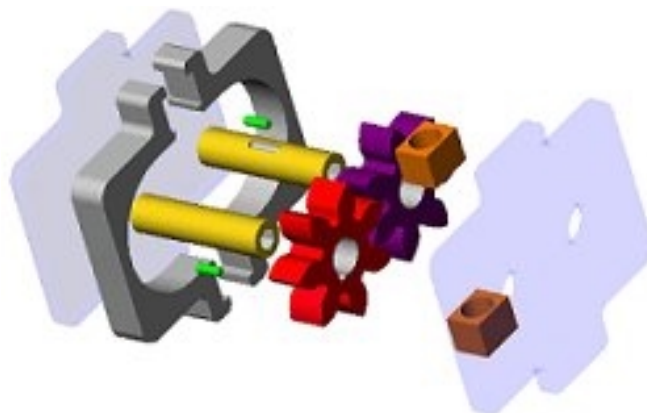


Creating DXF Files For The Waterjet

2D CAD Files For Your Gear Pumps

By Martin L. Culpepper



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I. INTRODUCTION:

This document will assist you in preparing your CAD files for machining on the Waterjet. If you experience problems with this tutorial contact a TA for help.

II. SOFTWARE FOR CREATING DXF FILES:

A DXF file is a standard format for 2D CAD. It is commonly used in industry to share ideas between people/companies/machines. You will make your DXF file using the CAD program in which you built the pump. We have included instructions for SolidWorks and ProEngineer.

III. WHAT IS REQUIRED

You need to **create 3 DXF files**, one for each part the Waterjet will cut. Therefore, you must follow the directions 3 times, once for each of the following 3 parts:

1. Endplate
2. Housing
3. Gear (for the 20002 offering of 2.000, we will make your dxf files)

IV. SUBMITTING FILES: NAMING CONVENTION

You may email the DXF files to me or hand them in on a disk. Please make sure you use the naming convention shown above, or you will have to re-submit.

I require the name of a group member (only one person) who will be responsible for submitting the files to me. **Do NOT leave spaces** in the file name.

GEAR:	Group_Member_Name-Gear.dxf
ENDPLATE:	Group_Member_Name-Endplate.dxf
HOUSING:	Group_Member_Name-Housing.dxf

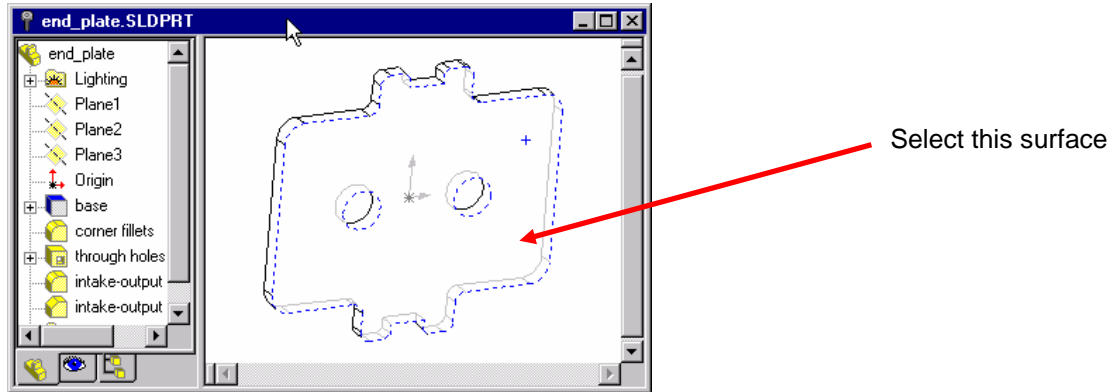
WARNING: DXF files are not tied to the original CAD models. If you make a change to the original part file, you **MUST** make a new DXF file to capture the changes. Changes made in part files are not reflected in DXF files!


