

DXF Export for Pro/ENGINEER Help

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DXF Export for Pro/ENGINEER Help

Welcome to *DXF Export for Pro/ENGINEER* Help!

This help file provides all the information you need to understand and use *DXF Export for Pro/ENGINEER*. Click the links below or use the Contents tab on the left to browse help topics. You can also scroll the topics on the above right hand pane.

The *DXF Export for Pro/ENGINEER* help file is divided into the following sections:

- [Introduction](#) - Introduction to *DXF Export for Pro/ENGINEER*.
- [Command Reference](#) - Quick reference of all the commands added by *DXF Export for Pro/ENGINEER*.
- [Tutorial](#) - Tutorial to get you started in the shortest possible time.

If you need any additional information please [contact](#) SYCODE.

Note:

- If you have trouble loading the *DXF Export for Pro/ENGINEER* plug-in file into the Pro/ENGINEER Wildfire environment, please refer to the [ReadMe](#) file.
- Please read the [License Agreement](#) before using the software.

Introduction

DXF Export for Pro/ENGINEER is an AutoCAD DXF file export plug-in for Pro/ENGINEER Wildfire. This plug-in gives Pro/ENGINEER Wildfire the ability to export 3D solid data from a Pro/ENGINEER part and assembly document to 3d solids or meshes in an AutoCAD DXF file.

Pro/ENGINEER already has the ability to export AutoCAD DXF files. However, the export feature is limited to 2D drawings only and cannot export parts and assemblies. *DXF Export for Pro/ENGINEER* reads the solid models from the current Pro/ENGINEER part or assembly and tessellates the solid bodies into individual triangular meshes. These meshes are then exported to a DXF file as 3D Polyface Mesh, 3D Face, Line or Point objects.

DXF Export for Pro/ENGINEER is very easy to use. Once installed, it must be loaded into the Pro/ENGINEER Wildfire environment. Follow the simple instructions in the "ReadMe" document which can be found in the installation folder. Once loaded, it adds new commands to Pro/ENGINEER Wildfire and a submenu called "DXFExport" to the Pro/ENGINEER Wildfire menu.

DXF Export for Pro/ENGINEER comes with detailed documentation as well as a [tutorial](#) to get you started in the shortest possible time.

If you need any additional information please [contact](#) SYCODE.

The DXF File Format

The AutoCAD DXF (Drawing Interchange Format, or Drawing Exchange Format) is a CAD data file format, developed by Autodesk as their solution for enabling data interoperability between AutoCAD and other programs. DXF was originally introduced in December 1982 as part of AutoCAD 1.0, and was intended to provide an exact representation of the data in the AutoCAD native file format, DWG (Drawing), for which Autodesk didn't (and doesn't) publish specifications.

DXF is probably one of the most widely supported vector formats in the world today. DXF is rich in features, including: support for 3D objects, curves, text, associative dimensioning, etc. Although DXF was developed to represent the data used in a CAD program, it is used by many programs as a "least common denominator" format for the exchange of many different types of data, most commonly vector-oriented information but also text and 3D polygons.

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ReadMe

This is the *DXF Export for Pro/ENGINEER* ReadMe file.

DXF Export for Pro/ENGINEER is a plug-in for Pro/ENGINEER Wildfire

Installation:

=====

DXF Export for Pro/ENGINEER is a Pro/TOOLKIT application and needs to be registered with Pro/ENGINEER Wildfire before it can be used. Once you register the plug-in with Pro/ENGINEER Wildfire, it will be automatically loaded every time you start Pro/ENGINEER Wildfire. Before you register DXF Export for Pro/ENGINEER it is important that you note down the folder where the plug-in was installed. The default installation location is "C:\Program Files\SYCODE\DXF Export for Pro ENGINEER".

Plug-in Registration:

=====

1) The plug-in installer copies a file called protk.dat into the installation folder. Open it in NotePad and observe the lines starting with "exec_file" and "text_dir".

2) The line starting with "exec_file" is the full path of the plug-in DLL. You need to choose the correct plug-in DLL depending upon your version of Pro/ENGINEER Wildfire. Modify the file path to match your plug-in installation folder and replace XXX in the file name as explained below:

For Pro/ENGINEER Wildfire 1.0 and 2.0, replace "XXX" with "DXFExport_VC60.dll"

For Pro/ENGINEER Wildfire 3.0, replace "XXX" with "DXFExport_VC71.dll"

For Pro/ENGINEER Wildfire 4.0 and 5.0 (32 Bit), replace "XXX" with "DXFExport_VC80.dll"

For Pro/ENGINEER Wildfire 4.0 and 5.0 (64 Bit), replace "XXX" with "DXFExport_VC80x64.dll"

For example, if you have installed DXF Export for Pro/ENGINEER in the default location and you are using Pro/ENGINEER Wildfire 4.0 (64 bit) the line will be as follows:

```
exec_file C:\Program Files\SYCODE\DXF Export for Pro  
ENGINEER\DXFExport_VC80x64.dll
```

IMPORTANT: This is a critical step. The plug-in will not load into Pro/ENGINEER Wildfire if this step is not carried out properly. To better understand this step you may want to browse to the installation folder (typically C:\Program Files\SYCODE\DXF Export for Pro ENGINEER) and take a look at the file names of the plug-in DLL's copied into it.

