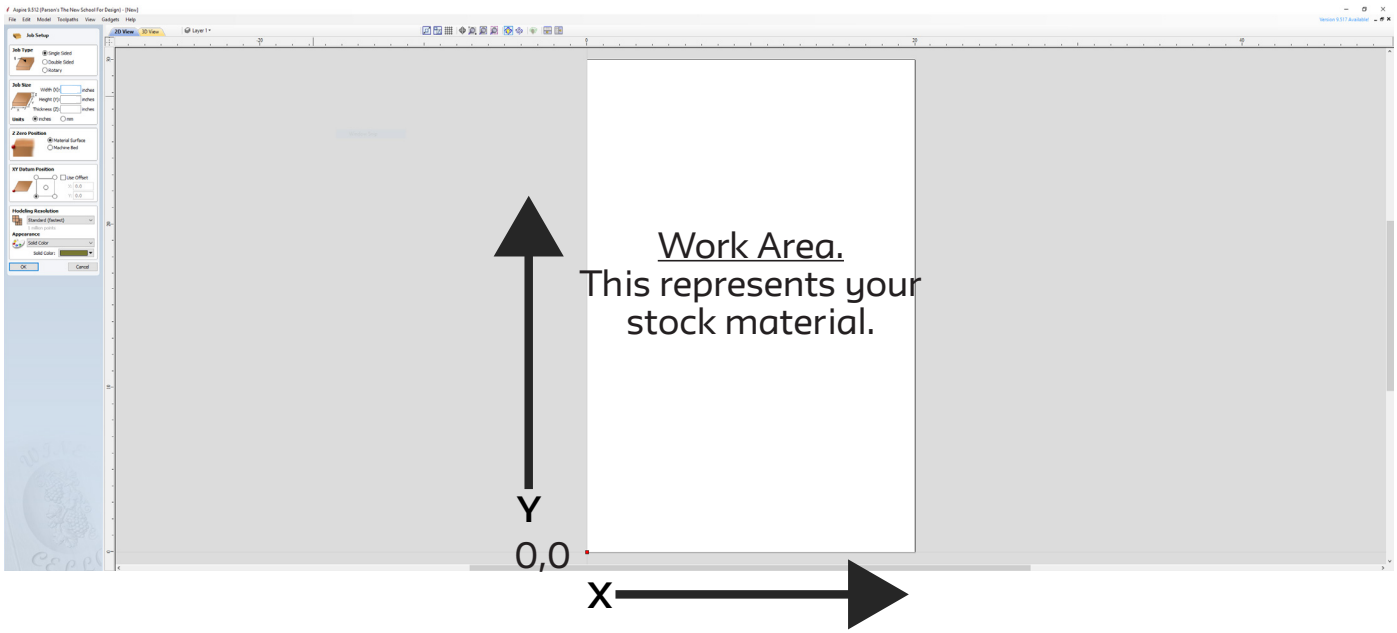
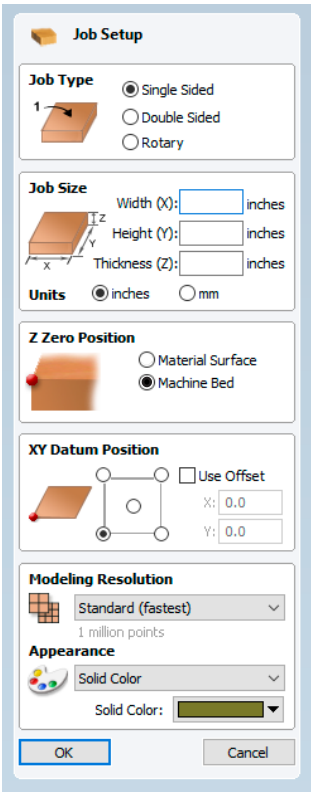
SETTING UP ASPIRE 9.5



Step 1: Filling out the Job Setup



Job Type

- Select "Single Sided"
  -Double Sided & Rotary are not supported

Job Size*

- X/Y Should be measured with a tape measure to the nearest 1/16th inch
- Z needs to be measured with a caliper to the nearest 0.001 inch (Use the **Thickest** measure of all four sides)

*note - The CNC machines only understand inches
DO NOT switch to millimeters

Z Zero Position

- Generally "Machine Bed" should be selected.
- The Technician will verify this.

XY Datum Position can remain as default setting.
Modeling Resolution can remain as default setting.

Select "**OK**" when finished.

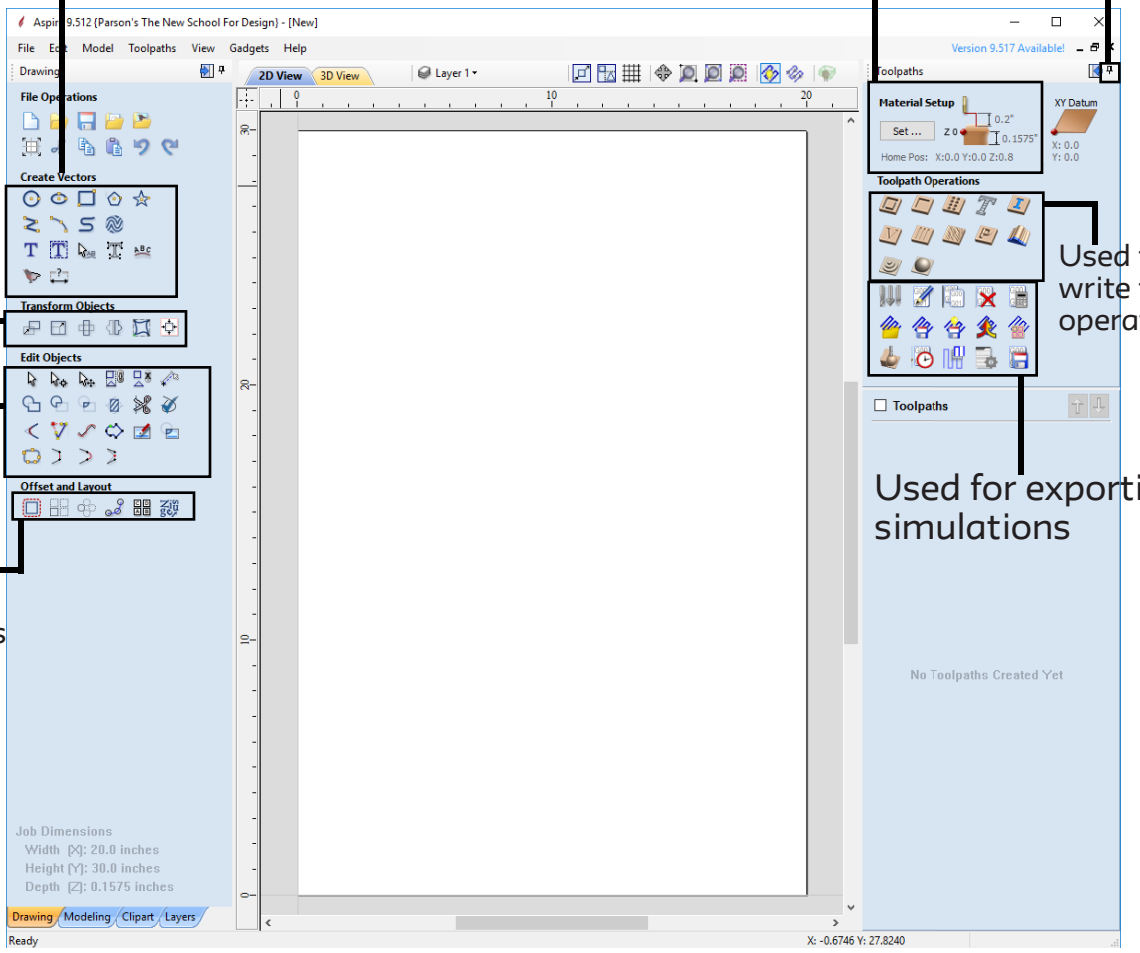
*X/Y has a max limit depending on machine.
Z has a max depending on the type of material, but should be set to your material's thickness exactly.

