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Introduction

Pro/ENGINEER® Wildfire™ 2.0 Student Edition
Welcome to the Pro/ENGINEER Wildfire 2.0 Student Edition (Pro/E SE)! We hope you enjoy using the world’s most popular 3D MCAD Solution.
This installation guide has been created to help you install and configure your Student Edition (available in English).

Pro/ENGINEER Wildfire 2.0 Student Edition Suite
The Student Edition of this software contains the following core PTC functional modules:

- Pro/ENGINEER® -Foundation Advantage™ *
- Design Animation
- Interactive Surfacing
- Mechanism Dynamics
- Advanced Assembly
- Peer-to-Peer Collaboration – powered by Groove Networks
- Behavioral Modeling
- Pro/ENGINEER Structure and Thermal Simulation (Powered by MECHANICA®)
- NEW! 3-D Drawing Capability

*Pro/ENGINEER Wildfire Student Edition does not include the following modules which are part of the commercial Pro/ENGINEER-Foundation Advantage Package: Pro/INTERFACE™ for CADDS®, Pro/INTERFACE™ for CATIA®, or Pro/PDGS®

Languages
Choose from English, French, German, Italian, Japanese and Spanish (NOTE: Simulation modules are not available in Italian or Spanish). Refer to the installation instructions for installing the correct language.
You will find the following included in the package:

- Pro/ENGINEER Wildfire 2.0 Student Edition. CD 1 and 2.
- PTC On-Line Help (included on the Pro/E SE CDs)
- Pro/ENGINEER Wildfire 2.0 Tutorials (PDF files). Available in English only.

Usage Guidelines for PTC SOFTWARE
Pro/E SE is available to individuals for personal or educational use only and may not be used for labs or other general or institutional purposes.
This product may not be used for commercial, professional, or for-profit purposes. It is compatible with the Pro/ENGINEER Wildfire 2.0 Education Edition found in many colleges.
Features Specific to PTC

PTC has included some features in the Pro/E SE software which make it uniquely suited for personal or educational use. None of these restrictions will inhibit your work while using the software for personal or educational use. Differences from the commercial software include:

- **Pro/E SE** does not have access to floating options.
- Plotted deliverables generated using **Pro/E SE** display a plot banner along the four page borders indicating that the object being printed was created with the Student Edition. This banner is present on **Pro/E SE** deliverables, including drawings.
- **Pro/E SE** will not read or write trail files.
- **Pro/E SE** will use an on-line registration process that will require a user to access the PTC website, accept the Student Edition license agreement, and obtain a unique registration key before the Student Edition software will run. Details on this process are included in this document.
- The software is compatible with the **Pro/E SE** found in college labs. Objects created with the commercial software can NOT be retrieved with the Student Edition and visa versa.
Installation and Product Support

Installation and Troubleshooting Support
Journey Publishing provides basic installation and troubleshooting support free of charge to all end-users who have purchased the product from Journey Publishing or one of its partners. Product Support is available from Journey Publishing for a fee.

Contact Options
Please read and carefully follow the steps outlined in this guide for installing and troubleshooting your software. Should you encounter software problems, visit our Student Edition web site at www.ProEStudent.com for advanced technical information and troubleshooting for Pro/ENGINEER® Wildfire™ Student Edition.

End-users in the United States, Canada, Mexico, Japan and Australia may contact Journey Publishing at 800.874.9001 or 1.972-481-2100 or by email to techsupport@JourneyEd.com or contact your Reseller where purchased. Hours of operation are 8:00 AM - 6:00 PM. Central Time, Monday through Friday. Europe customers please contact Journey Europe at 49.471.9489.444 or email to service@JourneyEdEurope.com

When contacting Journey Publishing, please have the following information about your software and computer’s configuration ready:

- Manufacturer and model of the computer on which you are installing your software.
- Operating System that is installed on your computer.
- Manufacturer and model of the graphics card installed on your computer.
- Amount of physical memory (RAM) your computer has.
- SWAP or Paging File settings of your computer.
- The version of PTC software you are installing.
- Date code for the PTC software that is located on the lower portion of your CDROM (i.e., “M020”).
- A valid email address for communication purposes.

