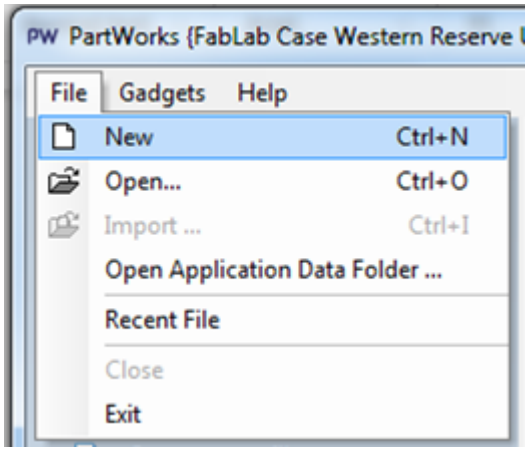
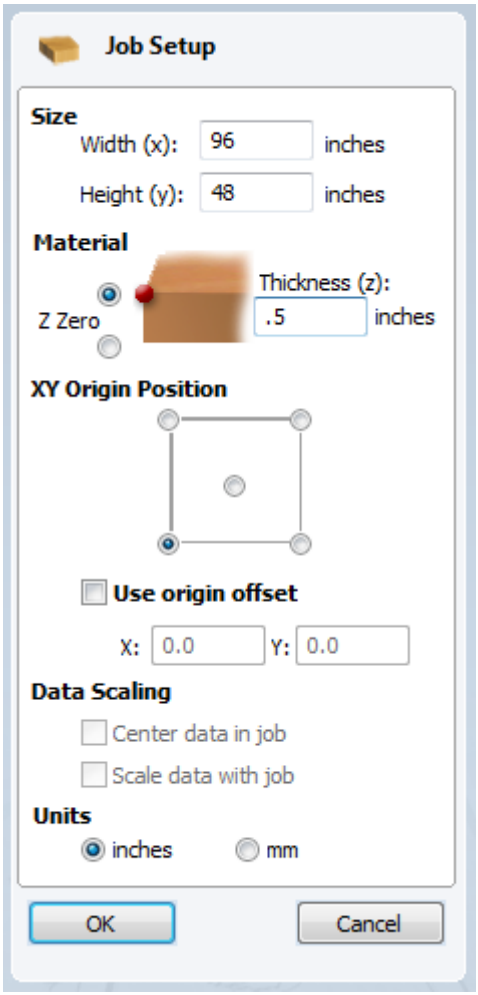
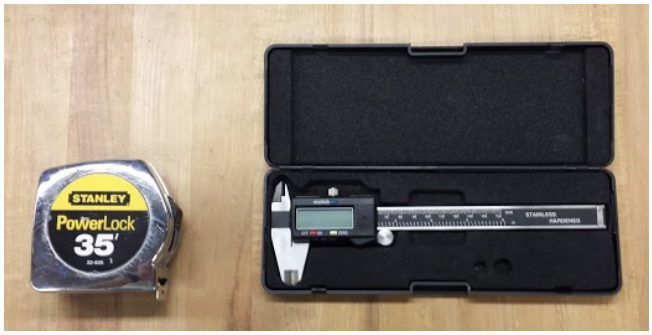
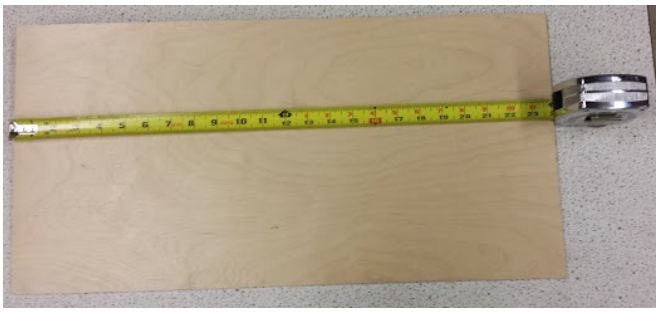
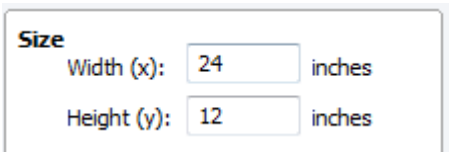
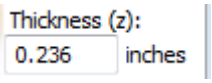
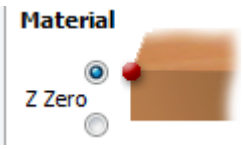
