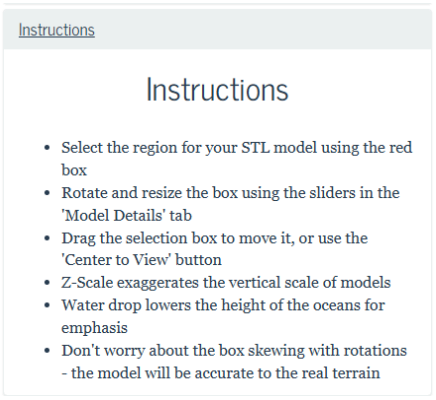
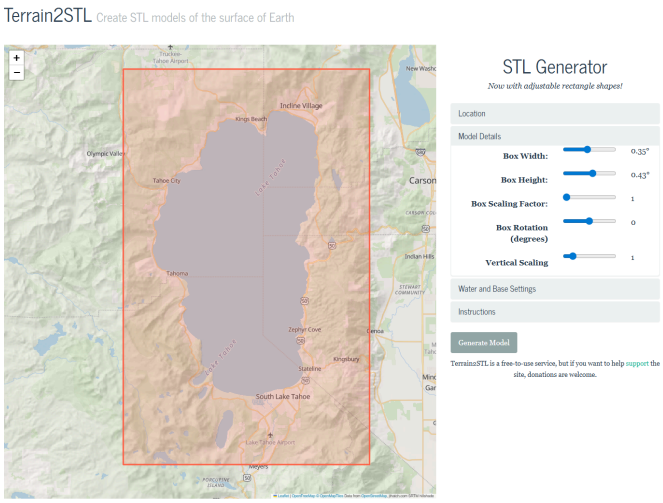
Topographic Maps Workflow - Using Terrain2STL with Aspire Vectric 12.5 and Makera Cavera ATC

Part 1: Terrain2STL

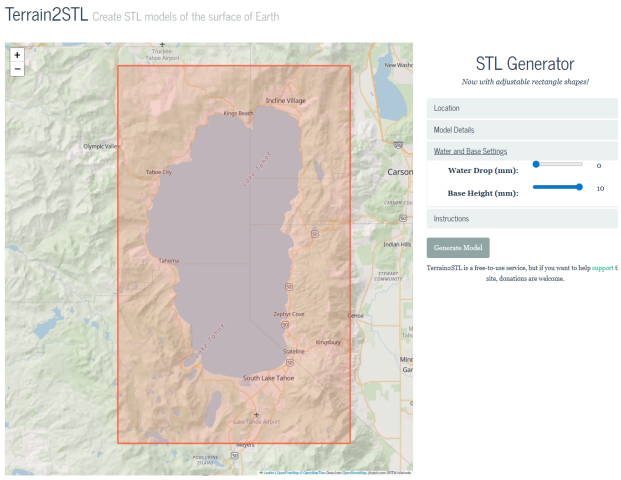
1. **Access the Website:**



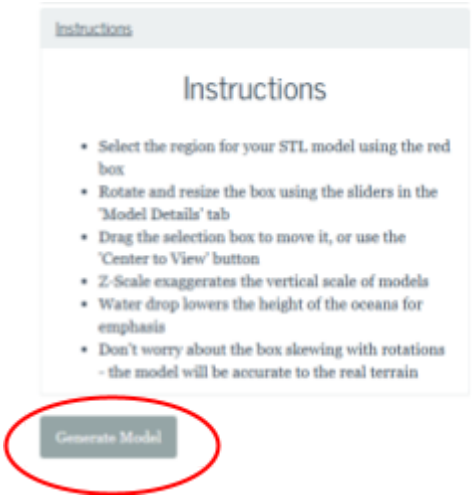
   Navigate to https://jthatch.com/Terrain2STL/.
2. **Find Your Location:**
   Use the map interface. You can click and drag to pan the map to the geographical area you are interested in. Under **Location**, **Click** on **Center to View**. This will create a red box.
3. **Define the *Model Area*: (see image below)**
   A red box on the map shows the area that will be turned into a 3D model.
   - **To Move the Box:** Click and drag the red box to the exact location you want.
   - **To Resize/Rotate the Box:** Use the sliders in the "Model Details" section (Box Width, Box Height, Box Rotation) to adjust the shape and orientation of your selection.



