# 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.
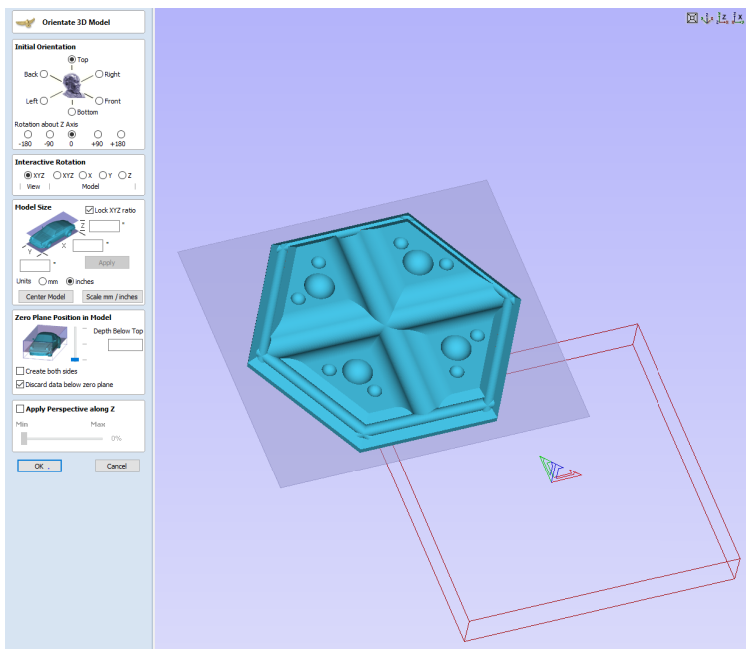Refer to the tips page to make sure your file is usable and ready.

Step 1: Import 3D Model
File -> Import...-> Import Component / 3D Model

Import the STL to Aspire that was created inside your preferred program
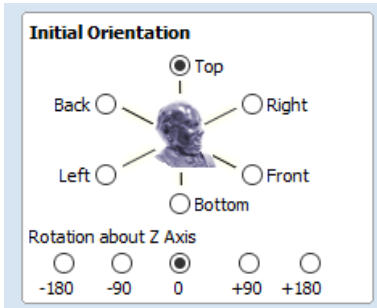(ie. Rhino/Fusion/Solidworks
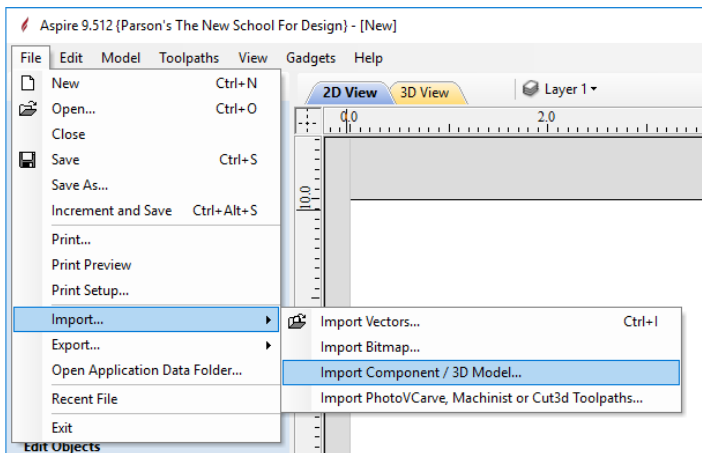
This will open a menu to orient and place your model into your stock.

Step 2: Configuring Orientation

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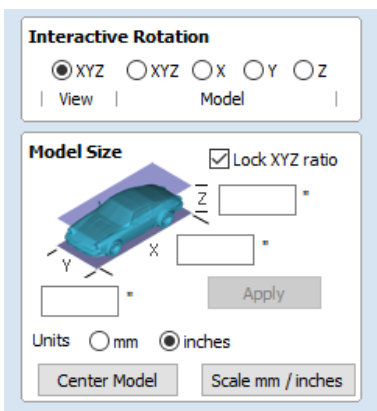Start from the top and work your way down one section at a time.

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.

