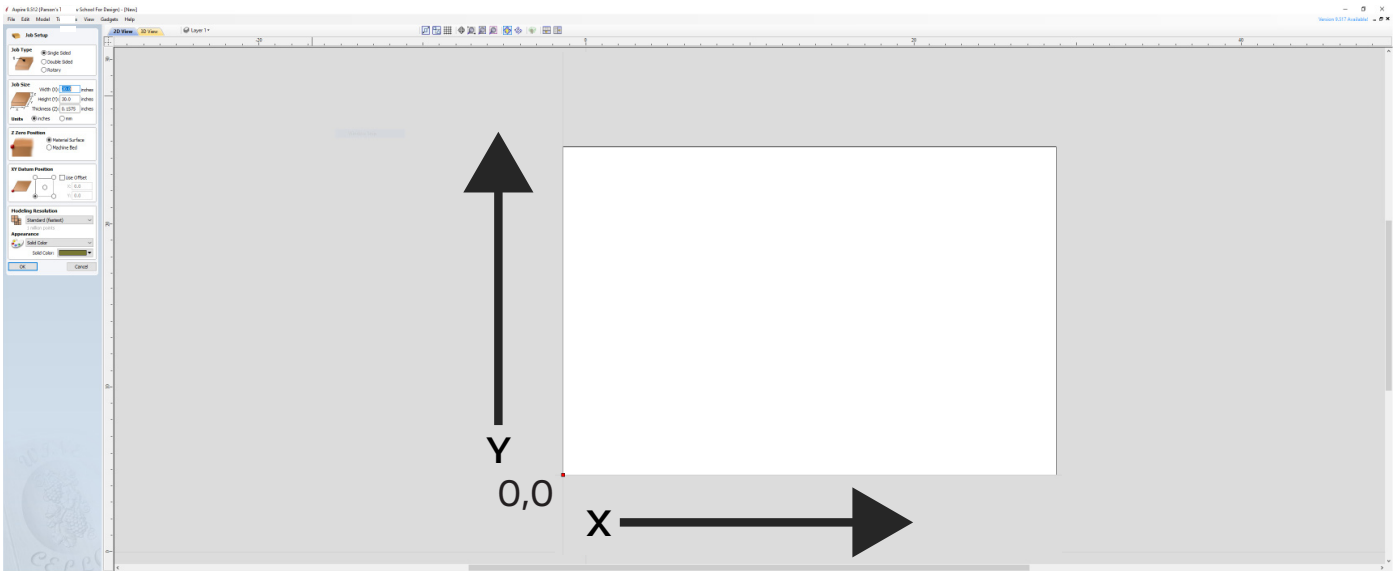
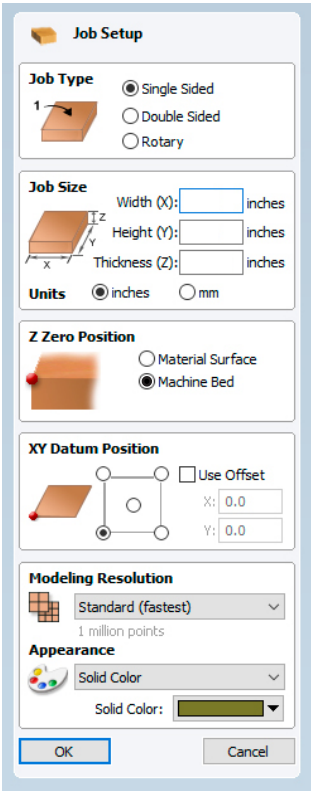
ASPIRE ASSEMBLY GUIDE

CAM PREP for furniture-like products



The longest dimension

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.

Click “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.

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 0.750" (3/4) thick plywood is a nominal measurement. Its stock measurement is generally between 0.710-0.740".

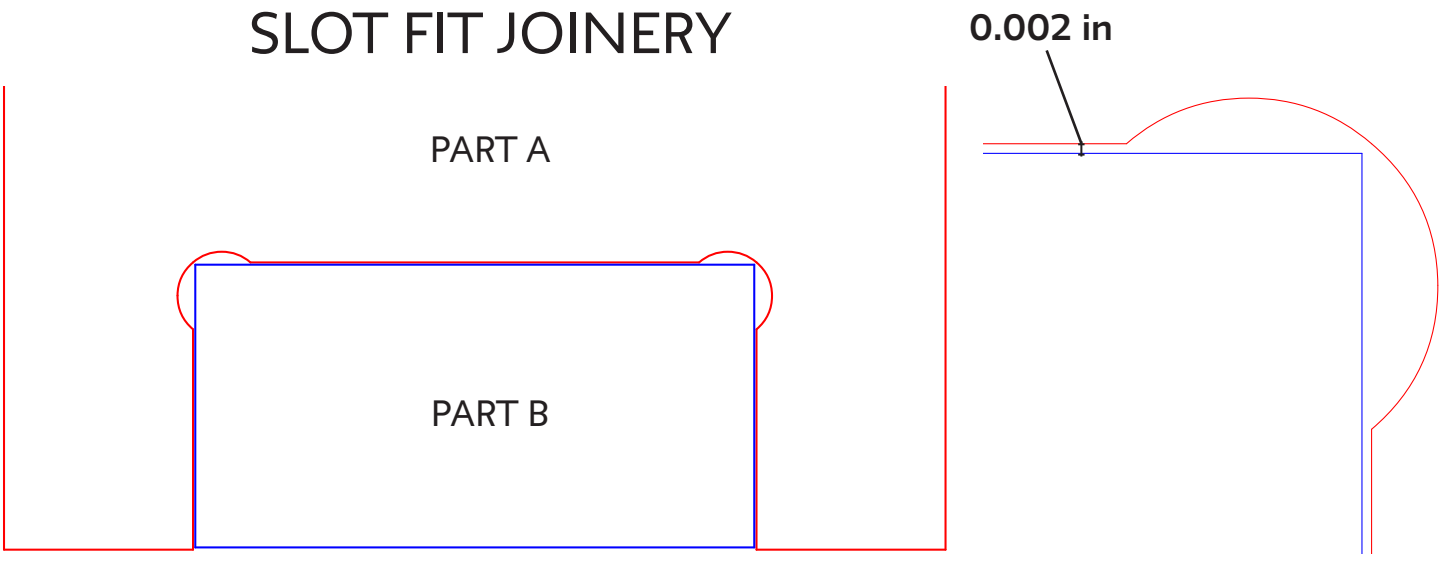
Material thickness affects joinery components. Therefore, you specific chosen material must be incorporated into the digital model when planning for CNC cuts and later assembly. Please remember that you might buy 3/4" plywood or MDF for your "furniture" but that thickness varies and 100% will NOT measure 0.750".

