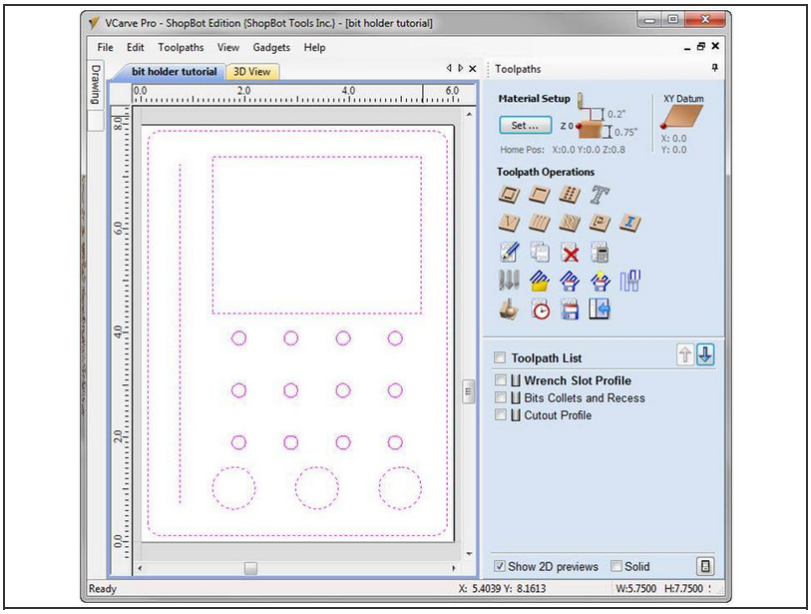
Create a Part - Section 2: Toolpaths and Saving Part File

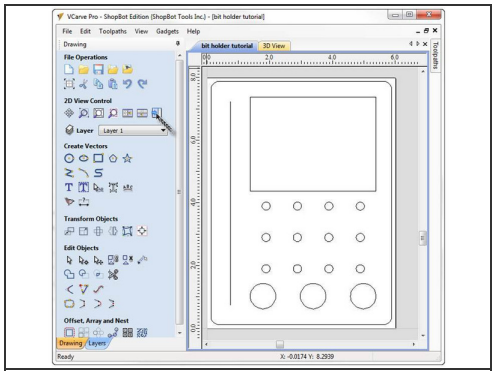
Written By: Nathanael

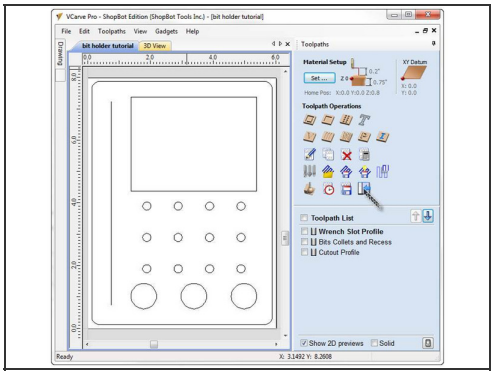
Step 1 — Toolpath Overview

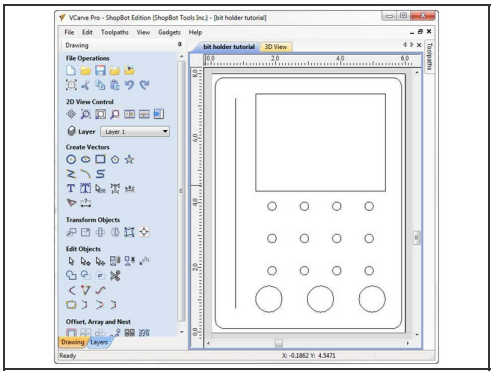


- Part 1 of this series showed how to draw vectors for a part design. In the next step, the vectors will be used to assign paths for the router bits to take.
- Two main types of tool paths will be used on this part.
- **Profile tool path**: The tool will follow a vector path to create a channel or a shape. This part uses two profile cuts; one for the wrench slot, and one to cut the piece from the remaining material.
- **Pocket tool path**: The tool will remove all material within a vector boundary. This part will have one pocket tool path operation for the rectangle and the holes.