In the event that you received a defective product, please contact your corresponding Journey Publishing division or reseller as referenced above.
Installing Pro/ENGINEER Wildfire 2.0 Student Edition

Minimum System Requirements

NOTE: MECHANICA® below refers to Thermal and Motion Simulation Modules; Pro/E refers to the rest of the product.

Operating System: Microsoft Windows 2000/XP (Home and Professional – 32 bit)
Main Memory: 256MB (512 MB or higher recommended)
Available Disk Space: 900MB (Pro/E), 1.2 GB (Pro/E & Pro/M)
Swap Space: 256 MB (1024 MB or higher recommended)
CPU Speed: 233 MHz (750 MHz or higher recommended)
Browser: Microsoft Internet Explorer 6.0 SP1 or later
Network: Microsoft TCP/IP, Ethernet network adapter
Monitor: 1024 X 768 (or higher) resolution support with 24-bit or greater color
File Systems: NTFS
Mouse: Microsoft approved 3 button mouse
Installation (Pro/M): For Windows 2000 and Windows XP install use the i486_nt executable
Distribution Media: CD-ROM or DVD Drive
Languages: When installing the product you will have the option to select from the following languages – English, French, German, Japanese, Italian (Pro/E only) or Spanish (Pro/E only)

*The following patch is required on Windows 2000 if using Service Pack 1:
Microsoft Knowledge Base # Q271976:
http://support.microsoft.com/default.aspx?scid=kb;/LN;Q271976

Warning!

Make sure that you exit all other applications before you attempt to begin the installation process, including disabling your screen saver. Failure to do so may result in a poorly installed application.

Users must be logged in as Administrator or logged on to an account with administrator rights on the computer.

If your machine was recently upgraded from Windows 95/98/Me to Windows 2000/XP the environment variable "winboottdir" may be incorrectly set. Normally this variable only exists on Windows 95/98.Me. However, if Windows 95/98/Me is upgraded to Windows 2000/XP, the environment variable is preserved, Pro/ENGINEER picks up the wrong OS type; and the software then fails to start. To address this issue, the user must delete the "winboottdir" variable. More information on these topics is available at www.ProEStudent.com.

IMPORTANT— Before you begin installing any of the software, you must register and license your copy of Pro/E SE. Detailed instructions for this process can be found in the Registering and Obtaining License for Pro/ENGINEER Student Edition section.
Starting PTC.Setup

Begin setup by inserting the Pro/ENGINEER Wildfire Student Edition CD into your CD-ROM drive. The autorun feature of the CD should start the setup process. If setup does not begin automatically, launch it by following these steps:

1. Click Start in the taskbar on your Windows desktop screen to access the Start menu, then click Run.
2. In the Run dialog box that pops up, select Browse.
3. In the Browse dialog box, click the down arrow on the right of the Look in: field to display options.
4. Select your CD drive, then click Setup.exe.

Performing steps 1-4 above causes the path d:\setup.exe (where d designates your CD-Rom-drive) to display in the Run dialog box (below).

5. Once the path displays, select OK to start the PTC.Setup.
Registering & Obtaining License for Pro/ENGINEER Wildfire Student Edition

Preliminary Steps

1. Start PTC.Setup as outlined in the Starting PTC.Setup section.
2. On the first Setup screen, double click on the language flag to install in the language you desire.
The basic Product information page will display. Double click on INSTALL NOW to begin installation.
Registering and Obtaining license … (continued)

The PTC Setup screen will display.

To begin installing Pro/E SE, follow the steps listed below. During installation you can click **Cancel** at any time to terminate the installation process.

You can select the portion of the program to install by double clicking on the title located on the right side of the screen.

Double Click on **“Pro/ENGINEER”** to begin installing Pro/ENGINEER.
The **License Configuration** dialog box displays.

3. Click on **Register for a PTC Online account**.
   (previous registered owners of Wildfire do not need to set up an account. Skip the online registration process and go to Create a new license step.)
Registering and Obtaining license … (continued)

Online Account Registration

After selecting the Register for a PTC Online account option, your default web browser opens and takes you to the PTC Online Account Registration window.

4. Complete the account registration form and then submit your registration.
   (This account is your account with PTC and is not related to JourneyEd)
Registration Confirmation

The **Confirmation** window displays after you submit your registration.

