

SolidWorks Intro Part 1c

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1. Convert a part to a dxf format

When you save a part in Solidworks it is saved as a 3D solid model with all the relationships among the entities, but all you need for the laser cutter is the 2d vector lines that define the paths that the laser should follow. In addition, you need to color the lines so you can tell the laser to etch the lines for the text and to cut the lines for the holes and the boundary. There are many ways to do this. This handout covers one of the easiest ways, but we will show you some others in later tutorials. What appear as extra steps in this process are not extra; they are necessary to convert the 3D part to a collection of lines.

1.1 What the colors mean to a laser cutter

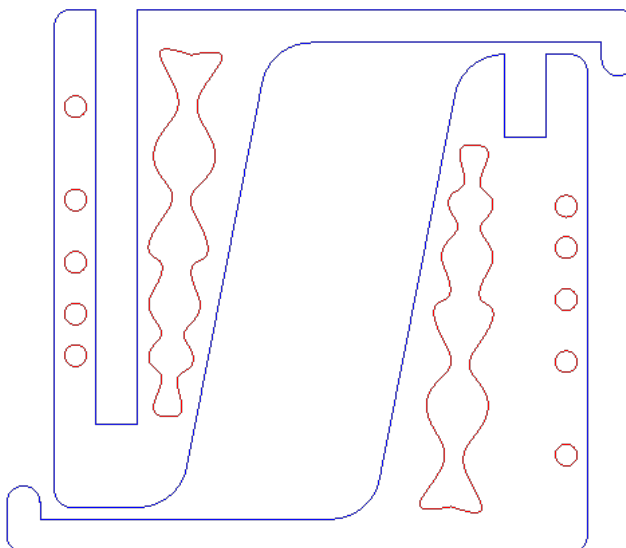
On older laser cutters, the color meanings are fixed:

- Red means “cut”
- Black or blue means “engrave”

Whether a laser cuts or engraves is a function of the power setting of the laser and the speed with which it moves. On modern laser cutters, the user can define powers and speeds for each available color. You can also specify the order in which colors are cut.

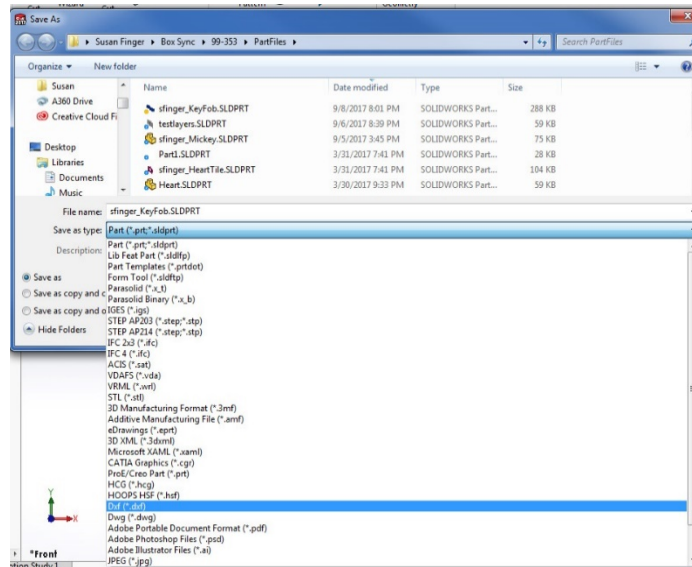
The general principle is to **engrave first, cut interior holes next, and cut the outline last.**

In the drawing of a business card holder, both the red and the blue lines would have power and speed set for cutting, but all the red lines would be cut first followed by the blue lines. This ensures that the parts do not move before the final cuts are made.



1.2 Save your part as a dxf

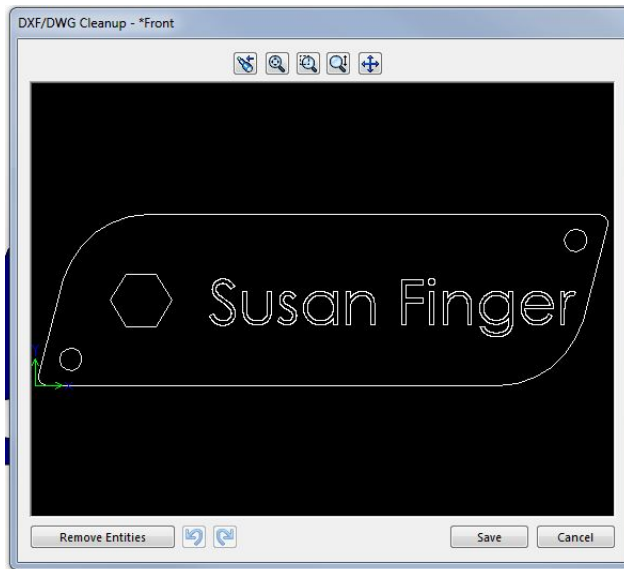
Select **File** → **Save As**. Select **dxf** from the list of file types.



