CNC Mill / RhinoCAM User Reference Guide

d[Fab] Lab
2018

GETTING STARTED

Setting up a RhinoCAM file can be a daunting task but this guide is here to help stream-line to process and get you on your way to independently creating CAM files.

In this guide you will learn how to set up a CAM file as well as generate tool-paths for the most common operations; drilling, profiling, pocketing, horizontal roughing, and parallel finishing.

① The first step to making anything with RhinoCAM is to first set it up in rhino. Begin by opening Rhinoceros 5 from the desktop or from the program list in the home button.

② Once Rhino opens, be sure to change your units to inches if they have not already been set. Type "Units" into the command bar to bring up the units tab, then select Inches from the pull down menu and click okay.
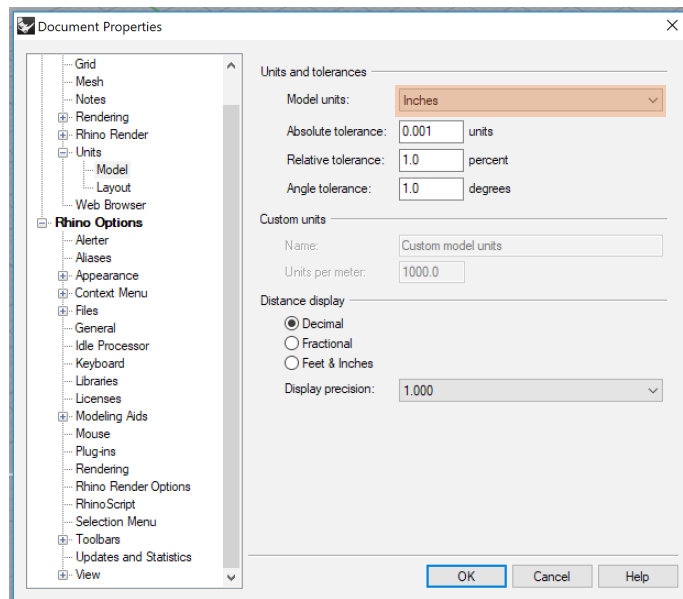
③ Often times RhinoCAM will not be open when you open Rhino, if this is the case simply type "RhinoCAM2017" into the command bar to open RhinoCAM.
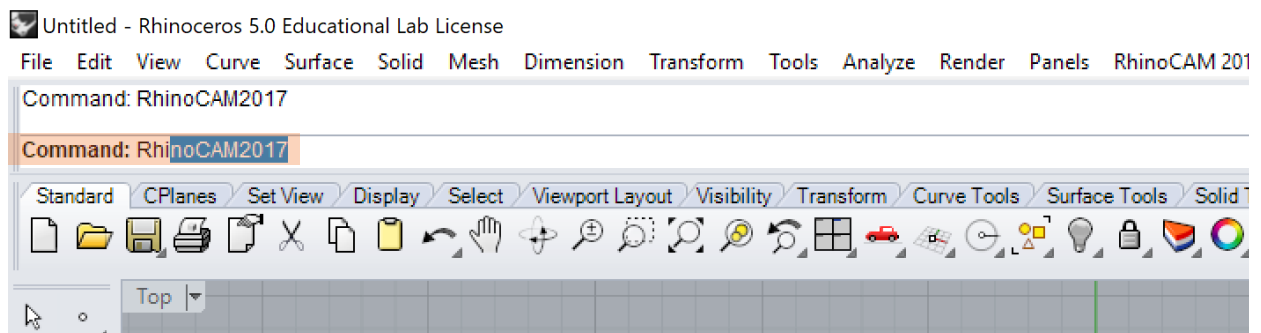
④ When it opens, make sure RhinoCAM is in "Mill" as its operation. If it is in "Turn", Click on it and switch it to Mill.
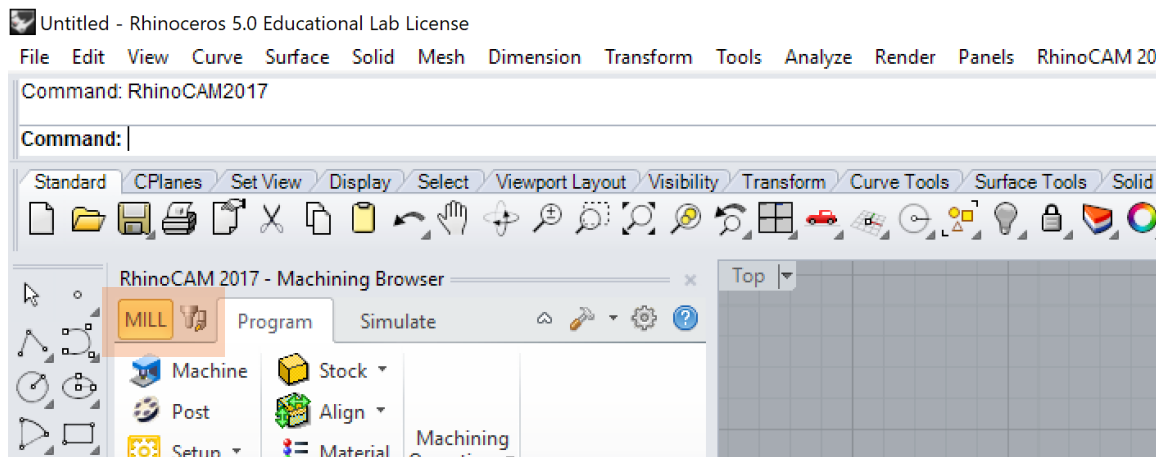
⑤ Next double click on the reel icon next to the word "Post" to bring up a dialogue box. Use the pull down tab to ensure that both the HAAS and the ShopBot are available to post to, and select the desired machine. IF one or both posts are missing alert a d[Fab] lab technician.
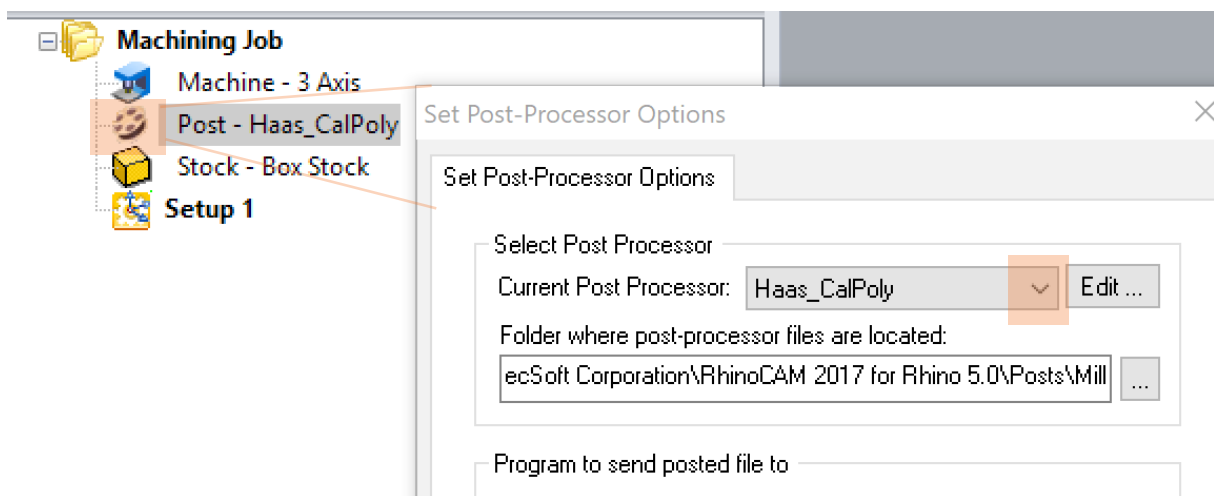
⑥ You are now ready to load in the desired geometry and begin setting up the CAM file. Use file and "import" to load in your file, alternatively you can begin with your file open and then run through steps 1-5.
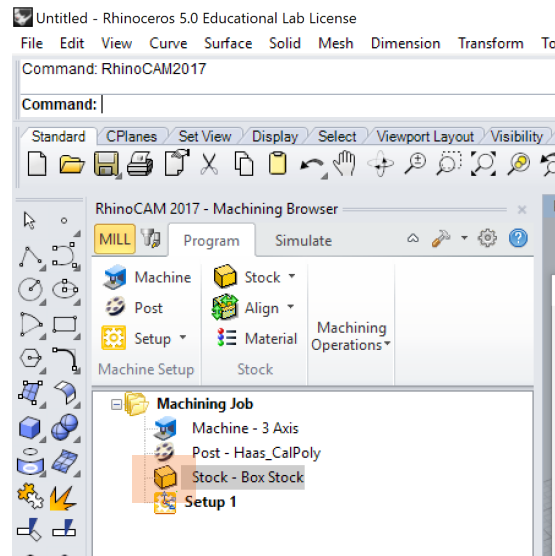
SETTING STOCK

Before you can create tool paths, it is critical that you set up an accurate and valid stock.
Maximum Stock Dimensions are: HAAS Mill: X=96" Y=48" Z=5.25"
                                           Shopbot Mill: X=48" Y=48" Z=4.5"

(1) To create a stock, double click on the orange box next to the word "stock" in the RhinoCAM tool bar.