Be sure to review the information on this page before closing your browser window.

An email confirmation of your registration will be sent to the email address you provided previously. This account confirmation is sent from PTC to the email address you provided in the account setup.
Creating a New License File
You now need to create a new license file. Follow the steps below to create it.

5. Display the PTC.Setup **License Configuration** dialog box again (see steps 1-3).

6. In the **License Configuration** dialog box, click on **Create a new license file**.

Your default web browser opens again and you are brought to a PTC website window that contains the **Enter Network Password** dialog box.

7. In the **Enter Network Password** dialog box, type the User Name and Password you established in your Online Account Registration (step 4), and click **OK**.
Pro/ENGINEER Student Edition

To continue the setup process you must register your copy of the Pro/ENGINEER Student Edition by accessing the URL below.

You will need this information to complete the on-line registration process:
- Your Host ID is
- Your Program ID is journeystudent

http://www.ptc.com/appserv/myprogram/index.jsp?host=

PTC - Online Services

User name: 
Password: 

[ ] Remember my password

OK  Cancel
Completing the Licensing Agreement

After clicking OK, the browser redirects you to the Pro/ENGINEER Student Edition Licensing web page. (This process may take a few moments.)

8. Mark the check-boxes to agree to both license agreements on the page. The License Agreement web page is actually two pages in size, but the graphic below shows only the first page of the screen displayed at this point in the registration process. Please note that the first box to check is located mid-page, and that the second box is at the bottom of the second page. To complete the agreement, scroll down the entire web page to find the second box to check.

9. When finished marking both boxes, click Continue.
Registering and Obtaining license … (continued)

After clicking Continue on the Pro/ENGINEER Student Edition Licensing web page, you are brought to the second part of the Pro/ENGINEER Student Edition Licensing page.

When you are filling out information on this page, note that:

- All Fields marked with a red asterisk (*) are required.
- Several fields are pre-populated with information you provided during online account registration: First Name, Last Name, Email Address, Host ID*.

10. Complete the form, then click the Submit Request button.

*The Host ID field will populate with the Host ID of the machine that you are currently on. If you will be using Pro/ENGINEER on a machine other than the one you are using to register the product, you will need to replace the Host ID in that field with the correct Host ID of the machine that will be running Pro/ENGINEER. For more information on how to obtain your Host ID, refer to the Basic Troubleshooting section located at the end of the guide.
Registering and Obtaining license ... (continued)

Registration Confirmation
After submitting your request form, you are brought to the final confirmation page thanking you for your request. Be sure to review the information on this page before closing your web browser.

Receiving Your License Pack by Email
At this point, you must wait for the email from PTC which contains your License Pack. It should arrive at the email address you provided within two hours.

Saving Your License Pack
It will be necessary for you to save your license pack to disk. When you receive your License Pack, it contains a file attachment under the name (or a similar name) of \textit{sw\_license\_email\_0000.txt}. Save this file to a location on your hard drive that you will remember and are able to access in the future (i.e., C:\TEMP). Due to the fact that email providers and their software function differently, we cannot provide instructions for this \textit{Save} process. Do not edit the name or contents of this file, as doing so can cause the license to become invalid and prevent Pro/E SE from starting.
Installing Pro/ENGINEER (continued)

After leaving the first Setup window by clicking Next, the License Configuration dialog box appears.

**IMPORTANT** To proceed beyond this point, you must have the file that was emailed to you after registration. If you have not yet registered your Pro/ENGINEER Student Edition product, click the link in the dialog box below to start the registration process. You must complete the registration process in order to receive your license file.

**Importing Your License File**

Once you have received your license file and saved it on your computer in a text file you must import it. Importing a file is a 2-step process. The first step is:

1. Click on the folder icon to the right of the License File Path field.
The second step to importing your license file is:

2. Navigate to the directory that contains your Pro/ENGINEER Student Edition License file (as shown in the Select File dialog box below), and click OK. The directory (Folders) path will vary depending on where you saved your license file and may not appear exactly as in the example below.

