Creating DXF Files For The Waterjet

2D CAD Files For Your Gear Pumps

By Martin L. Culpepper

CONTENTS:

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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.
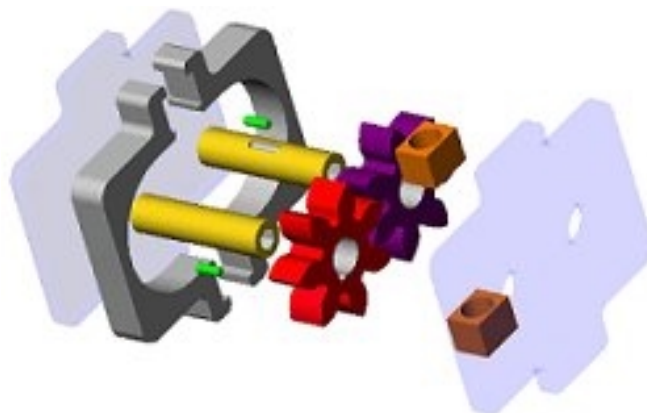
II. SOFTWARE FOR CREATING DXF FILES:

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!
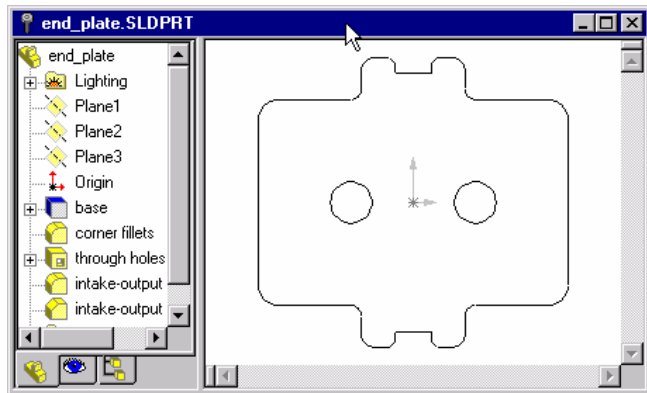
V. MAKING DXF FILES IN SOLIDWORKS

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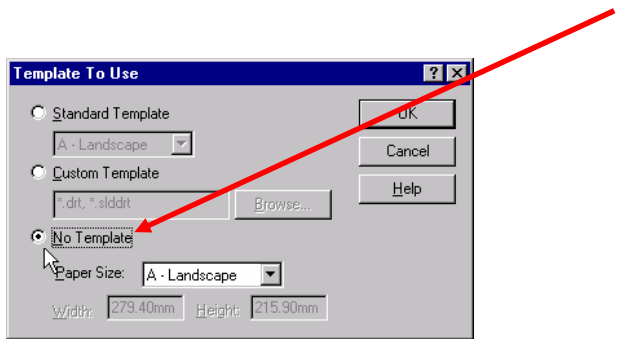
Select this surface
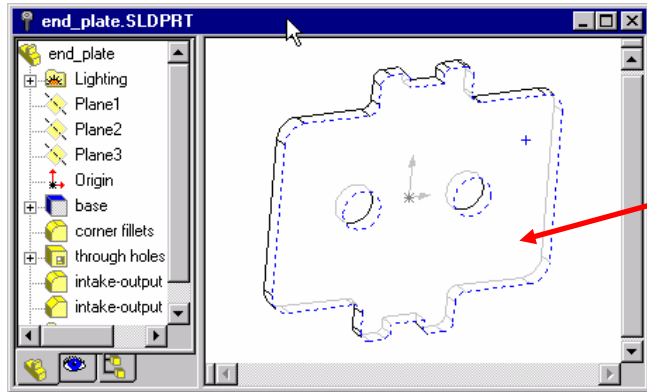
Then use the view normal tool button to orient this surface parallel to the screen as seen below.