QUICK REFERENCE & TIPS

Used to define material positions and Z safety moves.

Clicking this will prevent the toolpath menu from closing.

Used to create/edit/manipulate vectors.
We recommend using CAD or Illustrator to create vectors.



Used to scale & transform vectors

Used to move/select group/trim vectors

Used to offest & array vectors

Used to write the CAM operations

Used for exporting & simulations

THINGS TO CONSIDER

Stock Measurement Vs. Nominal Measurement

Nominal Measurment is what hardware/lumber yards use as the average across the stock. This makes up for tolerances through the manufacturing process.

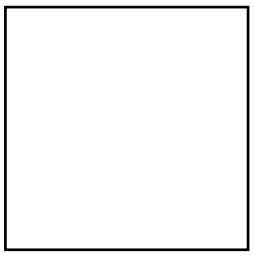
Stock Measurment is the true measurement of the material you will be using. With laminate material this can change from brand to brand. This is very important when you want to machine 3D files.

Ex. a 2”x4“ piece of lumber is a nominal measurement. Its stock measurement is generally closer to 1.5” x 3.5”

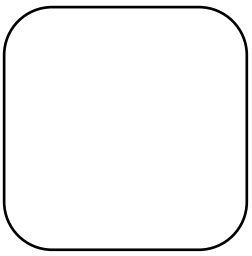
Internal Radii

This is what an **INSIDE** corner will look like when a tool goes into the area to cut.
Ex. 1/2” cutting tool will leave a 1/4” radius on a corner.

Original Design

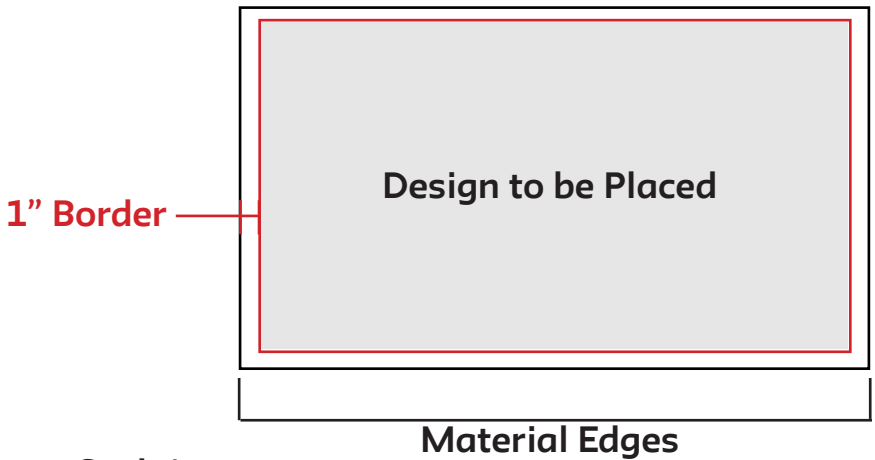


Internal Radii



Placement of Design

In order to secure material to the machine there **MUST** be the use of fasteners. Those might include screws, nails, tape, jigs or several of these methods combined. With that said, there has to be a border along the edge of any material that does **NOT** include any design components. This border shall be no less than **1”** set in from material edge.



Not Drawn to Scale

Order of Operations

When creating your cut file in Aspire you have to consider order of operations. The order of toolpaths created is very important.

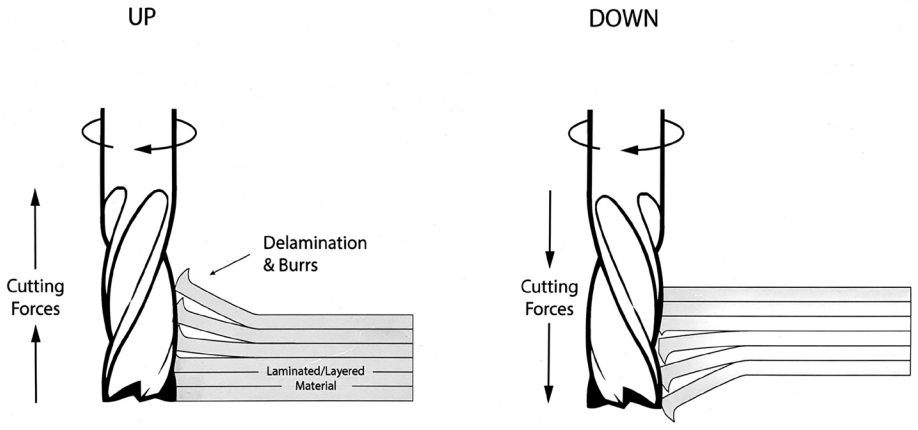
The rule of thumb: start with the smallest or least intrusive cuts first then work outwards or most instrusive cuts last.

Small profile cuts or pockets can be cut first. 3D surfacing toolpaths can run next (if applies to your design). Large profiles should always run last (we want the bulk of material connected for as long as possible before cut into).

Machining Plywood

When machining plywood a ***Down Spiral*** tool must first be used to protect the surface quality.
This only needs to be machined to break the first layer of veneer “0.008” is a good depth to start with it. This does slow down the process.

If surface quality is not important you can just use an ***Up Spiral*** tool for the whole cut, but it will blow/chip the plywood.



Climb Vs. Conventional

Conventional takes a less agressive chip from the material but leaves a fuzzier edge. We recommend using this for the bulk of the cutting. This preserves the tooling and is easier on the machine.