Tolerance

Assuming that the furniture-like product includes slots, tabs, and cross joinery, it is important to note that material "fit" also comes to affect these joinery components. This in turn, must be incorporated into the digital model when planning for CNC cuts and later assembly.

This is where **tolerance** comes into the picture. CNC cuts are fairly precise in relation to the digital model, so it is very important to be intentional with dimensioning. In order to avoid fits being either too snug or too loose, there has to be a "comfortable" spacing between connecting parts. A good rule of thumb is:

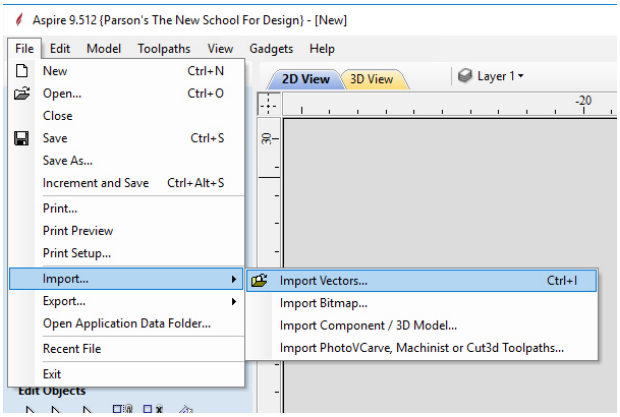
0.03-0.05 mm or 0.001-0.002 in



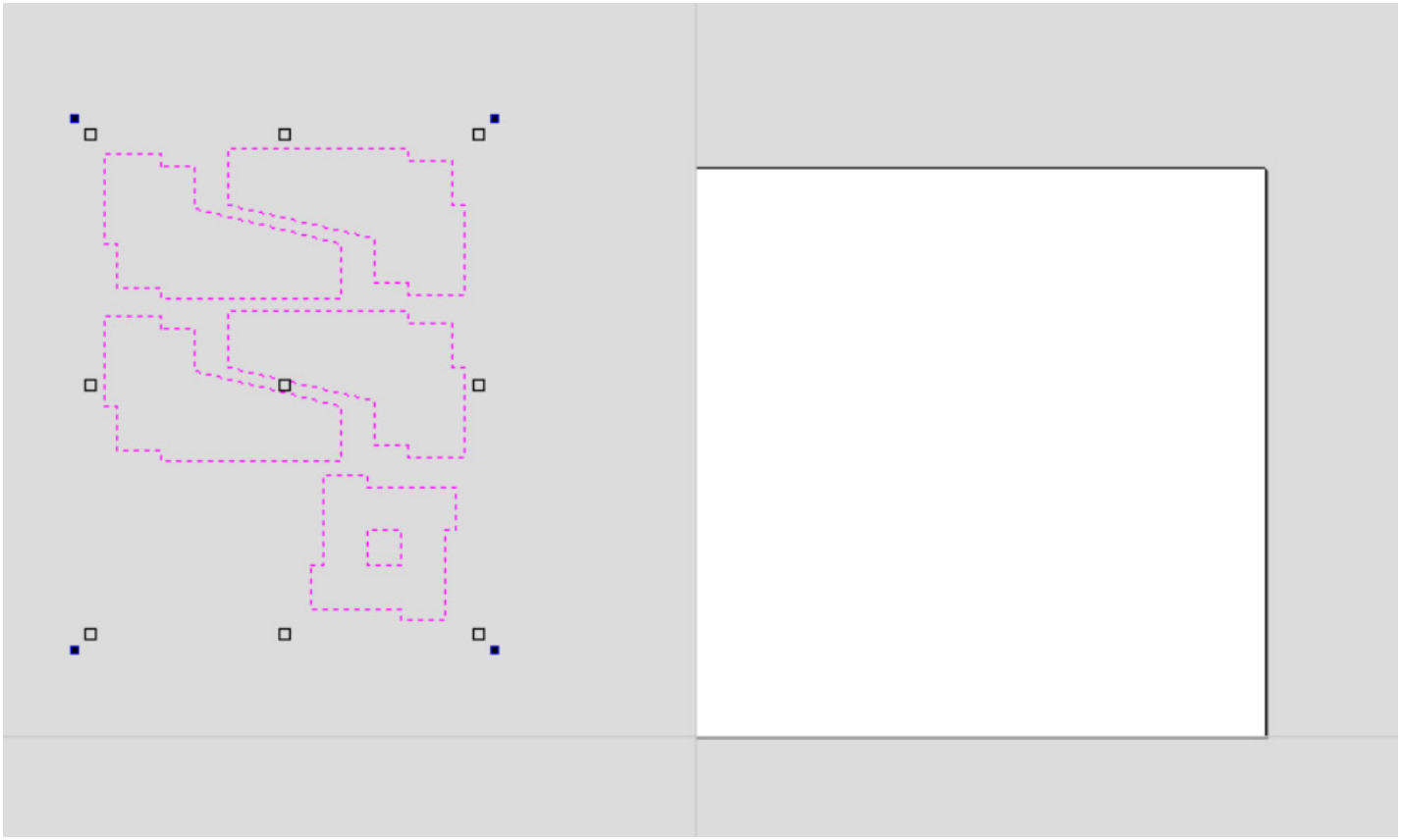
Step 1: Import Vectors

File -> Import...-> Import Vectors

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

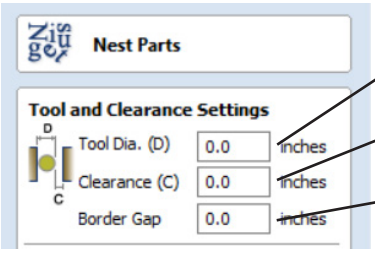


NOTE: Sometimes your imported vectors will appear off the artboard; this is normal. You can easily move them while selected (pink dashed lines) manually over to the artboard.