4. **Adjust Model Settings:**
   Before generating the model, you can fine-tune the settings in the "Model Details" and ***"Water and Base Settings"*** sections: **(see image below)**
   - **Vertical Scaling:** This slider (also called Z-Scale) exaggerates the height (mountains, valleys) of the terrain. A higher value makes the vertical features more pronounced. It is recommended to start with the default of 1 and adjust as necessary from there.

- **Water Drop (mm):** This lowers the height of water bodies (like oceans) to make coastlines stand out more. If you have water, it is recommended to start with the default of 0 and adjust as necessary from there.
- **Base Height (mm):** This adds a solid base underneath the terrain model, ensuring it's a solid, printable object. It is recommended to start with the default of 1 and adjust as necessary from there.
- 

5. **Generate the Model: (see image below)**
   Once you are satisfied with your selected area and settings, **click** the **Generate Model** button. The website will process the terrain data. The Generate Model button will begin generating and downloading your terrain .STL file as a .ZIP file.

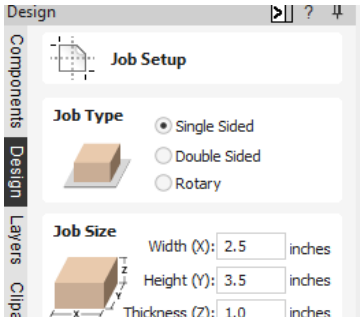


Part 2: Aspire Vectric 12.5 and Makera Cavera ATC

Phase 1: Job Setup & STL Import

(CAD: Computer Aided Design)

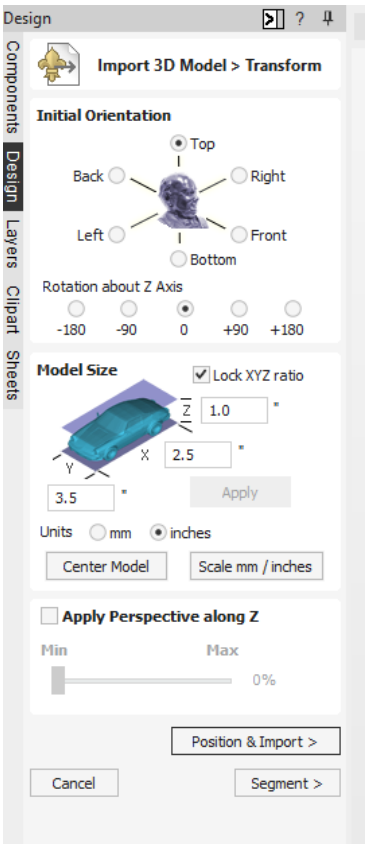
This phase is about preparing your workspace and bringing in your 3D model.



1. **Create New File:** Open Aspire.
2. **Define Material: (see image to the right)**
   a. Set the **Job Type** to Single Slided.

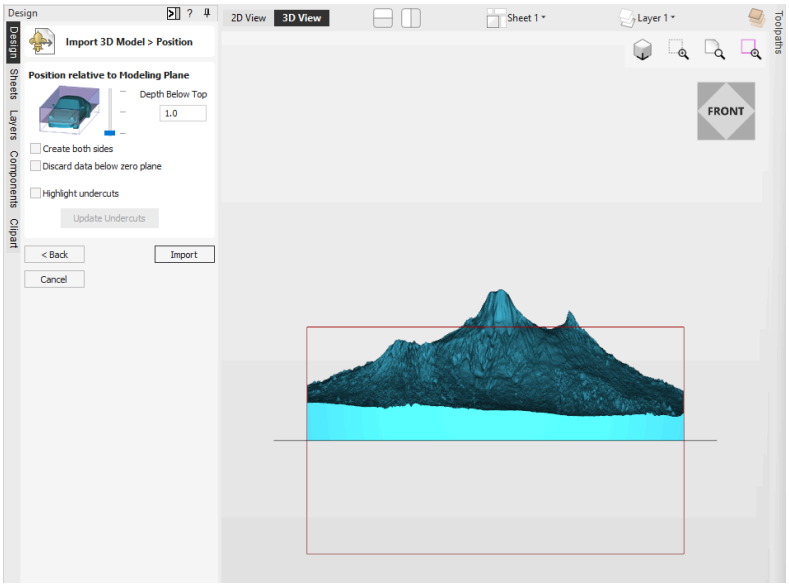
    b. Set your **Job Size (X, Y)** to match your physical stock. The **X axis** is to the right and left of the bed while the **Y axis** is to the back of the Carvera machine when looking at the bed.
    c. Set your **Thickness (Z)**. This is critical. Your imported STL model's height cannot be thicker than this value. [*As a general rule, this value should be the height (thickness) of your stock*.] See the example image to the right.
3. **Set Zero Origin:**
    a. **Z Zero Position:** Material Surface (top) for the Cavera.
    b. **XY Datum Position:** BottomLeft is often easiest when working with a single 3D model, as it helps you center the component easily.
4. **Set Model Resolution:**You can set the resolution to High or Very High for best 3D quality, but the software will run slower. It is okay to leave the resolution at **Standard** (fastest).
5. **Click OK**.

