Parametric Technology Corporation

Schools Edition
Pro/ENGINEER® Wildfire® 3.0
Installation and Administration Guide

May 2006

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Installing and Setting Up the Schools Edition

This Pro/ENGINEER Schools Edition guide provides instructions on the installation, implementation, and use of the Pro/ENGINEER Wildfire 3.0 software for teachers and students. This guide is subject to change without notice and without any commitment by Parametric Technology Corporation (PTC).

Topic **Page**

Overview of Pro/ENGINEER Schools Edition

Welcome to the Pro/ENGINEER Wildfire 3.0 Schools Edition. PTC hopes that you enjoy using the world's most popular 3D MCAD solution. This guide provides instructions on the installation, implementation, and use of the software. The Schools Edition Package includes the following items:

- Pro/ENGINEER Schools Edition software on one CD-ROM.
- Online Help for English, also on the CD-ROM. Other languages are available during an optional setup.
- Pro/ENGINEER tutorials on the Pro/ENGINEER landing page.

Usage Guidelines for Schools Edition Software

Pro/ENGINEER Schools Edition is available to schools in the PTC Design & Technology in Schools Program. PTC does not offer installation and troubleshooting support. See http://www.ptc.com/for/education/troubleshooting_faq.htm for a FAQ of common questions. Alternatively, students can ask their teacher for assistance. If you experience problems during installation, see the Troubleshooting section.

Note: You cannot use the Schools Edition software for commercial, professional, or for-profit purposes. The software is not compatible with commercial versions of Pro/ENGINEER.

Languages

Choose from English, Chinese (Traditional and Simplified), French, German, Italian, Japanese, and Spanish for the Pro/ENGINEER Schools Edition. See the installation instructions for installing the desired language.

File Compatibility of Pro/ENGINEER Editions

The different editions of Pro/ENGINEER and the compatibility between them are shown in the next table.

Pro/ENGINEER Edition	Training	Schools	Schools Advanced	University Plus	Student	Commercial
Training		No	No	No	No	No
Schools	No		Yes	Yes	Yes	Yes*
Schools Advanced	No	Yes		Yes	Yes	Yes*
University Plus	No	Yes	Yes		Yes	Yes*
Student	No	Yes	Yes	Yes		Yes*
Commercial	No	Yes*	Yes*	Yes*	Yes*	
* Requires the use of the Education-to-Commercial conversion tool, a part of University Plus.						

Features Specific to the Schools Edition

Pro/ENGINEER Schools Edition is different from the commercial software in the following ways:

- No access to floating options.
- Schools Edition banners on the borders of plots and drawings.
- Online registration and license generation process.
- No running of software over a network. You must install the software on every computer using Pro/ENGINEER.
- No software compatibility with commercial versions of Pro/ENGINEER. Objects created with the other Education Editions *are* compatible.

Meeting Platform Requirements

Before installing Pro/ENGINEER Schools Edition, your computer must meet the requirements in the next two tables. For platform details, see http://www.ptc.com/WCMS/files/30874/en/30874en_file1.pdf.

Operating System	Service Pack	Processor
Windows XP Professional and Home Edition, 32-bit	Base OS, Service Pack 1 and 2	INTEL Pentium/Xeon family and AMD Opteron family
Windows 2000, 32-bit	Base OS, Service Pack 1,2,3 and 4	INTEL Pentium/Xeon family

Minimum platform requirements follow. For computers upgraded from Windows 95/98/Me to Windows 2000/XP, delete the `winbootdir` variable, otherwise Pro/ENGINEER fails to start.

Platform	Minimum Requirement
Main Memory	256 MB (512 MB or higher)
Available Disk Space	900 MB (Pro/E), 1.2 GB (with Pro/ENGINEER Mechanica)
Swap Space	500 MB (1024 MB or higher)
CPU Speed	Pentium III 700 MHz (1 GHz or higher)
Graphics Cards	Cards must support OpenGL. For supported graphics cards, see http://www.ptc.com/WCMS/files/30874/en/30874en_file1.pdf. If your graphics card does not support OpenGL, see the section Configuring Graphics Acceleration.
Browser	Microsoft Internet Explorer 6.0 SP1 or later
Network	Microsoft TCP/IP, Ethernet network adapter
Monitor	1024 X 768 (or higher) resolution with 24-bit or greater color

Platform	Minimum Requirement
File Systems	NTFS
Mouse	Microsoft approved 3-button mouse
Distribution Media	CD-ROM or DVD drive

Installing Pro/ENGINEER Schools Edition

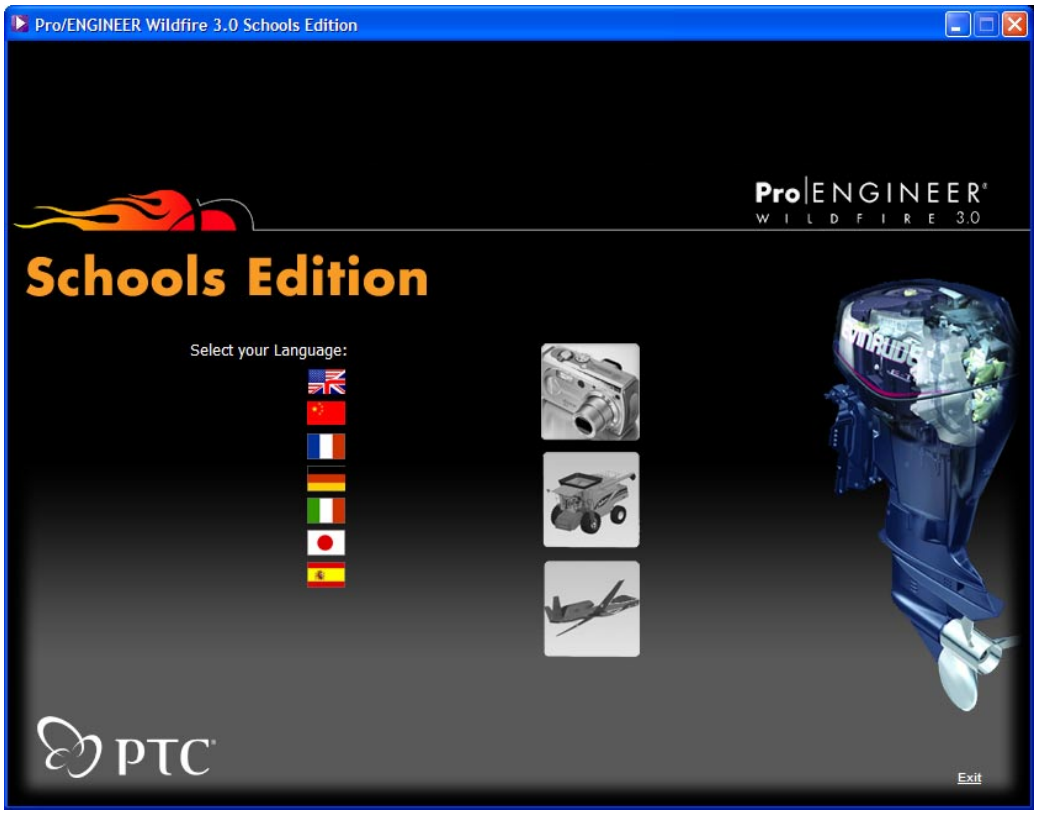
To perform the installation, use the Pro/ENGINEER Wildfire 3.0 Schools Edition CD. To streamline the process, first connect to the Internet and take the following actions first. Instructions follow in later sections.

- Open a PTC online account, if you do not have one.
- Generate a new software license.

Logging In with Administrator Privileges

To open the Pro/ENGINEER **Schools Edition** starting screen, follow these steps:

1. Close all applications and disable your screen saver. Failure to do so can produce a poorly installed application
2. Log in as administrator or log on to an account with administrator privileges.
3. Insert the Pro/ENGINEER Schools Edition CD into your CD-ROM drive. If installation does not start, see the Troubleshooting section at the end of this guide. The **Schools Edition** starting screen appears.



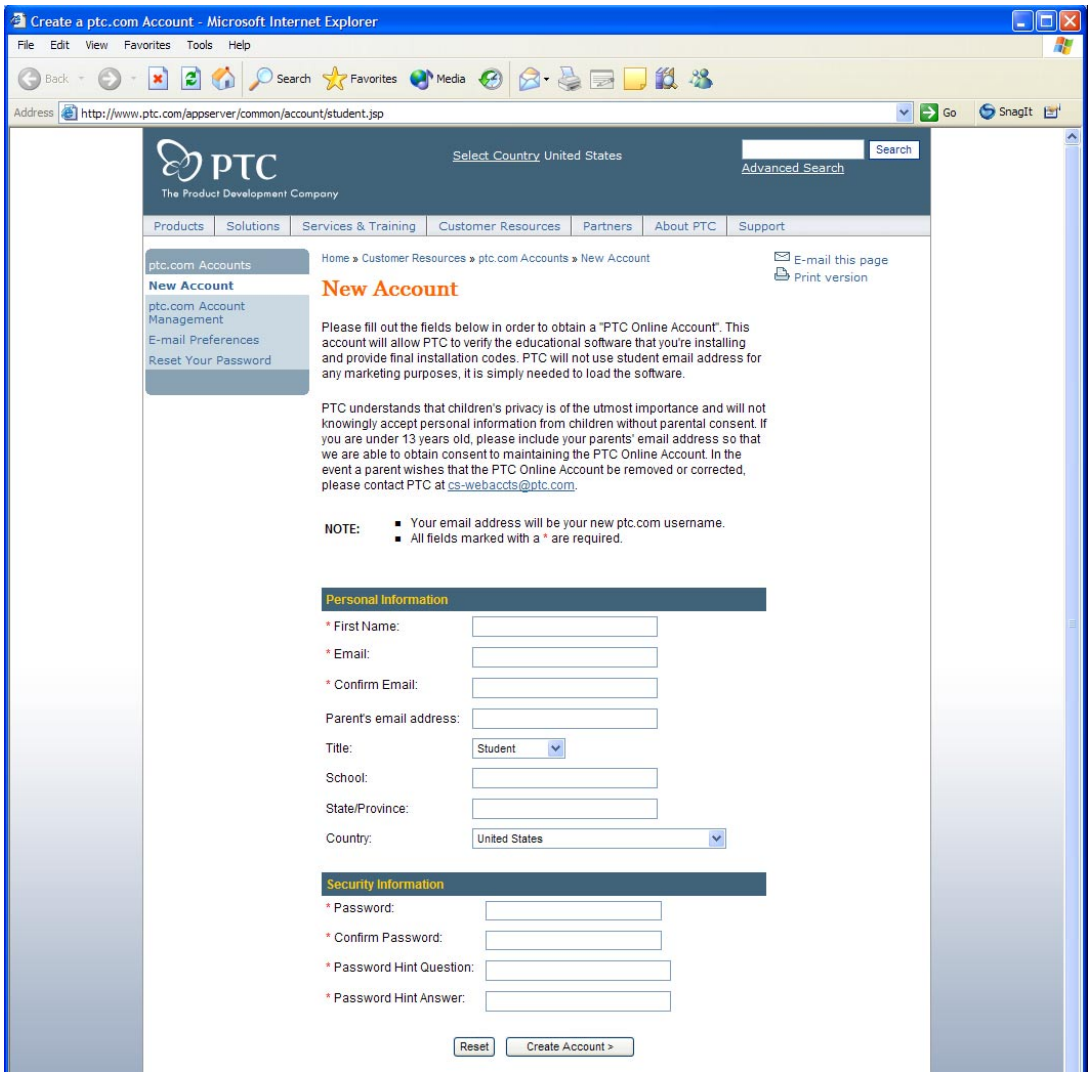
4. Click the country flag to choose the language for the software installation. The **Schools Edition** information screen appears.

