**Creating a DXF File for Use on the AFL Waterjet or Laser Cutter**

<u>Purpose</u>:

This document provides a basic framework for the creation of DXF drawing files, in Siemens NX 9.0, for use on the AFL waterjet or laser cutter. The DXF file contains the geometry needed to create cutting paths on each machine.

<u>Procedure</u>

1. Begin with the 3D model open for each part you'd like to create DXF drawing for. This will make it easier to extract the necessary geometry.
2. Click File → New, and select the Drawing Tab.
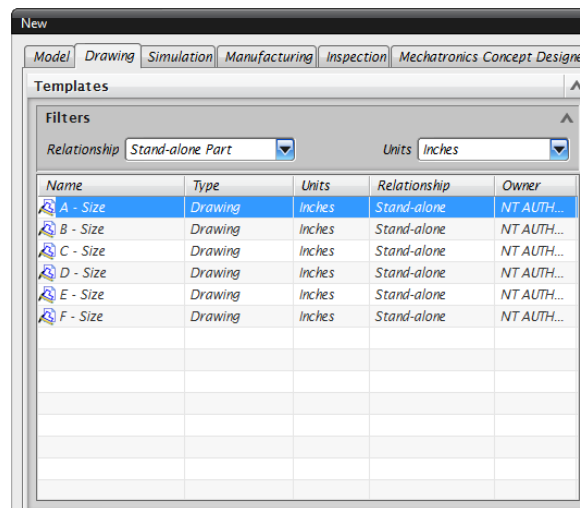
Specify inches as the unit for your drawing, as both machines operate using the imperial system. Select a drawing size (A, B, C, etc.) that will fully encompass your drawing. This step is not critical, but can save some frustration in the coming steps. Name your file, and locate the folder in which you'd like to save the drawing. Click OK once this is complete.

3. By default, NX will open a drawing sheet with a complex template (including a title block). If this template is retained, both cutters will attempt to cut the template shape during operation. The easiest way to remove the template is to add a new sheet.
   Select New Sheet from the ribbon.
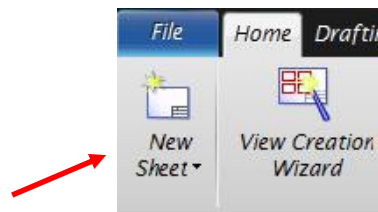
A dialog box will open. Select either Standard Size or Custom Size. **Do not select Use Template.** Select your desired size, being sure to retain a 1:1 scale.

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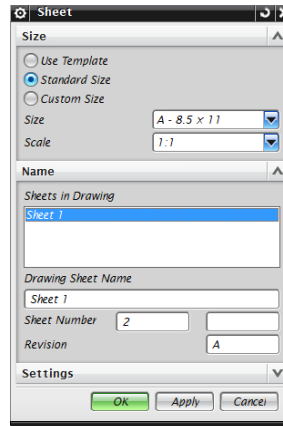
4. With the additional sheet created, simply delete the initial sheet that was created initially by NX. You will be left with a blank sheet to work with.

5. To specify the geometry you'd like to create in the drawing, click the View Creation Wizard button.

In the part section, select the part you would like to create a view of. If you have previously loaded all your parts in NX, you can find the part in the Loaded Parts section. Otherwise, you will have to open the proper file in this window.
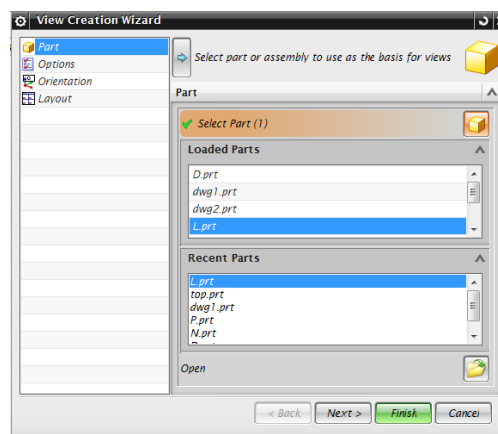

6. With the proper part selected, click options. On the View Boundary dropdown, select Manual. Deselect Auto-Scale to Fit, and change the scale to 1:1. This will ensure your part is cut the proper size. Deselect Process Hidden Lines, Show Centerlines, Show Silhouettes, and Show View Label.

7. In the orientation section, select the view required to show the 2D shape of the part you would like to cut. This will look similar to the image below:
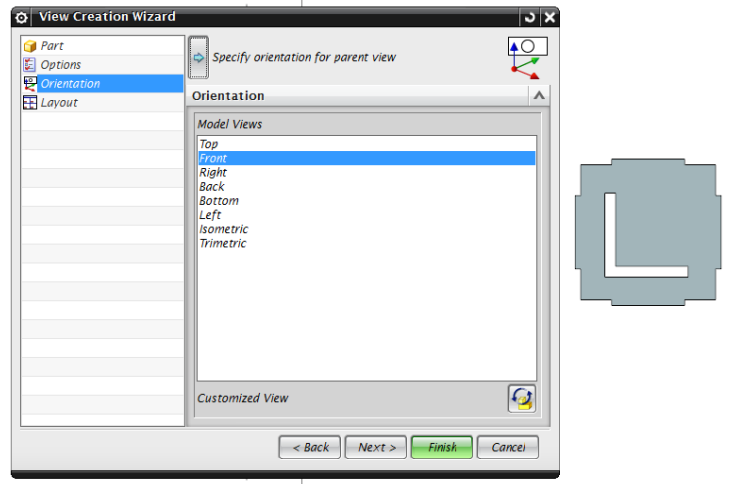