V. MAKING DXF FILES IN SOLIDWORKS

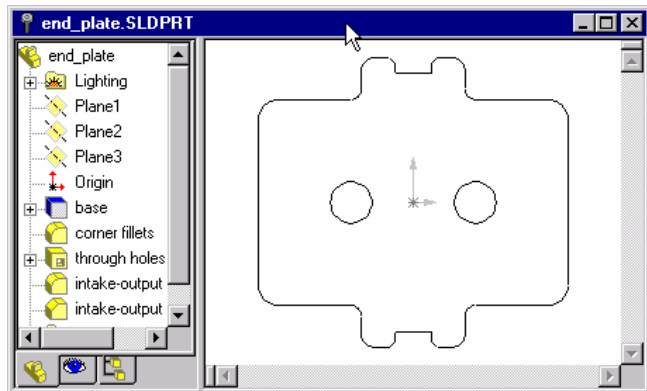
Start SolidWorks and open the part file for which you will make a DXF:

File > Open > part-file.sldprt (I.e. gear.sldprt, or housing.sldprt, or, endplate.sldprt)

With your cursor, select the surface which represents the shape to be cut by the Waterjet.

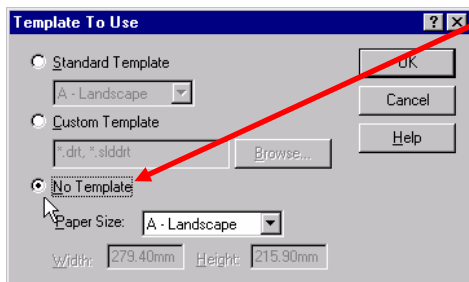


Then use the view normal tool button  to orient this surface parallel to the screen as seen below.



Now, open a new drawing **in this manner**:

File > New > Choose Drawing > Click OK > Choose No Template > Click OK



V. MAKING DXF FILES IN SOLIDWORKS CONT.

Before you move on, you must change the drawing length units to inches.

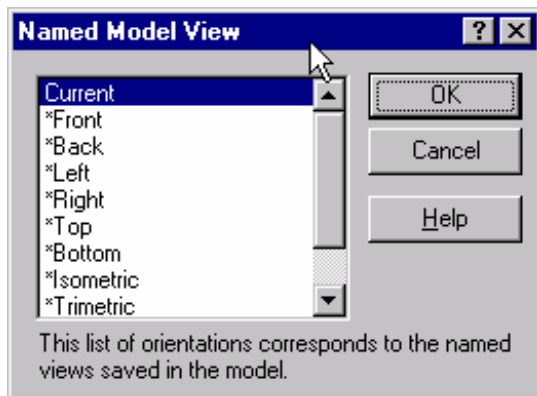
Tool > Options > Grid/Units Tab > Change the drawing dimensions to inches > Click OK

Now insert a view of the part file into the drawing. Make sure you are in the drawing file window:

Insert > Drawing Views > Named View > Switch windows to the part file > click in the working window

In the pop-up window which appears:

Select Current > Click OK > Return to the drawing file

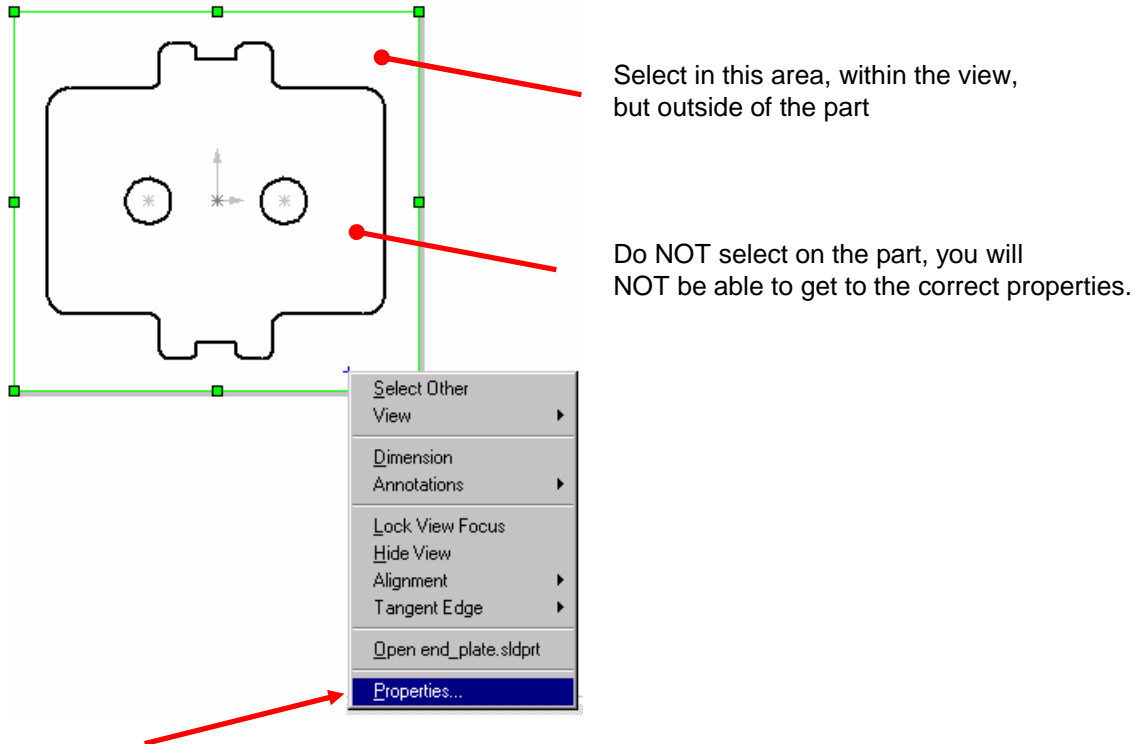


You should now be able to **drag and drop (by left clicking) the view into the drawing file's working window**. Do not be concerned as to the location, just “plop” it into the drawing file.

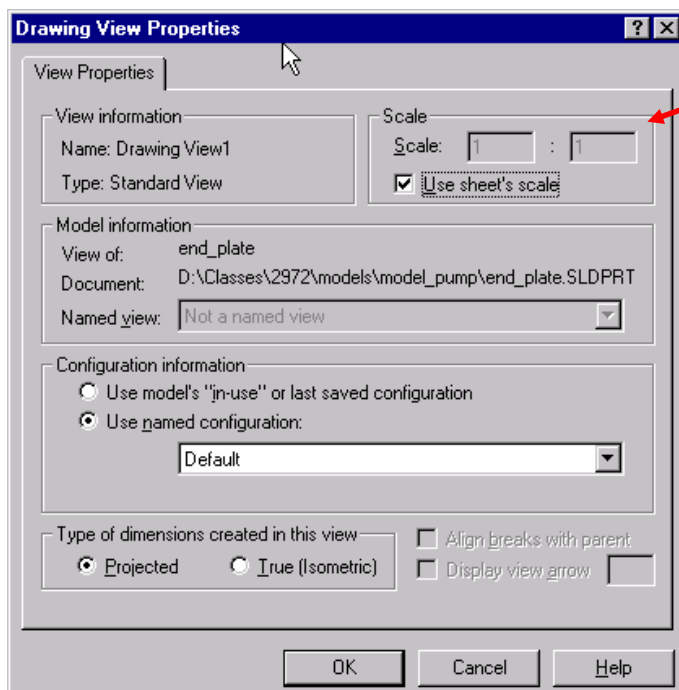
Close the part file (not drawing file) from which you acquired the view.

V. MAKING DXF FILES IN SOLIDWORKS CONT.

Right click on the drawing view, it should highlight in green.