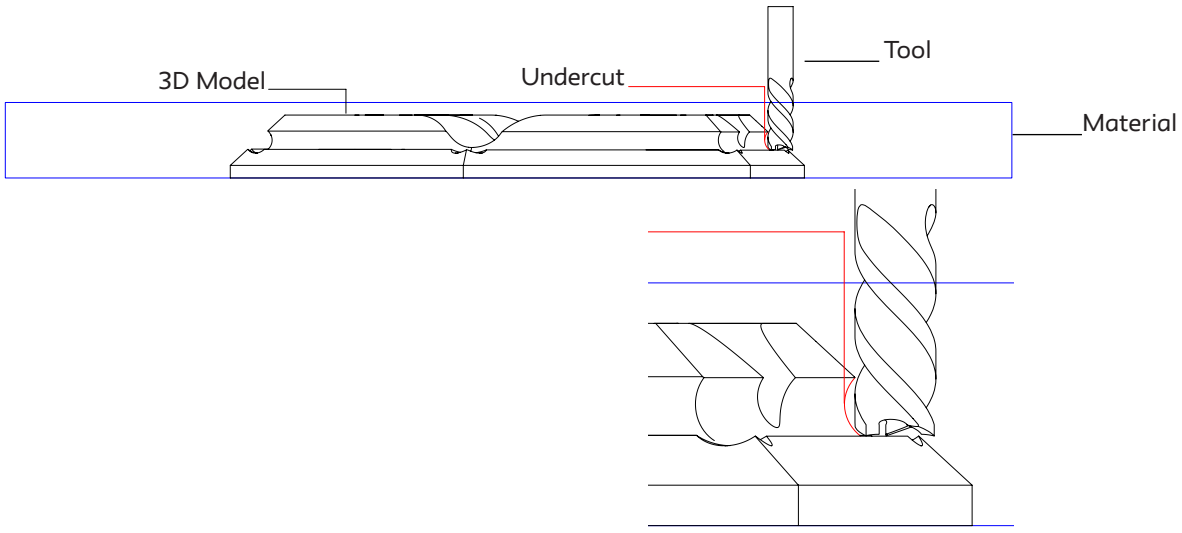
Climb takes a more agressive chip from the material but leaves a cleaner edge & can lead to tool breakage.
If you want to use climb we recommend using Conventional first to remove 90% of the material and create a finishing pass that is climb.

3D Machining

The **3D model** must be the **exact size or smaller** than your stock material.
If you are creating a digital model that is one inch thick, the stock material should than be at least one inch or slightly thicker.

Remember the stock material's thickness gets measured to the thousandths "**0.001**"

Undercuts are any angle that goes underneath the top surface of your model.
Our machines ***cannot*** cut undercuts and will leave a 90 degree cut instead.
Keep this in mind when designing.



2 Sided Machining - "Flip Files"

Flip files are done whenever a model has geometry on both sides requiring the stock to be flipped, reregistered, and machined on the 2nd side.
Often models can be sliced in half, machined, and then glued back together.

If you feel your model needs to be flipped, you will need to fill out a form for Technician approval. You will need to send it at least 1 week prior to you needing your file cut.
The Technican will get back to you to set up a time for a further consultation if approved.

The form can be found here: http://bit.ly/flipfile

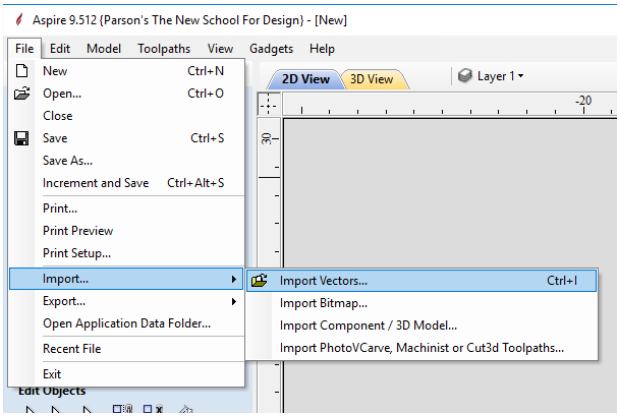
HOW TO WRITE A POCKET

A pocket uses a closed vector to machine down at a fixed depth. This results in the removal of material from the inside of a defined path.

Step 1: Import Vectors

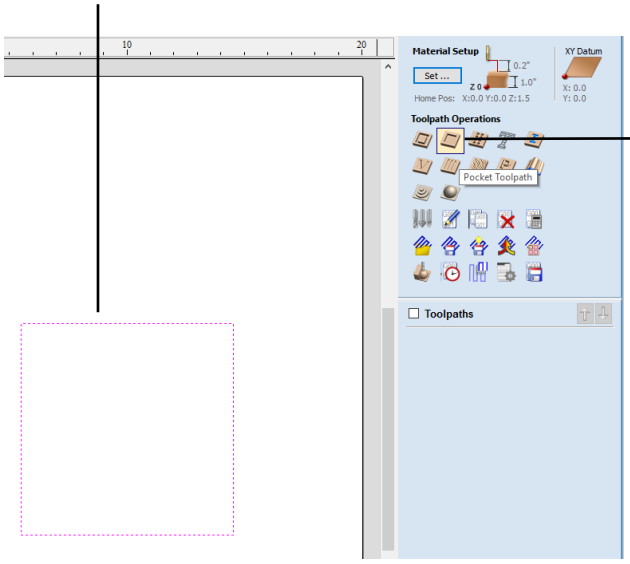
File -> Import...-> Import Vectors

Import the vectors into Aspire that were created inside your preferred program (ie. Rhino/Fusion/Illustrator)



Select the Pocket Toolpath on the right toolpath menu.

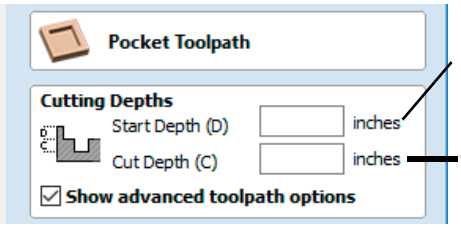
Make sure the vector you want to pocket is selected. (it will be a pink dashed line)



Step 2: Configuring Pockets

This will open a menu with a series of options to configure. Start from the top and work your way down one section at a time.

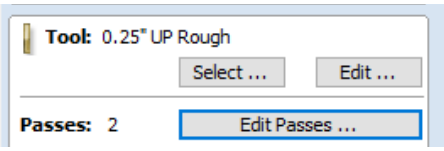
Cutting Depths



Start Depth(D) should always be set to 0.0 inches.
(the math is always calculated from the top of your material)

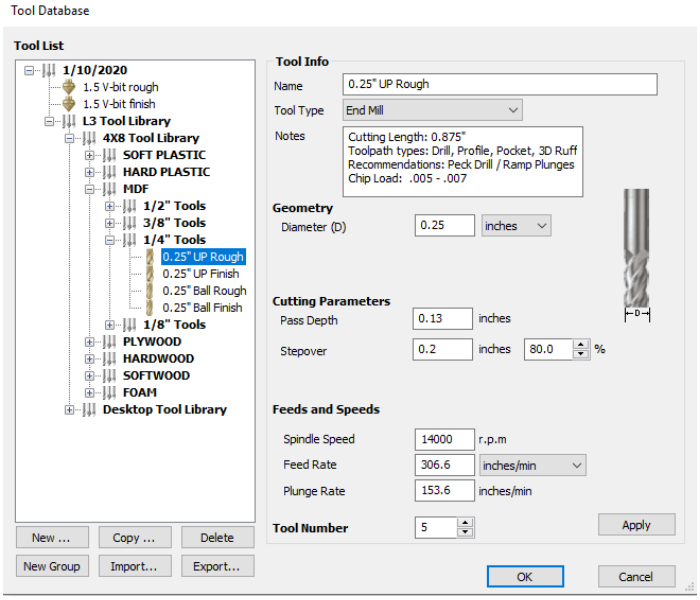
Cut Depth (C) this is set to how deep you want you pocket to go.

Tool Selection



Press **Select** to open a new window.
You will want to pick the correct tool for the type of material you are using.
Edit Passes controls how agressive the cut is. It is recommended to leave the default unless discussed with the technician.