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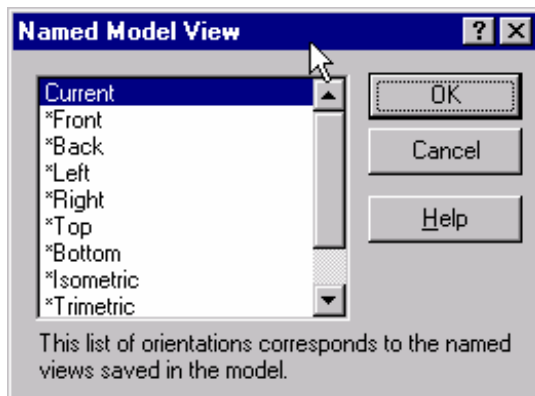
Tool > Options > Grid/Units Tab > Change the drawing dimensions to inches > Click OK

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:

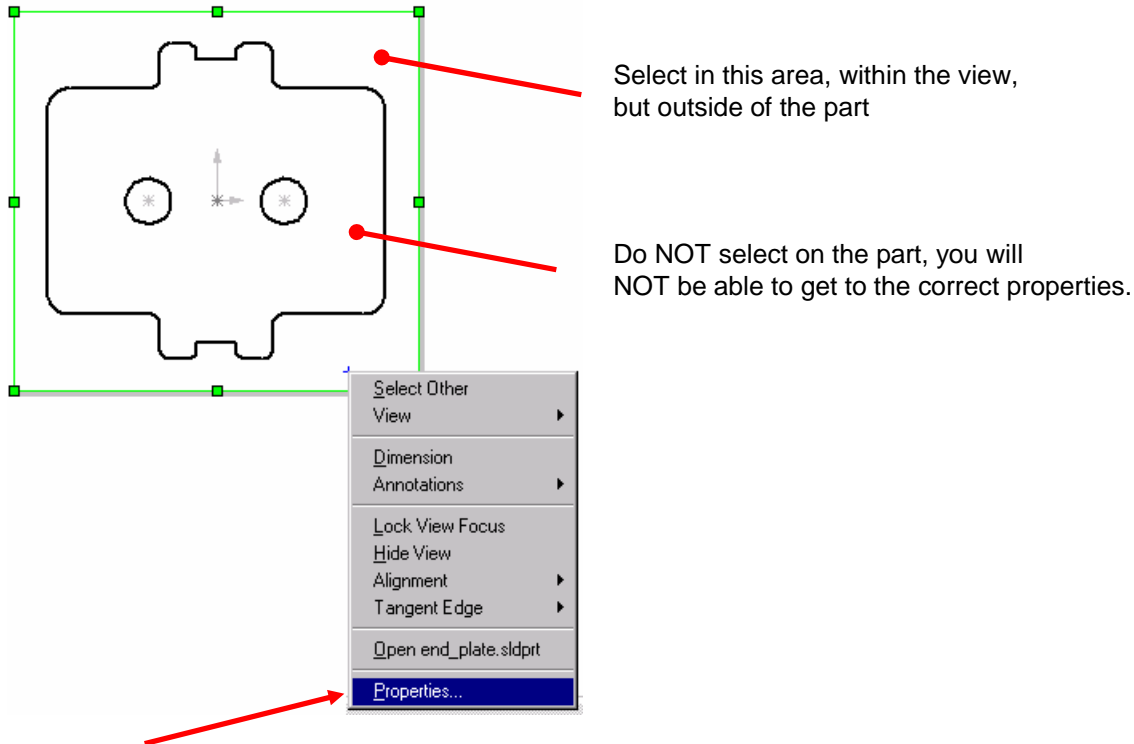**Select Current > Click OK > Return to the drawing file**

