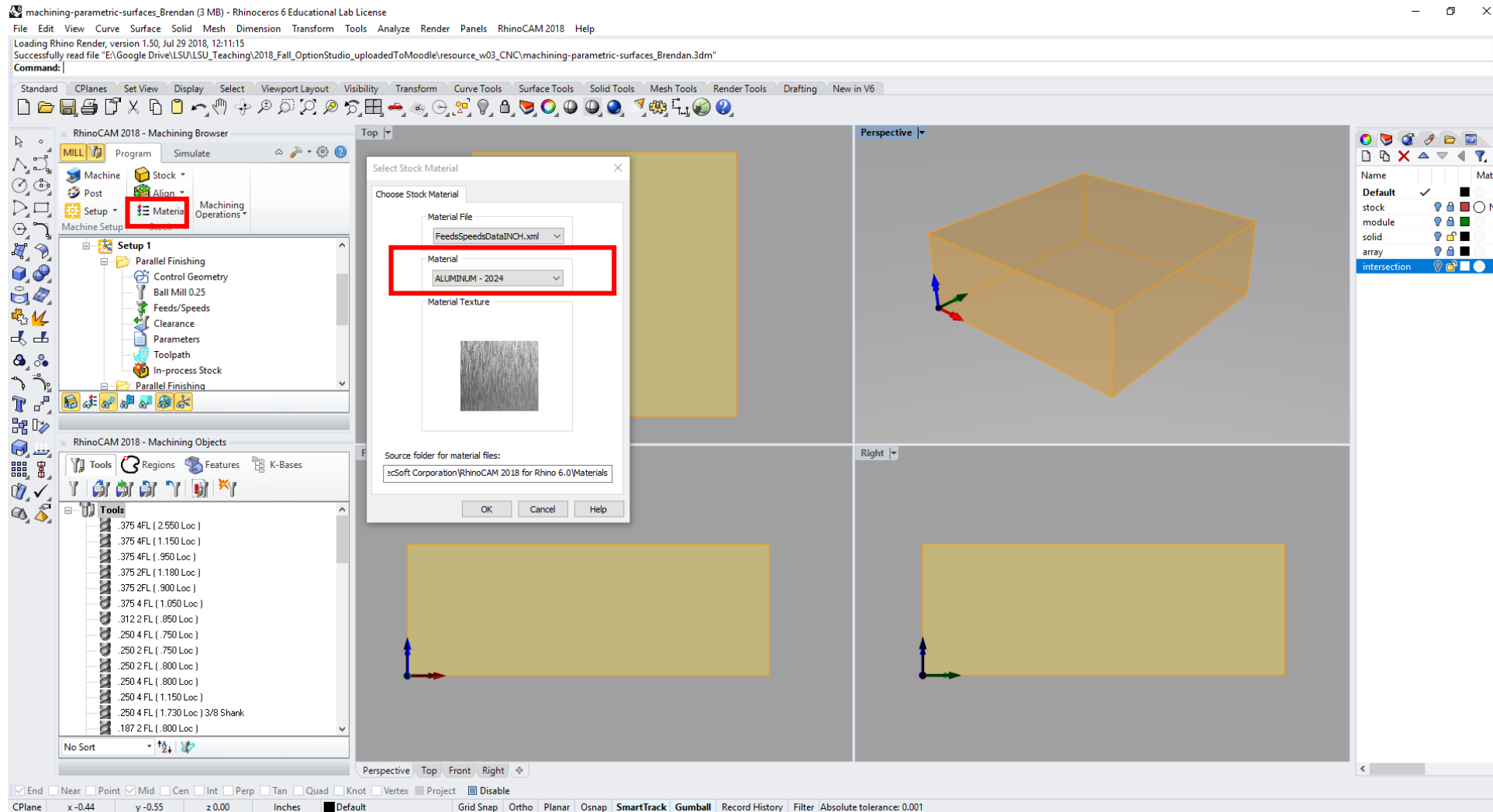
- You will be cutting your part from a stock geometry. We need to define the dimensions of the stock geometry.
- You can route different materials, and the feed rate and speed of the machine will be set accordingly. Therefore, you need to set up the material.
- The Work Zero is a specific point you will 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: $L=2.75$, $W=2.75$, $H=1$

