INSTALLATION GUIDE

Pro/ENGINEER® Wildfire™ Tryout Edition 2.0

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Thank you to Journey Ed. for providing the Installation Guide. If you would like to purchase a full-functioning version of Pro/ENGINEER Wildfire 2.0 Student Edition, please go to **www.ProEStudent.com** or call 1.800.874.9001

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1. Introduction

1.1 Pro/ENGINEER® Wildfire™ 2.0 Tryout Edition

Welcome to the **Pro/ENGINEER Wildfire 2.0 Tryout Edition** (**Pro/E TE**)! 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 Tryout Edition (available in English).

1.2 Pro/ENGINEER Wildfire 2.0 Tryout Edition Suite

The Tryout 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 Tryout 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®*

1.3 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 Tryout Edition. (Single CD)
- PTC On-Line Help is not included on the CD, but is available as an optional install from the setup launch page
- Pro/ENGINEER Wildfire 2.0 Tutorials are available from the landing page when you start Pro/E.

1.4 Usage Guidelines for PTC SOFTWARE

Pro/E TE is available to any individual that wants to try the new release of Pro/ENGINEER. Over the next 60 days, users will be able to experience the latest release of Pro/ENGINEER Wildfire 2.0 and it new enhancements and capabilities.

This product may not be used for commercial, professional, or for-profit purposes. It is not compatible with any other special edition of Pro/ENGINEER.

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1.5 Features Specific to PTC

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

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

2. Installation and Product Support

2.1 Installation and Troubleshooting Support

PTC does not offer any support for this special edition. Users are pointed to an online FAQ listing many of the common question users may have - **http://www.ptc.com/products/proe/studentedFAQ1.htm**

3. Installing Pro/ENGINEER Wildfire 2.0 Tryout Edition

3.1 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:	500 MHz (1 GB 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 "winbootdir" may be incorrectly set. Normally this variable only exists on Windows 95/98.Me. However, if Windows

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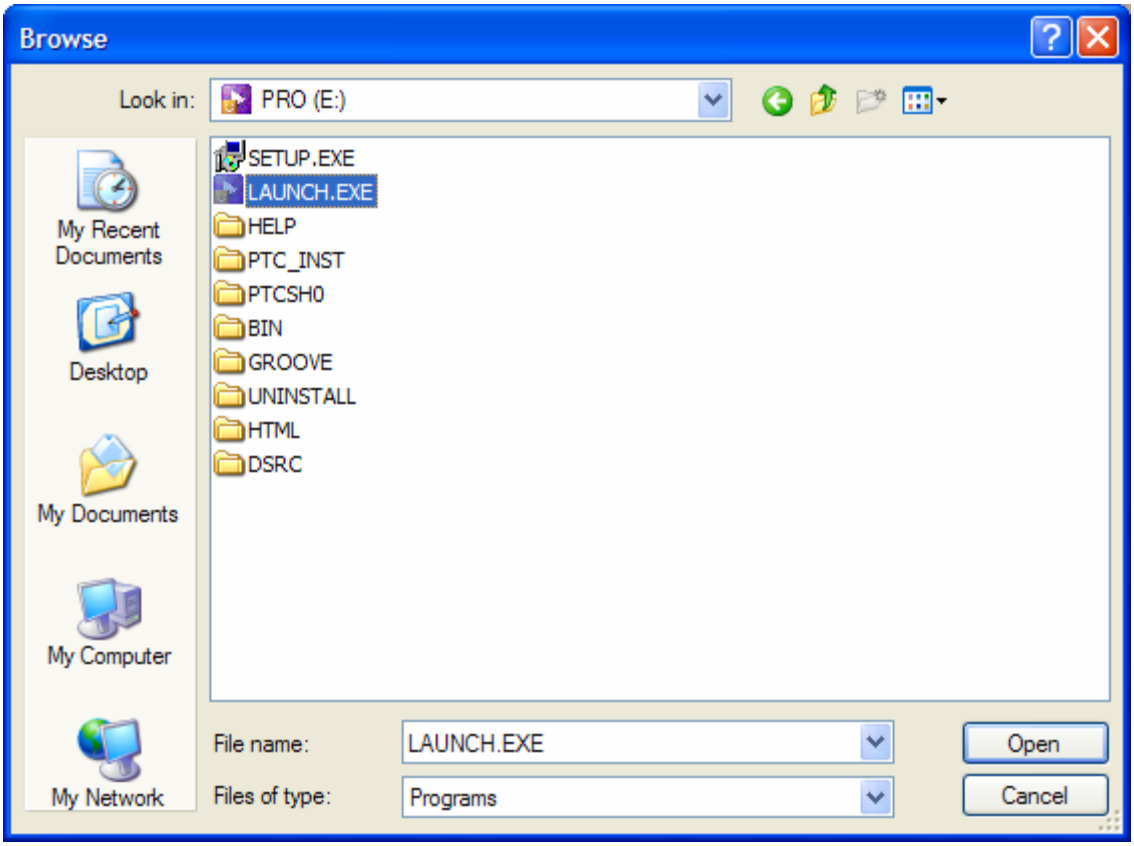
95/98/Me is upgraded to Windows 2000/XP, the environment variable is preserved, Pro/ENGINEER picks up the wrong OS type; and the software then fails to start. To address this issue, the user must delete the "winbootdir" variable.

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

3.2 Starting the PTC Launch Page

Begin setup by inserting the **Pro/ENGINEER Wildfire Tryout 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 **Launch.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.

5. Once the path displays, select **OK** to start the PTC Launch Page.

3.3 Preliminary Steps

Start the PTC launch page as outlined in the ***Starting the PTC Launch Page*** section.

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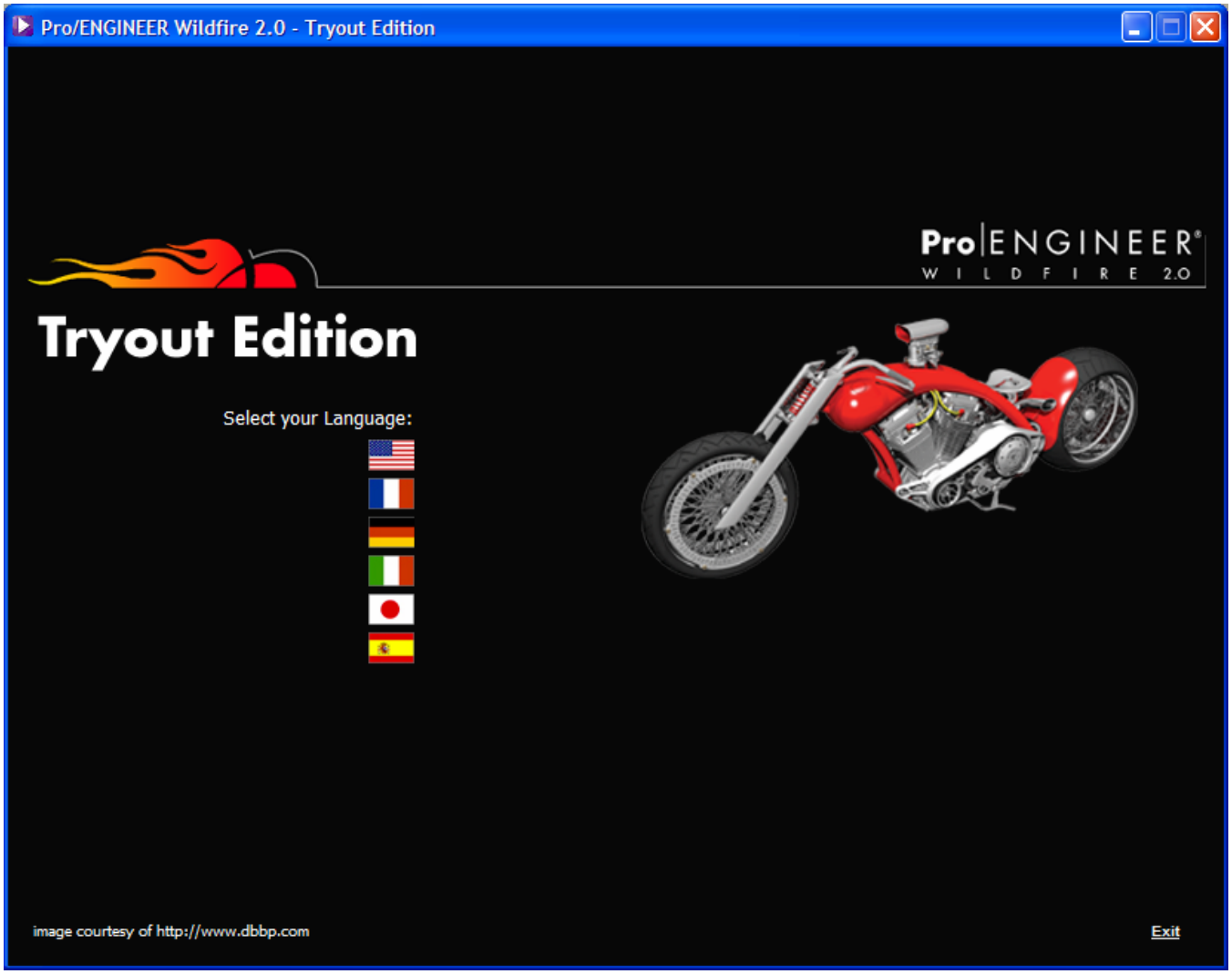
1. On the first **PTC Launch Page** screen, click on the language flag to install in the language you desire.



The basic Product information page will display. Click on **INSTALL NOW** that will initiate PTC.Setup that will begin the instillation steps for the **Pro/ENGINEER Wildfire Tryout Edition**.

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To begin installing Pro/E TE, follow the steps listed below.

Note: 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.

3.4 Installing Pro/ENGINEER Wildfire 2.0

1. Double Click on **"Pro/ENGINEER"** to begin installing Pro/ENGINEER.

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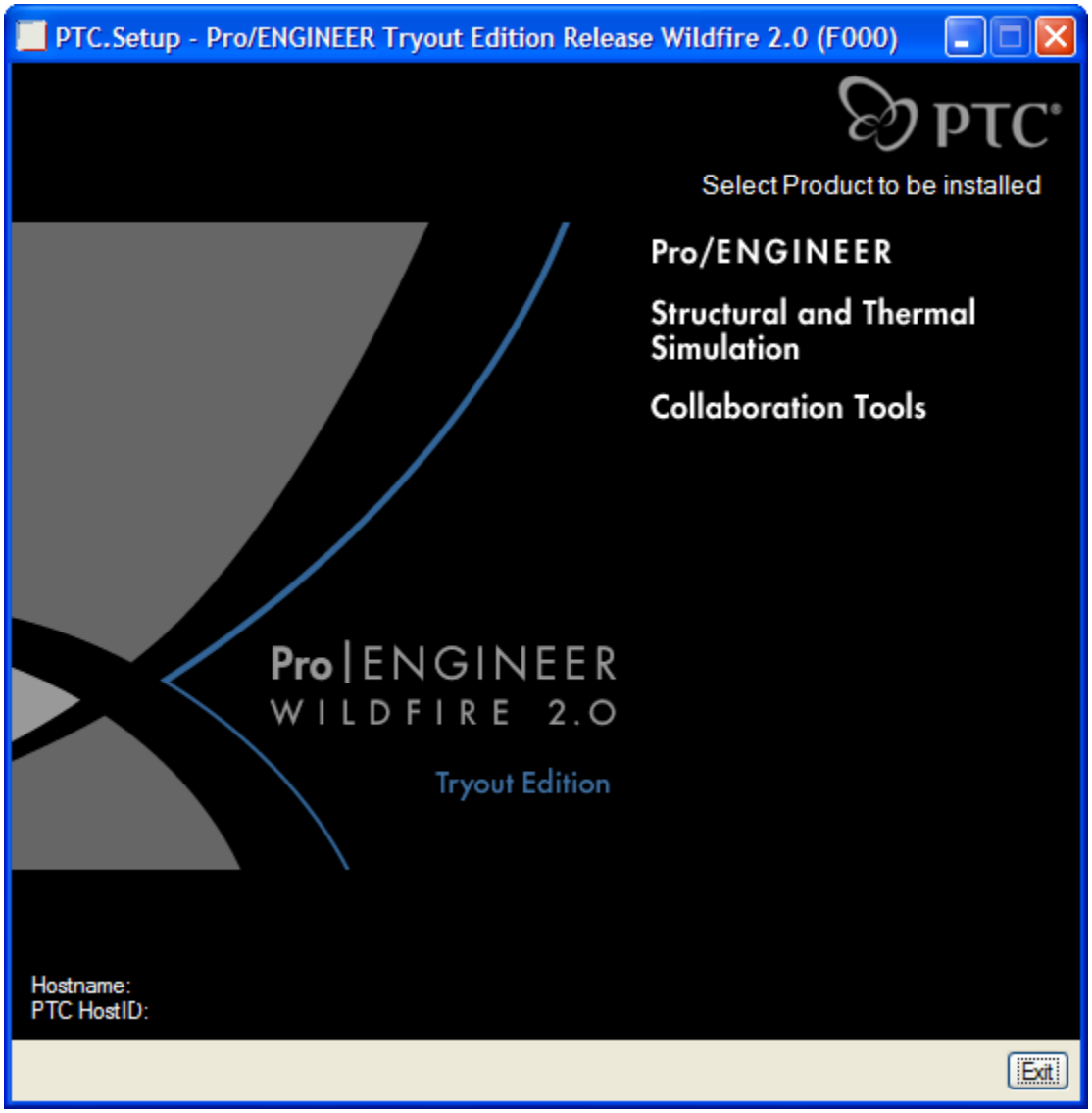