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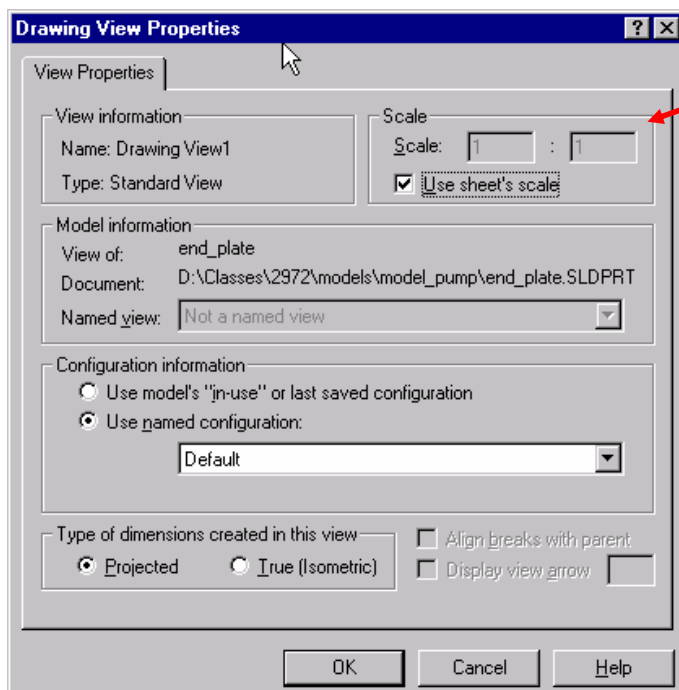
Close the part file (not drawing file) from which you acquired the view.

V. MAKING DXF FILES IN SOLIDWORKS CONT.

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

Select in this area, within the view, but outside of the part

Do NOT select on the part, you will NOT be able to get to the correct properties.

Select Other
View ▸
Dimension
Annotations ▸
Lock View Focus
Hide View
Alignment ▸
Tangent Edge ▸
Open end_plate.sldprt
Properties...

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**

Drawing View Properties

View Properties

View information
Name: Drawing View1
Type: Standard View

Scale
Scale: |1| : |1|
☑ Use sheet's scale

Model information
View of: end_plate
Document: D:\Classes\2972\models\model_pump\end_plate.SLDPRT
Named view: Not a named view

Configuration information
○ Use model's "in-use" or last saved configuration
◉ Use named configuration:
Default

Type of dimensions created in this view   ☐ Align breaks with parent
◉ Projected   ○ True (Isometric)   ☐ Display view arrow

OK   Cancel   Help

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)
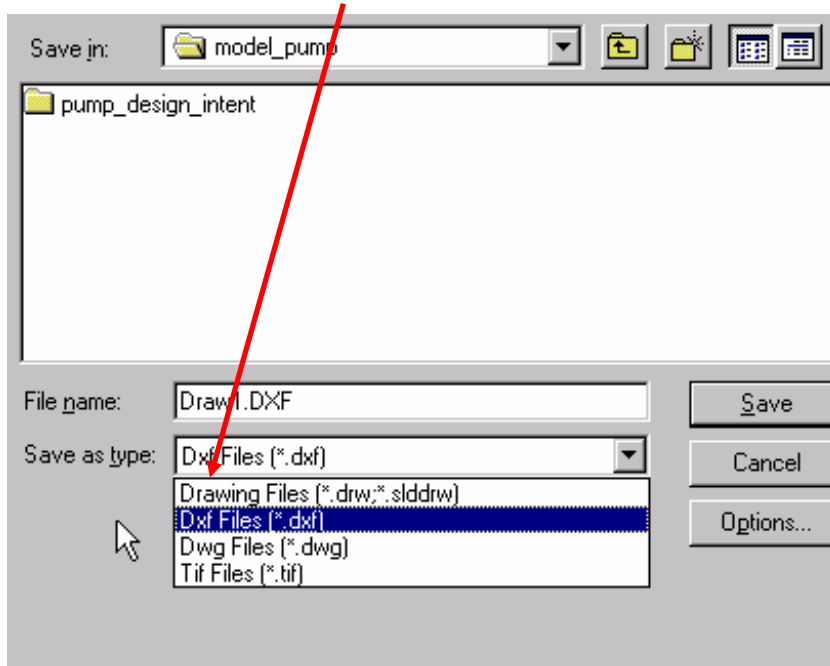
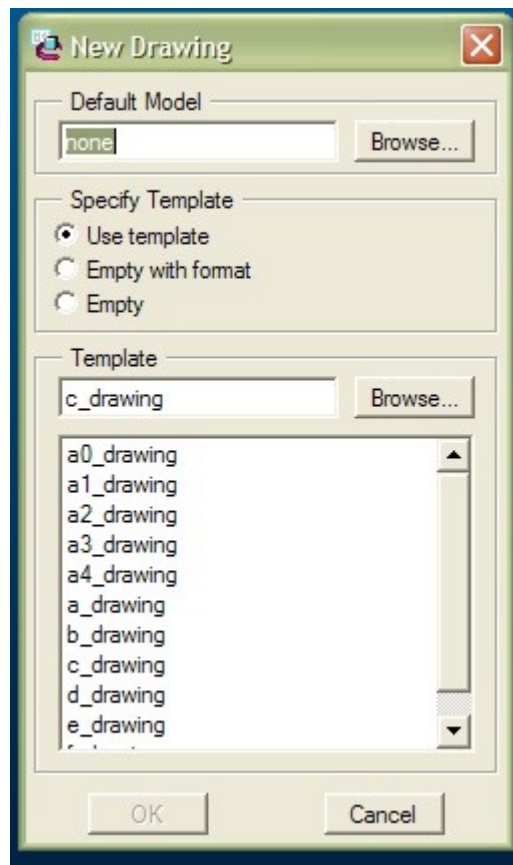## VI. STEPS IN MAKING DXF FILES IN PROENGINEER 2001