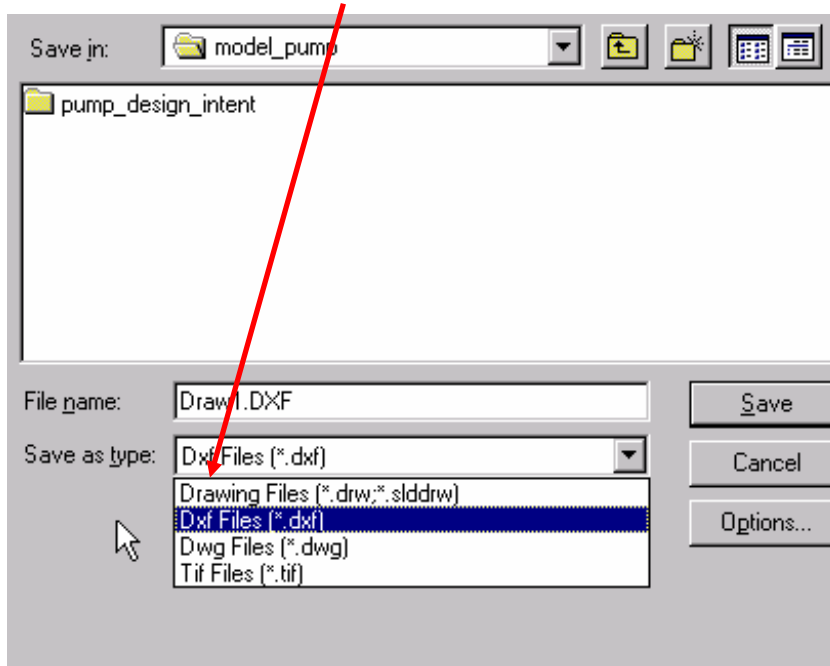
Choose **Properties** and make sure the scale at the top right of the pop-up window says **1:1**. If not, un-check the Use Sheet's Scale box, then change the scale to 1:1. **Click OK**



V. MAKING DXF FILES IN SOLIDWORKS CONT.

You can now save the drawing file as a DXF file (**read section IV before continuing!!!**):

File > Save As > Select DXF File > Give a file name with .DXF extension as explained below (V)



VI. STEPS IN MAKING DXF FILES IN PROENGINEER 2001

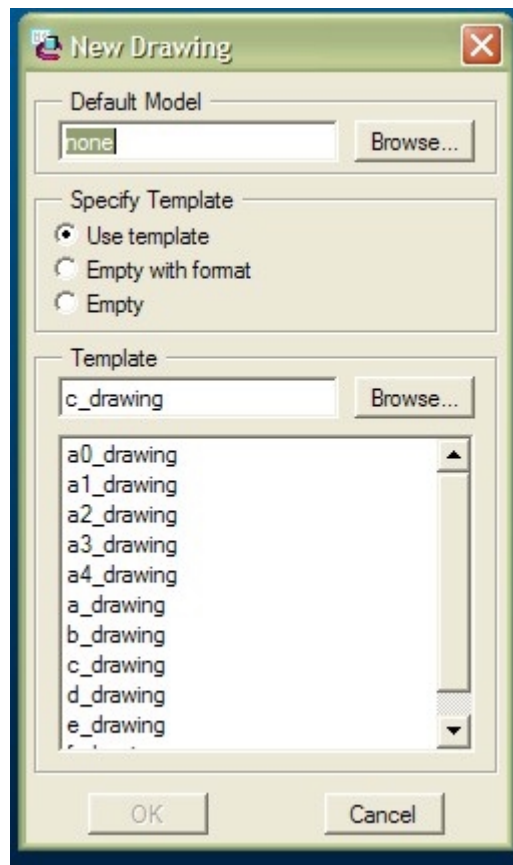
If ProEngineer is not open, start the program now.

Open the part file you want to convert to a DXF file.

Create a new drawing file (File->New).

In the “New” pop-up window, check the radio button for Drawing

The following dialog box should appear:



If the part you want to turn into DXF file is not listed under “default model” choose “browse” and find the part file you want.

Under “specify template” select the following options: Empty, then Landscape and Standard size = A

Click OK.