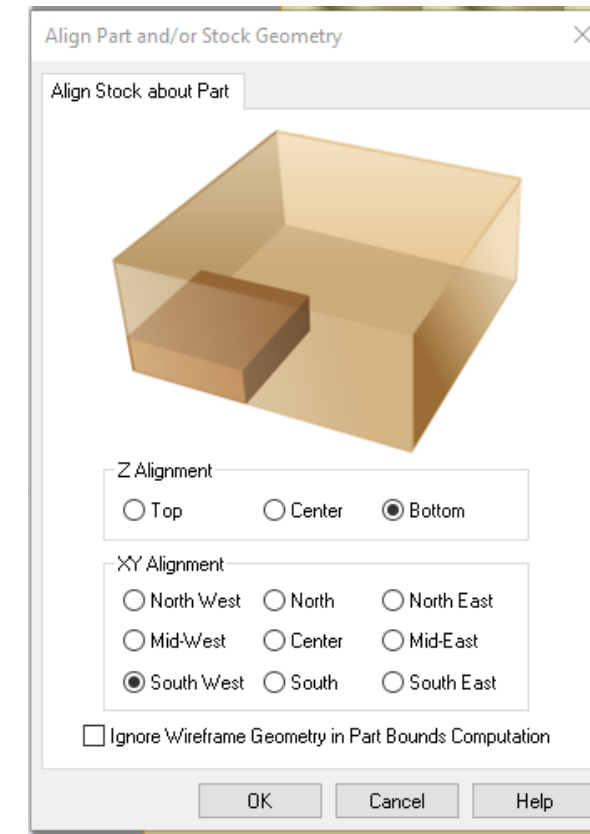
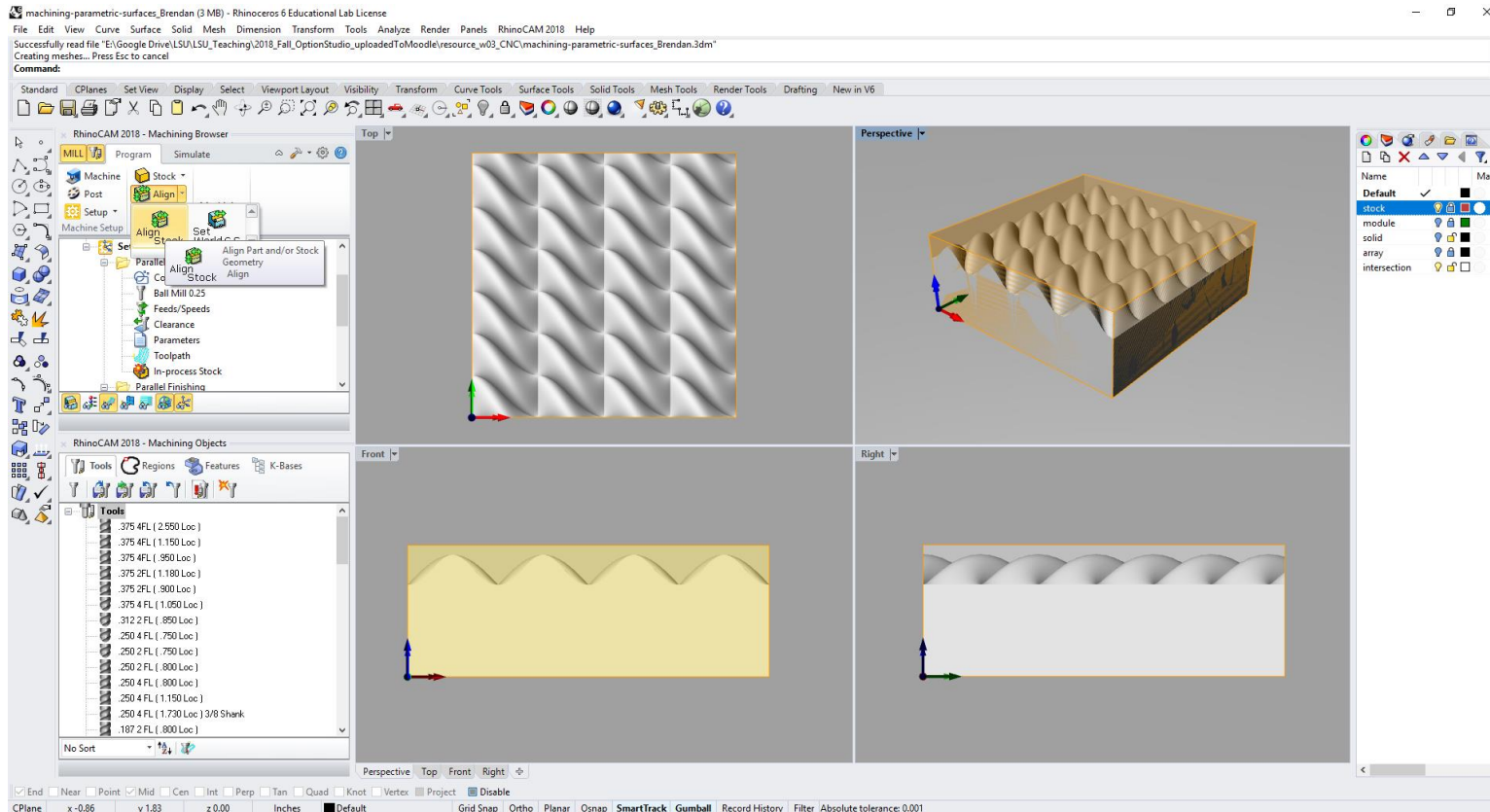
Note that the stock dimensions you enter are measured from the corner of the bounding box selected in this dialog. You can click on different corners.