3) The line starting with "text_dir" tells Pro/ENGINEER Wildfire where to look for supporting files required by the plug-in. Modify the folder path

to match your plug-in installation folder.

4) Save protk.dat and exit NotePad.

5) When Pro/ENGINEER Wildfire starts it looks for a file called protk.dat and loads the plug-ins listed in it. Pro/ENGINEER Wildfire looks for protk.dat in a folder called "text" located in the Pro/ENGINEER Wildfire installation folder. For example, in the case of Pro/ENGINEER Wildfire 4.0, the "text" folder may be "C:\Program Files\proeWildfire 4.0\text". Copy the protk.dat that you just edited to the Pro/ENGINEER "text" folder.

IMPORTANT: If there is already a file called protk.dat in the "text" folder, it means that you already have registered other plug-ins. Do not overwrite it. Instead, append the contents of the protk.dat file you just edited to the contents of the existing protk.dat file in the "text" folder.

Now DXF Export for Pro/ENGINEER is ready for use.

Usage:

=====

DXF Export for Pro/ENGINEER adds new commands to Pro/ENGINEER Wildfire. It also adds a submenu to the Pro/ENGINEER menu called "DXFExport" containing these new commands. Simply click on the menu items in this sub menu to run these new commands.

The new commands are:

- 1) Export - Exports a DXF file
- 2) Help - Display the DXF Export for Pro/ENGINEER help file
- 3) Register - Display the DXF Export for Pro/ENGINEER product registration dialog box
- 4) About - Display the DXF Export for Pro/ENGINEER about box

Support:

=====

For support, please E-Mail support@sycode.com

Contact

Mailing address	S1/116, Nova Cidade Complex, NH 17, Alto Porvorim, Goa - 403521 India
Phone	+91 9822689298
Fax	+91 832 2416615
E-Mail	info@sycode.com
Web Site	http://www.sycode.com

Command Reference

DXF Export for Pro/ENGINEER adds a new menu to Pro/ENGINEER Wildfire. The 'DXFExport' menu consists of the following commands:

Export	Exports a DXF file
Help	Displays the <i>DXF Export for Pro/ENGINEER</i> help file
Register	Registers your copy of <i>DXF Export for Pro/ENGINEER</i>
About	Displays the <i>DXF Export for Pro/ENGINEER</i> about box

