RhinoCAM Guide

UNLV School of Architecture
Fall 2023

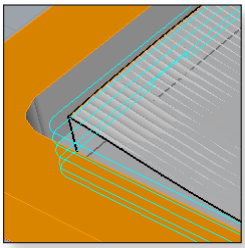
Introduction

RhinoCAM is a Rhino plugin that allows toolpaths to be created then exported to GCode for the CNC router. This is a step-by-step guide that shows how to create toolpaths using RhinoCAM.

This guide covers initial file setup and the most common machining operations used in RhinoCAM: ***profiling****,* ***pocketing****,* ***horizontal roughing****,* ***parallel finishing****,* ***drilling, and engraving****. Please note that this guide is intended to help introduce users to RhinoCAM and does not cover all the machining operations available in RhinoCAM. Also, users should be aware that not all machining operations reviewed by this guide may be applicable to every project. Users must determine what machining operations are appropriate for their project.*

Contents

I. Common Machining Operations

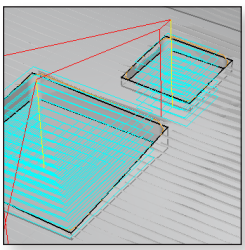


Profiling

Description: *Operation used to cut 2D linework and paths*
Tools: *1,3, and 4*

Typical Uses:

1. *Cut pieces out of sheet stock*
2. *Perform perimeter cut after surfacing operations*

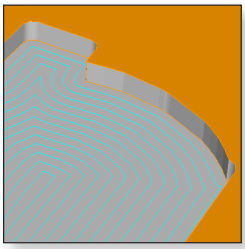


Pocketing

Description: *Operation used to surface 2D regions*
Tools: *1 and 3*

Typical Uses:

1. *Cut flat bottomed recessed areas into stock*

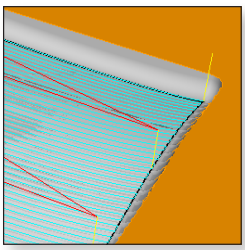


Horizontal Roughing

Description: *Roughing, or rough cutting, 3D surfaces*
Tools: *1 and 3*

Typical Uses:

1. *Prepare for parallel finishing operation*
2. *Create “stepped” surfaces*

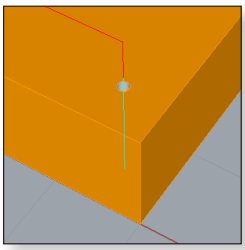


Parallel Finishing

Description: *Detail finishing of 3D surfaces*
Tools: *2 and 5*

Typical Uses:

1. *Creating smooth 3D surfaces*

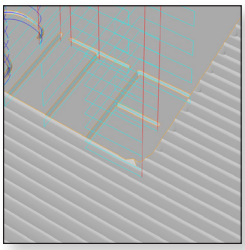


Drilling

Description: *Drilling holes*
Tools: *6*

Typical Uses:

1. *Create holes to locate material hold down screws*



Engraving

Description: *Engrave text or linework*
Tools: *7*

II. Feeds, Speeds, and Max Cut Depths

Tool 1: 1/4” Square End Mill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	150	150	120	120
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	2"			

Tool 2: 1/4” Ball End Mill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	145-190	145-190	120-170	120-170
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	1-1/2"			

Tool 3: 1/2” Square End Mill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	230	230	140	140
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	2"			

Tool 4: 1/2” Compression				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	270	270	190	190
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	2-1/2"			

Tool 5: 1/2” Ball End Mill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	190-240	190-240	190-220	190-220
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	2-1/2"			

II. Feeds, Speeds, and Max Cut Depths (Cont'd)

Tool 6: 1/8" Drill				
	Material			
	Foam	*MDF*	*Plywood*	*Hardwood*
Cut Feed Rate (IPM)	50	30	30	30
All other Feed Rates (IPM)	50			
Speed (RPM)	6,000			
Max Cut Depth	2"			

Tool 7: 1/4" Vee Mill				
	Material			
	Foam	*MDF*	*Plywood*	*Hardwood*
Cut Feed Rate (IPM)	80	50	30	30
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	3/8"			

III. Stepdowns & Stepovers

Max Stepover Distance: Horizontal Roughing & Pocketing				
	Material			
	Foam	MDF	Plywood	Hardwood
Stepover (% of bit diameter)	75	50	35	25

Max Stepover Distance: Parallel Finishing				
	Material			
	Foam	MDF	Plywood	Hardwood
Stepover (% of bit diameter)	75% - 50%	50% - 35%	35% - 25%	25% - 20%

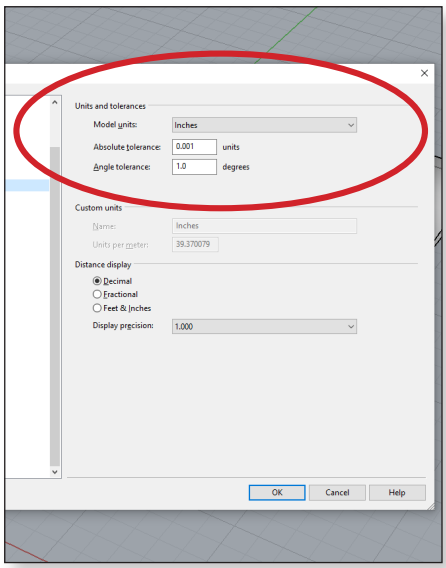
Max Stepdown Distance				
	Material			
	Foam	MDF	Plywood*	Hardwood*
Stepover (% of bit diameter)	75	50	25	25

* Tool 4 (1/2 Compression) stepdown percentages can be increased by 25%.

Part 1 File Setup

1. Launch Rhino 7 & Set Model Units to Inches

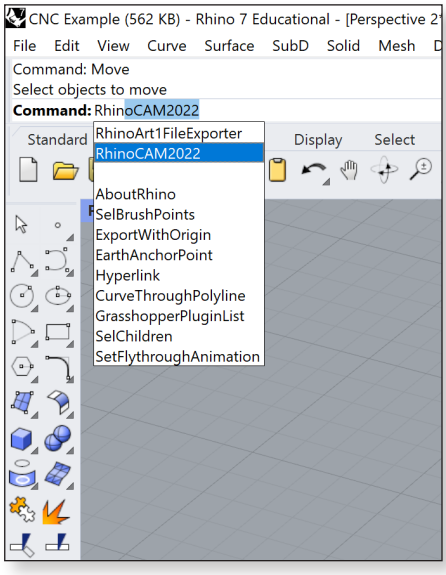
*Type **"units"** into the command bar to display the document properties dialog box.*



2. Launch RhinoCAM 2022

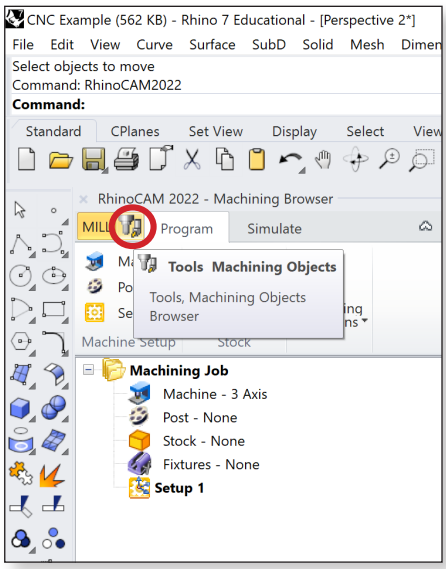
*If the RhinoCAM window is not visible, type **"RhinoCAM 2022"** into the command bar to activate the plugin.*

*Ensure **"MILL"** is activated at the top left of the RhinoCAM 2022 window.*



3. Open Tools Machining Objects

Select the icon with the image of a tool directly to the right of the "mill" icon at the top left of the RhinoCAM 2022 window to launch the tools machining objects window.

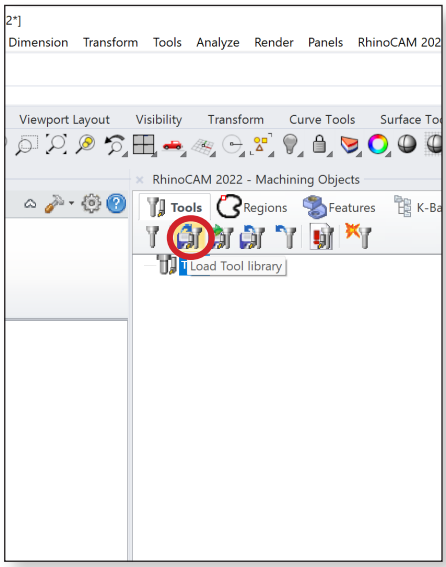


4. Load Tool Library

*Download the **".vkb"** file from: digitalfabricationlab.sites.unlv.edu/rhinocam.*

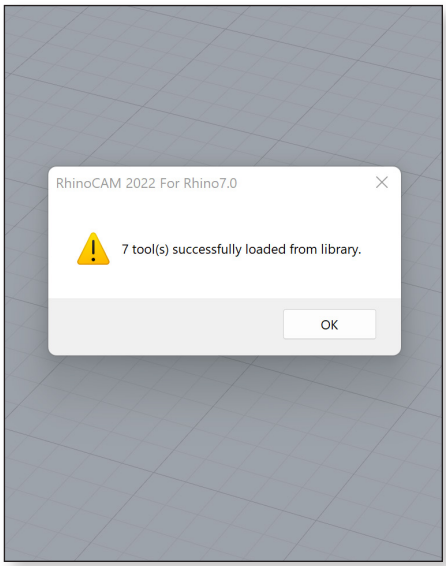
To load the library, click on the second icon from the left under the tools tab in the machining objects window.

*Locate the **".vkb"** file and click open in the lower right corner.*



5. Tools Successfully Loaded

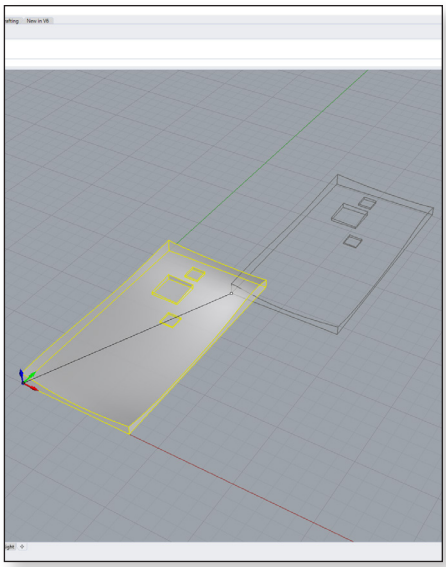
If the tool library is successfully loaded, the tool library will be listed in the machining objects window.



6. Move Parts to Origin 0,0,0

*Parts can be moved to the origin by using the **move** command and typing 0,0,0.*

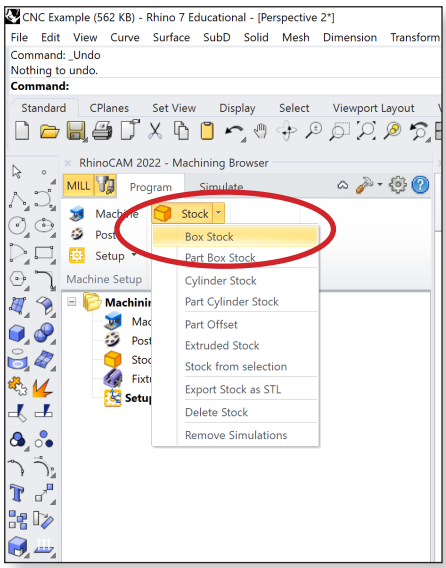
***Tip:** For best results, purge all unnecessary geometry from the file or begin with a fresh file.*



7. Set Box Stock

*Set the stock size that will be milled. Select the **"Stock"** icon and select **"Box Stock"** to launch the dialog window.*

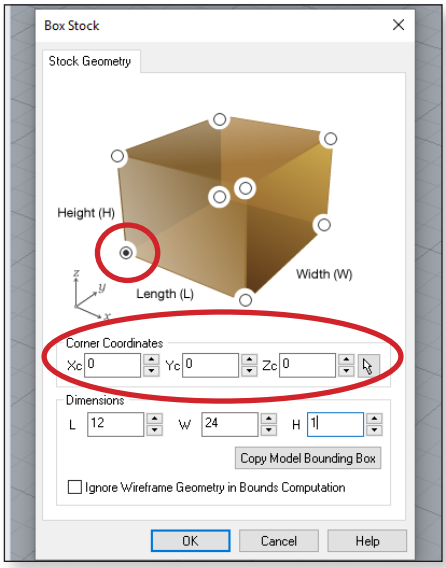
***Important:** Obtain the stock before programming and measure the stock thickness with calipers. If two or more pieces of stock are laminated (glued) together, the glue must be allowed to cure for **24 hours** prior to milling.*



8. Input Stock Dimensions

*The lower left corner of the stock should always be located at the origin. Thus, in the dialog box, the **"Corner Coordinates"** should read 0,0,0. Enter the length, width, and height in the **"Dimensions"** portion of the dialog box.*

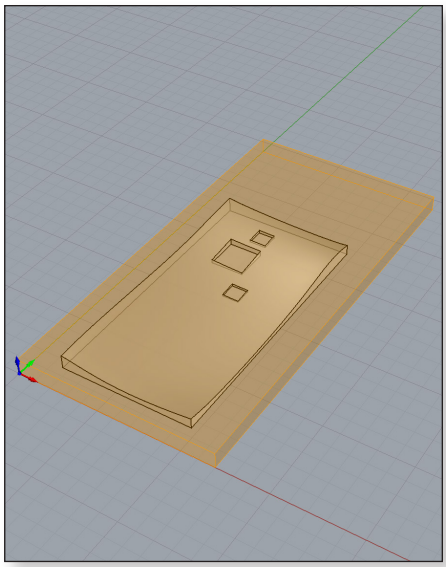
("L" corresponds to the dimension of stock along the X-axis; "W" corresponds to the dimension of stock along the Y-axis.)