(2) In the stock options tab you can set the location and dimensions of your stock
-The Stock must ALWAYS be located at Xc:0 Yc:0 Zc:0
-The Stock Z must ALWAYS be obtained through the use of calipers on your piece of material.
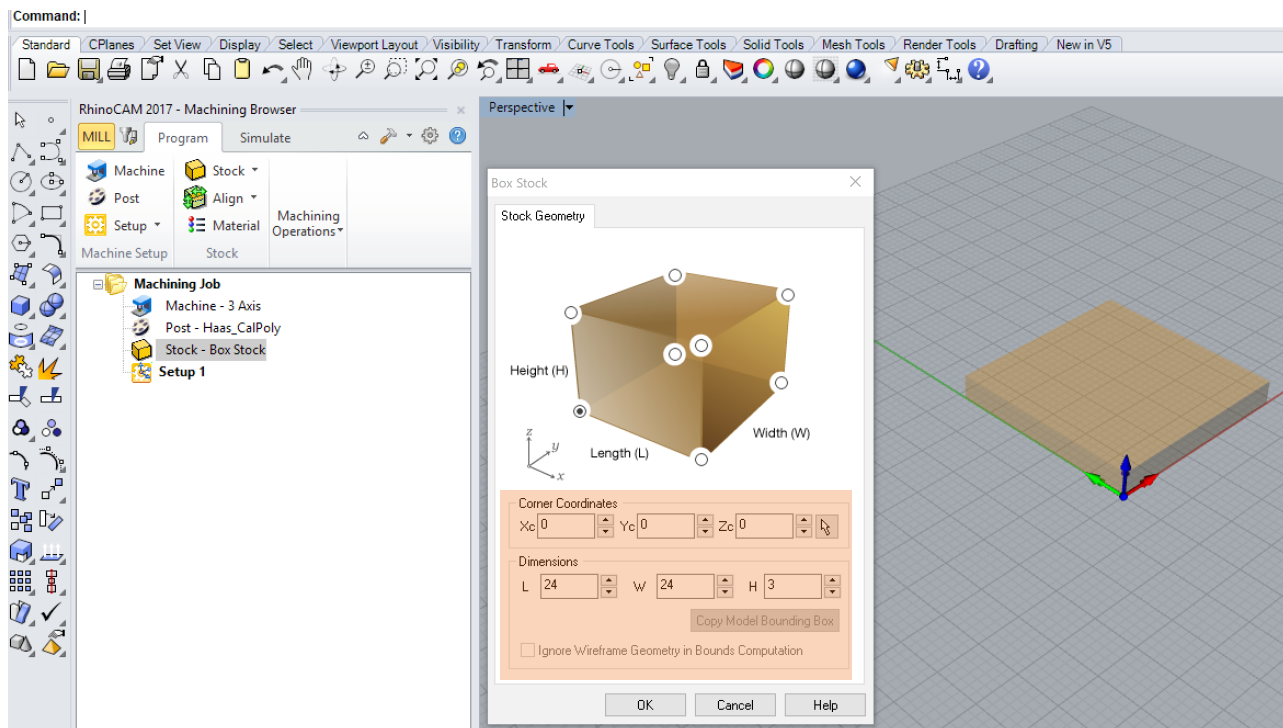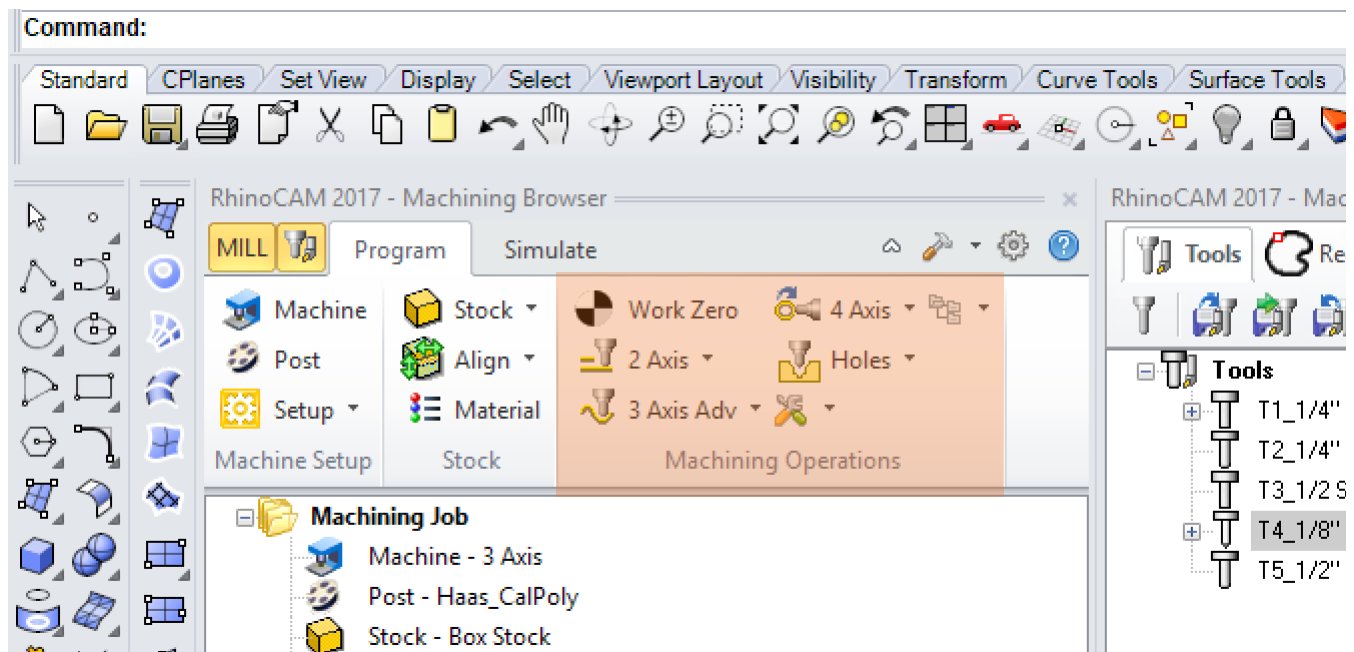
MACHINING OPERATIONS

Machining operations are the different functions the CNC Mill can perform. It is important to know the roles of each so that you can select and use the operation that best fits your need and geometry.

Operations are made up of three categories: HOLES, 2-AXIS, and 3-AXIS.

HOLES: We use only the drill operation from this category. You will use the drill command on most jobs to pre-drill your fastening screws that mount your stock to the spoil board. Additionally you may need to pre-drill holes for dowels or other hardware to be fitted to your piece later, you would use a holes operation for this.

2-AXIS: This category includes profiling and pocketing. Profiling is used to cut a 2-d shape out of the stock, like scissors on a dotted line this performs profile cuts to produce the desired shape from the stock. Pocketing is used to generate areas of stock that have been cut out like the footprint of a building on a site or a shape that has been extruded into the stock.

3-AXIS: Finally we have our 3d operations, Horizontal roughing and parallel finishing. These are frequently, but not always used together to create a variety of 3d surfaces and forms. It is important to know our mill is only a three axis mill and therefore cannot handle any undercuts, so overhanging geometry is not possible. 3d operations are used to create curvilinear surfaces and topography.
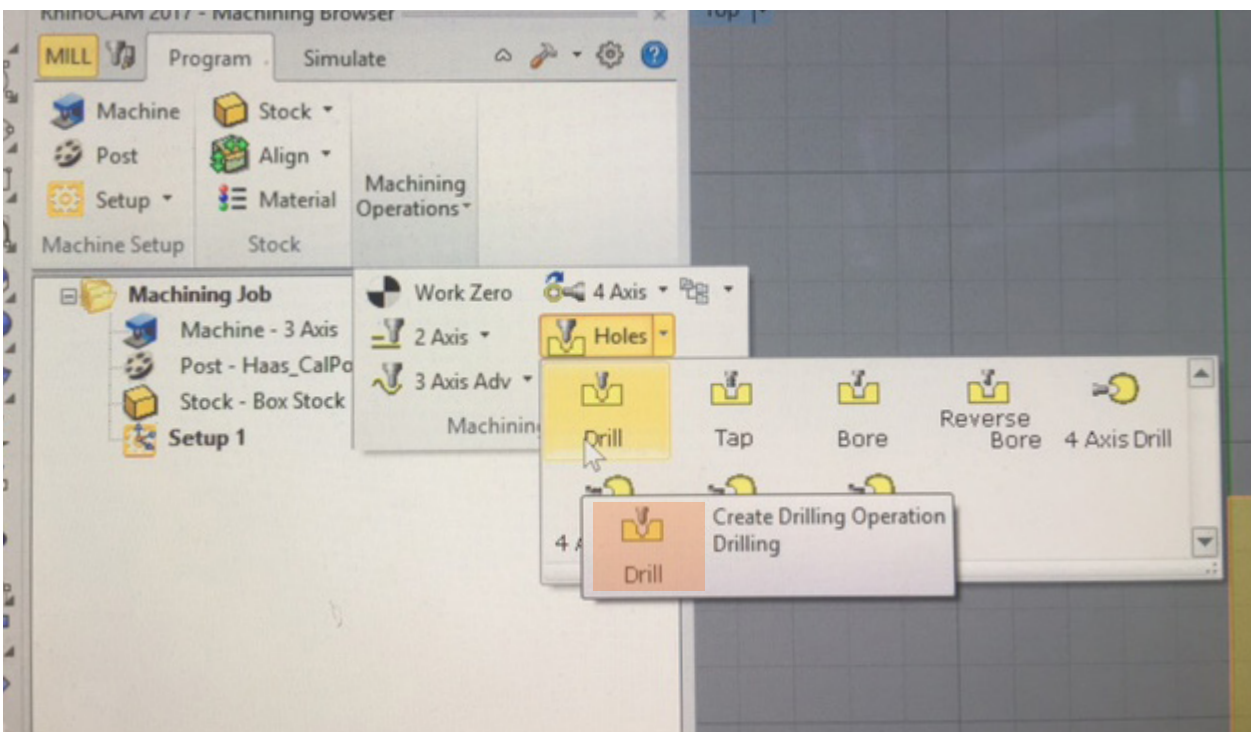
DRILLING

