Creating a DXF File Using AFL Supported CAD Packages

Purpose:

This document provides a basic framework for saving files from AFL supported CAD packages as a .dxf file, in order to be used on the waterjet in Purdue's AFL.

Software Packages:

- AutoCAD 2013
- AutoDesk Inventor 2013
- CATIA v5R20
- Siemens NX 8.5/9.0
- CREO (Pro/E) 5.0
- Solidworks 2011

Basic Procedure:

A .dxf file extension is a basic 2-D graphics file. The easiest way to get a correct file is to create a drawing of the top view of whatever profile you want to cut out of your material, and then simply perform a "Save As" to a .dxf file. This works for any software package.

Create 2-D Image (Drawing) → 'Save As' Option → Choose .dxf file extension

Because a .dxf file is a 2-D graphics file, you cannot save a 3-D model as a .dxf file. Therefore you must create a drawing first.

AutoCAD 2013:

Given that AutoCAD is used primarily for 2-D drawings, simply save the file as a .dxf extension.

AutoDesk Inventor 2013:

Inventor Fusion 2013 (available on the Demonstration Studio computers) saves .dxf files so that it can read them, but they cannot be read by FlowPath. It is necessary to save the 3-D model as a .stp file, which can then be imported into another CAD package and used.

1. File → Save As → Type name and use a .stp file extension

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Catia v5R20:

Starting from a 3-D model, follow these step

1. Create a Drawing file (File → New → Drawing)
2. Choose whatever style of drawing sheet is needed to fit your part
3. Click on the View Creation Wizard

4. Select a Projection Method (any will do)
5. Click Next, and then Finish
6. Go back to the 3-D Model (Window → Select whichever window holds the 3-D Model)
7. Click on whichever side of the 3-D Model you want as the "Top View" (to be waterjet cut)
8. Delete all extraneous information (other views, words, etc...) until only the top view remains
9. File → Save As → Type name and use a .dxf file extension

Siemens NX 8.5/9.0:

Starting from a 3-D model,

1. Make sure the 3-D model is open in the NX session
2. Click File → New → Drawing Tab
3. Choose a drawing size that encompasses your part
4. Close the initial box that asks for title block information
5. Click on the View Creation Wizard
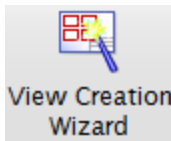
6. Choose your part from the "Loaded Parts" option
7. Click on the Orientation Tab in the Option Tree on the left side of the dialogue box
8. Choose whichever orientation shows the view you wish to be cut
9. File → Save As → Type name and use a .dxf file extension

* Note: It will be necessary to delete a few extraneous lines in FlowPath once opened

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CREO (Pro/E) 5.0:

From 3-D model

1. Open the 3-D model in Pro/E
2. File → New → Drawing
3. Make sure the 3-D model name is in the "Default Part" box
4. Choose a drawing size
5. Delete the pre-defined views (everything on the drawing)
6. Under the "Layout" tab up top, select the "General" button in the Model View Choices

7. Click on any point inside the drawing
8. Under "View Type" Category (default when open), select the view you want to show for waterjet cutting and select "Apply" at the bottom of the dialogue box
9. Under "View Display" Category, click the "Display Style" drop down box and select the "Hidden" option.
10. File → Save a Copy → Type name and use a .dxf file extension
11. Click "OK" on the dialogue box that opens.

* **NOTE**: Pro/E will not save the size correctly. Make sure you know how long any dimension is (preferably a   straight line) and once the file is opened in FlowPath, it can be scaled down. Also make sure to delete any unwanted lines in FlowPath (i.e. coordinate system).

Solidworks 2011:

From a 3-D model

1. Open the 3-D model in Solidworks
2. File → New → Drawing
3. Select the size of the drawing
4. Double click on the name of the Solidworks file while in the Model View (this is brought up by default for every new drawing)
5. Selected the preferred view under the "Orientation Tab" while still in the Model View dialogue box
6. Place the view on the drawing
7. Click the green arrow at the top of the Model View dialogue box
8. File → Save As → Type name and use a .dxf file extension