9. Locating Geometry Within the Box Stock

*The piece of stock must be larger than the part to be milled. This allows for a **1”** perimeter border to be left around the stock for attachment to the spoil board.*

***Important:** It is difficult to get stock perfectly aligned to the origin and stock edges may not be true. If an absolute size is desired for the final part, it is always advisable to perform all milling operations, then cut the final piece out of the stock at the end of the job.*



10. Postscript

*Download the **“.spm”** file from: digitalfabricationlab.sites.unlv.edu/rhinocam.*

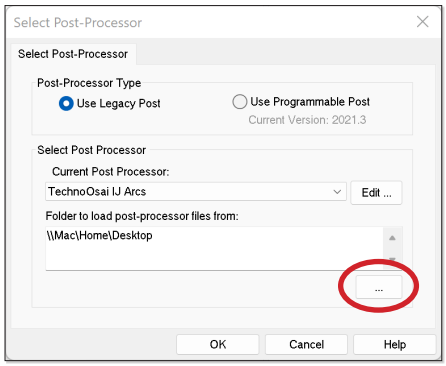
*Select **“Post”**.*

11. Loading Postscript

*Select **“...”** in the dialog box.*

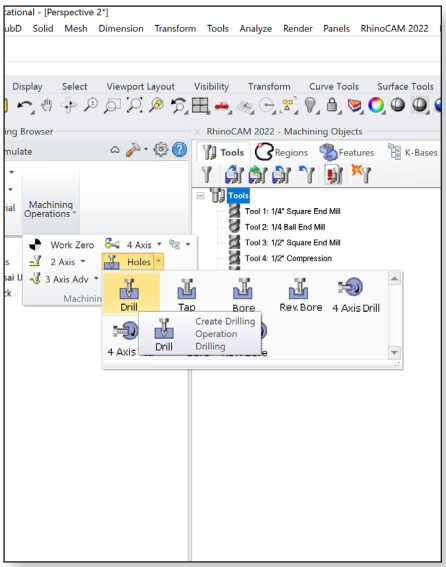
*Redirect to the folder where the **“TechnoOsai IJ Arcs”** postscript is located. Select **“OK”**.*

*The **“TechnoOsai IJ Arcs”** postscript should populate under current post processor. Select **“OK”**.*



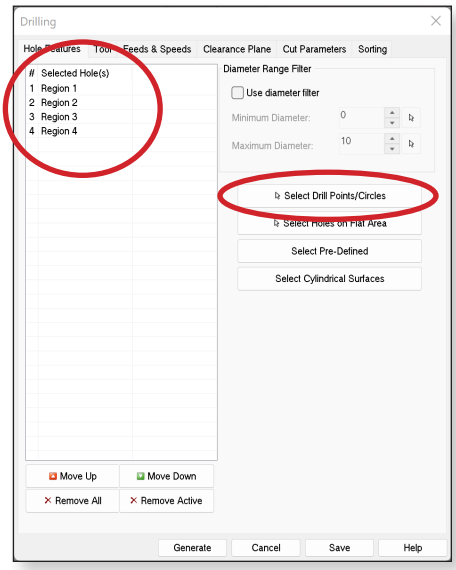
Part 2 Drilling

1. Locate the Drill Machining Operation Icon



*Under the program tab in the RhinoCAM dialog box, there is a **"Machining Operations"** drop down menu. **"Drill"** can be found under the **"Holes"** drop down menu.*

2. Selecting Geometry



*When the drill icon is selected, a dialog box will appear with a series of tabs located at the top. **It is imperative to input information and check settings in every tab.***

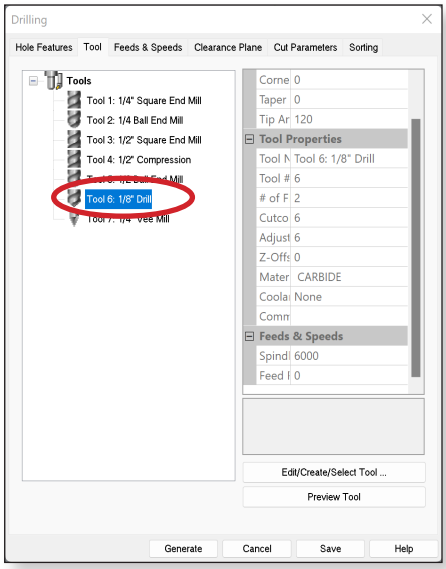
*The first step is to select the geometry to perform the operation. Click **"Select Drill Points/Circles"**; the dialog box will disappear. Select the appropriate geometry, then press enter.*

*After the dialog box reappears, it will display what has been selected in the previous step in the list on the left. Once satisfied with the selection, click the **"Tool"** tab at the top to continue to the next step.*

3. Select Tool For Drilling Operation

The tool library will appear in the list on the left.

*Select the **1/8" Drill (Tool 6)** for this operation. Click the **"Feeds & Speeds"** tab on the top to continue to the next step.*



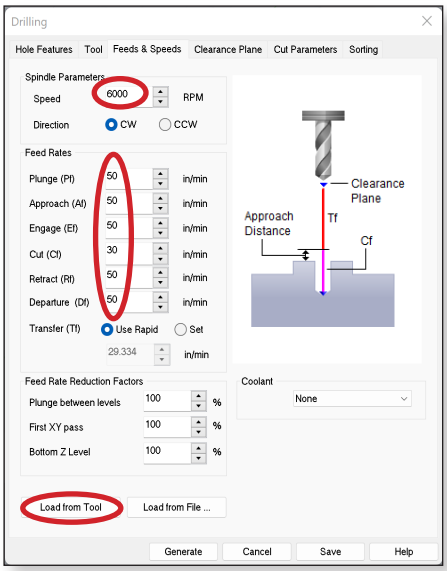
4. Set Feeds & Speeds

Feeds & speeds correspond to the rate at which the CNC router moves the tool across the part and the RPM (revolutions per minute) that the router bit spins.

*Feeds & speeds tables for each tool and material combination are located in a previous section of this guide. Use **50 in/min** for all other feed rates.*

Tool 6: 1/8" Drill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	50	30	30	30
All other Feed Rates (IPM)	50			
Speed (RPM)	6,000			
Max Cut Depth	2"			

***Tip:** Clicking **"Load from Tool"** will auto populate the 'speed' and all feed rates except 'Cut (CF)'.*

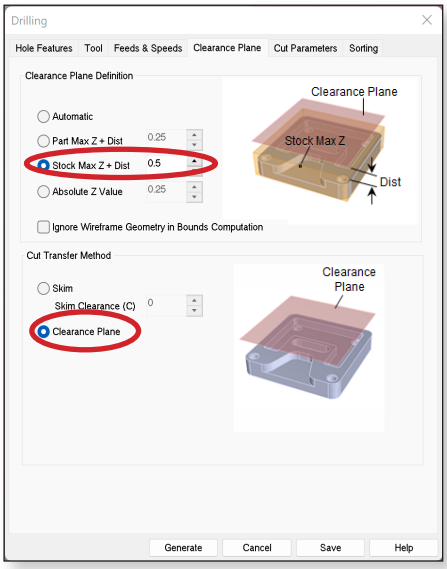


5. Clearance Plane

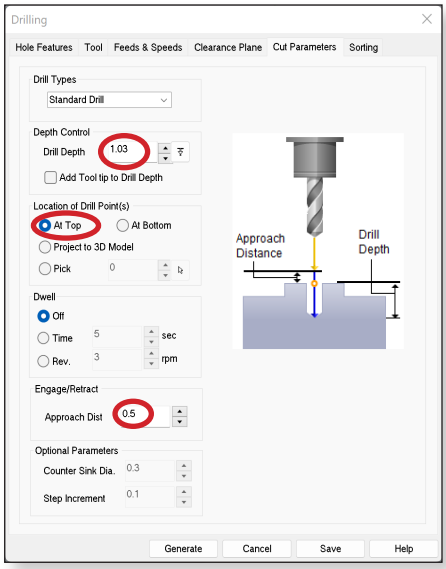
Establishing a clearance plane determines how far the router bit will be away from the stock during transfers. It is important that the clearance plane is always above the stock so that there is no contact during transfer.

*Set the clearance plane **0.5"** above the maximum height (or max Z value) of the stock.*

*Under cut transfer method, select **"Clearance Plane"**.*



6. Set Cut Parameters



*When drilling basic holes, select **"Standard Drill"**.*

*Input the depth of the hole you want to drill under **"Drill Depth"**.*

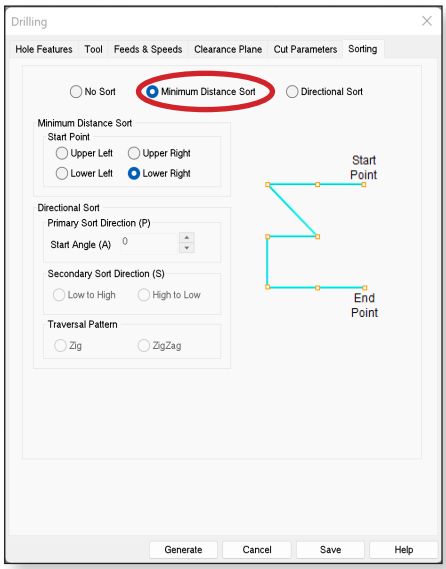
***Tip:** If the intent is to cut completely through the stock, it is advisable to add **0.03"** to the total cut depth to account for variations in the stock. In other words, if the stock was 1" thick, input the total cut depth as **1.03"**.*

*Set the location of drill points. Always locate drill points at the **top** of the part.*

*The dwell setting provides a delay during the drilling operation. Set dwell to **"off"**.*

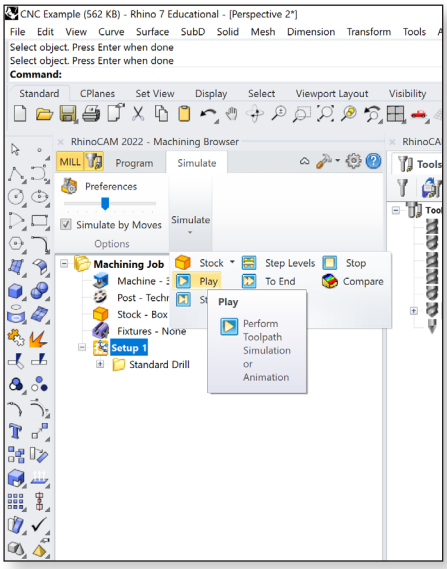
*The approach distance is the distance above the part where the specified feed rate is applied. Set the approach distance to **0.5"**.*

7. Set Sorting



*Sorting establishes the order that multiple holes are drilled. To optimize job time, it is recommended to set the sorting to **"Minimum Distance Sort"**.*

8. Simulate

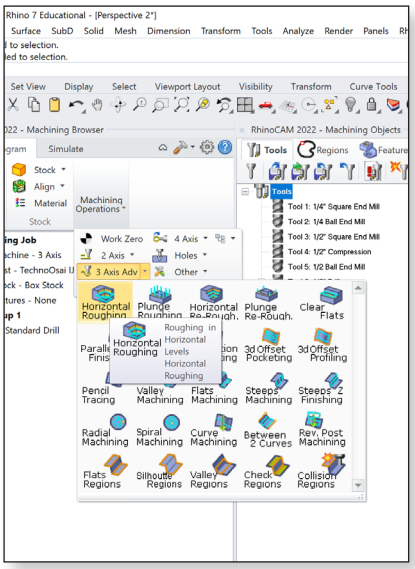


*If **"Setup 1"** is selected, all tool operations will be simulated. To start the simulation, select **"Play"** under the simulate tab at the top of the machining browser.*