When you press **Save**, you are not done. In the left panel, you will see that you have to say which view you want to export.



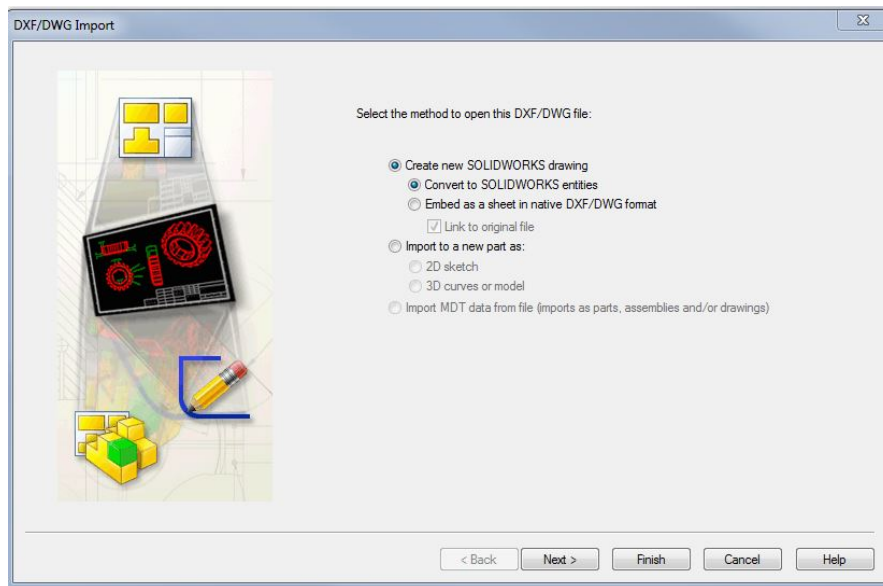
Even though the current view is the front view, you can play it safe by unchecking “Current” and checking “Front.” Click the green checkmark to finish saving.



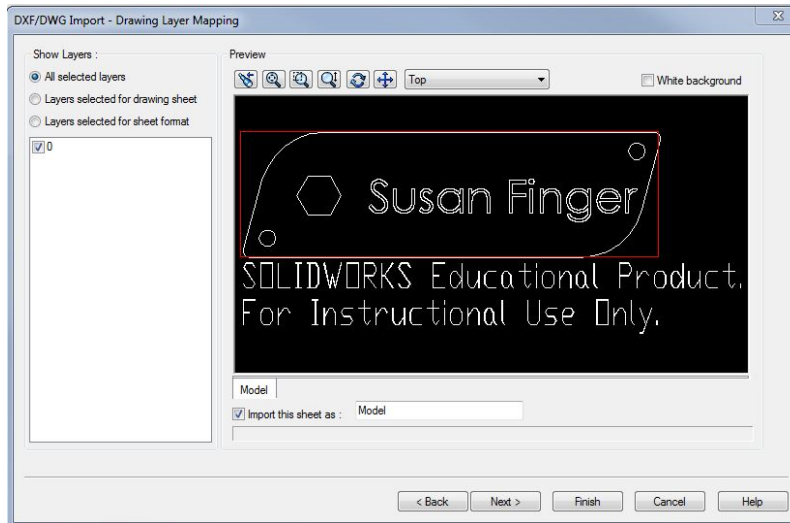
Click **Save**.