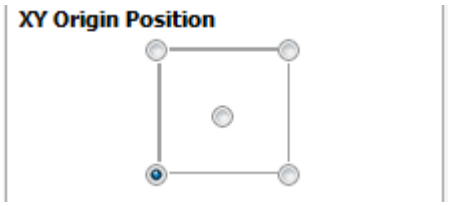
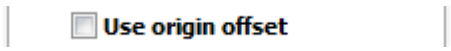
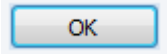
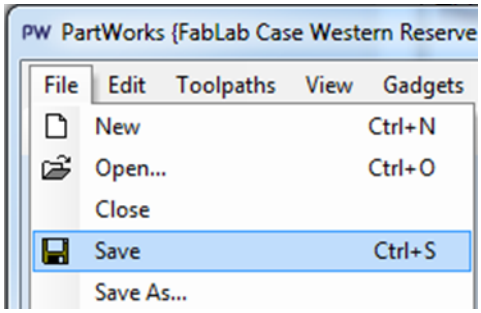
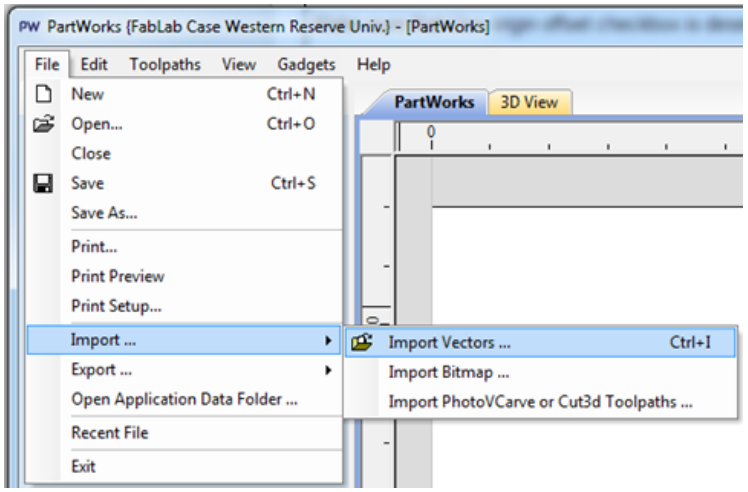
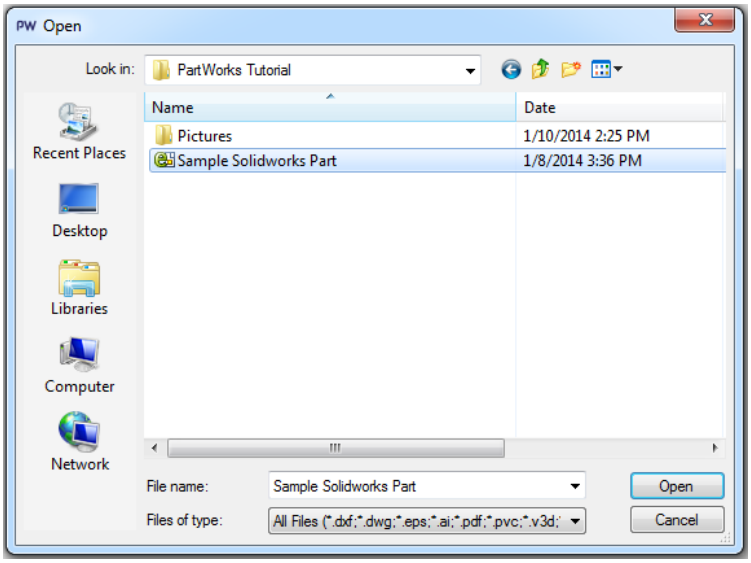
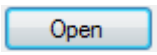
Setup the V Carve File

