

# Chapter 1

---

## Introduction to Pro/ENGINEER Wildfire 5.0

### Learning Objectives

**After completing this chapter, you will be able to:**

- Understand the advantages of using Pro/ENGINEER Wildfire 5.0.
- Understand the bidirectional associative property.
- Know the system requirements of Pro/E.
- Learn various important terms and definitions in Pro/E.
- Learn various important options in the File menu.
- Understand the importance of Model Tree.
- Understand the functions of mouse buttons.
- Learn about the default toolbars.
- Understand the functions of browser.
- Understand the Appearance Gallery.
- Understand the rendering stages in Pro/ENGINEER.
- Change the color scheme of the background in Pro/ENGINEER.

## INTRODUCTION TO Pro/ENGINEER Wildfire 5.0

Welcome to Pro/ENGINEER Wildfire 5.0. If you are a new user of Pro/ENGINEER software package, you are going to join hands with thousands of users of this high-end CAD/CAM/CAE tool worldwide. If you are a user of the previous releases of this software, you are going to upgrade your designing skills with the tremendous improvement in this latest release.

Pro/ENGINEER Wildfire 5.0 is a powerful software used to create complex designs with great precision. The design intent of any three-dimensional (3D) model or an assembly is defined by its specification and its use. You can use the powerful tools of Pro/ENGINEER Wildfire 5.0 to capture the design intent of any complex model by incorporating intelligence into the design. Once you understand the feature-based, associative, and parametric nature of Pro/ENGINEER Wildfire 5.0, you can appreciate its power as a solid modeler.

To make the designing process simple and quick, this software package has divided the steps of designing into different modules. This means each step of the designing is completed in a different module. For example, generally a design process consists of the following steps:

- Sketching using the basic sketch entities.
- Converting the sketch into features and parts.
- Assembling different parts and analyzing them.
- Documenting parts and the assembly in terms of drawing views.
- Manufacturing the final part and assembly.

All these steps are divided into different modes of Pro/ENGINEER Wildfire 5.0; namely, the **Sketch** mode, **Part** mode, **Assembly** mode, **Drawing** mode, and **Manufacturing** mode.

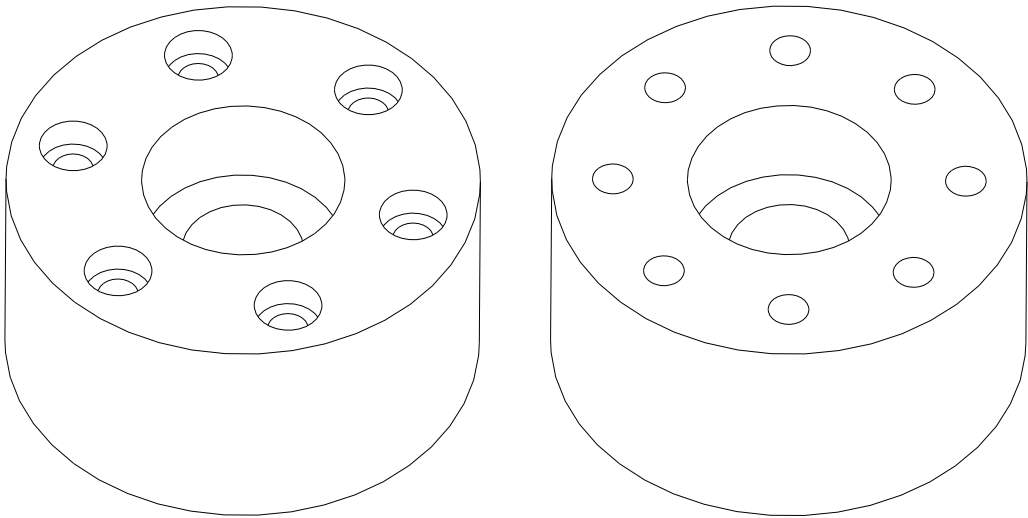
Despite making various modifications in a design, the parametric nature of this software helps preserve the design intent of a model with tremendous ease. Pro/ENGINEER allows you to work in a 3D environment and calculates the mass properties directly from the created geometry. The new feature in this release includes dynamic editing of features, which makes editing easier. You can also switch to various display modes like wireframe, shaded, hidden, and no hidden at any time with ease as it does not affect the model but changes the appearance of the model only. The new release of Pro/ENGINEER Wildfire enables the real-time rendering of solid models. Also, new appearances and scenes have been added to the library. It also includes shadows, reflections, perspective views and animations of the exploded state.

## FEATURE-BASED NATURE

Pro/ENGINEER Wildfire 5.0 is a feature-based solid modeling tool. A feature is defined as the smallest building block and any solid model created in Pro/ENGINEER Wildfire 5.0 is an integration of a number of these building blocks. Each feature can be edited individually to bring in any change in the solid model. The use of the feature-based property provides greater flexibility to the parts created. For example, consider the part shown in Figure 1-1. It consists of a counterbore hole at the center and six counterbore holes around it at some Bolt Circle Diameter (BCD).

Now, consider a case where you need to change all the outer counterbore holes to drill holes keeping the central counterbore hole and the BCD for the outer holes same. Also, you need to change the number of holes from six to eight. In a nonfeature-based software package, you need to delete the entire part and then create a new part as per the new specifications.

Whereas, Pro/ENGINEER Wildfire 5.0 allows you to make this modification by just modifying some values in the same part, see Figure 1-2. This shows that the solid parts created in Pro/ENGINEER Wildfire 5.0 are a combination of various features that can be modified individually at any time.



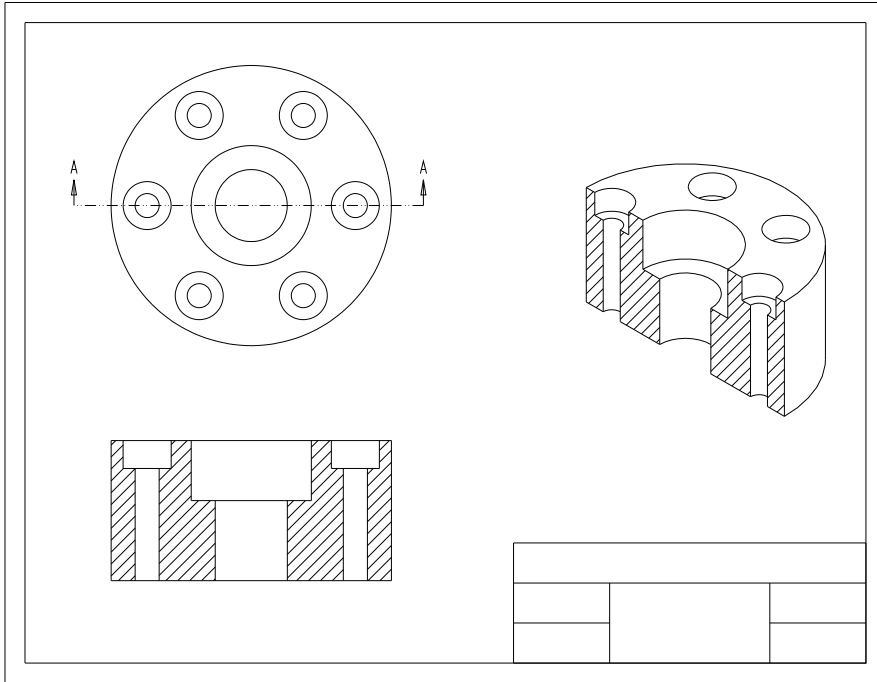
**Figure 1-1** Model displaying the counterbore holes **Figure 1-2** Model after making the modifications