You must have a PTC online account to generate a license. Create your online account now, if you have not already done so.

Opening a PTC Online Account

If you already have an online PTC account, skip to the next section.

1. In the previous screen under **Installation Instructions**, click **PTC online account** in step 3. The **New Account** screen opens.



2. Fill in any empty text boxes.
3. Click **Create Account**. A confirmation screen appears.
4. Review and print this confirmation. A confirmation of your account is sent to your and your parent's e-mail addresses.

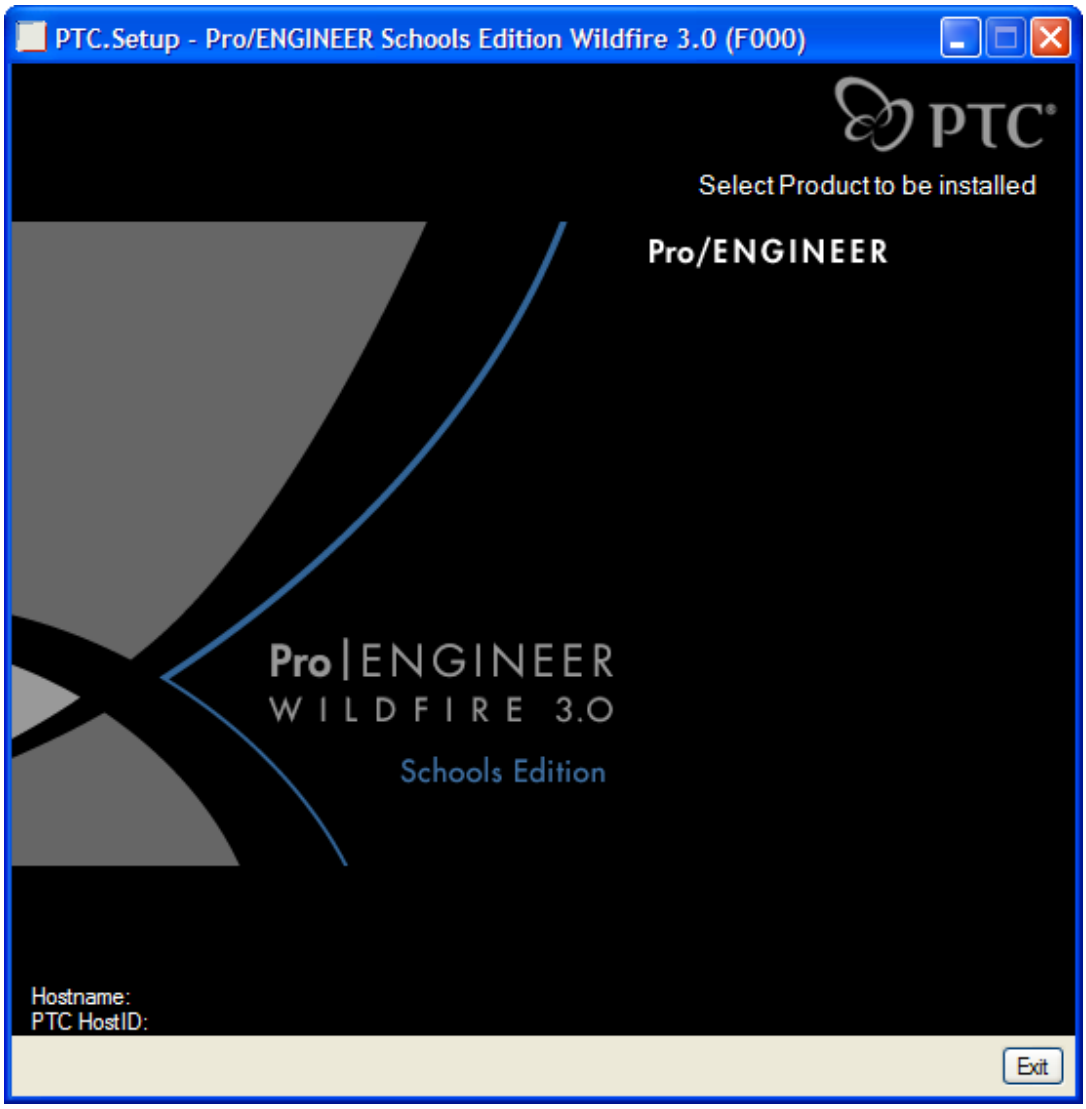
Generating a New License File

The method of generating a license file depends on whether the computer on which Pro/ENGINEER will run is connected to the Internet. From the **Schools Edition** information screen shown previously, follow these steps.

- No Internet for installed software—Skip to the section Generate an Offline License File.
- Internet connection for the installed software—Follow the next steps.

1. Click **Install Now** to begin the installation for Pro/ENGINEER Schools Edition. The **PTC.Setup** screen appears.

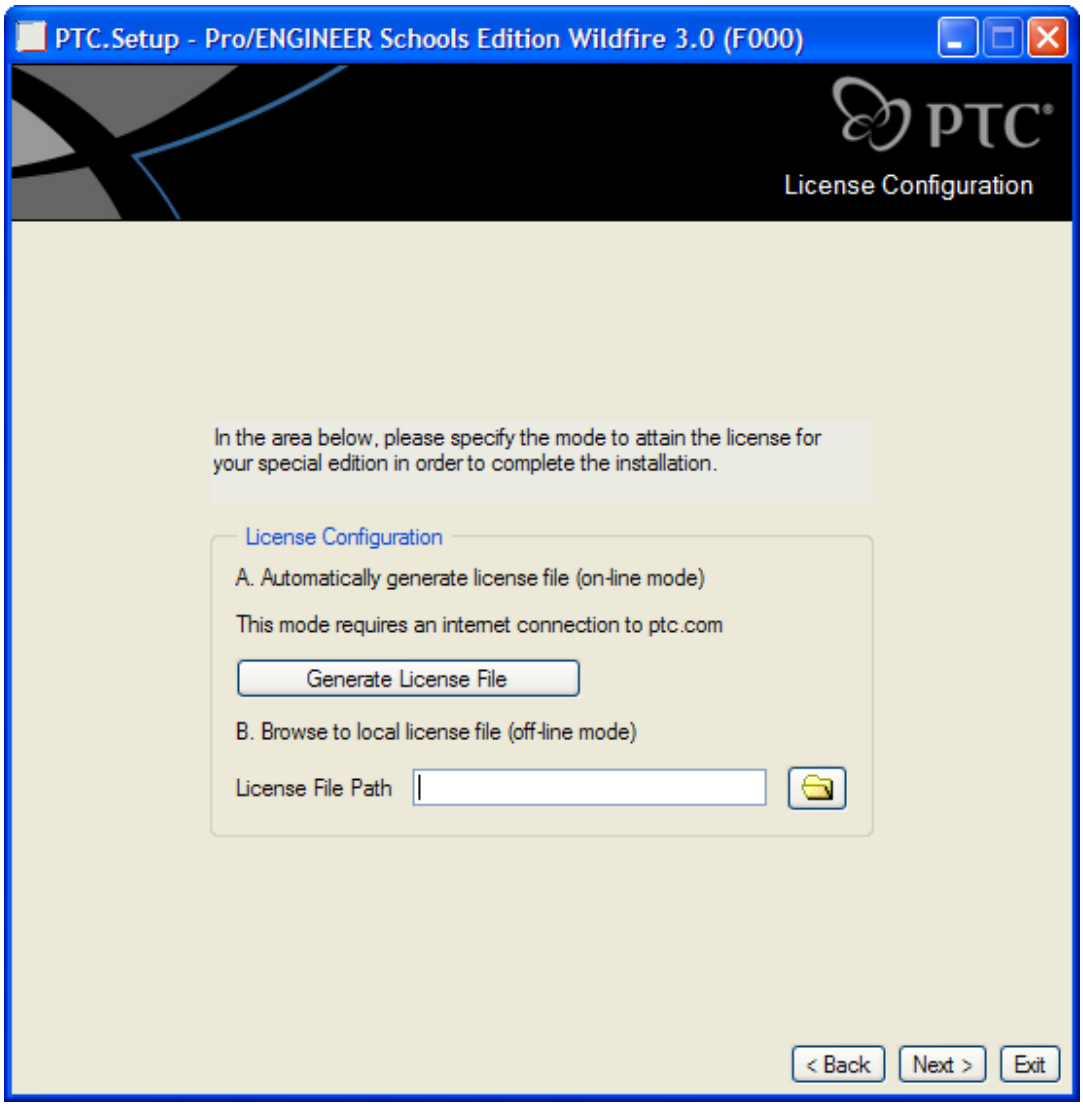
   Note: During installation, you can click **Exit** at any time to stop the process.



Generate an Online License

If the computer for Pro/ENGINEER installation is connected to the Internet, use online license generation:

1. Click **Pro/ENGINEER** to install the Pro/ENGINEER Schools Edition software. The **PTC.Setup** screen for License Configuration appears.



2. Click **Generate License File**. A prompt appears.
3. Log in with your user name and password. PTC.Setup uses your PTC HOST ID and Schools Edition ID to generate the license file. The path appears in the **License File Path** box.
4. Click **Next**. The **Define Installation Components** screen appears.

Note: Now skip to the section Selecting the Pro/ENGINEER Modules to Install.