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.

VI. STEPS IN MAKING DXF FILES IN PROENGINEER 2001 CONT.

You should now have a blank drawing sheet/screen. In the Menu Manager, choose
View -> Add view -> General -> Full view -> No Xsection -> Scale -> Done

You will be prompted to place the new view on the drawing (somewhere on the screen), do this by clicking in the center of the drawing window.

Now you will be prompted to enter the scale of the view. **It is important that the DXF file be the same scale (i.e. size) as the part you want to make. Key in a value of 1.0 for the scale.**

After you key in the scale, an isometric view of your part will appear along with the following dialog box:
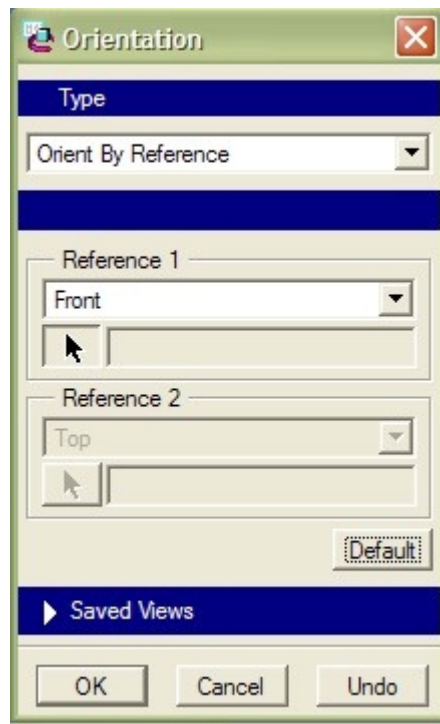
This dialog will allow you to change the orientation of the view (i.e. from isometric to front or side) so that it is a view of the part you wish to convert to DXF. You need to choose two surfaces/planes (see two selection boxes in figure above) to tell ProE how to orient the part. **You must orient the part so that we see only the 2D cross section that is to be cut.**
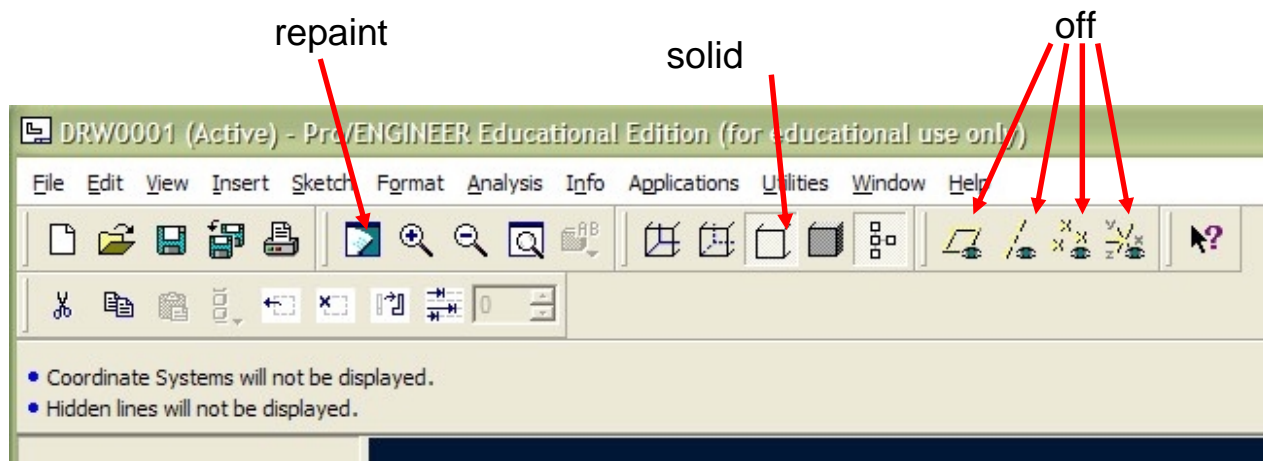
VI. STEPS IN MAKING DXF FILES IN PROENGINEER 2001 CONT.