Begin by using the "Point" command in Rhino to draw points (minimum of 4) at least 1" away from the edge of the stock and any geometry. Place points at the TOP of the stock.
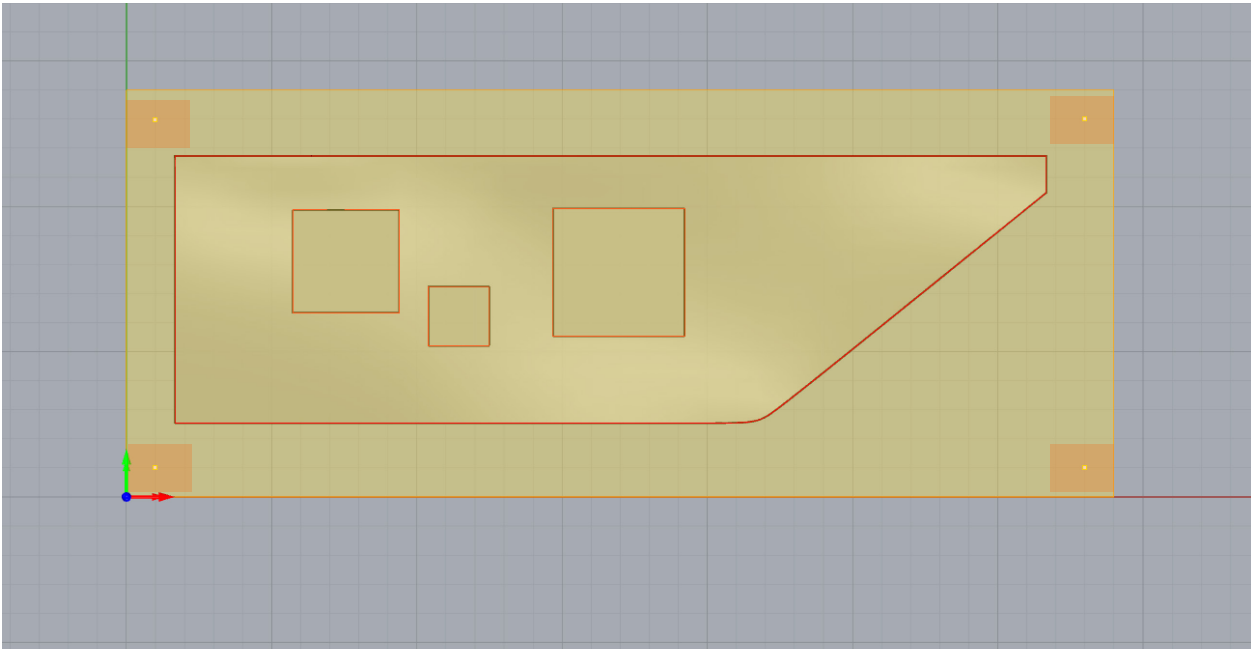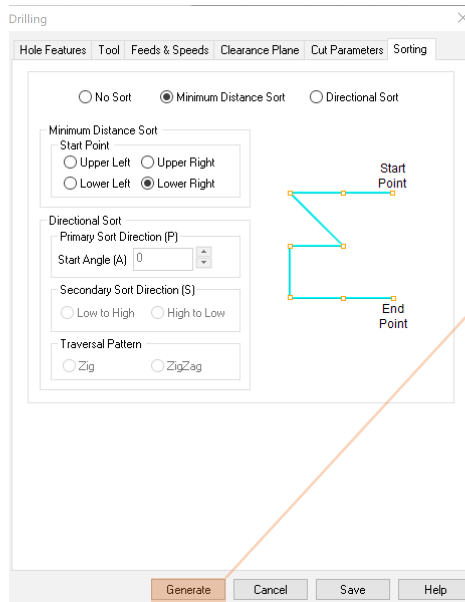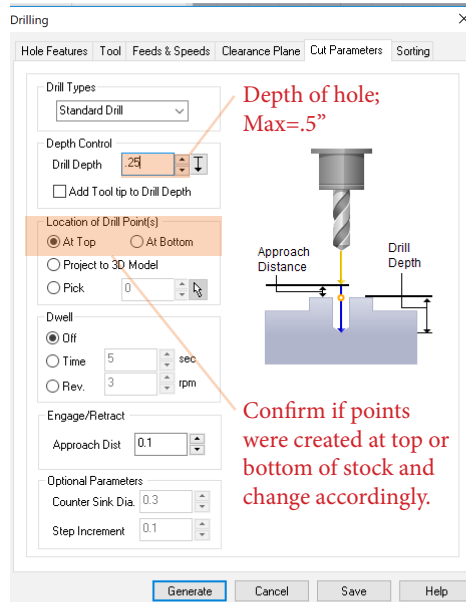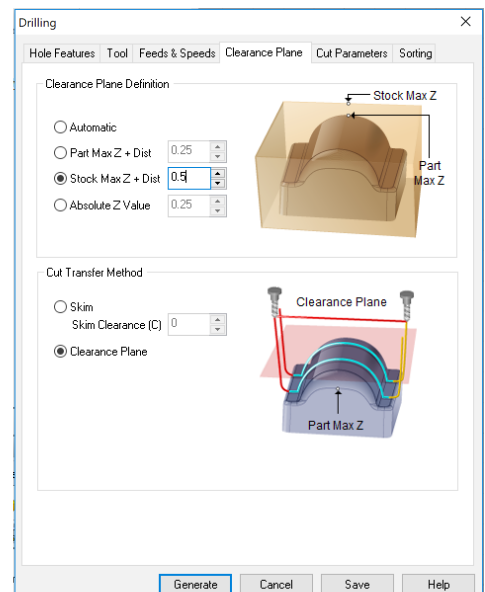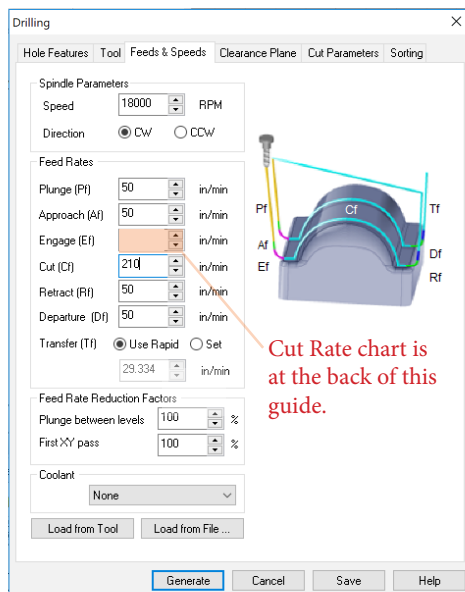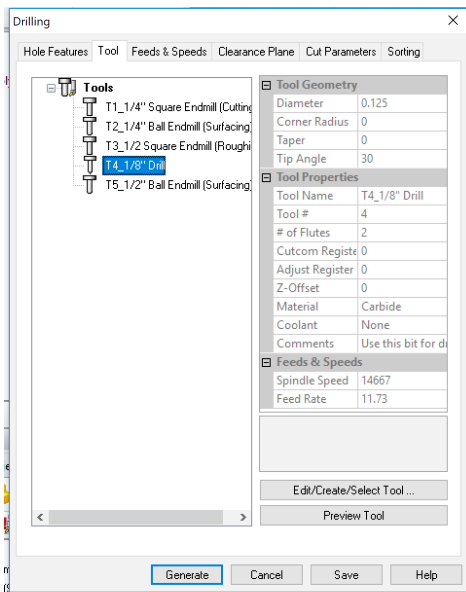

Now it is time to set up the operation. Click on the "Machining Operations" button then the "Holes" tab, lastly select Drill and a menu box will open.

③ Once the operation is created, it will ask you for the control geometry. Click on "Select Drill Points/Circles" and then left click on all the desired points, when done right click or press enter to finish selecting.
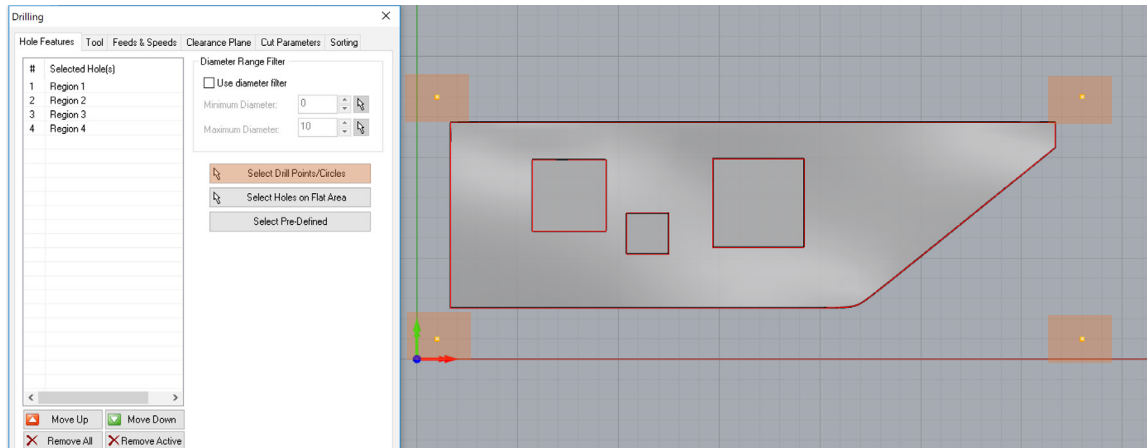
④ Next go through each of the settings tabs in the operation and make sure they match the information below:

Cut Rate chart is at the back of this guide.

Depth of hole; Max=.5"

Confirm if points were created at top or bottom of stock and change accordingly.

When all settings are complete click generate to create the tool path!

HORIZONTAL ROUGHING

① Begin by using the "Make2D" command or "Dupfaceborder" command in Rhino to create a profile line. Then locate the profile line at the TOP of the stock aligned with your cut piece. This is your control geometry.
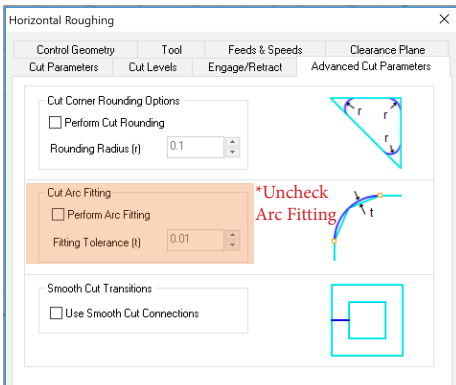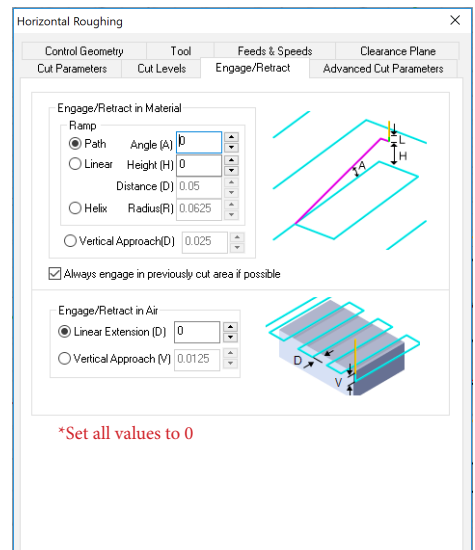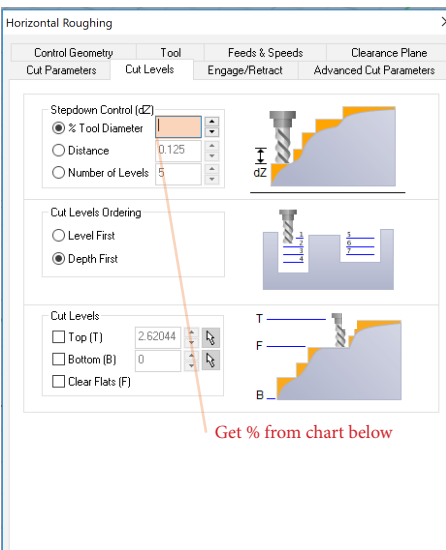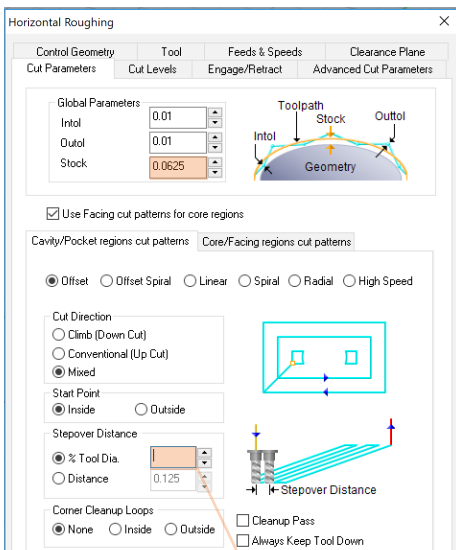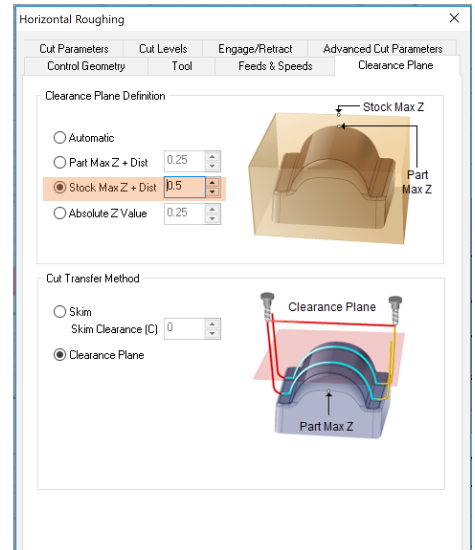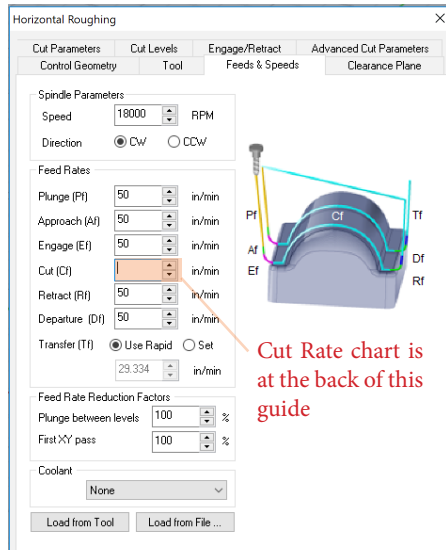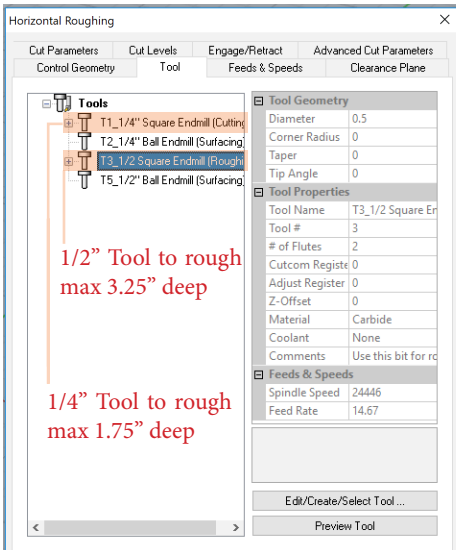