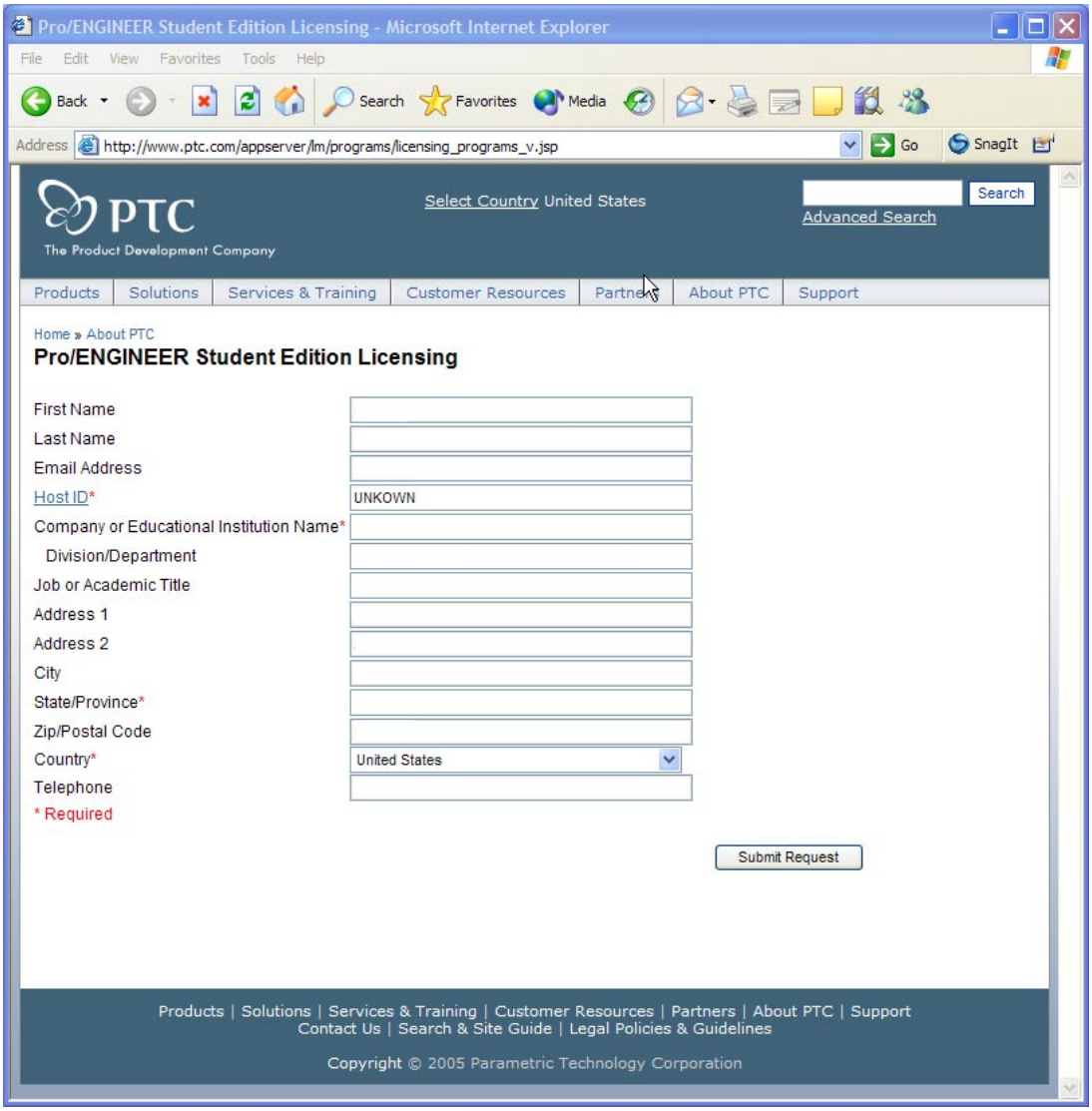
Generate an Offline License

If the computer on which Pro/ENGINEER is to be installed is not connected to the Internet, use offline license generation. Perform these steps on a computer connected to the Internet. You must then move the license to the computer on which the software will be installed.

1. Click **Generate License for Offline Mode** at the bottom of the **Schools Edition** information screen shown previously. A prompt appears.
2. Log in with your user name and password. The **Pro/ENGINEER Schools Edition Licensing** screen opens. Most of the text boxes are filled, except for the **HostID**.

3. In the **HostID** textbox, type the Host ID for the computer on which the Schools Edition will be installed.

   Note: For more information on your Host ID, see the Troubleshooting section.



4. Click **Submit Request.** A confirmation screen appears.

5. Review the information before closing your browser.

Continue with the next two sections to save you License Pack. Then you can install the Schools Edition on your offline computer.

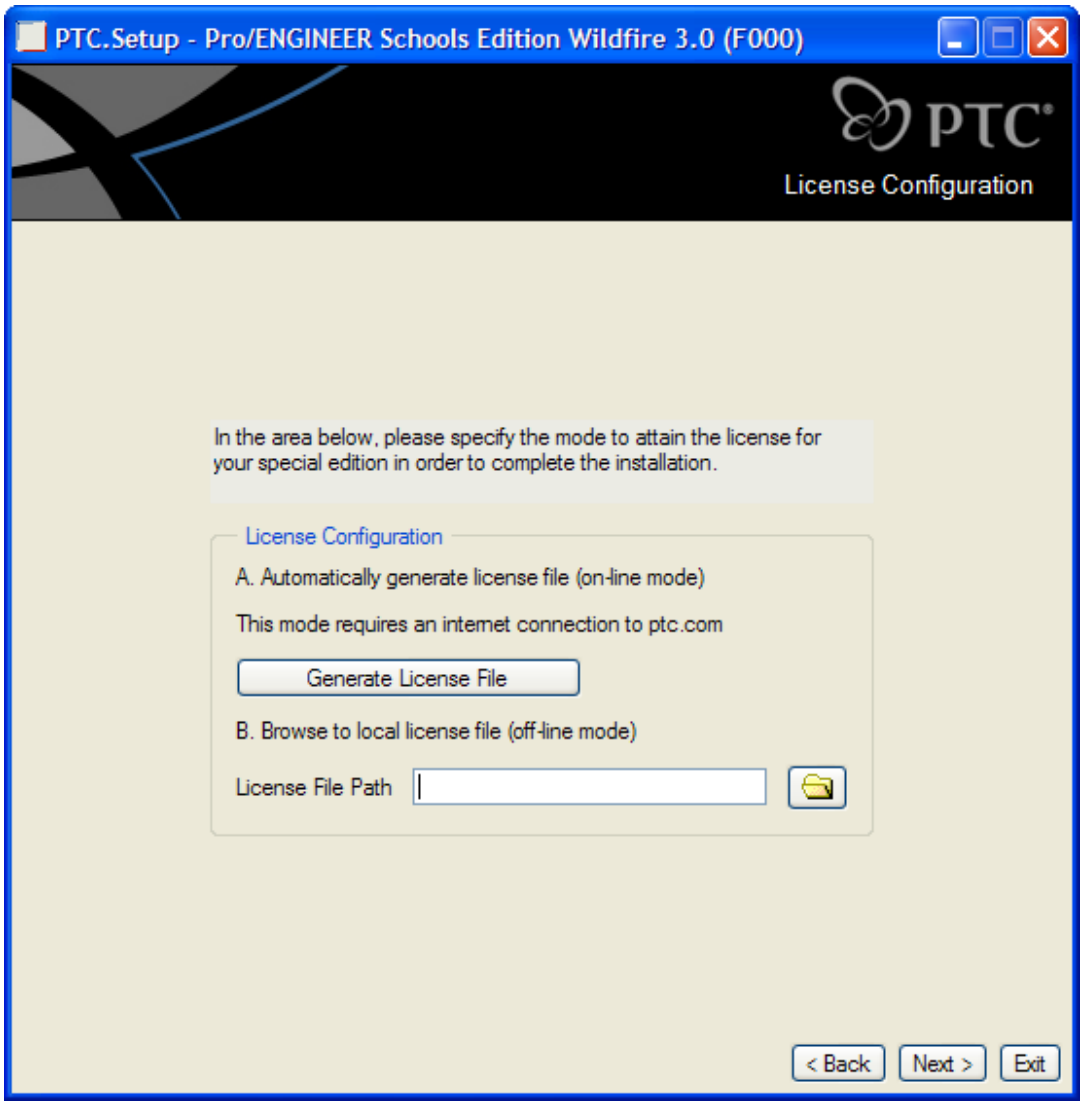
Saving Your License Pack for Offline Use of Pro/ENGINEER

Your License Pack for offline use arrives by e-mail within 2 hours. It contains a file attachment named `sw_license_email_0000.dat` or something similar. Save the file to the offline computer in an easily remembered location, for example, `C:\TEMP`.

Caution: Do not edit the name or contents of this license file. Changes to it make the license invalid and prevents Pro/ENGINEER Schools Edition from starting.

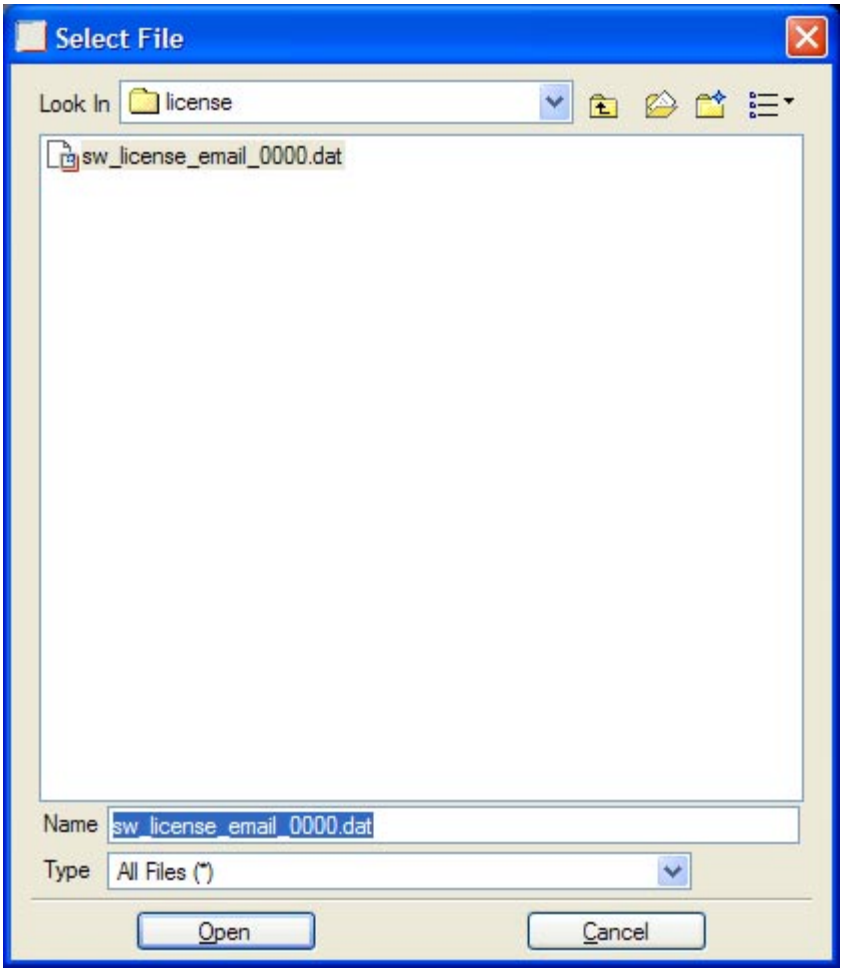
To open the Pro/ENGINEER **Schools Edition** starting screen, follow these steps:

1. Close all applications and disable your screen saver. Failure to do so can produce a poorly installed application
2. Log in as administrator or log on to an account with administrator privileges.
3. Insert the Pro/ENGINEER Schools Edition CD into your CD-ROM drive. If installation does not start, see the Troubleshooting section at the end of this guide. The Schools **Edition** starting screen appears.
4. On the **Schools Edition** starting screen, click the language flag for the desired language. The **School Edition** information screen appears.
5. Click **Install Now**.
6. Click **Pro/ENGINEER** to install the Pro/ENGINEER Schools Edition software. The **PTC.Setup** screen for License Configuration appears.



7. Click the Folder icon to the right of the **License File Path** box.

8. Navigate to the directory with your Pro/ENGINEER Schools Edition license file as shown in the **Select File** dialog box. The directory (Folder) path depends on where you saved your license file. It may not appear exactly as in the example.



9. Select the license file and then click **Open**. The **Select File** dialog box closes the path appears in the **License File Path** box on the **PTC.Setup** screen for License Configuration.