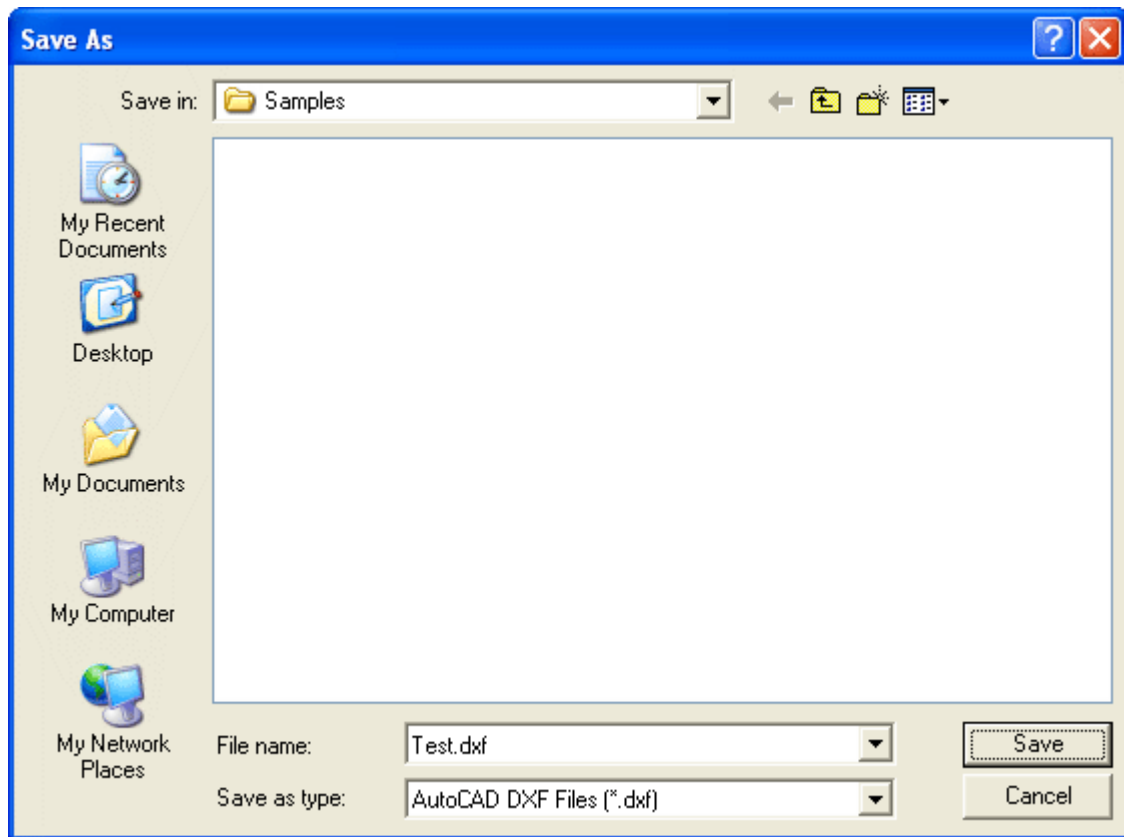
Export

Menu Button DXFExport > Export

This command tessellates the active Pro/ENGINEER part or assembly and exports the resulting meshes to a PolyFace Mesh objects in an AutoCAD DXF file.

To export to a DXF file

1. Select *Export* from the "DXFExport" menu.
2. The Pro/ENGINEER Wildfire "Save As" dialog box is displayed. Enter the output file name of a DXF file and click *Save*.



3. The "DXF Export for Pro/ENGINEER Options" dialog box is displayed. Set the options as desired and click *Ok*.

Help

Menu Button DXFExport > Help

This command displays the help file. The [Welcome](#) page is a good place to start.

About

Menu Button DXFExport > About

This command is used to display the *DXF Export for Pro/ENGINEER* About box. The About box lists the product and its version.



Register

Menu Button DXFExport > Register

This command is used to register your copy of *DXF Export for Pro/ENGINEER*.

You need a license key to start using the software. A license key can be of two types: Trial or Permanent. A trial key is issued to you instantly and completely free of cost so that you may try the software before you purchase a license. A permanent key is issued after you have purchased a license.

SYCODE License Server

Product Name: DXF Export for Pro/ENGINEER Wildfire 4.0

Host ID: 00-1D-7D-FD-AD-1D

License Key: jSAn0D100C01p6c0F044D040AAq2y308M
Fh707Gc1A57Q0D40A0ID71A0P93C160A1
4T5E75785U0BDtC1d07Bh50ijD8AvRcC3

=== LICENSE KEY PARAMETERS ===
Key type: Trial
Expires: 25-Jul-2009
Runs remaining: 10

How do I get a Trial Key?

Simply click the 'Request Key' button and select 'TRIAL KEY' as the type of key being requested. Fill the form and submit it. A trial key will be immediately sent to you by email.

How do I get a Permanent Key?

To get a permanent key you need to have already purchased a license. If you have not yet purchased a license you can do so by clicking the 'Buy Online' button. After you place your order click the 'Request Key' button and select 'PERMANENT KEY' as the type of key being requested. Fill the form and submit it. Your order will be verified and a permanent key will be sent to you by email within one business day.

IMPORTANT:

- (1) Please ensure that your computer is connected to the internet before you request for a key.
- (2) Please be sure to enter a valid email address as the key will be sent to you by email.

DXF Export for Pro/ENGINEER Help

(3) To avoid disruption in your work-flow we urge you to place your order before the trial period ends.

If you need anything else please send an email to register@sycode.com

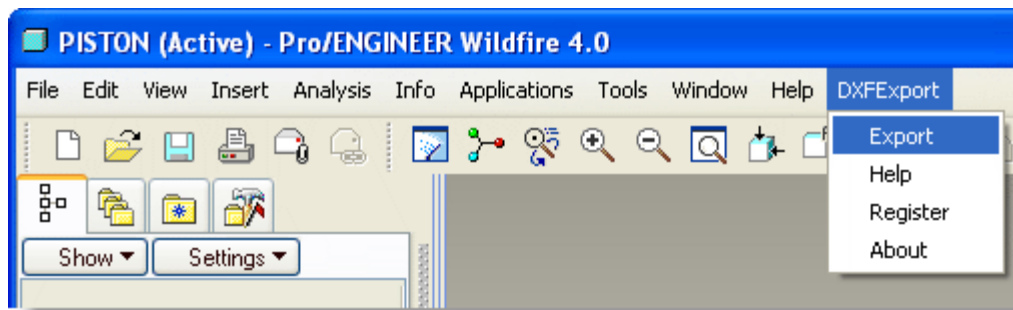
Tutorial

This tutorial provides the necessary information and hands-on experience to help you get started with *DXF Export for Pro/ENGINEER*.