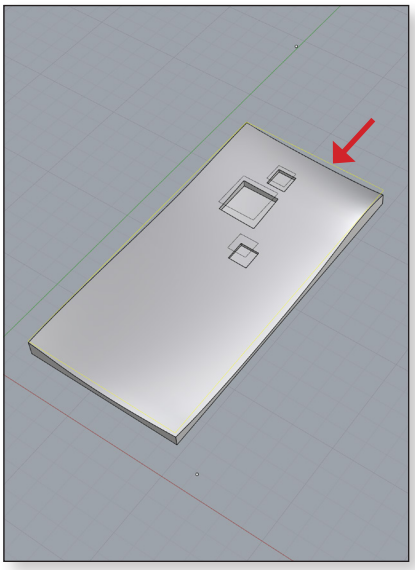
Part 3 Horizontal Roughing

1. Locate the Horizontal Roughing Operation Icon

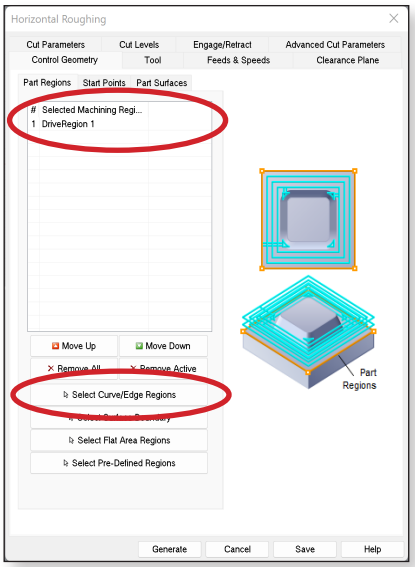
*Under the program tab in the RhinoCAM dialog box, there is a **"Machining Operations"** drop down menu. **"Horizontal Roughing"** can be found under the **"3 Axis"** drop down menu.*



Important: *To establish drive regions, use polylines located in a plane at the top of the stock. One way to create these polylines is to activate the top view, use the command **"Make2D"**, then relocate and align the linework with the top of the stock. It is always necessary to ensure curves are joined together.*



2. Selecting Geometry

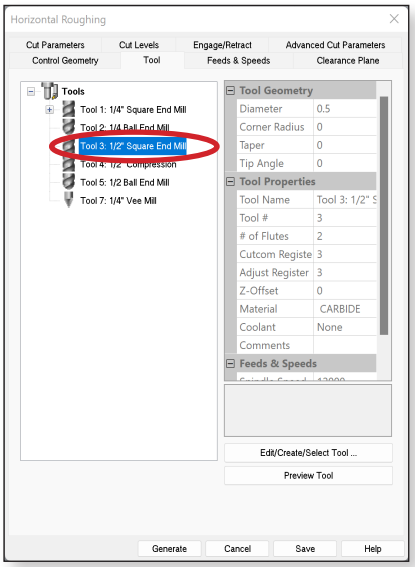


When the horizontal roughing icon is selected, a dialog box will appear with a series of tabs located at the top. ***It is imperative to input information and check settings in every tab.***

The first step is to select the geometry to perform the operation. Click ***"Select Curve/Edge Regions"****; the dialog box will disappear. Select the appropriate geometry, then press enter.*

After the dialog box reappears, it will display what has been selected in the previous step in the list on the left. Once satisfied with the selection, click the ***"Tool"*** *tab at the top to continue to the next step.*

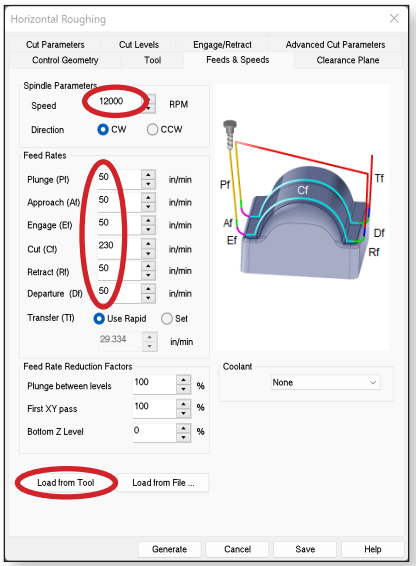
3. Select Tool For Horizontal Roughing Operation



The tool library will appear in the list on the left.

Select the ***1/2" Square End Mill (Tool 3)*** *for this operation. Always use the 1/2" Square End Mill if possible because it affords removing large quantities of material quicker than the 1/4" Square End Mill. Click the* ***"Feeds & Speeds"*** *tab on the top to continue to the next step.*

4. Set Feeds & Speeds



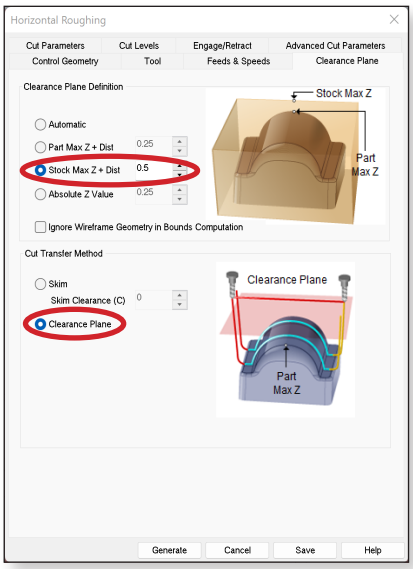
Feeds & speeds correspond to the rate at which the CNC router moves the tool across the part and the RPM (revolutions per minute) that the router bit spins.

Feeds & speeds tables for each tool and material combination are located in a previous section of this guide. Use ***50 in/min*** *for all other feed rates.*

Tool 3: 1/2" Square End Mill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	230	230	140	140
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	2"			

Tip: *Clicking* ***"Load from Tool"*** *will auto populate the 'speed' and all feed rates except 'Cut (CF)'.*

5. Clearance Plane

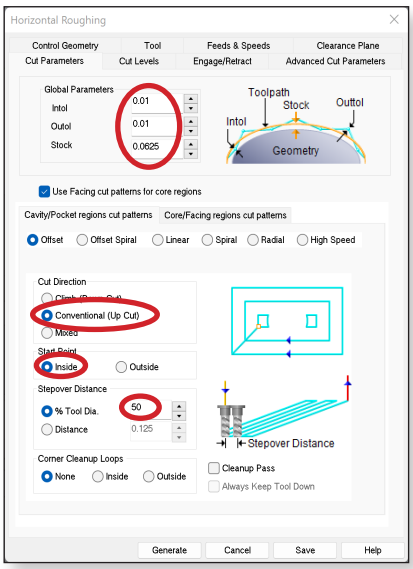


Establishing a clearance plane determines how far the router bit will be away from the stock during transfers. It is important that the clearance plane is always above the stock so that there is no contact during transfer.

*Set the clearance plane **0.5"** above the maximum height (or max Z value) of the stock.*

*Under cut transfer method, select **"Clearance Plane"**.*

6. Set Cut Parameters



*The **"Intol"** and **"Outol"** should be set to **0.01**.*

*The **"Stock"** setting under global parameters is the amount of material left beyond the finished part, or in other words, the amount of material left between the material being removed and final geometry. Set the stock to **0.0625" (1/16")**.*

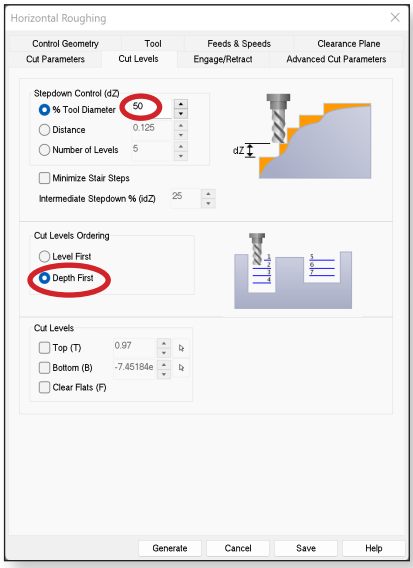
*Select **"Offset"** under the cavity/pocket regions cut patterns.*

*For cut direction, select **"Conventional (Up Cut)"**.*

*Select **"Inside"** under start point.*

*Input the **"Stepover Distance"** as a percentage of the tool diameter. The tables for each tool and material combination are located in a previous section of this guide.*

7. Set Cut Levels



Stepdown controls the depth of material removed with each pass. The tables for each tool and material combination are located in a previous section of this guide.

*Select **"Depth First"** under cut level ordering.*

Max Stepover Distance: Horizontal Roughing & Pocketing				
	Material			
	Foam	MDF	Plywood	Hardwood
Stepover (% of bit diameter)	75	50	35	25

Max Stepdown Distance				
	Material			
	Foam	MDF	Plywood*	Hardwood*
Stepover (% of bit diameter)	75	50	25	25

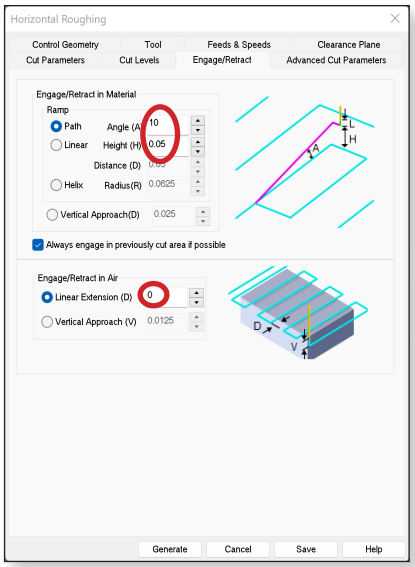
* Tool 4 (1/2 Compression) stepdown percentages can be increased by 25%.

8. Engage/Retract

Engage and retract are parameters used to program a tool reaching a certain depth over a sloped path, rather than directly plunging to the specified depth in a single spot.

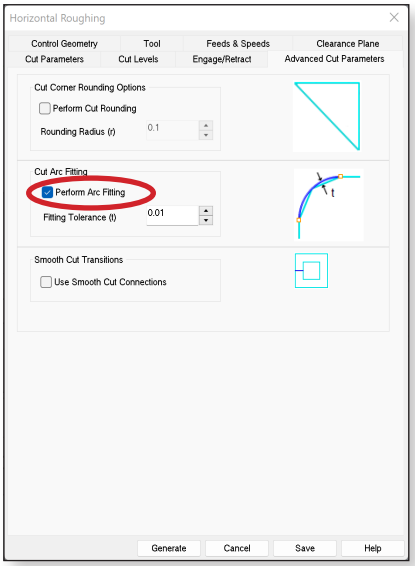
*Ensure **"Path"** is selected with default angle and height values. Set the **"Angle (A)"** to **10** and **"Height (H)"** to **0.05**.*

*Set **"Linear Extension (D)"** to **0**.*



9. Advanced Cut Parameters

Ensure **"Perform Arc Fitting"** *is selected.*

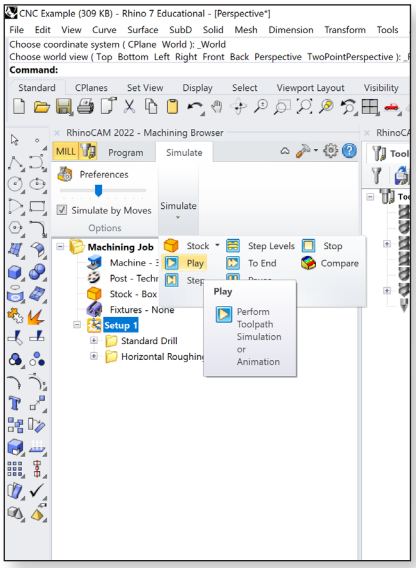


10. Simulate

Once the tool operation is programmed, it can be visually inspected through a virtual simulation to ensure the desired result. To begin a simulation, select the operation to be simulated under the machining job.