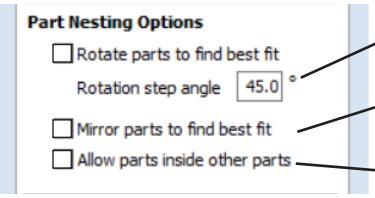


Step 2: Nesting Parts

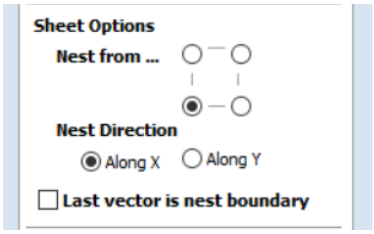
Select the **Nesting Parts** icon on the left side menu.
Found under Offset and Layout



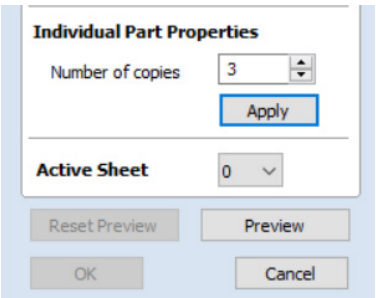
Tool Diameter(D) this will equal the tool you decide to cut with.
Clearance (C) this should be slightly larger than tool used. i.e. tool=0.25” / clearence=0.30”
Border Gap this is the spacing offset from the edge of the artboard. No portion of design should intrude on this space.



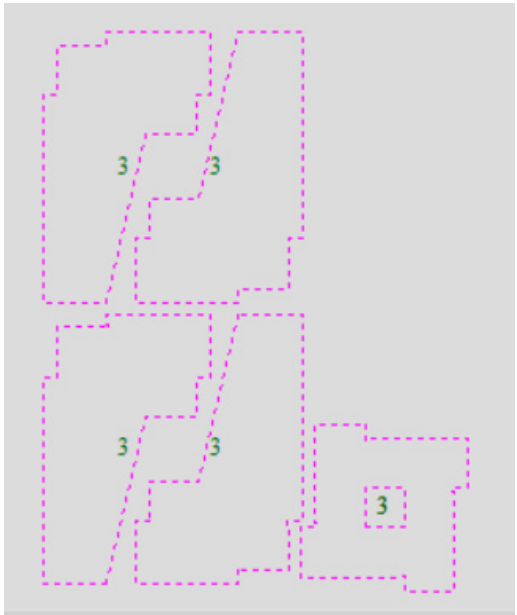
Rotate Parts this will allow parts to optimize spacing by rotating said parts.
Mirror Parts this will allow parts to flip vertical/horizontal for best fit. DO NOT check if need single direction parts.
Parts inside parts this will allow parts inside other parts. CHECK if have scrap parts inside that could hold smaller parts. DO NOT check if interior parts are not scraps.



Sheet Options DO NOT CHANGE



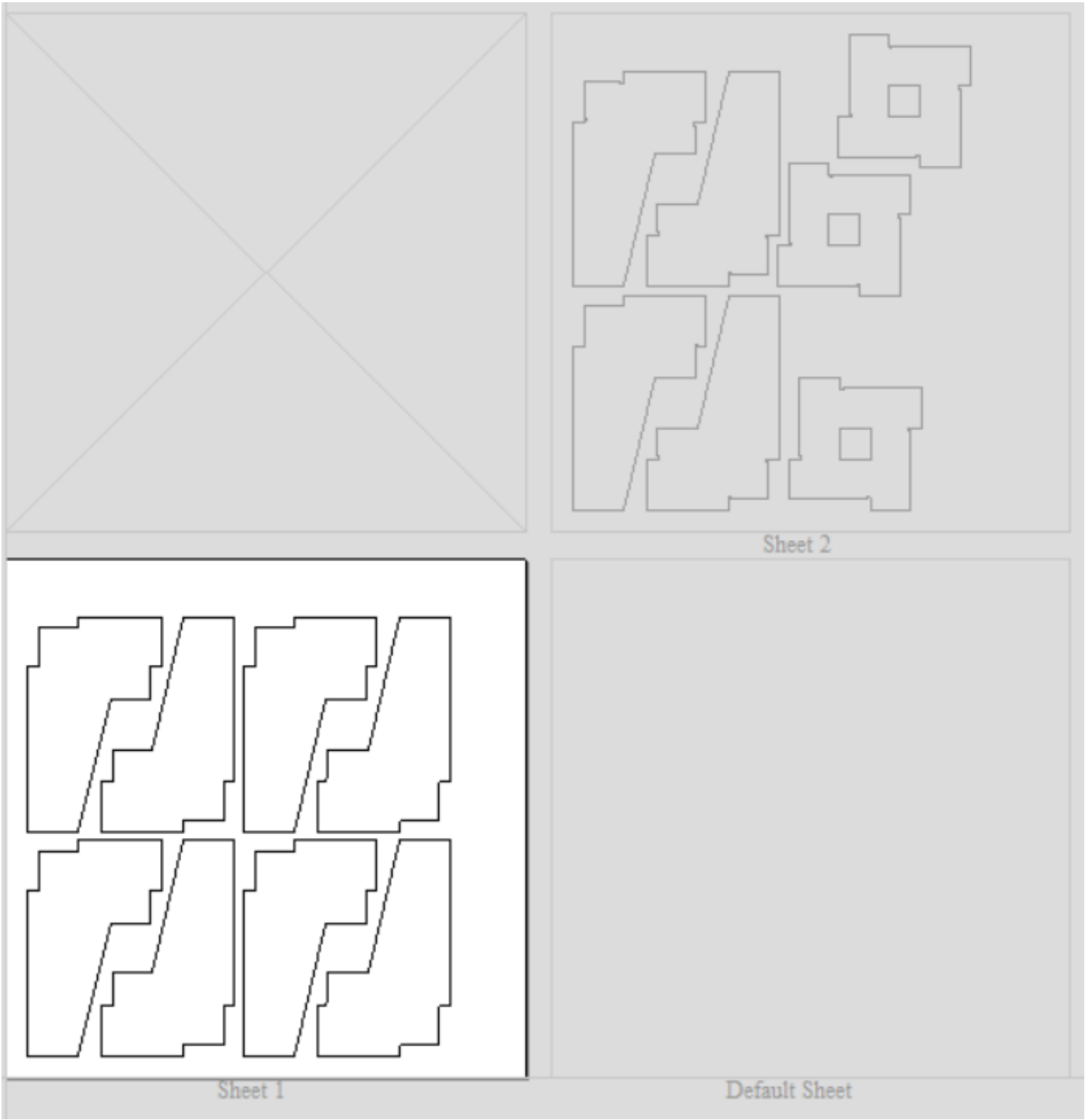
Individual Part Properties
Here you can define how many copies of the original parts you may need. Select original parts, define number of copies and click **apply**.