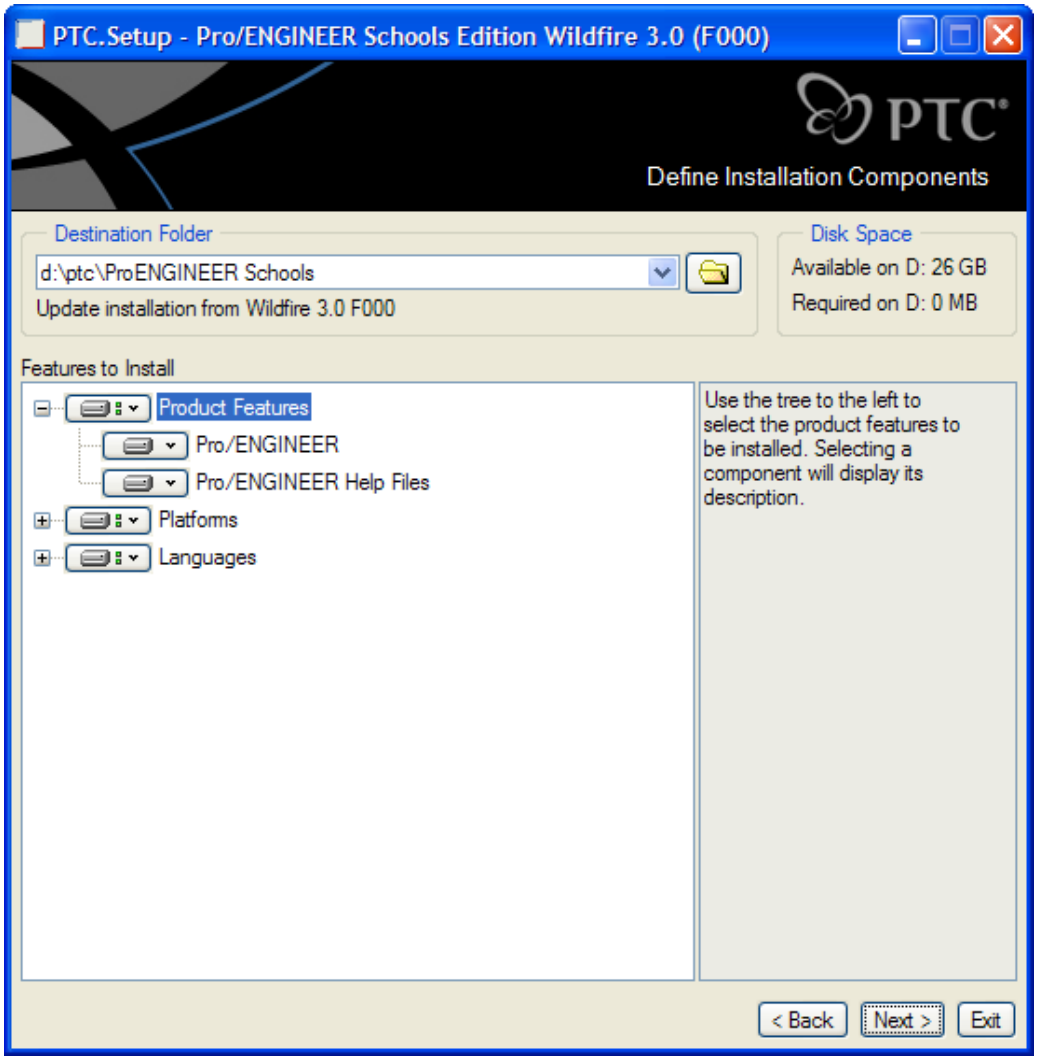
   Note: To be prepared for a reinstallation, write down in a safe place the path of the license file. PTC strongly recommends this action.

10. **Click Next**. The **Define Installation Components** screen appears.

Selecting the Pro/ENGINEER Modules to Install

In the **Define Installation Components** screen, you can choose not to install some modules of Pro/ENGINEER.

1. Right-click a module to omit it from the installation, and click **do not install**.

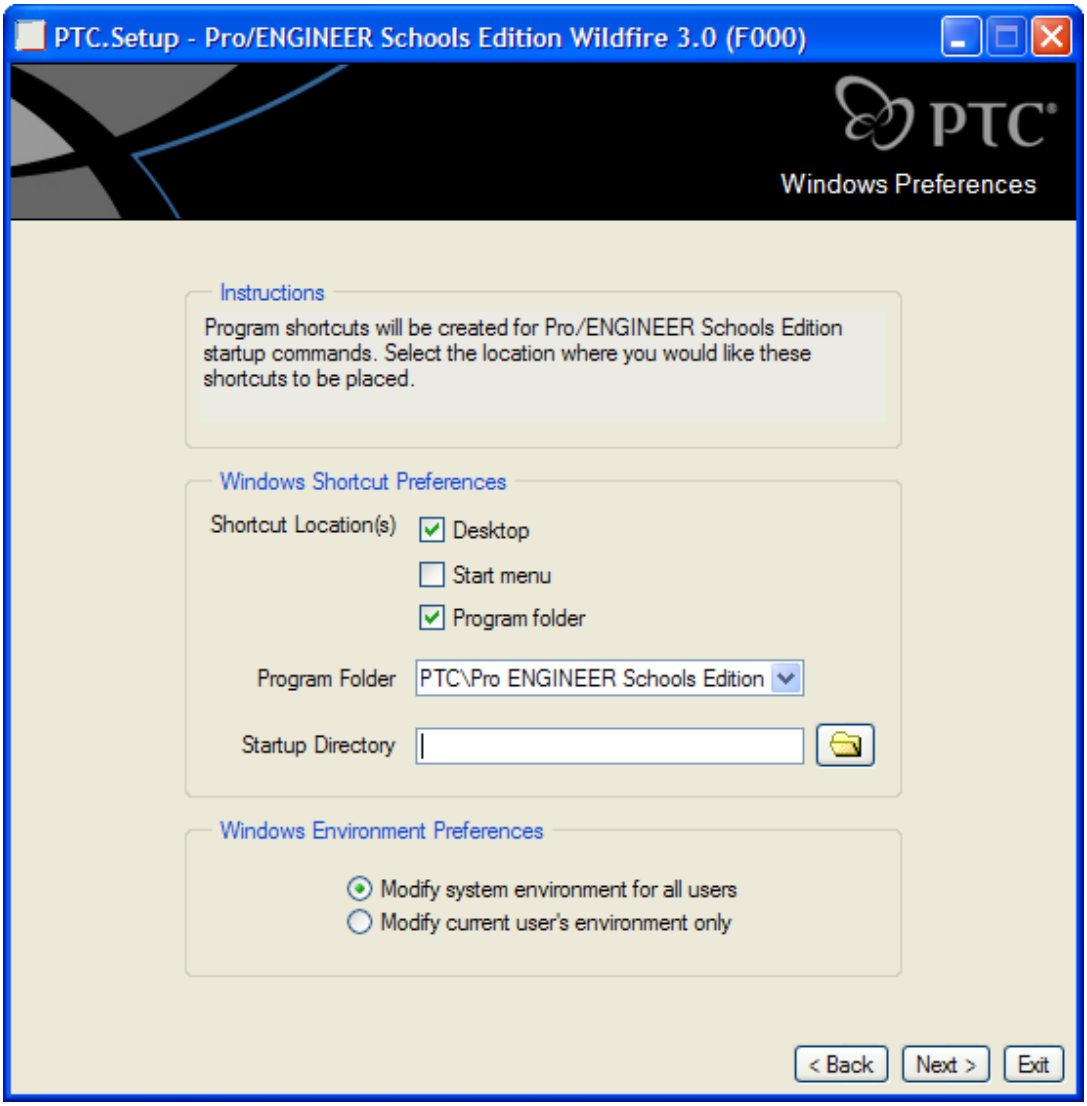


Destination Folder	`C:\Program Files\ProENGINEER Schools Edition`. To change this default location, type another path, or click **Browse** to select another path. PTC strongly recommends a new installation. The path cannot be an existing PTC directory, or PTC.Setup assumes you are trying to update or reconfigure Pro/ENGINEER Schools Edition.
Platforms	The only platform available in Pro/ENGINEER Schools Edition is i486_nt. This option is the default.
Languages	Check any additional languages for installation. Additional language packs require more hard drive space.
Pro/ENGINEER (default)	Keep this component to install the files necessary to run the software for Pro/ENGINEER Schools Edition.

2. When finished specifying the installation directory, click **Next**. The **Windows Preferences** screen appears.

Defining Windows Preferences

1. In the **Windows Preferences** screen shown below, you define your Windows preferences.



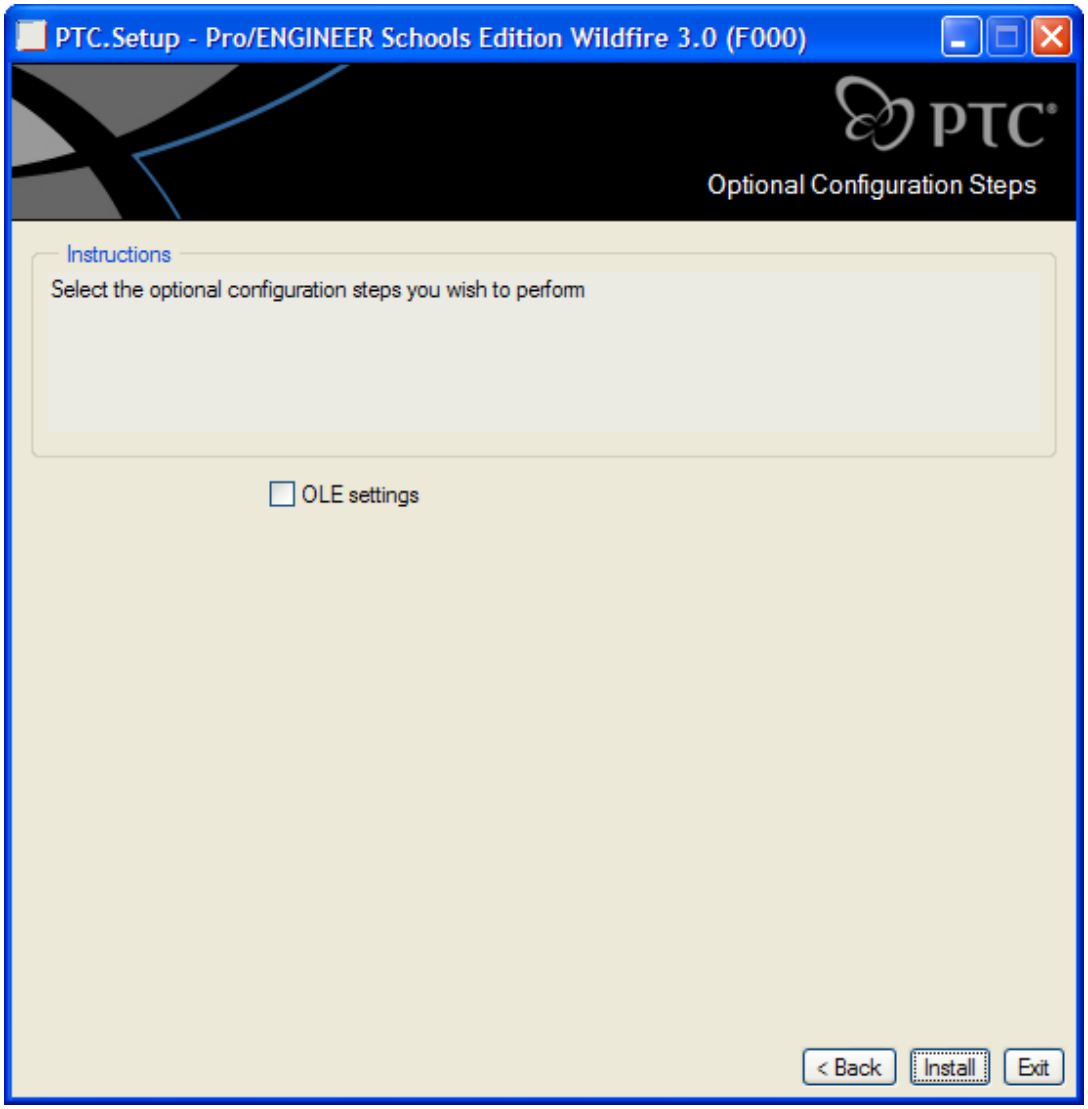
2. Select your **Shortcut Locations**.
3. In the **Startup Directory** text box, type the full path for the location of the shortcuts.
4. Under **Windows Environment Preferences**, choose to modify the system environment for all users or the current user.
5. Click **Next**. The **Optional Configuration Steps** screen appears.