Step 2 — Switch to Toolpath Tab

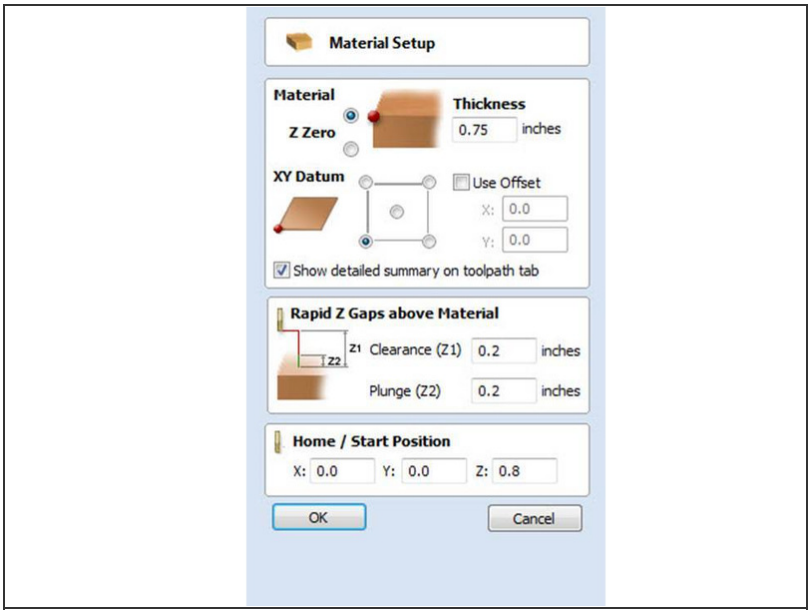






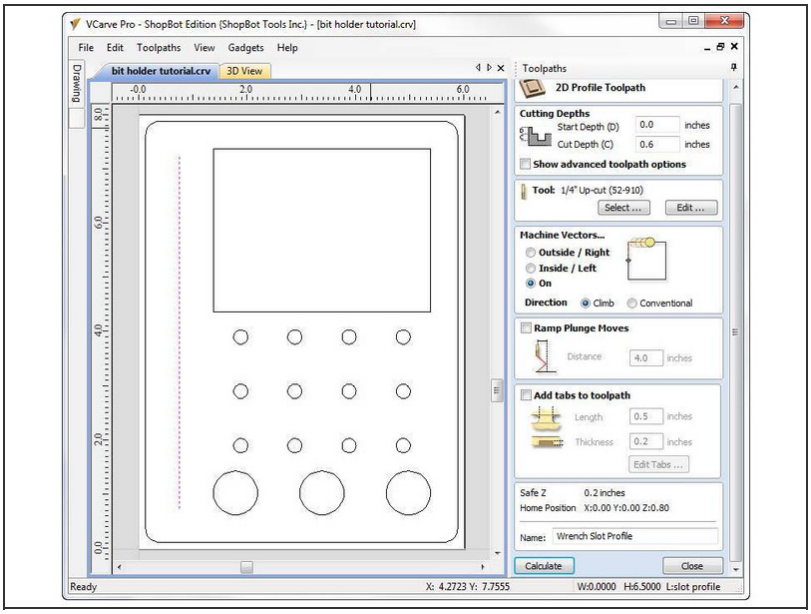
- Switching between the **Drawing** and **Toolpath** panes is simple. Under **2D View control** look for a box with a blue arrow. In the **Drawing** pane moving the cursor over this icon will show **Switch to toolpaths tab**.
- Clicking the tab will close the **Drawing** pane and open the **Toolpaths** pane on the other side of the VCarve Pro window.
- There is a corresponding icon on the **Toolpaths** side that switches to the **Drawing** pane. This is useful if a vector needs to be edited.

Step 3 — Material Setup



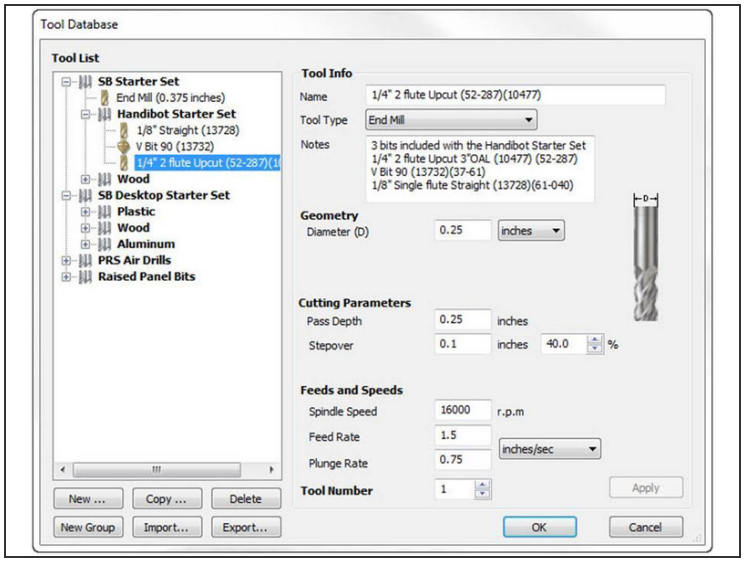
- Select **Set** to bring up the **Material Setup** pane. This pane shows settings that effect the start point of the bit, where the bit should be zeroed, and how far about the material the bit will travel when not cutting.
- Usually Handibot part files should be created with the **Material Z Zero** at the top. Make sure the top option is selected for this part file.
- The **Thickness** should be **0.75 inches**. This was entered during **Job Setup** in Part 1.
- **XY Datum** establishes the reference point for the bit. The lower left corner should be selected and corresponds with the front left corner of the Handibot.
- **Rapid Z Gaps above Material** sets the distance the bit will move about the bit when not cutting. The default, **0.2 inches** is optimal for this part. Parts using clamps or other potential obstructions would require a higher value.
- **Home/Start Position** defines where the Handibot will start each toolpath. Typically X and Y should be 0 and Z should be 1 or less.
- Once all values are verified, select **OK**.