6. **Import 3D Model:**
    a. Go to the **Modeling tab** (the one with the blue/green shapes).
    b. Click the **"Import a Component or 3D Model"** icon (looks like a folder with a blue arrow).
    c. Select your .stl file.
7. **Orient 3D Model -**
    a. **Imported 3D Model > Transform**
        i. This is the **most important step** for an STL workflow. A new 3D orientation window will appear.
        ii. **Initial Orientation:** Set the Initial Orientation to **Top**. Leave the **Rotation about Z Axis** at “0” or change it to move the orientation of your 3D model.
        iii. **Model Size:** Adjust the model's **Width, Height, Depth**. Use the "Lock" icon, which needs to be unchecked, to change the X, Y, and Z sizes individually to the stock size. Use the appropriate size for your stock. In our example, stock is 1”. [*As a general rule, the Z height* **or** *be less than or equal to the thickness of your stock for the Carvera.*]
        iv. **Click Apply and then Center Model.**
        v. **Apply Perspectiver along Z:** Ignore. Leave it unchecked. **Click Position and Import>**.



b. **[Beginner]-Import 3D Model (.stl file) > Position:**

i. Next, using the slider bar position model Depth Below above the purple horizontal cutting plane. In other words, make sure your model's Z height size is equal the Depth Below size (Red circles below).



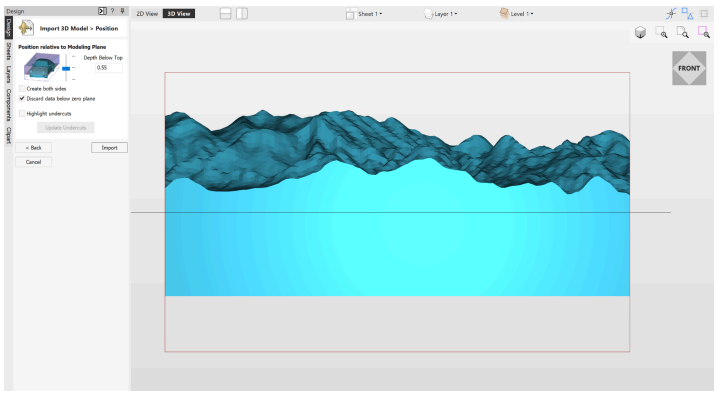
ii. Using the view cube the top right corner the screen, **Click** on **FRONT VIEW** to the view of your model compared to the cut line.

The pictures below are for your conceptual understanding, but these are separate images. You will NOT see these all together.

iii. **Click Import** while on the **Position relative to the Modeling Plane**.

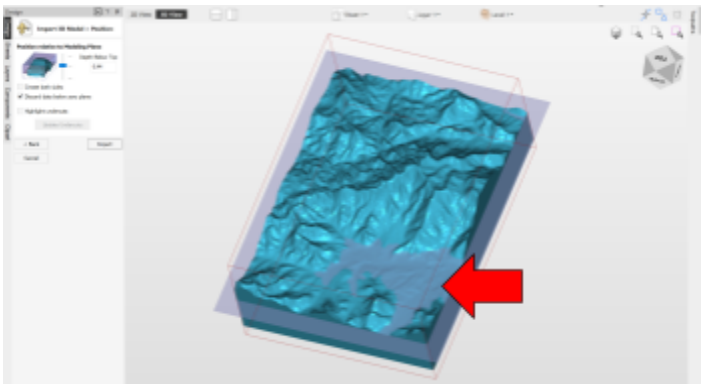
c. **[Advanced] - Import 3D Model (.stl file) > Position:**

i. Next, using the slider bar, position the purple horizontal cutting plane at the appropriate depth to maximize the vertical relief of your model while maintaining a reasonable base height of the model. In other words, make sure that your model's Z height size is slightly below the lowest relief point of your model. (Red circles below).