## BIDIRECTIONAL ASSOCIATIVE PROPERTY

There is bidirectional associativity between all modes of Pro/ENGINEER Wildfire 5.0. The bidirectional associative nature of a software package is defined as its ability to ensure that if any modifications are made in a particular model in one mode, the corresponding modifications are also reflected in the same model in other modes. For example, if you make any change in a model in the **Part** mode and regenerate it, the changes will also be highlighted in the **Assembly** mode. Similarly, if you make any change in a part in the **Assembly** mode, after regeneration, the change will also be highlighted in the **Part** mode. This bidirectional associativity also correlates the two-dimensional (2D) drawing views generated in the **Drawing** mode and the solid model created in the **Part** mode of Pro/ENGINEER Wildfire 5.0. This means that if you modify the dimensions of the 2D drawing views in the **Drawing** mode, the change will be automatically reflected in the solid model and also in the assembly after regeneration. Likewise, if you modify the solid model in the **Part** mode, the changes will also be seen in the 2D drawing views of that model in the **Drawing** mode. Thus, bidirectional associativity means that if any modification is made to any one application, it changes the output of all the other modes related to the model. This nature relates the various modes in Pro/ENGINEER.

Figure 1-3 shows the drawing views of the part shown in Figure 1-1 generated in the **Drawing** mode. The views show that the part consists of a counterbore hole at the center and six counterbore holes around it.

Now, when the part is modified in the **Part** mode, the modifications are automatically reflected in the **Drawing** mode, as shown in Figure 1-4. The views in this figure show that all outer counterbore holes are converted into drilled holes and the number of holes is increased from six to eight.

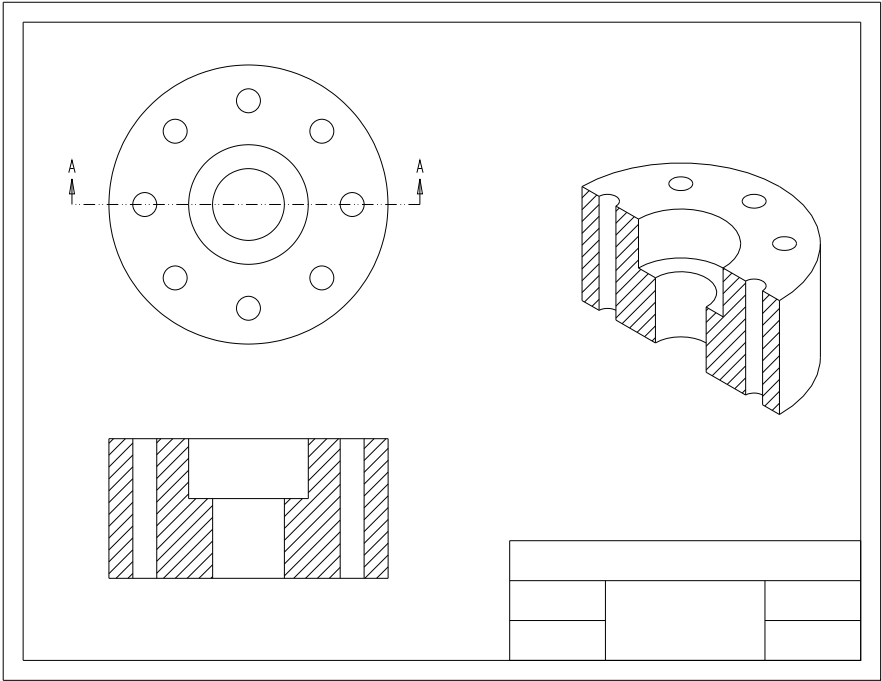


*Figure 1-3 Drawing views of the model before modifications*

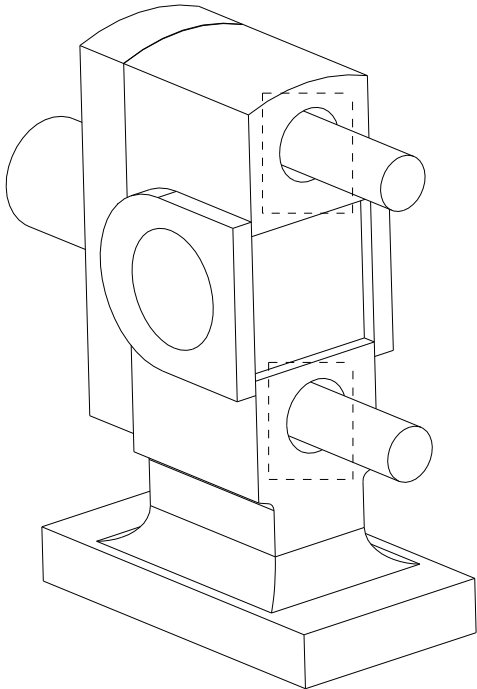
Figure 1-5 shows the Crosshead assembly. It is clear from the assembly that the diameter of the hole is more than what is required (shown using dotted lines). In an ideal case, the diameter of the hole should be equal to the diameter of the bolt.

The diameter of the hole can be changed easily by opening the file in the **Part** mode and making the necessary modifications in the part. This modification is reflected in the assembly, as shown in Figure 1-6. This is due to the bidirectional associative nature of Pro/ENGINEER.