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



Align part and stock material



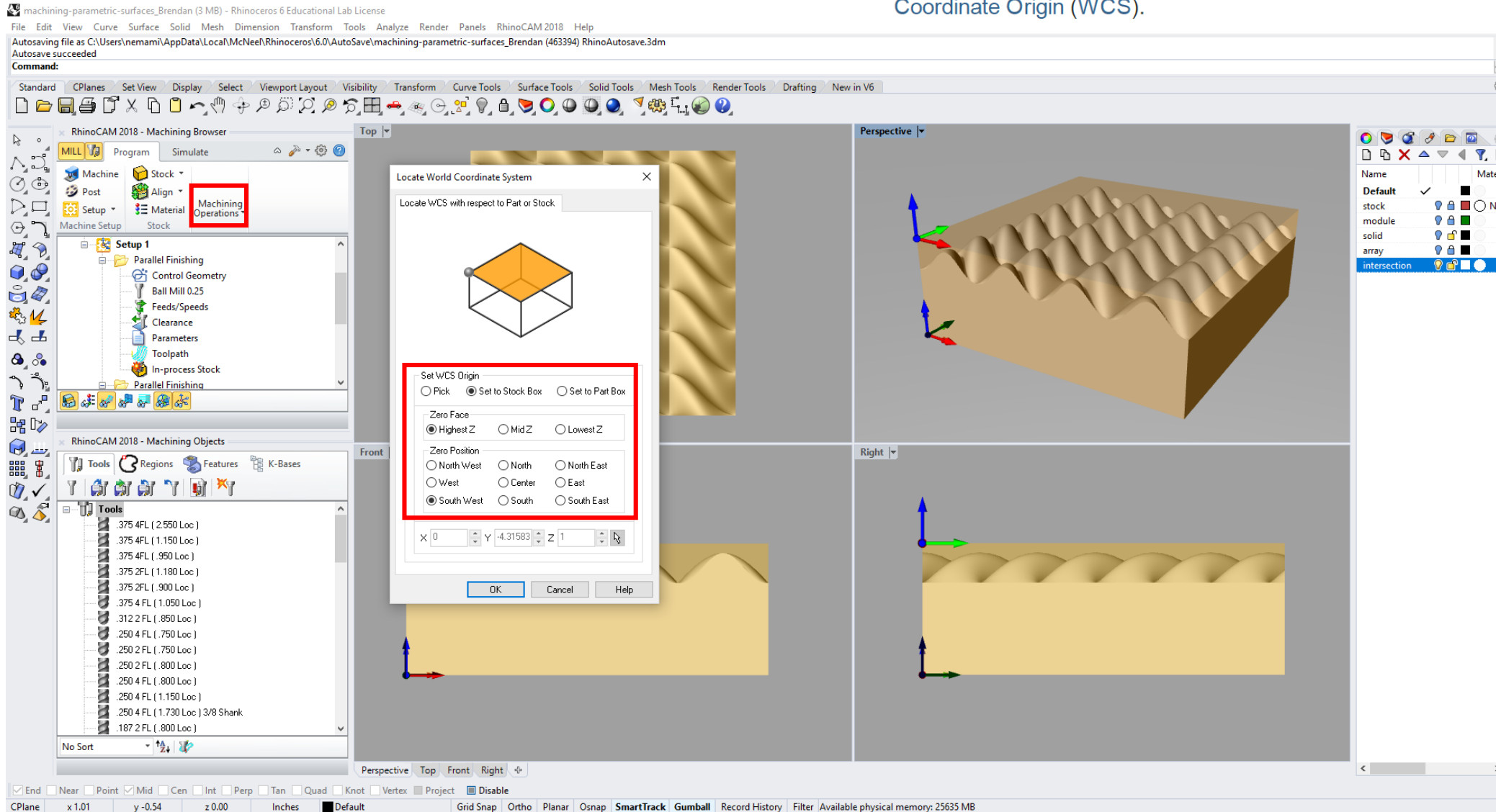
- Let's say you are cutting a small part out of a bigger stock material. You need to align it to the stock so that you know which portion of the stock will be milled.
- In this example, the stock and part size match (2.7" by 2.7" by 1")

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

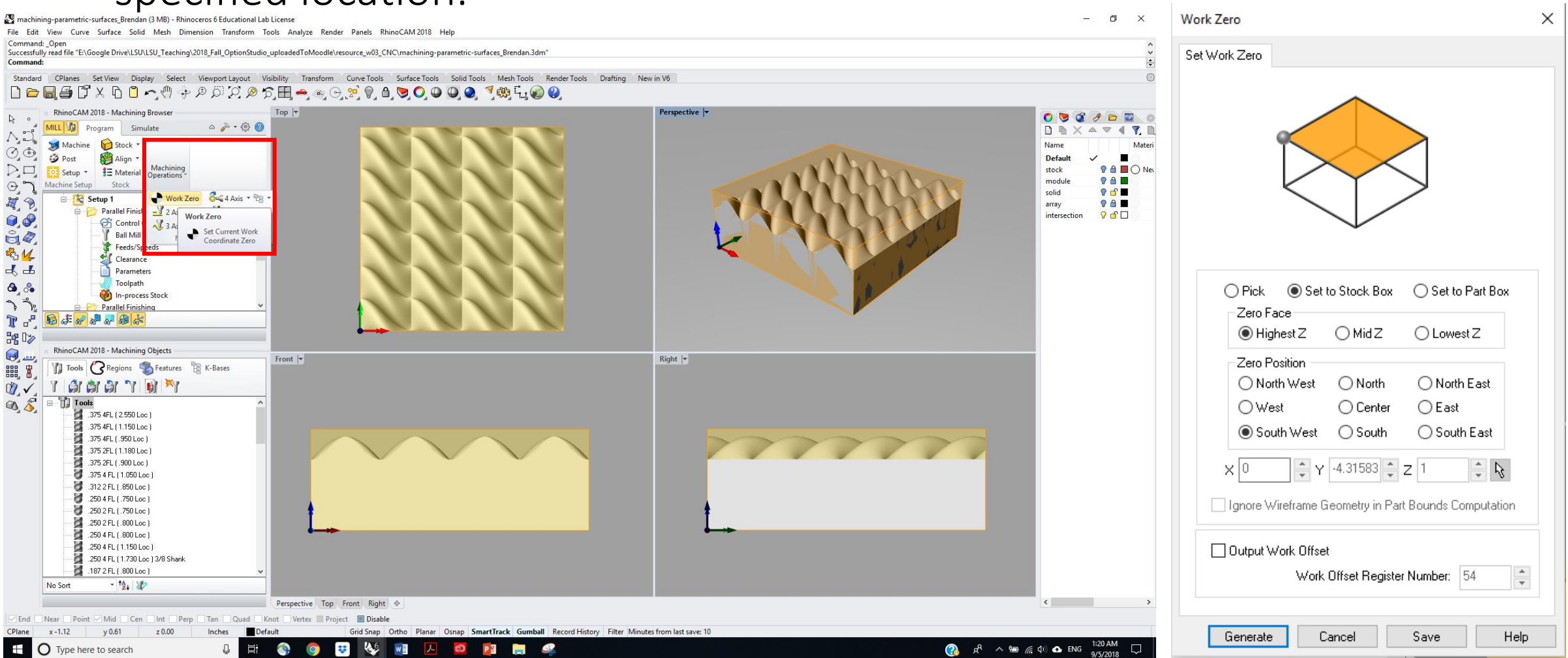
Click on Align and select “set world CS” then pick “set to stock material” and “SouthWest” on top or “highest z”

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.