ii. Using the view cube in the top right corner of the screen, **Click** on the **FRONT VIEW** to see the view of your model compared to the cut line.

iii. Again, using the view cube, **adjust the view cube to an isometric view** to be able to view your model to be sure you are not cutting away part of the bottom relief like in the image above.



Your STL file is now imported as a single 3D "Component" in your Component Tree. You will see it in the 3D view.

Phase 2: Sizing your Component

Now you must position the 3D model towards the top of the stock.

1. **Click on the Component tab:**
    - Click on the name of your .STL file name to access the **Component Properties**.You may need to right click on the name of the imported .STL file that is under **Level 1** below the **Components** section. You can also access this by right clicking then selecting the **Properties.**
    - Click the **"Shape Height"** and change the value to **1.0**. This will make the Z height of your model more pronounced. Next, Click the **"Base Height"** and change the value to **0.25**. This will raise the height of your imported model in the material so that it is closer to the top of stock
    - **Click close.**

Phase 3: Positioning & Creating a Boundary

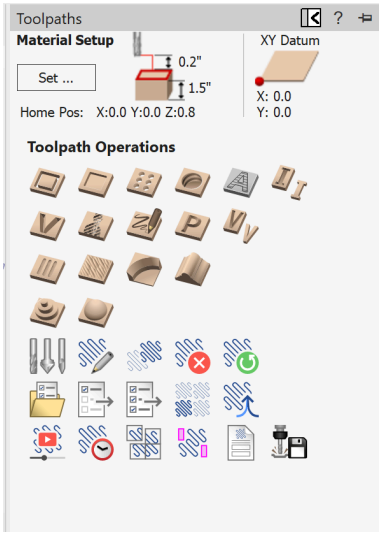
Now you must position the 3D model within your material and tell Aspire the *exact area* you want to machine.

2. **Position the Component:**
    - Switch to the **Design tab**. Now **click on the 2D view**.
    - Click the **Alignment Tool** (under Transform Objects). Click the **"Center"** tool. This will perfectly center your imported model in the material you defined in Phase 1.
    - Click Close
3. **Create a Boundary:**
    - While on the 2D view go to the **Create Vectors section** under Design.
    - Click the **" Draw Rectangle"** icon. 
    - You will create a rectangle around the boundary of your model. This rectangle will be the size of your stock, such as 2.5” (X) by 3” (Y).
    - **Why this is crucial:** This profile is what you will use to trim your finished product.

Phase 4: 3D Toolpath Generation (CAM: Computer Aided Manufacturing)

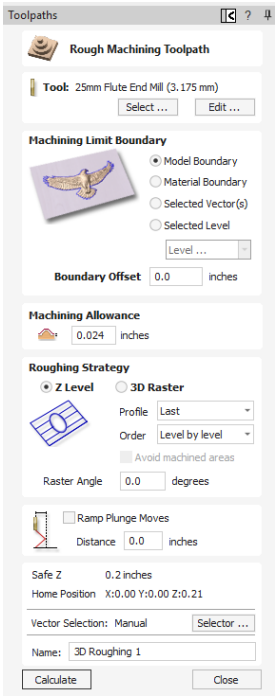
This is where you create the actual cutting instructions. For a 3D model, this is almost always a two-part process: a "Roughing" pass and a "Finishing" pass.

1. **Switch to the Toolpaths Tab** (top right). **(see image to the right)**
2. **Select Your Boundary:** In the 2D view, click on the 3D model image.
3. **Pin the Toolpaths Tab Open** using the "pin" icon so it doesn't close.