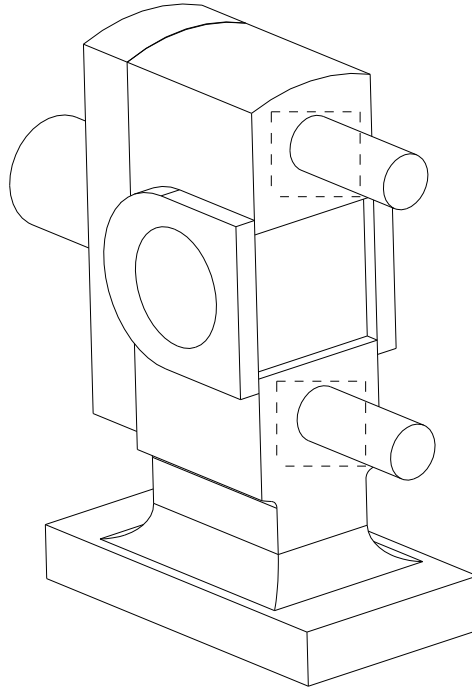
Since all modes of Pro/ENGINEER Wildfire 5.0 are interrelated, it becomes very easy to modify your model at any time. This makes the application software more user-friendly.



**Figure 1-4** Drawing views of the model after modifications



**Figure 1-5** Diameter of the hole and the bolt



*Figure 1-6 Model after modifying the diameter of the hole*

## PARAMETRIC NATURE

Pro/ENGINEER Wildfire 5.0 is parametric in nature, which means that the features of a part become interrelated if they are drawn by taking the reference of each other. You can redefine the dimensions or the attributes of a feature at any time. The changes will propagate automatically throughout the model. Thus, they develop a relationship among themselves. This relationship is known as the parent-child relationship. So if you want to change the placement of the child feature, you can make alterations in the dimensions of the references and hence change the design as per your requirement. The parent-child relationship will be discussed in detail while discussing the datums in later chapters.

## SYSTEM REQUIREMENTS

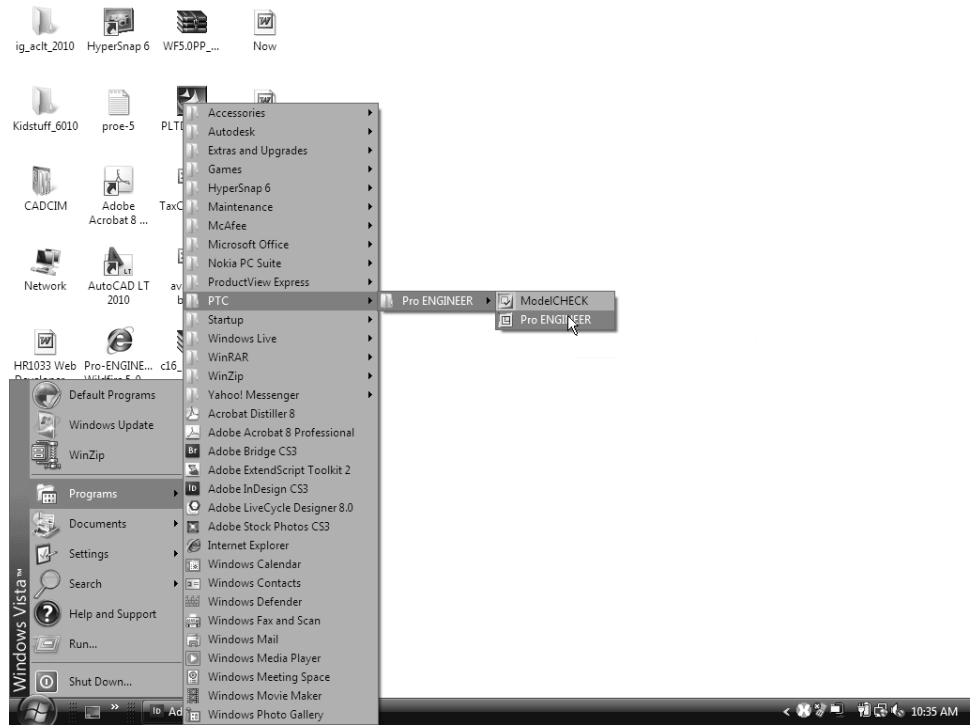
The system requirements for Pro/ENGINEER Wildfire 5.0 are given below.

1. Operating System: Windows XP Home Edition, XP Professional Edition, XP Professional x64 Edition, Windows Vista (Professional, Ultimate, and Enterprise Editions), Solaris (64 bit only).
2. Monitor: 1280 x 1024 (or higher) resolution support with 24-bit or more.
3. Processor: 500MHz minimum (2.4GHz or higher recommended).
4. Memory: 256MB RAM minimum (1024MB or higher recommended).

5. Swap Memory: 500MB minimum (2048MB or higher recommended).
6. Hard disk space: 2.00GB minimum (3.0GB or higher recommend).
7. An ethernet adapter interface card or network card.
8. Microsoft approved 3-button mouse.
9. Microsoft Internet Explorer 6.0 or later.

## GETTING STARTED WITH Pro/ENGINEER Wildfire 5.0

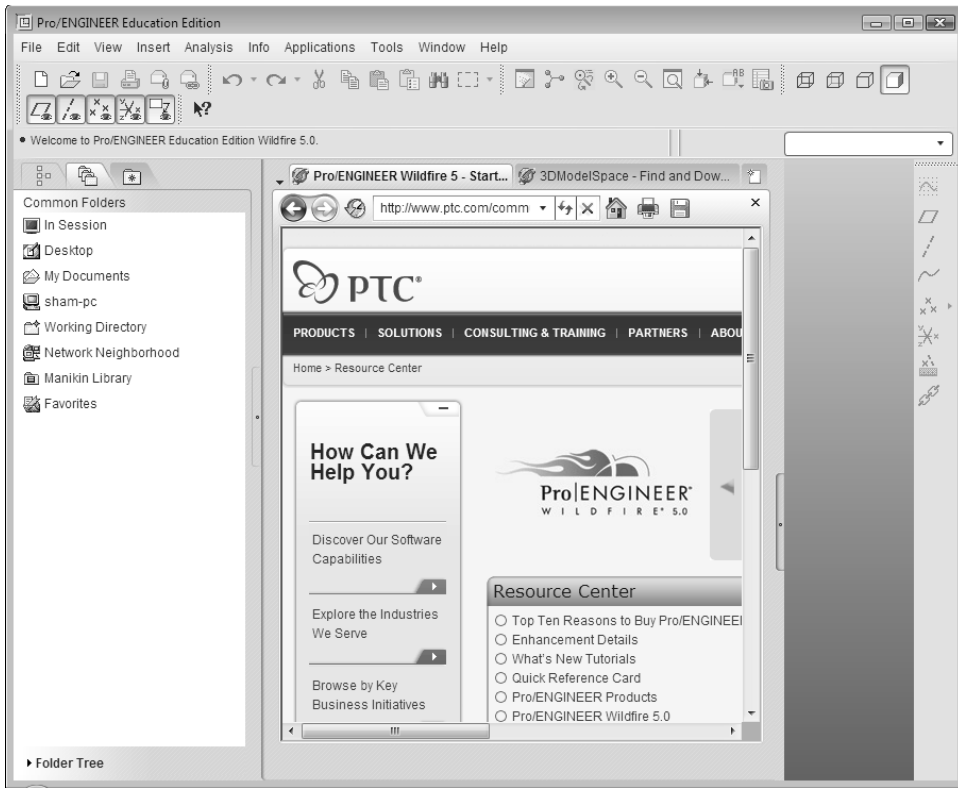
Once you have installed Pro/ENGINEER Wildfire 5.0 on your system, there are two options to start it. The first option is to choose the **Start** button at the lower left corner of the screen and then choose **Programs > PTC > Pro ENGINEER > Pro ENGINEER**, as shown in Figure 1-7.