At this time it **is very important** that you turn all datum planes, points, surfaces, axis… off.
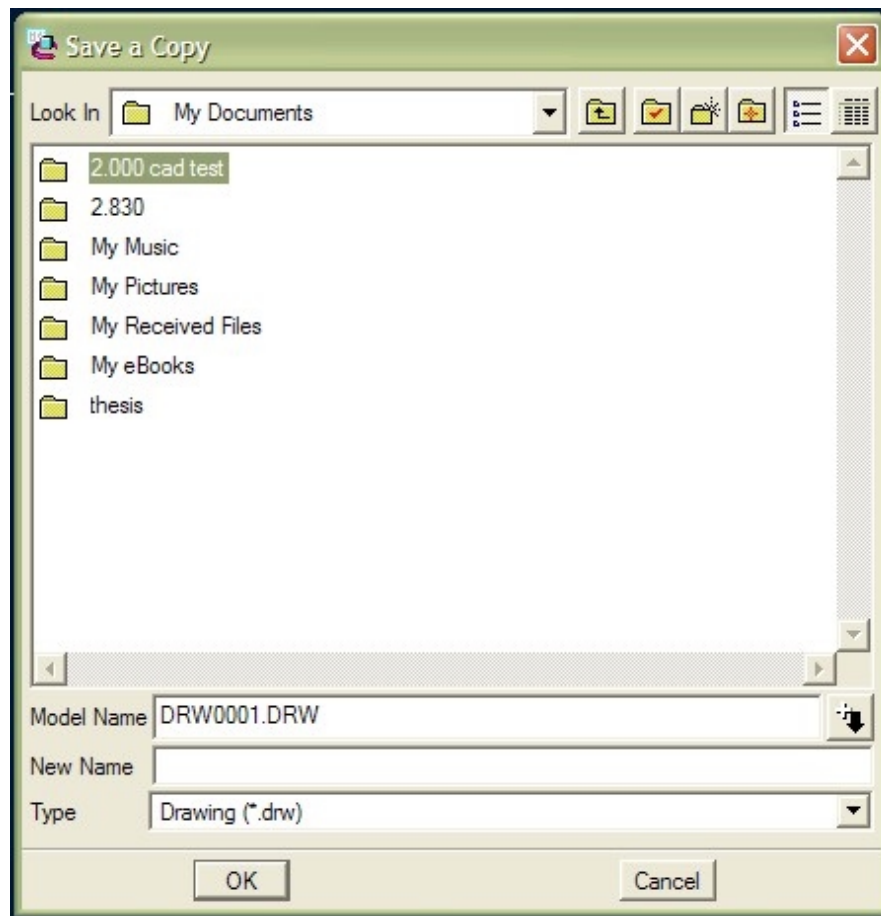
Now choose the non-shaded solid format for the view.

Then hit the repaint button… (see below).

VI. STEPS IN MAKING DXF FILES IN PROENGINEER 2001 CONT.

Next, go File>Save a copy.

The following dialog should appear: (see below)

Step 1: Under "type" choose dxf…

Step 2: Under "new name" choose a name for your dxf file according to the naming convention in section IV. Make sure you use the file name convention in section IV.

Click OK…

You're done!