Configuring the OLE Settings

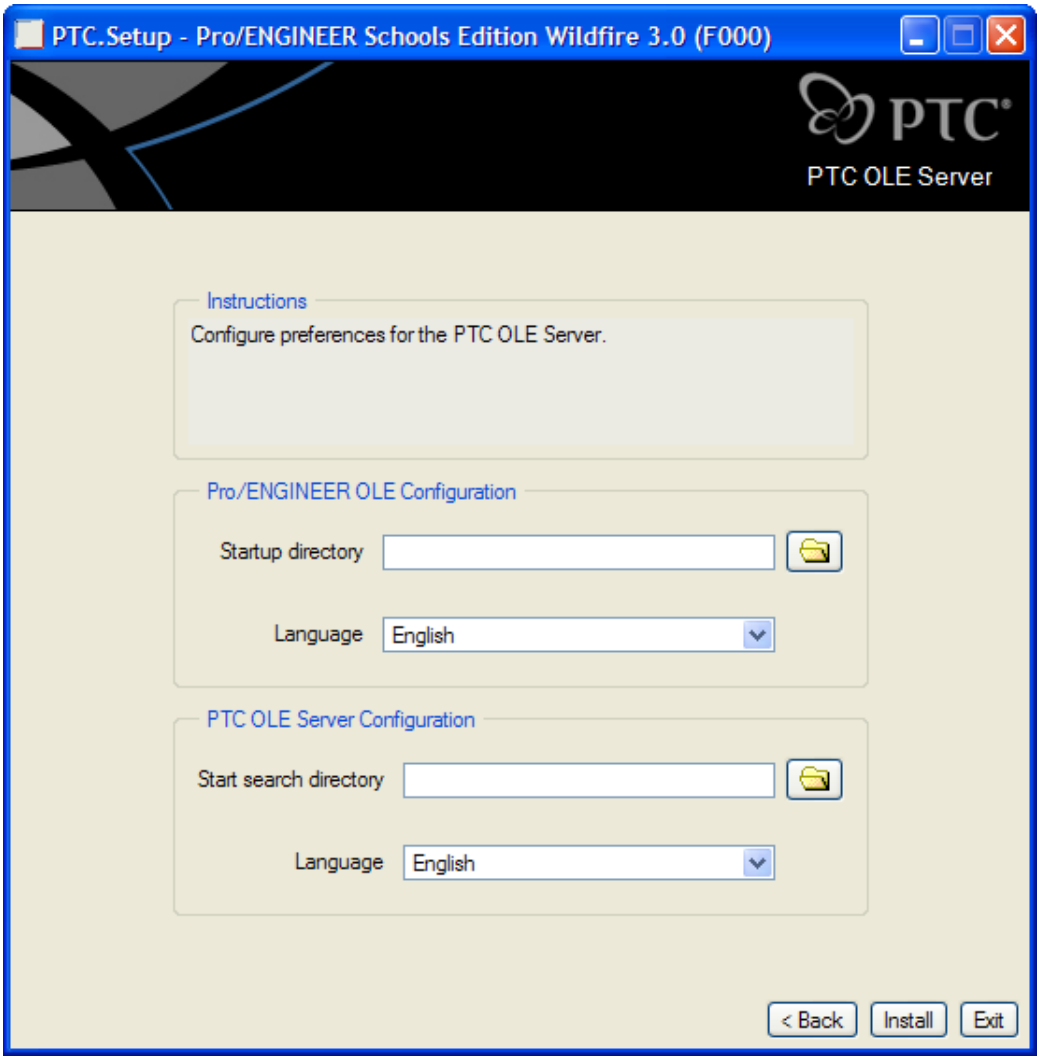
Use the **OLE settings** checkbox to configure Pro/ENGINEER Schools Edition as an OLE (Object Linking and Embedding) Server. With this setting, you can import Pro/ENGINEER files (parts, assemblies, and drawings) into Microsoft documents (Word, Excel, and PowerPoint). You cannot import Microsoft files into Pro/ENGINEER.

If you choose not to use OLE settings, skip to the next section.

The **Optional Configuration Steps** screen below is open from the last step in the previous section. Proceed with the configuration for OLE settings.



1. Select the **OLE settings** check box to customize your installation.
2. Click **Install**. The **PTC OLE Server** screen appears. You can fine-tune the options, although all of the default values work.

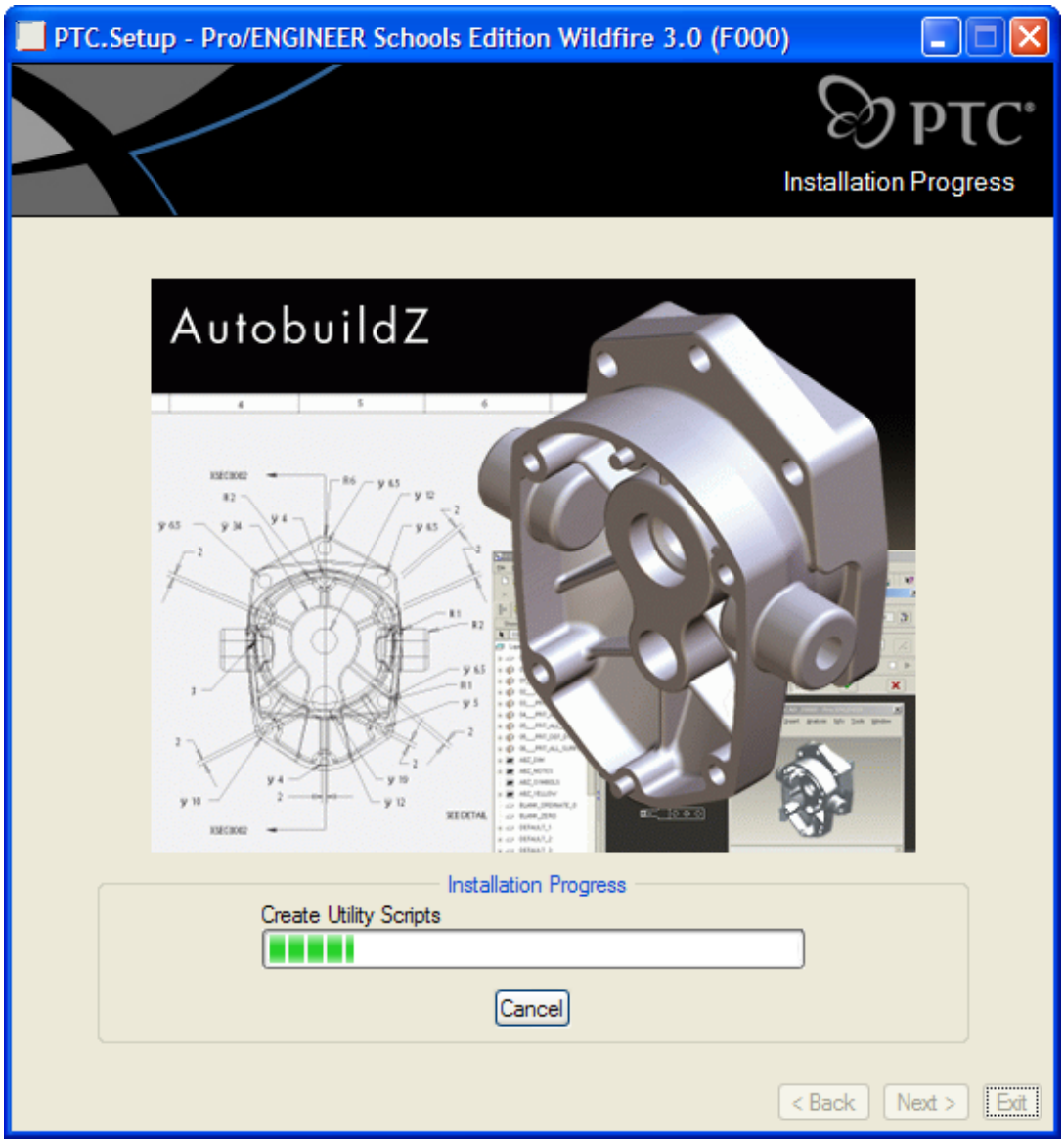


3. In the **Startup directory** text box, enter the full path to the Pro/ENGINEER directory. Typically, this is the Pro/ENGINEER working directory.
4. From the **Language** list, choose your language if other than the given language.
5. In the **PTC OLE Server Configuration**, enter the startup directory of the PTC OLE Server. The Microsoft application (PTC OLE Container) searches for PTC objects in this directory.
6. In the **Language** text box for the server, keep the language or choose another.

Completing Pro/ENGINEER Configuration and Installation

You have almost completed the Pro/ENGINEER installation. To proceed, click one of the following commands:

- Click **Back** to review information.
- Click **Install** to continue installation.
- Click **Exit** to stop the installation.



When Pro/ENGINEER stops, click **Exit**.

Congratulations! You can now use Pro/ENGINEER Wildfire 3.0 Schools Edition.

Configuring and Setting Up Pro/ENGINEER

Pro/ENGINEER provides a highly customizable environment with configuration files to define system-specific and application-specific settings, such as required drafting standards and system color schemes. If a configuration option is in more than one configuration file, Pro/ENGINEER applies the most recently loaded or recently read setting.

- **Protected system configuration files**—At startup, Pro/ENGINEER first reads in a protected system configuration file called `config.sup`. The configuration option settings in this file *cannot* be overwritten by the `config.pro` file of an individual user.
- **Configuration files**—Next, Pro/ENGINEER searches for and reads in configuration files `config.pro`, `config.win`, and `menu_def.pro` from the following directories in the following order:
  - **`<loadpoint>/<filename>`**–Pro/ENGINEER installation directory. Your system administrator can put configuration files here to support school for Windows configuration settings, formats, and libraries. Users starting Pro/ENGINEER from this load point, use the values in this file.
  - **Login directory**—The home directory for your login ID. By placing your configuration files here, you can start Pro/ENGINEER from any directory. You do not need to a copy of the configuration files in each directory.
  - **Startup directory**—Your current or working directory when you start Pro/ENGINEER.
- **Local configuration files**—Last to be read are `config.pro`, `config.win`, and `menu_def.pro` in your startup directory. Therefore, they override any conflicting configuration file option entries on your computer.

Using config.sup and config.pro Files

You can create and store custom configuration files in your working directory. These files can apply to specific projects or to individual user settings.

Config.sup

`Config.sup` is a protected system configuration file. Use this file to ensure that students cannot change configuration settings, such as specified templates, directory settings, and so forth. The system administrator sets configuration options in the `config.sup` file for school-wide use. Values in this file cannot be overridden by the `config.pro` file.