1. Open V Carve	
2. Click *File >> New.*	
3. You will now see the Job Setup window on the left sidebar. This sidebar allows us to tell V Carve about the dimensions of your material.	
4. Get a tape measure and calipers. Please Note: It is imperative to actually measure your material. Materials are sold using “nominal” dimensions, for example 96 by 48 by ½ inches, however they are just about never exactly these dimensions. To use a CNC machine, accurate dimensions are needed.	

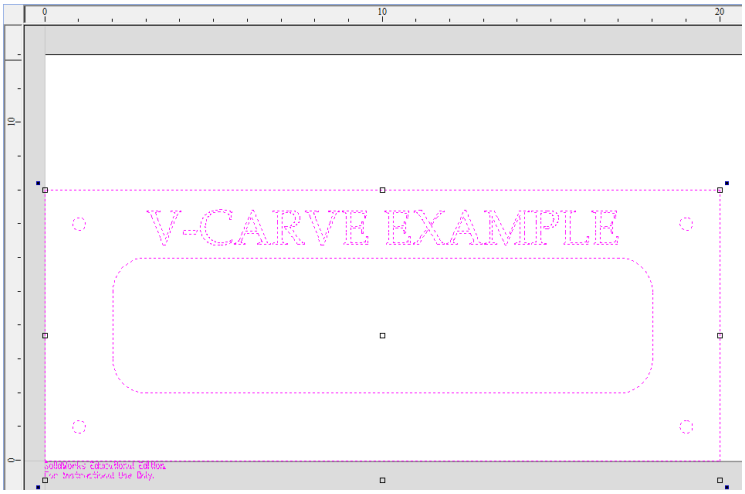
5. Use the tape measure to measure the length and width of your material.	
6. In V Carve, enter the dimensions of your material in inches. These are the dimensions you measured with the tape measure. The maximum dimensions are: 96 inches and 48 inches in X and Y respectively.	
7. Use the calipers to measure the thickness of your material. Squeeze the jaws of the calipers snugly against your material to get an accurate measurement. Do this in several places on your material and note the maximum thickness you measure.	
8. Next, enter the Z dimension of your material in inches. This is the thickness you measured with the calipers. The maximum dimension is 6 inches.	
9. Make sure the the Z Zero is set to the top of your material and not the bottom.	
10. Make sure the XY Origin Position is set to the bottom left.	
11. Make sure that Use origin offset checkbox is	

deselected.	
12. Make sure that inches is selected as the Unit.	
13. Click the *OK* button.	
15. Now is a good time to save your work. Click *File* >> *Save*, give your file a good name, and save it on the desktop or somewhere you can easily find it.	

Import your design

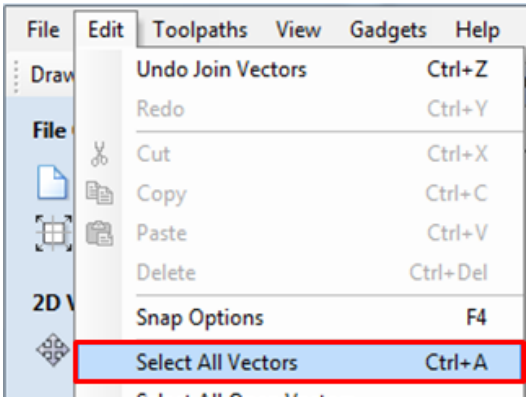
16. Click *File* >> *Import* >> *Import Vectors.*	
17. Use the file browser to find your design file. The AI file you produced in Adobe Illustrator.	
18. Click Open.	

19. Your design should appear in V Carve. You may need to zoom out (use the scroll wheel on the mouse) to see your complete design.

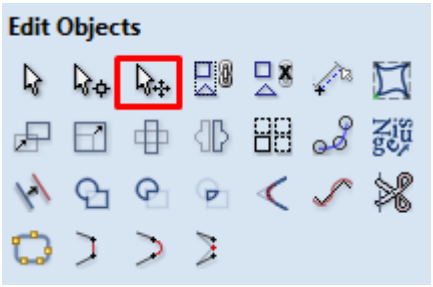


Move Design Onto Material

20. Click *Edit >> Select All Vectors*.



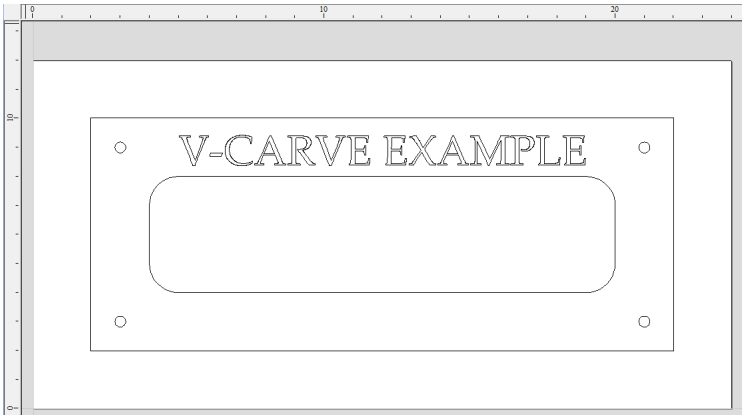
21. Click the Move / Scale / Rotate tool.