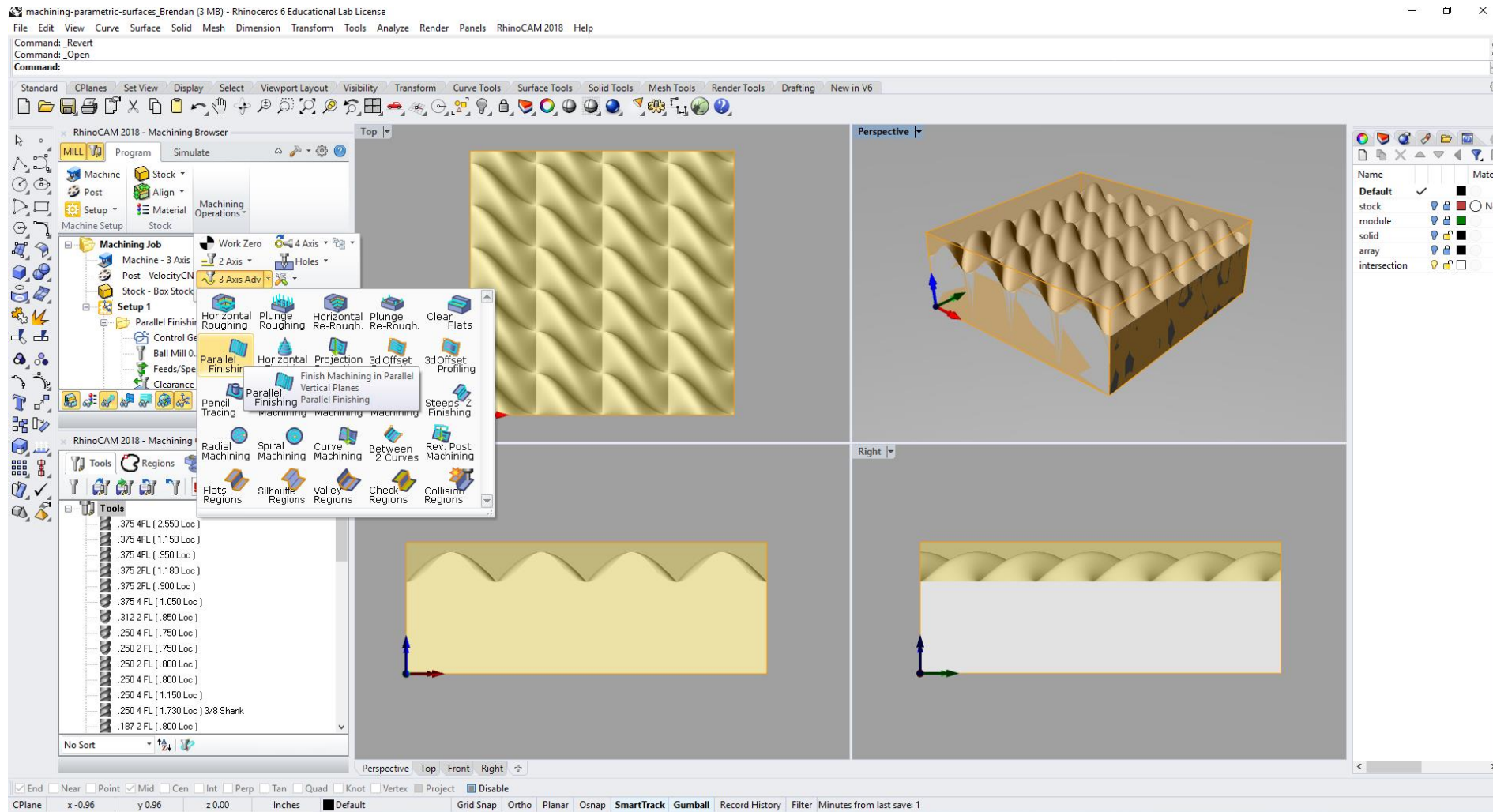


This sets the machine home to the top of the stock material and the southwest corner of the stock geometry.