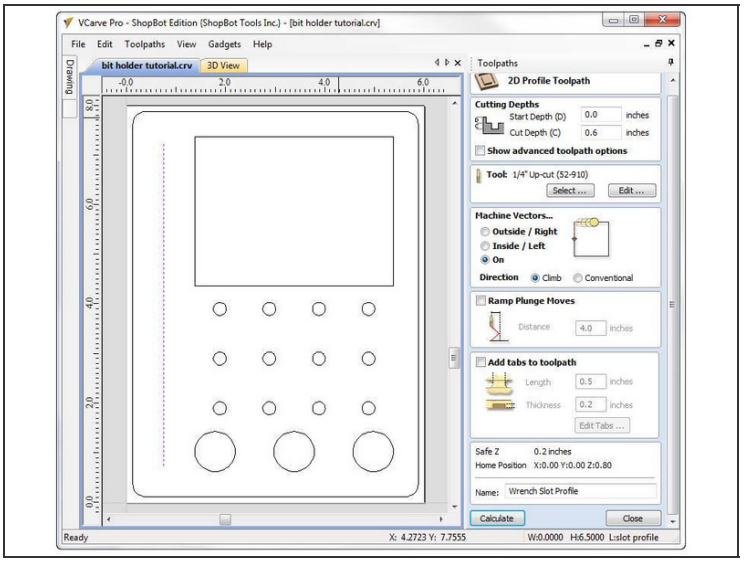
Step 4 — Wrench Slot



- The first path will be a profile tool path that follows the line on the left of the part. It will form a slot to place the collet wrench in.
- Select the line at the left of the part. It will turn magenta.
- Click on the **Profile Toolpath** icon. It is the first icon on the left under **Toolpath Operations**. The **2D Profile Toolpath** pane will open.
- The first parameter to enter is the cutting depth of the bit you'll use. **Start depth** is the distance into the material the cut will start. For this part it should be **0 inches**.
- The **Cut Depth** of this toolpath will be **0.6 inches**.
- This toolpath does not require advanced toolpath options.

Step 5 — Wrench Slot - Tool Selection



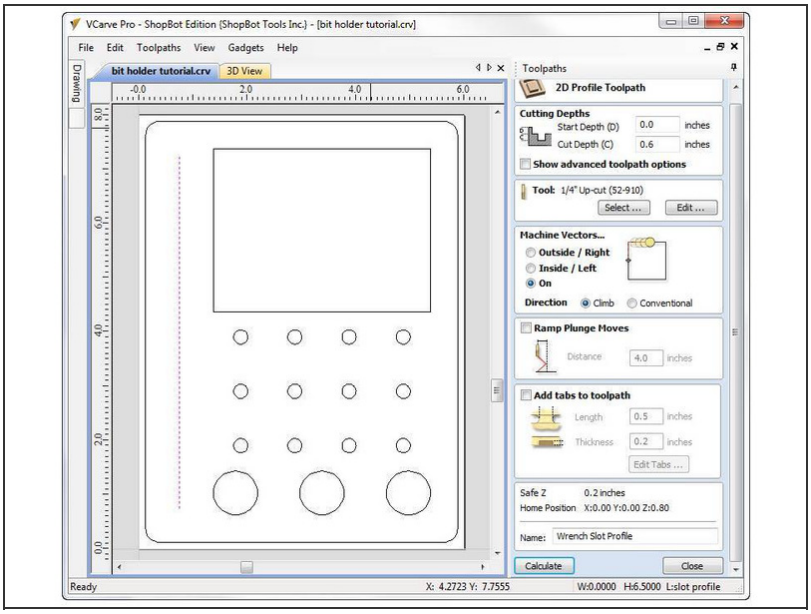


- In the **Tool** section, press **Select**. The **Tool Database** window should appear.
- Expand the **Handibot Starter Set**, and select the **1/4“ 2 flute Upcut** bit and click **OK**. The parameters entered during bit creation will work well for this cut file.
- The bit will now be displayed after Tool.