Note: The `config.sup` file must be in the `text` directory in the Pro/ENGINEER load point.

To ensure the `config.sup` is always correct and up-to-date, create a `.bat` file to start Pro/ENGINEER and copy the `config.sup` file to the local computer's text directory. An example of a `.bat` file is in the section Using Trail Files.

Config.pro

By default, Pro/ENGINEER uses a set of predefined `config.pro` settings. When you make modifications to the default settings, a `config.pro` file is created in `C:\Program Files\proeWildfire 3.0\text\`. An example `config.pro` file is in the `pro_standards\config_files` directory.

Creating Configuration Options

To view, modify, or add `config.pro` settings, follow these instructions:

1. On the main toolbar of Pro/ENGINEER, click **Tools** > **Options**. The **Options** dialog box opens.
2. Clear the **Show only options loaded from file** check box. All options are available.
3. Click one or more configuration options from the left column to apply them.
4. Click **Apply**.
5. Save a backup copy of this file to a secure backup directory or disk.

Configuring Graphics Acceleration

One entry in the `config.pro` is for graphics cards. The default is set as follows: `use_software_opengl = yes`, so that Pro/ENGINEER uses OpenGL software for graphics. Check http://www.ptc.com/WCMS/files/30874en_file1.pdf to see if you have a compatible graphics card. If you do, change the `config.pro` entry to `from yes` to `no` to use that graphics card.

Configuring Pro/ENGINEER to Be Like Pro/DESKTOP

You can change `config.pro` options and make Pro/ENGINEER similar to Pro/DESKTOP. Click **Tools** > **Options** and use the **Configuration Properties** dialog box. Change each default setting with an asterisk in the next table to its opposite.

Config.pro Option	Description	Value
`sketcher_blended_background`	Controls whether to use the blended in 3D Sketcher. The option has no effect if the blended background is turned off.	No*/Yes
`sketcher_starts_in_2d`	Defines initial model orientation in Sketcher. Yes–(2D orientation) Looking directly at section (sketching) plane. No–(Orientation unchanged) Sketch directly on the 3D part.	No/Yes*
`sketcher_refit_after_dim_modify`	Refits section after dimension modification in 2D section or when creating the first feature.	No/Yes*

Setting the Config.pro Files

- **.dtl**—By default Pro/ENGINEER sets the drafting standard to ASME / ANSI. To use another standard, add a setting into the `config.pro` file to point Pro/ENGINEER at the required standard. The `config.pro` option is `drawing_setup`. Country-specific files are provided for your convenience.
  - **UK**—`BS8888.dtl` in `pro_standards\config_files`
  - **Australia**—AS1100`.dtl` in `pro_standards\config_files`
- **tree.cfg**—The `tree.cfg` file controls how information is displayed in the Model Tree. This file is in `pro_standards\config_files`. You can control the Model Tree display by inputting the config.pro option mdl_tree_cfg_file and pointing it at the particular tree.cfg in the pro_standards\config_files directory.

- **syscol.scl**—To change the color scheme for Pro/ENGINEER, click **View** > **Display Settings** > **System Colors**. The color scheme is saved in the `syscol.scl` file. An alternative `syscol.scl` file is in the `pro_standards\config_files` directory. To use this syscol file, add the configuration option `system_colors_file` into your `config.pro` file and point it at the required file.

  Note: You can use a sample `syscol.scl` file for making Pro/ENGINEER drawings with a white background like Pro/DESKTOP. To enable this color scheme, set the `system_colors_file` into your `config.pro` file and point it at the required file in the `pro_standards\config_files` directory. You must restart Pro/ENGINEER for the changes to take place.

- **config.win**—The `config.win` file is a database file. It stores window configuration settings, such as toolbar visibility and Model Tree location.

Using Trail Files

Every session of Pro/ENGINEER creates a *trail file*. This file contains all user actions in that session of Pro/ENGINEER. Delete these trail files on a regular basis to free up disk space. By default these trail files are created in the **Start in** directory defined within the Pro/ENGINEER shortcut icon. To view or change this configuration option, right-click the Pro/ENGINEER icon and choose **Properties**.

IMPORTANT: The best location of the trail file directory is on the hard drive of the local computer. Pro/ENGINEER records every command and option in the trail file. Using the trail file directory over the network causes high network traffic.

- To assist with system management, consider setting up a dedicated directory for trail files in the `config.pro`. Then the teacher, student, or IT (Information Technology) Administrator can delete the contents of this directory on a regular basis.

- To set the trail file directory, set the `trail_dir` configuration option in the `config.pro` file to the required directory. Use the full path, for example, `D:\trail\proe_trail_files.`

For an automated approach, you can include a routine in Pro/ENGINEER startup to delete trail files. Create a batch file to do this:

1. Click **Start** > **All Programs** > **Accessories** > **Word Pad.**
2. Type the following commands:

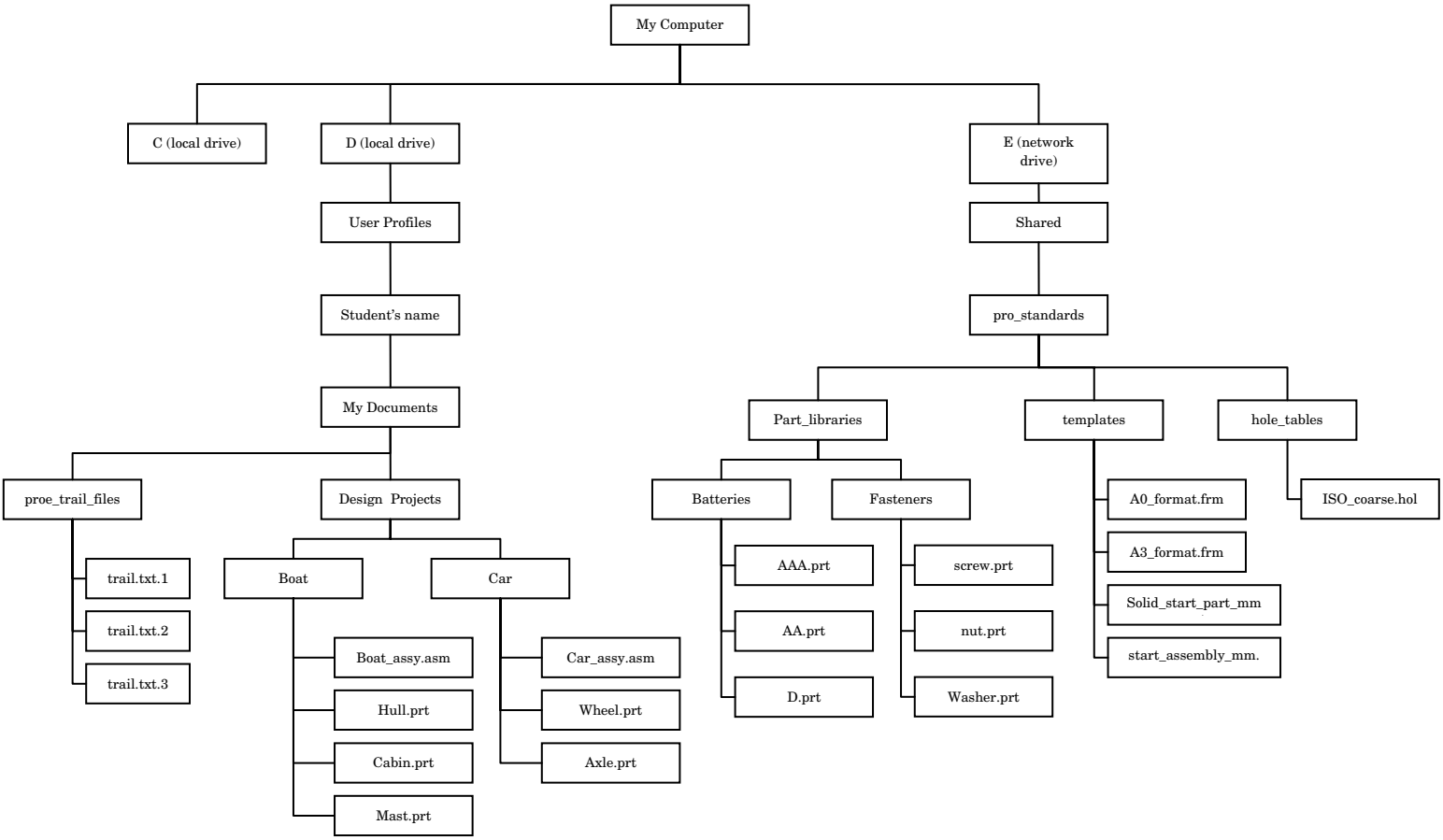
```
echo off.
REM Batch file created to delete Pro/ENGINEER trail files and then start Pro/ENGINEER
REM first set current working directory
set CWD=%cd%
echo.
echo Removing Pro/ENGINEER Trail Files from your directory
echo.
cd /d "c:\pro-standards\trail_files"
erase trail.txt.*
cd "%cwd%"
echo.
echo Now starting Pro/ENGINEER Wildfire 3.0 Schools Edition
echo.
" C:\Program Files\ProENGINEER Schools Edition\bin\proe.exe"
```

3. Click **Save As.** For the **Save as type,** choose **Text Document** and give the file a meaningful name, for example, `proe_clean-trail_start.bat.`

Setting Search Paths

Pro/ENGINEER requires a list of directories in which to search for file types, such as parts or drawings. By default Pro/ENGINEER looks in the current working directory. If parts and assemblies are in other directories, you must list the path explicitly in the `config.pro` or the `search_path.pro`. If you want to use the `search_path.pro` configuration option, the path to `search_path_file` must be in the `config.pro` file.

PTC recommends using the full path for all directories. For example; consider the directory structure in the next figure.



The search path entries required in either the `config.pro` file or the `search_path.pro` file are as follows:

```
E:\shared\pro_standards\part_libraries\Batteries
```

```
E:\shared\pro_standards\part_libraries\Fasteners
```

Note: All students must set their own working directory within Pro/ENGINEER to the specific project. Therefore, teachers need not add design project directories for students, such as `Boat` and `Car`, to the search path.

Pro/ENGINEER does not search subfolders or subdirectories. Every folder or directory with Pro/ENGINEER files must be included in `search_path.pro`. Pro/ENGINEER searches for files in a specific order:

- In session (files opened and then closed but *not* erased)
- In the directory from which the file or assembly is specifically opened

- In the defined working directory
- In directories listed in the search_path.pro file

To ensure you are always using the correct Pro/ENGINEER files, erase the files that are not displayed. See the section Erasing Pro/ENGINEER Files.

Using Template Files

Pro/ENGINEER uses a number of template files whenever you create a new part, assembly, or drawing. The default template files are in the template directory within the Pro/ENGINEER program directories. The default templates are all in Imperial units, for example, inlbs_part_solid.prt and e_drawing.drw.

For metric versions of files, add the following configuration options into your config.pro file and set the entry to point at the required files.

- template_designasm—Specifies the designated template assembly
- template_drawing—Specifies the drawing used as the default drawing template
- template_solidpart—Specifies the model used as the defaults part template

Note: Please see the pro_standards section.