IN CASE OF REINSTALLATION: JOURNEY STRONGLY RECOMMENDS THAT YOU WRITE DOWN IN A SAFE PLACE, THE FILE LOCATION PATH THAT YOU SAVED THE LICENSE FILE TO. JOURNEY CANNOT ASSIST YOU WITH WHERE YOU SAVED IT ON YOUR COMPUTER SHOULD YOU NOT REMEMBER THE LOCATION.
Define Installation Components Options

After clicking OK in the Select File dialog box, it disappears from the screen to display the Define Installation Components window. There is a momentary pause at this point as the program searches to detect your system’s settings.

When auto-detection at the Define Installation Components window has been completed, you are presented with the options shown in the table below.

1. If you want to follow the default installation, in English, click Next.
   You have the option to not install some of the portions of the program by right clicking on that portion and selecting “do not install”. After you have set your options, click the Next button at the bottom of the window to continue.
You will then be able to select your Shortcut preferences. Once completed click the "NEXT" button.

**Destination Folder**
- This is the directory in which Pro/ENGINEER is installed. The default installation directory is `C:\Program Files\ProENGINEER Student Edition`. If you wish to change the default location, you may simply type in another directory path, or select the browse button to the right of the path to select another location. For new installations, this cannot be an existing PTC directory. If the directory exists, PTC.Setup assumes you trying to update or reconfigure an existing copy of Pro/ENGINEER Student Edition. PTC strongly recommends that a NEW installation be performed.

**Platforms**
- The only platform available in Pro/E SE is i486_nt. This option is selected by default.
| **Languages** | – Check any additional languages that you would like installed. If you wish to install additional language packs, more hard drive space will be required. |
| **Components** | – Select or clear the components that you want to install by selecting the check box next to the appropriate component. The available components are as followed |
| **Pro/ENGINEER (default)** | – Keep this component to install the files necessary to run the software for Pro/E SE. |
| **Pro/ENGINEER Help Files (default)** | – Keep this component to install the Pro/E SE Help Files in the product installation directory. To view context-sensitive Help, you must install these HTML files. |
| **Subcomponents** | – There are no subcomponents in the Pro/E SE. |
The **Optional Configuration Steps** dialog box now displays. Click **INSTALL** to begin the default settings installation

**OR** check the OLE settings if you want to customize your installation and follow the steps below.

---

**OLE Settings**

- By keeping the default OLE settings box checked on the Optional Configuration Steps window you can configure Pro/ENGINEER Student Edition as an OLE (Object Linking and Embedding) Server. This allows the user of the client application to import Pro/ENGINEER files (parts, assemblies, and drawings) into Microsoft documents (Word, Excel, and Power Point). This does not configure Pro/ENGINEER as an OLE container. That is, you cannot import Microsoft files into Pro/ENGINEER.
2. After selecting the OLE settings, click **NEXT** to proceed to the next screen

**PTC OLE Server Options/Configuration**

With **OLE Settings** checked, then the **PTC OLE Server** dialog box and options appears (see below). You may fine-tune the options to your own taste; however, all of the default values will work.

![PTC OLE Server Configuration](image)

**Pro/ENGINEER OLE Configuration**

- **Startup command** – Specify the command used to start Pro/ENGINEER (the default will be accurate).
- **Startup directory** – Specify the default startup directory for Pro/ENGINEER. Typically, this is the Pro/ENGINEER working directory.
- **Language** – Keep Language settings as default unless you plan to use another language.

**PTC OLE Server Configuration**

- **Start search directory** – Allows you to specify the startup directory of the PTC OLE Server. The Microsoft application (PTC OLE Container) searches for PTC objects in this directory.
- **Language** – Keep Language settings as default unless you plan to use another language.
Completing Pro/ENGINEER Configuration

Congratulations! Once you reach this point you have finished the OLE custom configuration on the Pro/ENGINEER installation. The **Installation Progress** window (below) now greets you.

3. Click **Install** to start the installation process, or choose one of the following options:
   - Click **Back** to review information
   - Click **Install** to begin installation
   - Click **Exit** to cancel PTC.Setup

   During installation, you can click **Cancel** at any time to terminate the installation process.
Installing Pro/ENGINEER (continued)

Changing CD-ROMS During Installation

PTC.Setup copies files and sets permissions. Around 60% of the way through installation, a dialog box like the one below pops up, prompting you to insert the second CD-ROM. When this happens,

4. Either replace the current Pro/ENGINEER CD-ROM with the second disk, or click **Browse** and provide the path to the drive containing the second disk.