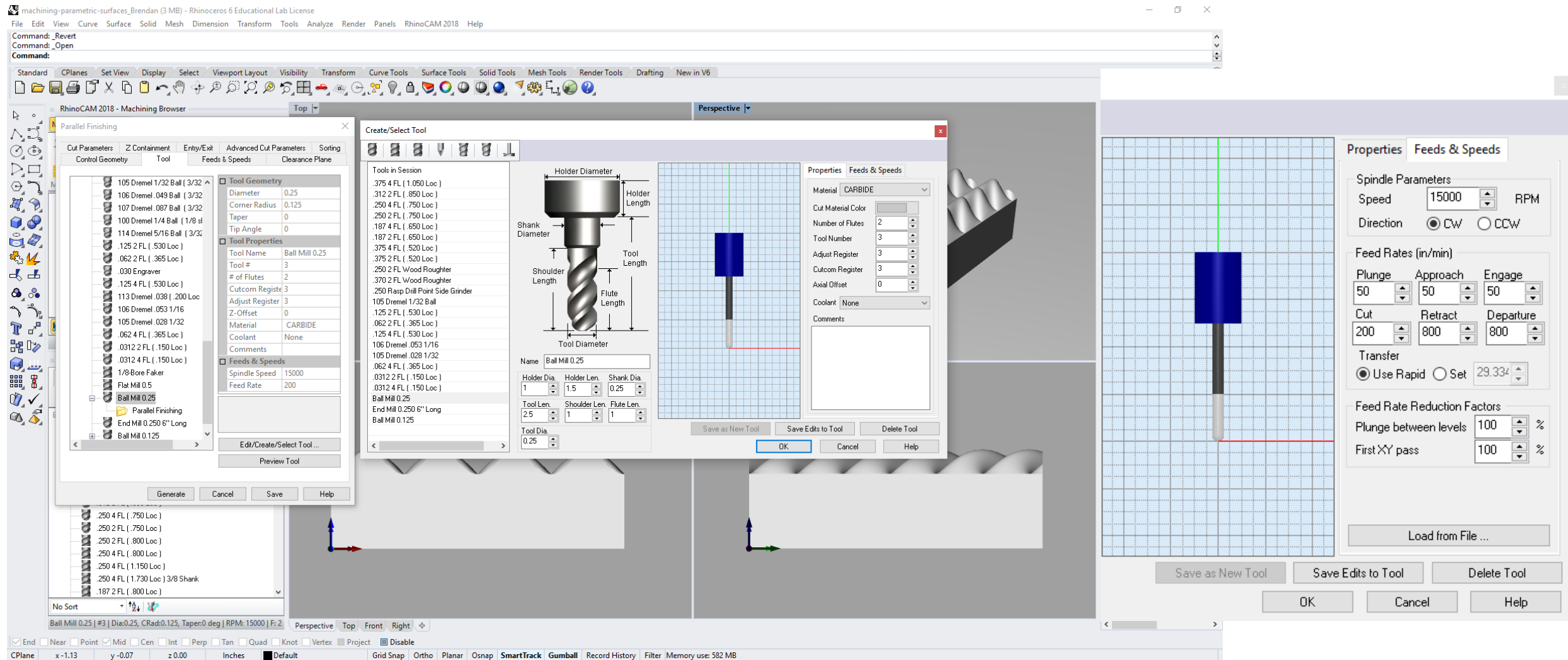
Step 04: set the toolpath

- 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.

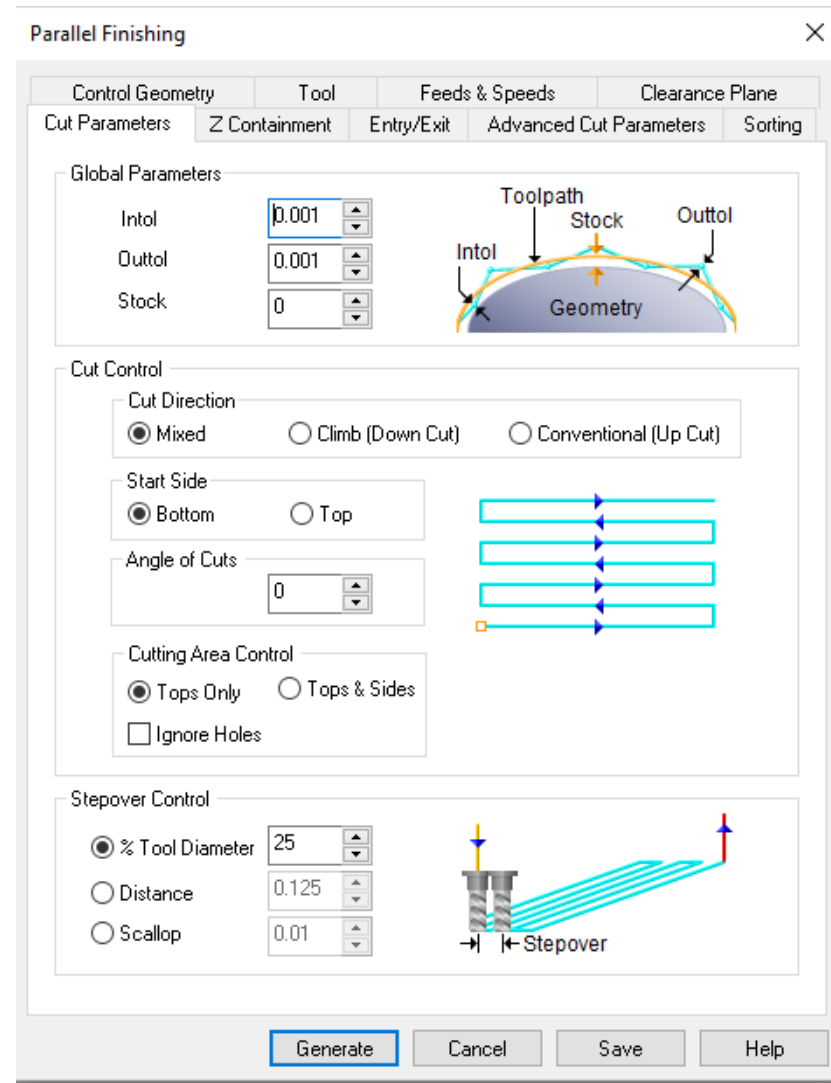
Setting the first toolpath: Machine operations-> 3-axis Adv -> Parallel finishing