8. Click Finish to create the part view. The result should be the 2D geometry that will be cut using the waterjet or laser cutter.
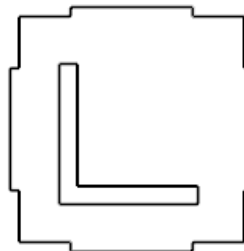
9. Repeat steps 5 – 8 for each part you would like to place on the drawing file. The arrangement of drawings on the sheet can be modified by clicking and dragging the bounding box that surrounds the part representation.


Bounding Box
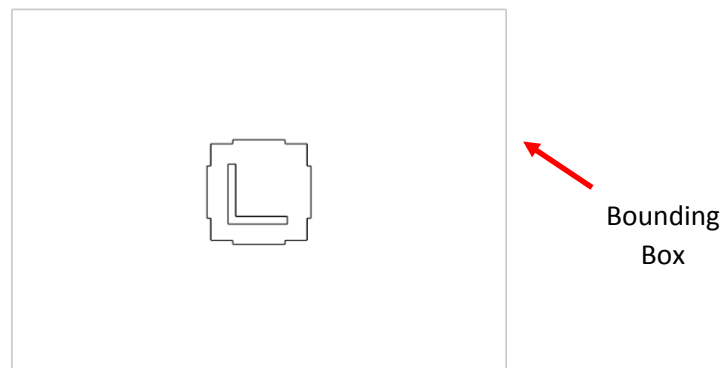
10. The resulting drawing should look similar to what is shown below.
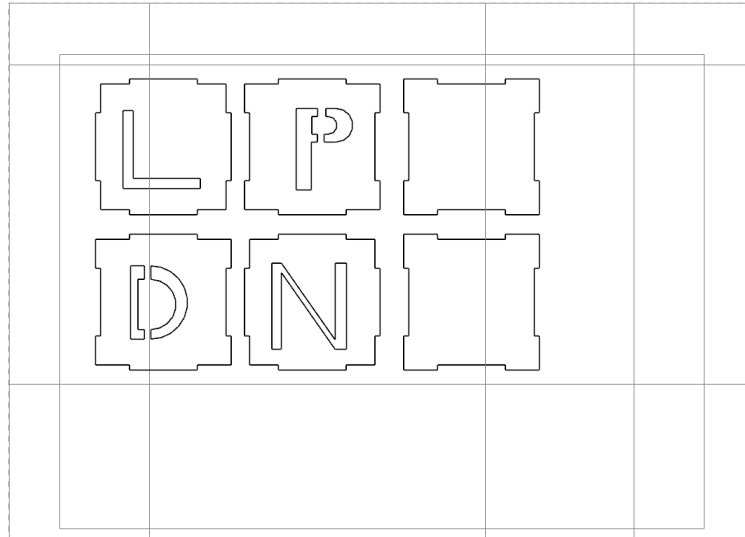
Once each of the parts/views have been created, we can convert this file over to a format that the machines can read.

11. While many CAD packages allow you to convert a drawing file to a DXF simply by saving as a DXF, NX does process this conversion appropriately. Instead, click File → Export → AutoCAD DXF/DWG. This will launch the export wizard.
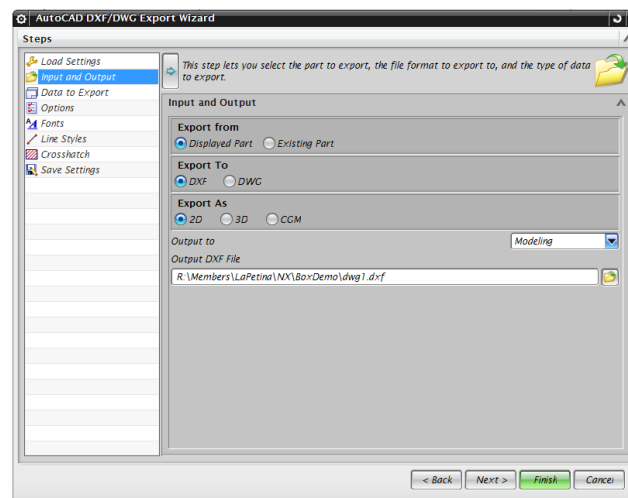
In the Input and Output section, specify:
- Export from Displayed Part
- Export to DXF
- Export as 2D
- Output to Modeling

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Specify the output file name and location, and click Finish. You need not change any of the other settings in this wizard. A command prompt will open, do not close this window. Once it closes on its own, locate your file. It should now be ready to use on the waterjet or laser cutter!

Note: Some extraneous lines may be generated by NX that will need to be deleted on the waterjet/laser control. This is unavoidable and easy to fix.