![Insert a new CD dialog box](image)

When installation reaches around 99%, another window like the one below pops up, prompting you to re-insert the first CD-ROM. When this happens,

5. Either replace the current Pro/ENGINEER CD-ROM with the first disk, or click **Browse** and provide the path to the drive containing the first disk.

![Insert a new CD dialog box](image)
Once you have performed both of these actions the installation process will continue until it is complete.

6. Click **Next** to install the next selected component or to exit PTC.Setup
Installing Structural & Thermal Simulation (Pro/MECHANICA)

Install Pro/ENGINEER before Pro/MECHANICA

IMPORTANT – Since the Student Edition of Pro/MECHANICA only runs in integrated mode, Pro/ENGINEER must be installed before you can install Pro/MECHANICA. Failure to do so will prevent Pro/MECHANICA from functioning.

During installation, you can click Cancel at any time to terminate the installation process.

1. Select Structural and Thermal Simulation by clicking on it on the display screen.
Installing Pro/MECHANICA (continued)

Defining Installation Components

You will be brought to the **Define Installation Components** screen, you will notice a pause as the program searches to detect your system’s settings. This pause occurs during both Pro/ENGINEER and Pro/MECHANICA installations. Once auto-detection is complete, you are presented with several options (see next page).
Installing Pro/MECHANICA (continued)

Available Options

The options available on the Define Installation Components screen are as follows:

**Installation Directory** – **IMPORTANT** This is the Directory in which Pro/MECHANICA is to be installed. Since Pro/MECHANICA Student Edition only runs in integrated mode, the installation directory must contain the path to a valid Pro/ENGINEER Student Edition loadpoint.

- If you continue this installation immediately after Pro/ENGINEER installation, the installer populates this field with the same information you provided when installing Pro/ENGINEER.
- If you installed Pro/ENGINEER previously and are returning to install Pro/MECHANICA, the default path for the Installation Directory will be C:\Program Files\ProENGINEER Student Edition. This is the default path for Pro/ENGINEER installations, but if you choose to install Pro/ENGINEER to a location other than the default path, you must to manually browse to that folder by clicking the folder icon located to the right of the Installation Directory field.

*If you fail to do this properly and click Next, you will receive the error shown in the picture below. Should this occur, correct the Installation Directory to point to the valid Pro/ENGINEER loadpoint.*

**Platforms** – The only platform available in Pro/MECHANICA Student Edition is i486_nt. This option is selected by default.

**Languages** – Check any additional languages that you would like installed. If you wish to install additional language packs, more Hard Drive space will be required.

**Components** – The only component that is available in the Student Edition is Pro/MECHANICA (default): Keep this component to install the files necessary to run the software for Pro/MECHANICA

**Subcomponents** - There are no subcomponents in the Student Edition.

2. After all configuration options have been set on the Define Installation Components screen, click the Next button to continue.
Installation Startup
When you have finished configuring the Pro/MECHANICA installation, you are greeted with the Installation Progress window.

The display will show the separate modules as they install and the completion percentage.

During installation, you can click Cancel at any time to terminate the installation process.
3. After 100% completion click **Next** to return to the Wildfire Display setup screen.

**NOTE:** Pro/Mechanica does not create any shortcuts even if properly installed. For more information, see the section *Running Pro/Mechanica Student Edition for the First Time.*
Installing Collaboration Tools

Preliminary Steps

To select Collaboration Tools for installation click on it from the Wildfire setup Display screen.

You are taken directly to the Define Installation Components screen.
Defining Installation Components

Define Installation Components screen will display. You will notice a pause as the program searches to detect your system’s settings. This pause occurs during both Pro/ENGINEER and Pro/MECHANICA installations. Once auto-detection is complete, you are presented with several options (see next page).
Available Options

The options available on the Define Installation Components screen are as follows:

**Destination Folder** – This is the directory where Collaboration Tools are installed. The default installation directory is `C:\Program Files\PTC Collaboration Tools`. If you wish to change the default location, you may simply type in another directory path, or select the browse button to the right of the path to select another location. For new installations, this cannot be an existing PTC directory. If the directory already exists, PTC.Setup assumes you trying to update or reconfigure an existing copy of Pro/ENGINEER Student Edition.