22. Click and drag your design to where you want it to be cut on your material.

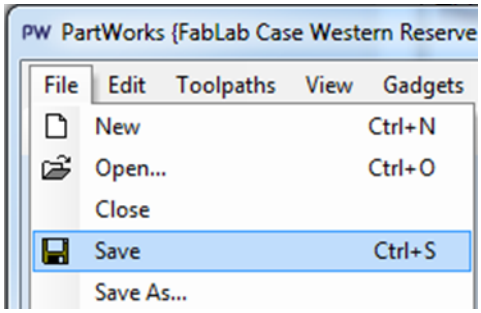
Use the arrow keys on your keyboard for fine adjustments. Note that holding SHIFT while pressing an arrow key will move in very large increments, while holding CTRL while pressing an arrow key will move in very small increments.

Verify your design is entirely over the material and not sticking out anywhere.



23. Now is a good time to save your work.

Click *File* >> *Save*.

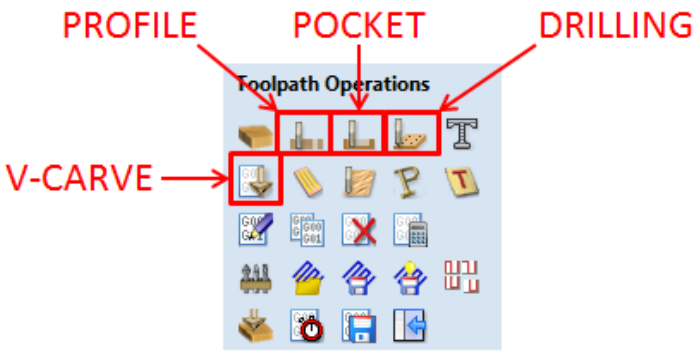


We will use a Profile toolpaths (you can skip this next section)

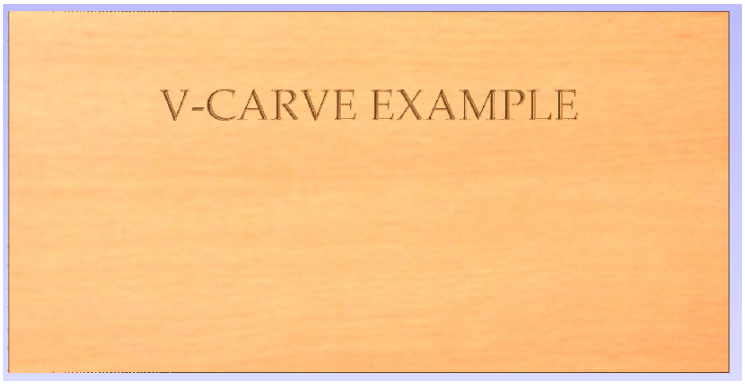
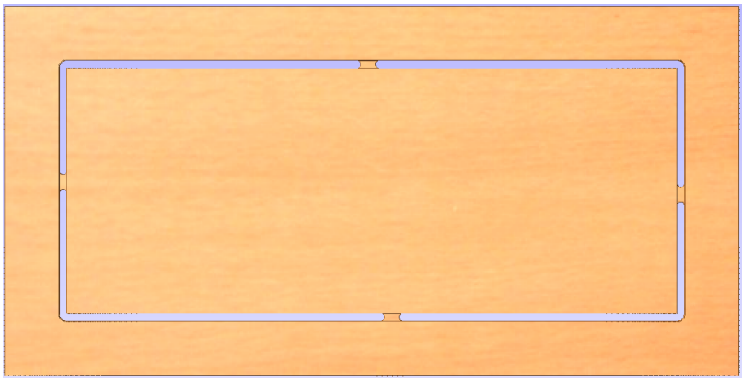
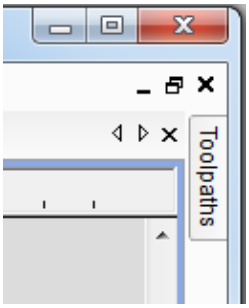
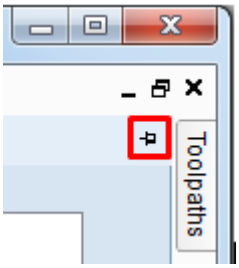
Introduction to Toolpaths