If you do not have the Handibot Starter Set, complete the **Edit the Tool Database** tutorial.

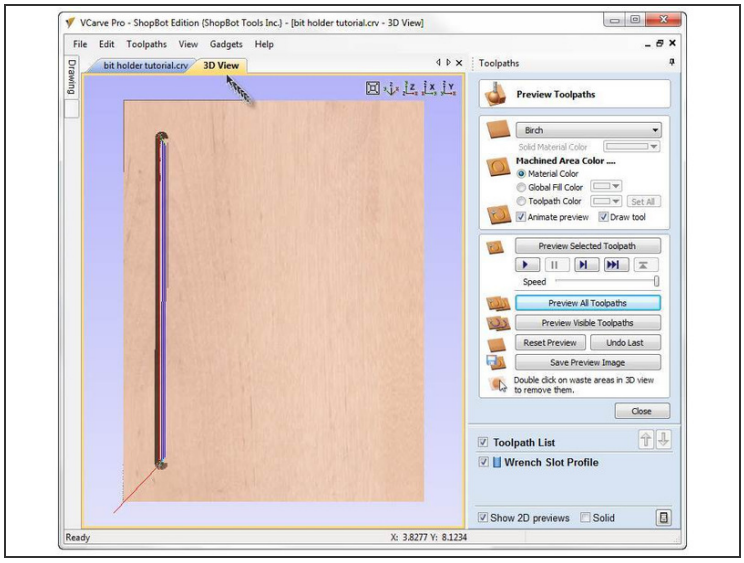
- At this point it is good practice to click **Edit** and check the tool parameters. Some tool paths require different settings than the default values. Changes made to the parameters in **Edit** will not be made to the normal tool parameters.
- It is not necessary to change any of the default values for this toolpath. Click **Cancel** to close the **Edit Tool** window.

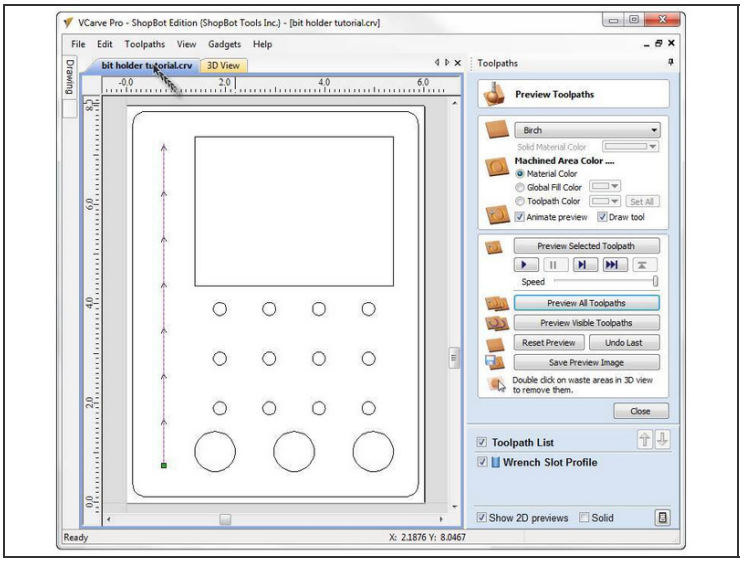
Step 6 — Wrench Slot - Final Toolpath Parameters



- **Machine Vectors** sets the location the bit will be in relation to the vector. In this case the center of the bit should follow directly on the line. The cut **Direction** is not significantly different for this tool path.
- **Ramp Plunge Moves** allow the bit to move down into the material at an angle. This is helpful for bits that do not have cutting edges covering the base, and are not capable of drilling straight down. Ramp plunge moves are not required for this tool path.
- **Add Tabs to Toolpath** allows small pieces of material to be left hold the material in place during a through cut. The topic will be covered in the last cutout profile toolpath. The tick box for this section should not be selected now.
- **Name** this toolpath **Wrench Slot Profile**.
- After reviewing the parameters, click **Calculate**. The **Preview Toolpaths** pane will appear.