1.3 Open the dxf file as a drawing

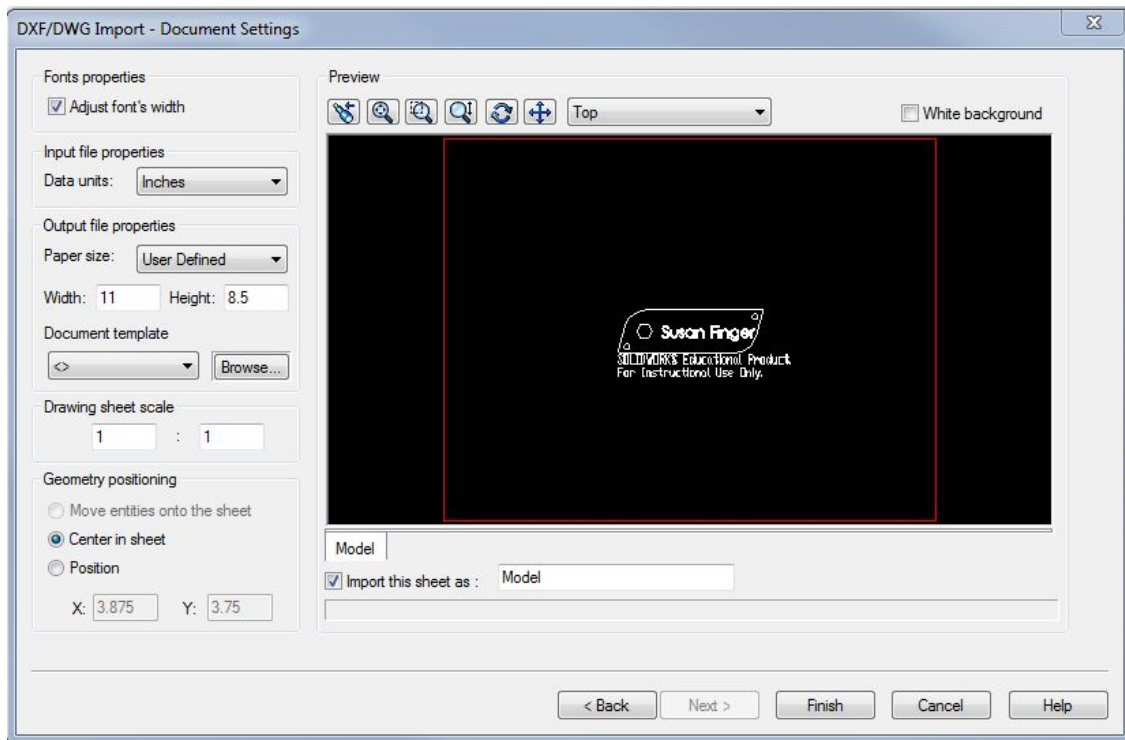
Select **File** → **Open** and select your DXF file. You should get the following dialog box:



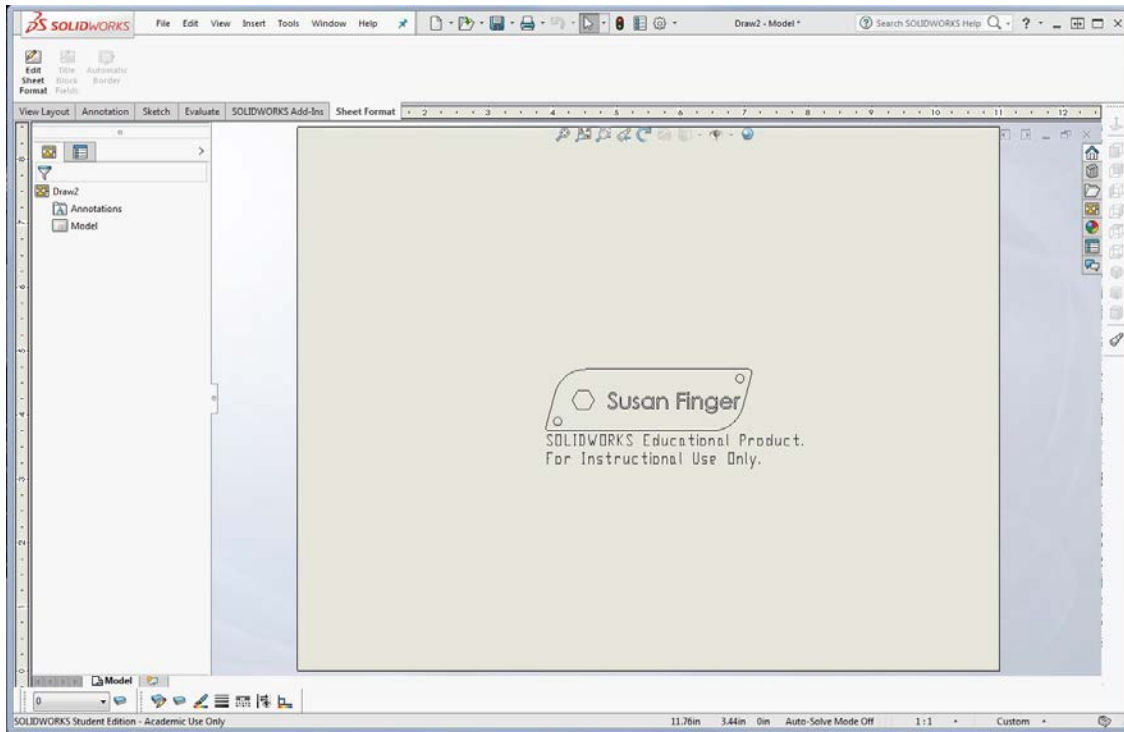
The defaults are OK. You want to create SolidWorks drawing. When you click **Next**, you should get a popup that looks a lot like the one that you had before, except it has some extra glop in it. Don't worry about that for now.



Click **Next** to get the window that lets you set the size and scale of your drawing. Be sure the **Units** are Inches. Select **User Defined** from the Paper size pull down menu. Set the **Paper size** to 4 by 4. Be sure the scale is 1:1. Finally, click next to **Center in Sheet**.