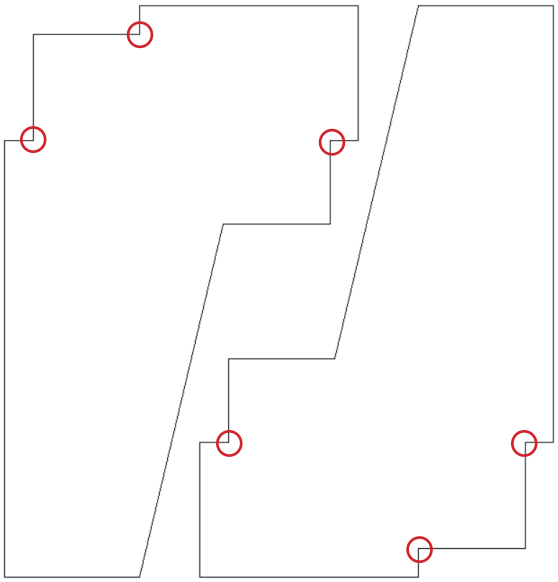
Active Sheets
By clicking **PREVIEW** aspire will provide a visual of how many sheets your copies would require. At this point you would know how many additional sheets of material you might need at the same size or you may need to buy a larger sheet of material.

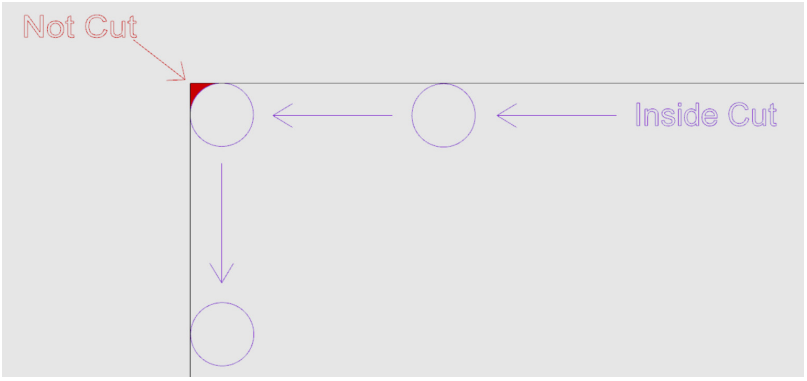
i.e. you have bought a 4'x4' sheet of plywood but need two or you could buy a 4'x8' sheet to hold all copies of parts.

Once you are satisfied with how things look click **OK**.



IMPORTANT: CNC cannot cut sharp 90 degree interior corners

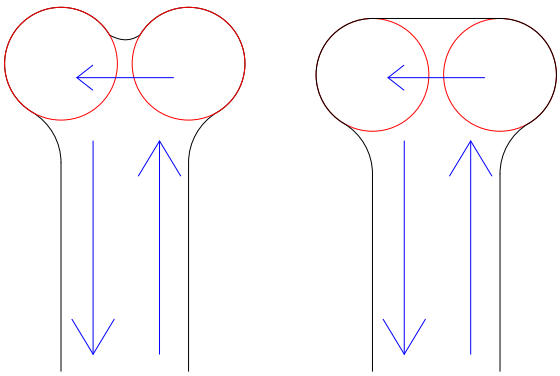




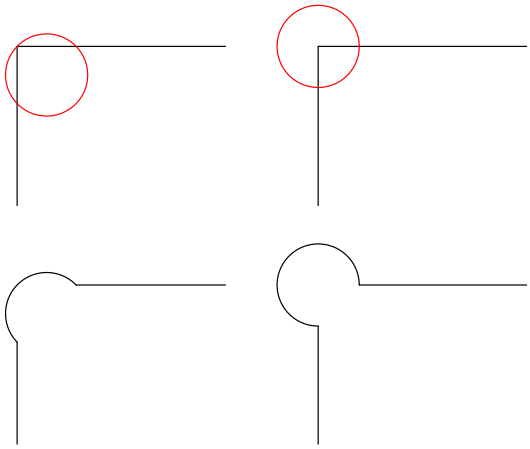
CNC Solutions

These options are decided by preference and appearance. As shown in the image to the right solutions are simply dog bone or t-bone. Each option has a variety of styles which can be seen below.

