3D Routing Guide

PART 2: 3D software, V-Carve 8

--IMPORTANT NOTE-- The 3D software now used for the 3D routing workflow is "V-Carve 8" (both 2D and 3D interfaces are now native to V-carve)

- The same steps should still apply, you will just need to open V-Carve instead of Partworks 3D
- As of this time V-Carve 8 can only be accessed from the computers in the West Mall basement.
- For the Time being Partworks 3D is still installed on the Build Lab work station, but it can only be accessed by opening an existing Partworks 3D file.

Video: CNC Training - 3D

The 3D software is a little simpler than the 2D because you typically only have to assign one toolpath for the entire surface. It does start to get complicated, however, when your model’s thickness is greater than the cut length of the bit you are using. In order to make models deeper than the cut length of the bit you are using, please see the instructions for Slicing 3D Models. However, please note that the cut length of the bit is not the same as the plunge depth of the bit. The cut length is the distance from the bottom of the bit to the CNC head. This depth varies by bit, but is typically 2 1/2-3 inches. Any model with a stepdown greater than the cut length must be sliced. This most commonly occurs at the edge of the model outline.

1. Cutting foam is much faster and much less risky than cutting wood so you will need to run a few jobs with foam before you can use other materials.

3D modeling conventions (Rhino, SketchUp, 3D scanner, etc.)

1. Check that your surface geometry is within the dimensions of your material and that it is a single, logical solid.
2. Unlike the 2D version, do not draw a box to represent your material.
3. Export your surface as an .stl (stereolithography) file

---VCarve 8 Alternate Workflow---

1. Before you can import a .stl file you must first set your material size and thickness as is described in the Enter Surface Properties section below.
2. once you have done this click "ok"

3. In the following window, you will first need to navigate to the Modeling tab.
4. and then click on the folder icon at the top of the window that opens.

---End of VCarve 8 Alternate Workflow---

Open Partworks 3D

1. Open Partworks 3D software by clicking on Start --> All Programs --> Shopbot --> Partworks 3D

2. Open the 'partworks license code & install instructions.txt' file in the Shopbot Share (\arch-data.austin.utexas.edu\shopbot - open to all UTSOA users) - copy and paste the license code into the license window.

3. Once in Partworks, go to File --> Open and select the .stl file you just exported.

4. Check that the x and y directions are the same as in your original file and that it will sit on the bed in that direction.

Enter Surface Properties

1. Check that your units are in inches—you can use the scale mm to inches button and/
or rescale your surface if needed.

2. You can reorient the surface if desired, but it is recommended that you do these type of manipulations in your 3D modeling software instead. Assuming your model was imported correctly (side to be machined is facing up), under Top Surface, select Top and under Sides to Machine, select Top.

3. Hit 'Apply' button

4. Hit 'Next' button
Enter material properties

1. Fill out the material properties menu.
2. Zero the X and Y axes to this corner
3. Enter your material’s boundaries and the thickness
4. Zero the Z axis at the top
2. ShopBot will assume material size based on the extents of your model, but you will need to adjust this to your specific material.

3. Change the thickness of material to actual thickness based on your caliper measurement.

4. Under Machining Margins, if you select ‘Symmetrical’ equal to zero the bit will cut a rectangle the same dimensions as the extents of your model at a clipping plane depth so that your surface will protrude above a ground zero cut plane. You can extend the margins by increasing the number next to Symmetrical. If you select ‘Use Model Silhouette’, the bit will leave the top surface intact so that your model is embedded within the original surface of your material, you can also increase that margin, and it will offset the outline of your model.

5. ‘Depth of Model Below Surface’ set at zero will float the highest point of your model to the top of your material. To be careful, however, it’s recommended to set that value at 0.02 in order to avoid height variations of your material.

6. Set the Cut Plane thoughtfully depending on your bit and your desired thickness.

‘Bottom’ is the most common selection.

7. Click ‘Apply’

8. Click ‘Next’

Roughing toolpath

This is not needed for foam. After you have run a few jobs with foam, you can cut wood, and you will most likely want to create a roughing toolpath (it saves time). Come talk to IO Central when you are ready for this step.

Finishing toolpath

1. Under Bit Selection, select an appropriate bit, pass depth, feed rate, plunge rate, stepover, etc. See chart labeled Material Spreadsheet attached to back of this tutorial for selection criteria.

2. Choose a raster angle that is perpendicular to the dominant grain of your surface.

3. Select Create an extra pass at 90-degrees for increased resolution

4. Click ‘Calculate’ to preview the bit’s travel paths.

5. The time estimation is not accurate unless you keep track of similar projects and establish a scale factor.
6. Click ‘Next’

**NOTE:** Increased resolution is highly dependent on the type of model and material you are using. It can be achieved with a smaller stepover rate, a smaller bit, and/or increased passes. All of these techniques will take more time. Creating an extra toolpath at 90-degrees is generally the most effective method.
Cut Out Toolpath

1. If you want the Router to cut out your model after it has routed the surface, select Create Cut Out Toolpath

2. Enter the properties similar to all other toolpaths (bit selection, pass depth, etc.)

3. Click ‘Calculate’

4. Click ‘Next’
Preview and check toolpaths

1. Click the Finishing Toolpath Preview button and confirm that the toolpath is doing what you want

2. If it looks correct, click ‘Next’

3. If something looks wrong, click the icons at the top to navigate to previous screens and change applicable settings. After you have made changes, be sure to hit Calculate, and proceed until the Finishing Toolpath Preview looks good.
Save your files

1. Select ‘Finishing Toolpath Save’ to save a .sbp file for use at the machine.

2. Also go to File --> Save to save a .3dv file so you can go back and tweak parameters in Partwork 3D.

Next: ?Bits and Speeds

?Using the CNC