24. This tutorial covers 4 different kinds of toolpaths:

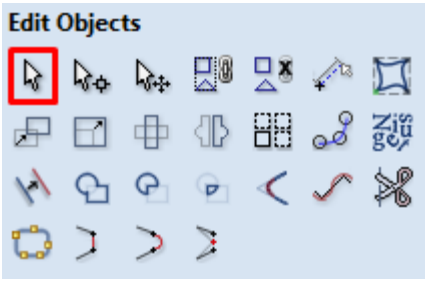
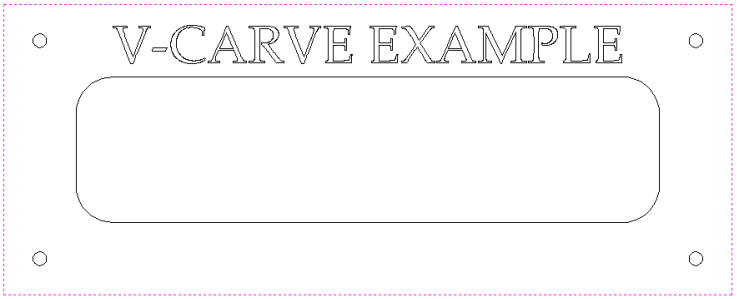
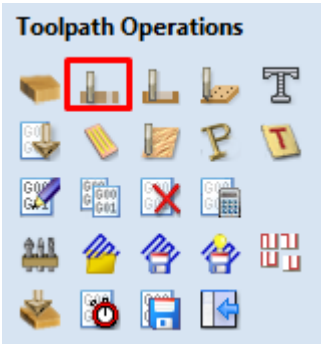
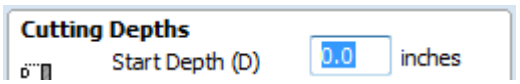
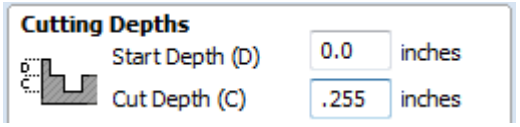
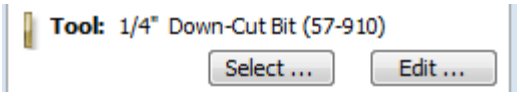
- Drilling toolpaths
- Pocket toolpaths
- V-Carve toolpaths
- Profile toolpaths



25. Drilling toolpaths are used to rapidly drill many holes. The shopbot can hold any drill bit from 3/32 inches in diameter to 5/8 inches in diameter.

26. Pocket toolpaths are used to remove material within the area of a closed shape.	
27. V-Carve toolpaths are typically used for engraving words into a design. The v-shaped profile of the tool produces clean lettering with sharp looking internal corners.	
28. Profile toolpaths are used to cut out shapes.	
29. Click the Toolpaths tab.	
30. Click the pin icon to keep the Toolpath sidebar open.	

Create all profile toolpaths

31. Click the Selection tool.	
32. Hold the SHIFT key while clicking on vectors you would like to cut out. If you would like to cut out different vectors using different bits, only select the vectors for one of the bits at this time.	
33. Click Profile Toolpath	
34. Make sure the Start Depth is set to 0 inches	
35. Set the Cut Depth equal to the thickness of the material as measured with the calipers, plus 0.005 inches to make sure the bit really does go all the way through your material. Be very careful when entering this value, a slight typo can result in the ShopBot cutting too deep and ruining the table. **The settings that we should use with the mdf is** **Start Depth 0.0** **Cut Depth 0.8** **WARNING:** your Cut Depth may not be any longer than the flute length for the tool you plan on using. If you need to cut deeper, you must use a tool with a longer flute length. Ignoring this may cause broken bits.	
36. Click the “Select” button	

37. The settings for the bit should be:

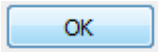
Tool Type: End Mill

Geometry
Diameter (D) 0.25

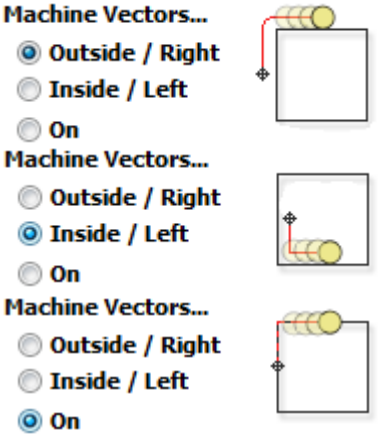
Cutting Parameters
Pass Depth 0.125
Stepover 0.19 | 76%