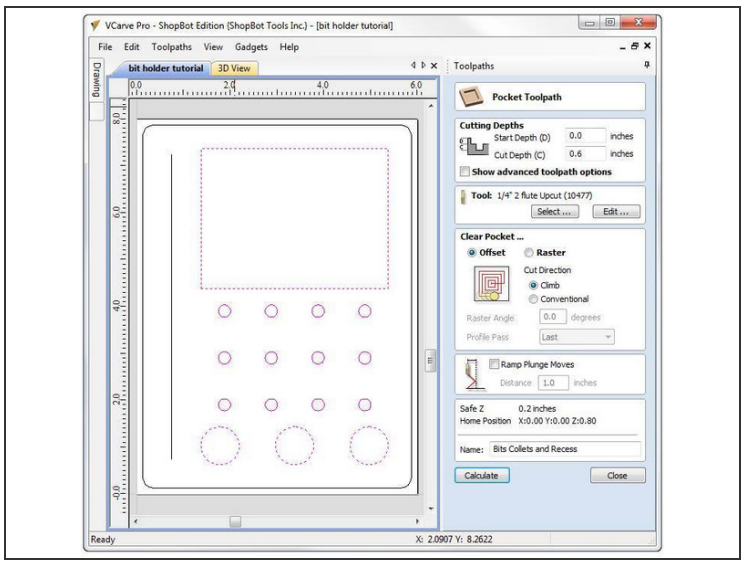
Step 7 — Preview Wrench Slot Toolpath

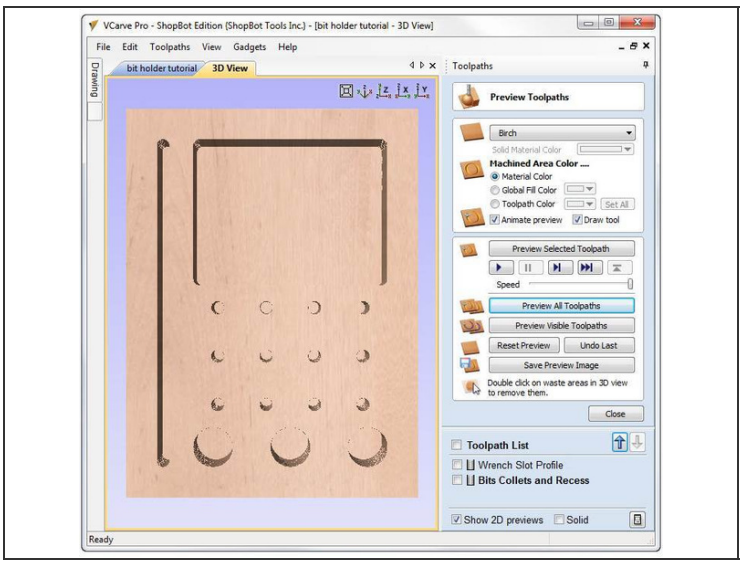




- The **Preview Toolpaths** pane provides a visual representation of the results of the toolpath.
- Click **Preview Selected Toolpath** to see the slot this toolpath makes.
- Click **Reset Preview** to clear the material.
- Toggle between the 2D and 3D tabs at the top of the display window to see both the drawing and the model.
- Click **Close** to return to the **Toolpaths** pane.