② Now it is time to set up the operation. Click on the "Machining Operations" button then the "3 Axis Adv" tab, lastly select Horizontal Roughing and a menu box will open.

③ Once the operation is created it will ask you for the control geometry. Click on "Select Curve/ Edge Region" and then left click on all the desired curves, when done right click or press enter to finish selecting.
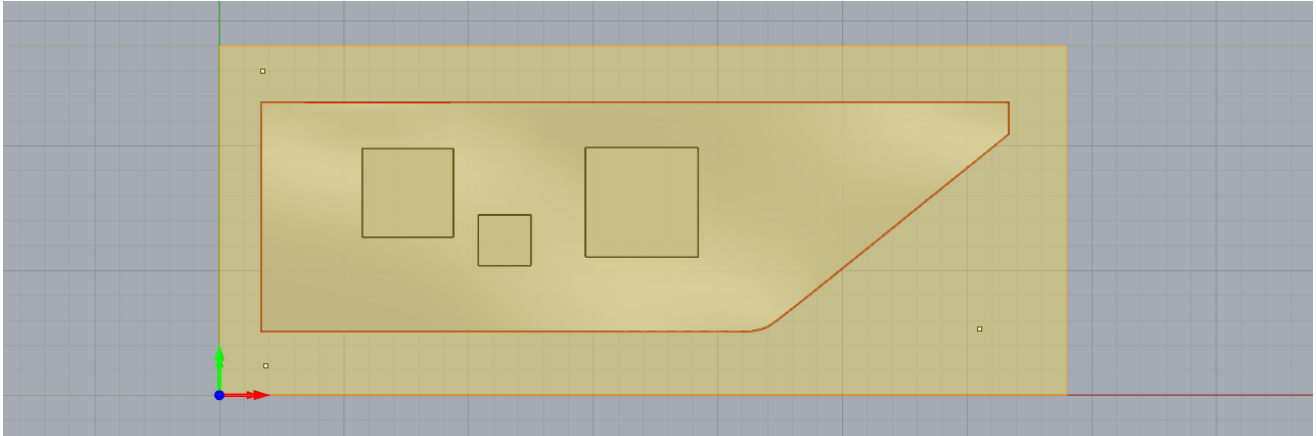
1/2" Tool to rough max 3.25" deep

1/4" Tool to rough max 1.75" deep

Cut Rate chart is at the back of this guide

Get % from chart below

Get % from chart below

*Set all values to 0

*Uncheck Arc Fitting

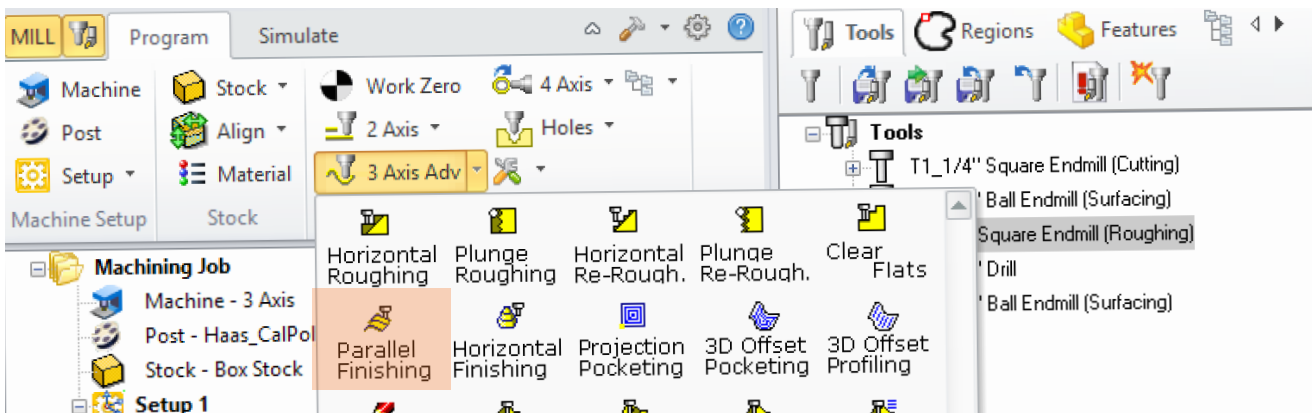When all settings are complete click generate to create the tool path!

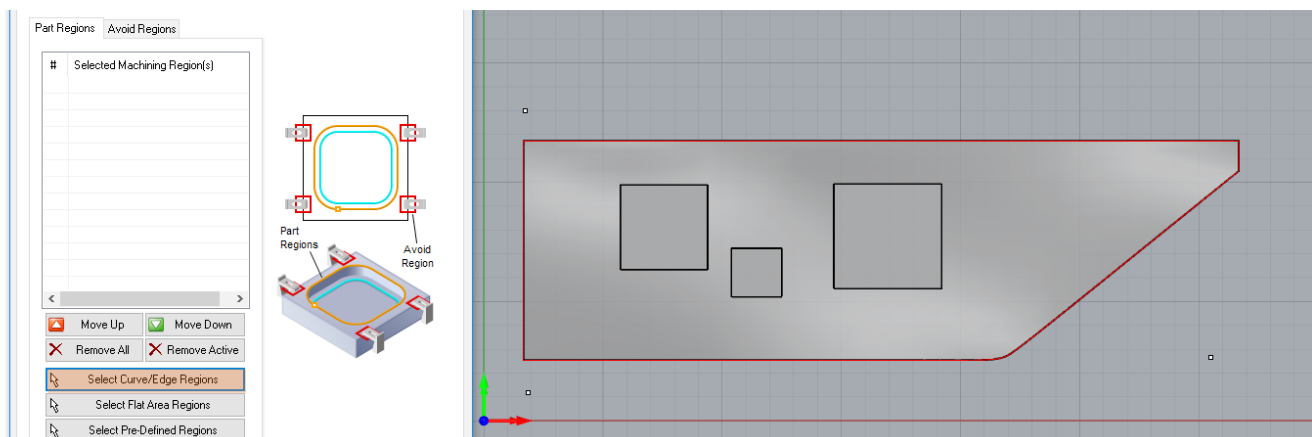| Material	Blue Foam	ULMDF	Plywood	Hardwood
Stepover	75%	50%	35%	25%
Stepdown	200%	100%	50%	50%