Feeds and Speeds
Spindle Speed 12000
Feed Rate 2.36
Plunge Rate 0.75

38. Click “OK”.



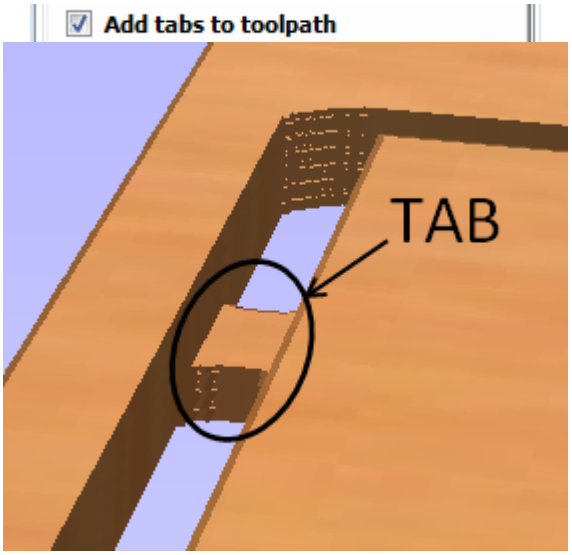
39. Select the appropriate Machine Vector depending on whether you would like the cutting bit to follow the Outside of your vectors, the Inside, or directly On your vector.



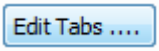
40. Select “Climb” cut direction, it will generally give you the best cut. The exception is if you are cutting foam, in which case select “Conventional” cut direction.

41. Enable “Add tabs to toolpath”.

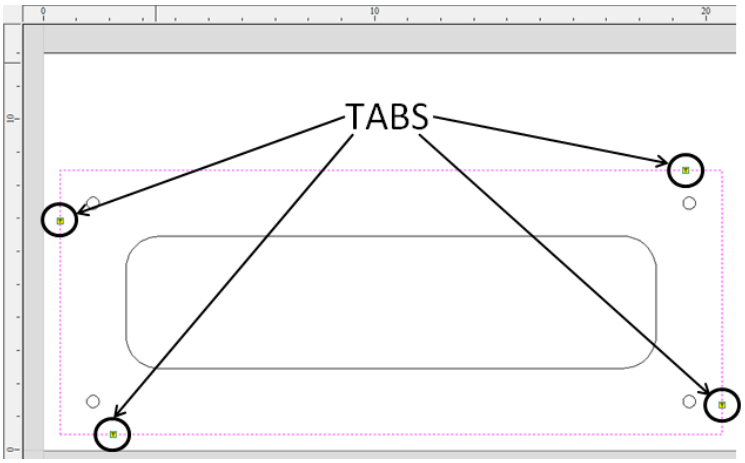
Tabs are small bits of material left uncut, which anchor your cutout to your workpiece in order to prevent the cutouts from freely moving while the bit is still cutting.



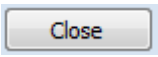
42. Click the “Edit Tabs” button



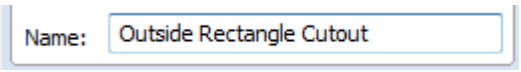
43. Add tabs by clicking anywhere on the selected vectors. It is considered good practice to add tabs near each corner, AND one tab every 24 inches or so.



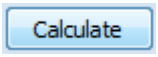
44. Click the "Close" button to close the Tabs sidebar.



45. Optionally, give your toolpath a unique name. This is helpful if you have many different profiling toolpaths and you would like to easily identify them.