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

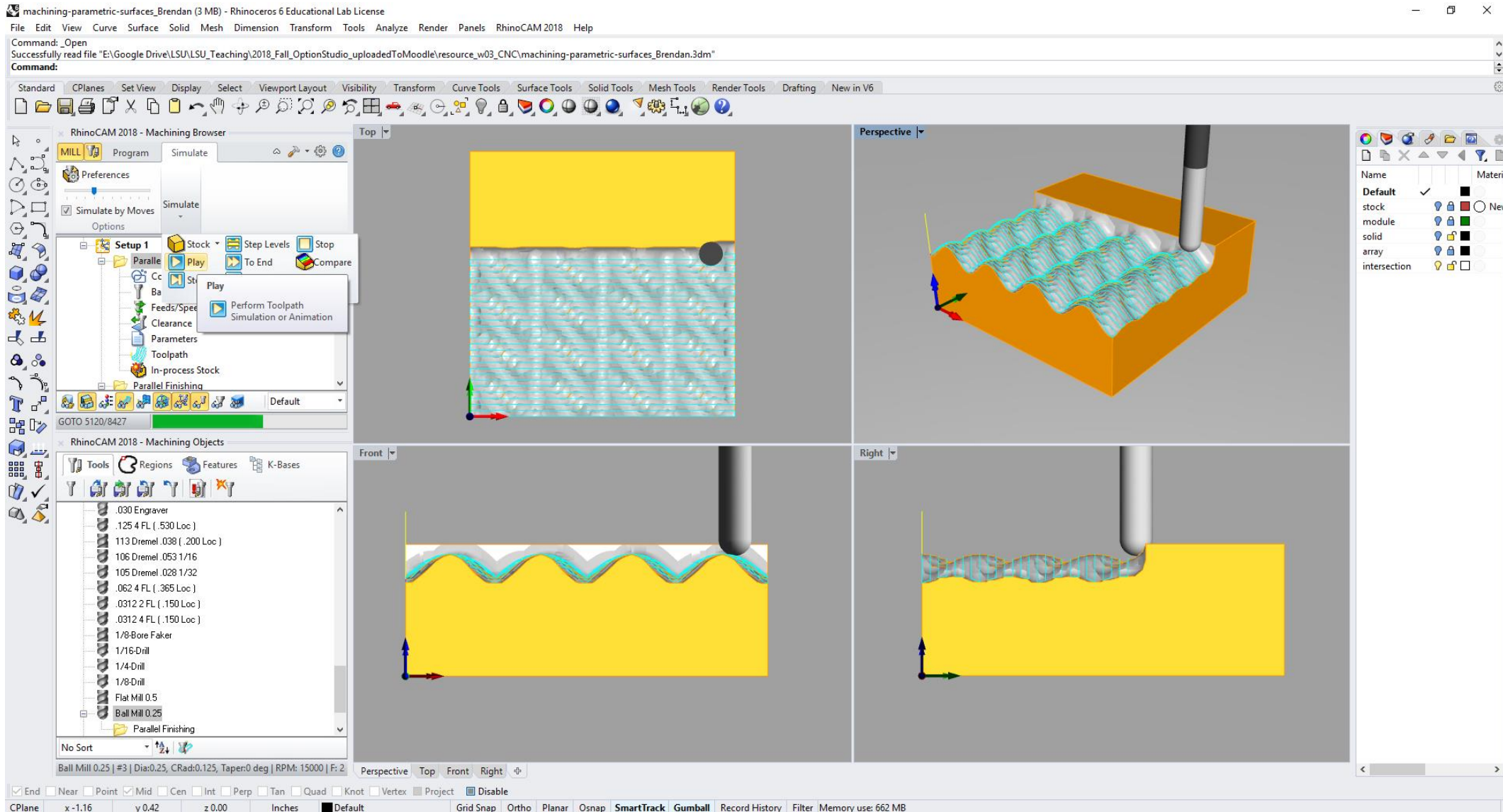


In the CUT parameters, set stepover control to 25% of tool diameter

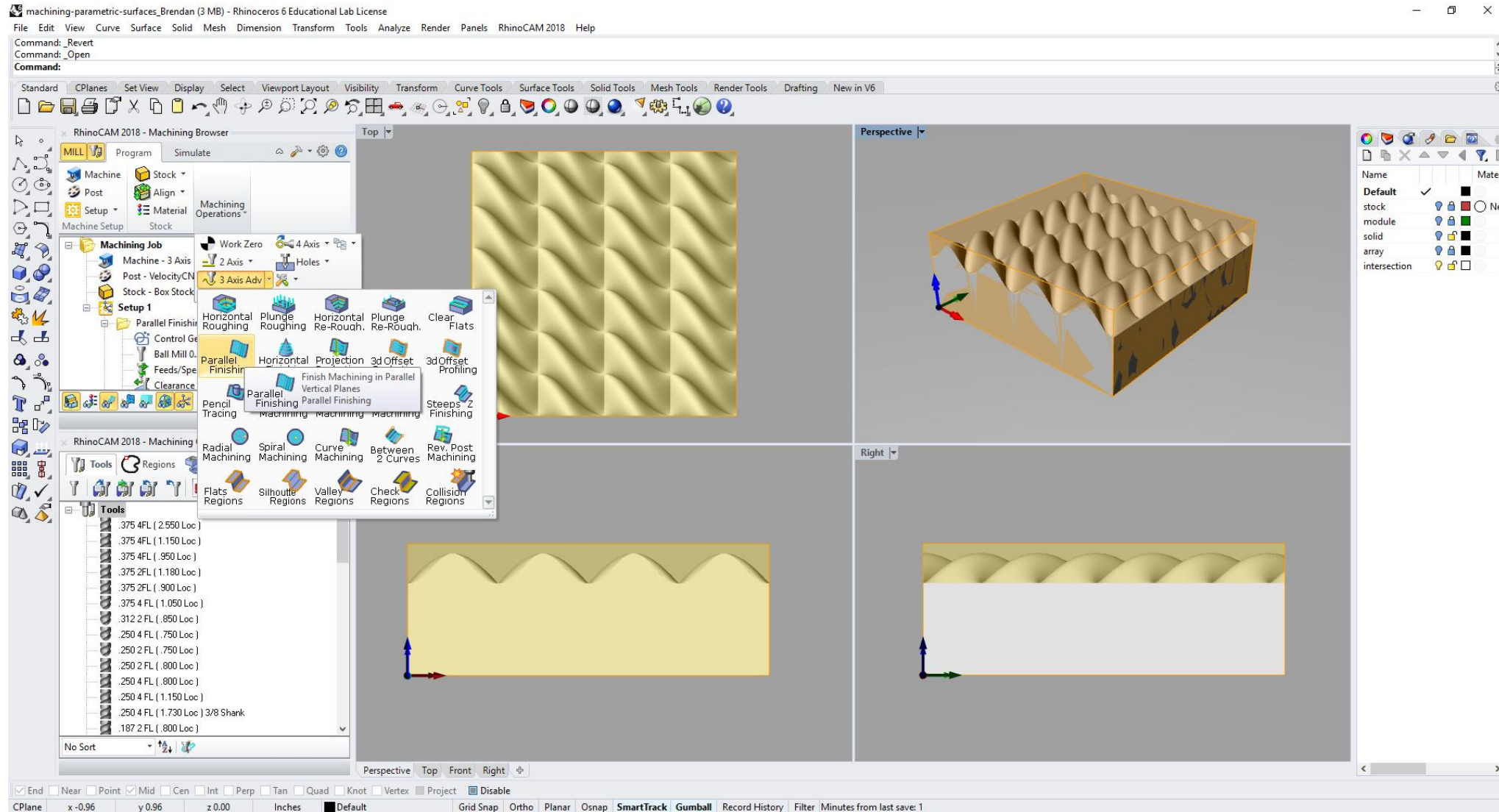


hit generate toolpath

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 (.1050 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 Rougher

.370 2 FL Wood Rougher

.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

Shoulder Length

Tool Length

Flute Length

Tool Diameter

Name

Ball Mill 0.125

Holder Dia.