PARALLEL FINISHING

① Begin by using the "Make2D" command or "Dupfaceborder" command in Rhino to create a profile line. Then locate the profile line at the TOP of the stock aligned with your cut piece. This is your control geometry.
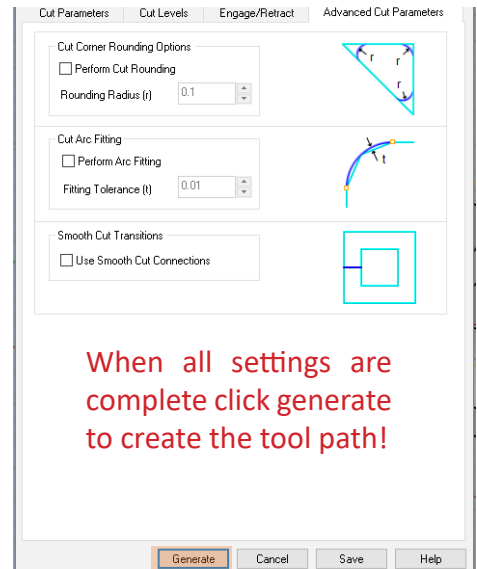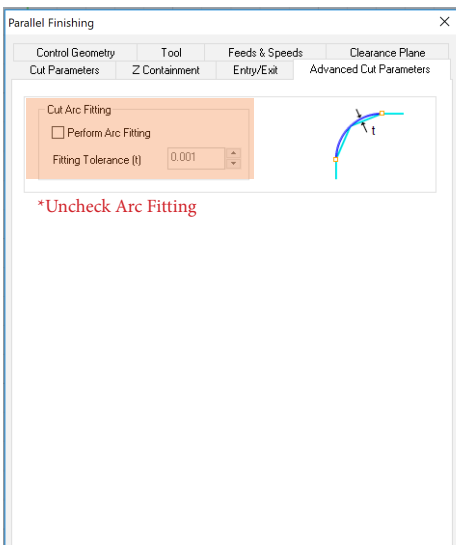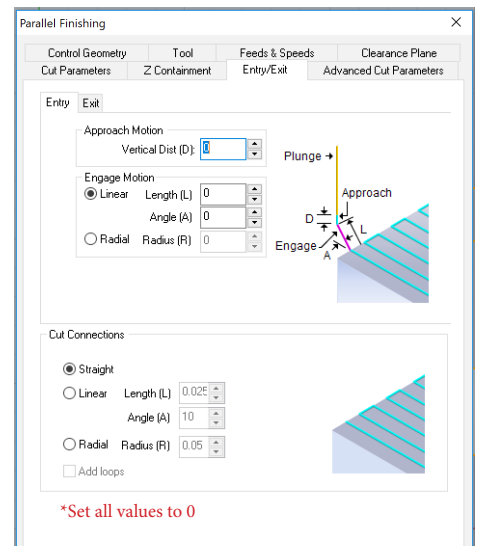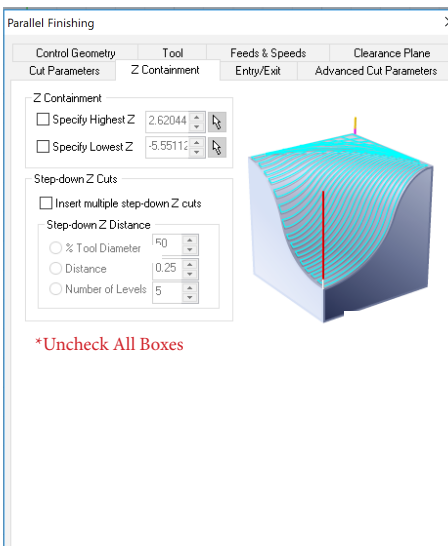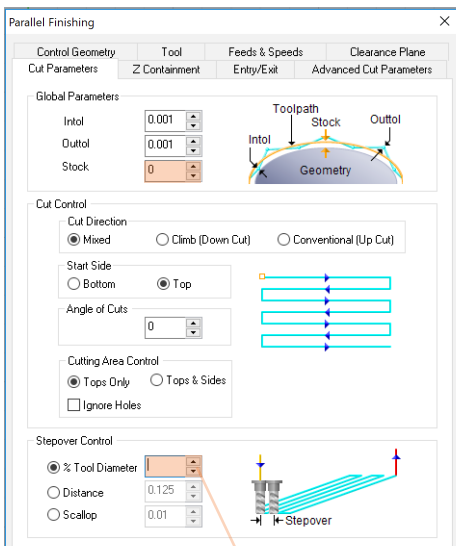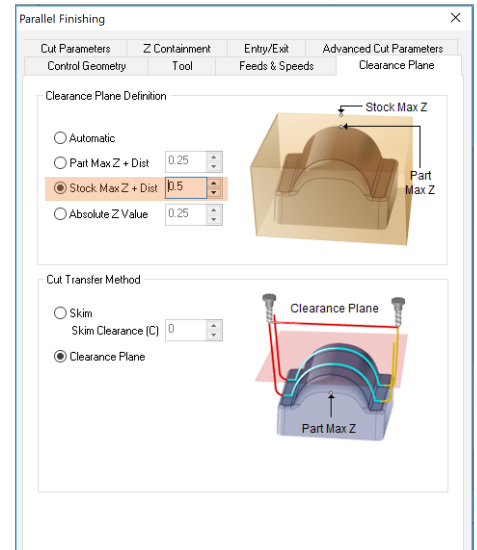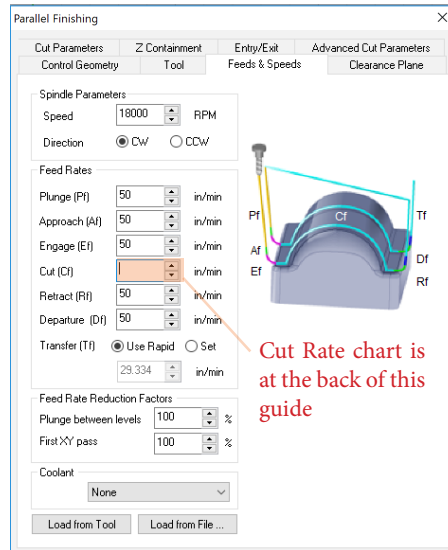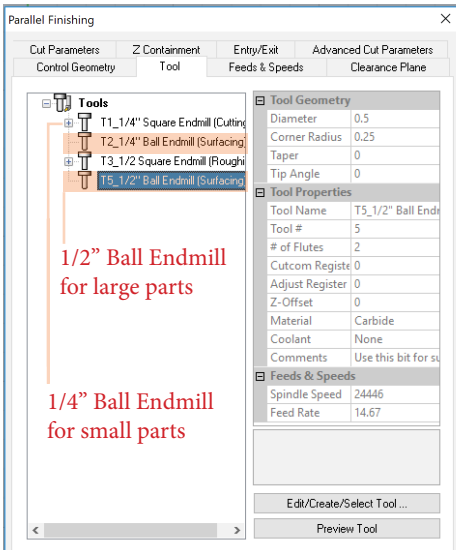

② Now it is time to set up the operation. Click on the "Machining Operations" button then the "3 Axis Adv" tab, lastly select Horizontal Roughing and a menu box will open.

③ Once the operation is created it will ask you for the control geometry. Click on "Select Curve/ Edge Region" and then left click on all the desired curves, when done right click or press enter to finish selecting.
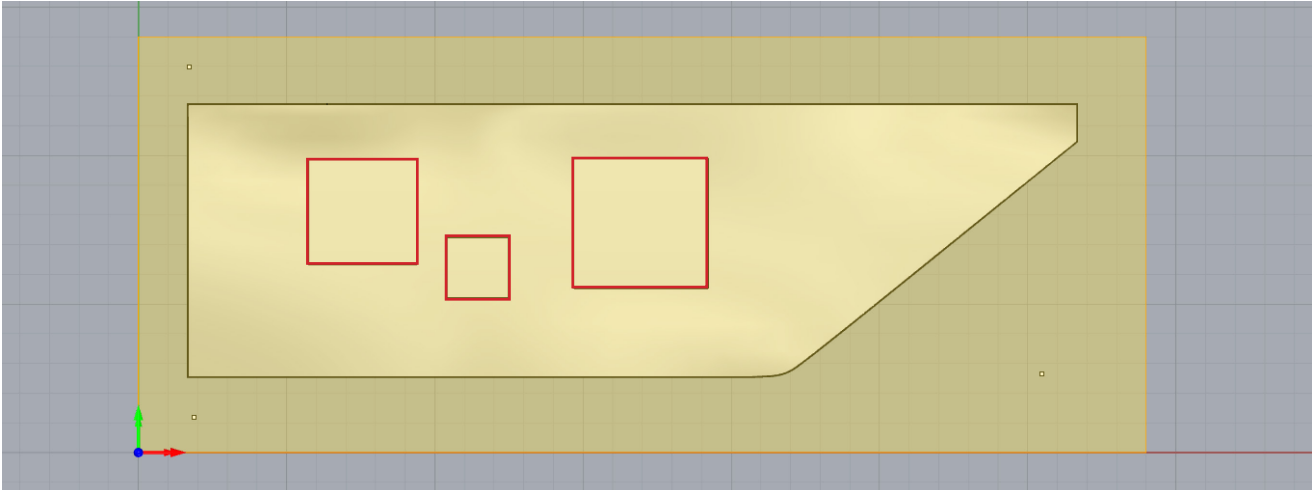
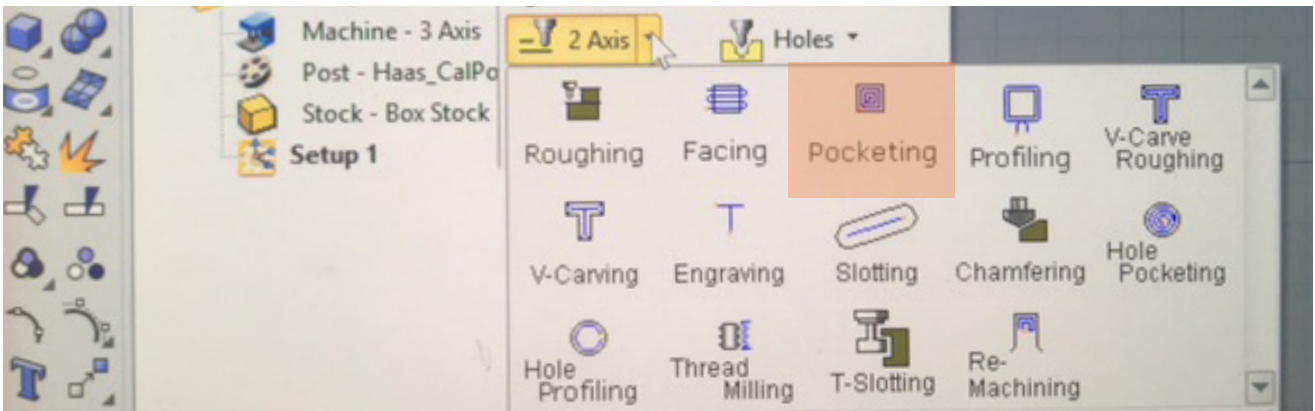**④** Next go through each of the settings tabs in the operation and make sure they match the information below:

1/2" Ball Endmill for large parts

1/4" Ball Endmill for small parts

Cut Rate chart is at the back of this guide

Get % from chart below

*Uncheck All Boxes

*Set all values to 0

*Uncheck Arc Fitting

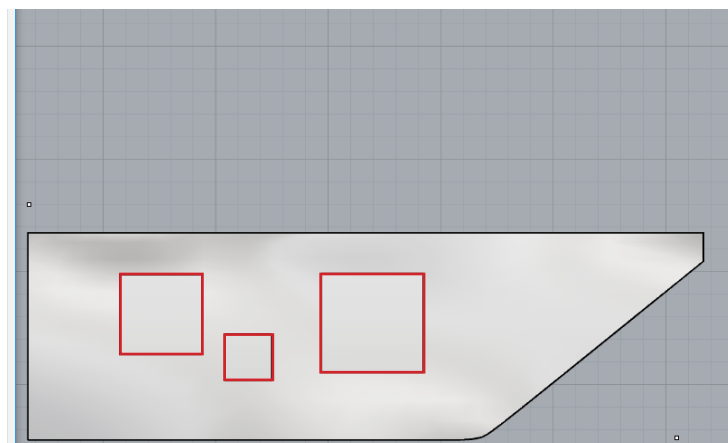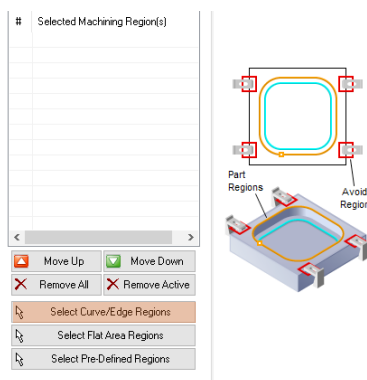When all settings are complete click generate to create the tool path!