*Figure 1-7 Windows screen with the task bar and application icons*

The second option to start Pro/ENGINEER Wildfire 5.0 is by double-clicking on its shortcut icon on the desktop of the computer. Note that the icon will be created on the desktop only if the option for displaying the icon on the desktop is selected while installing the software.

Figure 1-8 shows the screen that appears when you start Pro/ENGINEER Wildfire 5.0.



*Figure 1-8 Initial screen appearance after starting Pro/ENGINEER Wildfire 5.0*

## IMPORTANT TERMS AND DEFINITIONS

Some important terms that will be used in this book while working with Pro/ENGINEER Wildfire 5.0 are discussed next.

### Entity

An element of section geometry is called an entity. The entity can be an arc, line, circle, point, conic, coordinate system, and so on. When one entity is divided at a point, then the total number of entities is said to be two.

### Dimension

It is the measurement of one or more entities.

### Constraint

Constraints are logical operations that are performed on the selected geometry to make it more accurate in defining its position and size with respect to the other geometry.

### Parameter

It is defined as a numeric value or any definition that defines a feature. For example, all dimensions in a sketch are parameters. The parameters can be modified at any time.



## Relation

A relation is an equation that relates two entities.

## Weak Dimensions and Weak Constraints

Weak dimensions and weak constraints are temporary dimensions or constraints that appear in gray color. These are automatically applied to the sketch when it is drawn using the **Intent Manager**. They are removed from the sketch without any confirmation from the user. The weak dimensions or the weak constraints should be changed to strong dimensions or constraints if they seem to be useful for the sketch. This only saves an extra step of dimensioning the sketch or applying constraints to it.

## Strong Dimensions and Strong Constraints

Strong dimensions and strong constraints appear in white color. These dimensions and constraints are not removed automatically. All dimensions added manually to a sketch are strong dimensions.



**Tip:** When several strong dimensions or constraints conflict, Pro/ENGINEER makes the constraints and dimensions appear in red color, and prompts you to delete one or more of them.

## FILE MENU OPTIONS

The options that are displayed when you choose **File** from the menu bar are discussed next.

### Set Working Directory

A working directory is a directory on your system where you can save the work done in the current session of Pro/ENGINEER Wildfire 5.0. You can set any directory existing on your system as the working directory. Before starting the work in Pro/ENGINEER Wildfire 5.0, it is important to specify the working directory. If the working directory is not selected before saving an object file, then the object file will be saved in a default directory. This default directory is set at the time of installing Pro/ENGINEER Wildfire 5.0. If the working directory is selected before saving the object files that you create, it becomes easy to organize them. In Pro/ENGINEER Wildfire 5.0, the working directory can be set in two ways:

### Using the Navigator

When you start a Pro/ENGINEER Wildfire 5.0 session, the navigator is displayed on the left of the drawing area. This navigator can be used to select a folder and set it as the working directory.

Right-click on the folder that you need to set as the working directory; a shortcut menu will be displayed, as shown in Figure 1-9. Choose the **Set Working Directory** option from this shortcut menu to set the selected folder as the working directory. To make a new folder, choose the **New Folder** option from the shortcut menu.

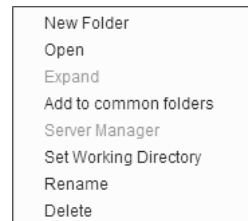
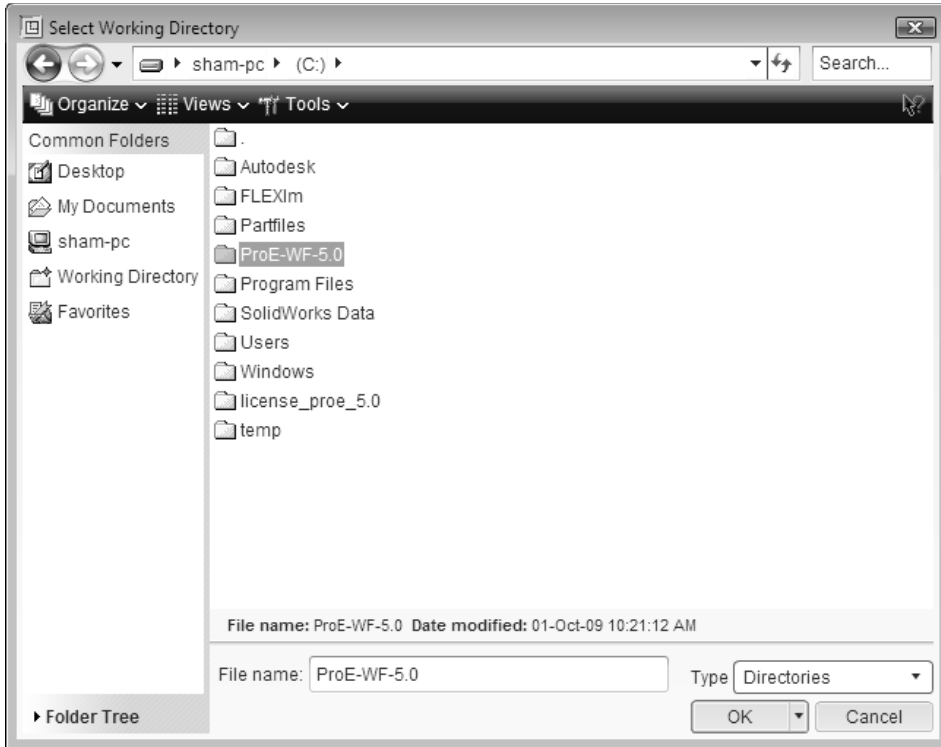


Figure 1-9 Shortcut menu

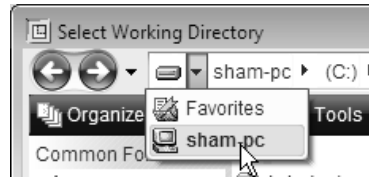
## Using the Select Working Directory Dialog Box

To specify a working directory, choose **File > Set Working Directory** from the menu bar; the **Select Working Directory** dialog box will be displayed, as shown in Figure 1-10. Using this dialog box, you can set any directory as the working directory.