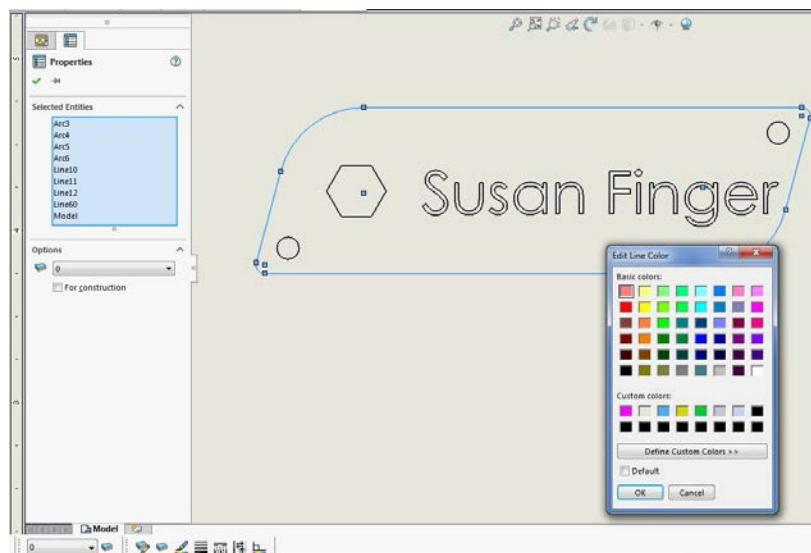
Click **Finish**. Your view should now look like the screen shot below -- except that I probably have more tool bars pinned in various places and your key fob will have your name.



Select the extra text about SOLIDWORKS and delete it. Select **View** → **Toolbars** → **Line Format**. This toolbar will appear at the bottom on your SolidWorks window.

You can use the **Zoom to Area** tool to get a closer view of your part.

1. Use Shift Left Mouse Button to select all the lines on the perimeter of the part.
2. Click the **Line Color** tool (the paintbrush) in Line Format toolbar
3. Select a color
4. Press OK
5. Press the green check mark

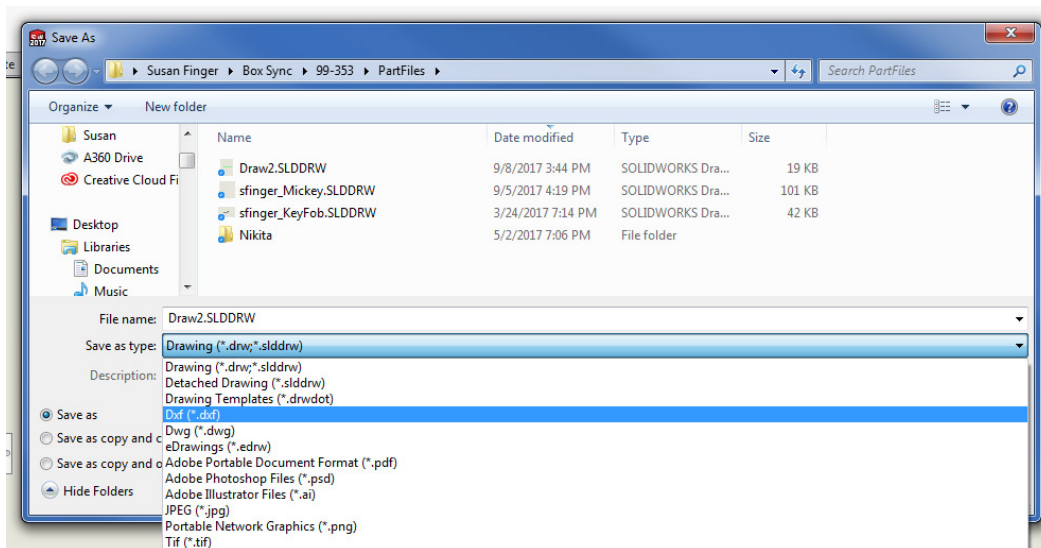


Repeat these steps to make the interior holes all the same color (but different from the perimeter). Note that you have to select each line. If you miss a line, just repeat the process for that line.

To select the lines of the text, press the shift key and drag the mouse until you have all the text selected. Set a new color for the text.



Save your keyfob one final time. The default file type will be a SLDDRW (SolidWorks Drawing), but you need a DXF file. You will probably have to give it a slightly different name in order to save it since original DXF file may still be open.



Make a copy of your final dxf file on a USB stick, a google drive, send it to yourself in email, ... You will need to import the file on the laser cutter computer, so put it somewhere where you will be able to find it.

Finally, upload a copy of the dxf file to Canvas for today's assignment.