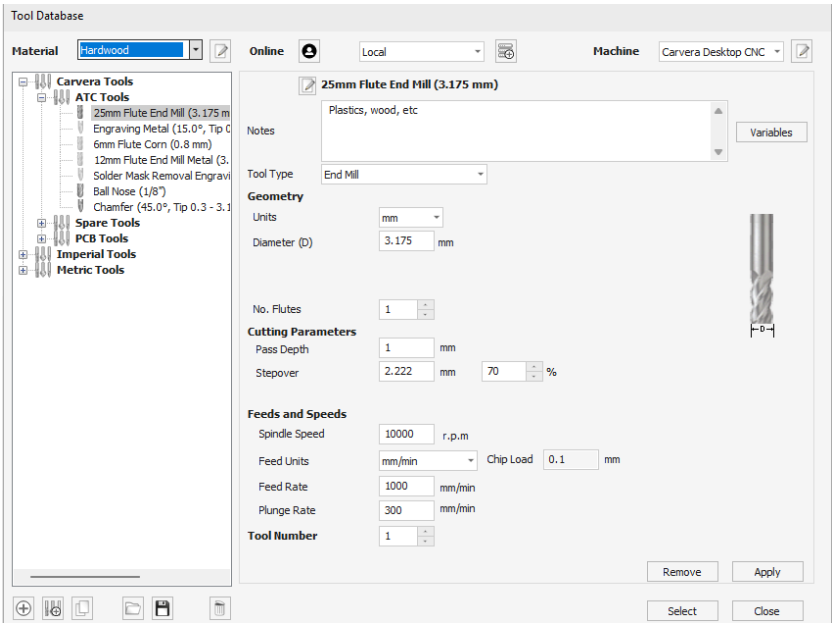


Step 4A: The 3D Roughing Toolpath (Clearing Material)

The goal here is to quickly remove the bulk of the "unused" material with a large, strong bit.

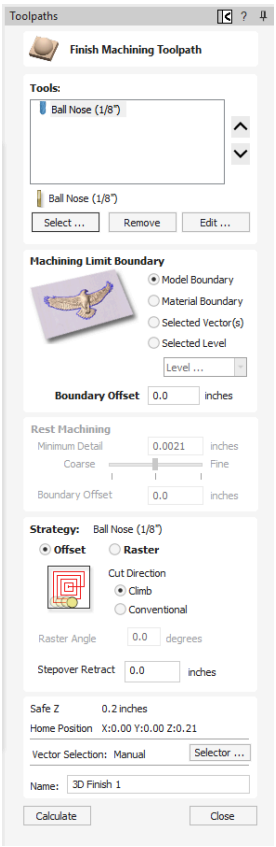


1. Click the **3D Roughing Toolpath** icon. 
2. **Tool:** The Material needs to be set to Hardwood. Select a large 25 mm Flute End Mill (3.175 mm) under the Carvera Tools subsection Example Tools. This is also known as **1/8**” **End Mill**. This is set to **Tool Number 1** for the ATC**. (see image below)**
3. **Machining Limit Boundary:** Select Selected Vectors. This tells Aspire to *only* machine the area *inside* the vector you have selected.
4. **Machining Allowance:** Set a small value (e.g., 0.024"). This leaves a thin "skin" of material for the finishing bit to clean up, preventing it from "chattering" or breaking.
5. **Strategy:** Choose Z Level (efficient for "stairstepping" down) or 3D Raster (good for flatter models).
6. **Name your toolpath** (e.g., "3D Rough - 0.25 Endmill or 25 mm Flute End Mill") and click Calculate. **(see image on the right)**



Step 4B: The 3D Finishing Toolpath (The Detail Pass)

The goal here is to use a small bit to slowly go over the entire model, creating the smooth, detailed final surface.