*Figure 1-10 The Select Working Directory dialog box*

Choose the arrow at the upper left corner of the dialog box, as shown in Figure 1-11; the flyout will be displayed. This flyout displays some of the drives present on your computer along with the **Favorites** folder, as shown in Figure 1-11. The **Favorites** folder contains all directories that you saved as favorites. The procedure to save the favorite directories will be discussed later. When the **Select Working Directory** dialog box is invoked by default, it displays the contents of the default directory. However, you can change the default directory that appears every time you open this dialog box. Various options in the **Select Working Directory** dialog box are discussed next.



*Figure 1-11 The flyout with some drives*



**Tip:** The **Select Working Directory** dialog box has some of the properties of the Microsoft Windows operating system. You can set the working directory using this dialog box by browsing through directories and folders. You can also rename a file, directory, or a folders name in this dialog box. You can create a new directory using this dialog box.

**File name**

The **File name** edit box displays the name of the directory selected in the **Select Working Directory** dialog box. You can select a directory using the flyout as discussed earlier or enter the path of any existing directory in this edit box.

**Type**

The **Type** drop-down list has two options, **Directories** and **All Files**. If you select the **Directories** option, all directories present are listed, and if you select the **All Files** option, then all files along with the directories are listed in the dialog box.

**Organize**

When you choose the **Organize** button from the **Select Working Directory** dialog box, a flyout will be displayed. The options in this flyout are used to create a new directory or rename an existing directory. You can also cut, copy, paste, and delete the existing folders using the options in the flyout. Moreover, you can add any existing folder in the **Common Folders** by using the **Add to common folders** option in this flyout, refer to Figure 1-12.

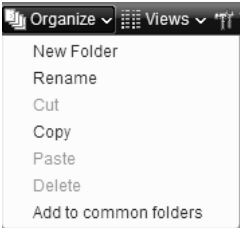


Figure 1-12 The **Organize** button flyout

**Views**

When you choose the **Views** button from the **Select Working Directory** dialog box, a flyout will be displayed. The options in this flyout are discussed next.

**List.** The **List** option is used to view the contents of the current folder or drive. These include files and folders in the form of a list.

**Details.** The **Details** option is used to view the contents of the current folder or drive in the form of a table, which displayed the name, size, and date on which it was last modified.

**Tools**

When you choose the **Tools** button from the **Select Working Directory** dialog box, a flyout will be displayed, as shown in Figure 1-13. The options in this flyout are discussed next.

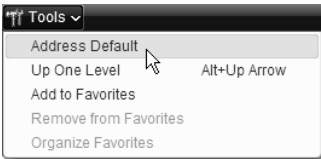
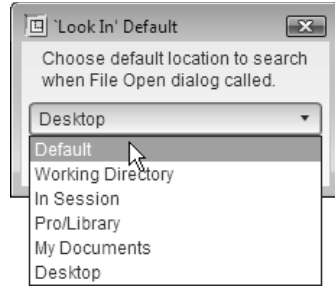


Figure 1-13 The **Tools** flyout

**Address Default.** When you select this option, the '**Look In**' **Default** dialog box will be displayed. Figure 1-14 shows this dialog box with the options in the drop-down list. If you select the **Default** option from the drop-down list and then invoke the **File Open** dialog box, it will display the directory that is set as default. If you select the **Working Directory** option from the drop-down list and then invoke the **File Open** dialog box, it will display the working directory that is set. If you select the **In Session** option and then invoke the **File Open** dialog box, the **File Open** dialog box will open with the **In Session** folders selected by default. Similarly, you can set the **Pro/Library** as the working directory.



*Figure 1-14 The '**Look In**' **Default** dialog box with options in the drop-down list*

**Up One Level.** The **Up One Level** option allows you to move one level up in the directory. Choose this option; a directory that is one level above the current directory will be displayed. Alternatively, press ALT+Up arrow keys to move one level up. You can also choose the arrow button on the left of the required directory in the address bar to display all folders in it.

**Add to Favorites.** The **Add to Favorites** option allows you to add the folders in the **Favorites** folder.

**Remove from Favorites.** The **Remove from Favorites** option allows you to remove the folders from the **Favorites** folder. This option is not enabled by default. To enable this option, you first need to add the folder in the **Favorites** folder using the **Add to Favorites** option.

**Sort By.** In the **Select Working Directory** dialog box, the **Directories** option in the **Type** drop-down list is displayed by default. From this drop-down list, if you select the **All Files** option and then choose the **Tools** button; a flyout will be displayed with the **Sort By** option.

The **Sort By** option is used to list all files in the directory in an order to facilitate the process of searching a file. When you choose the **Sort By** option, a cascading menu is displayed. In the cascading menu, there are two options, **Model Name** and **Markup/Instance Name**. If you choose the **Model Name** option, the file list will be sorted out alphabetically by the model name in the **Select Working Directory** dialog box. The **Markup/Instance Name** option sorts out the file list by specific markups or instance names in the **Select Working Directory** dialog box.

### Common Folders and Folder Tree

The **Common Folders** and **Folder Tree** tabs are available on the left of the **Select Working Directory** dialog box. The **Common Folders** contains folders such as **Desktop**, **My Documents**, **Computer**, **Working Directory**, and **Favorites**. You can add more folders in the **Common Folders** by using the **Add to common folders** option available in the

options of the **Organize** button. The **Folder Tree** contains all the drives available on your computer along with their contents. You can also set the working directory by using the **Folder Tree**. By default, the **Folder Tree** is in the collapsed state. To expand it, you need to click on the arrow that is available on the left of the **Folder Tree**. The **Working Directory** and **Favorites** folders available in the **Common Folders** are discussed next.

**Working Directory.** This folder is used when you have already set the working directory. You may browse through the directories in the **Select Working Directory** dialog box, but when you choose this folder, the directory selected previously as the working directory is displayed in the list box.

**Favorites.** This folder is used to save the location of the directories that are to be used frequently. You just need to specify the working directory to be used frequently and save its location by selecting the **Favorites** folder.

If you want to select one of the favorite working directories, then select the **Favorites** folder from the **Common Folders**; the list of all directories that were saved as favorites will be displayed in the list box. Select the required favorite directory and choose **OK**; the selected favorite directory will be set as the current working directory.