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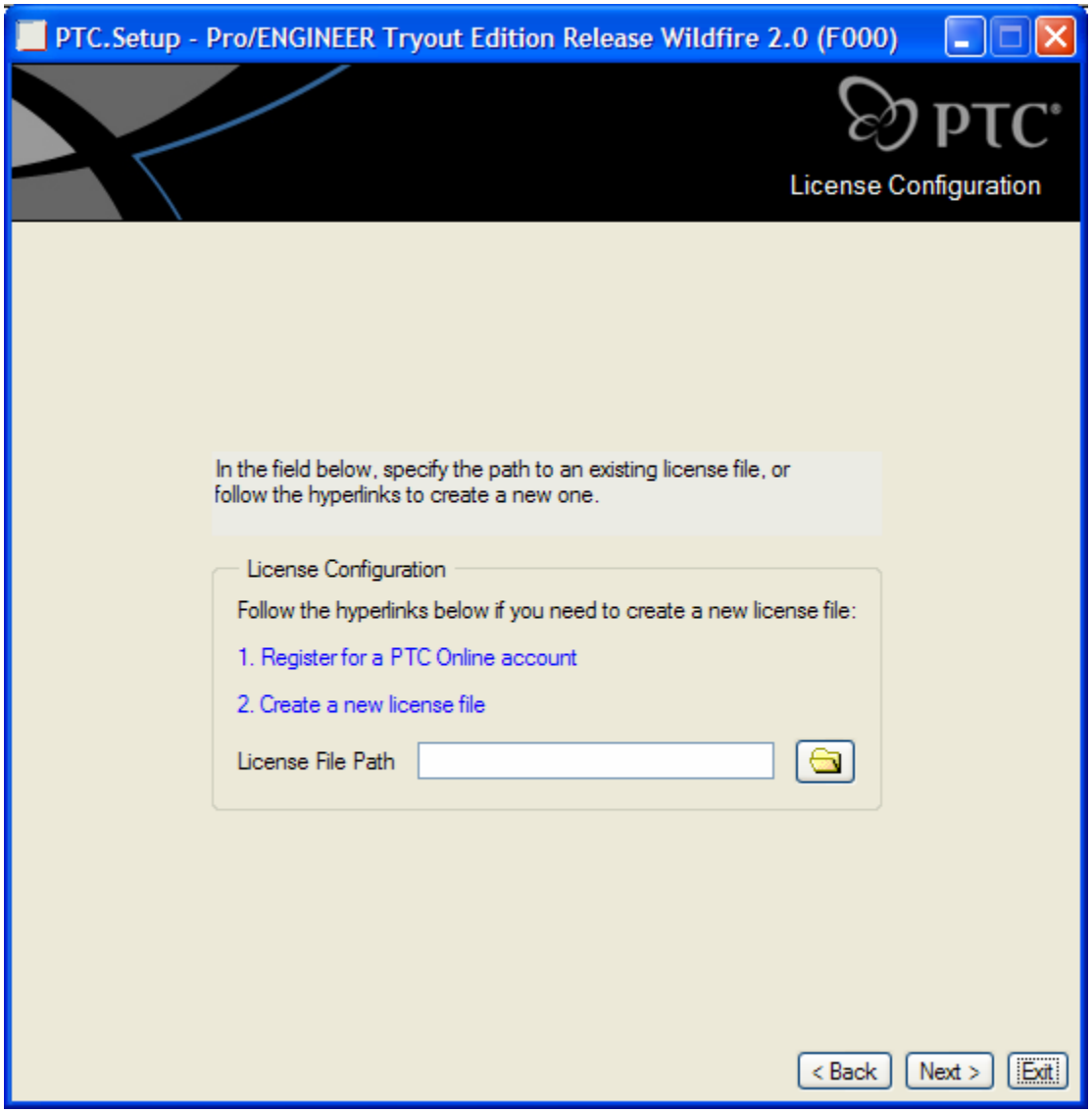
2. The License Configuration dialog box displays. Click on **Register for a PTC Online account**.



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. Complete the account registration form and then submit your registration.

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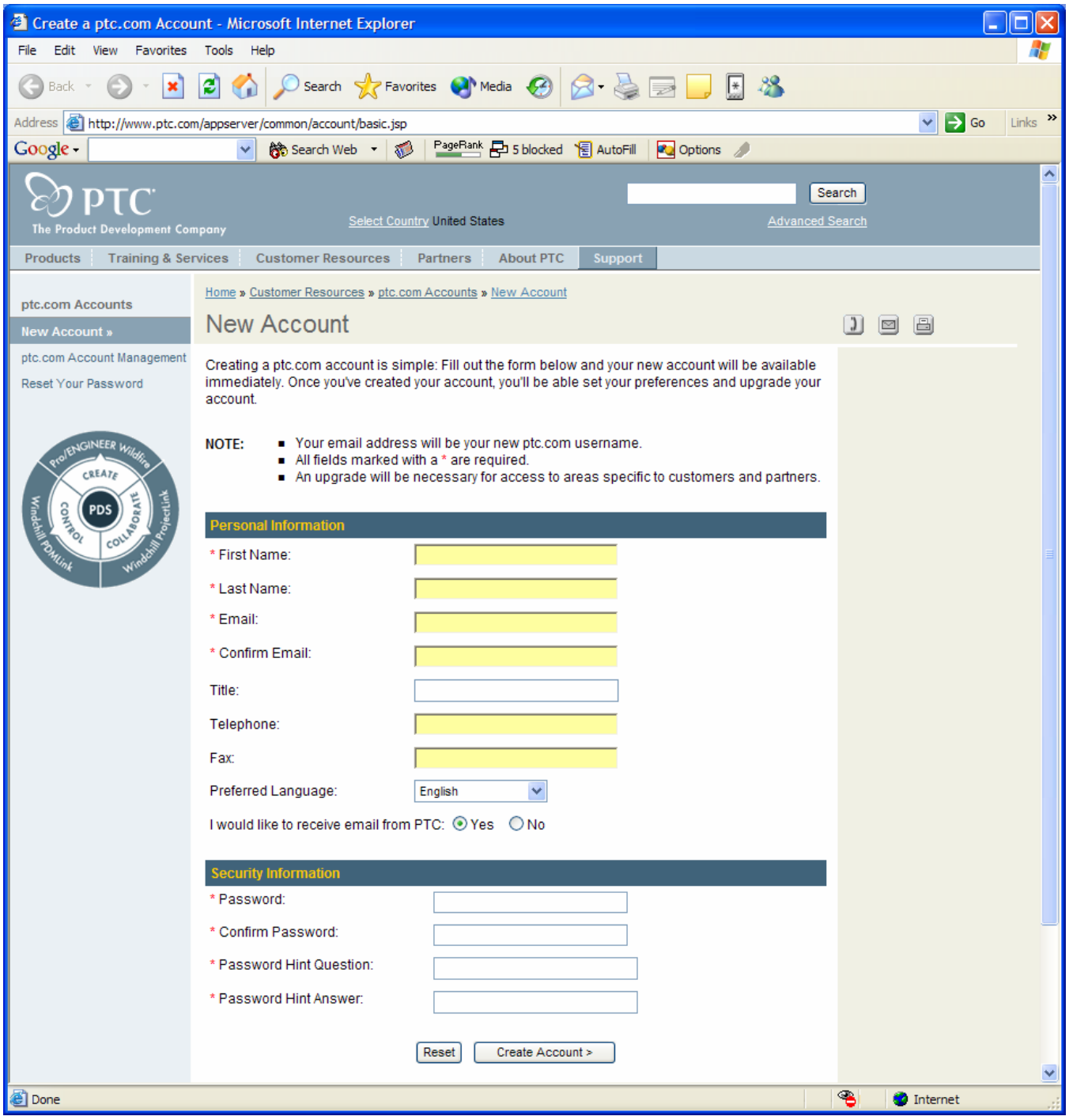

Once submitted a Confirmation window will be displayed. Be sure to review the information on this page before closing your browser window.

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.

Once an account has been created, you will be able to generate your license file.

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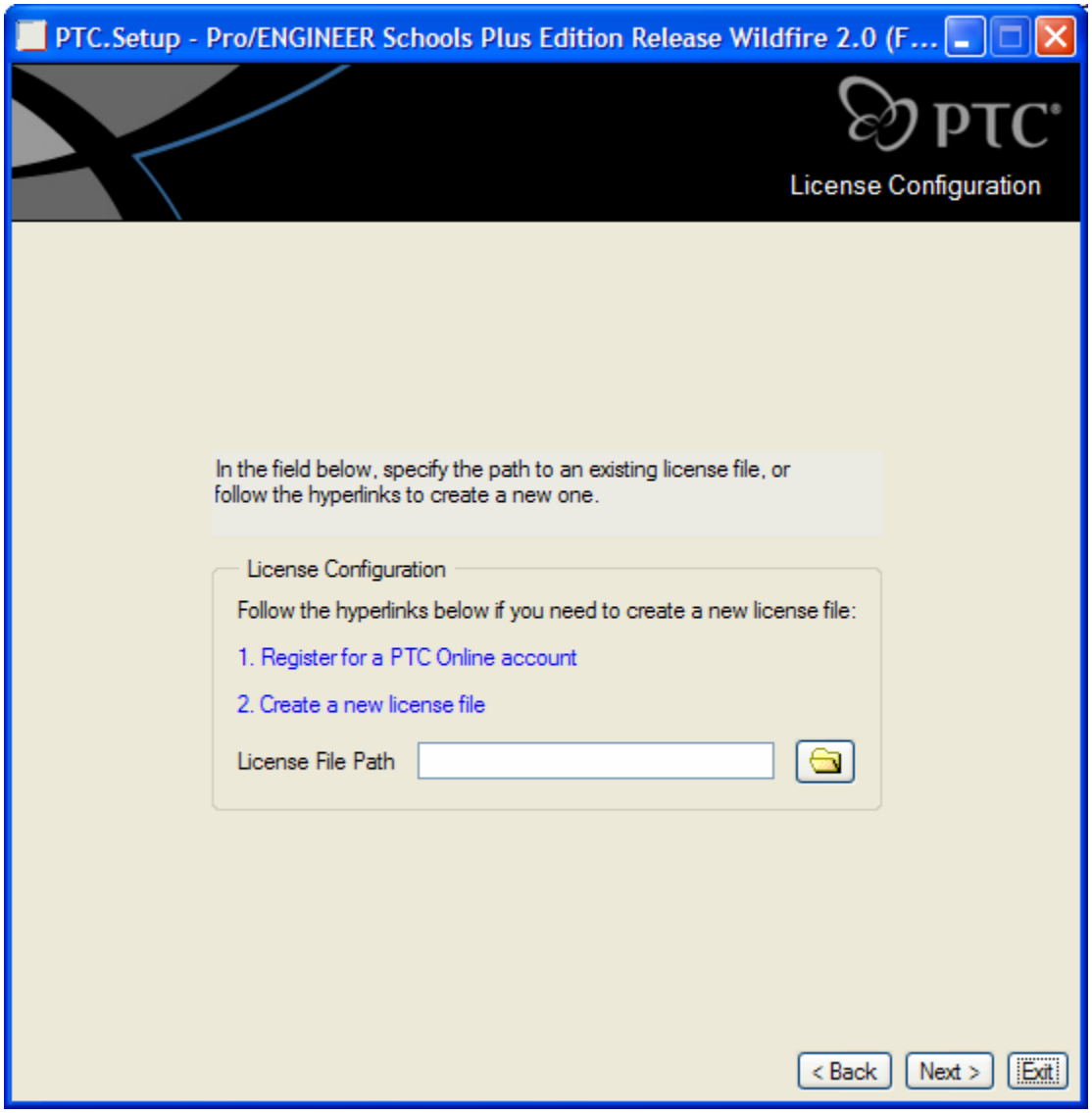
3. Creating a New License File

You now need to create a new license file for the Pro/E TE. From the License Configuration Screen in PTC.Setup, select the **Create a New License File** option.



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

In the Enter Network Password dialog box, type the User Name and Password you established in your Online Account Registration (step 2), and click OK.

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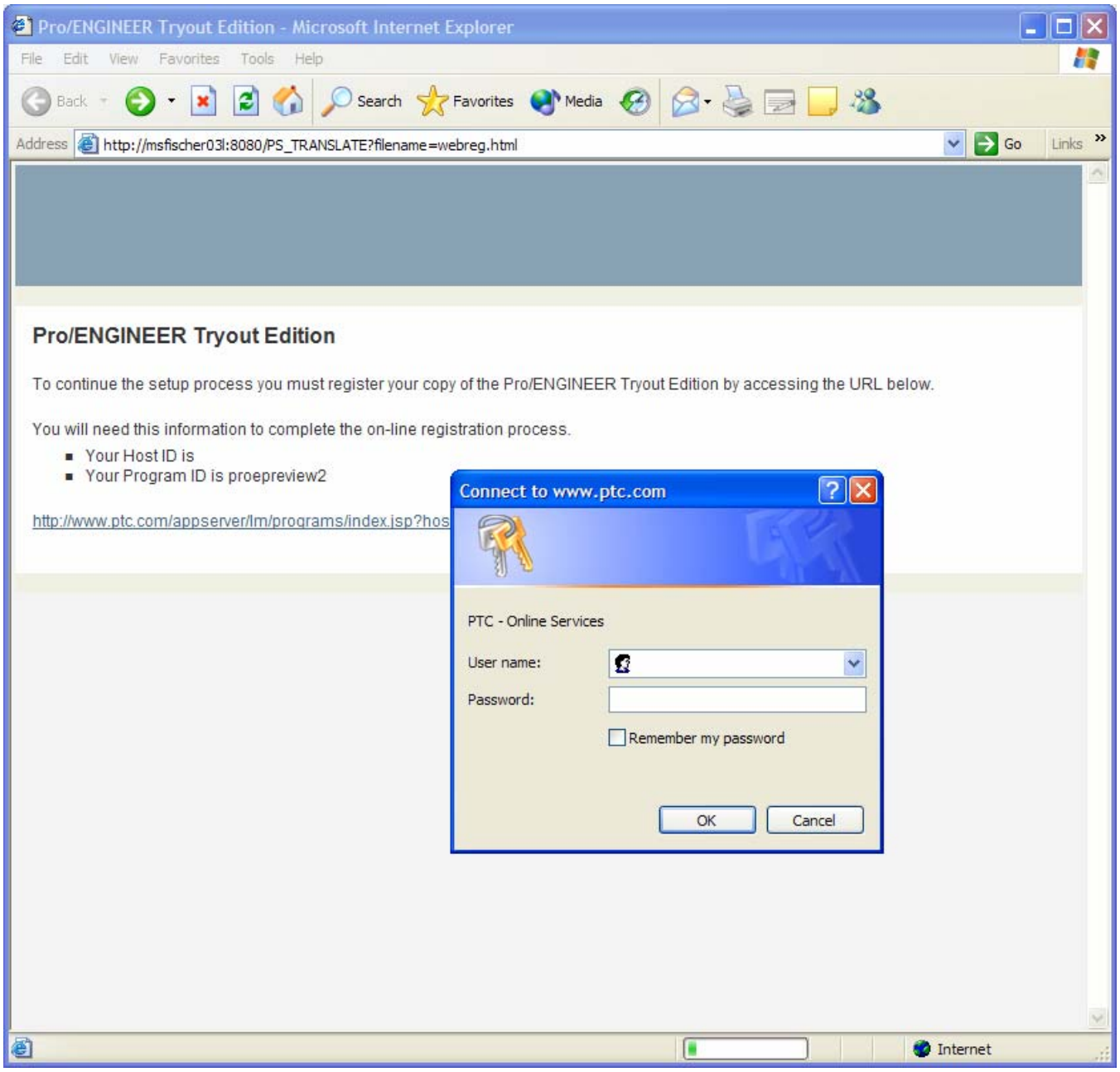

After clicking **OK**, the browser redirects you to the **Pro/ENGINEER Tryout Edition Licensing** web page. (This process may take a few moments.)