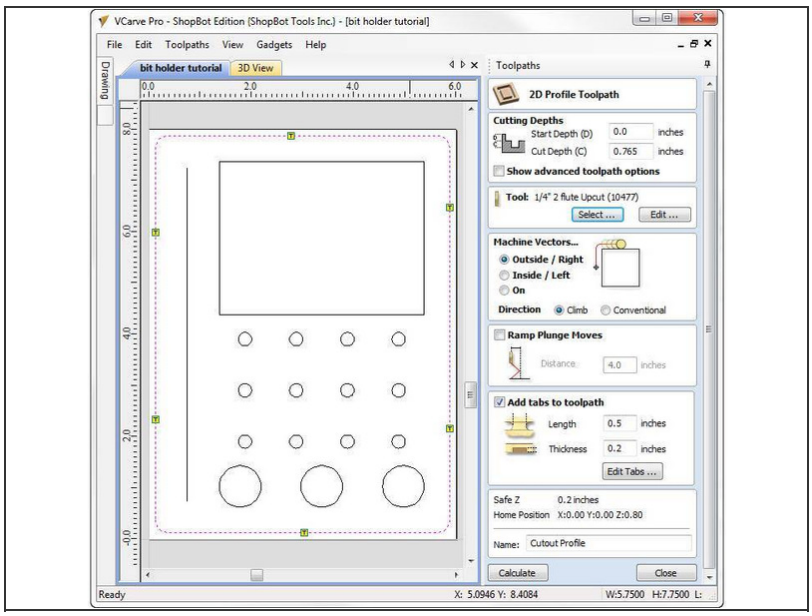
Step 8 — Pocket Toolpath Setup





- Select the **2D view** window. Click, drag, and hold to create a sweep selection rectangle around the small rectangle and all circles. Their color should change to magenta.
- Select the **Pocket Toolpath** icon. It is the second icon from the left under **Toolpath Operations**.
- Set the **Start Depth** will be **0.0 inches**, and the **Cut Depth** to **0.5 inches**.
- Select the **1/4“ 2 flute Upcut** bit and click **OK**. The parameters entered earlier will work well for this cut file. The bit will now be displayed after **Tool**.
- For this toolpath choose **Offset** and **Climb**. There is no need to use ramp plunge moves with the cutter.
- **Name** this toolpath **Bits Collets and Recess**.
- Click **Calculate** after reviewing all parameters. Use the **Preview Toolpaths** pane to review the new toolpath.

Step 9 — Cutout Profile Toolpath Setup

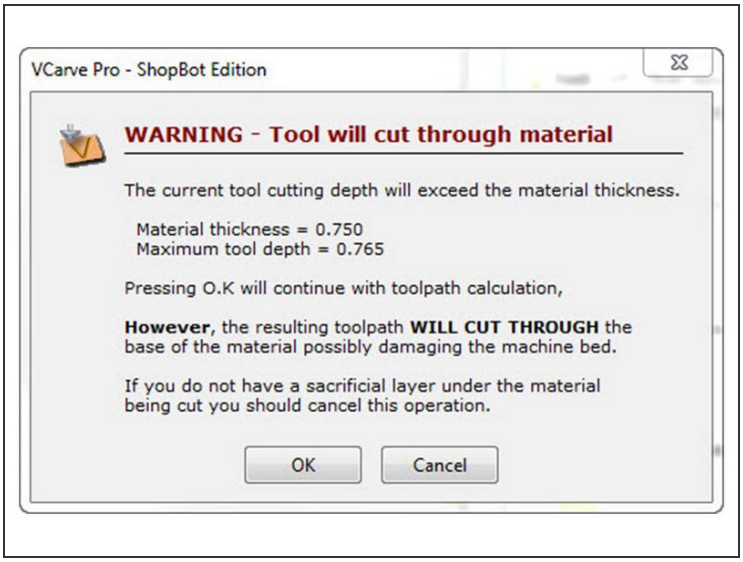


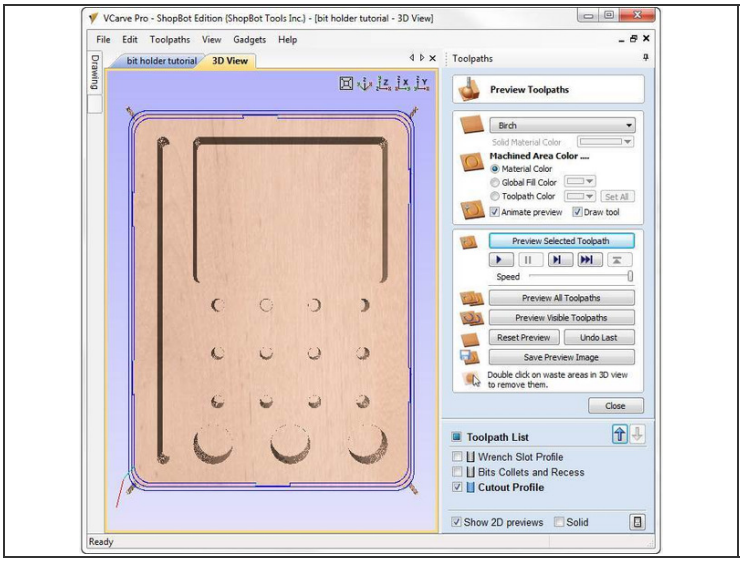
- The remaining toolpath will cut the part from the material using a profile toolpath. The path will use tabs to prevent the part from moving as it is cut from the material.
- In the 2D view select the outer rectangle.
- Click on the **Profile Toolpath** icon. It is the first icon on the left under **Toolpath Operations**. The **2D Profile Toolpath** pane will open.
- The **Start Depth** is **0.0 inches**.
- This toolpath is a **Through Cut**. It will cut all the way through the material which is 0.75 inches. To compensate for difference in material thicknesses and ensure a complete through cut, it is a good idea to add a small amount of extra cut depth.

ⓘ The total cut depth will vary depending on the bit and material choice, but a value of 0.01 to 0.02 extra is a great starting point. Note that this will cause the bit to cut slightly below your workpiece. For this reason it is a good idea to use a noncritical surface, or spoil board, below the material.

- For this toolpath set the **Cut Depth** to **0.765 inches**. This takes into account 0.75 inches of material thickness plus 0.015 inches of through cut.