Holder Len.

Shank Dia.

Tool Len.

Shoulder Len.

Flute Len.

Tool Dia.

1

1.5

0.125

2.5

1

1

0.125

Properties

Feeds & Speeds

Material

CARBIDE

Cut Material Color

Number of Flutes

2

Tool Number

3

Adjust Register

3

Cutcom Register

3

Axial Offset

0

Coolant

None

Comments

Save as New Tool

Save Edits to Tool

Delete Tool

OK

Cancel

Help

Properties

Feeds & Speeds

Spindle Parameters

Speed

15000

RPM

Direction

☒ CW

☐ CCW

Feed Rates (in/min)

Plunge

50

Approach

50

Engage

50

Cut

200

Retract

800

Departure

800

Transfer

☒ Use Rapid

☐ Set

29.334

Feed Rate Reduction Factors

Plunge between levels

100

%

First XY pass

100

%

Load from File ...

Save as New Tool

Save Edits to Tool

Delete Tool

OK

Cancel

Help

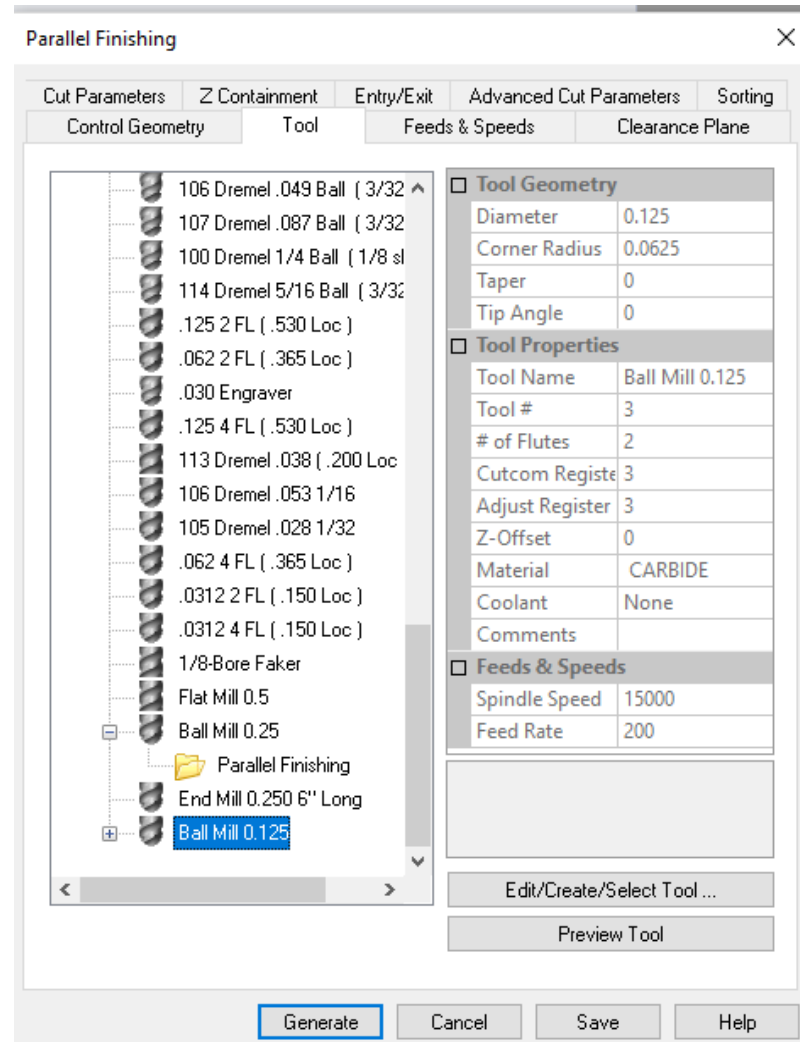
Mill 0.125 | #3 | Dia:0.125, CRad:0.0625, Taper:0 d