#### Note

*An object in Pro/ENGINEER Wildfire 5.0 is defined as a file that is created using any of its modes such as **Part**, **Drawing**, **Sketch**, and so on.*

## New



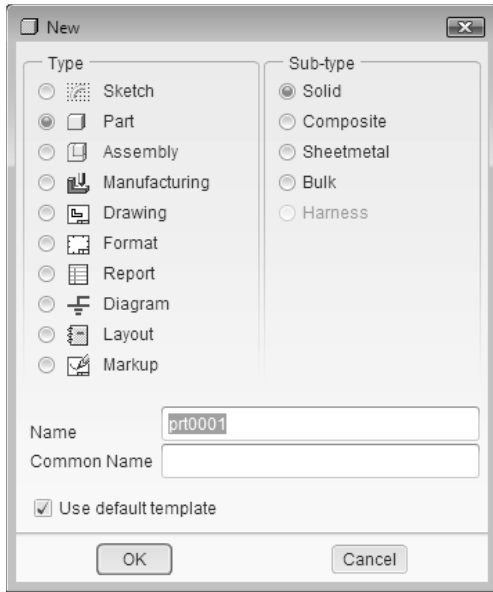
To create a new object, choose **New** from the **File** menu or choose the **New** button from the **Top Toolchest**; the **New** dialog box will be displayed, as shown in Figure 1-15.

This dialog box displays various modes available in Pro/ENGINEER Wildfire 5.0. In this dialog box, by default, the **Part** mode radio button is selected and a default name of the object file is displayed in the **Name** edit box. You can also enter a as desired new name for the object file.

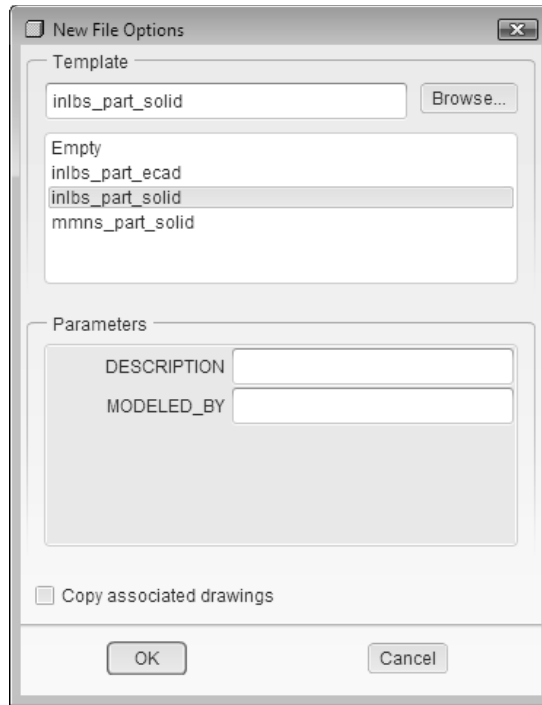
When you select the **Part**, **Assembly**, or **Manufacturing** radio button in this dialog box, the subtypes of the respective modes will be displayed under the **Sub-type** area of this dialog box.

Accept the default settings in the **New** dialog box by choosing the **OK** button; the **New File Options** dialog box will be displayed, as shown in Figure 1-16. Using this dialog box, you can select the predefined templates in Pro/ENGINEER Wildfire 5.0 or create a user-defined template. You can also open an empty template provided in the **New File Options** dialog box. In this case, you need to create the datum planes and the coordinate systems manually.

If, the measuring units for creating models is inches. Select **inlbs\_part\_solid** from the list in the **Template** area and then choose **OK** from the **New File Options** dialog box. On doing so, the three default datum planes and a coordinate system will be displayed in the drawing area. Also, the **Model Tree** will appear on the left of the screen, as shown in Figure 1-17.



*Figure 1-15 The New dialog box*



*Figure 1-16 The New File Options dialog box*

## Open



The **Open** button is used to open an existing object file. When you choose the **Open** option from the **File** menu or choose the **Open** button from the **File** toolbar in the **Top Toolchest**, the **File Open** dialog box will be displayed, as shown in Figure 1-18. The selected working directory will be displayed in it. Note that the **Preview** area is not displayed by default. To view the **Preview** area, choose the **Preview** button. Most of the options in this dialog box are the same as discussed in the **Select Working Directory** dialog box. The rest of the options in this dialog box are discussed next.

## Tools

When you choose this button, a flyout will be displayed. The options available in this flyout are the same as the options discussed in the **Tools** button of the **Select Working Directory** dialog box, except the **All Versions** and **Show Instances** options. These two options are discussed next.

### All Versions

This option, when selected, displays all versions of an object file. In Pro/ENGINEER Wildfire 5.0, the file once saved will generate a new version of it with an extension 1. An object file is not copied on another object file but a new version of it is created. Therefore, every time you save an object using the **Save** option, a new version of it is created on the disk in the current working directory.