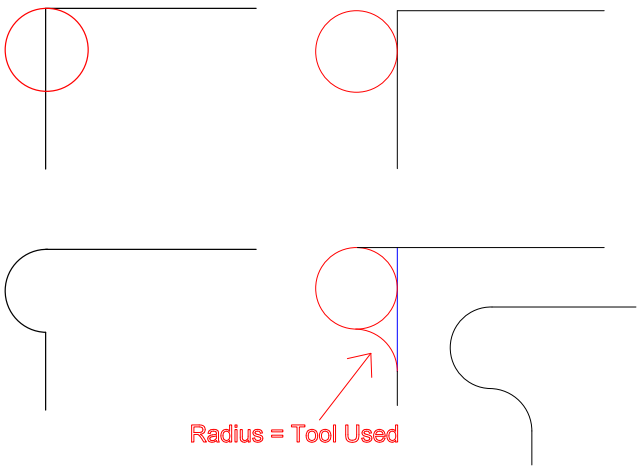
Dog Bone vs. T Bone



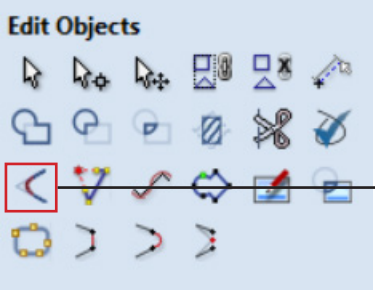
Dog Bone Fillet Options



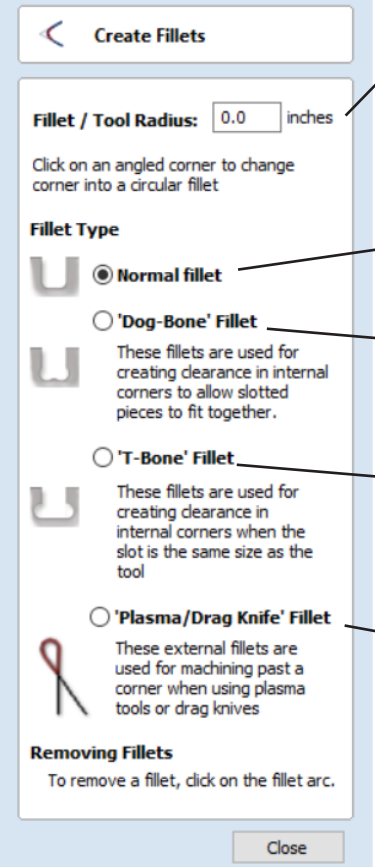
T-Bone Fillet Options



Step 3: Fillet Inside Corners



Select the **Fillet** icon on the left side menu.
-Found under Edit Objects



Fillet / Tool Radius this should equal the radius of the tool you choose for CNC cuts.

Normal this will allow you to round over sharp corners but do not need to fit into other pieces.

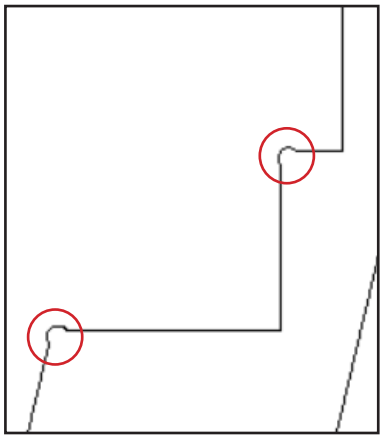
Dog-Bone this will allow you to create rounded cutaways on internal corners to allow varying pieces to slot-fit together.

T-Bone this will allow you to create rounded cutaways on internal corners to allow same-size pieces to interconnect.

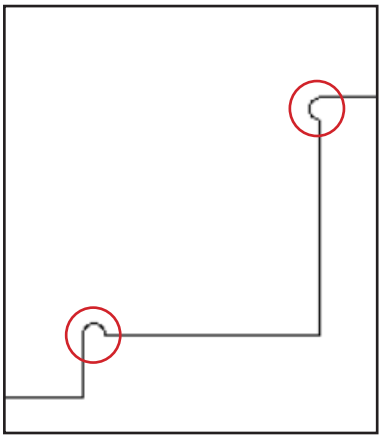
Plasma/Drag Knife Making Center CNC Shops do not use this function