*If **"Setup 1"** is selected, all tool operations will be simulated. To start the simulation, select **"Play"** under the simulate tab at the top of the machining browser.*

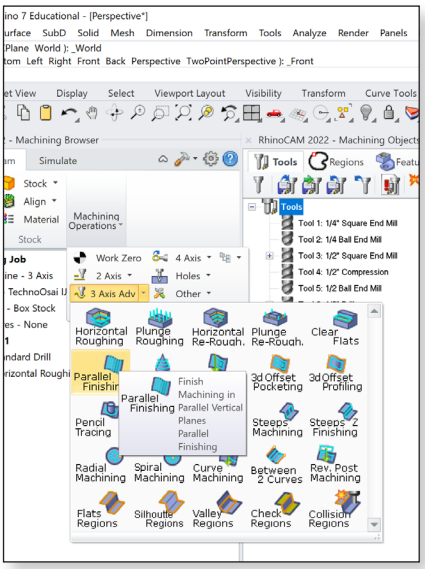
***Tip:** The order of machining operations under setup 1 matters and is organized sequentially. Ensure the machining operation to occur first is located at the top of the list.*



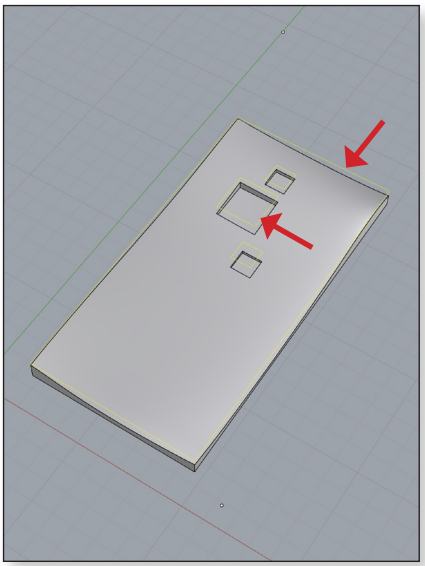
Part 4 Parallel Finishing

1. Locate the Parallel Finishing Operation Icon

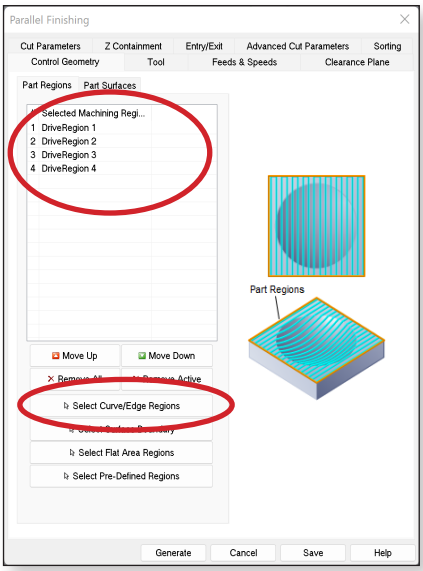
*Under the program tab in the RhinoCAM dialog box, there is a **"Machining Operations"** drop down menu. **"Parallel Finishing"** can be found under the **"3 Axis"** drop down menu.*



Important: *To establish drive regions, use polylines located in a plane at the top of the stock. One way to create these polylines is to activate the top view, use the command **"Make2D"**, then relocate and align the linework with the top of the stock. It is always necessary to ensure curves are joined together.*



2. Selecting Geometry

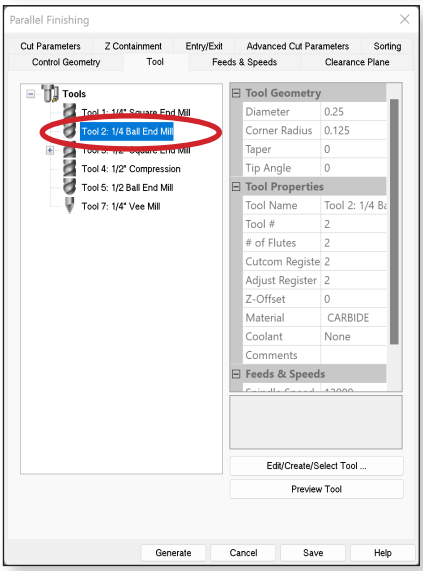


When the parallel finishing icon is selected, a dialog box will appear with a series of tabs located at the top. ***It is imperative to input information and check settings in every tab.***

The first step is to select the geometry to perform the operation. Click ***"Select Curve/Edge Regions"****; the dialog box will disappear. Select the appropriate geometry, then press enter.*

After the dialog box reappears, it will display what has been selected in the previous step in the list on the left. Once satisfied with the selection, click the ***"Tool"*** *tab at the top to continue to the next step.*

3. Select Tool For Parallel Finishing Operation

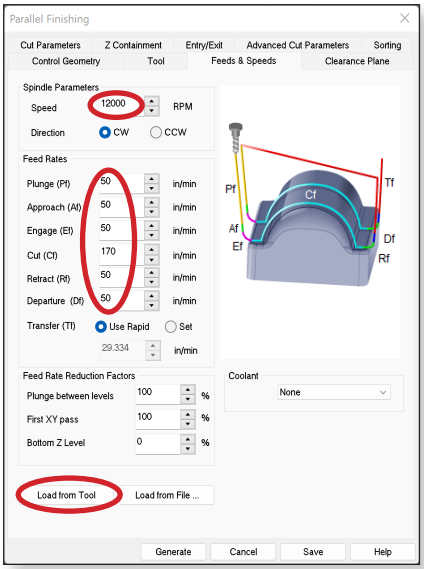


The tool library will appear in the list on the left.

Select the ***1/4" Ball End Mill (Tool 2)*** *or* ***1/2" Ball End Mill (Tool 5)*** *for this operation. Click the* ***"Feeds & Speeds"*** *tab on the top to continue to the next step.*

Tip*: The 1/2" Ball End Mill removes large quantities of material quicker than the 1/4" Ball End Mill, thus saves cutting time.*

4. Set Feeds & Speeds



Feeds & speeds correspond to the rate at which the CNC router moves the tool across the part and the RPM (revolutions per minute) that the router bit spins.

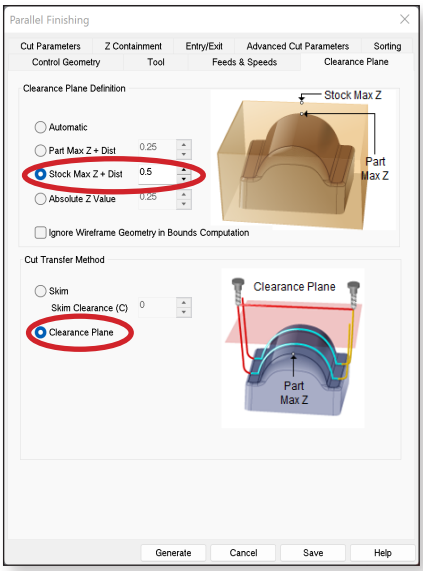
Feeds & speeds tables for each tool and material combination are located in a previous section of this guide. Use ***50 in/min*** *for all other feed rates.*

Tool 2: 1/4" Ball End Mill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	145-190	145-190	120-170	120-170
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	1-1/2"			

Tool 5: 1/2" Ball End Mill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	190-240	190-240	190-220	190-220
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	2-1/2"			

Tip*: Clicking* ***"Load from Tool"*** *will auto populate the 'speed' and all feed rates except 'Cut (CF)'.*

5. Clearance Plane

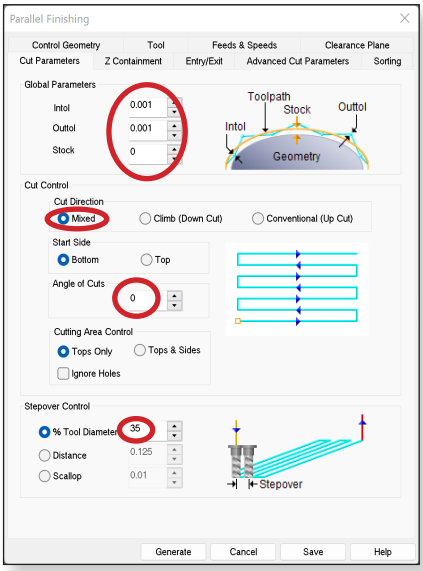


Establishing a clearance plane determines how far the router bit will be away from the stock during transfers. It is important that the clearance plane is always above the stock so that there is no contact during transfer.

*Set the clearance plane **0.5"** above the maximum height (or max Z value) of the stock.*

*Under cut transfer method, select **"Clearance Plane"**.*

6. Set Cut Parameters



*The **"Intol"** and **"Outol"** should be set to **0.001**.*

*The **"Stock"** setting should be set to **0**.*

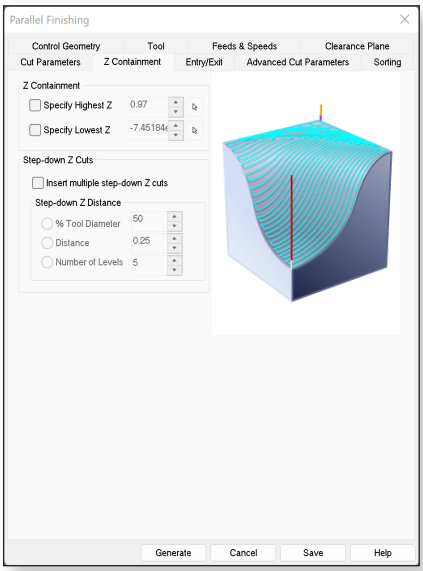
*For cut direction, select **"Mixed"**.*

*The angle of the parallel finishing lines can be change by entering a new value under **" Angle of Cuts"**.*

Input the **"Stepover Distance"** *as a percentage of the tool diameter. The tables for each tool and material combination are located in a previous section of this guide.*

Max Stepover Distance: Parallel Finishing				
	Material			
	Foam	MDF	Plywood	Hardwood
Stepover (% of bit diameter)	75% - 50%	50% - 35%	35% - 25%	25% - 20%

7. Z Containment

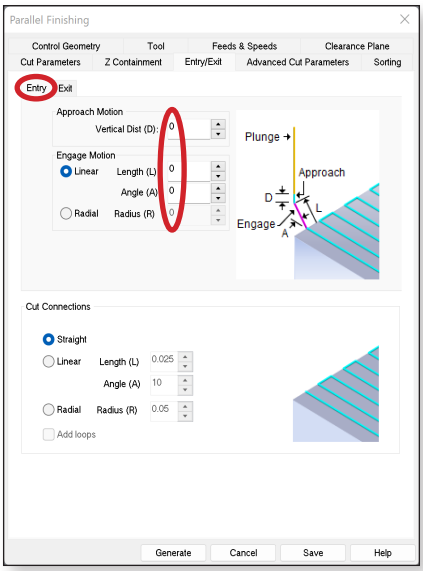


Z Containment is an advance feature used to restrict the level of a cut. Z Containment is typically not applicable.

8. Entry

Entry and exit parameters are used to allow a tool to reach a certain depth (or retract from a certain depth) over a sloped path rather than directly plunging (or withdrawing) in a single spot.

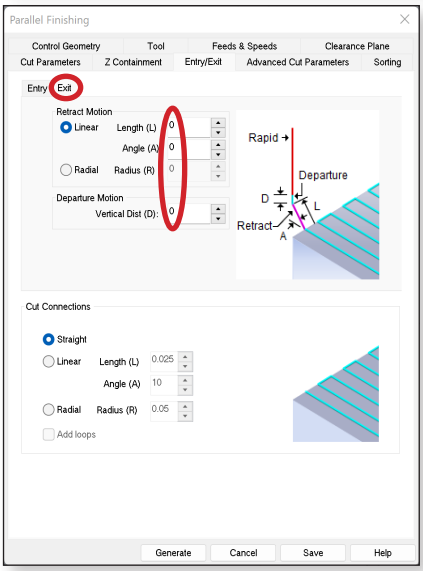
*We recommend these values be set to **"0"** as they have little impact on the type of machining typically done.*



9. Exit

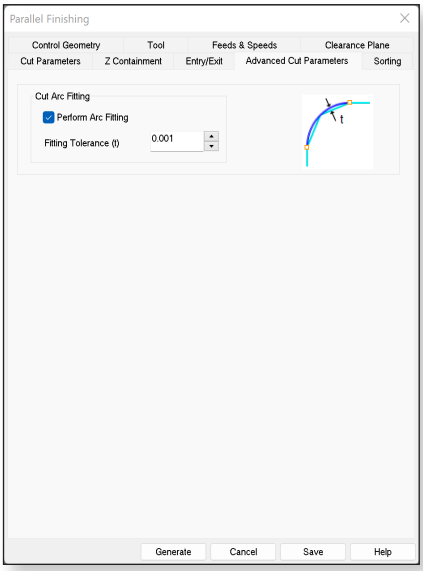
Entry and exit parameters are used to allow a tool to reach a certain depth (or retract from a certain depth) over a sloped path rather than directly plunging (or withdrawing) in a single spot.