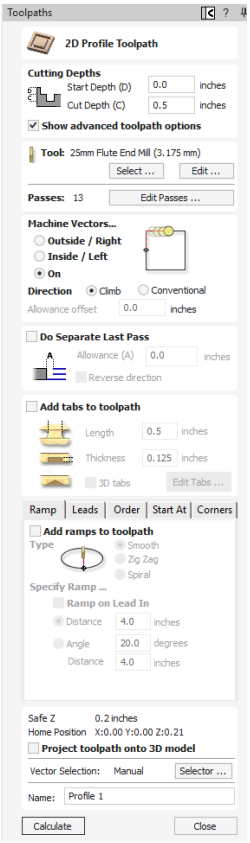
1. Click the **3D Finishing Toolpath** icon.  (Your boundary vector should still be selected).
2. **Tool:** The Material needs to be set to Hardwood. Select a small **⅛" Ball Nose** bit. The smaller the bit, the more detail you get, but the *longer* it takes.This is set to **Tool Number 6** for the ATC. **(see image below)**
3. **Machining Limit Boundary:** Again, select Selected Vectors.
4. **Strategy:**
    - Raster: A good, all-around strategy. Set the Raster Angle (e.g., 0 degrees) to go back and forth along the X-axis.
    - Offset: Good for models that are circular or oval.
5. **Name your toolpath** (e.g., "3D Finish - 0.125 Ballnose") and click Calculate. **(see image to the right)**

Phase 5: 2D Profile Toolpath Generation (CAM)

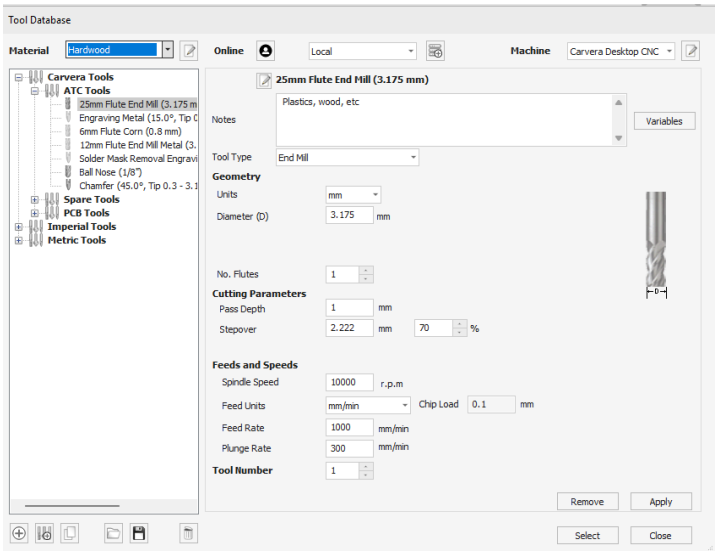
This is where you create the actual cutting instructions. For a 3D model, this is almost always a two-part process: a "Roughing" pass and a "Finishing" pass.

1. **Select Your Boundary:** In the 2D view, click on the **rectangular** model profile.
2. **Toolpaths:** Select the 2D Profile Toolpath. 

The 2D Profile Toolpath

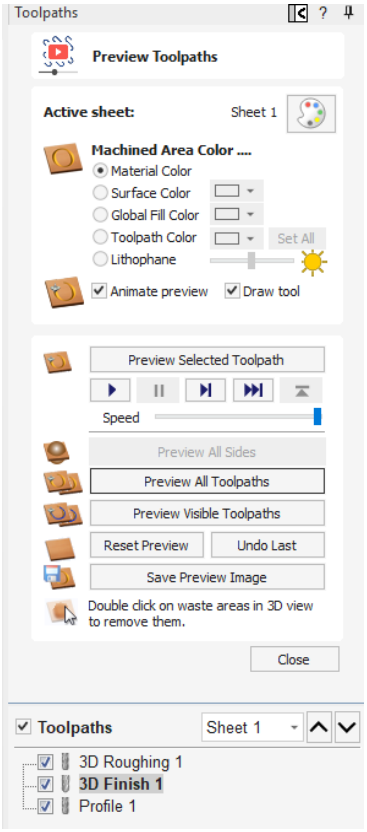


3. Click the **2D Roughing Toolpath** icon. 
4. **Cutting Depths:** Make the Star Depth 0 and the Cut Depth 0.5. This tells Aspire to *only* machine the specific depth.
5. **Tool:** The Material needs to be set to Hardwood. Select a large 25 mm Flute End Mill (3.175 mm) under the Carvera Tools subsection Example Tools. This is also known as ⅛” **End Mill**. This is set to **Tool Number 1** for the ATC. **(see images to right and below)**