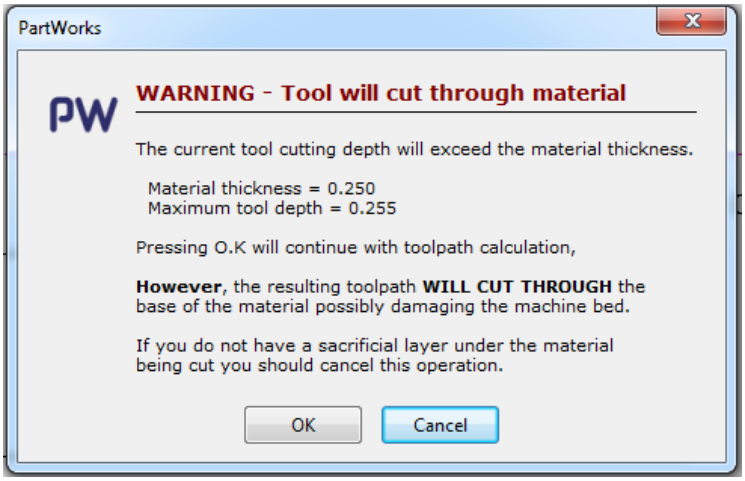


46. Click Calculate.

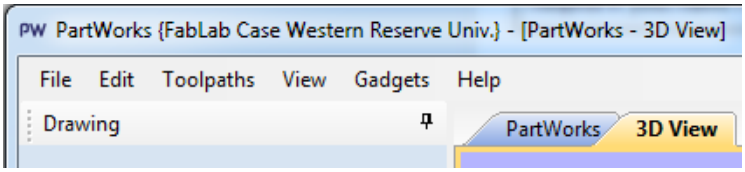


47. You will receive a warning message that your Cut Depth is deeper than the thickness of your material. This is expected, and is nothing to worry about. The bit will go entirely through the material, and very slightly go into the MDF that covers the ShopBot table. This MDF layer is a sacrificial layer, also called a "spoil board", and is periodically replaced.

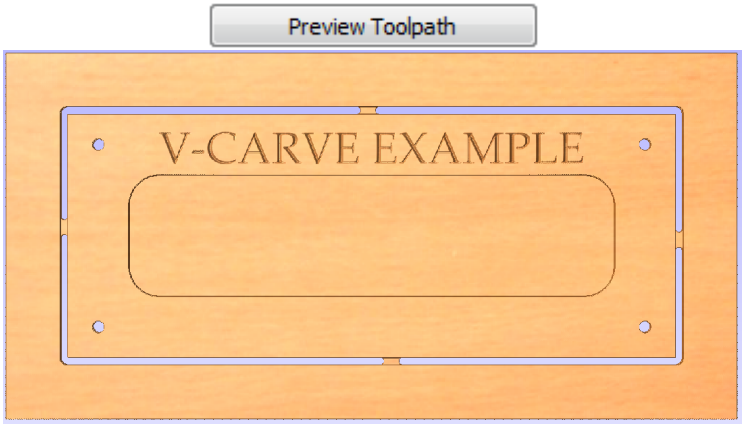
Click OK to continue.

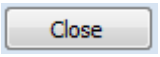
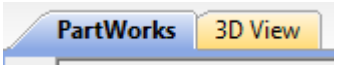
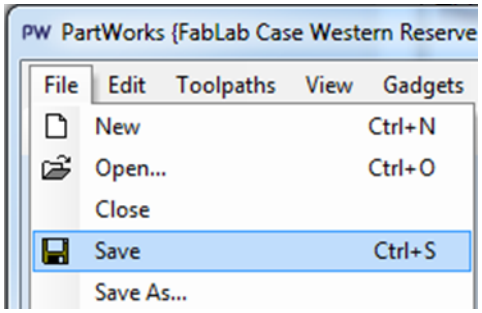


48. Notice that V Carve has automatically switched from *V Carve* design view to *3D View*.

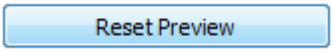
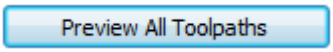
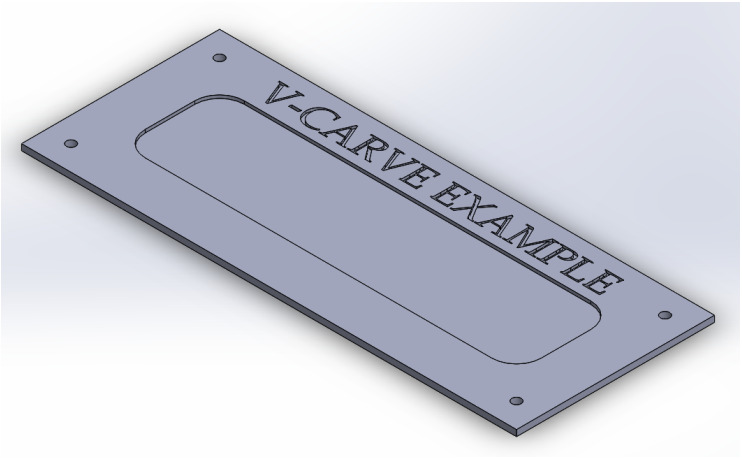