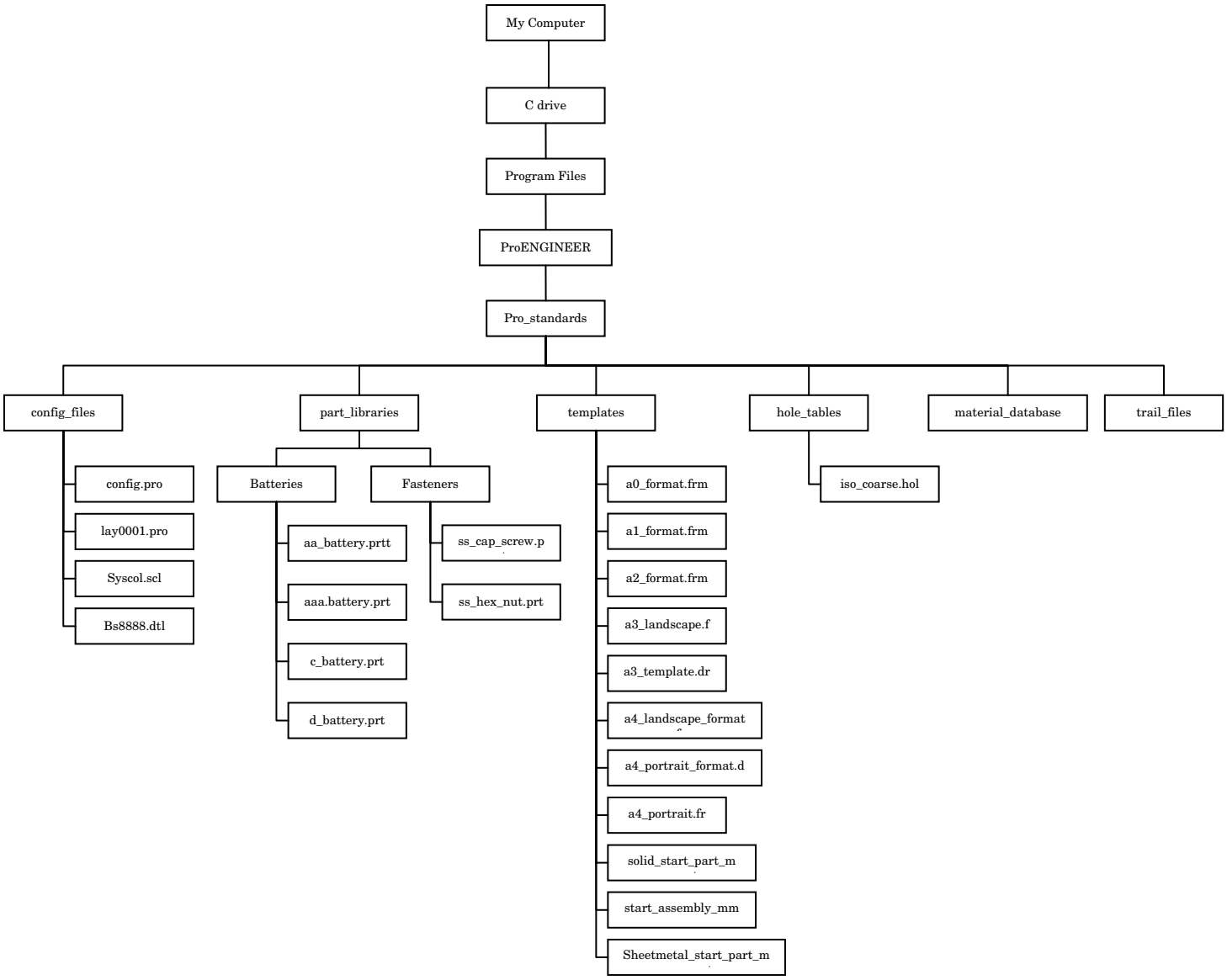
To use customized templates, such as drawing formats, create a single shared directory for all the templates. See the suggested directory structure in the next section. PTC provides example metric start parts and ISO drawing formats under the pro_standards/templates directory.

Developing a Directory Structure

The default Pro/ENGINEER directory is C:\Program Files\ProENGINEER Schools Edition. All the default template files, such as start parts and Imperial drawing templates, are in ProENGINEER Schools Edition\templates. Default configuration settings do not require a config.pro file or anything else. However, if you make any modifications to the config.pro file, create the relevant configuration file in C:\Program Files\ProENGINEER Schools Edition\text.

As an instructional aid, sample configuration files, start parts, drawing templates, and other files are under Program Files\ProENGINEER Schools Edition\pro_standards.

For all student to have access to the same data, you can place the pro_standards directory on a network drive. Then, make the relevant settings of the configuration options in the config.pro file point at this network directory.



Running Pro/DESKTOP and Pro/ENGINEER on the Same Computer

Note: If you are using Pro/DESKTOP 8.0 Service Pack 2 (datecode 2005400 build 1067), ignore this section.

When you run both Pro/DESKTOP and Pro/ENGINEER on the same computer, you must reregister Pro/DESKTOP *each time* you run Pro/ENGINEER. To reregister Pro/DESKTOP, open a DOS window and type the following commands:

```
C% cd \Program Files\PTC\ProDESKTOP 8.0\Program
C% prod.exe \regserver
```

Using Pro_Standards in Classrooms

To help implement Pro/ENGINEER Wildfire Schools Edition, PTC works with teachers around the world to provide preconfiguration for easier deployment within the classroom. A preconfiguration directory called `pro_standards` is in the `<loadpoint>\ProENGINEER Schools Edition\` directory.

Within the ProENGINEER Schools Edition directory are the following directories:

```
config_files
hole_sizes
material_database
part_libraries
templates
trail_files
```

Copy the pro_standards directory to a centrally accessible drive or directory. All students can then access common templates, part libraries, configuration options, and so forth. The purpose of each directory, its contents, and how to use it follows.

config_files

Pro/ENGINEER Wildfire uses different files to configure how Pro/ENGINEER looks and functions. Typically a Pro/ENGINEER System Administrator with knowledge of configuration options sets up these files. To help teachers implement Pro/ENGINEER Wildfire Schools Edition, a preconfigured set of configuration files comes with the software. You can modify the preconfigured files as you become more familiar with Pro/ENGINEER's operation.

config.pro

One of the most important configuration files in Pro/ENGINEER is the config.pro file. This file *must* be on each computer under <loadpoint>\ProENGINEER Schools Edition\text\. The teacher or an IT (Information Technology) Administrator should modify the config.pro within Pro/ENGINEER to suit educational requirements. Save the file in a secure location, and copy it to each computer running Pro/ENGINEER.

The config.pro file contains a list of directories that Pro/ENGINEER searches to find drawing formats, drafting standards, templates, and so forth. The teacher or IT Administrator must edit the relevant directory settings within the config.pro file. For example, the configuration option template_solidpart specifies which template file that Pro/ENGINEER uses for new parts. Provide the full path in the configuration options.

Note: If you have moved the pro_standards directory to a centrally accessible drive or directory, modify this setting to point Pro/ENGINEER at the new location.

To modify the config.pro file, follow these steps:

1. Start Pro/ENGINEER.
2. Click **Tools** > **Options**.
3. Choose each option and modify its value.
4. Click **Add/Change** after each change.
5. Click **Apply** or **OK**. The newly modified config.pro file is saved for future use

Note: You must copy the final config.pro file onto every computer running Pro/ENGINEER. Place it in the directory <loadpoint>\ProENGINEER Schools Edition\text\.

search_path.pro

The search_path.pro file specifies where Pro/ENGINEER searches for any parts, assembly files, and other files that are not in the working directory. In Word Pad or

Note Pad, you can edit the search_path.pro file. Use the full path for the configuration option. A search_path.pro file comes with the software. It contains the paths to sample parts, such as, pro_standards/part_libraries/fasteners.

The location of the search_path.pro file is specified in the config.pro file. You must modify the path in this file to suit your own directory structure.

bs8888. dtl

Pro/ENGINEER does not come with a predefined UK BS8888 drafting standard. The bs8888.dtl file for the Schools Edition configures Pro/ENGINEER's drafting functions to replicate the BS8888 drafting standard.

iso.dtl

A copy of the ISO drafting standard configuration file, iso.dtl, comes with Pro/ENGINEER Schools Edition. The installation of the Schools Edition copies it into the pro_standards directory. This location ensures that students are using the same iso.dtl file.

lay0001.pro

The lay0001.pro configuration file specifies on which layers to place specific entities such as datum planes.

syscol.scl

The syscol.scl configuration file defines the colors for use for different types of geometry, graphics, text, and so forth, in Pro/ENGINEER.

tree.cfg

The tree.cfg configuration file determines what Pro/ENGINEER displays or hides within the Model Tree. For Pro/ENGINEER to find these configuration files, you must specify their location in the config.pro file.

hole_sizes

With Pro/ENGINEER, you can insert standard holes, that is, M10 tapped hole with a countersink. Pro/ENGINEER comes with a set of ISO, UNC, and UNF tables of hole sizes. An ISO coarse table contains the most commonly used ISO hole sizes. You must set the config.pro option of hole_parameter_file_path with the ISO_coarse.hol file to point at your centrally located pro_standards directory. Otherwise, Pro/ENGINEER does not find the file.

material_database

With solid modeling in Pro/ENGINEER, you can represent parts and assemblies as real solid components with accurate Mass properties. You can assign a diverse range of material properties to each solid part. Pro/ENGINEER comes with a sample set of materials that are in the directory <loadpoint>\ProENGINEER Schools Edition\text\materials-library. You can also define additional materials and store them in this central directory.

You can provide student access a common material library. Copy the contents of this directory into the centrally accessible pro_standards\material_database directory.

To find the pro_standards\material_database directory, set the config.pro option of pro_material_dir to point at your centrally located pro_standards\material_database directory.

part_libraries

As part of best practice, use standard parts. For example, an assembly can contain a number of nuts and bolts. Rather than have students create their own models of a nut or bolt, you can have them use standard sets of nuts and bolts. Sample part libraries for nuts, screws, and commonly used batteries come with the software.

To find the part_libraries directory, you must add the full path names of each directory to the search_path.pro file. For classroom installations, place this directory on a central server.

templates

For use in classrooms, place the template directory on a central server. When creating a new part, assembly, or drawing, Pro/ENGINEER uses templates, or start parts. These templates have predefined units, drawing borders, and so forth. Pro/ENGINEER comes with a default set of Imperial start parts and drawing formats. A metric set of sample start parts and drawing formats is also available.

To include these template files in the Pro/ENGINEER user interface, you must set the relevant configuration options in the config.pro file with the path to the pro_standards\templates directory.

```
start_model_dir
template_solidpart
template_designasm
template_drawing
```

trail_files

Every time you starts a session of Pro/ENGINEER, a trail file is created. This trail file records all actions during that session. These trial files accumulate, and you must delete them. To make the deletion easier, follow these steps:

1. Create a trail_files directory within each user account.
2. Set the configuration option to direct Pro/ENGINEER to create the user-specific trail files in the appropriate user-specific directory.
3. Set a trail_dir configuration option in the config.pro file to point at the users own trail_files directory.

Managing Files and Your Work Environment

You can manage your files and work environment in a number of ways.

Using Pro/ENGINEER File Types

The most widely used file extensions in Pro/ENGINEER are as follows:

.prt—part files
.asm—assembly
.drw—drawing
.frm—drawing format

Naming Files

A name of a file, feature, drawing view, datum plane, sketch, or any file within Pro/ENGINEER *cannot* contain any of the following characters: & / . , ; : # % $ ( ). In addition, you cannot use a space in a file name, as in `top bracket.prt`. Use an underscore (_) instead as in `top_bracket.prt`.