Fill in the required information and click Submit.

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*.**

This will process your request and an email will be sent within 2 business hours.

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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.*

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.

4. 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.

Note: Saving Your License Pack

When you receive your License Pack email, it will contain a file attachment under the name (or a similar name) of ***sw_license_email_0000.dat***. 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 *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 TE** from starting.

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 Tryout 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.

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

a. Click on the folder icon to the right of the **License File Path** field.

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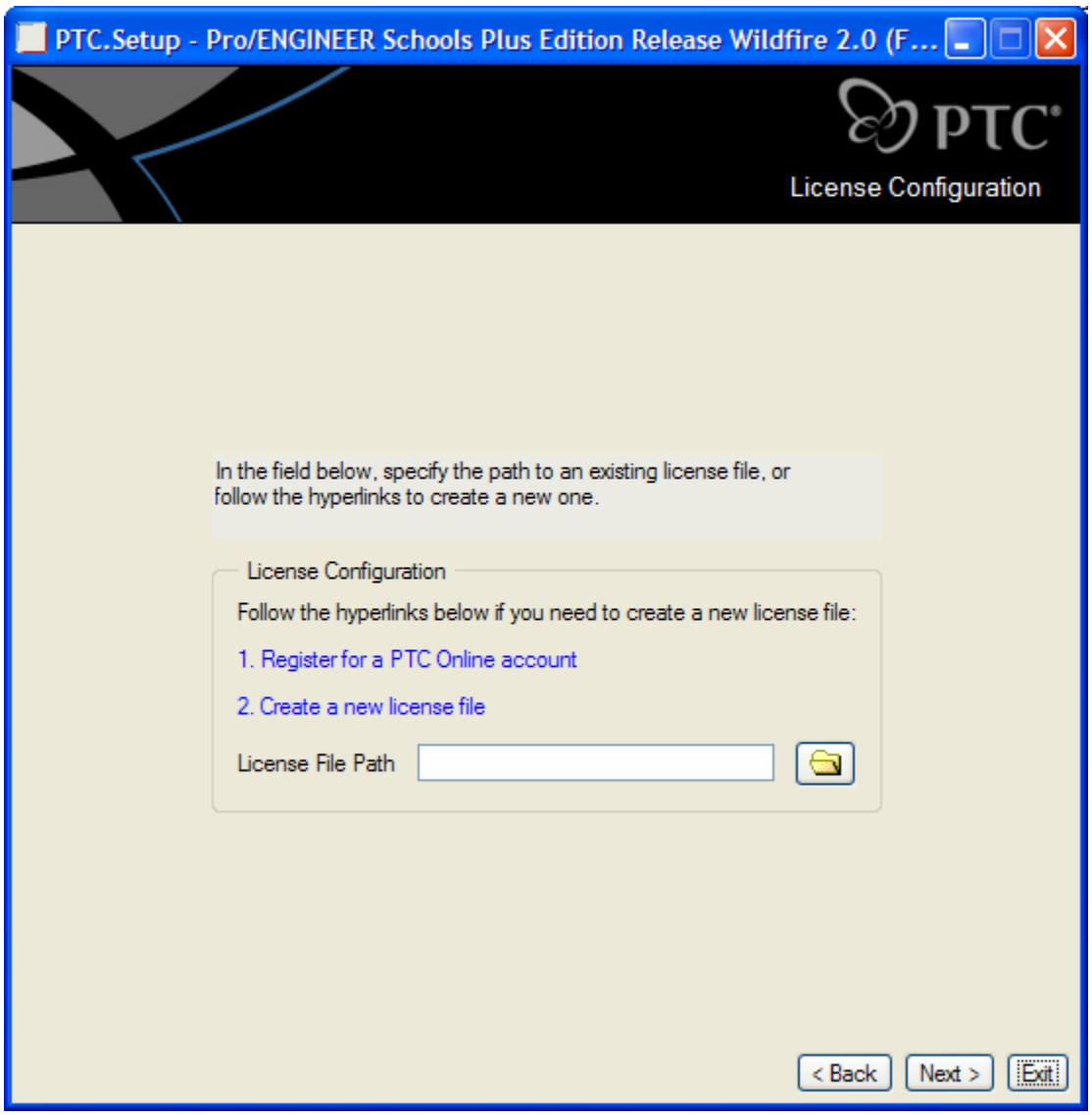

b. The second step to importing your license file is navigate to the directory that contains your Pro/ENGINEER Tryout Edition License file (as shown in the Select File dialog box below), and click **OK**. The directory (Folders) path will vary depending on where you saved your license file and may not appear exactly as in the example below.

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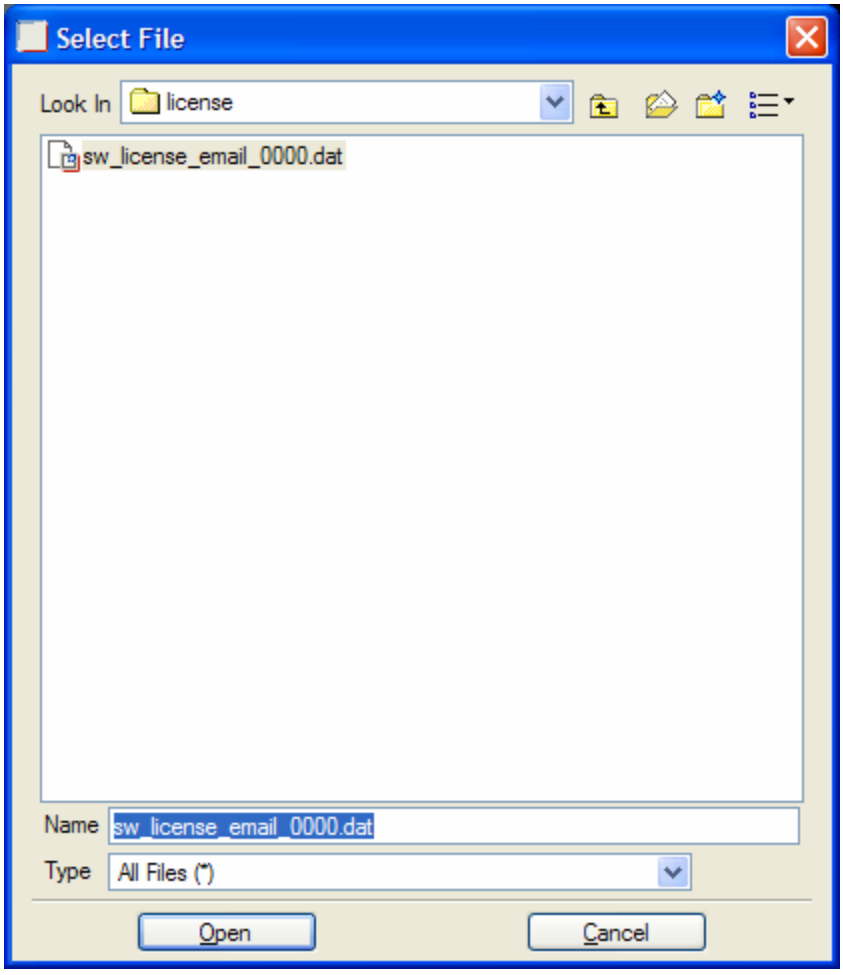

Note: In case of re-installation, PTC strongly recommends that your write down in a safe place the file location path that you saved the license file to.

5. 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. Select the installation directory to install Pro/E TE.

If you want to follow the default installation, in English, click **Next**.

Note: 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

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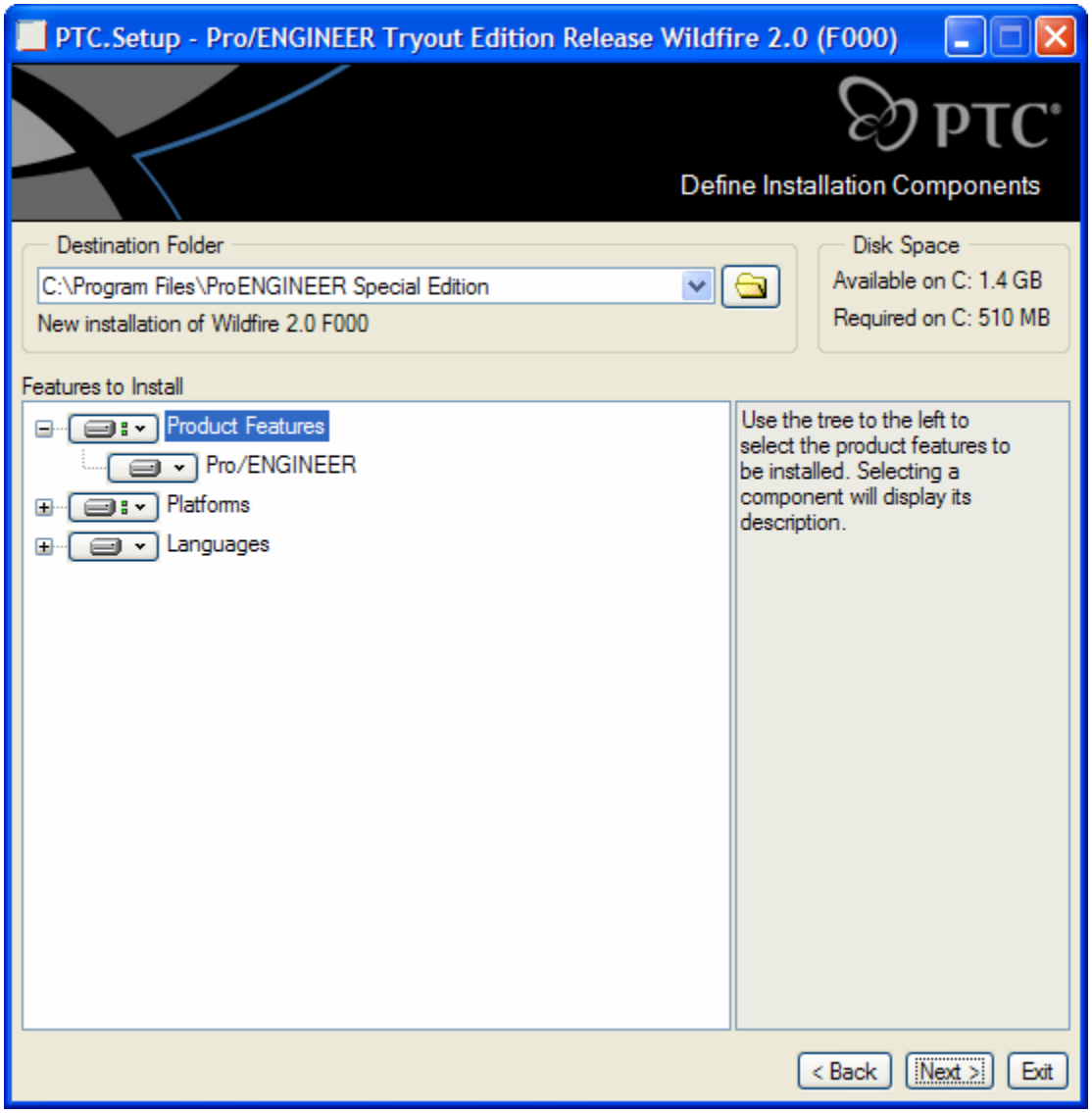

Destination Folder	**–** This is the directory in which Pro/ENGINEER is installed. The default installation directory is C:\Program Files\ProENGINEER Special Edition. If you wish to change the default location, you may simply type in another directory path, or select the browse button to the right of 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 Tryout Edition. PTC strongly recommends that a NEW installation be performed.
Platforms	**–** The only platform available in Pro/E TE 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 ant to install by selecting the check box next to the appropriate component.