Tool Database



This is where you pick a tool for the active toolpath operation.

note Please make sure to select the correct Tool Library!!!
4x8 Tool Library vs. Desktop Tool Library
Based on what machine you plan to use.

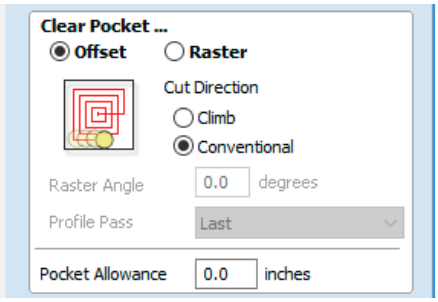
Up Rough tooling is used for most general purpose operations.

Ball Finish tooling is generally used for 3D finishing.

In this example, we have selected the ***4x8 MDF 1/4" UP Rough***.

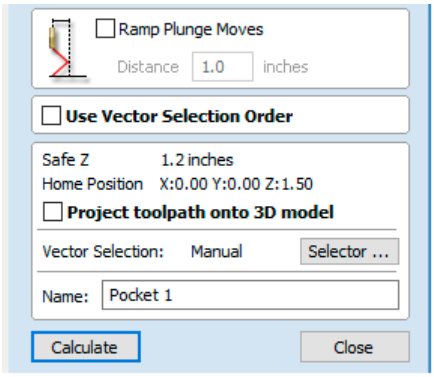
Click **OK** to continue.

Clear Pocket



This defines the method that the machine will clear away the material.
There are advantages & disadvantages to both.
Rastering with the grain of wood can result in a better cut.
Offests can create a better tooling mark for organic shapes.
We recommend using Conventional cut direction.
*read "**Things to Consider**" for more information.

Ramp Plunge Moves

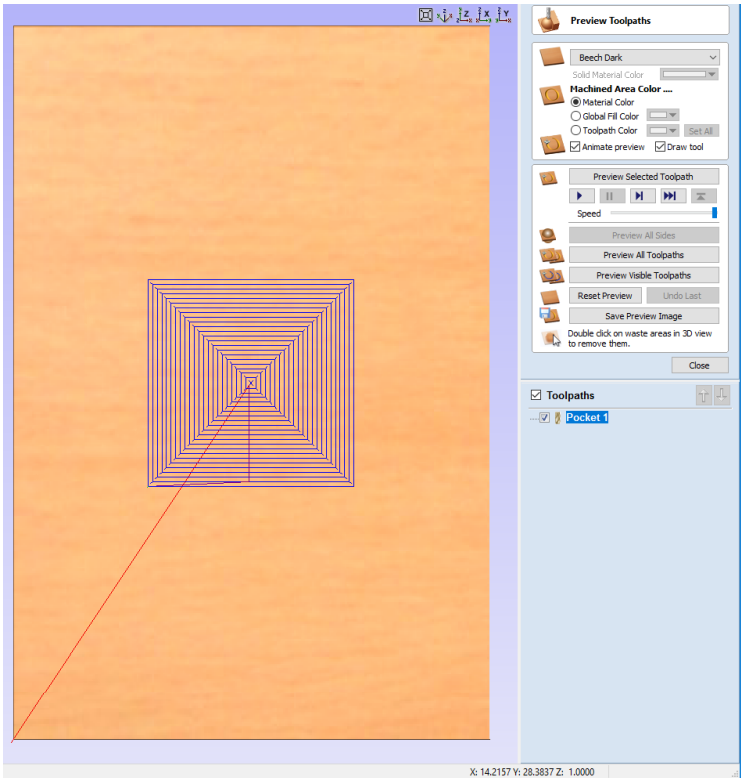


These are not required, but can create a more gentle approach and can improve surface quality. This will increase the cut time and is not supported on the Desktops.

It is recommended to rename the operation to keep track of where you are in your file.

Ex. “0.25_up_Pocket_1”

Press **Calcuate** to finish the operation.



After it finishes calculating, you can simulate & preview the results.

Click **Preview Visible Toolpaths**

If you want to edit the pocket after viewing, right click the name of the operation and click **edit** or double click the name.

This will allow you to make changes to the operation and recalculate.

When finished, click **Close** to close the preview and Click **2D view** to continue writing your operations.

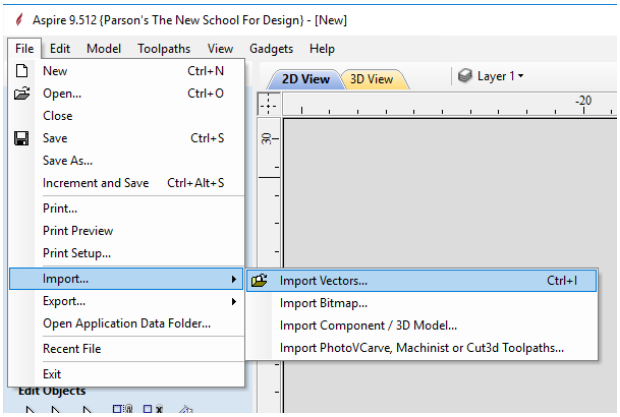
HOW TO WRITE A PROFILE

A profile uses a closed vector to machine down at a fixed depth. This results in a cut-out piece from either the inside, outside, or on the defined path.

Step 1: Import Vectors

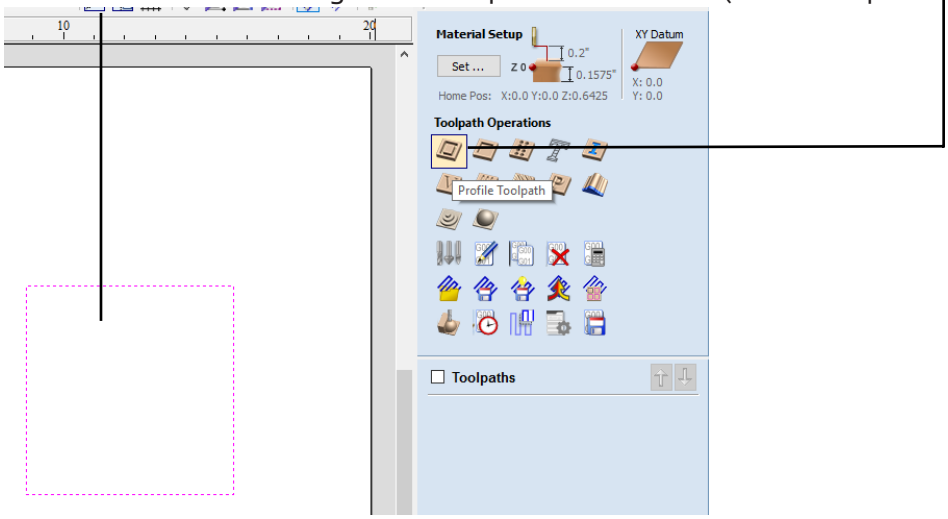
File -> Import...-> Import Vectors

Import the vectors into Aspire that were created inside your preferred program (ie. Rhino/Fusion/Illustrator)



Select the Profile Toolpath on the right toolpath menu.