POCKETING

① Begin by using the "Make2D" command or "Dupfaceborder" command in Rhino to create a closed curve. Then locate the closed curve at the TOP of the stock aligned with your pockets. This is your control geometry.
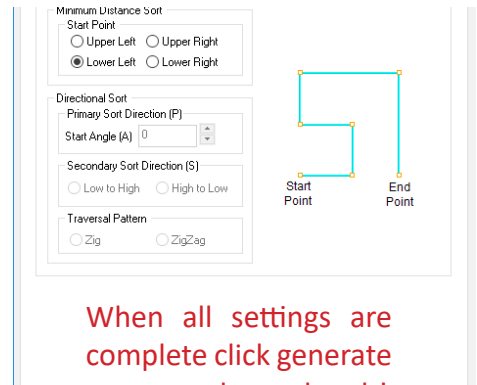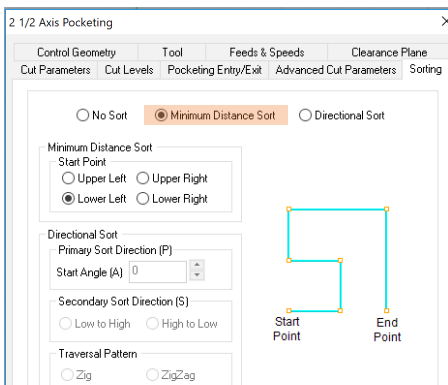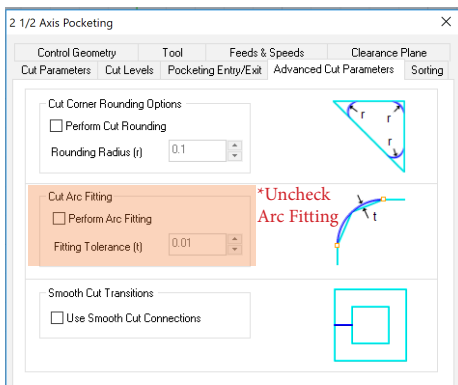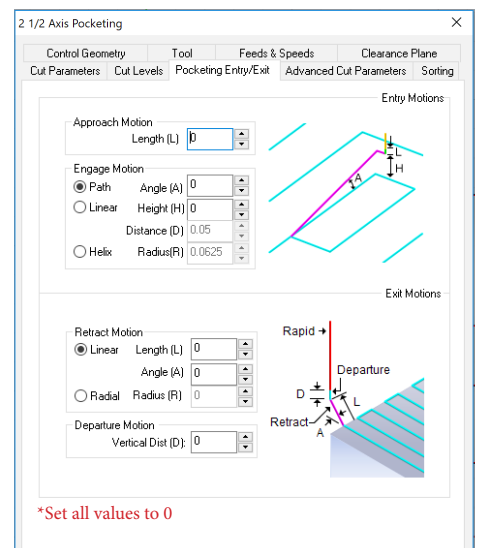

② Now it is time to set up the operation. Click on the "Machining Operations" button then the "2 Axis" tab, lastly select Pocketing and a menu box will open.

③ Once the operation is created it will ask you for the control geometry. Click on "Select Curve/ Edge Region" and then left click on all the desired curves, when done right click or press enter to finish selecting.
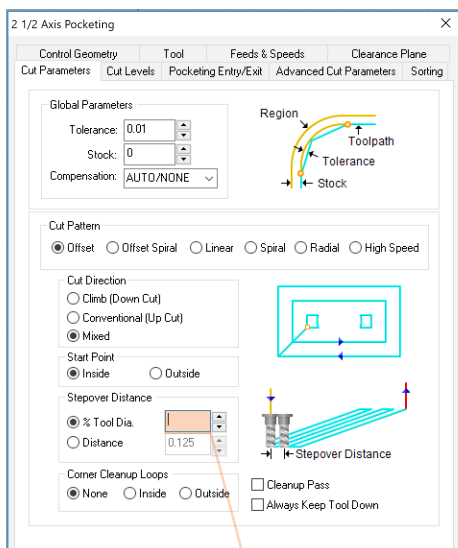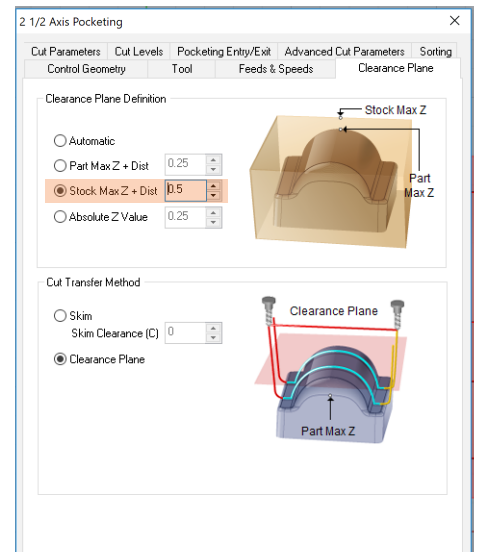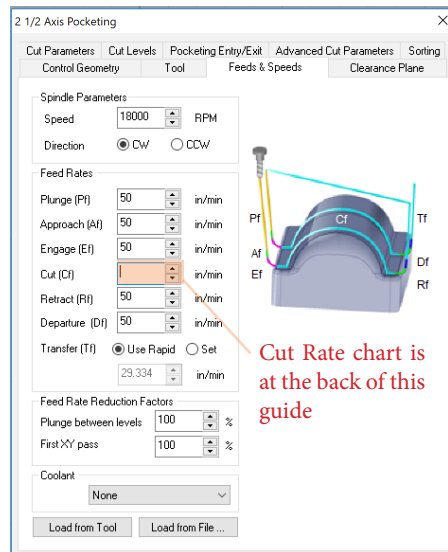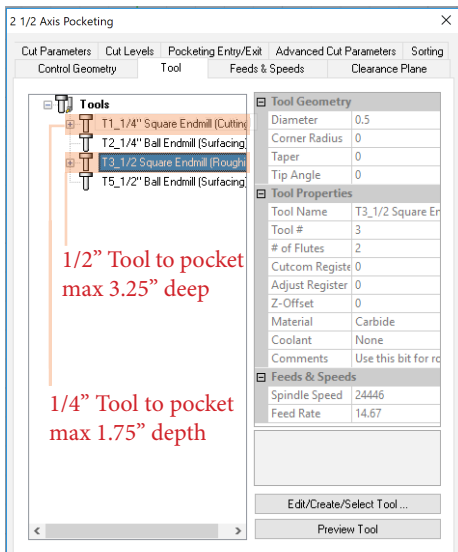
④ Next go through each of the settings tabs in the operation and make sure they match the information below:

Tool tab:
1/2" Tool to pocket max 3.25" deep

1/4" Tool to pocket max 1.75" depth

Feeds & Speeds tab:
Cut Rate chart is at the back of this guide

Cut Parameters tab:
Get % from chart below

Cut Levels tab:
Use tool to measure pocket depth or type in

.25" MAX for 1/2" Tool
.125" MAX for 1/4" Tool

Pocketing Entry/Exit tab:
*Set all values to 0

Advanced Cut Parameters tab:
*Uncheck Arc Fitting

Sorting tab:
When all settings are complete click generate to create the tool path!
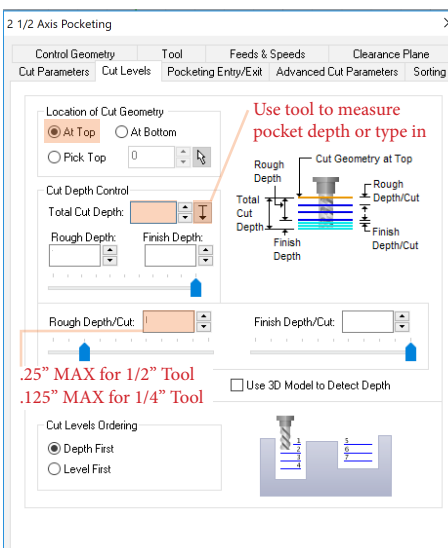
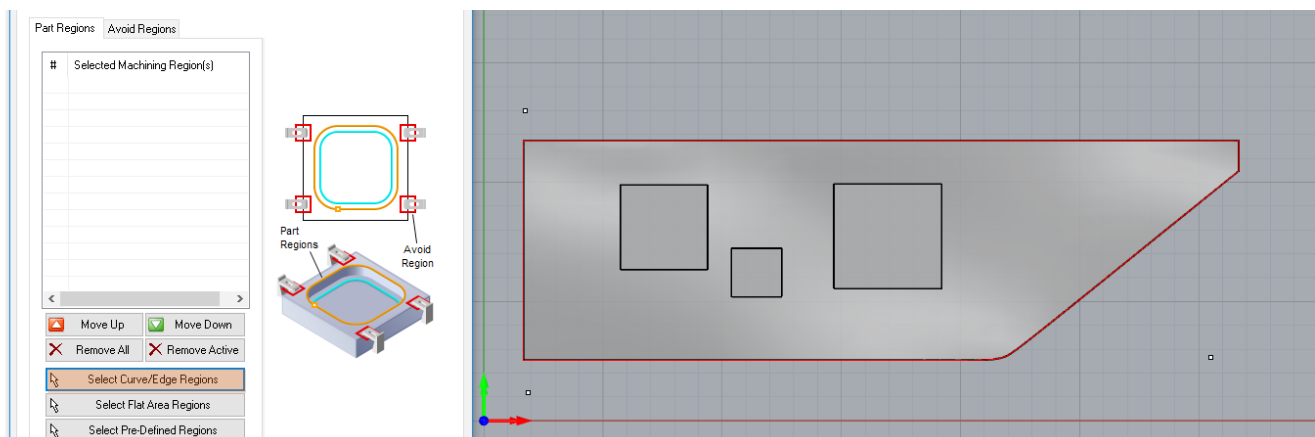| Material	Blue Foam	ULMDF	Plywood	Hardwood
Stepover	75%	50%	35%	25%
Stepdown	200%	100%	50%	50%