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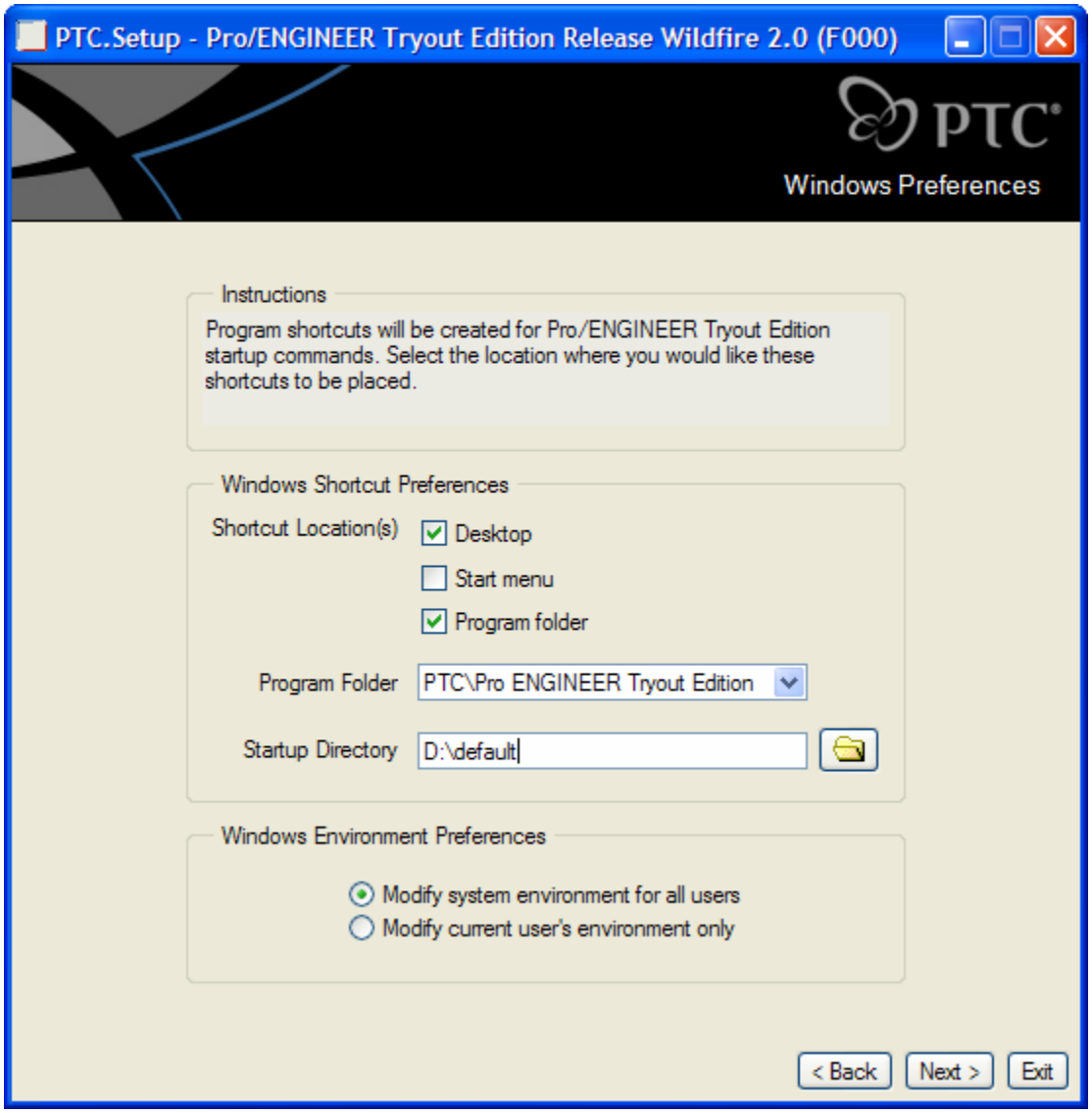
	The available components are as followed
Pro/ENGINEER (default)	– Keep this component to install the files necessary to run the software for Pro/E TE.
Subcomponents	**–** There are no subcomponents in the Pro/E TE.

6. Define Windows Preferences Options

You will then be able to select your Shortcut preferences. Once completed click the "**NEXT**" button



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7. Optional Configuration Steps

The **Optional Configuration Steps** dialog box now displays. Click **INSTALL** to begin the default settings installation and go to step 9 **OR** check the OLE settings if you want to customize your installation and follow the steps below.

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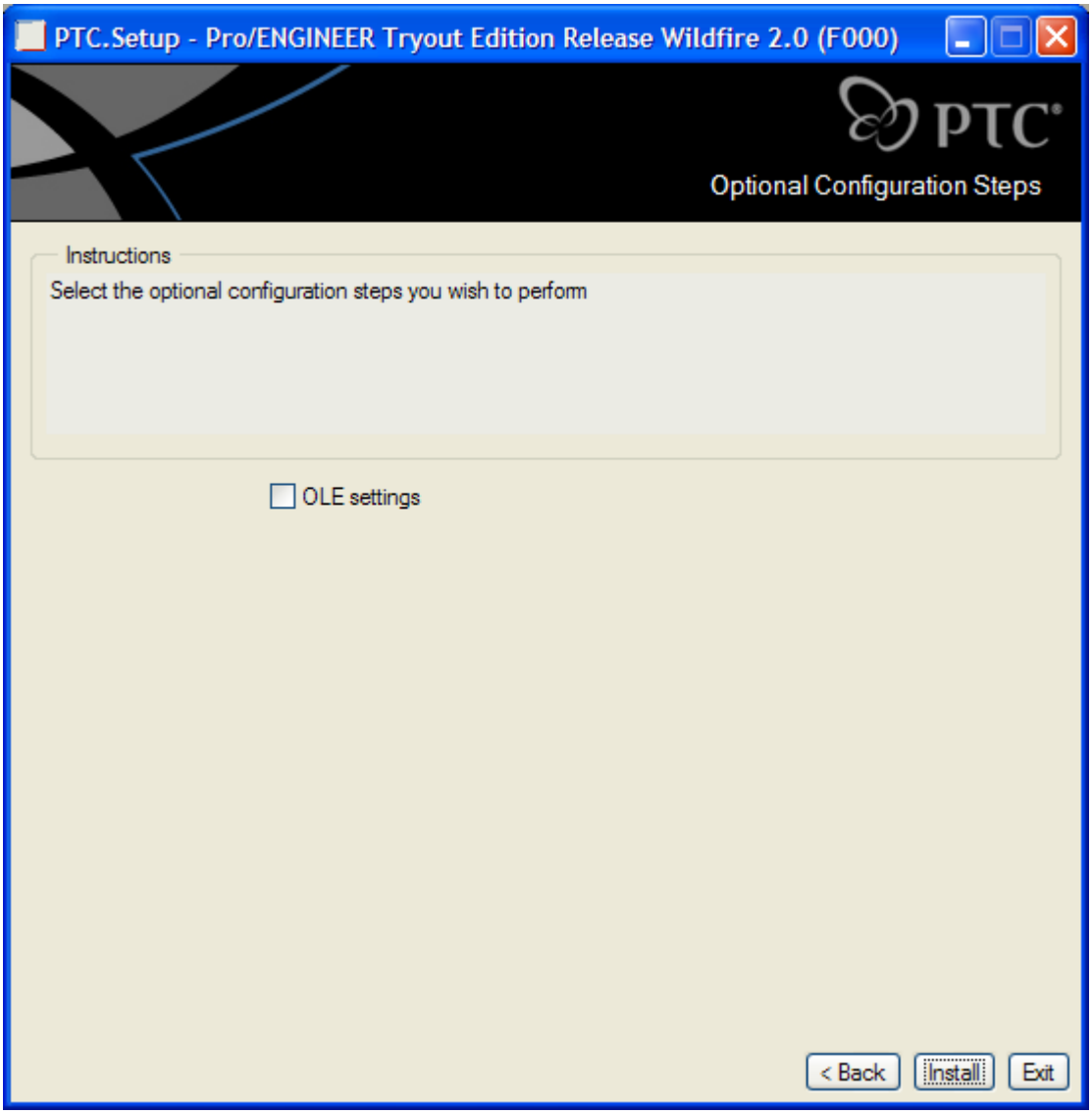

OLE Settings	**–** By keeping the default OLE settings box checked on the Optional Configuration Steps window you can configure Pro/ENGINEER Tryout Edition as an OLE (Object Linking and Embedding) Server. This allows the user of the client application to import Pro/ENGINEER files (parts, assemblies, 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.

Optional: 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.

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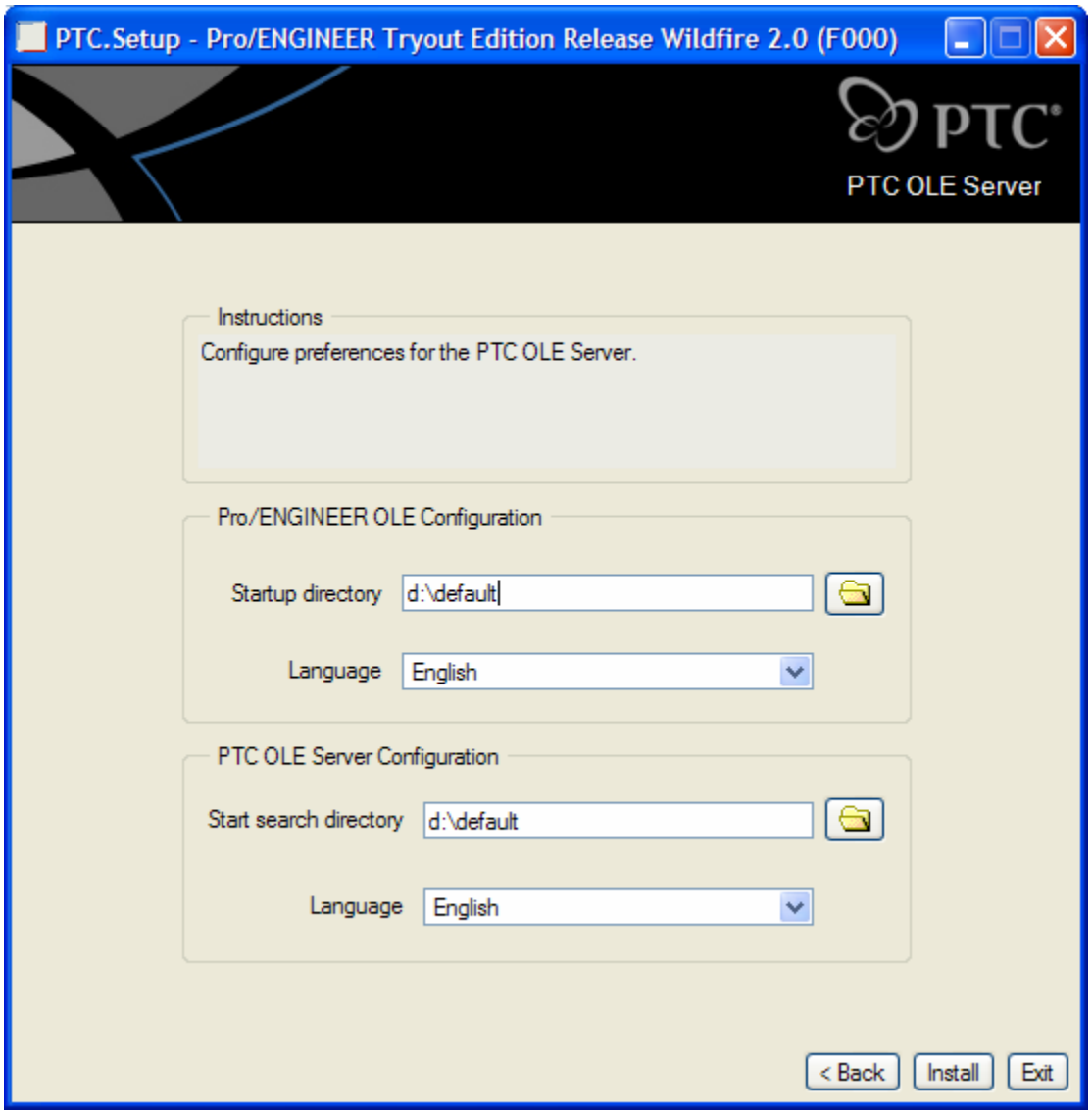

Pro/ENGINEER OLE Configuration

- 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.

8. Completing Pro/ENGINEER Configuration

Congratulations! Once you reach this point you have finished the OLE custom configuration on the Pro/ENGINEER installation.

Click **Install** to start the installation process, or choose one of the following options:

- Click **Back** to review information

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- Click **Install** to begin installation
- Click **Exit** to cancel PTC.Setup

The **Installation Progress** window (below) now greets you.

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

9. Once Pro/ENGINEER has been installed, Click Next to install the next selected component or to exit PTC.Setup

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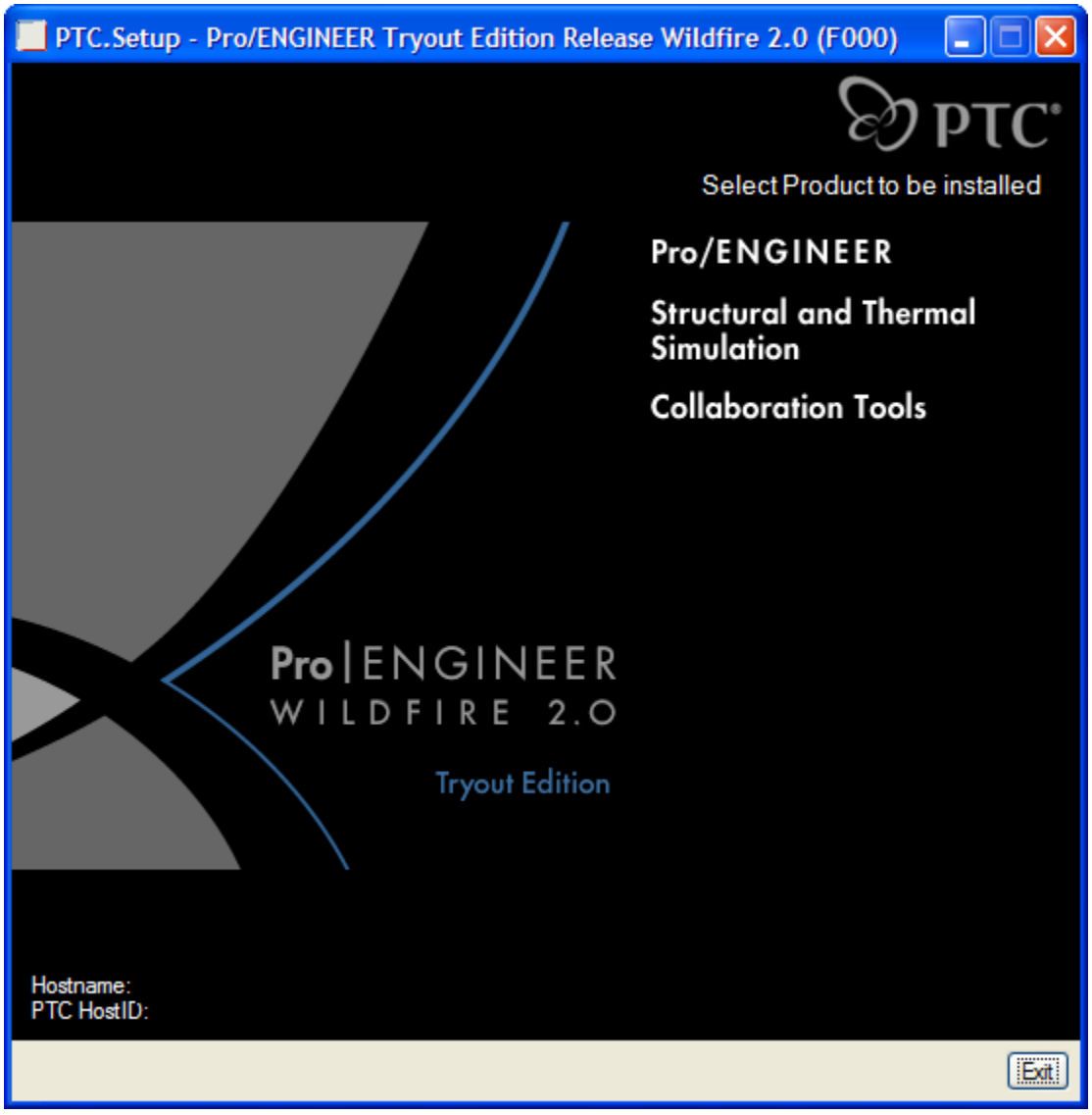
3.5 Installing Structural & Thermal Simulation (Pro/MECHANICA)

