

Installation Strategies

by Eric Horn

Most software installation is fairly straightforward. You insert a CD and set up the program without regard to how the software will be used. However, this is not a useful way to install Pro/ENGINEER software. As most Pro/ENGINEER users are aware, you must take into account how you will use Pro/E and where everything will reside. In addition to the importance of the PTC set-up routine, there are factors to consider after the software is installed. This article will focus on those factors.

Do not assume you are ready to work immediately after inserting the Pro/ENGINEER 2001 CD into your system and installing it on your computer at "D:\ptc\proe2001." Instead, stop and think about how you work on a day-to-day basis. Consider where you want to do your work. If you are not using a data management system, such as Pro/INTRALINK, you will work on your hard drive or a network drive. This method is risky because you have a greater chance to lose data. It is important to set up a working directory or your computer will default to "D:\ptc\proe2001\bin," and this is not an efficient way to work. Each time you launch a session, working and system files will be saved to this location. It is recommended *not* to save any files into the Pro/E installation location.

For this example we will use "D:\work" as the location for the working directory. You can place this directory anywhere, but make sure it is created as a new and separate directory and make sure it is not located in the installation location. To set it up, create a shortcut on your desktop

to launch Pro/ENGINEER. Right-click on that icon and select **Properties**. You should see the shortcut properties. Enter the directory you just created under **Start in**, as shown in *Figure 1*. Each time you launch a session of Pro/ENGINEER, this location will be used as the working folder.

If you have been working with Pro/ENGINEER, you likely have a set of config files you use. The config files can be stored in one of two locations. There are two locations Pro/ENGINEER will search to load these files—the "D:\ptc\proe2001\text" directory and the "D:\work" directory. Use the first option if you want the config files to load every time and you do not want to modify them. Doing this will allow you to create a supplemental config file in your working directory that can complement your existing config file or override the options in it. It is important to note that config files load in a sequence. The latest config file will override previous options that are the same. The "D:\ptc\proe2001\text" directory is read first, followed by "D:\work". Keep this in mind when creating your config files and when making changes. The two most common config files are a config.pro, which is used to store all the config options and settings, and a config.win, which is used to store screen options for toolbars and button locations. There also is a menu.def, which is used to customize the side-menu functions.

At this point, your files are set up in a way that will keep you productive. Next, you want to take a look at how you are storing Pro/ENGINEER standard files. This is your library of files that you regularly use. Do not confuse this with a library of components. The standards include files such as drawing formats, material files, drawing notes, drawing symbols, etc. Correctly setting this up in advance can save time and avoid headaches.

You can build this standards area as you work in Pro/ENGINEER. You may want to start by creating a formats folder and building from there. Following are the steps involved to build a good foundation for a Pro/E installation. This will provide a place for the standard files you use daily.

First, choose a location to store your standards. If you installed Pro/ENGINEER software at "D:\ptc\proe2001," create a folder called "D:\ptc\pro_std" as a logical place for standards storage. This will create a folder called "pro_std" in the same location as the software was installed. The standards are placed here because these are files you do not change on a daily basis. In addition, placing them in this folder keeps them from crowding the work area and safe from unintentional modification.

Next create a subfolder within the pro_std folder to store the files. Create the following folder within this location to use as a basis for your standards, see *Figure 2*:

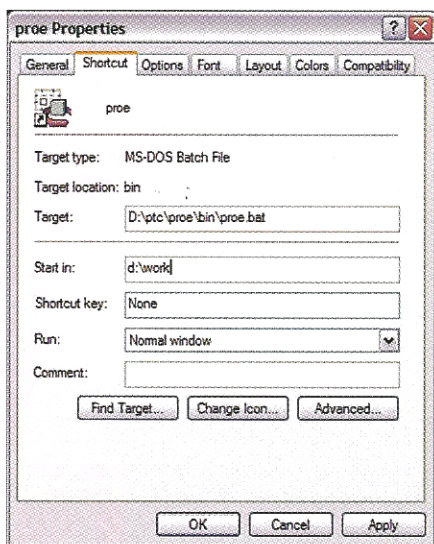


Figure 1.

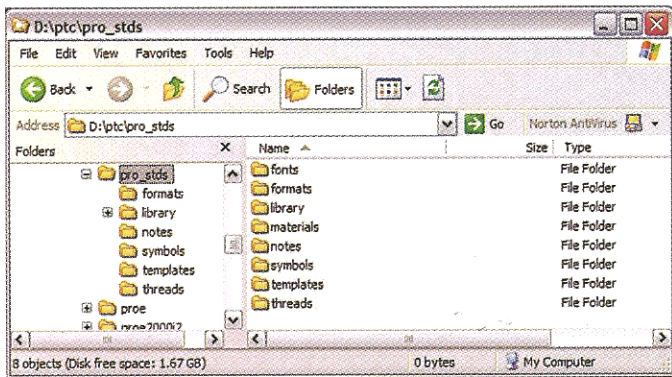


Figure 2.

- Fonts
- Formats
- Library
- Materials
- Notes
- Symbols
- Templates
- Threads

This will give you a solid foundation from which to start. For some of these folders you can set a config option to locate the folder on your system. These config options are listed below next to the folder names, as shown in Figure 3:

- Fonts (pro_font_dir)
- Formats (pro_format_dir)
- Library (pro_library_dir)
- Materials (pro_material_dir)
- Notes (pro_note_dir)
- Symbols (pro_symbol_dir)

The templates folder is set aside to store start parts and assemblies. The reason there is no option for the directory is that you must specify the file name itself. The following config options are used to set the template for each component type:

- Template_designasm**
- Template_sheetmetalpart**
- Template_solidpart**
- Template_drawing
- Template_ecadasm
- Template_ecadpart
- Template_mfgcast
- Template_mfgmold
- Template_mold_layout

The first three, which are listed in bold type, are the three most commonly used templates. This config option will specify a default assembly, part and sheetmetal template file to use. When you create a new assembly, part or sheetmetal part, it automatically will use the template specified in the config file to use instead of asking each time.

How to configure different options in Pro/ENGINEER is essential to keep in mind when installing and setting up a working environment on your system. Once everything is in place, you will be much more produc-

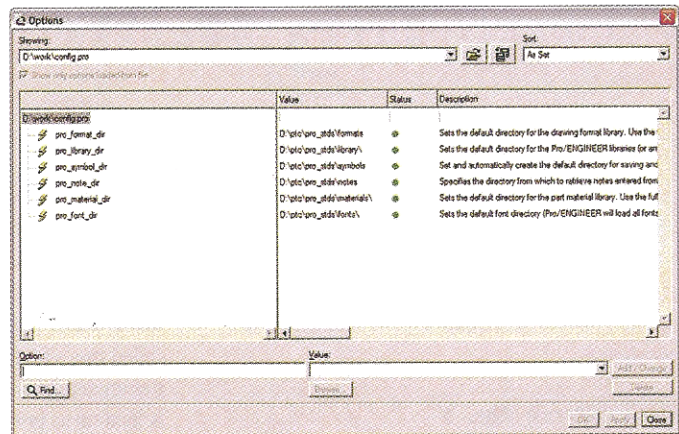


Figure 3.

tive. Do not forget to place new standards in your standards location as you are working. Most of these file types do not change significantly from previous versions of Pro/ENGINEER, but it is always a good idea to open and check the compatibility whenever you install a new version.

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