Step 10 — Cutout Profile Toolpath Parameters



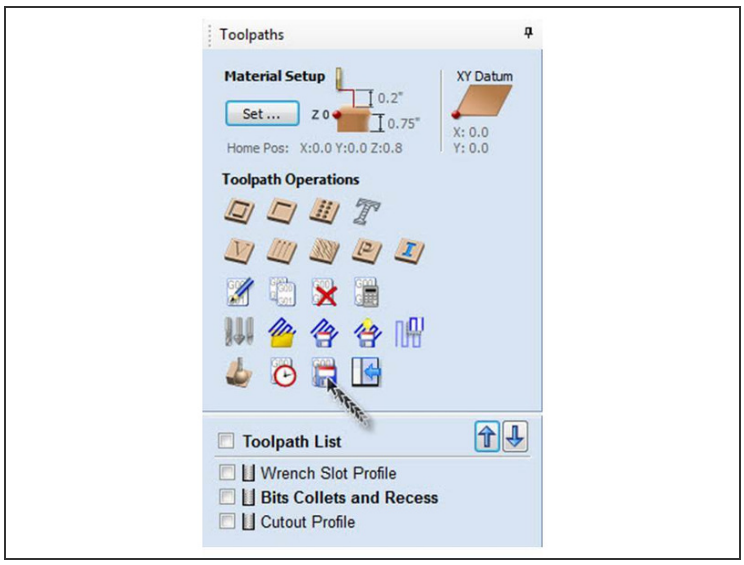


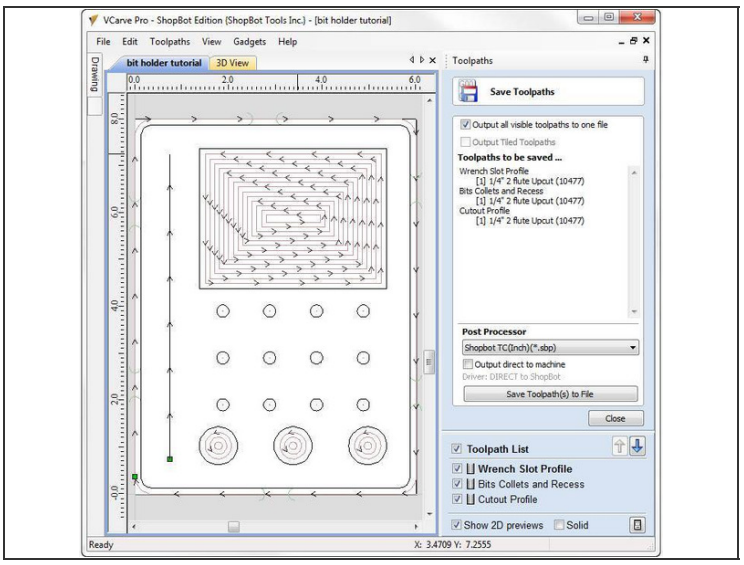
- Select the **1/4“ 2 flute Upcut** bit and click **OK**. The parameters entered earlier will work well for this cut file. The bit will now be displayed after **Tool**.
- This toolpath needs to maximize the size of the part. For this the bit should travel on the outside of the vector. Under **Machine Vectors** chose **Outside/Right**, and for **Direction** chose **Climb**.
- **Tabs** are a method of retaining the part within the extra material by leaving a bridge of material between them. Without Tabs, large parts may shift as they are cut free, leading to a small gouge or notch along the edge. Smaller parts can be destroyed or thrown by the cutter.
- Select **Add tabs to toolpath** by filling the checkbox. Yellow T icons should appear on the vector. Set the tab **Length** to **0.5 inches** and the **Thickness** to **0.2 inches**.
- Clicking **Edit Tabs** allow you to change the position, spacing, and configuration of the tabs. The default locations should be good for this part file.

⚠ Click **Calculate** after reviewing all parameters. The warning shown should appear. **This warning is normal, but read it every time it is shown.** Make sure the maximum tool depth does not exceed the material depth by more than 0.02 inches.

- Click **OK** to clear the warning window. Use the **Preview Toolpaths** pane to review the new toolpath.