Note: Install Pro/ENGINEER before Structural and Thermal Simulation

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

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

2. Select **Structural and Thermal Simulation** by clicking on it on the display screen.



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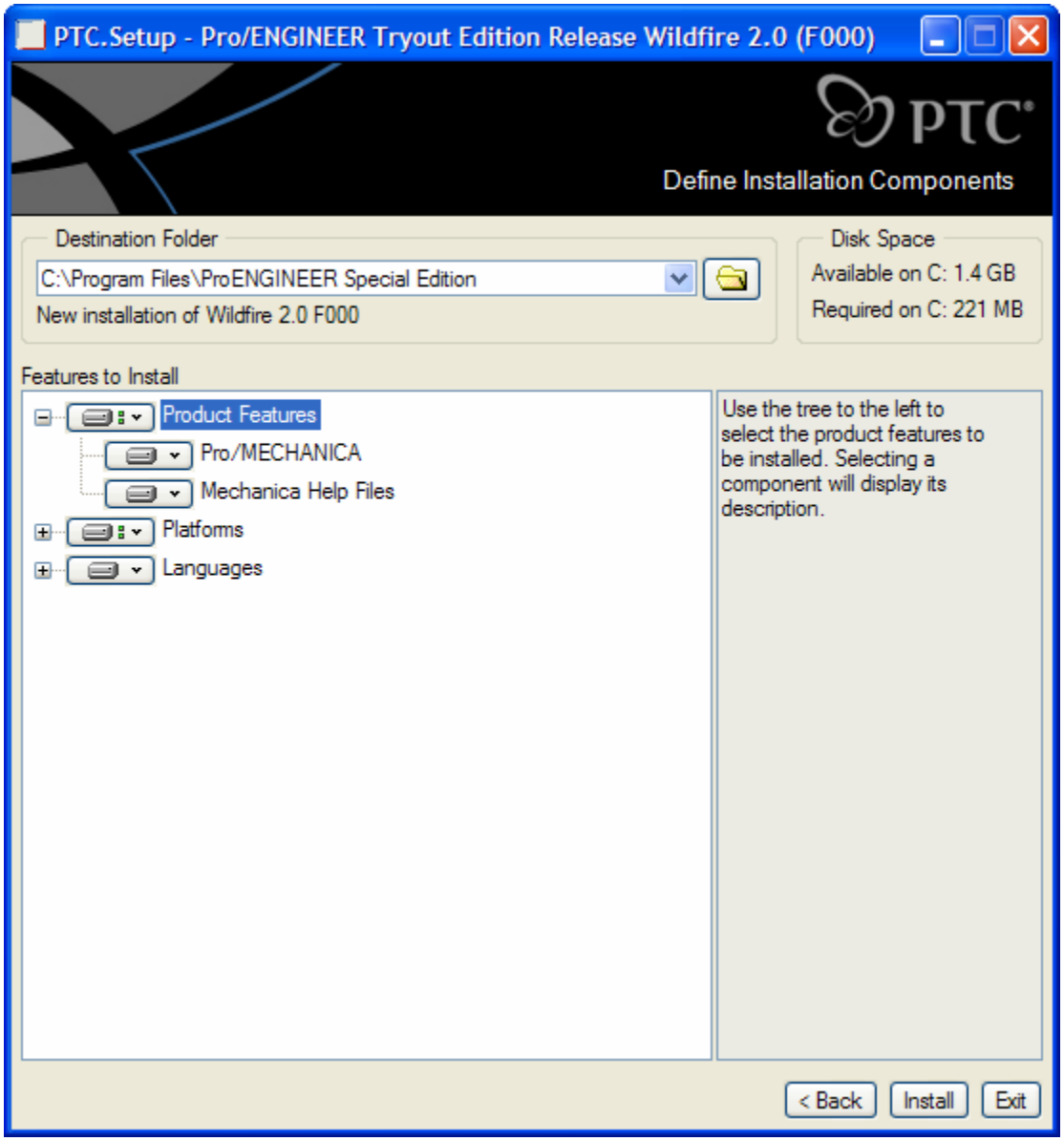
3. **Defining Installation Components**

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 Structural and Thermal Simulation installations. Once auto-detection is complete, you are presented with several options



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

Installation Directory – ****IMPORTANT**** This is the Directory in which Structural and Thermal Simulation is to be installed. Since Structural and Thermal Simulation Tryout Edition only runs in integrated mode, the installation directory must contain the path to a valid Pro/ENGINEER Tryout 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 Tryout Edition. This is the

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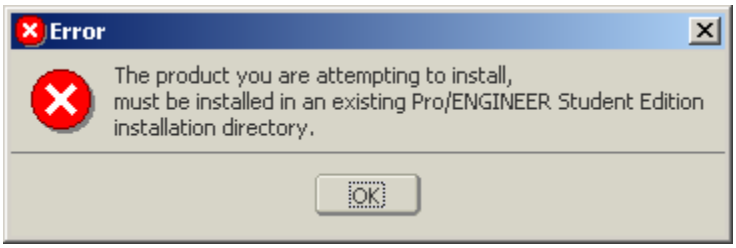
default path for Pro/ENGINEER installations, but if you choose to install Pro/ENGINEER to a location other than the default path, you must to manually browse to that folder by clicking the folder icon located to the right of the Installation Directory field.

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 Structural and Thermal Simulation Tryout 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 Tryout Edition is Pro/MECHANICA (default): Keep this component to install the files necessary to run the software for Structural and Thermal Simulation
Subcomponents	**–** There are no subcomponents in the Pro/E TE.

After all configuration options have been set on the **Define Installation Components** screen, click the **Next** button to continue.

4. **Installation Startup**

When you have finished configuring the Structural and Thermal Simulation installation, you are greeted with the **Installation Progress** window.

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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.

5. After 100% completion click **Next** to return to the Wildfire Display setup screen.

Note: Structural and Thermal Simulation does not create any shortcuts even if properly installed. For more information, see the section **Running Structural and Thermal Simulation Tryout Edition for the First Time**.

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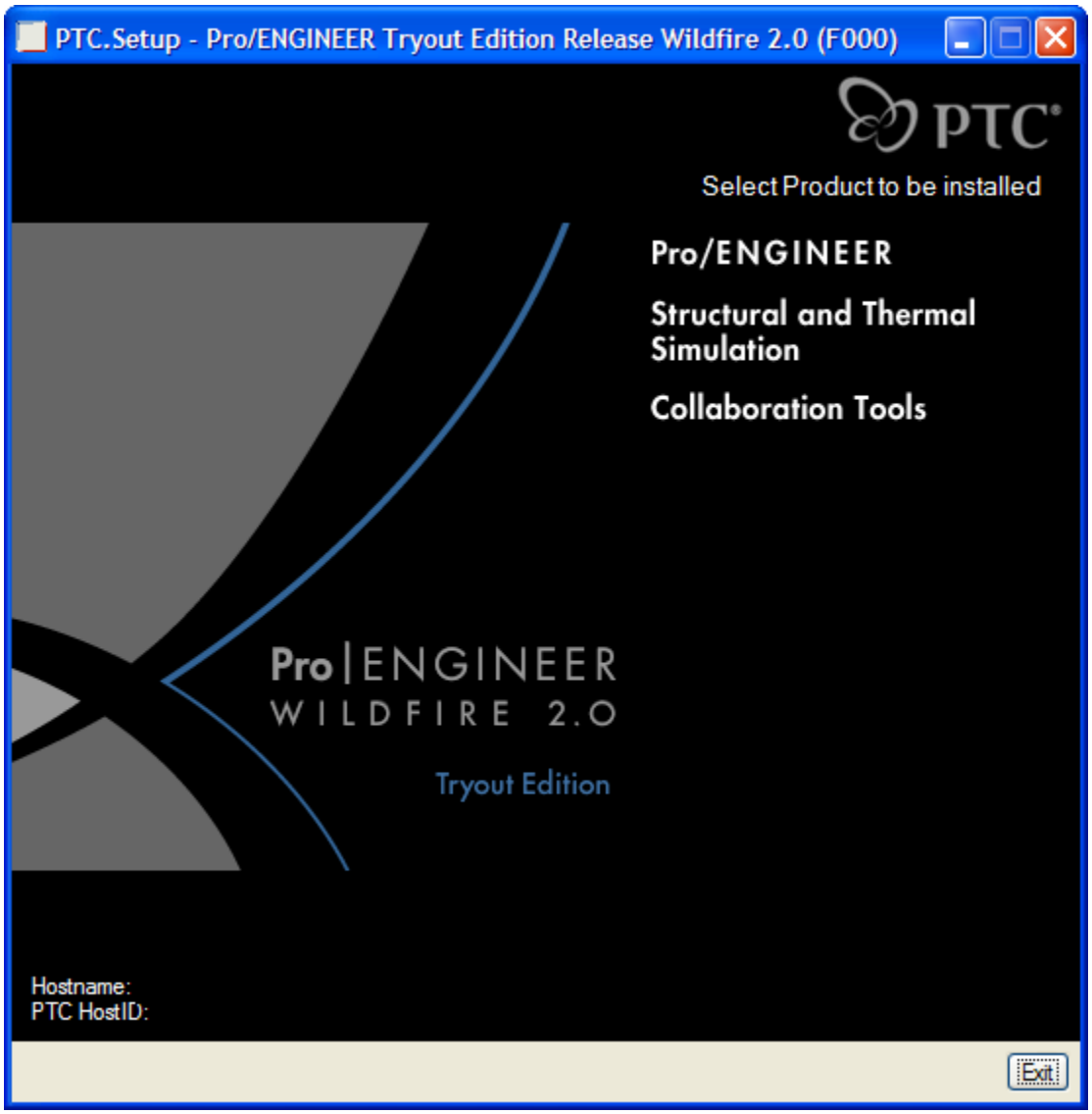
3.6 Installing Collaboration Tools

1. Select Collaboration Tools for installation click on it from the Wildfire setup Display screen.



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

2. **Defining Installation Components**

Note: You will notice a pause as the program searches to detect your system's settings. This pause occurs during both Pro/ENGINEER and Structural and Thermal Simulation installations. Once auto-detection is complete, you are presented with several options (see next page).

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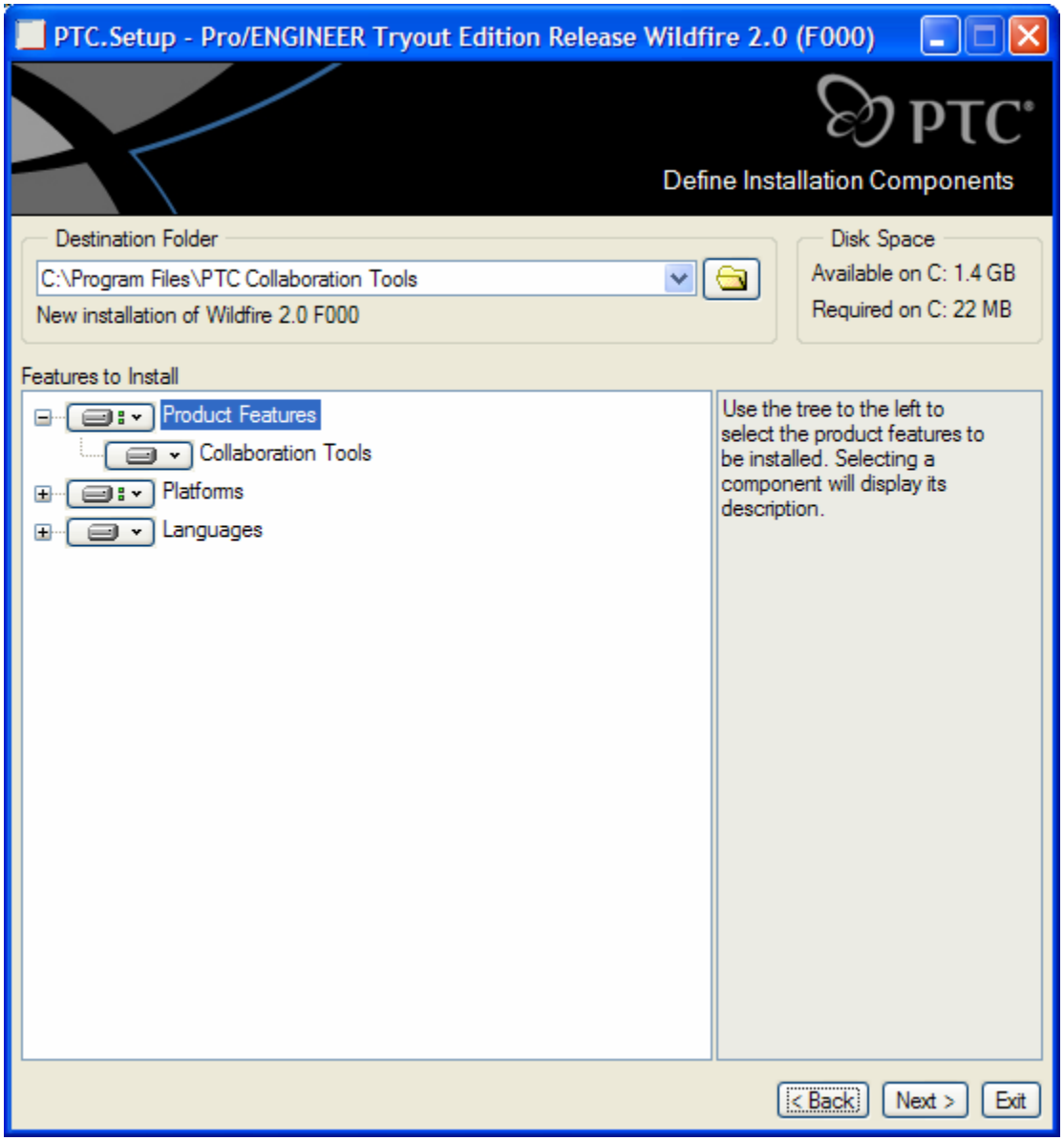

The options available on the **Define Installation Components** screen are as follows:

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 Tryout Edition.
Platforms	**–** The only platform available for the Tryout Edition of Collaboration Tools is i486_nt. This option is selected by default.
Languages	– Check any additional languages that you would like installed.