**Platforms** – The only platform available for the Student Edition of Collaboration Tools is i486_nt. This option is selected by default.

**Languages** – Check any additional languages that you would like installed. If you wish to install additional language packs, more hard drive space will be required.

**Components** – The only component that is available in the Student Edition is *Collaboration Tools* (default). Keep this component to install the files necessary to run Collaboration Tools.

**Subcomponents** - There are no subcomponents in the Student Edition.

4. After you have set all configuration options this screen, click the Next button to continue.
Completing the Collaboration Tools Installation

Select your shortcut options if different than default settings

1. Click **Install** to start the installation process, or select one of the following options:
   - Click **Back** to review information
   - Click **Install** to begin installation
   - Click **Exit** to cancel PTC.Setup
The installation process will begin and show the installation progress.

During installation of Collaboration Tools, you are notified of the *Groove* software installation process. Installation instructions appear in the next section, *Installing Groove*. 
Installing Groove

During the installation process of Conference Center Server (Collaboration Tools), a Groove Wizard appears.

*NOTE*: Journey Publishing, Inc. recommends that you use the version of Groove software supplied with the PTC software. Journey does not support the update of the Groove software outside the PTC.Setup installation utility.

1. Click **Next** to continue the installation of Groove software.
Installing Groove (continued)

A **Choose Destination Location** dialog box displays, and allows you to select a folder where files are to be installed.

2. Click **Next** to create the default destination folder, or click **Browse** to select a different destination folder.
The **Choose User Data Location** dialog box now appears, giving you options for folder creation.

3. Click **Next** to create the user data folder, or click **Browse** to select a different folder.
The next screen contains a dialog box allowing you to **Choose System Data Location**.

4. Click **Next** to create the system data folder, or click **Browse** to select a different folder.
Installing Groove (continued)

After making your data location selection, an **Installation Options** window opens, and offers you two selections.

5. Choose one or more of the following installation options:
   - **Install Groove for All Users** – Allows all users to run the Grove software
   - **Add Groove to Desktop** – Creates a shortcut for Grove software on your desktop*

   *We do not recommend that you add the Groove shortcut to your desktop, as you will not be starting Conference Center Server this way. For more information see the section titled Running Conference Center Server for the First Time.

6. After selecting your option(s), click **Next**.
7. Specify the Program folders, then click Next.
The **Check Setup Information** window opens with the installation information for the Groove software.

8. Click **Back** to go to the previous windows and change the installation settings, or click **Next** to continue.
Continuing on with the installation, the **Setup Status** window now appears. When installation reaches 100% the following window will appear. Allow the program to complete its task.

Groove needs to run an optimization procedure; this may take several minutes.

When the following Information dialog box appears click **Finish**
Installing Collaboration Tools (continued)

You will be returned to the PTC.Setup window to complete the remaining Collaboration tools install process.

Upon Installation Completion click the NEXT button to return to the ProE Setup display or click Exit if you have installed all three modules of the software.

**NOTE:** Journey Publishing, Inc. recommends that you use the version of Groove software supplied with the PTC software. Journey does not support the update of the Groove software outside the PTC.Setup installation utility.
Running Pro/MECHANICA for the First Time

1. Launch Pro/ENGINEER by clicking on the **Pro/ENGINEER Student Edition** shortcut located either on your Desktop or your Start menu.

2. Once Pro/ENGINEER has started, create a new part or assembly.

3. Once the part or assembly has been created, go to the **Applications** pull-down menu and select **Mechanica**, as shown here.

   After selecting **Mechanica**, a **Units Info** dialog box will appear.

4. Click **Continue** to proceed.
After a few moments the **Menu Manager** appears in the upper right-hand corner with **MECHANICA** as the title. The **FEM mode** is unchecked and **Indep Mec** is greyed out.

5. Select the **Structure** option.

The icons on the right hand side of the screen change to MECHANICA icons and the **Menu Manager** looks like the one below. **Congratulations! You are now in Pro/MECHANICA.**
Using Conference Center Server (Collaboration Tools) for the First Time

Conference Center can be started from its shortcuts, but this method is not recommended. Journey recommends that you start Conference Center by using the following technique.