Step 1: Start Pro/ENGINEER Wildfire.

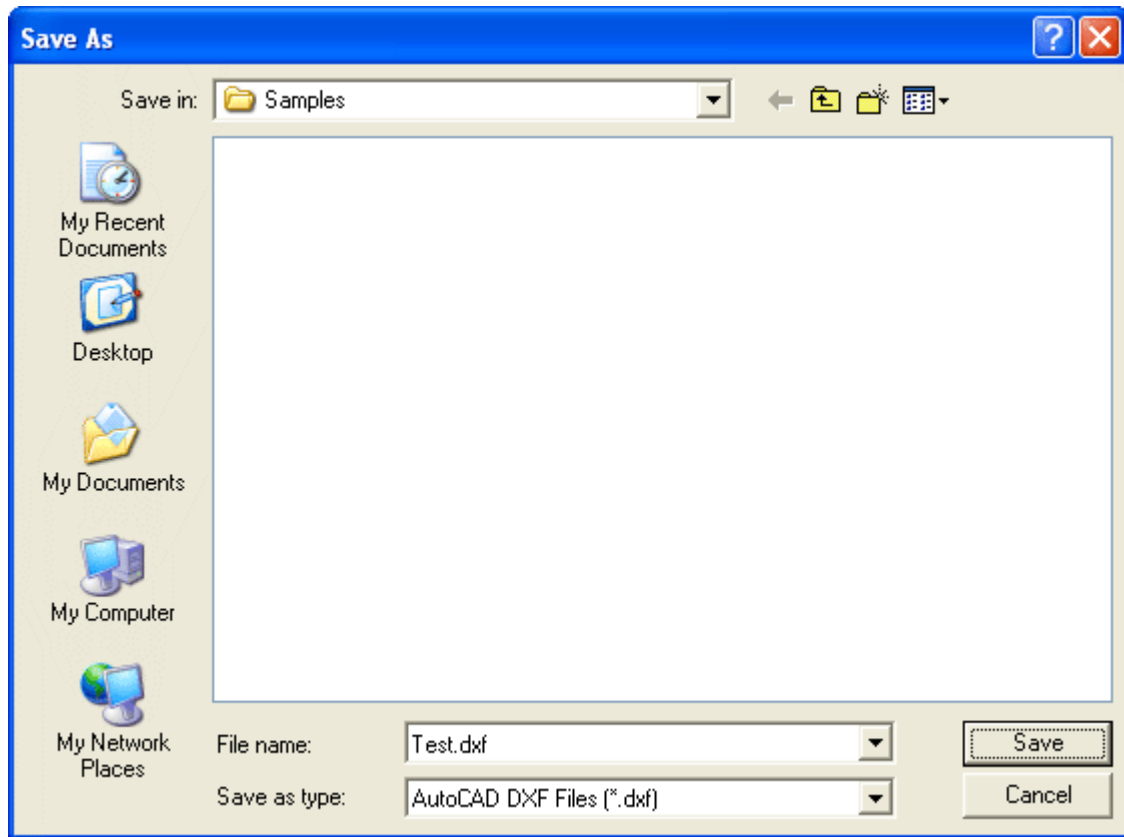
Step 2: Open the file "C:\Program Files\DXF Export for Pro ENGINEER\Samples\Piston.sat" as a part document.

Step 3: Select *Export* from the "DXFExport" menu.

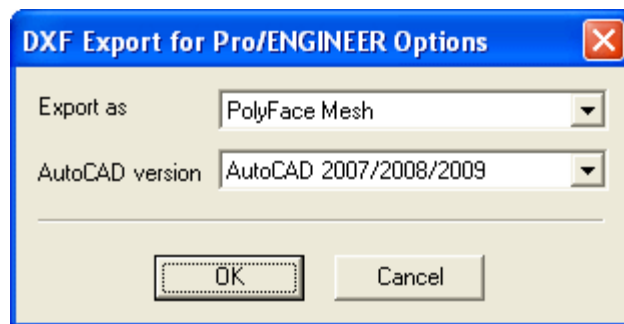


Step 4: The Pro/ENGINEER "Save As" dialog box is displayed. Enter the output file name of a DXF file and click Save.

DXF Export for Pro/ENGINEER Help



Step 5: The "DXF Export for Pro/ENGINEER Options" dialog box is displayed. Set the options as desired click *Ok*.



The DXF file will be created in the location you specified.