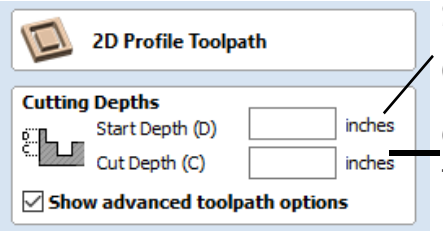
Make sure the vector you want to pocket is selected. (it will be a pink dashed line)



Step 2: Configuring Profiles

This will open a menu with a series of options to configure. Start from the top and work your way down one section at a time.

Cutting Depths

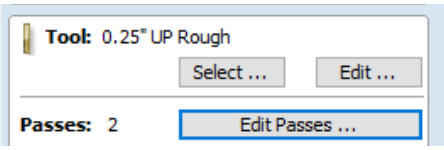


Start Depth(D) should always be set to 0.0 inches.

(the math is always calculated from the top of your material)

Cut Depth (C) this is set to how deep you want your profile to go.

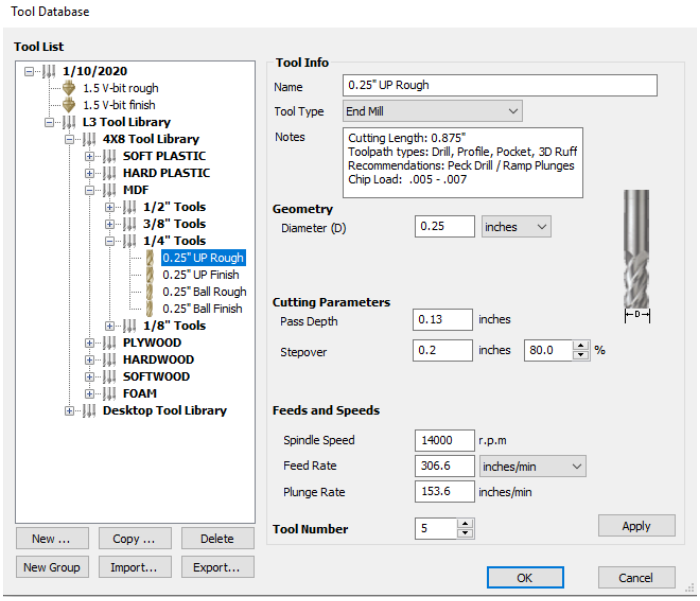
Tool Selection



Press **Select** to open a new window.
You will want to pick the correct tool for the type of material you are using.

Edit Passes controls how agressive the cut it. It is recommended to leave thedefault unless discussed with the technician.

Tool Database



This is where you pick a tool for the active toolpath operation.

note Please make sure to select the correct Tool Library!!!
4x8 Tool Library vs. Desktop Tool Library
Based on what machine you plan to use.

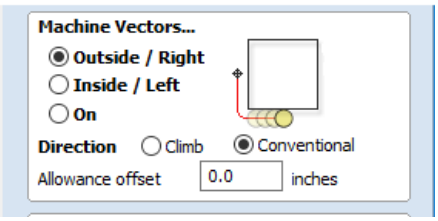
Up Rough tooling is used for most general purpose operations.

Ball Finish tooling is generally used for 3D finishing.

In this example, we have selected the ***4x8 MDF 1/4" UP Rough***.

Click **OK** to continue.

Machine Vectors...



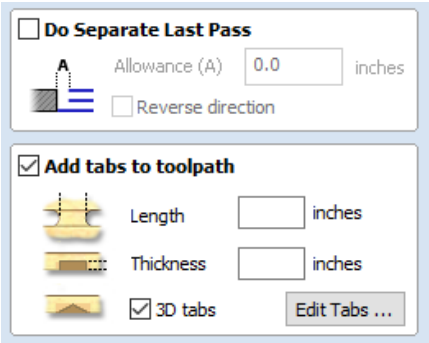
This defines how the machine will follow the selected vector.

Outside / Right will have the tool follow the vector on the outside of the vector. This is generally used to cutout shapes/vectors.

Inside / Left will have the tool follow the vector on the inside of the vector. This is generally used to create windows/dropouts from main features.

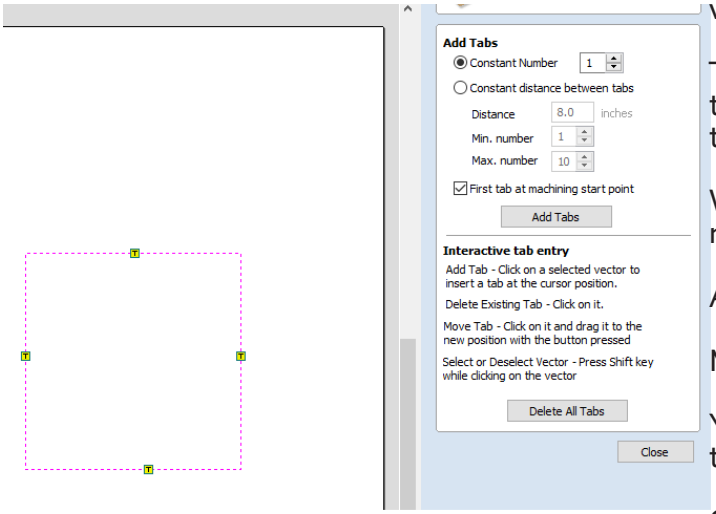
On will have the tool follow the vector on top of the vector. This is generally used for text and will have the same width of the chosen tool's diameter.

Tabs



DO NOT USE "Do Seperate Last Pass." This section can be skipped.
Check **Add tabs** to toolpath.
These are used to hold the loose part in place so it does not fly out and cause injury or damage to the machine or to your work.
Length depends on how many tabs added 0.5 is a good average
Thickness should be around 50% of your total thickness
Check **3D tabs**
Click **Edit Tabs...**

Tabs Menu



*Tabs also prevent vibration & chatter and prevents tool breakage.

This will open a new window to add tabs to your vector.

There is an automated function that will generate tabs across your vector. This does not always place tabs in the best locations.

We recommend looking through and optimizing to make removing tabs easier.

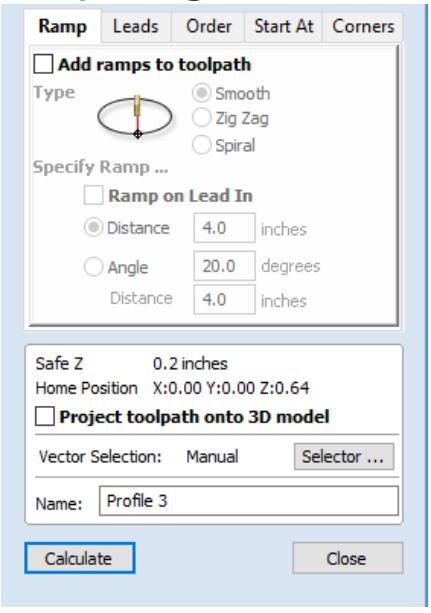
Avoid corners and internal curves.

Make sure any long vectors have tabs attached.

You can manually edit curves by clicking the vector to add.

Clicking on the "**T**" to delete tabs or click and hold to move tabs.

Ramp Plunge Moves



These are not required but can create a more gentle approach and can improve surface quality.

