

Creating a DWG/DXF File for use with the Laser Cutter System/AWJ

Laser cutter line color settings:

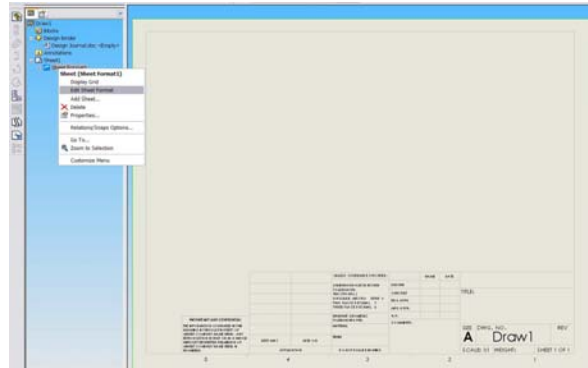
1. Cut lines (through cuts): CYAN
2. Etch lines: RED
3. Score lines (deeper than etch): YELLOW
4. Line thickness: 0.000"



SolidWorks 2006

NOTE: In SolidWorks, DWG and DXF files MUST be created from the drawing document (*.slddrw).

1. From your solid model, create a SolidWorks drawing document of your part (*.slddrw).
2. Delete all borders/annotations from the template if needed.
 - a. In the feature navigator, right click the name of the sheet or sheet format and go to **Edit Sheet Format**. All borders will turn color from gray to blue. Select and delete all annotations and borders. When finished, right click the sheet format again and click **Edit Sheet** to exit the sheet editor.
3. Insert the appropriate views of your parts noting spacing and layout.
4. Be sure to **delete** all centerlines. Hide all origins and sketches. Select **View -> Hide All Types**.
5. Bring up the **Line Format** toolbar. **Tools -> Customize**. Check **Line Format** under **Toolbars** tab.
6. Select edges of your models if necessary – in the **selection filter** toolbar). See figure to the right.
7. Select all edges. Go to the **Line Format** toolbar. Select **Line Thickness** icon and select the thinnest setting available (first setting directly under the word “default”). All edges should become thinner.
8. Select edges (see step 6) that correspond to a certain color setting. Use the **Line Color** icon in the **Line Format** toolbar to change edges to cyan, red, or yellow depending on your design intent.
9. DWG Creation:
 - a. **File > Save As**
 - b. In the **Save As Type** drop-down box, select **DWG Files (*.dwg)**
 - c. Type an appropriate file name of your choice
 - d. Select **OK**
10. DXF Creation:
 - a. **File > Save As**
 - b. In the **Save As Type** drop-down box, select **DXF Files (*.dxf)**
 - c. Type an appropriate file name of your choice
 - d. Select **OK**



NX 5.0.1.x

1. NX 5.0.0.x is currently NOT supported for DXF creation. Please use NX 4.0.x or 5.0.1.x
2. DXF Creation:
 - a. From the model (*.prt) file
 1. A solid geometric shape will output 3D in the DXF file. Sketches/curves will output 2D curves.
 2. Create your sketch. Put your model in the TOP orthographic view or whichever orthographic view you require.
 3. **Save** the *.prt file
 4. **File > Export > DXF/DWG**
 5. Under heading **Export to**, make sure **DXF** is selected.
 6. Specify the DXF file name and path under the heading **DXF File**
 - b. From the drawing (*.prt) file [*Method 1*]
 1. If you are using the RPI template with borders and wish to delete them within NX:
 - a. Format > Layer Settings
 - b. Turn layers 255 and 256 from **Visible** to **Selectable**.
You may now delete these extra curves.
 2. **Save** the *.prt file
 3. **File > Export > 2D Exchange**
 4. See settings below. Under DXF file, select the directory and name to save.