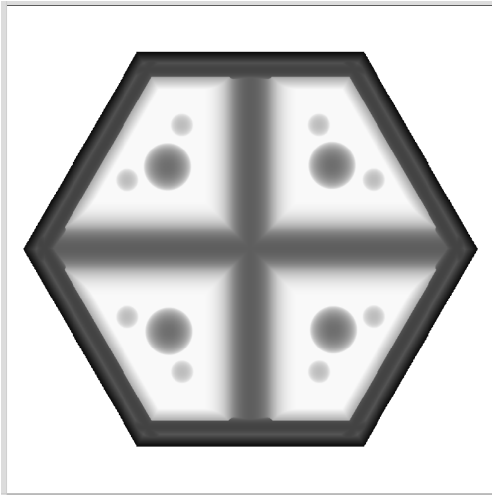## Zero Plane Position in Model
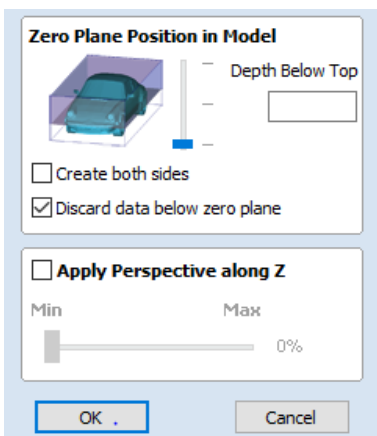
This is used to set how low you want your model to sit in the stock. Almost always you will want to bring the slider to the bottom.

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
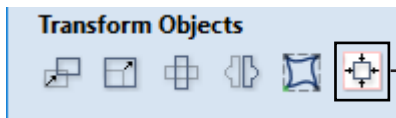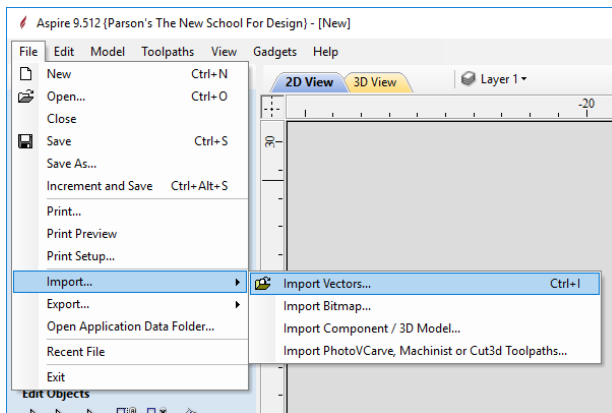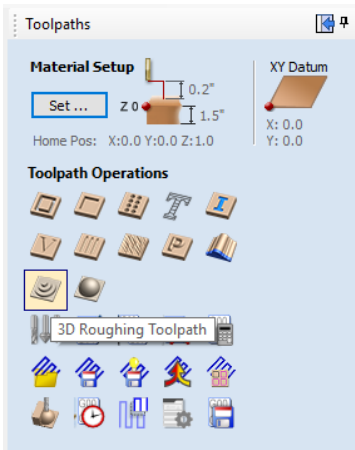

You may need to align your vectors to your model.

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.
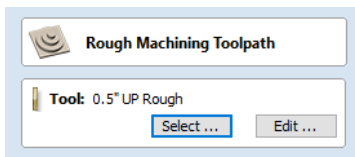
Step 5: 3D Roughing Toolpath

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.
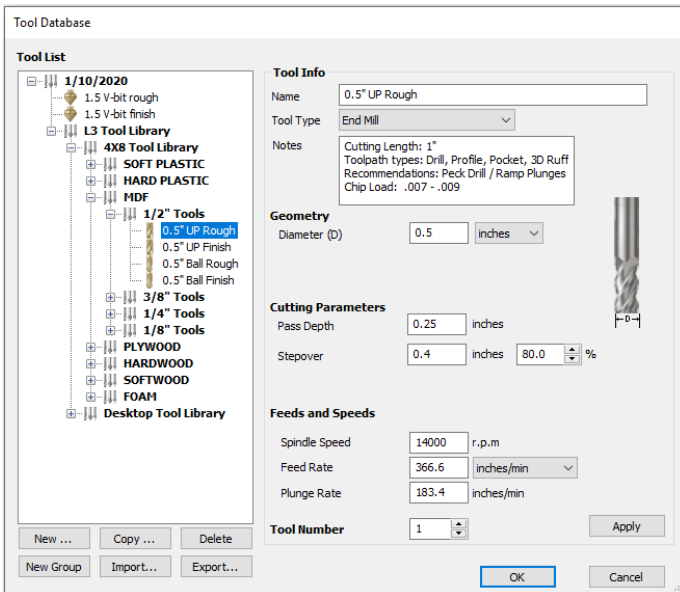
Tool Selection

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.

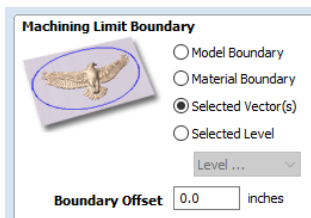*__Up Rough__* tooling is used for most general purpose operations.

__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.
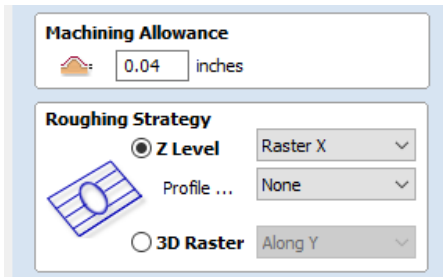Click **OK** to continue.