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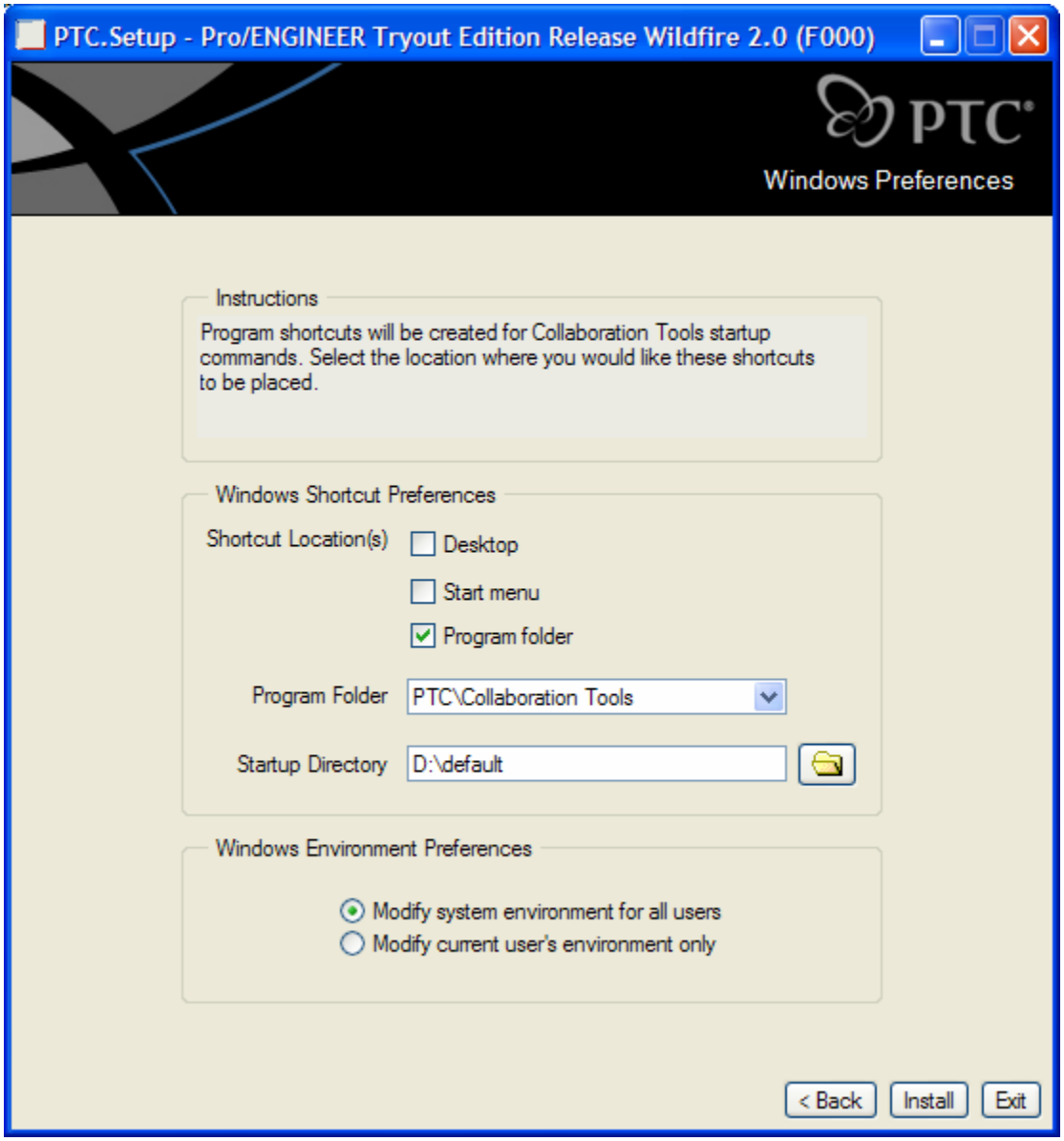
	If you wish to install additional language packs, more hard drive space will be required.
Components	**–** The only component that is available in the Tryout Edition is *Collaboration Tools* (default). Keep this component to install the files necessary to run Collaboration Tools.
Subcomponents	**–** There are no subcomponents in the Pro/E TE.

After you have set all configuration options this screen, click the **Next** button to continue.

3. **Completing the Collaboration Tools Installation**



Select your shortcut options if different than default settings. Click **Install** to start the installation process, or select one of the following options:

- Click **Back** to review information
- Click **Install** to begin installation

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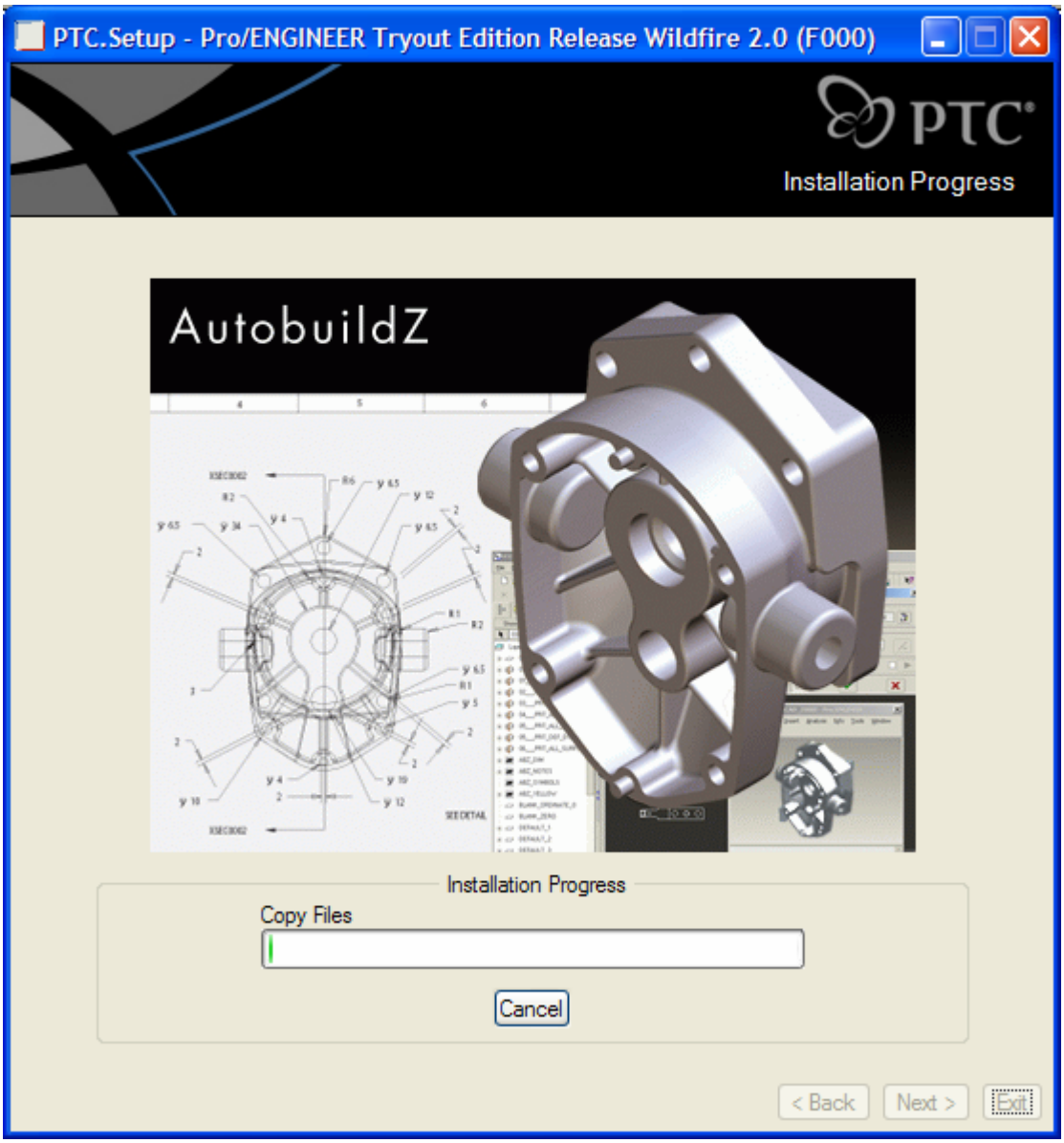
- Click **Exit** to cancel PTC.Setup

4. 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: PTC recommends that you use the version of Groove software supplied with the PTC software. PTC does not support the update of the Groove software outside the PTC.Setup installation utility.

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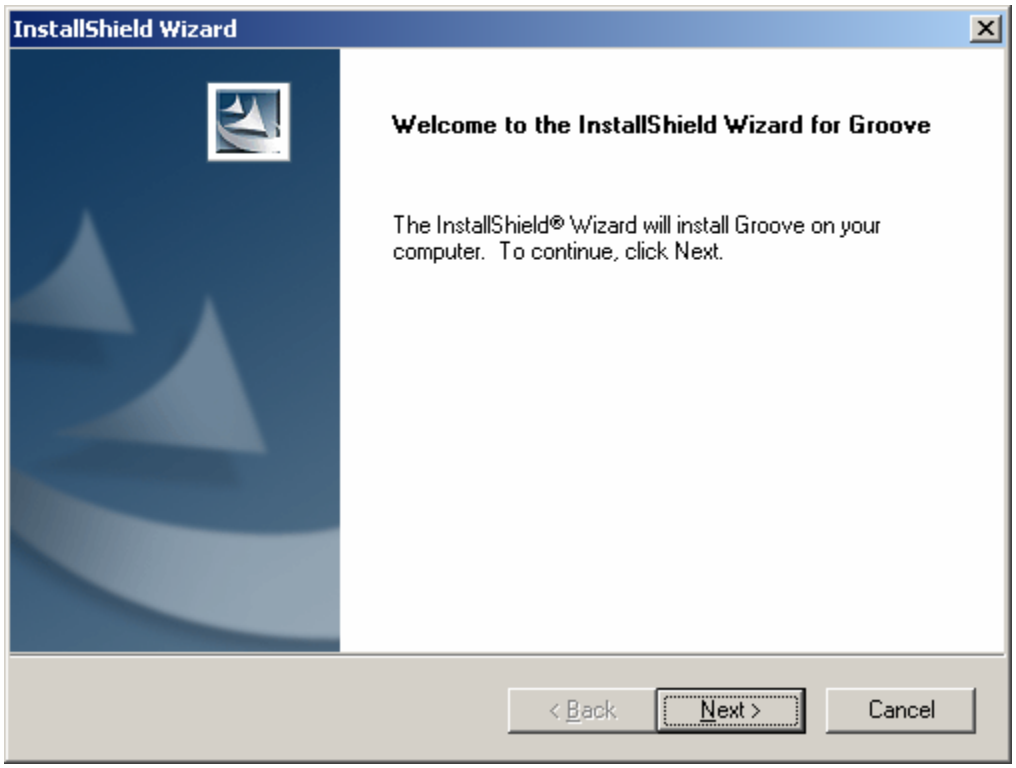
a. Click **Next** to continue the installation of Groove software.



b. Choose Destination Location

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

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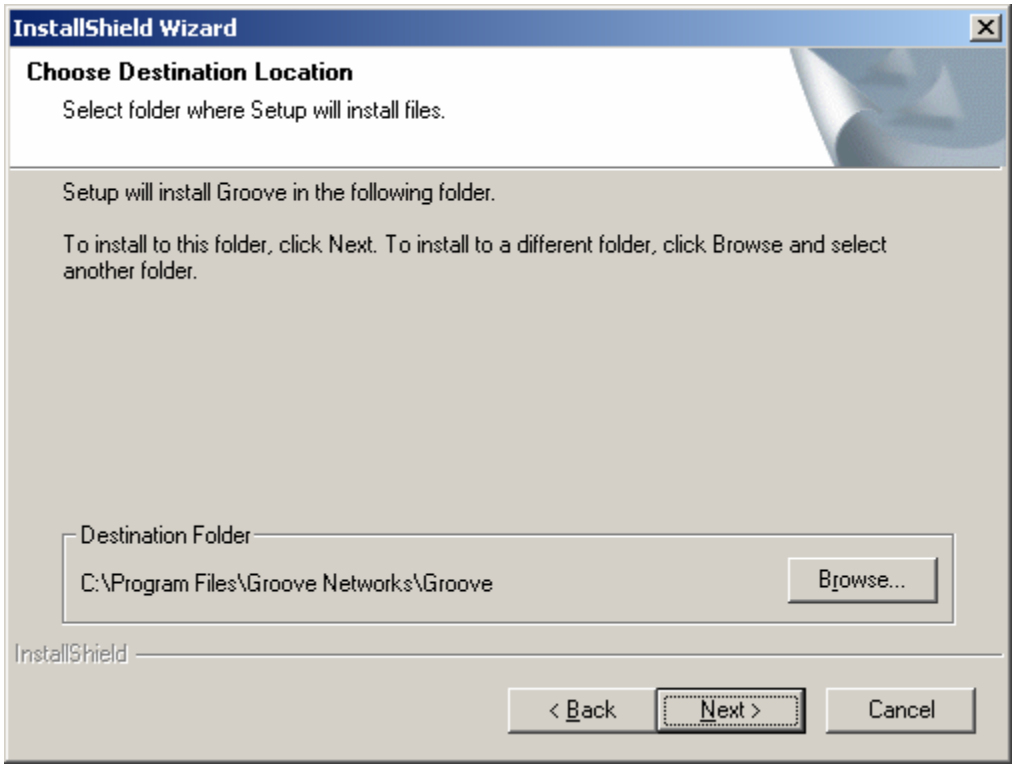

Click **Next** to create the default destination folder, or click **Browse** to select a different destination folder.

Note: PTC recommends that use the default location specified in the Groove install

c. **Choose User Data Location**

The **Choose User Data Location** dialog box now appears, giving you options for folder creation.