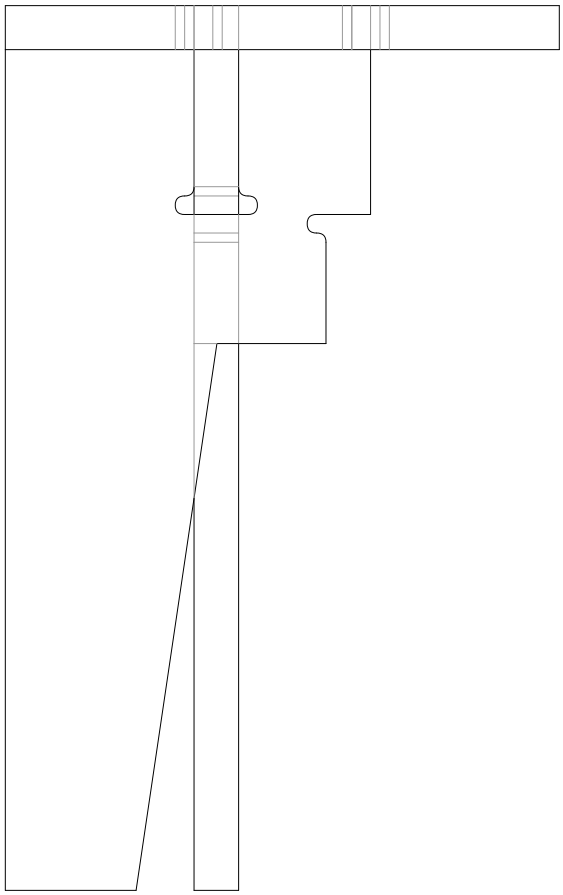
Click **Close**

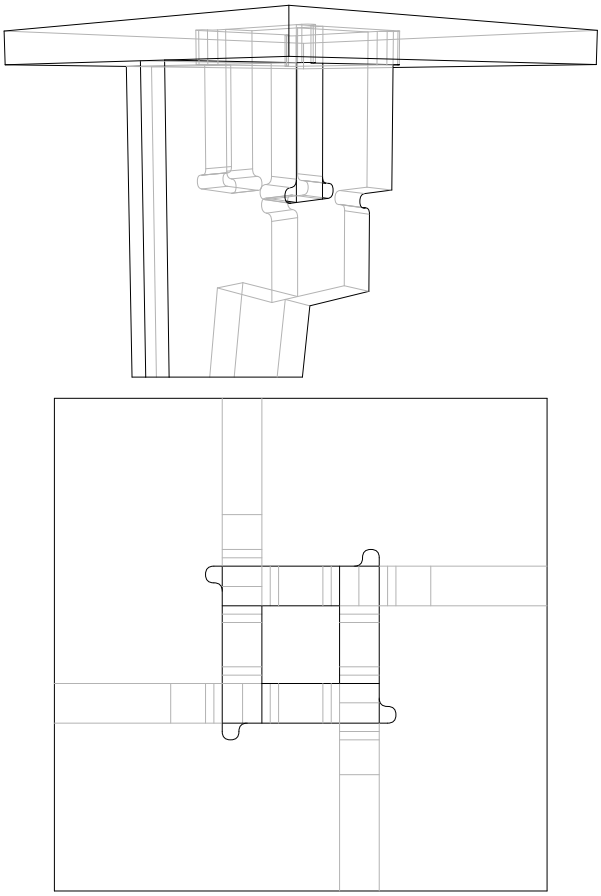
Dog-Bone Fillet



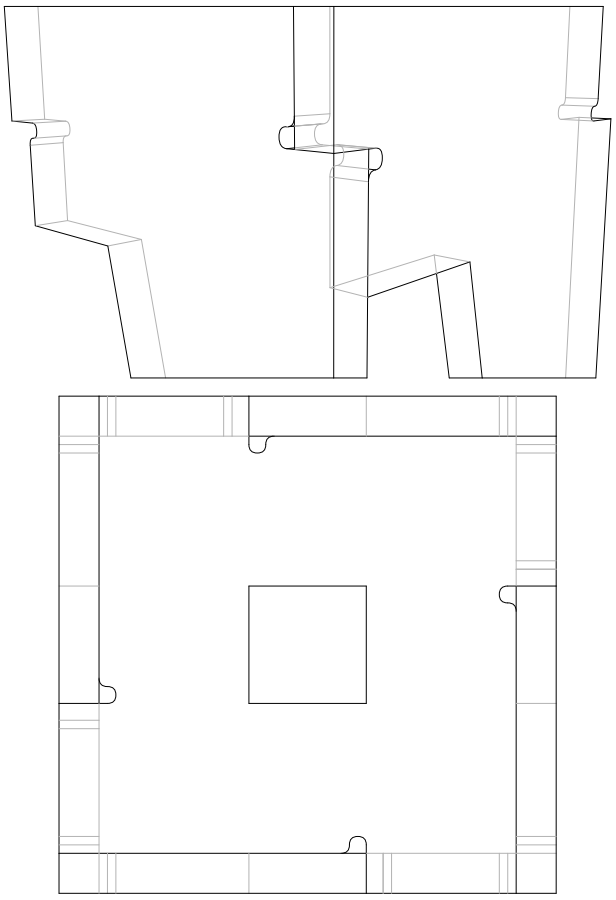
T-Bone Fillet

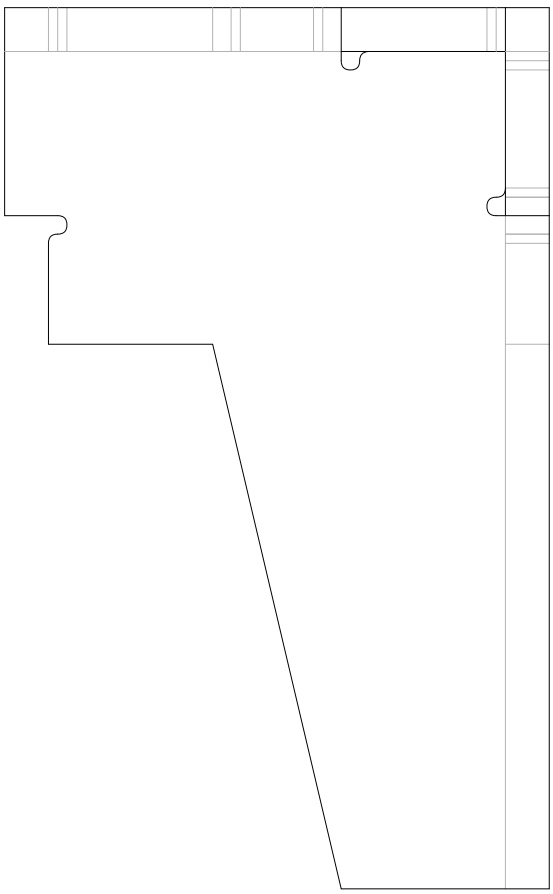






Examples of Joinery using Fillet Methods



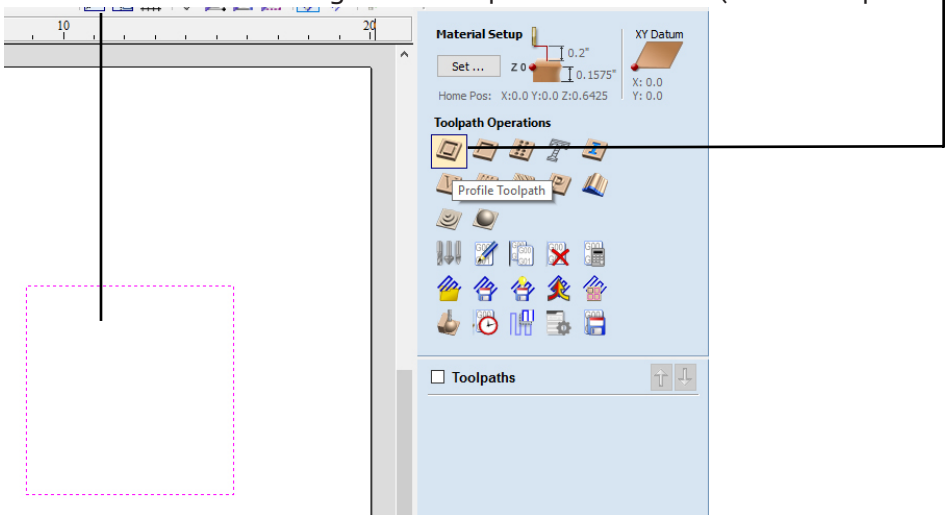


Step 4: Configuring Profiles

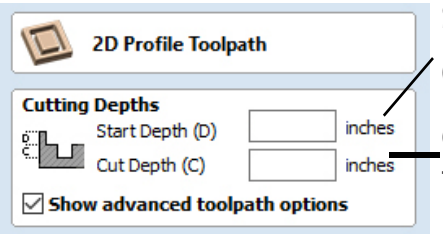
This will open a menu with a series of options to configure. Start from the top and work your way down one section at a time.