*We recommend these values be set to **"0"** as they have little impact on the type of machining typically done.*



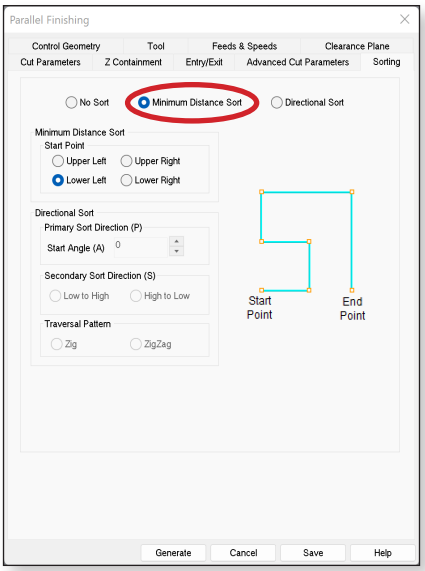
10. Advanced Cut Parameters

Ensure **"Perform Arc Fitting"** *is checked.*



11. Set Sorting

*Sorting establishes the order that parallel finishing operations are completed. To optimize job time, it is recommended to set the sorting to **"Minimum Distance Sort"**.*

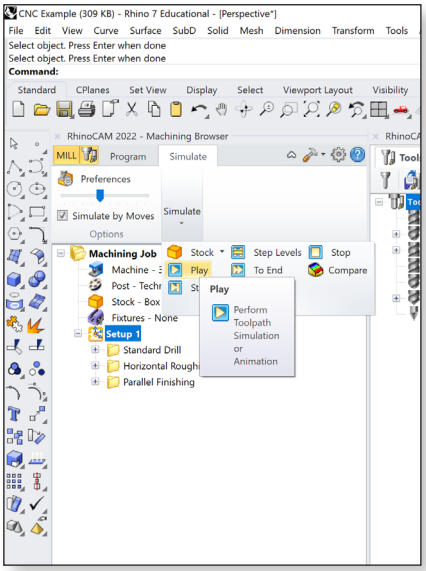


12. Simulate

Once the tool operation is programmed, it can be visually inspected through a virtual simulation to ensure the desired result. To begin a simulation, select the operation to be simulated under the machining job.

*If **"Setup 1"** is selected, all tool operations will be simulated. To start the simulation, select **"Play"** under the simulate tab at the top of the machining browser.*

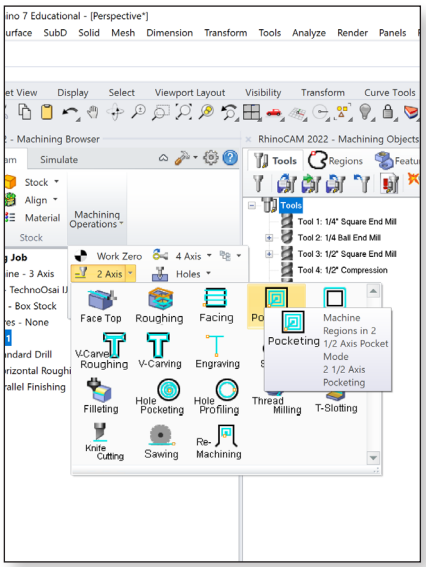
***Tip:** The order of machining operations under setup 1 matters and is organized sequentially. Ensure the machining operation to occur first is located at the top of the list.*



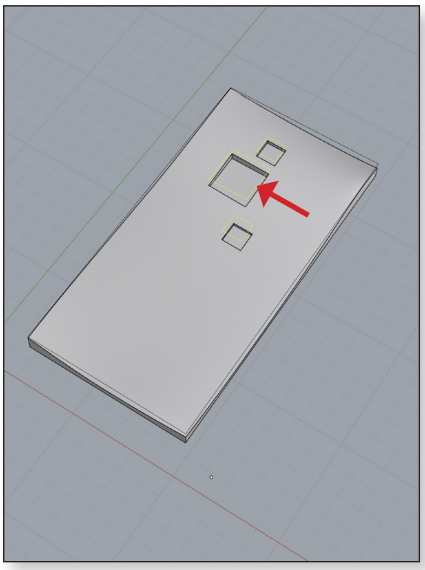
Part 5 Pocketing

1. Locate the Pocketing Operation Icon

*Under the program tab in the RhinoCAM dialog box, there is a **"Machining Operations"** drop down menu. **"Pocketing"** can be found under the **"2 Axis"** drop down menu.*



Important: *To establish drive regions, use polylines located in a plane at the top of the stock. One way to create these polylines is to activate the top view, use the command **"Make2D"**, then relocate and align the linework with the top of the stock. It is always necessary to ensure curves are joined together.*

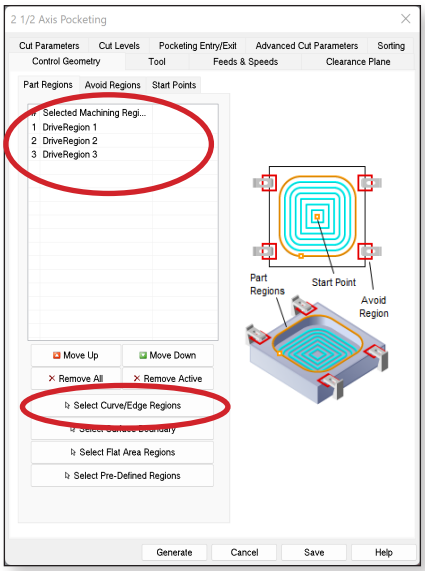


2. Selecting Geometry

When the pocketing icon is selected, a dialog box will appear with a series of tabs located at the top. ***It is imperative to input information and check settings in every tab.***

The first step is to select the geometry to perform the operation. Click ***"Select Curve/Edge Regions"****; the dialog box will disappear. Select the appropriate geometry, then press enter.*

After the dialog box reappears, it will display what has been selected in the previous step in the list on the left. Once satisfied with the selection, click the ***"Tool"*** *tab at the top to continue to the next step.*

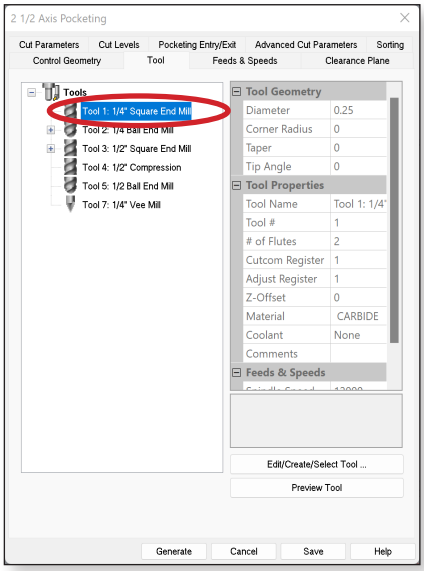


3. Select Tool For Pocketing Operation

The tool library will appear in the list on the left.

Select the ***1/4" Square End Mill (Tool 1)*** *or* ***1/2" Square End Mill (Tool 3)*** *for this operation. Click the* ***"Feeds & Speeds"*** *tab on the top to continue to the next step.*

Tip: *The 1/2" Ball End Mill removes large quantities of material quicker than the 1/4" Ball End Mill, thus saves cutting time.*



4. Set Feeds & Speeds

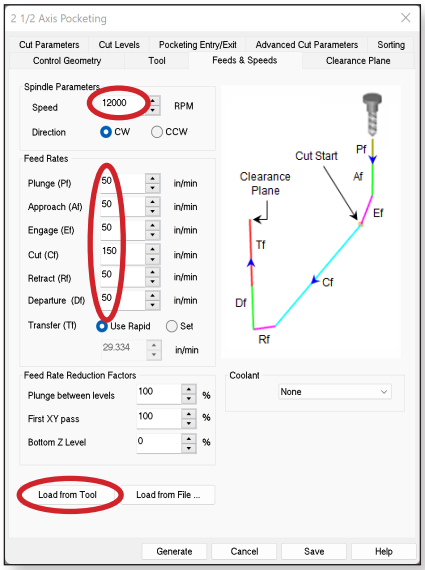
Feeds & speeds correspond to the rate at which the CNC router moves the tool across the part and the RPM (revolutions per minute) that the router bit spins.

Feeds & speeds tables for each tool and material combination are located in a previous section of this guide. Use ***50 in/min*** *for all other feed rates.*

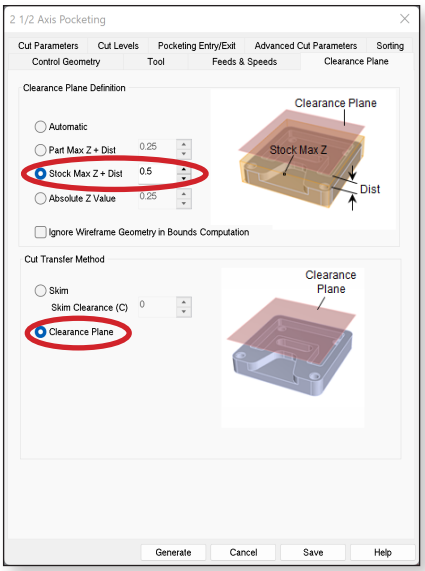
Tool 1: 1/4" Square End Mill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	150	150	120	120
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	2"			

Tool 3: 1/2" Square End Mill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	230	230	140	140
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	2"			

Tip: *Clicking* ***"Load from Tool"*** *will auto populate the 'speed' and all feed rates except 'Cut (CF)'.*



5. Clearance Plane

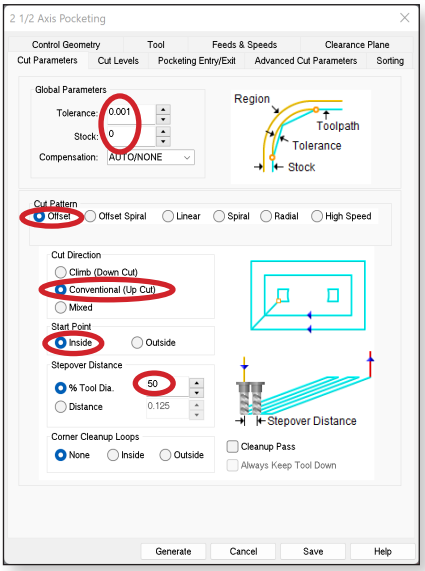


Establishing a clearance plane determines how far the router bit will be away from the stock during transfers. It is important that the clearance plane is always above the stock so that there is no contact during transfer.

*Set the clearance plane **0.5"** above the maximum height (or max Z value) of the stock.*

*Under cut transfer method, select **"Clearance Plane"**.*

6. Set Cut Parameters



*The **"Tolerance"** should be set to **0.001**.*

*The **"Stock"** setting should be set to **0**.*

*Select **"Offset"** under cut pattern.*

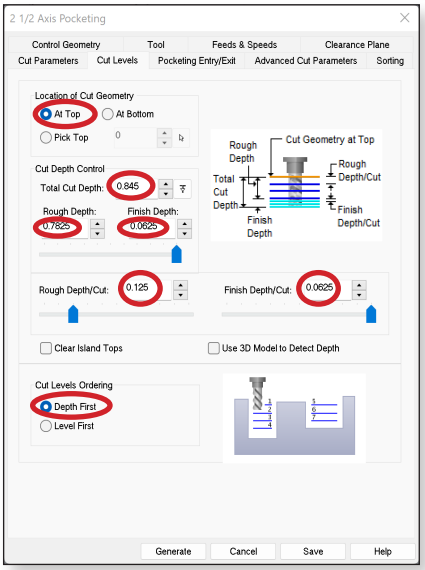
*For cut direction, select **"Conventional (Up Cut)"**.*

*Select **"Inside"** under start point.*

*Input the **"Stepover Distance"** as a percentage of the tool diameter. The tables for each tool and material combination are located in a previous section of this guide.*

Max Stepover Distance: Horizontal Roughing & Pocketing				
	Material			
	Foam	MDF	Plywood	Hardwood
Stepover (% of bit diameter)	75	50	35	25

7. Set Cut Levels



*Always locate cut geometry at the **"top"** of the part.*

***"Total Cut Depth"** specifies the total overall desired depth of a pocket. This is divided into two parts: **"Rough Depth"** and **"Finish Depth"**. Input **"Finish Depth"** as **1/16" (0.0625")**; **"Rough Depth"** will auto populate.*

***"Rough Depth/Cut"** and **"Finish Depth/Cut"** specify the depth of each cut level. The maximum stepdown tables for each tool and material combination are located in a previous section of this guide. (If using a compression bit, the minimum rough depth per cut must be **0.25"**.)*

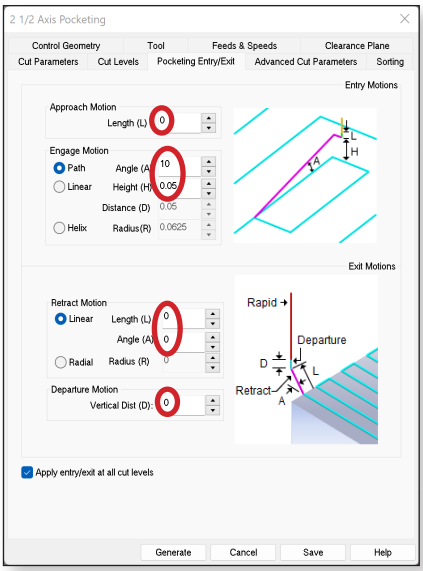
*Select **"Depth First"** under cut level ordering.*

Max Stepdown Distance				
	Material			
	Foam	MDF	Plywood*	Hardwood*
Stepover (% of bit diameter)	75	50	25	25

* Tool 4 (1/2 Compression) stepdown percentages can be increased by 25%.