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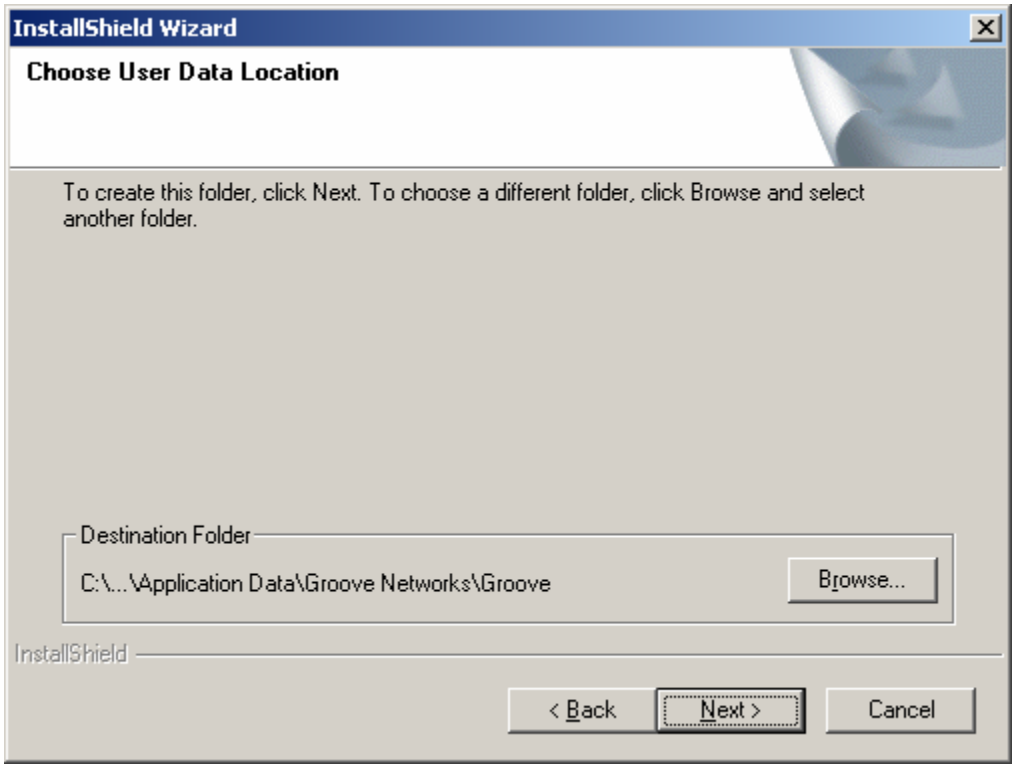

Click **Next** to create the user data folder, or click **Browse** to select a different folder.

Note: PTC recommends that use the default location specified in the Groove install

d. Choose System Data Location

The next screen contains a dialog box allowing you to **Choose System Data Location**.

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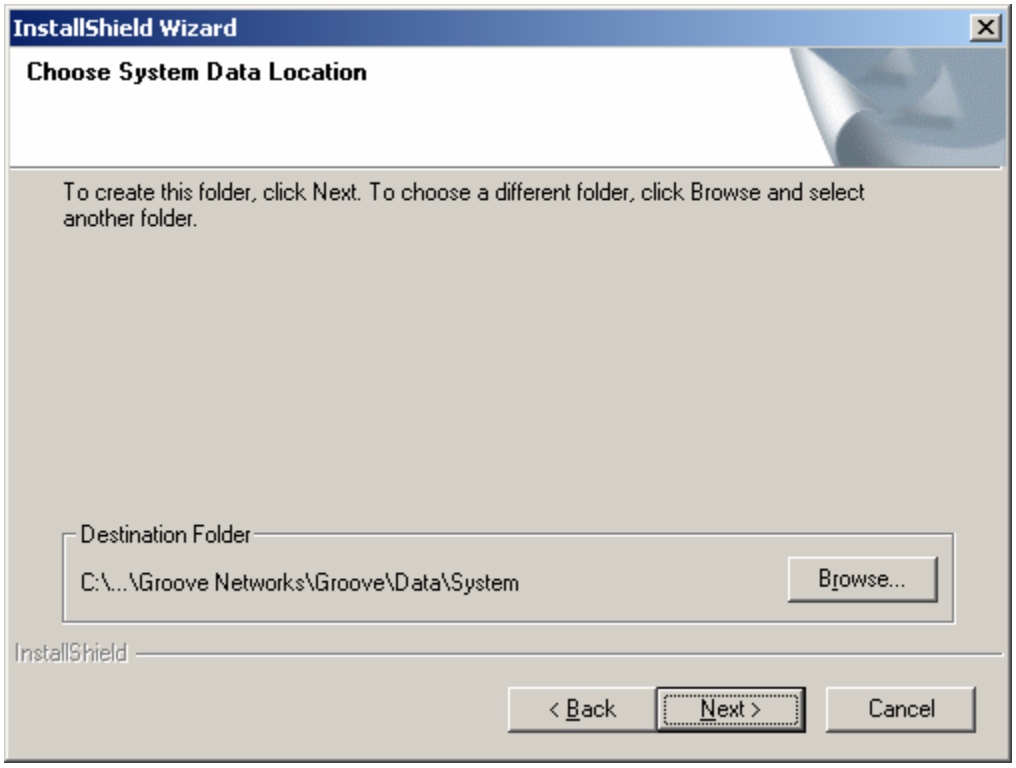

Click **Next** to create the system data folder, or click **Browse** to select a different folder.
Note: PTC recommends that use the default location specified in the Groove install

e. Installation Options

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

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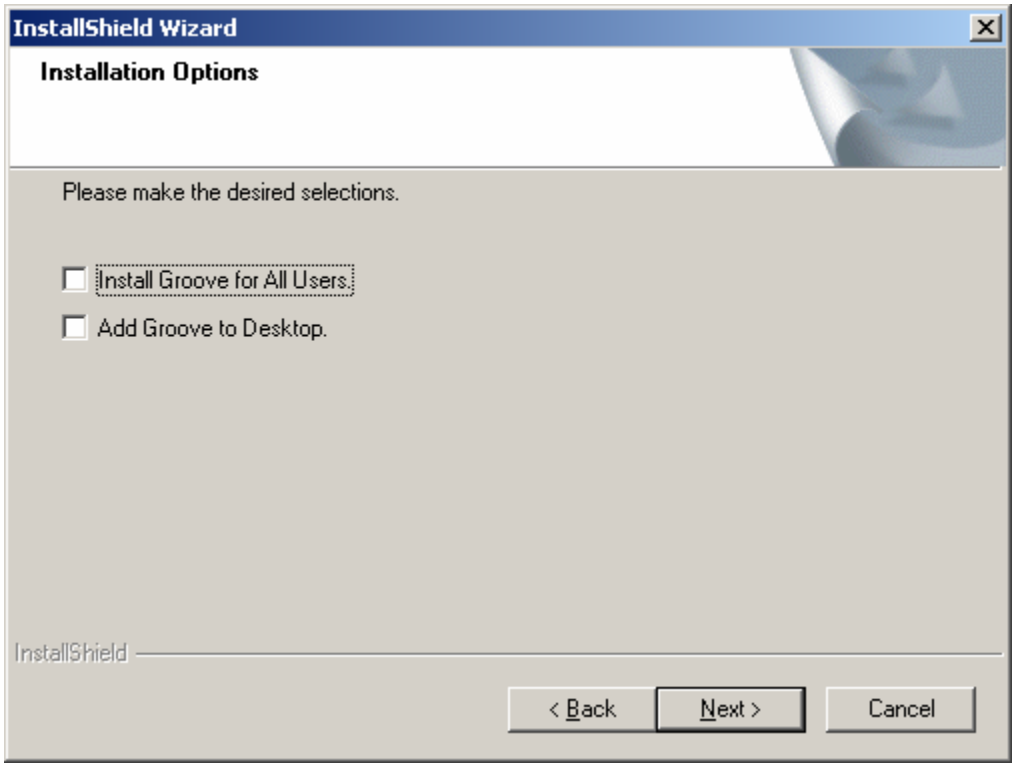

Choose one or more of the following installation options:

- **Install Groove for All Users –** Allows all users to run the Grove software (recommended)
- **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.*

After selecting your option(s), click **Next**.

f. Program Folder

Specify the Program folders, then click **Next**

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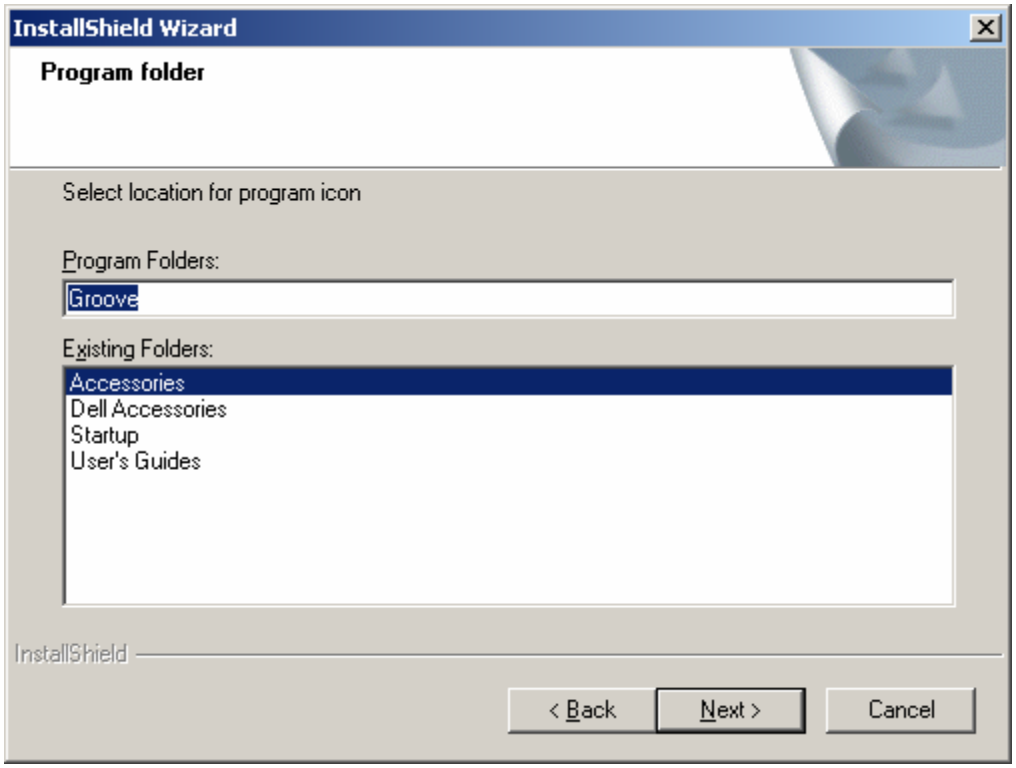
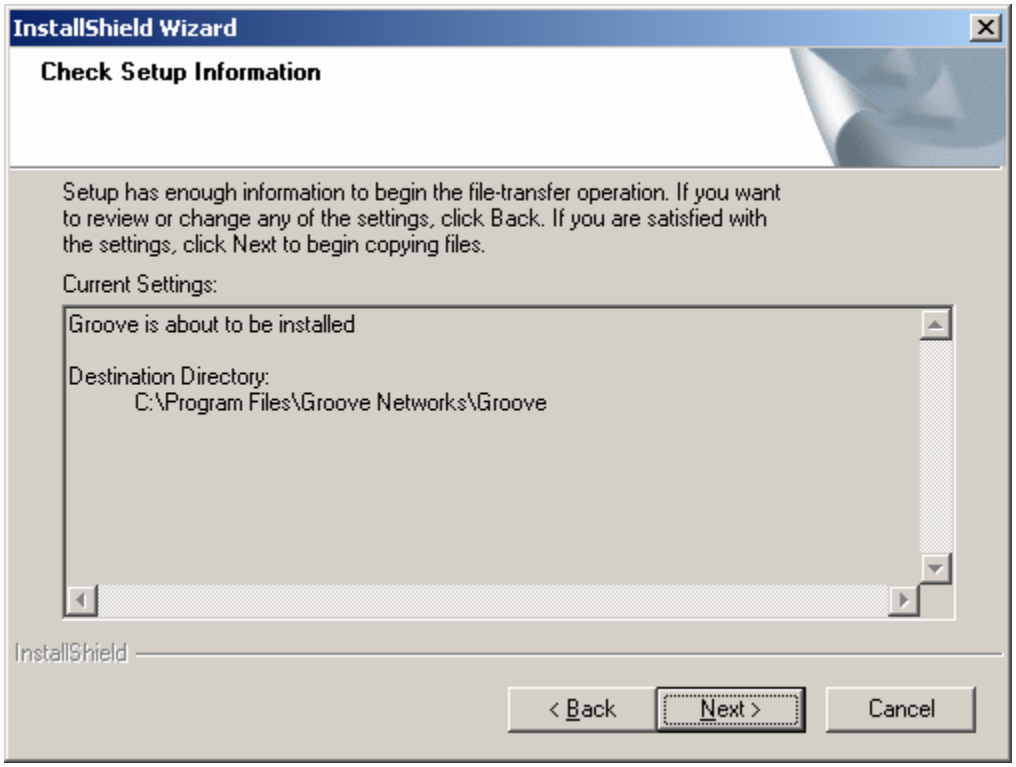


g. Check Setup Information

The **Check Setup Information** window opens with the installation information for the Groove software.