Select the **Profile Toolpath** on the right toolpath menu.

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



Cutting Depths

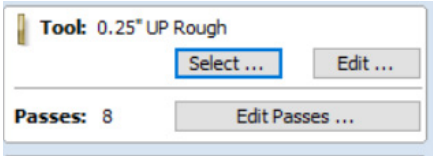


Start Depth(D) should always be set to 0.0 inches.

(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.

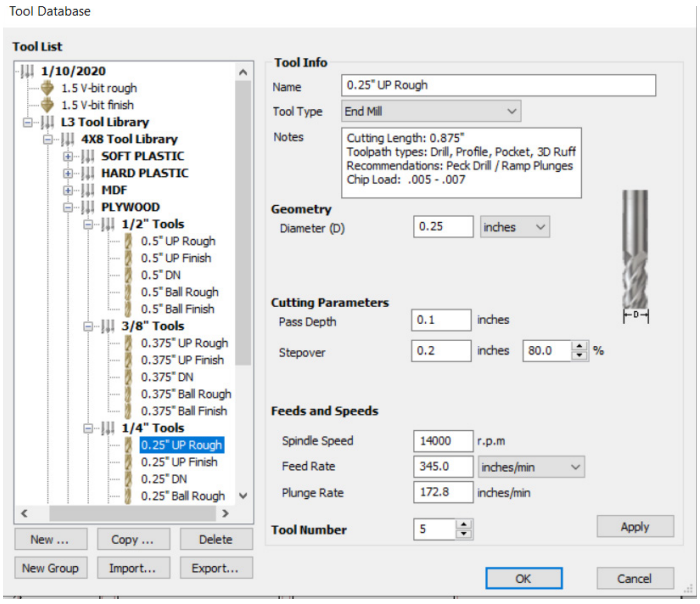
Tool Selection



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.

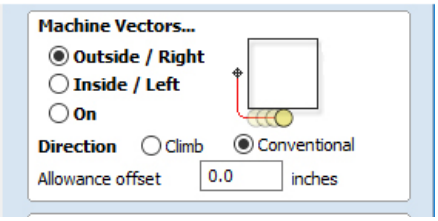
Up Rough tooling is used for most general purpose operations.

Ball Finish tooling is generally used for 3D finishing.

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

Click **OK** to continue.

Machine Vectors...



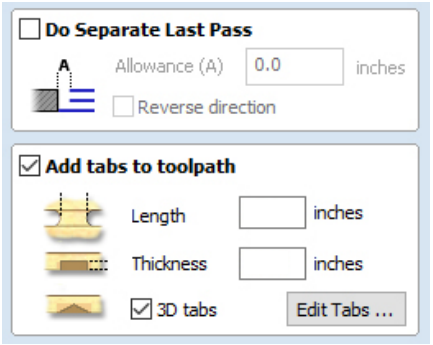
This defines how the machine will follow the selected vector.

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.

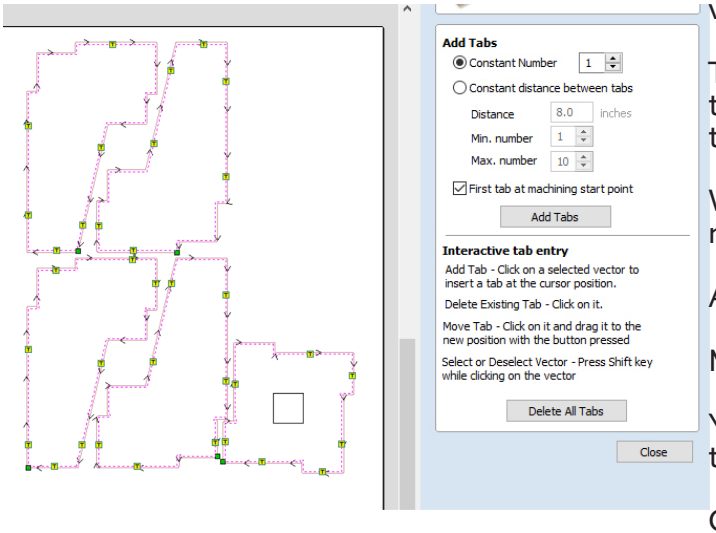
Length depends on how many tabs added 0.75 is a good average

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.

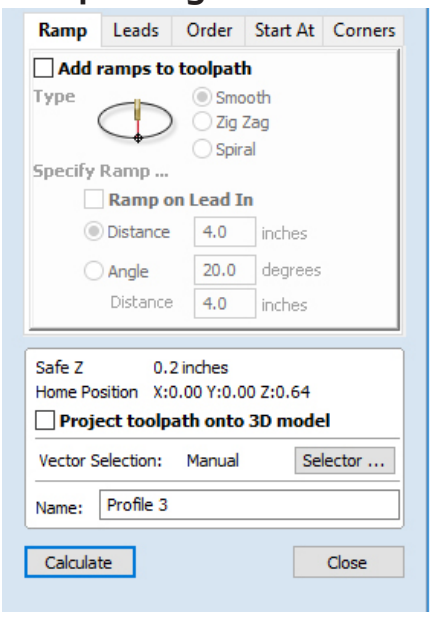
Avoid corners and internal curves.

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.