PROFILING

① Begin by using the "Make2D" command or "Dupfaceborder" command in Rhino to create a profile line. Then Locate the profile line at the TOP of the stock aligned with your cut piece. This is your control geometry.
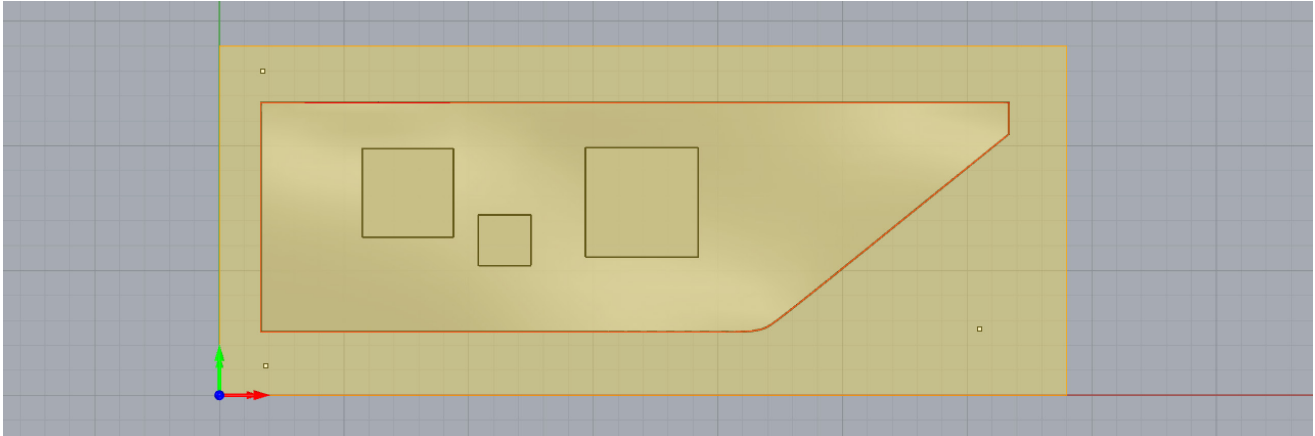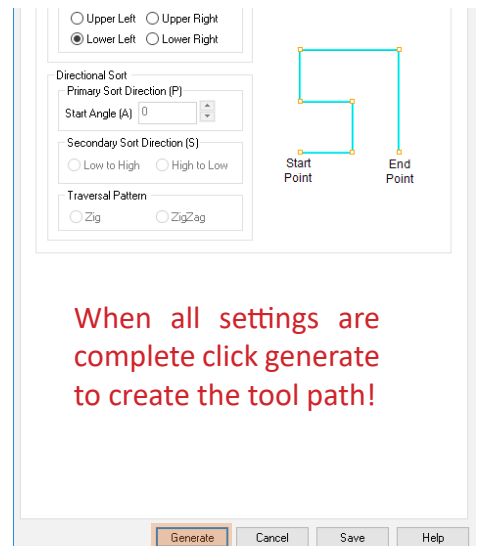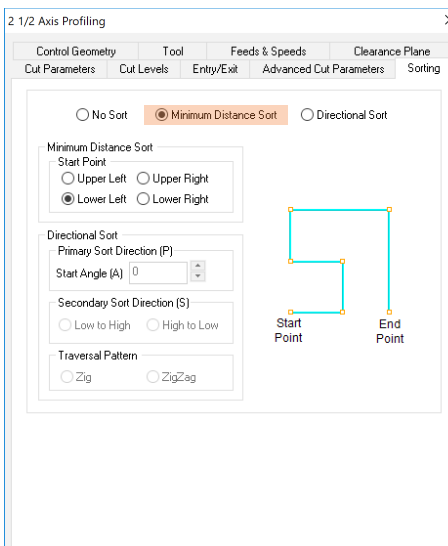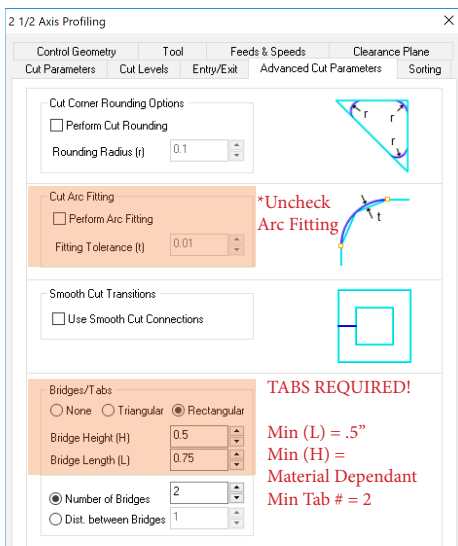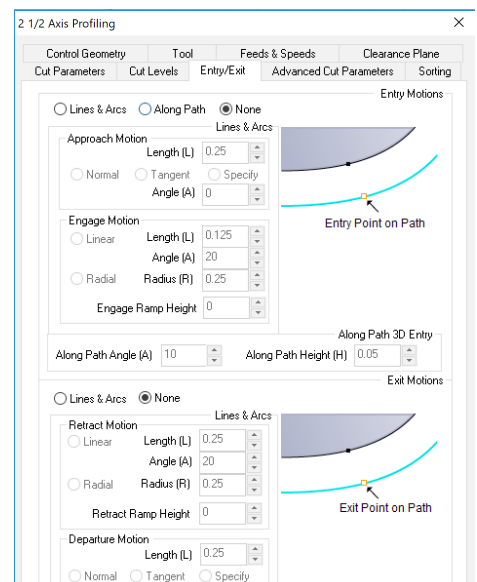

② Now it is time to set up the operation. Click on the "Machining Operations" button then the "2 Axis" tab, lastly select "Profiling" and a menu box will open.
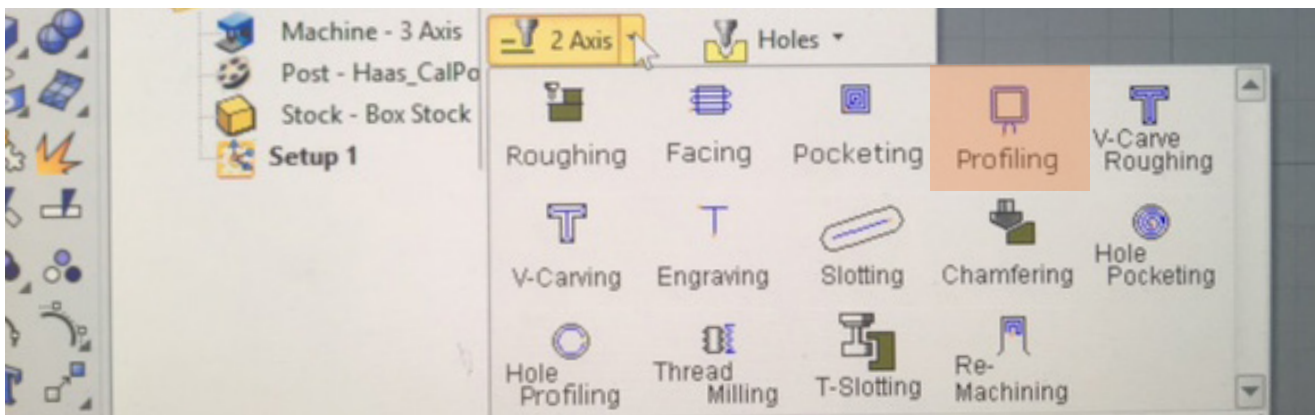
③ Once the operation is created it will ask you for the control geometry. Click on "Select Curve/ Edge Region" and then left click on all the desired curves, when done right click or press enter to finish selecting.

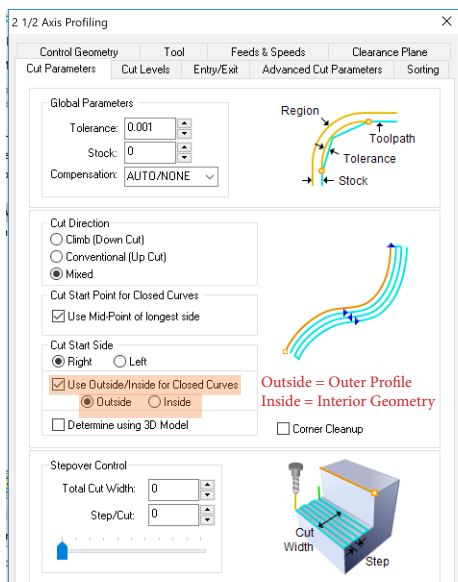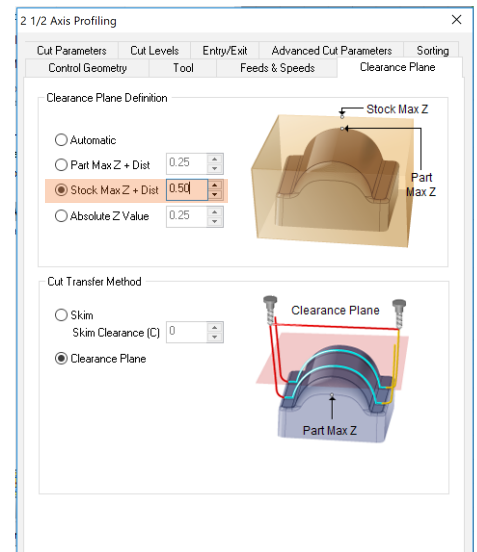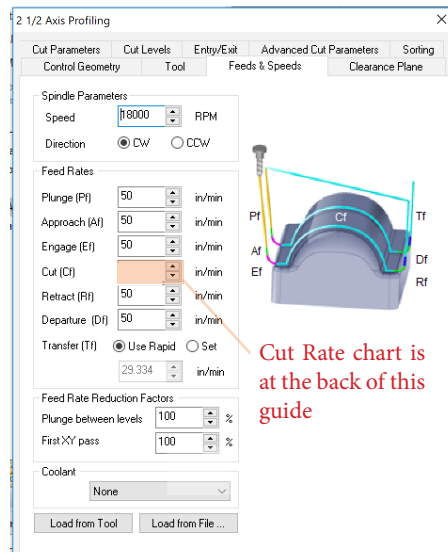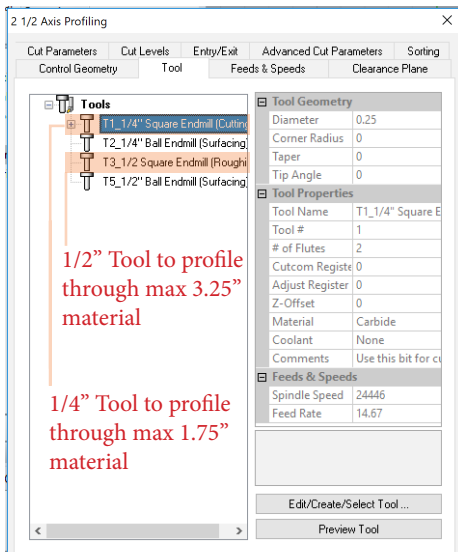④ Next go through each of the settings tabs in the operation and make sure they match the information below:

1/2" Tool to profile through max 3.25" material

1/4" Tool to profile through max 1.75" material

Cut Rate chart is at the back of this guide

Outside = Outer Profile
Inside = Interior Geometry

Material thickness + 0.03"

Autofills

.125"

Autofills

.25" MAX for 1/2" Tool
.125" MAX for 1/4" Tool

*Uncheck Arc Fitting

TABS REQUIRED!