Max Stepdown Calculation: *50% of 1/4" bit diameter = 1/8" or 0.125"*

8. Pocketing Entry/Exit



Engage and retract are parameters used to program a tool reaching a certain depth over a sloped path, rather than directly plunging to the specified depth in a single spot.

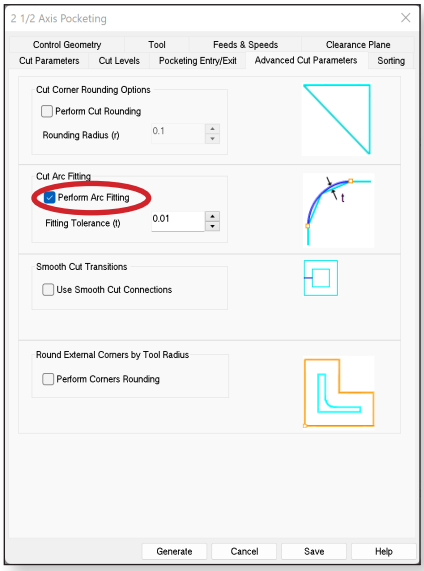
*Set **"Approach Motion Length (L)"** to **0**.*

*Ensure **"Path"** is selected with default angle and height values. Set the **"Angle (A)"** to **10** and **"Height (H)"** to **0.05**.*

*Ensure all **"Retract Motion Values"** are set to **0**.*

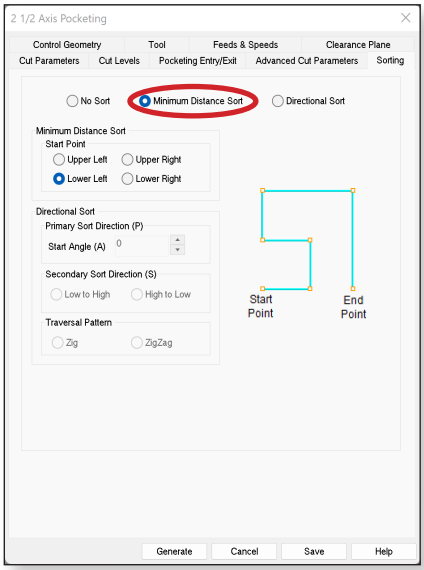
*Check **"Apply entry/exit at all cut levels"**.*

9. Advanced Cut Parameters



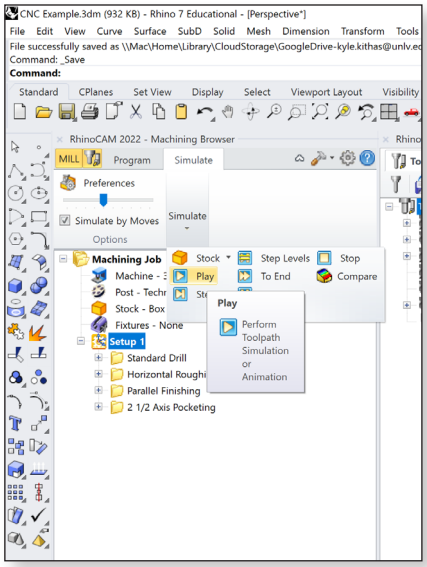
Ensure **"Perform Arc Fitting"** *is selected.*

10. Set Sorting



*Sorting establishes the order that pocketing operations are completed. To optimize job time, it is recommended to set the sorting to **"Minimum Distance Sort"**.*

11. Simulate



Once the tool operation is programmed, it can be visually inspected through a virtual simulation to ensure the desired result. To begin a simulation, select the operation to be simulated under the machining job.

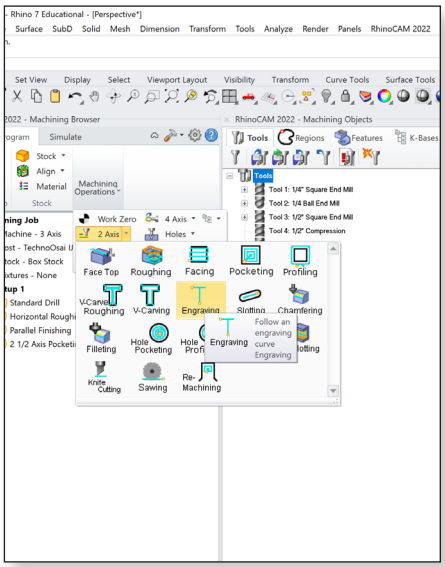
*If **"Setup 1"** is selected, all tool operations will be simulated. To start the simulation, select **"Play"** under the simulate tab at the top of the machining browser.*

***Tip:** The order of machining operations under setup 1 matters and is organized sequentially. Ensure the machining operation to occur first is located at the top of the list.*

Part 6 Engraving

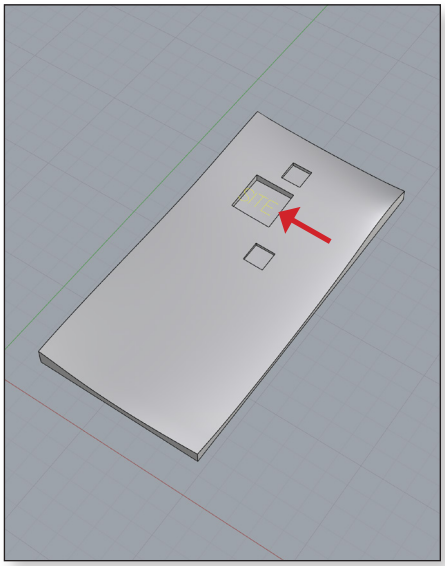
1. Locate the Engraving Operation Icon

*Under the program tab in the RhinoCAM dialog box, there is a **"Machining Operations"** drop down menu. **"Engraving"** can be found under the **"2 Axis"** drop down menu.*

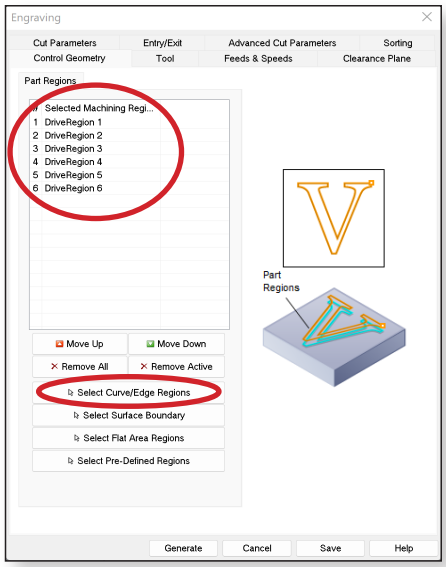


***Important**: To establish drive regions, use polylines located in a plane at the top of the stock. One way to create these polylines is to activate the top view, use the command **"Make2D"**, then relocate and align the linework with the top of the stock. It is always necessary to ensure curves are joined together.*

Tip**: Text can be created using the **"Text"** command in Rhino. The recommended font to use is **"SLF-RHN Architect".



2. Selecting Geometry

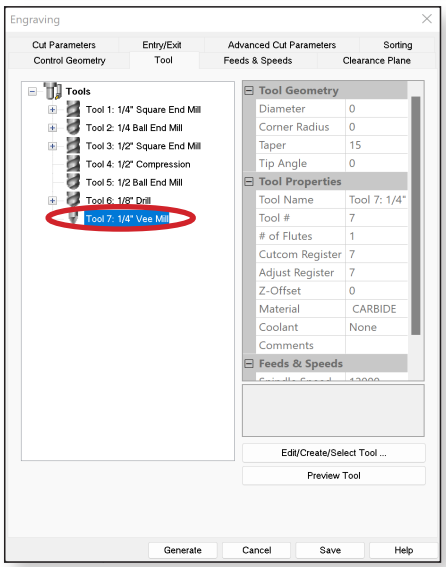


When the engraving icon is selected, a dialog box will appear with a series of tabs located at the top. ***It is imperative to input information and check settings in every tab.***

The first step will be to select the geometry to perform the operation. Click ***"Select Curve/Edge Regions"****; the dialog box will disappear. Select the appropriate geometry, then press enter.*

After the dialog box reappears, it will display what has been selected in the previous step in the list on the left. Once satisfied with the selection, click the ***"Tool"*** *tab at the top to continue to the next step.*

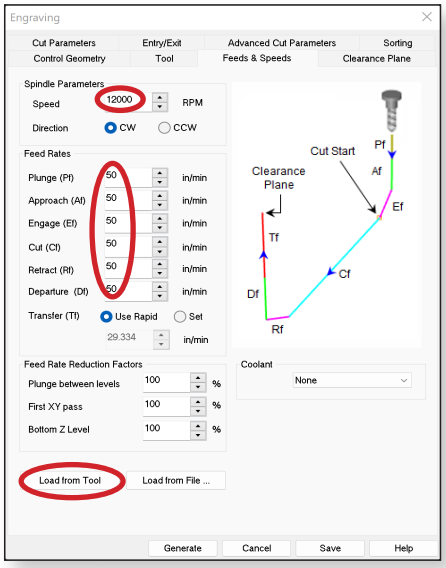
3. Select Tool For Engraving Operation



The tool library will appear in the list on the left.

Select the ***1/4" Vee Mill (tool 7)****.*

4. Set Feeds & Speeds



Feeds & speeds correspond to the rate at which the CNC router moves the tool across the part and the RPM (revolutions per minute) that the router bit spins.

Feeds & speeds tables for each tool and material combination are located in a previous section of this guide. Use ***50 in/min*** *for all other feed rates.*

Tool 7: 1/4" Vee Mill				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	80	50	30	30
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	3/8"			

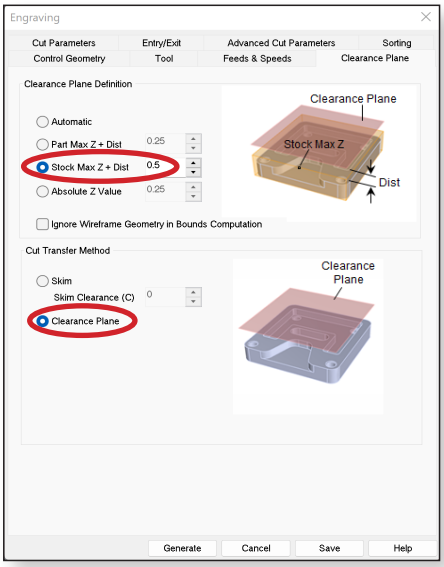
Tip: *Clicking* ***"Load from Tool"*** *will auto populate the 'speed' and all feed rates except 'Cut (CF)'.*

5. Clearance Plane

Establishing a clearance plane determines how far the router bit will be away from the stock during transfers. It is important that the clearance plane is always above the stock so that there is no contact during transfer.