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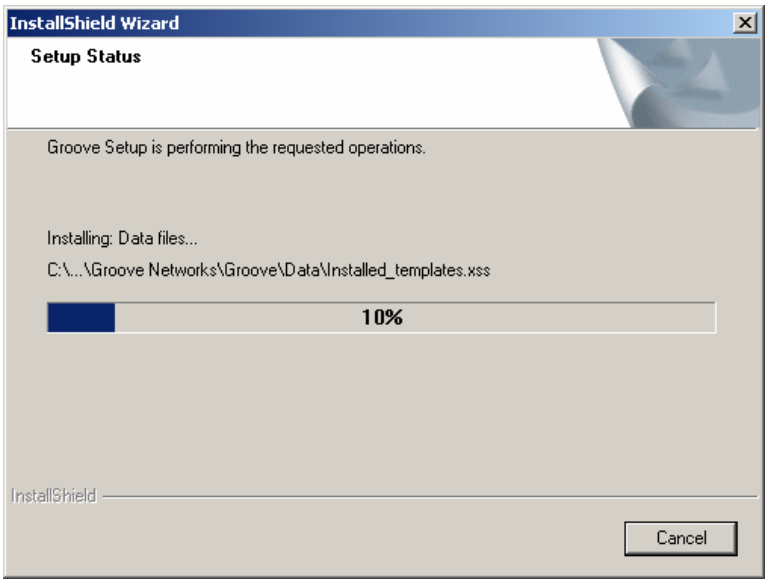
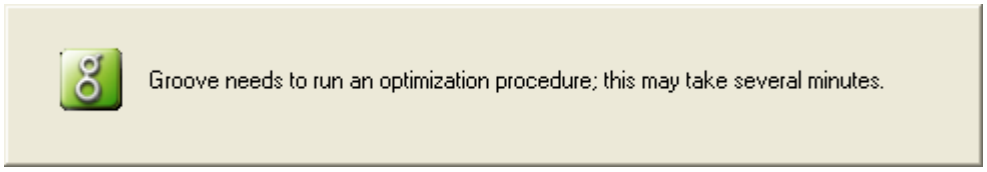
Click **Back** to go to the previous windows and change the installation settings, or click **Next** to continue which will start the installation of the Groove Software.

h. Setup Status

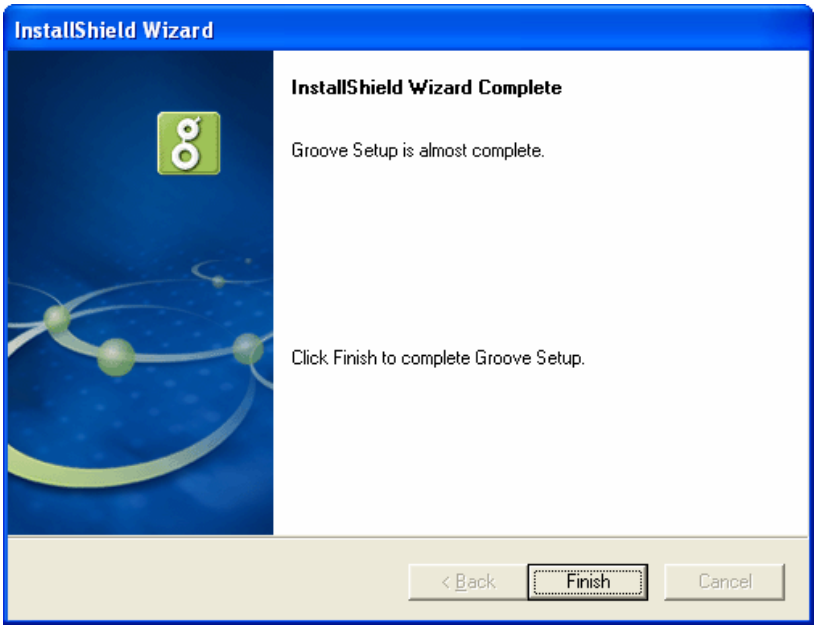
The **Setup Status** window now appears.



When installation reaches 100% the following window will appear. Allow the program to complete its task.



When the following Information dialog box appears click **Finish**



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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.

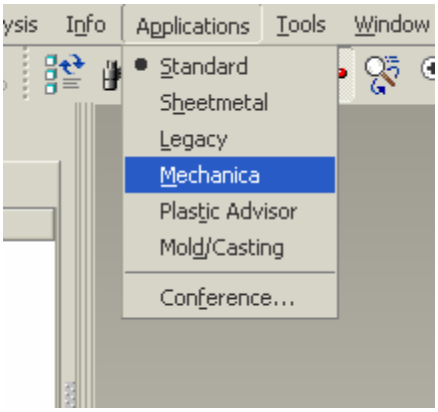
Note: PTC recommends that you use the version of Groove software supplied with the PTC software. PTC does not support the update of the Groove software outside the PTC.Setup installation utility.

4.0 Running Pro/MECHANICA for the First Time

1. Launch Pro/ENGINEER by clicking on the **Pro/ENGINNER Tryout 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.

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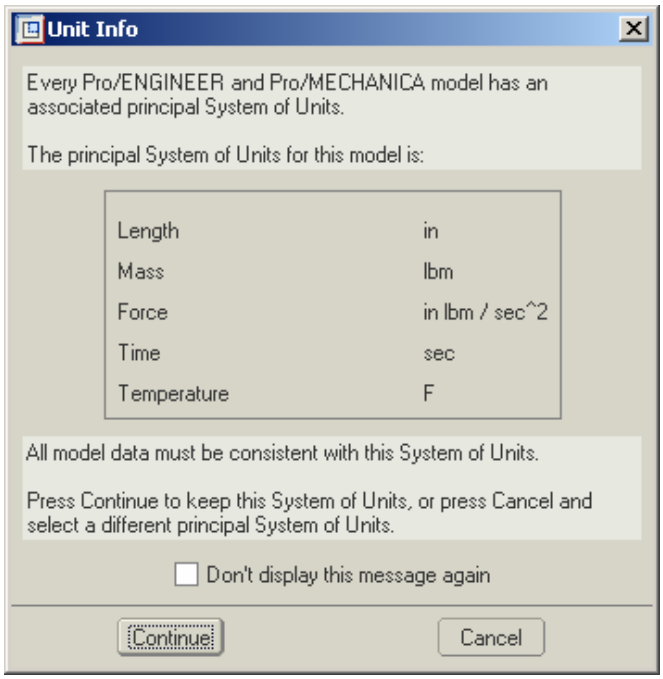

After a few moments the Menu Manger appears in the upper right-hand corner with MECHANICA as the title. The FEM mode is unchecked and **Indep Mec** is grayed out.

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 Structural and Thermal Simulation – Powered by Pro/MECHANICA.**

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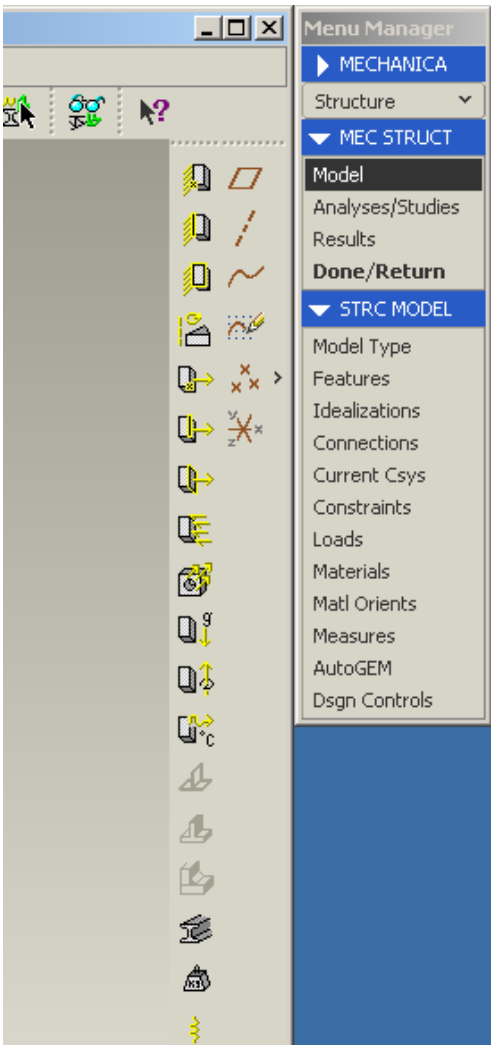

5.0 Using Design Conference for the First Time

The Conference Center is a standalone application that can link to Pro/ENGINEER. It can be started in two ways; either from its shortcuts or directly from within Pro/ENGINEER (as described below).

1. Launch Pro/ENGINEER by clicking on the **Pro/ENGINNER Tryout 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

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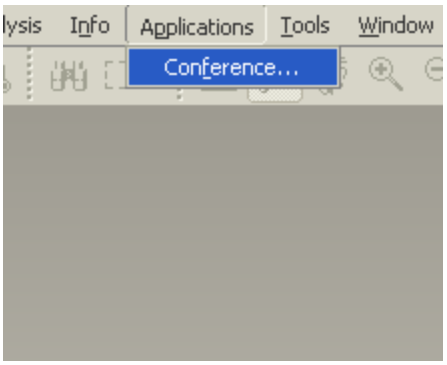
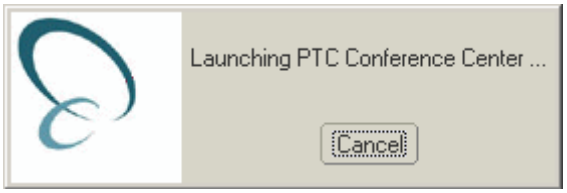
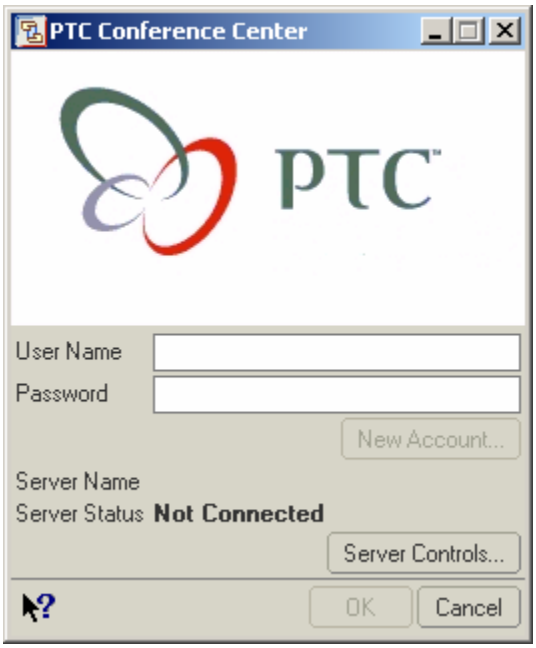


A small window appears, informing you that **Conference Center …** is launching.



The next window to open is the **PTC Conference Center** window.



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

3. Click the Start button to start the Server.

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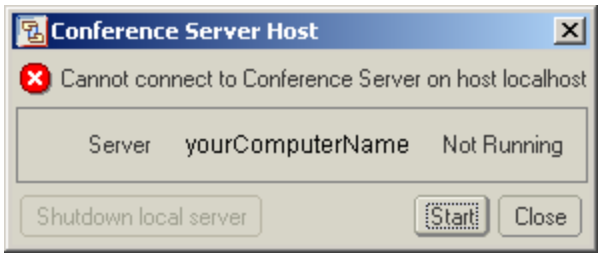
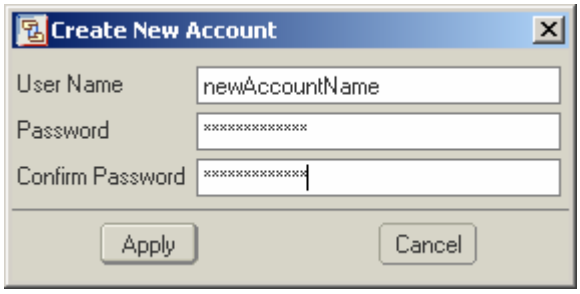
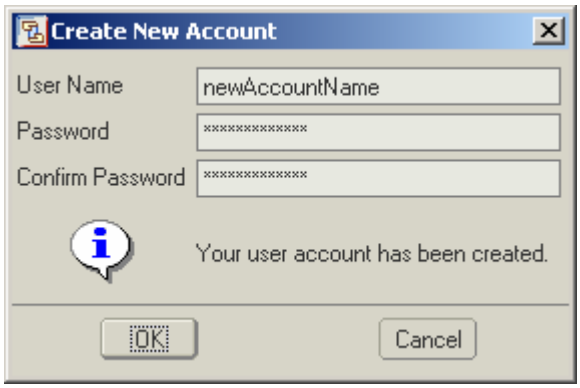


After a brief moment, the Service Status on the Conference center window will change to Connected.

4. Click the New Account button and enter the information requested to create a new account.



Once you have completed entering your information, click Apply. 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).



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

5. After confirming that your user account has been created, the PTC Conference Center dialog box displays on your screen.

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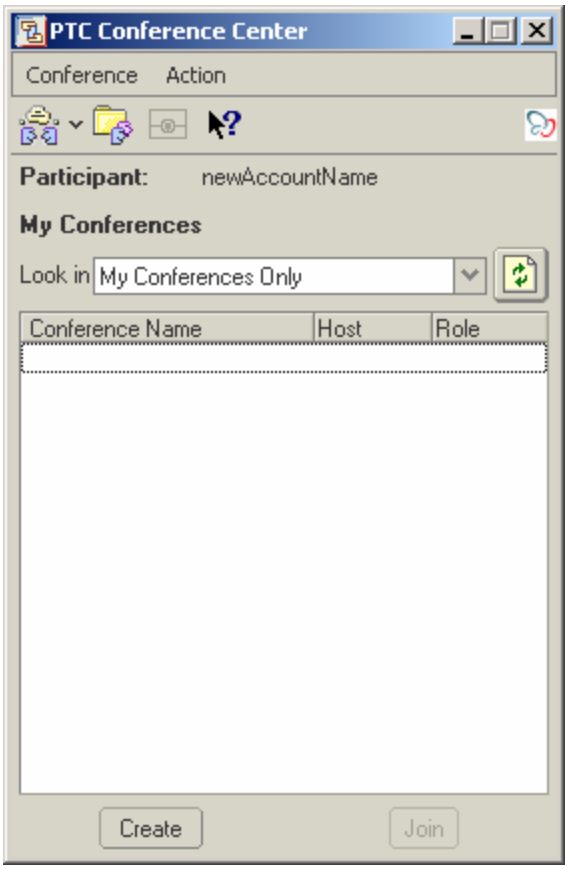

Congratulations! You are now ready to use Design Conferencing.

6.0 Basic Troubleshooting

The following common Q and A (Questions and Answers) have been provided to help you get up and running with your new Pro/E TE software. For additional troubleshooting assistance, please go to ***http://www.ptc.com/products/proe/studentedFAQ1.htm***

Q1. How do I locate my computer's Host ID?

A1. Insert your Pro/ENGINEER Tryout 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 Installing Pro/ENGINEER Wildfire 2.0 Tryout Edition 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

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

A2. 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:

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Physical Address. . . . . : 00-0A-03-D6-28-39 → This is your Host ID

Q2. What does the error "error in plpfhost" refer to?

This error may occur for one of two reasons.

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

Q3. What does the error "could not detect network adapter" refer to?

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

Q4. When attempting to start Pro/ENGINEER, you see an MS-DOS window that contains the message: Cannot Find <Loadpoint>\i486_nt\filename

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 Tryout Edition", but if you installed Pro/ENGINEER into a directory other than the default you loadpoint will need to reflect that change.

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