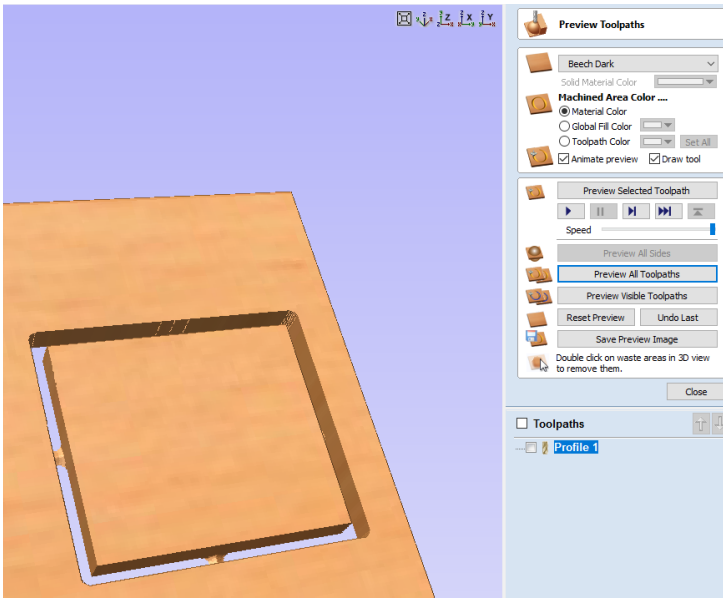
This will increase the cut time and is not supported on the desktops.

It is recommended to rename the operation to keep track of where you are in your file.

Ex. "0.25 UP profile"

Press **Calculate** to finish the operation.

Simulate



After it finishes calculating, you can simulate & preview the results.

Click **Preview Visible Toolpaths**

If you want to edit the pocket after viewing, right click the name of the operation and click **edit** or double click the name.

This will allow you to make changes to the operation and recalculate.

When finished, click **Close** to close the preview and click **2D view** to contine writing your operations.

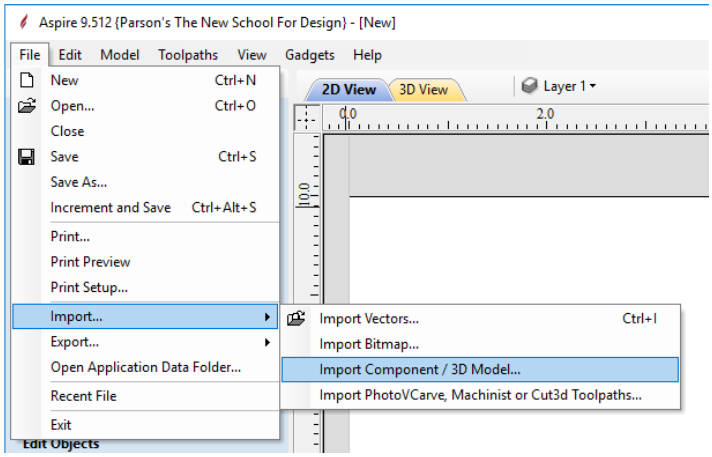
HOW TO WRITE A 3D MACHINING OPERATION

3D Machining uses an STL & Vectors to machine out a 3D digital model.
This is easy to write but needs to have proper prep work done.
Refer to the tips page to make sure your file is usable and ready.

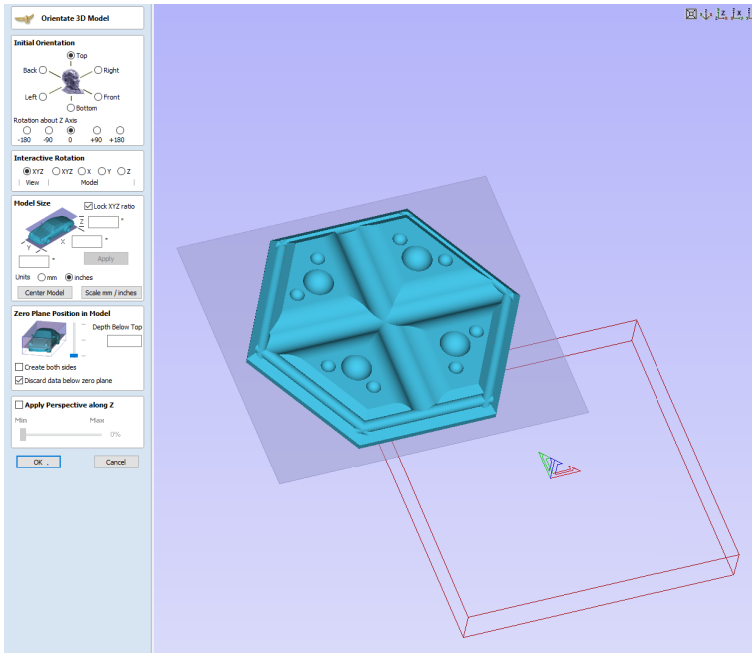
Step 1: Import 3D Model

File -> Import...-> Import Component / 3D Model

Import the STL to Aspire that was created inside your preferred program (ie. Rhino/Fusion/Solidworks



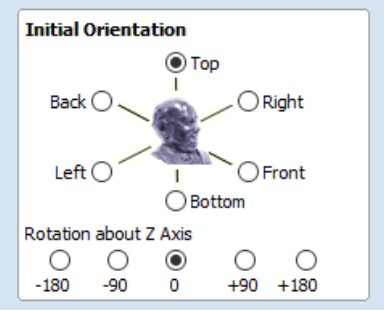
This will open a menu to orient and place your model into your stock.



Step 2: Configuring Orientation

This will open a menu with a series of options to configure.
Start from the top and work your way down one section at a time.

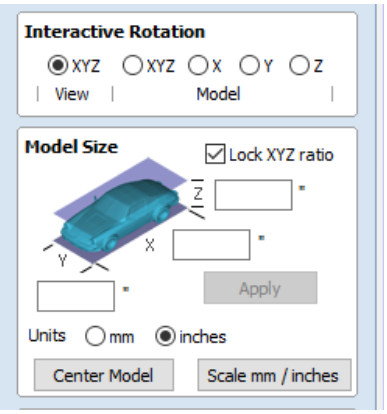
Cutting Depths



Use these selections to orient your part if it imported at an odd angle.

Rotating around the Z axis can align your model across the X/Y axis.

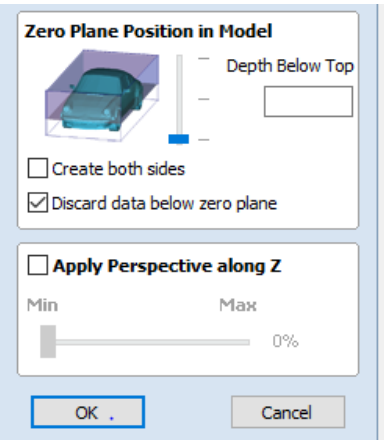
Model Size



You can leave the Interactive Rotation as default.
If you need to adjust the scale of your model this is the section to do it.It is recommended to have your model properly scaled in CAD before this stage.

Remember when using 3D models your STOCK MATERIAL **must** be larger than your digital model.