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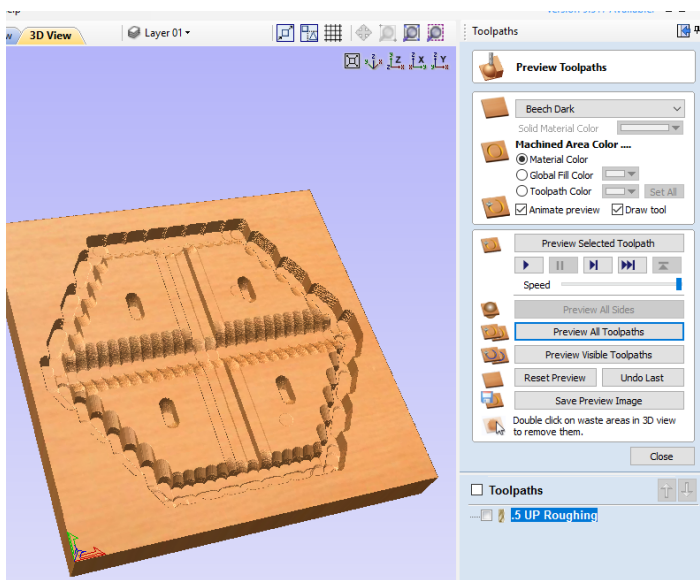
Ramp Plunge
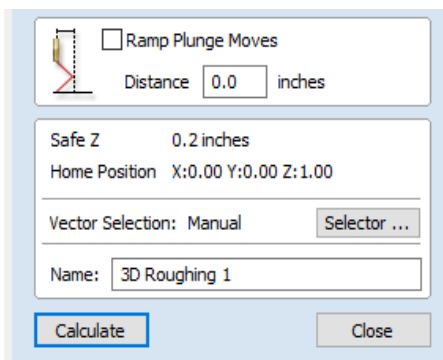
Ramp plunges are not required but can create a better surface. This will also increase the machining time.

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.
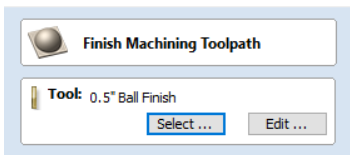
Step 6: 3D Finishing Toolpath
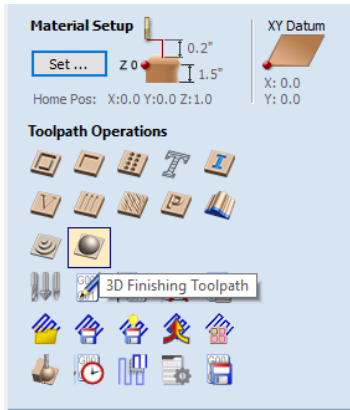
From the toolpath menu, Select **3D Finishing Toolpath**.

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.

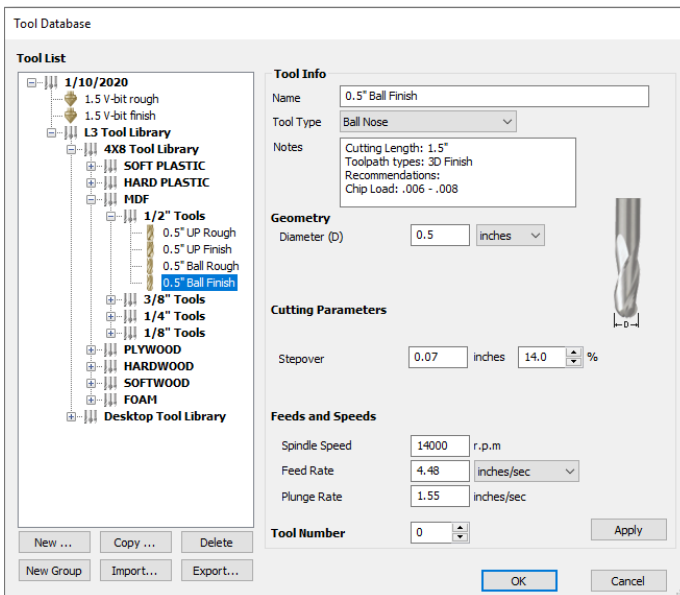Make sure your vector is selected. It will be a dashed pink line when selected.

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.

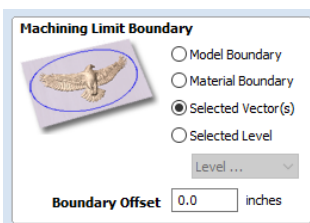_**Up Rough**_ tooling is used for most general purpose operations.

**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.