Min (L) = .5"
Min (H) =
Material Dependant
Min Tab # = 2

When all settings are complete click generate to create the tool path!
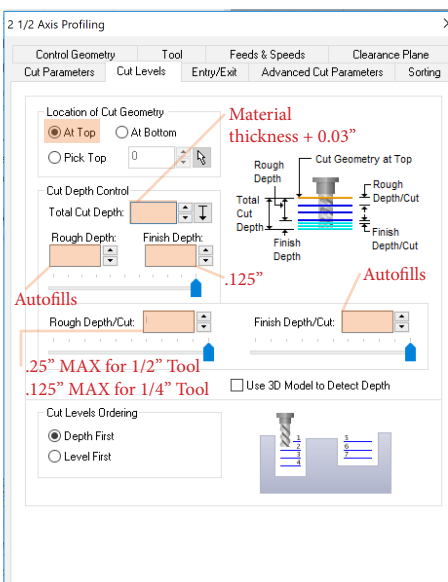
CUT RATES CHART

| CUT RATES					
Tool #	Onsrud Part #	Blue Foam (imp)	ULMDF (imp)	Plywood (imp)	Hardwood (imp)
1 (1/4" square)	48-179	180	150	150	150
2 (1/4" ball)	52-280B	215	200	200	150
3 (1/2" square)	48-183	250	200	200	200
5 (1/2" ball)	52-360B	250	250	250	250

REVIEWING CAM FILE

① Check the generated toolpaths and confirm that the mill will be doing exactly what you intend. Below are examples of what each of the toolpaths should look like for each of the machining operations that you set up:
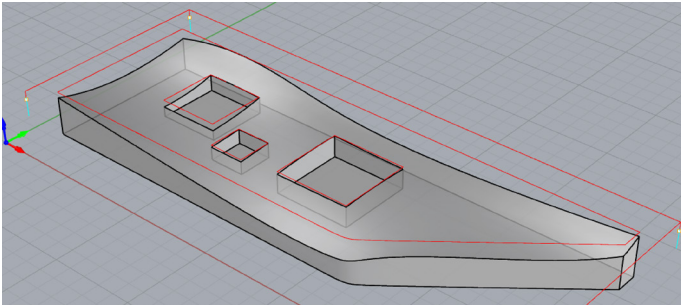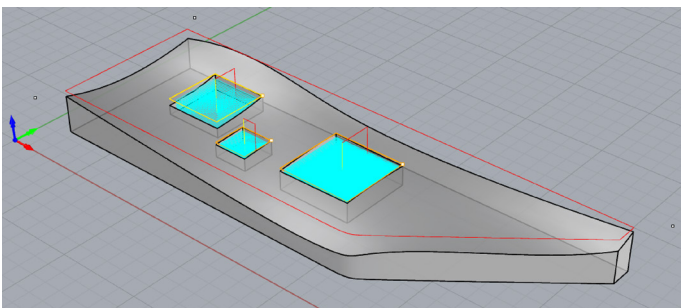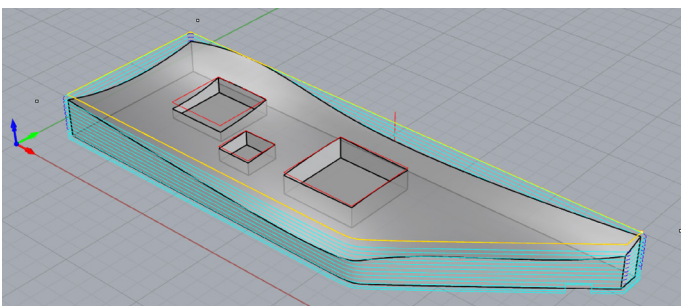

DRILLING TOOLPATHS
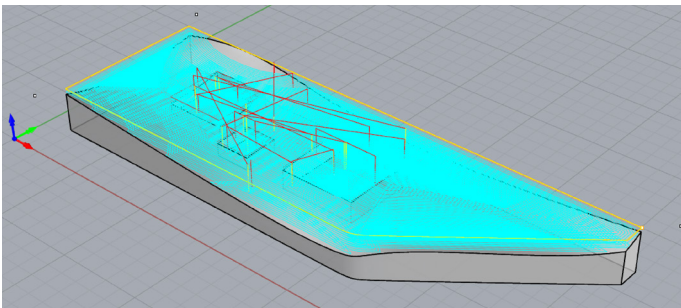
HORIZONTAL ROUGHING TOOLPATHS
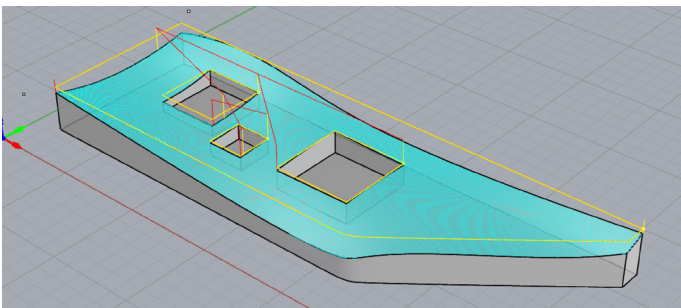

PARALLEL FINISHING TOOLPATHS

2 1/2 AXIS POCKETING TOOLPATHS

2 1/2 PROFILING TOOLPATHS

② Run the SIMULATE function to preview the mill job. Firstly, make sure that all operations under the "Setup" are in the correct order. Next, highlight Setup 1. Open the "Simulate" tab and press the "Play" button to watch the mill cut out your part. If the CAM file will not simulate or if any regions of the stock turn red, notify a d[Fab] lab technician.
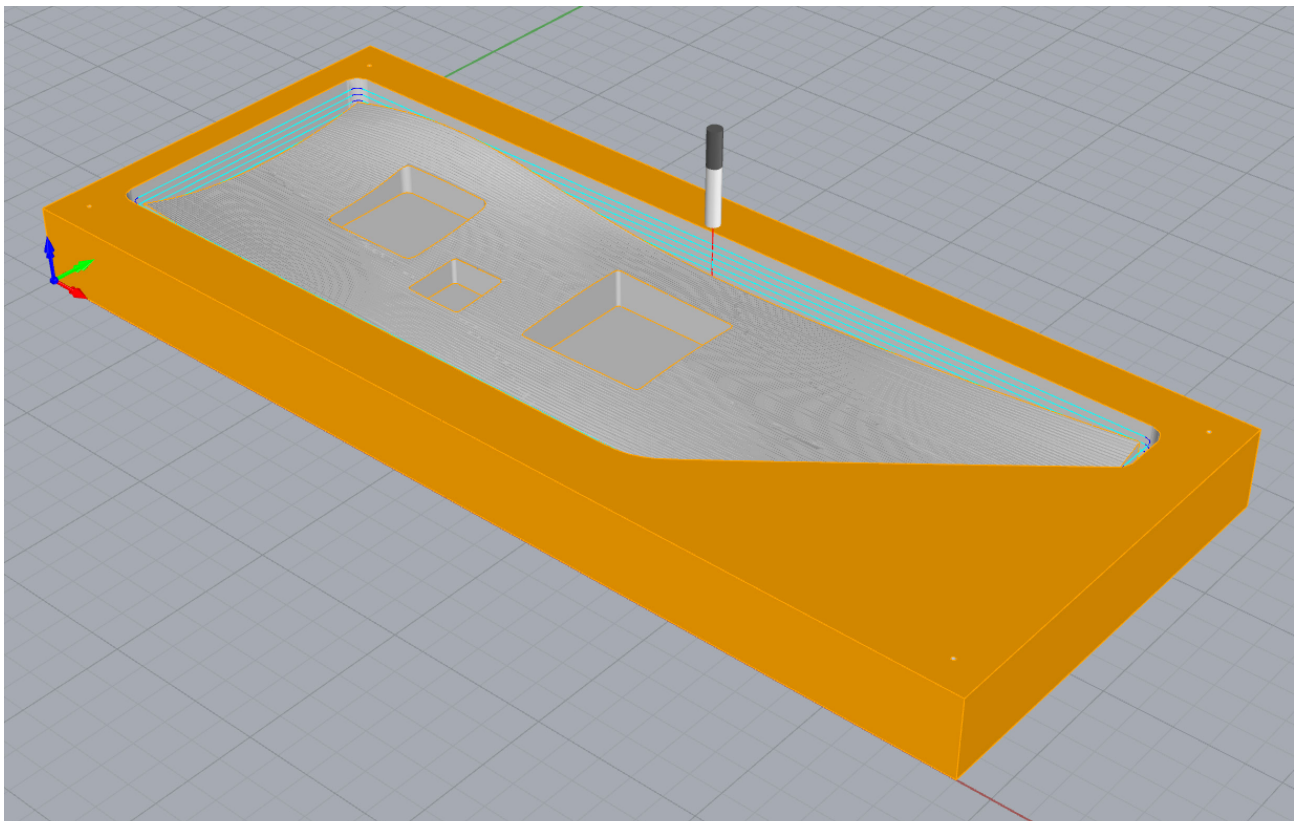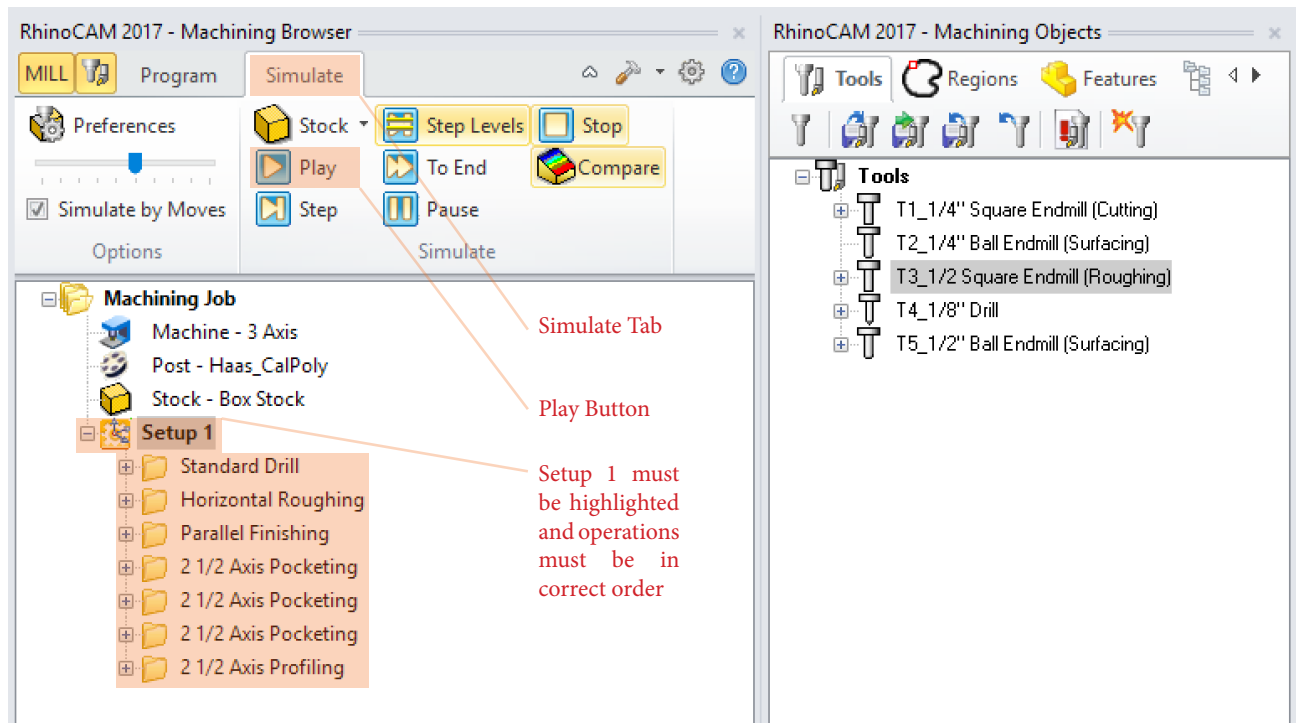
③ If your CAM file passes the simulation mode, you are ready to have your file reviewed by a d[Fab] lab technician. At this point, the technician will review your model, stock, and operations to set up the "Post" files and get you on the calendar to use the mill. Until you have been officially cleared by an *authorized* lab technician, you DO NOT have a mill appointment. Once your fill out a CNC Router Pre-Flight Form and get your file cleared, you will be set up with an appointment.