*Set the clearance plane **0.5"** above the maximum height (or max Z value) of the stock.*

*Under cut transfer method, select **"Clearance Plane"**.*



6. Set Cut Parameters

*The **"Tolerance"** should be set to **0.001**.*

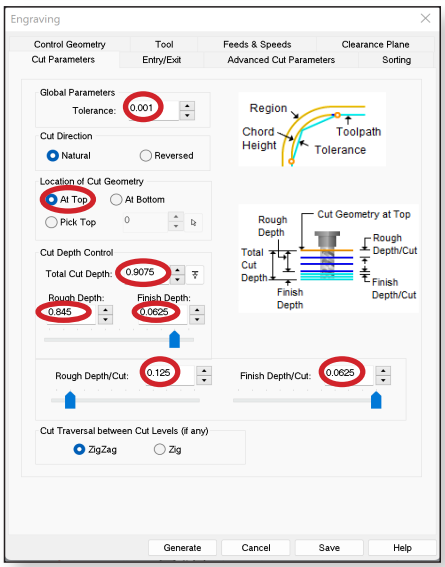
*Always locate cut geometry at the **"top"** of the part.*

***"Total Cut Depth"** specifies the total overall desired depth of an engraving. This is divided into two parts: **"Rough Depth"** and **"Finish Depth"**. Input "**Finish Depth"** as **1/16" (0.0625")**; **"Rough Depth"** will auto populate.*

***"Rough Depth/Cut"** and **"Finish Depth/Cut"** specify the depth of each cut level. The maximum stepdown tables for each tool and material combination are located in a previous section of this guide.*

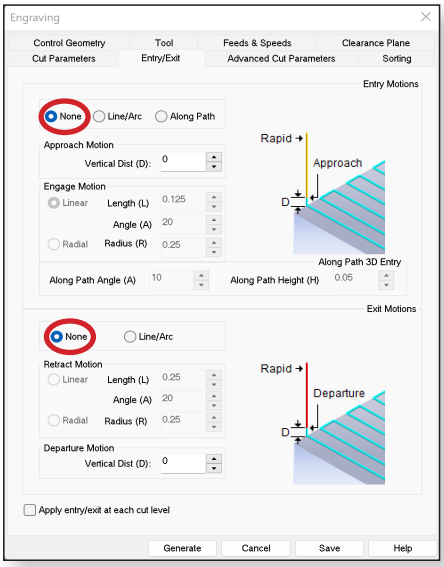
Max Stepdown Distance				
	Material			
	Foam	MDF	Plywood*	Hardwood*
Stepover (% of bit diameter)	75	50	25	25

***Max Stepdown Calculation:** 50% of 1/4" bit diameter = 1/8" or 0.125"*



7. Entry/Exit

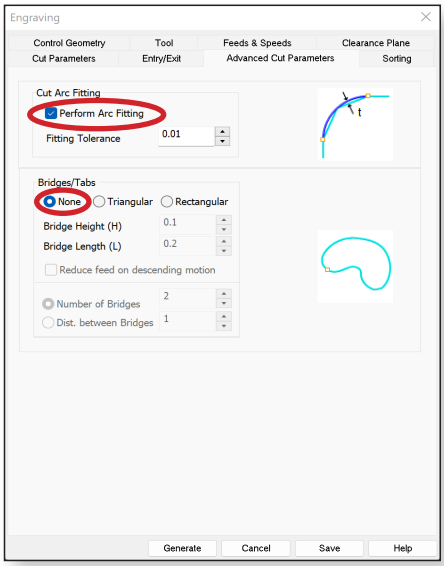
*Select **"None"** for both entry motions and exit motions.*



8. Advanced Cut Parameters

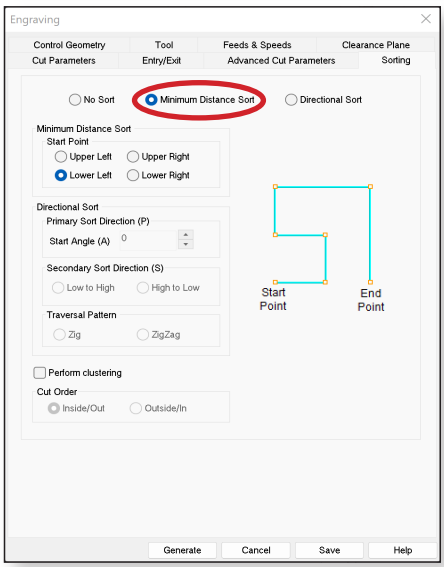
*Ensure **"Perform Arc Fitting"** is selected.*

*Select **"None"** for Bridges/Tabs.*



9. Set Sorting

*Sorting establishes the order that engraving operations are completed. To optimize job time, it is recommended to set the sorting to **"Minimum Distance Sort"**.*

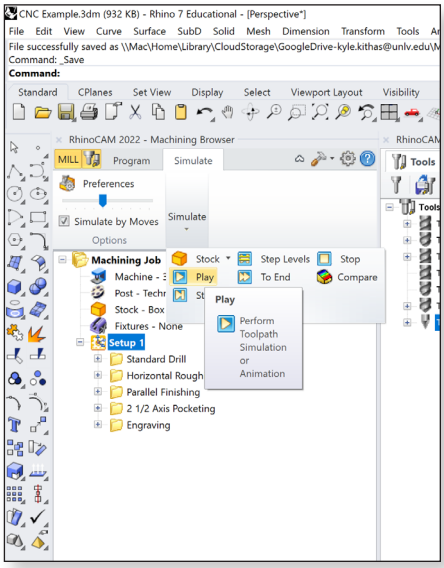


10. Simulate

Once the tool operation is programmed, it can be visually inspected through a virtual simulation to ensure the desired result. To begin a simulation, select the operation to be simulated under the machining job.

*If **"Setup 1"** is selected, all tool operations will be simulated. To start the simulation, select **"Play"** under the simulate tab at the top of the machining browser.*

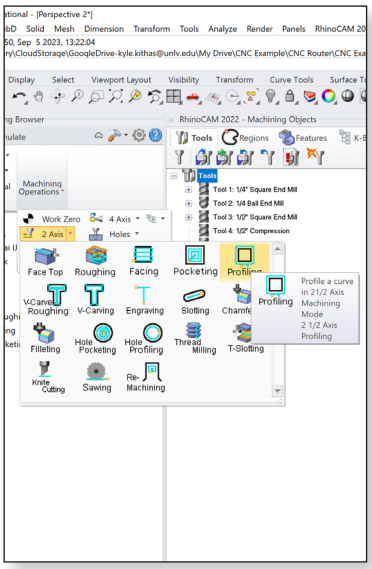
***Tip:** The order of machining operations under setup 1 matters and is organized sequentially. Ensure the machining operation to occur first is located at the top of the list.*



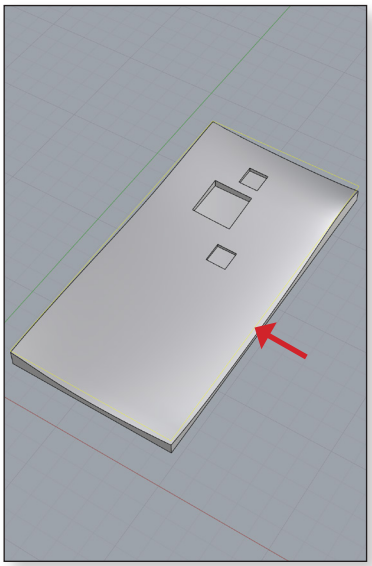
Part 7 Profiling

1. Locate the Profiling Operation Icon

*Under the program tab in the RhinoCAM dialog box, there is a **"Machining Operations"** drop down menu. **"Profiling"** can be found under the **"2 Axis"** drop down menu.*



***Important**: To establish drive regions, use polylines located in a plane at the top of the stock. One way to create these polylines is to activate the top view, use the command **"Make2D"**, then relocate and align the linework with the top of the stock. It is always necessary to ensure curves are joined together.*



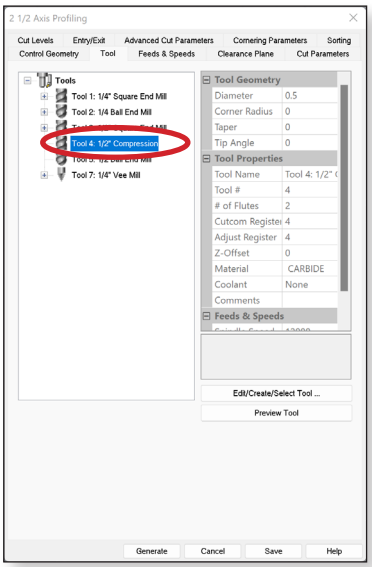
2. Selecting Geometry

When the profiling icon is selected, a dialog box will appear with a series of tabs located at the top. ***It is imperative to input information and check settings in every tab.***

The first step will be to select the geometry to perform the operation. Click ***"Select Curve/Edge Regions"****; the dialog box will disappear. Select the appropriate geometry, then press enter.*

After the dialog box reappears, it will display what has been selected in the previous step in the list on the left. Once satisfied with the selection, click the ***"Tool"*** *tab at the top to continue to the next step.*

3. Select Tool For Profiling Operation



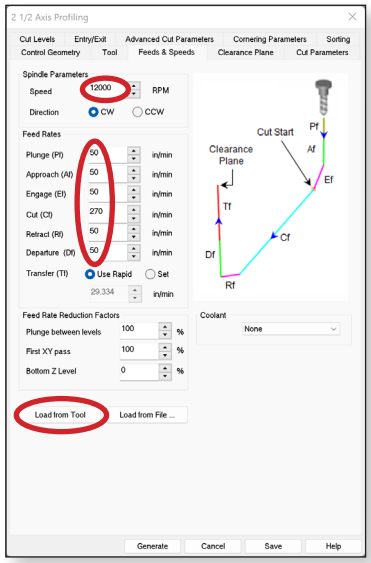
The tool library will appear in the list on the left.

Select the ***1/2" compression bit (tool 4)*** *if cutting completely through the material.*

If any of the following are true, the 1/4" or 1/2" square end mills (tools 1 and 3) should be used for profiling:

1. *The material is less than* ***0.25"*** *thick*
2. *The depth of profile cut is less than* ***0.25"***
3. *Width of profile cut needs to be* ***0.25"***

4. Set Feeds & Speeds



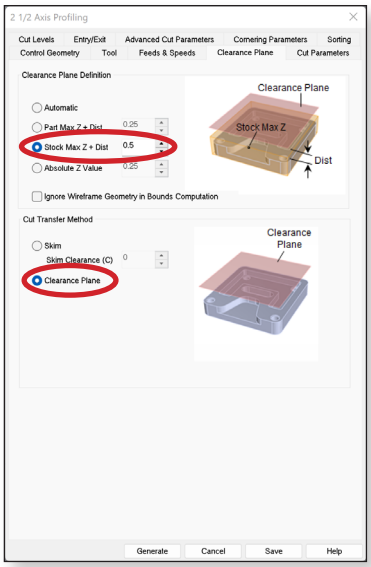
Feeds & speeds correspond to the rate at which the CNC router moves the tool across the part and the RPM (revolutions per minute) that the router bit spins.

Feeds & speeds tables for each tool and material combination are located in a previous section of this guide. Use ***50 in/min*** *for all other feed rates.*

Tool 4: 1/2" Compression				
	Material			
	Foam	MDF	Plywood	Hardwood
Cut Feed Rate (IPM)	270	270	190	190
All other Feed Rates (IPM)	50			
Speed (RPM)	12,000			
Max Cut Depth	2-1/2"			

Tip: *Clicking* ***"Load from Tool"*** *will auto populate the 'speed' and all feed rates except 'Cut (CF)'.*

5. Clearance Plane

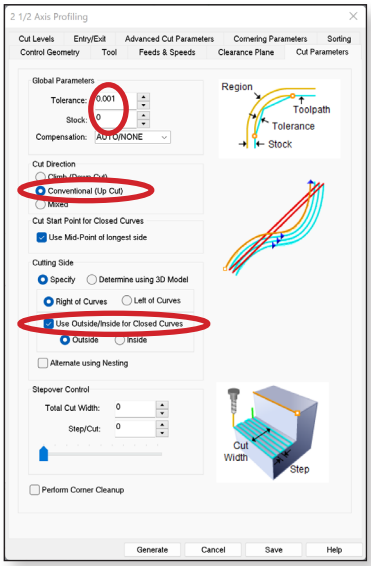


Establishing a clearance plane determines how far the router bit will be away from the stock during transfers. It is important that the clearance plane is always above the stock so that there is no contact during transfer.

*Set the clearance plane **0.5"** above the maximum height (or max Z value) of the stock.*

*Under cut transfer method, select **"Clearance Plane"**.*

6. Set Cut Parameters



*The **"Tolerance"** should be set to **0.001**.*

*The **"Stock"** setting should be set to **0**.*

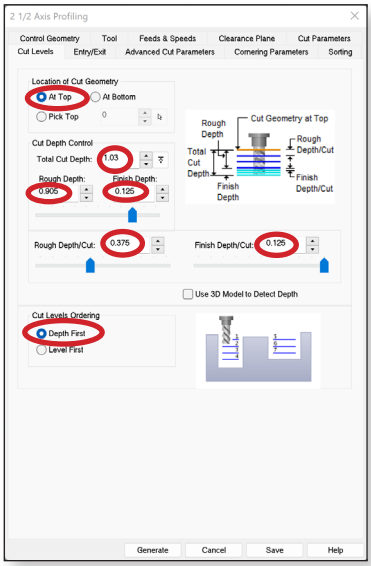
*For cut direction, select **"Conventional (Up Cut)"**.*

*Select **"Outside"** or **"Inside"**. To cut out parts, select "Outside". To cut out holes or voids within parts, select "Inside".*

*For open curves, ensure **"Use Outside/Inside for Closed Curves"** is not selected and select **"Right"** or **"Left"** side.*

*For programming organization, it is advised to keep profile cuts the same width as the tool diameter. Therefore, both **"Total Cut Width"** and **"Step/Cut"** should be set to **"0"**. If a cut is desired to have a greater width than the tool diameter, it is recommended to use a pocketing operation.*

7. Set Cut Levels



*Always locate cut geometry at the **"top"** of the part.*

***"Total Cut Depth"** specifies the total overall desired profile depth. This is divided into two parts: **"Rough Depth"** and **"Finish Depth"**. Input **"Finish Depth"** as **1/8" (0.125")**; **"Rough Depth"** will auto populate.*

***Tip:** If the intent is to cut completely through the stock, it is advisable to add **0.03"** to the total cut depth to account for variations in the stock. In other words, if the stock was 1" thick, input the total cut depth as **1.03"**.*

***"Rough Depth/Cut"** and **"Finish Depth/Cut"** specify the depth of each cut level. The minimum **"Rough Depth/Cut"** must be **0.25"** for a compression bit.*

*Select **"Depth First"** under cut level ordering.*

Max Stepdown Distance				
	Material			
	Foam	MDF	Plywood*	Hardwood*
Stepover (% of bit diameter)	75	50	25	25

* Tool 4 (1/2 Compression) stepdown percentages can be increased by 25%.