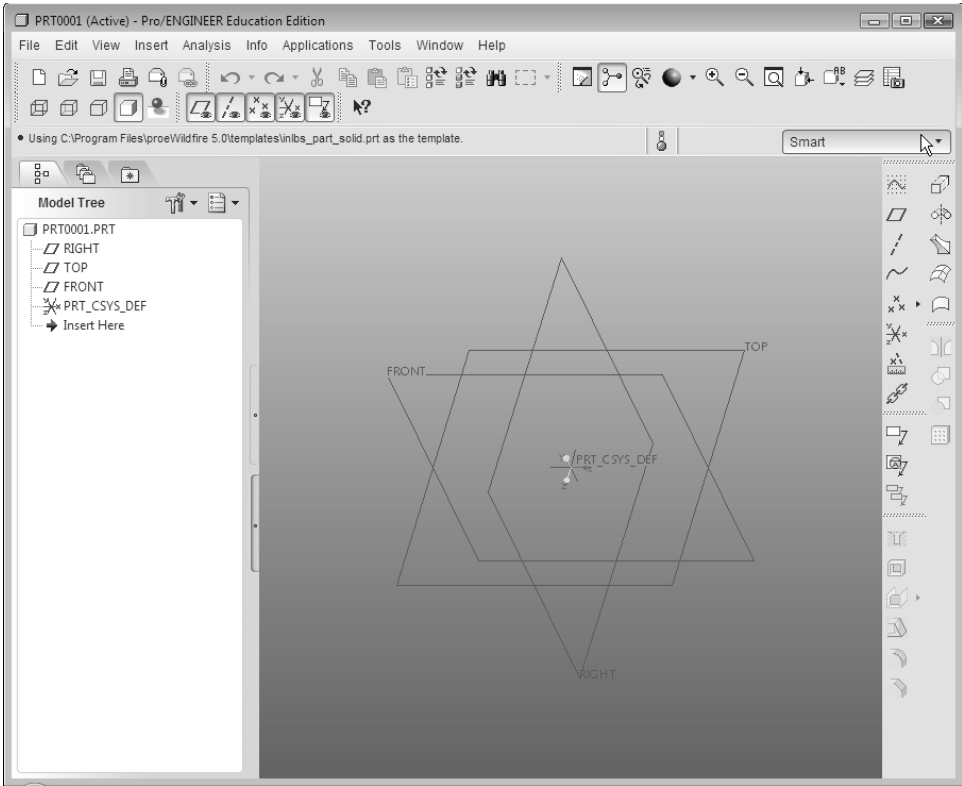


Figure 1-17 The initial screen appearance after entering the **Part** mode

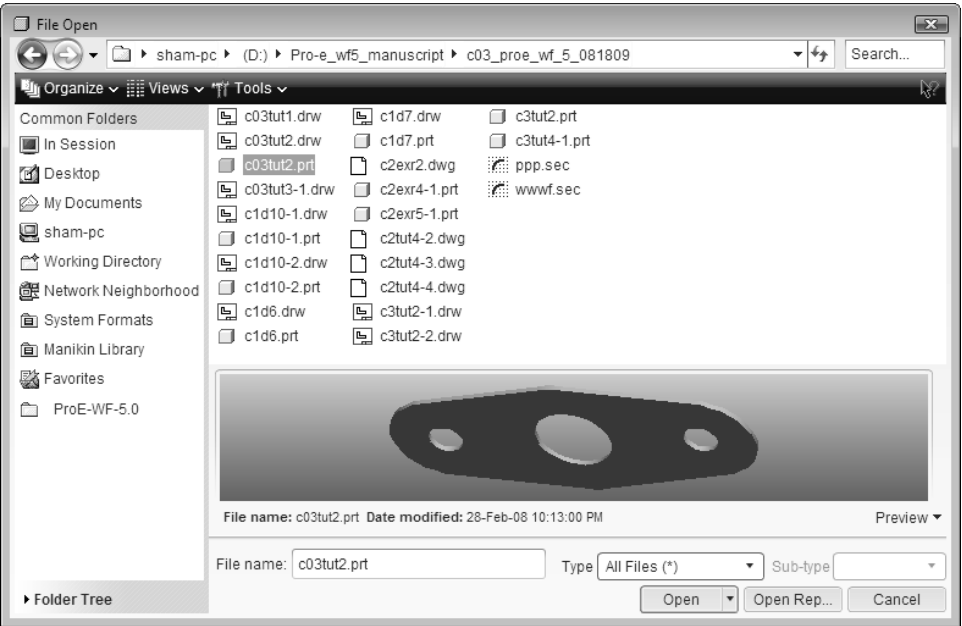


Figure 1-18 The **File Open** dialog box with the **Preview** area

### Show Instances

The **Show Instances** option, when selected, displays all instances of the object file. Select the required file and then select the **Show Instances** option from the **Tools** button; all the instance of the selected file will be displayed.

### File name

In the **File name** edit box, you can enter the name of the existing object file that you want to open.

### Type

The **Type** drop-down list contains the file formats of various modes available in Pro/ENGINEER Wildfire 5.0. Also, it contains other file formats that can be imported in Pro/ENGINEER Wildfire 5.0. These file formats include IGES, SET, STEP, DWG/DWF, Medusa, Inventor, Parasolid, Rhino, and so on. By default, the **Pro/ENGINEER Files** option is selected in this drop-down list. As a result, you can open the files created in any mode of Pro/ENGINEER. However, if you select a specific mode from this drop-down list, only the files of the corresponding mode will be displayed. For example, if you select **Part** in the drop-down list, then only the .prt files will be displayed. This makes the selection and opening of the files easy.

### Preview

The **Preview** button is used to preview the model to be opened. By choosing this button, you can preview the selected model in the **File Open** dialog box. This feature of the **File Open** dialog box helps in previewing the model before actually opening it. You can zoom, pan and rotate the model in the preview. Also, you can change the appearance (shaded, wireframe, no hidden, and hidden line) of the model, switch the model between the orthographic and perspective views, change the orientation type (dynamic, anchored, delayed, velocity, fly through, and standard) of the model, set the number of frames per second, and refit the preview in the preview screen.



#### Note

*Assembly files with the file extension .asm can also be previewed by using the **Preview** button. If you are not able to see the preview of the assembly files in the preview area, then choose the **Refresh** button on the upper right of the **File Open** dialog box to resize the assembly according to the preview area.*

*There is no Command prompt in Pro/ENGINEER Wildfire 5.0. However, you are provided with prompts in the message area. Whenever you have to enter any numerical value or text, a **Message Input Window** will be displayed in the message area.*

### In Session

The **In Session** folder is available in the **Common Folders**, on the upper left of the **File Open** dialog box. When you choose the **In Session** folder, all the object files that are in the current session will be displayed in the display area. The object files that you open in Pro/ENGINEER Wildfire 5.0 in the current session are stored in its temporary memory. This temporary memory is stored in a folder named **In Session**. Once you exit Pro/ENGINEER Wildfire 5.0, the contents of this folder are deleted automatically. However, the original files are not removed from their actual location.



## Erase

The **Erase** option is used to delete the files from the temporary memory stored in the **In Session** folder. To invoke this option, choose **File > Erase** from the menu bar; a cascading menu will be displayed with three options, **Current**, **Not Displayed**, and **Component Reps**. As discussed earlier, all files opened in a session of Pro/ENGINEER Wildfire 5.0 are saved in the temporary memory. You can use the **Erase** option to erase files from the temporary memory. The options that are displayed in the cascading menu are discussed next.

### Current

The **Current** option is used to erase the file opened and displayed in the drawing area. On choosing this option, the **Erase Confirm** message box will be displayed, prompting you to confirm the erasing of the file, as shown in Figure 1-19.

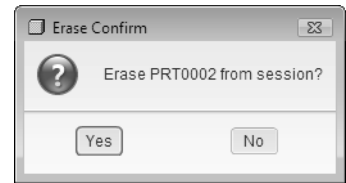


Figure 1-19 The Erase Confirm message box

### Not Displayed

This option is used to erase the files that are in the **In Session** folder, but not displayed presently on the screen. On choosing this option, the **Erase Not Displayed** dialog box will be displayed, as shown in Figure 1-20. You can select the files from this dialog box to erase them.

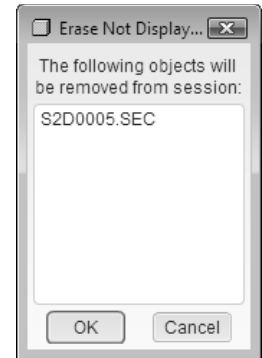


Figure 1-20 The Erase Not Displayed dialog box

### Component Reps

This option removes the unused simplified representations from the **In Session** folder. When you choose this option from the **File** menu, a message will be displayed with the message, All the objects which were not displayed have been erased.