Zero Plane Position in Model



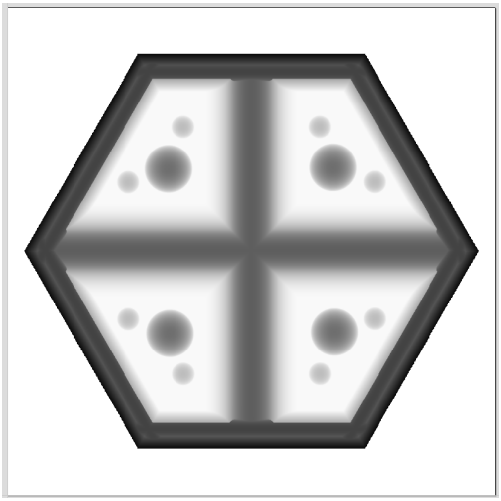
This is used to set how low you want your model to sit in the stock. Almost always you will want to bring the slider to the bottom.

This sets your model's flat bottom to the bottom of the stock. Removing excess material from the top.

You can skip the Apply Perspective along Z.

Press **OK** to continue.

Step 3: Click 2D view to go to the next steps



This will give you a grayscale top side view of your model.

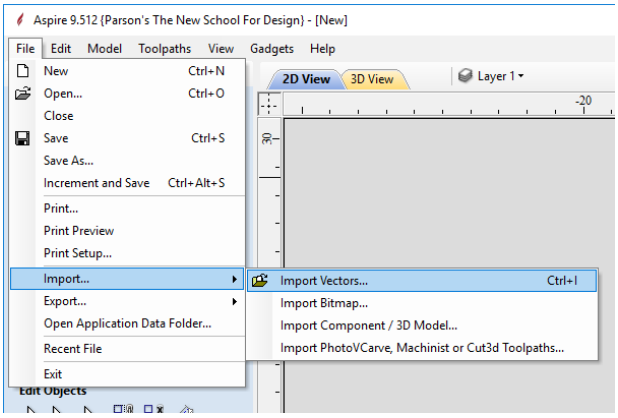
This will allow you to align and import your vectors. The vectors are used to constrain the tool to the model.

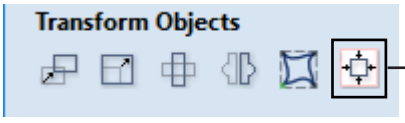
The vectors will also be used to cut out your model using the profile operation.

Step 4: Import Vectors

File -> Import...-> Import Vectors

Import the vectors to Aspire that were created inside your preferred program (ie. Rhino/Fusion/Solidworks



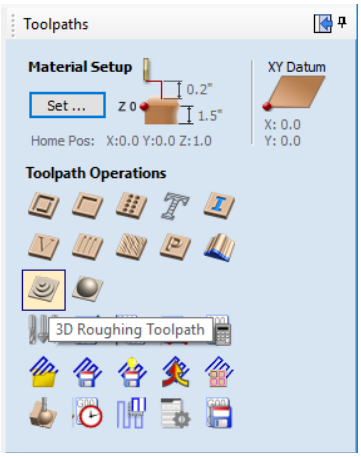


You may need to align your vectors to your model.

This can be done using the **Center Align** tool under the **Transform Objects** section.

Once the vectors are aligned to the model you can move on to writing the 3D roughing operation.

Step 5: 3D Roughing Toolpath

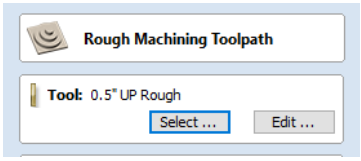


From the toolpath menu, Select 3D Roughing Toolpath.

This will open a menu with a series of options to configure. Start from the top and work your way down one section at a time.

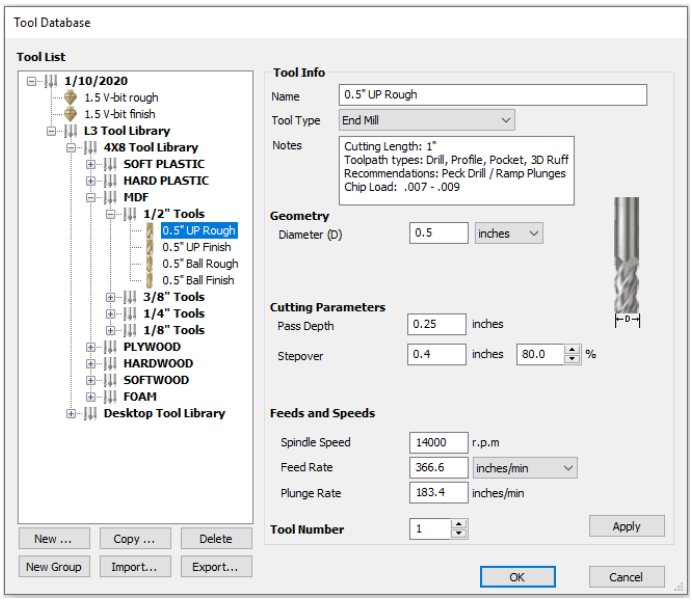
Make sure your vector is selected. It will be a dashed pink line when selected.

Tool Selection



Press **Select** to open a new window.
You will want to pick the correct tool for the type of material you are using.

Tool Database



This is where you pick a tool for the active toolpath operation.

****note**** Please make sure to select the correct Tool Library!!!
4x8 Tool Library vs. Desktop Tool Library
Based on what machine you plan to use.

Up Rough tooling is used for most general purpose operations.

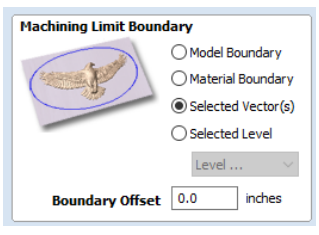
Ball Finish tooling is generally used for 3D finishing.

In this example, we have selected the ***4x8 MDF 1/2" UP Rough***.

*If you are using 3-4"foam Select the LONG General bit instead.

Click **OK** to continue.

Machining Limit Boundary

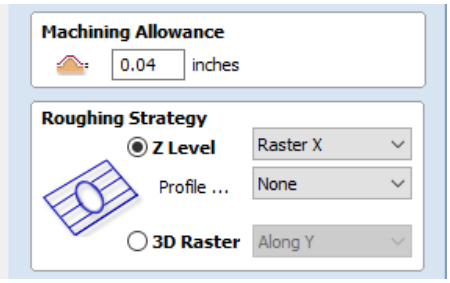


You will want to select **Selected Vector(s)**

If your vector is not selected, this is the time to do it!

Boundary Offset can be adjusted to let the tool move past the boundary. This is often used if it's not capturing certain detail but we still want it to only machine slightly past the boundary.

Roughing Strategy



You can leave the Machining Allowance as its default.

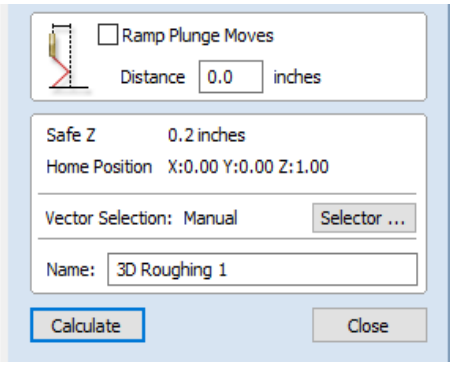
Roughing Strategy approaches how to remove material. The roughing part is meant to quickly remove large swathes of material, but leave a very crude model.