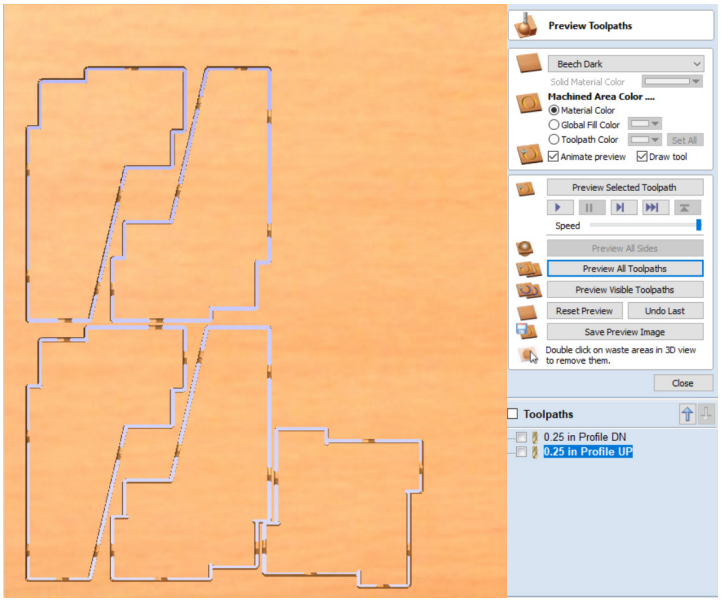
This will increase the cut time and is not supported on the desktops.

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.

Step 5: Simulate and Save



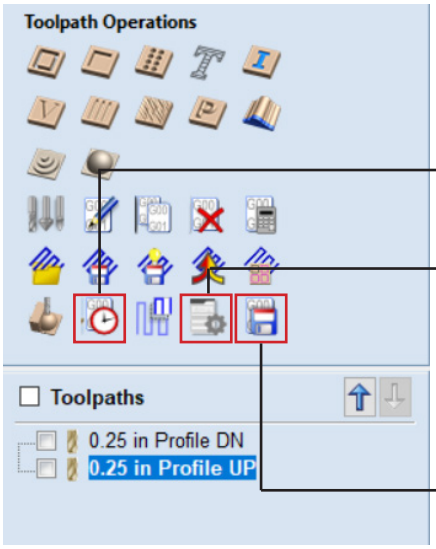
After it finishes calculating, you can simulate & preview the results.

Click **Preview Visible Toolpaths**

If you want to edit the profile 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.

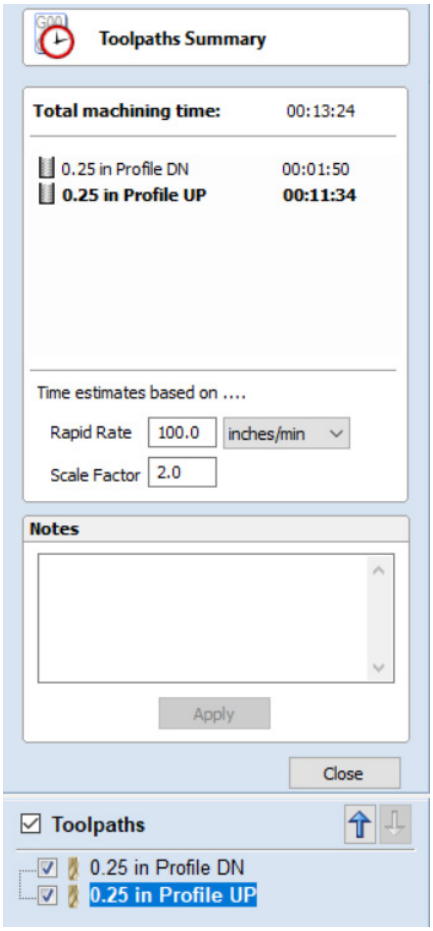


These saving options are found on the right side menu.
-Found under **Toolpath Operations**

Toolpath Summary this is where you can get a cut time estimate.

Create Job Sheet this is where you can save a quick browser friendly job summary. Please use naming convention: **MMDD_FirstName_LastName**

Save Toolpaths this is where you save the cut file.



Toolpath Summary

Total machining time: 00:00:00

Tools Used

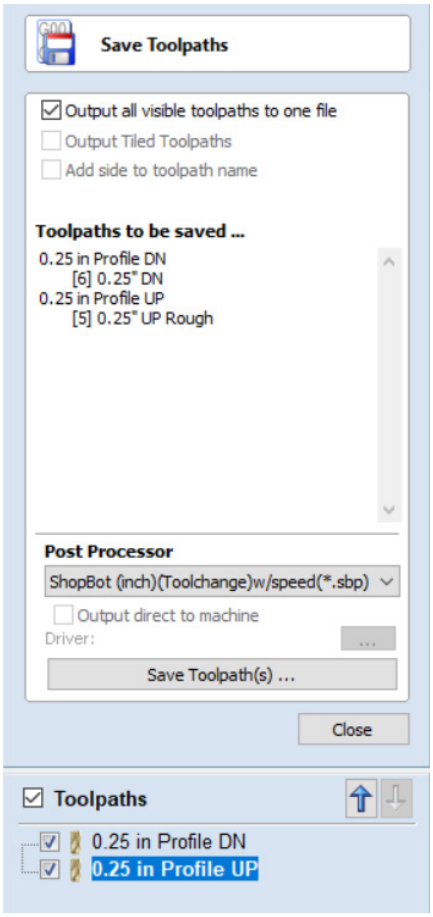
Time estimates based on...

Rapid Rate: 100.0 inches/min
*make sure this is always set to inches unit

Scale Factor:
*set to 2.0 if mostly 2D file

Make sure to check all tools in order to get an accurate cut time estimate.

Click **CLOSE** when done



Save Toolpaths

Check box to “Output all visible toolpaths to one file”

Preview: Toolpaths to be saved...

-this will show:
-tool type
-tool size
-tool number

Make sure the Post Processor is Shopbot (inch) (Toolchange)w/speed(*.sbp) or Shopbot TC

Click **SAVE Toolpath(s)...**
*Use naming convention
MMDD_FirstName_LastName

Click **CLOSE** when done