Step 11 — Saving a Part File - Choose toolpaths

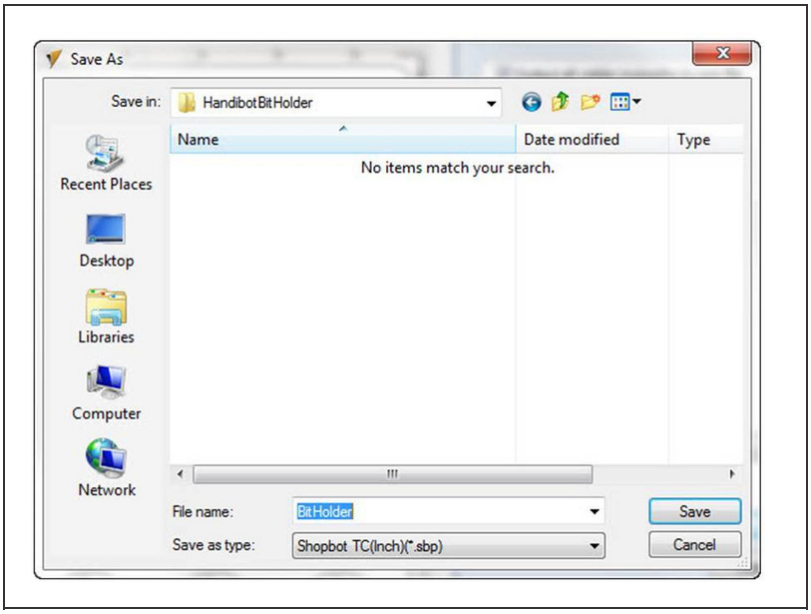




- Now that all toolpaths have been created, they must be saved in a format that the Handibot is able to recognize. This is known as **post processing** and uses a small program to do the conversion.
- On the **Toolpaths Pane** find the **Save Toolpath** icon. It is in the bottom row of **Toolpath Operations** and looks like a blue diskette.
- Look for the **Toolpath List** at the bottom of the pane. Each of the toolpaths that have been created has a check box next to it. Check each one, or check the box next to **Toolpath List** to select all at one time.
- All three toolpaths should show under **Toolpaths to be saved**. Since all tool paths use the same bit they can all be run in the same cut file.

ⓘ Note that the toolpaths will run in the order they are displayed. It is best practice to run the cutout profile last, as this optimizes rigidity for other toolpaths. To change the order of toolpaths, use the arrow buttons next to **Toolpath List**.

Step 12 — Saving a Part File - Choose a Post Processor

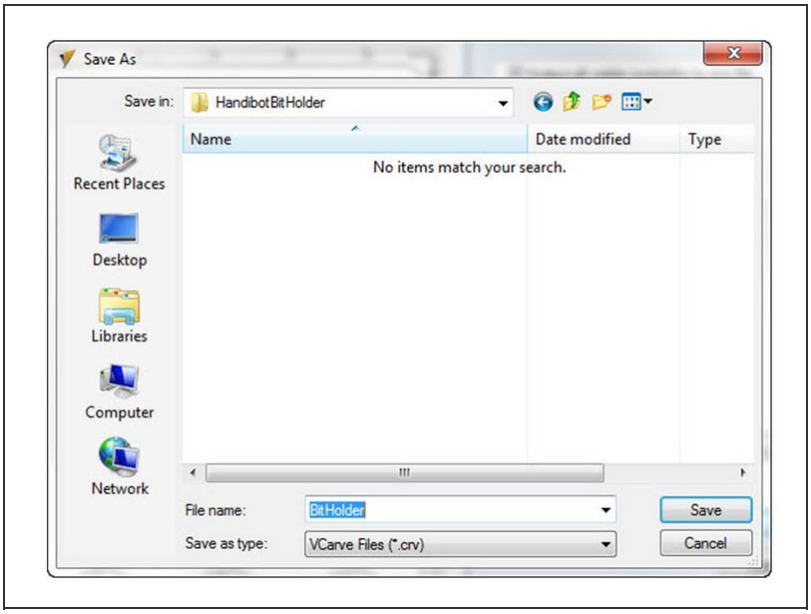


- In the **Post Processor** pull-down box, locate **Shopbot TC (Inch)(*.sbp)**. This is the most up to date postprocessor that works best with the Handibot.

ⓘ Note that while the ShopBot TC (Inch) post processor will support tool changes, it is best to save separate part files for pieces that use more than one bit.

- Save the part file in a location of your choice. Creating a folder named **Handibot Projects** with specific project subfolders is helpful. This file will save as a ShopBot Part file, and has a .sbp extension.

Step 13 — Save the Design File



- VCarve design files should be saved for future editing. The design file can be used for making edits should the part file ever need changed.
- The best place to save the design file is in the same folder as the part file for this project.
- Go to **File** at the top left of the VCarve Pro window and chose **Save**. Save this file in the same project subfolder as the part file. The design file will save as a VCarve file, and will have a .crv file extension.
- Design and toolpath work is now complete. Switch to ShopBot 3 to run the part file.

This document was last generated on 2016-01-29 10:08:29 AM.