Z Level uses a type of pocketing that follows the shape of your model. This is generally a lot faster than the 3D Raster option, but can leave a lot of leftover material. This is a great choice for soft materials.

3D Raster follows the topology of your model. This removes a lot more material, but takes a lot longer to machine. This is a great choice for hard materials.

Both options allow you to select what axis you want to machine across. This is subjective to the user and in some cases does not matter. In other cases, like hardwood, it would be best to match the direction of the wood grain.

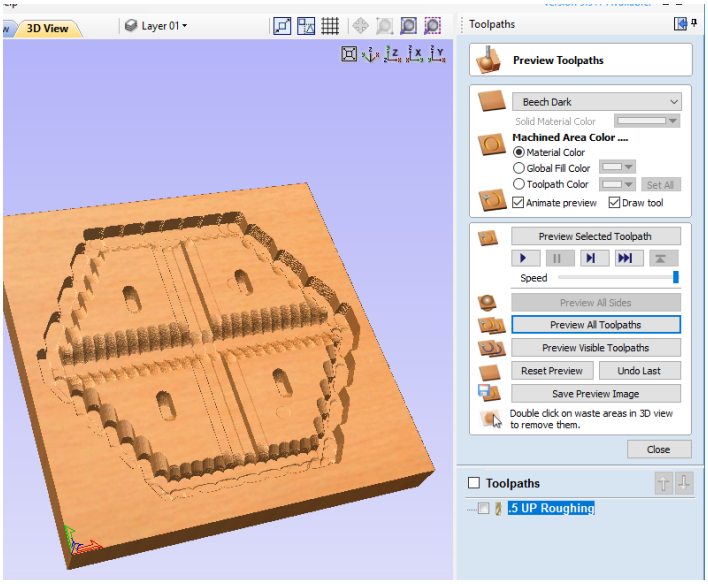
Ramp Plunge



Ramp plunges are not required but can create a better surface. This will also increase the machining time.

It is recommended to rename the operation to keep track of where you are in your file.
Ex. "***0.5 UP Roughing***"

Press **Calculate** to finish the operation.



After it finishes calculating, you can simulate & preview the results.

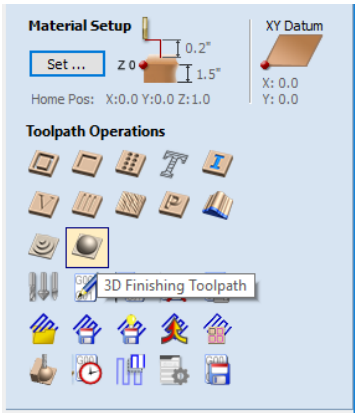
Click **Preview Visible Toolpaths**

If you want to edit the roughing after viewing:

Right click the name of the operation and click edit or double click the name. This will allow you to make changes to the operation and recalculate.

When finished, click **Close** to close the preview and click **2D View** to continue writing your operations.

Step 6: 3D Finishing Toolpath

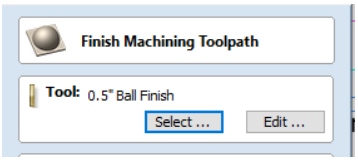


From the toolpath menu, Select **3D Finishing Toolpath**.

This will open a menu with a series of options to configure. Start from the top and work your way down one section at a time.

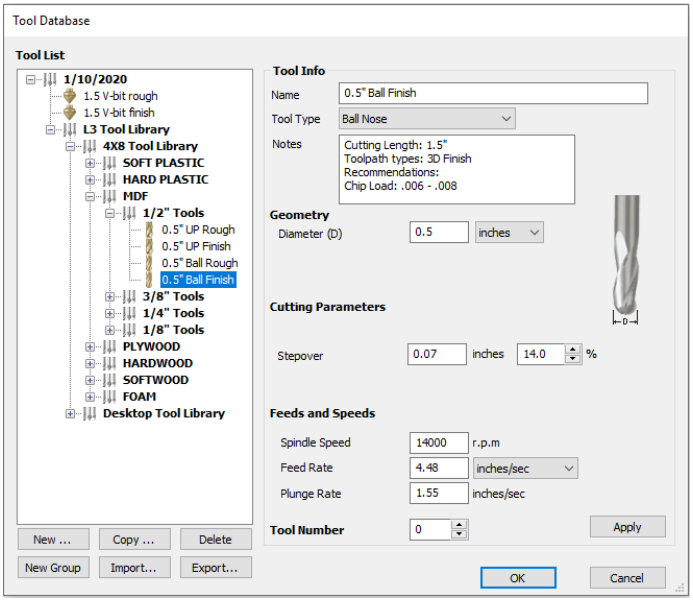
Make sure your vector is selected. It will be a dashed pink line when selected.

Tool Selection



Press **Select** to open a new window.
You will want to pick the correct tool for the type of material you are using.

Tool Database



This is where you pick a tool for the active toolpath operation.

note Please make sure to select the correct Tool Library!!!
4x8 Tool Library vs. Desktop Tool Library
Based on what machine you plan to use.

Up Rough tooling is used for most general purpose operations.

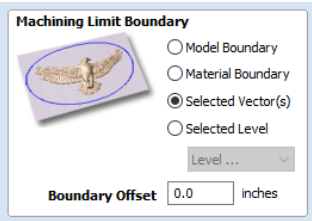
Ball Finish tooling is generally used for 3D finishing.

In this example, we have selected the ***4x8 MDF 1/2" Ball Finish***.

*If you are using 3-4"foam Select the LONG Ball Finish bit instead.

Click **OK** to continue.

Machining Limit Boundary

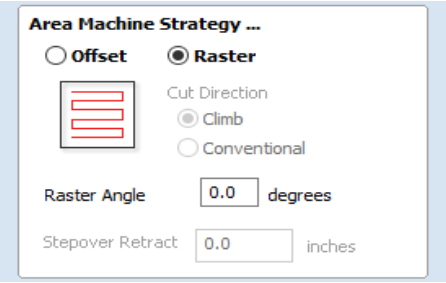


You will want to select **Selected Vector(s)**

If your vector is not selected, this is the time to do it!

Boundary Offset can be adjusted to let the tool move past the boundary. This is often used if it's not capturing certain detail but we still want it to only machine slightly past the boundary.

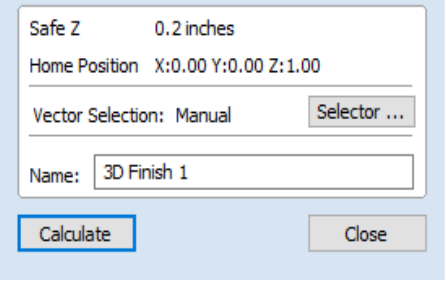
Finishing Strategy



This defines the method that the machine will clear away the material.
There are advantages & disadvantages to both.

Rastering with the grain of wood can result in a better cut. This method is often best for large shallow curves.

Offset can create a better tooling mark for organic shapes. This method is often best for steep slanted curves.



It is recommended to rename the operation to keep track of where you are in your file.

Ex. "***0.5 UP BN Finish***"

Press **Calculate** to finish the operation.