***Max Stepdown Calculation:** 75% of 1/2" bit diameter = 3/8" or 0.375"*

8. Entry/Exit

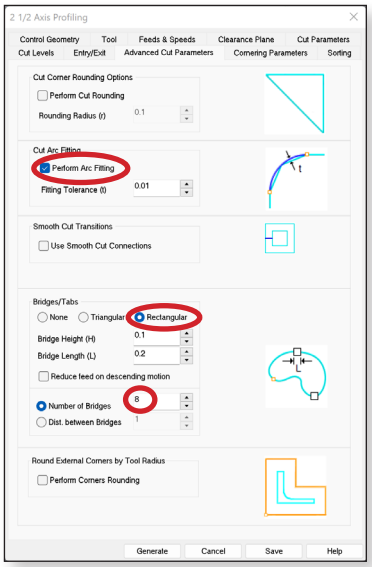
*Select **"None"** for both entry motions and exit motions . Ensure **"Apply entry/exit at each cut level"** is unchecked.*

9. Advanced Cut Parameters

*Ensure **"Perform Arc Fitting"** is selected.*

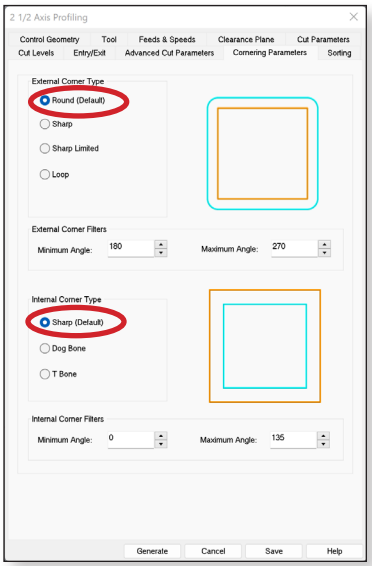
***Bridging** is required for profile cuts. Bridges retain the connection between the stock and piece being milled. The purpose of bridging is to ensure parts don't move during the milling process.*

*A minimum of **(4)** bridges per side are required, but more bridges are advisable if the part is very large.*

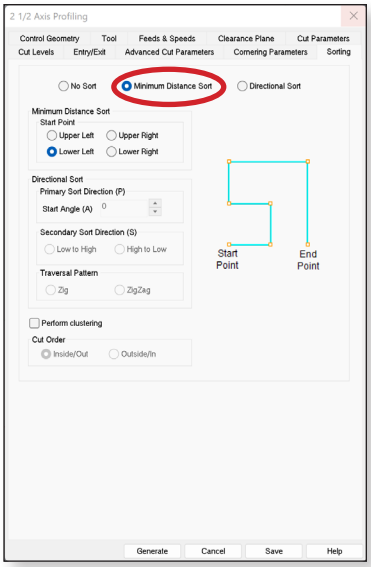


10. Cornering Parameters

The defaults are recommended for corner parameters.

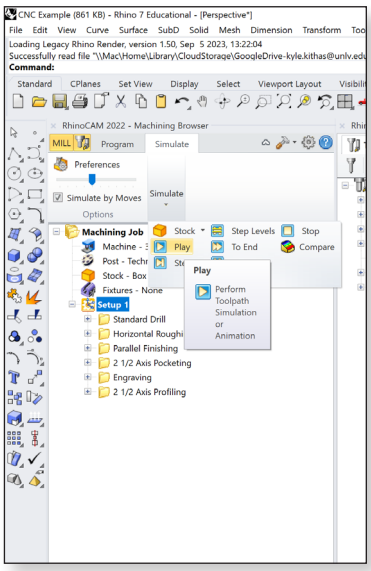


11. Set Sorting



Sorting establishes the order that pocketing operations are completed. To optimize job time, it is recommended to set the sorting to ***"Minimum Distance Sort"****.*

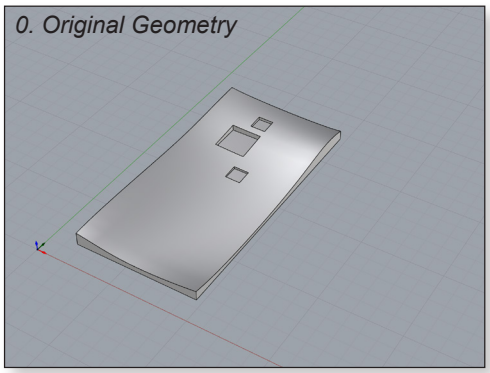
12. Simulate

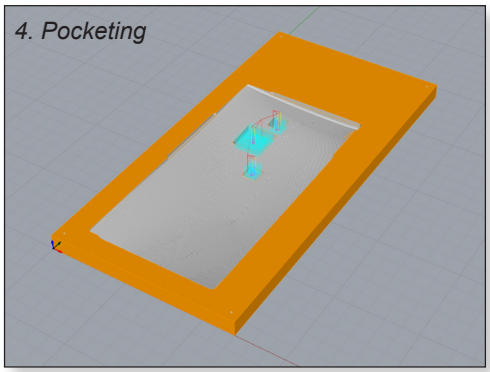


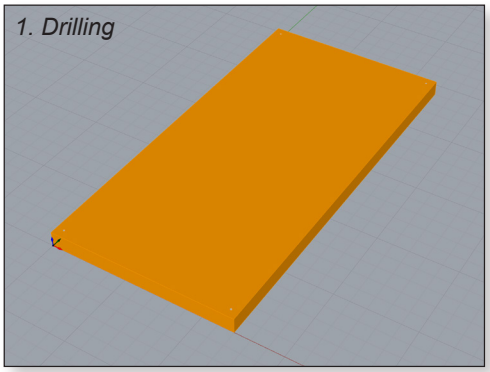
If ***"Setup 1"*** *is selected, all tool operations will be simulated. To start the simulation, select* ***"Play"*** *under the simulate tab at the top of the machining browser.*

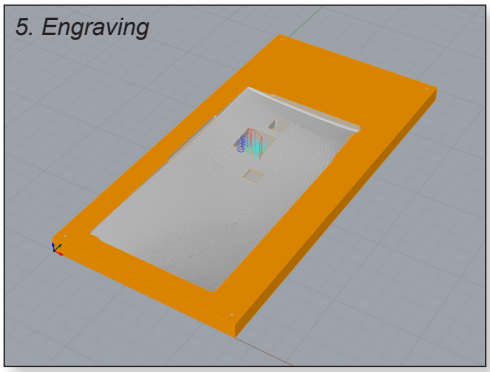
Tip*: The order of machining operations under setup 1 matters and is organized sequentially. Ensure the machining operation to occur first is located at the top of the list.*

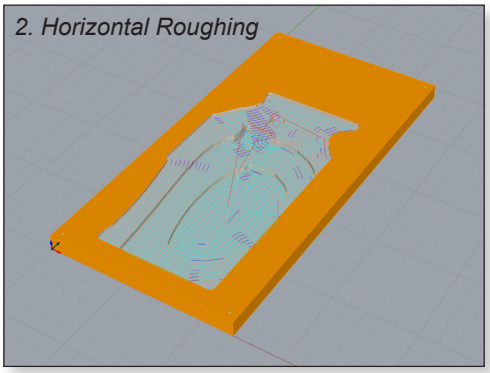
Part 8 Simulation Overview

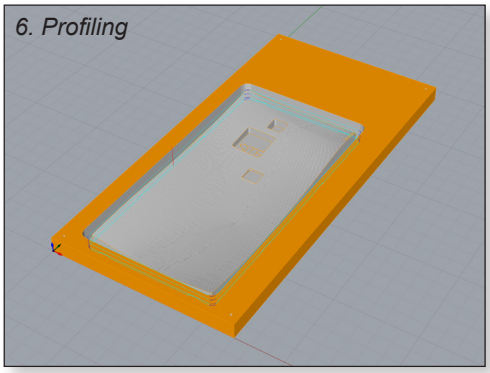


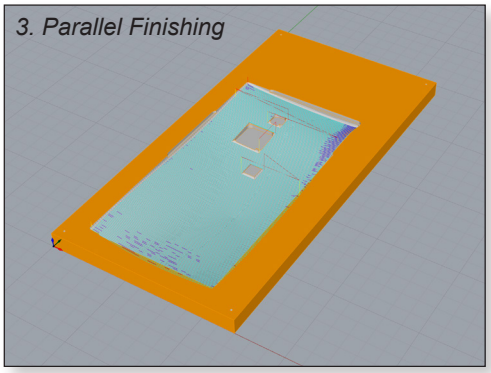








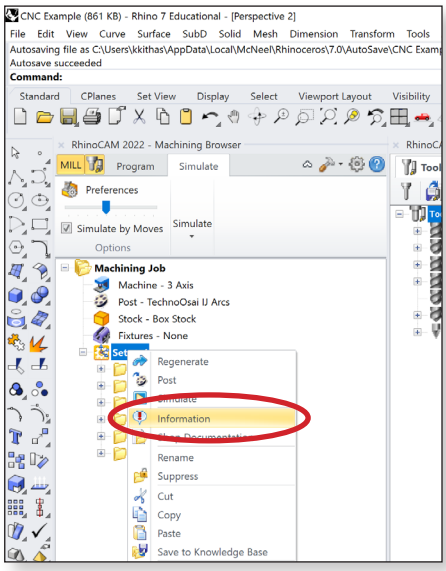




Part 9 Estimating Time & Posting

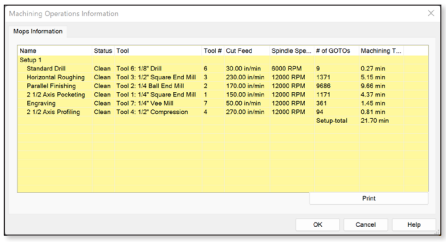
1. Information

Once the tool operations are programmed, right click “Setup 1” and select information.



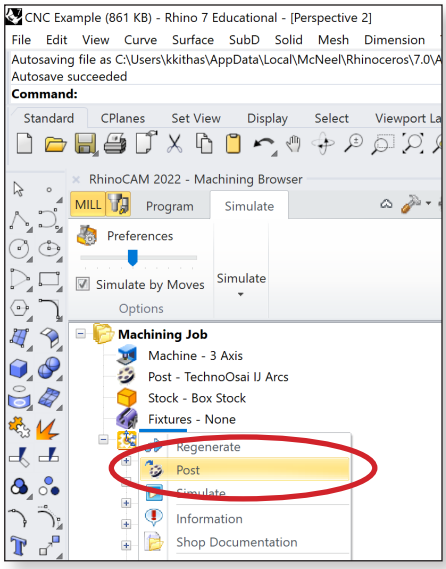
2. Machining Operations Information

Selecting information will display a dialog box showing each tool operation, the tool required for that operation, and machine time. The total machining time will also be displayed.



3. Post

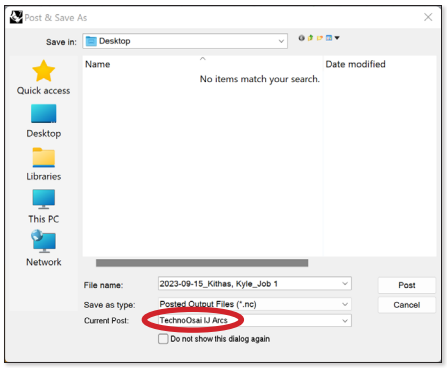
*The job can be **"posted"**, or converted to G-code, by right clicking **"Setup 1"** and selecting **"Post"**.*



4. Saving Post

Save the posted file in the following format:
[year-mm-dd]_[last name, first name]_[job #]

*Ensure the **"TechnoOsai IJ Arcs"** is listed as the current post.*



5. Post

After the file is posted, it will automatically open in Notepad.