Pro/ENGINEER does not differentiate between uppercase of lowercase characters. For example, when opening a new part, you enter the file name `Front_Wheel.prt`. Pro/ENGINEER sees and refers to this part as `FRONT_WHEEL.PRT`, and on disk this part is seen as `front_wheel.prt`.

Erasing Pro/ENGINEER Files

Pro/ENGINEER holds the part, assembly, or other file in memory after you save and close it. Erase files from memory when you have finished with them. To remove a file from memory, click **File > Erase** > **Not Displayed** and choose the file to erase. Otherwise, when you open a file, Pro/ENGINEER first searches for it in memory and could open an unwanted file, as the example below explains.

Suppose that you open `wheel.prt` from `C:/my documents/projects/car` and then close it. Next, you change directory to `C:/my documents/projects/truck` and select a file call `wheel.prt` from this directory. Pro/ENGINEER looks first in memory, sees a file called `wheel.prt`, and opens that file. As a result, the wheel.prt of the car would open and not the wheel.prt of the truck.

Working with Versioned Pro/ENGINEER Files

Every time you save a file in Pro/ENGINEER, a sequential version of the file is created. For example; if you open a newly created part called `antenna.prt` and save it, `antenna.prt.1` is created, and then `antenna.prt.2` and so on with each successive save. These versioned files can consume disk space. Purge files on a frequent basis on each directory with part files, assembly files, and drawing file.

To add the Purge utility to the Windows Explorer menu, follow these steps:

1. Open Windows Explorer.
2. Click **Tools** > **Folder Options** and click the **File Types** tab.
3. In the **File Types** list, click **Folder** and then click **Advanced**. The **Edit File Type** dialog box opens.
4. Click **New.**
5. In the **Action** box, type the name for the command, for example, `ProePurge`.
6. Browse to the bin directory in the Pro/ENGINEER `loadpoint` directory and click `purge.bat`.
7. In the **Application used to perform action** text box, at the end of the command type a space and then `*.*`. For example, …`bin\purge.bat *.*`

The **ProePurge** command now appears on the shortcut menu. Right-click a folder and choose **ProePurge** to remove all old version files from this folder.

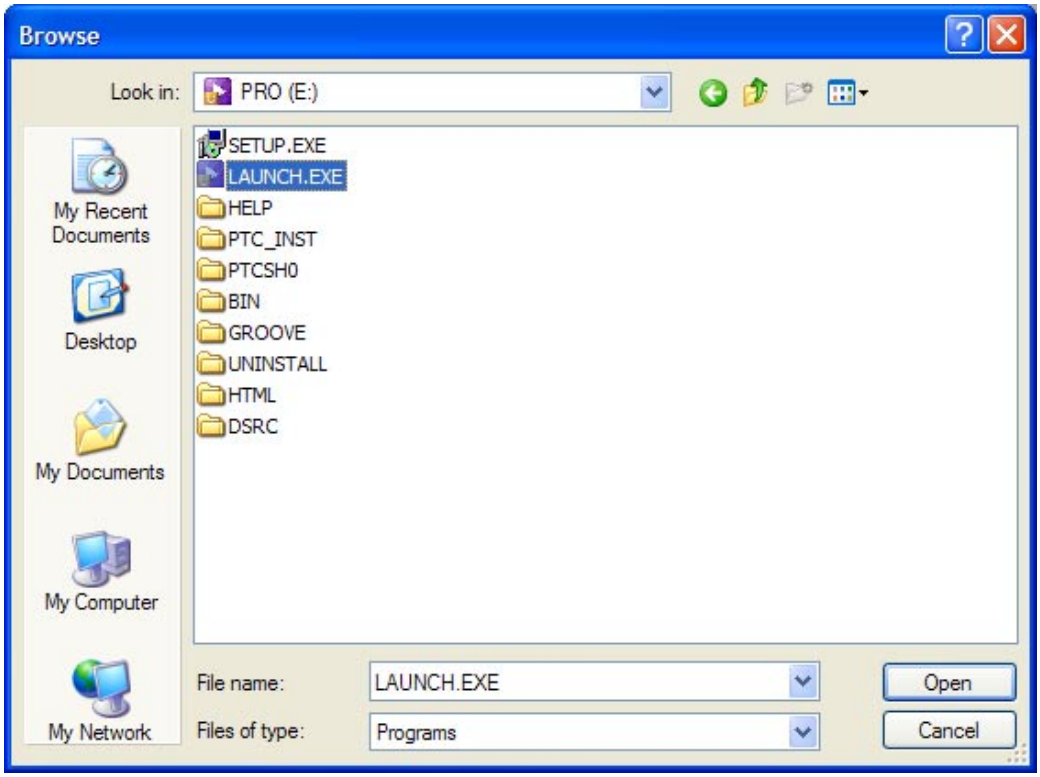
Troubleshooting

Common questions and answers can help you get up and running with your Pro/ENGINEER Schools Edition. For additional troubleshooting assistance, see the Troubleshooting FAQ for the PTC Education Programs on the **Schools Program** page.

Setup did not begin when I inserted the CD. What do I do?

If PTC.Setup does not begin automatically, follow these steps:

1. Click **Start > Run** on the Windows task bar. The **Run** dialog box opens.
2. Click **Browse.** The **Browse** dialog box opens.



3. For the **Look in** list box, click the down-arrow and select your CD-ROM drive.
4. Click **LAUNCH.EXE** in the **File name** box.
5. Click **Open** in the **Browse** dialog box. After the path appears in the **Run** dialog box, click **OK**. The **Schools Edition** starting screen appears.

How do I locate my computer's Host ID?

The Pro/ENGINEER Schools Edition license is locked to the Host ID, which is the network physical address of your computer. You can find out your Host ID during installation or from the **Start** menu.

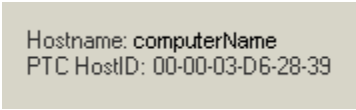
During Installation

The Host ID appears in the bottom-left corner of the first screen during Pro/ENGINEER installation.

1. Insert your Pro/ENGINEER Schools Edition CD-ROM in to the CD-ROM drive of the computer on which you want to install Pro/ENGINEER.

2. Start PTC.Setup as described in the section Installing PRO/ENGINEER Schools Edition.

3. On the first installation screen, in the lower-left corner, you will see a 12-digit number on the second line similar to that in the figure below.



   The 12-digit number with hyphens is your Host ID.

From the Start Menu

The Host ID appears when you enter the `ipconfig/all` command from a DOS window.

1. Click **Start > Run,** type `cmd`, and press ENTER.)

2. At the prompt, type `ipconfig/all` and press ENTER. The Windows IP Configuration information appears.

3. Locate the line with the words *Physical Address*. The 12-digit number with hyphens is your Host ID.

   Physical Address. . . . . : 00-11-22-33-44-55

What does the error in plpfhost mean?

A `plpfhost` error can occur for one of two reasons:

- A required Ethernet network adapter is not installed or configured properly. See your hardware vendor for installation and configuration information.
- Media Sense within Windows 2000 causes the PTC Host ID (Ethernet network adapter address) to not be detected. It disables the network card. Use the instructions in the Windows 2000 document Q239924 to disable media sensing.

Why does my Schools Edition not start?

If your computer was upgraded from Windows 95/98/Me to Windows 2000/XP, the variable `winbootdir` could have been preserved rather than deleted. Delete the `winbootdir` variable and your Pro/ENGINEER Schools Edition will start.

How do I disable Media Sense for TCP/IP in Windows 2000?

To disable Media Sense for TCP/IP in Widown 2000, see the article from Microsoft:

http://support.microsoft.com/default.aspx?scid=kb;EN-US;q239924

What does the message “Could not detect network adapter” mean?

See your hardware vendor for information on installing and configuring the network adapter if you get this message: `Could not detect network adapter`. An Ethernet network adapter is required to start the Pro/ENGINEER Schools Edition.

Why does Pro/ENGINEER not find the load point at startup?

If you get the message `Cannot Find <Loadpoint>\i486_nt\filename`, the `PATH` environment variable for the Pro/ENGINEER load point is either is not set or is incorrect. You may have a syntax or spelling error. To correct this, follow these steps:

1. Click **Start > Settings > Control Panel**. (For Windows XP, click **Start > Control Panel**). The Control Panel appears.
2. Double-click **System**. (XP users, if you do not see **System** in the **Control Panel**, click **Switch to Classic View** in the upper-right corner.)
3. Click the **Advanced** tab and click **Environment Variables**. The **Environment Variables** page appears.
4. Look under **System variables** for the variable PATH. Check to see if `PATH` contains `<loadpoint>\bin`. (All values are separated by a semicolon. Therefore, if there are other paths before and after Pro/ENGINEER, the Pro/ENGINEER path will have semicolons both before and it).

   Note: The load point is the directory into which Pro/ENGINEER was installed. If you accepted the default location, then your load point will be `C:\Program Files\ProENGINEER Schools Edition`. If you installed Pro/ENGINEER into a directory other than the default load point, you must reflect that change.
5. Between the semicolons, fix the PATH variable to the correct location of your load point.

Why does my computer select a wrong network connection?

If your computer automatically selects the wrong network connection, for example, WiFi instead of your preferred land-line connection, your computer is most likely using Media Sensing. Consider disabling Media Sensing. For information on how Media Sensing works and how to disable it, see the following link:

http://support.microsoft.com/default.aspx?scid=kb;EN-US;q239924.

Note: Ethernet cable 10b2 or coaxial cable RG-58 is not a connection-based media. Windows does not attempt to detect a connect state if this type of cabling is used. Also, NetBEUI and IPX do not recognize Media Sense.

Examples of some side effects exist for disabling the Media Sensing feature follow.

- Your computer has two network adapters ,and the Media Sensing feature is enabled. If one network adapter does not work, it is unbound, and associated routes are removed. As a result, all traffic goes through the other network adapter, assuming a default gateway is there.
- You are a roaming (portable) user. With the Media Sensing feature, you can connect to any network and have everything work, without restarting, releasing and renewing, and so on. After disabling Media Sensing and restarting your computer, Windows still shows the Network Disconnected icon on the task bar. In addition, the ipconfig command still shows a `Media State .....: Cable Disconnected` message when the cable is disconnected. However, the Network Interface is bound to TCP/IP. You can verify this by looking at the route table. Use the `route print` command to show the interface IP address. (You can also use the `ping` command to get the IP address assigned to the NIC).

Why is the wrong Host ID displayed on Windows 2000?

The wrong Host ID is displayed when a Windows 2000 computer is not physically connected to the network. The Ptcstatus utility shows the license server up with available licenses before Pro/ENGINEER is started. As a result, a Pro/ENGINEER licensing error occurs. Ptcstatus displays `COUNTED after attempt to run Pro/ENGINEER`.

`Ptclmgrd.log` shows the wrong `\Host ID`. When a machine is booted off the network, the registry shows the network cards in an incorrect order at the following path:

`HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows NT\CurrentVersion\NetworkCards`

When you connect to the network, the order of the cards is correct when the network card that goes with the generated FLEXnet license is in the first folder. When you are disconnected from the network, the network card folders must be put in the correct order.

This issue is related to the Media Sensing feature of Windows 2000. Disabling the Media Sensing resolves the issue. For more information on Media Sensing, see http://support.microsoft.com/support/kb/articles/Q239/9/24.ASP.