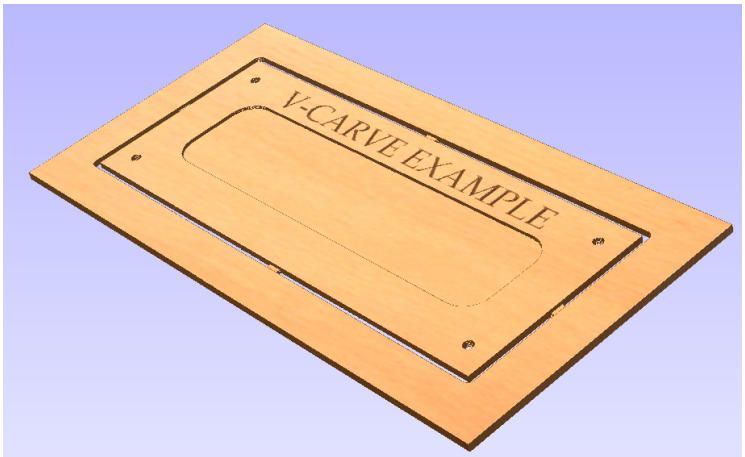
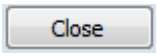
49. Click the Preview Toolpath button to preview what your material will look like after this toolpath has run.



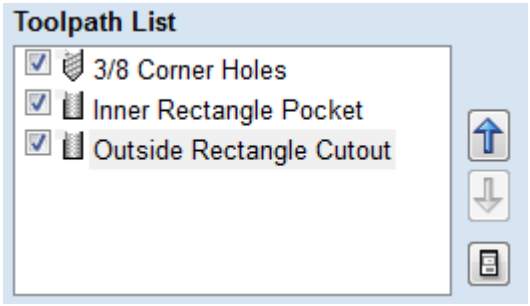
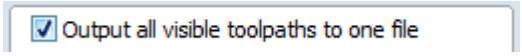
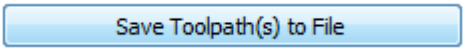
50. Click the Close button to exit the 3D Preview sidebar.	
51. Click the V Carve tab to return to the design view.	
52. If you want to create additional profile toolpaths, repeat steps 31 through 51.	
53. Now is a good time to save your work. Click *File* >> *Save*.	

Preview all toolpaths

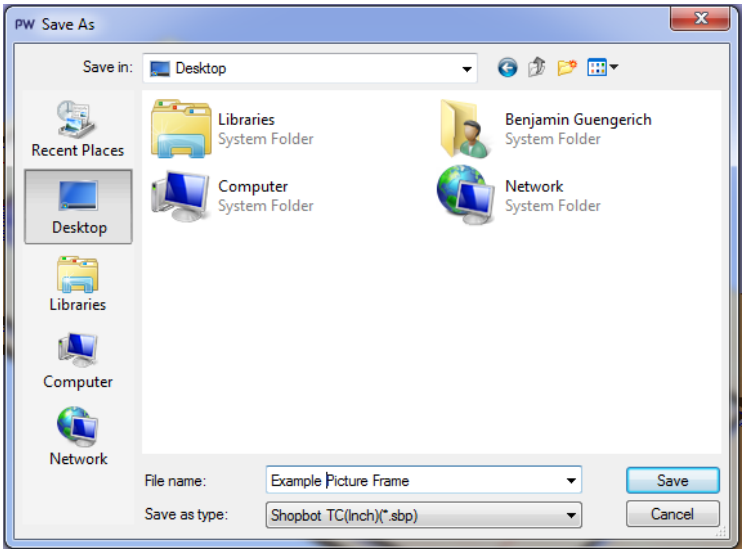
54. Click the “Preview Toolpaths” icon.	
55. Click the “Reset Preview” button.	
56. Click the “Preview All Toolpaths” button.	
57. Verify your preview looks as you expected. If not, go back and try again and see if you can figure out what went wrong. Otherwise, try to find a teaching assistant who is familiar with V Carve and ask them for help.	

	
58. Click the Close button to exit the 3D Preview sidebar.	

Export your toolpaths

59. Make sure all of your toolpaths are selected.	
60. Click the “Save Toolpath” icon.	
61. Make sure that “Output all visible toolpaths” is enabled.	
62. Click the “Save Toolpaths to File” button.	

63. Give your file a unique name and save it on the desktop or somewhere you can easily find it.



You are done!