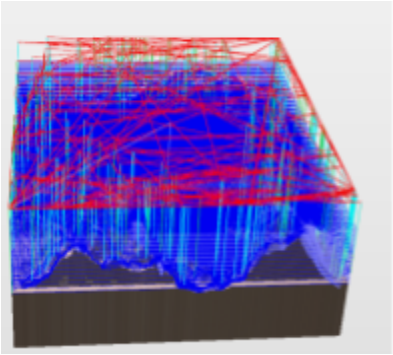


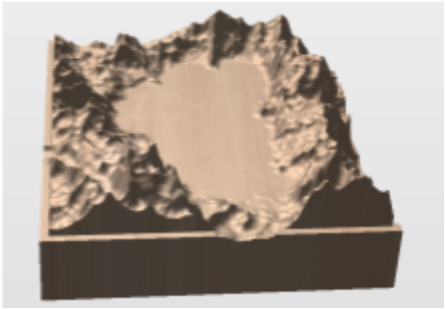
6. **Machine Vectors:** Select **On** the line and the Direction of **climb**.
7. **Do Separate Last Pass:** Ignore it. Leave it unchecked.
8. **Add tabs to toolpath:** Not Needed.
9. **Name your toolpath** (e.g., "2D Profile - ⅛” Endmill" or 25 mm Flute End Mill) and click Calculate. **(see image below)**

Phase 6: Simulation & Exporting

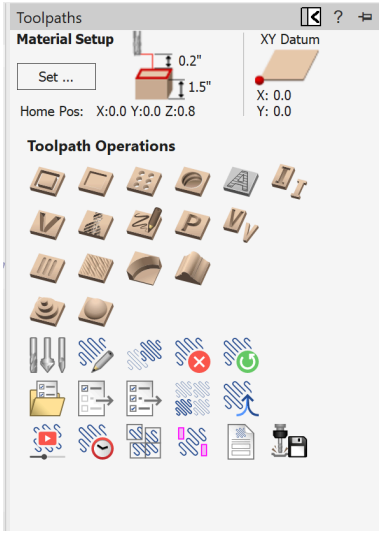


1. **Preview ALL Toolpaths**  **:**
   - This is your most important safety check. Click the **Preview all Toolpaths** button. **(see image to the right)**
   - Aspire will run a full 3D simulation. First, you'll see the Roughing pass "hog out" the material in steps. Then, you'll see the Finishing pass clean it up. You can preview each file independently, together, etc. **(see images below)**
   - **Check for errors:** Does it look correct? Did you miss any spots? Is the detail what you expected? The images below are examples of the Preview Toolpaths. **(see images below)**

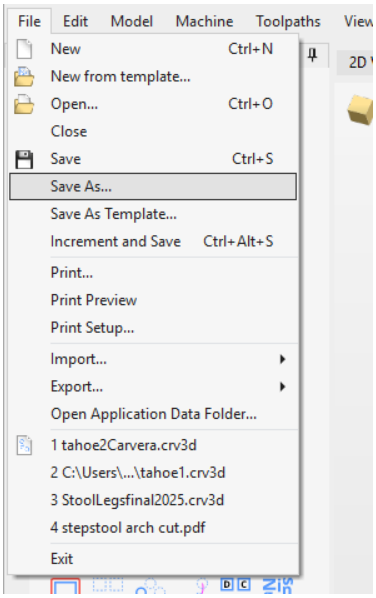




   - **Toolpaths Summary**  **:** Click on the Toolpaths Summary icon to learn how long your total machining time will take to complete your file. **(see image to the right)**



2. **Save Your Project File:** While u Go to File > Save As... and save your .crv3d project file. This saves all your work you have completed as an Aspire VCarve file to the computer. **(see image below)**



3. **Save Your G-Code (Toolpath Files):**
    - Click the **Save Toolpaths** button (floppy disk icon). 
    - This workflow creates **Visible toolpaths to one file** for your machine.
    - ***Check Toolpaths:*** *Select* "3D Rough," then select "3D Finish", and then select “2D Prolfie”. -> Click Save Toolpath(s).
    - Choose your **Machine -> Carvera Desktop CNC Machine.**
    - Choose your **Post-Processor**(the "driver" for your Carvera ATC CNC machine) → **Carvera ATC (mm) (*cnc)** -> Click Save Toolpath(s). This will save the .gcode as a .cnc Makera Carvera file to the computer. **(see image to the right)**
    - You will now have the .gcode (or .nc, etc.) file to run at your CNC machine, as a file that will automatically run each toolpath one after another.