Perspective Top Front Right

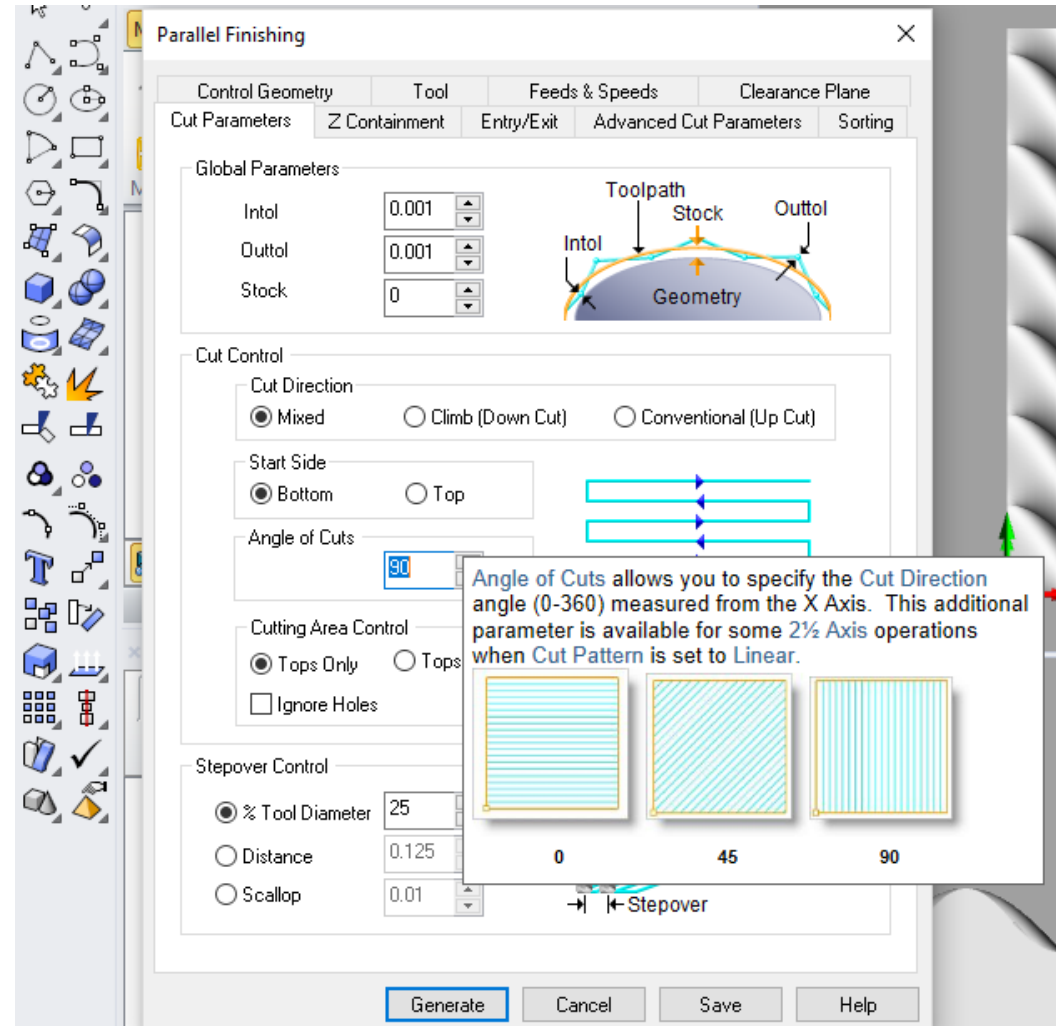
ir ☐ Point ☐ Mid ☐ Cen ☐ Int ☐ Perp ☐ Tan ☐ Quad ☐ Knot ☐ Vertex ☐ Project ☐ Disable

.00 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")

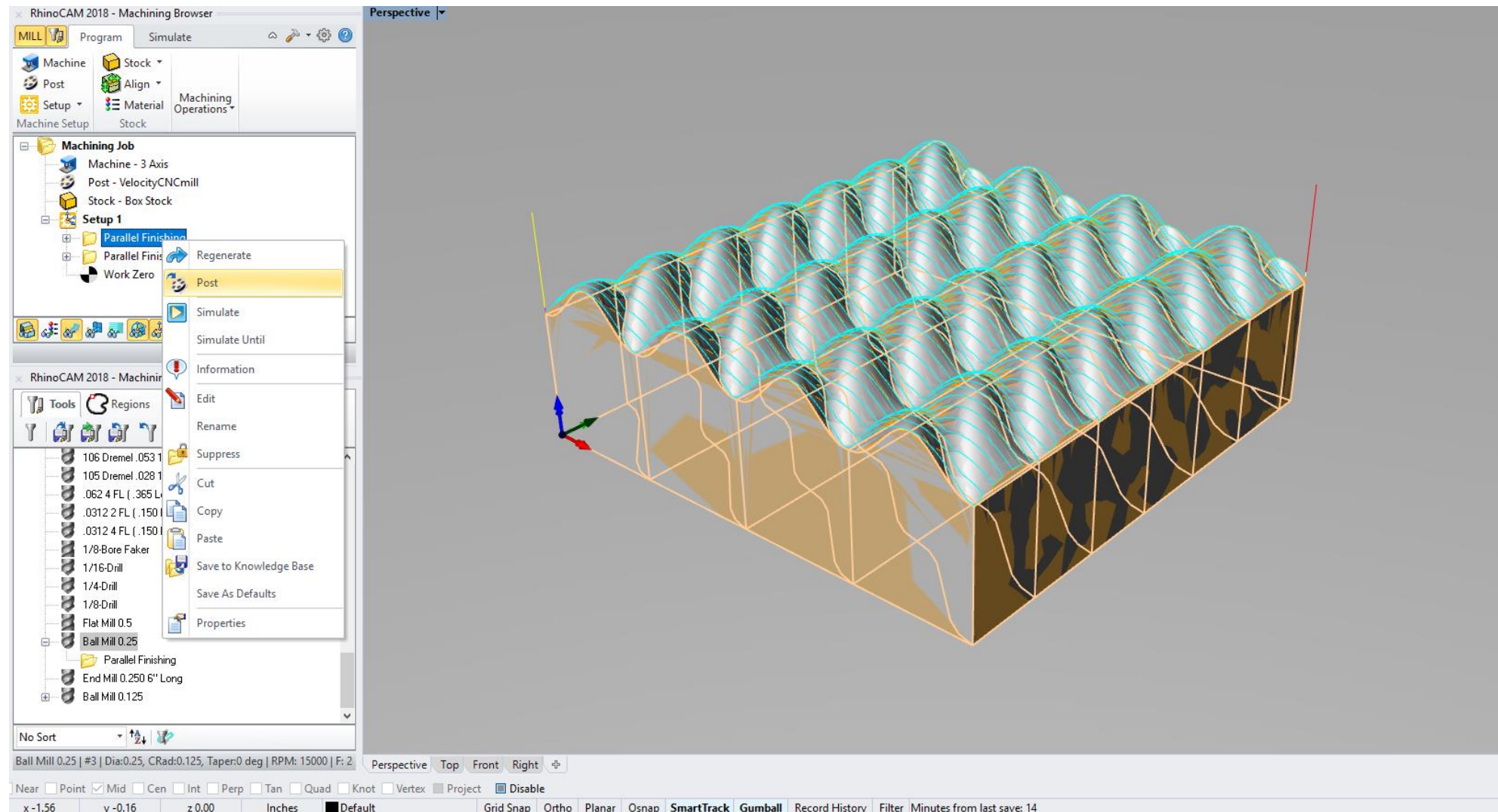


Set the angle of cut to 90 degrees. This second parallel finish is perpendicular to the first toolpath



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