1. Launch Pro/ENGINEER by clicking on the Pro/ENGINEER Student Edition shortcut located either on your Desktop or your Start menu.

2. Once Pro/ENGINEER has started, go to the Applications menu and select Conference as shown below.
   
   ![Conference Center Launch Window](image)

A small window appears, informing you that Conference Center … is launching.
The next window to open is the **PTC Conference Center** window.

![PTC Conference Center window](image)

After the **PTC Conference Center** window displays, an error message similar to the one below pops up.

3. Click the **Start** button to start the Server.

![Conference Server Host error message](image)

After a brief moment, the Service Status on the Conference center window will change to **Connected**.
4. Click the **New Account** button and enter the information requested to create a new account.

5. Once you have completed entering your information, click **Apply**.

![Create New Account](image1)

After applying your information, a confirmation appears, letting you know that your new account has been created. (This process can take some time to complete).

6. Click on **OK** to complete this procedure.

![Create New Account](image2)
After confirming that your user account has been created, the **PTC Conference Center** dialog box displays on your screen.

![PTC Conference Center dialog box](image)

**Congratulations! You are now ready to use PTC Conference Center.**
Basic Troubleshooting

The following common Q and A (Questions and Answers) have been provided to help you get up and running with your new ProE SE software. For additional troubleshooting assistance, please go to www.ProEStudent.com.

Question
How do I locate my computer’s Host ID?

Answer 1
Insert your Pro/ENGINEER Student Edition CD-ROM in to the CD-ROM drive of the computer you wish to install Pro/ENGINEER on.

Start PTC.Setup as outlined in the Starting PTC.Setup section.

On the first installation screen look in the lower left-hand corner of the screen, you will see something similar to this:

![Hostname: computerName
PTC HostID: 00-00-03-D6-28-39](image)

The 12-digit number on the 2nd line is your Host ID.

Answer 2
Open a windows command prompt (Start > Run > type: CMD > press [enter])

At the C: \ > prompt type: ipconfig /all and press [enter]

This will return the Windows IP Configuration information

Locate the line that says:

Physical Address. . . . . . . . . : 00-0A-03-D6-28-39

This is your Host ID

Question
What does the error “error in plpfhost” refer to?

Answer
This error may occur for one of two reasons.

1. An Ethernet network adapter is not installed or configured properly. An Ethernet network adapter is required to start The Pro/ENGINEER Student Edition. Refer to your hardware vendor for information on installing and configuring the network adapter.

2. A new feature implemented within Windows 2000, Media Sense, causes the PTC Host ID (Ethernet network adapter address) to not be detected due to the network card being disabled.

Use the following documents to disable media sensing:

Windows 2000 Q239924 –

How to Disable Media Sense for TCP/IP in Windows 2000

http://support.microsoft.com/default.aspx?scid=kb;EN-US;q239924
Basic Troubleshooting (continued)

Question
What does the error “could not detect network adapter” refer to?

Answer
An Ethernet network adapter is required to start The Pro/ENGINEER Student Edition. Refer to your hardware vendor for information on installing and configuring the network adapter.

Question
When attempting to start Pro/ENGINEER, you see an MS-DOS window that contains the message:
Cannot Find <Loadpoint>\i486_nt\filename

Answer
This is caused by the PATH environment variable either not set or being set incorrectly due to syntax or spelling error. You can correct this by doing the following:

Go to: Start > Setting > Control Panel (if you are using Windows XP you may only have to go to Start > Control Panel)
Double Click on System (XP users, if you do not see System in the Control Panel click in the upper right-hand corner where it says “Switch to Classic View”
Go to the Advanced Tab
Click on Environment Variables
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/ENGINEERS the Pro/ENGINEER path will have semicolons both before and it)

NOTE: The loadpoint will be equivalent to the directory Pro/ENGINEER was installed in. If you accepted the default location then your loadpoint will be “C:\Program Files\ProENGINEER Student Edition”, but if you installed Pro/ENGINEER into a directory other than the default you loadpoint will need to reflect that change.