### Delete

This option removes the selected file permanently from the hard disk. To invoke this option, choose **File > Delete** from the menu bar. The cascading menu is displayed with two options, **Old Versions** and **All Versions**.



**Tip:** Suppose you open an assembly that has a component named Nut. Close the assembly and now open another assembly that has a component named Nut. Now, there are chances that the second assembly you choose to open may open with the Nut that was present in the previous assembly. This is because the component with the file named Nut was already present in the memory of Pro/ENGINEER (in session) before opening the second assembly.

To avoid this error of assemblies, you should erase the files in the current session of Pro/ENGINEER before opening the second assembly.

## Old Versions

This option is used to delete all old versions of the current file. When you choose the **Old Versions** option, you are prompted to enter the name of the object file of which the old versions have to be deleted. When the **Message Input Window** is displayed, enter the object file name in this window. All versions of that file will be deleted from the hard disk except the latest version.

## All Versions

This option is used to delete all versions including the current file from the hard disk. When you choose the **All Versions** option, a warning is displayed stating that performing this function can result in the loss of data. This option is chosen when the file is opened and is displayed in the drawing area.

## Save



The **Save** option is used to save the objects present in the **In Session** folder or an object in the drawing area. When you choose the **Save** option from the **File** menu or the **Save** button on the **File** toolbar, the **Save Object** dialog box will be displayed. Also, the name of the current object will be displayed in the **Model Name** edit box. Choose the **OK** button from the **Save Object** dialog box to save the object.



### Note

*Remember that all object files saved using the **Save** option in the **File** menu should be present in the **In Session** folder. Otherwise, Pro/ENGINEER Wildfire 5.0 will display an error informing you that the file does not exist in the current session.*

*It should be noted here that the file opened currently on the screen may not be in the **In Session** folder, but in the current session. So, the file will be saved without any error.*

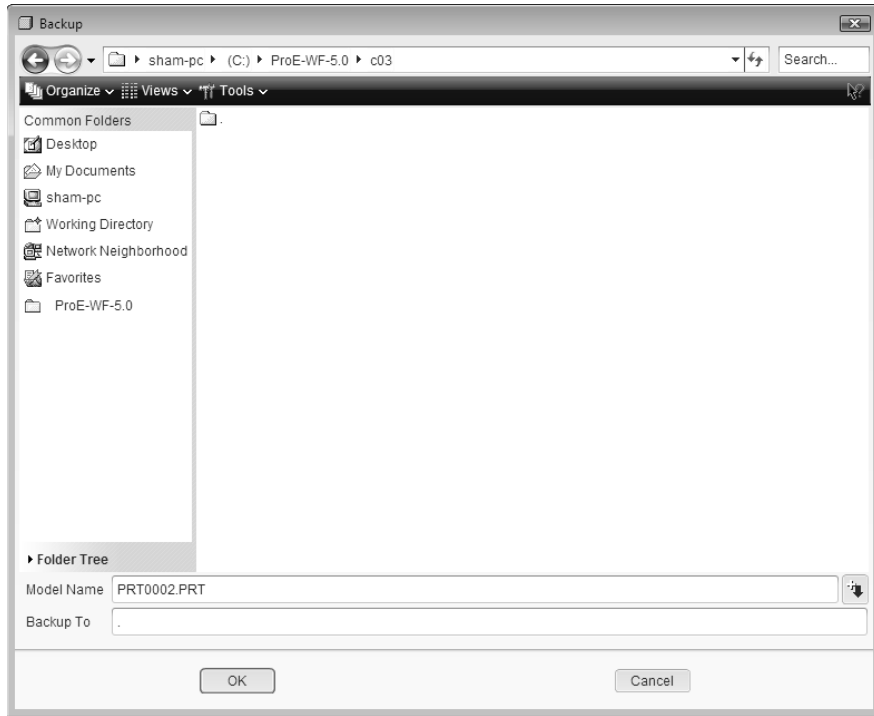
## Save a Copy

The **Save a Copy** option is used to save a copy of the current object in the same working directory or in some other directory. When you choose this option from the **File** menu, the **Save a Copy** dialog box will be displayed and you need to specify the new name of the object file to be saved as a copy and the name of the target directory in the **Save a Copy** dialog box. You can browse through the directories and select the target directory. The file will be saved in the selected directory.

Using this option, you can also export a file in other file formats such as Inventor file, pdf, ACIS, Wavefront file, and so on. After specifying the name of the new file and the target directory, choose the file format in which you want to export the file from the **Type** drop-down list in the **Save a Copy** dialog box.

## Backup

The **Backup** option of the **File** menu is used to create a backup copy of an object file in memory. When you choose this option, the **Backup** dialog box will be displayed, as shown in Figure 1-21.

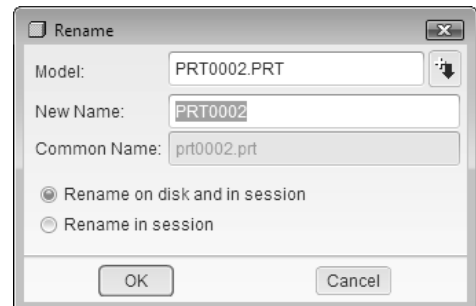


*Figure 1-21 The Backup dialog box*

In the **Model Name** edit box, the name of the file for which you want to take a backup is displayed. In the **Backup To** edit box, the name of the directory is specified where the object will be saved as a backup. If you create the backup of an assembly or a drawing object, Pro/ENGINEER Wildfire 5.0 will save all its dependent files in the specified directory. For example, if you create the backup of a drawing file in a different directory, then the part file of the drawing will also be created automatically in the same directory where backup of the drawing file is created.

## Rename

The **Rename** option in the **File** menu is used to rename the currently active object on the screen. To rename an object, choose this option; the **Rename** dialog box will be displayed, as shown in Figure 1-22. Specify the new name of the object in the **New Name** edit box and choose **OK**.



*Figure 1-22 The Rename dialog box*

## MANAGING FILES

As discussed earlier, a new file is generated whenever you save an object. The number of files generated is directly proportional to the number of times you save that object. So, these files occupy a lot of disk space. The latest version of the object is of use and should be stored. Latest version refers to the highest number that is suffixed with the file name of that object. The rest of the files called old versions should be deleted from the hard disk, if they are not required.



### Note

*To save disk space, you should keep deleting the old versions of a file. This is done using the **Delete** option in the **File** menu.*

## MENU MANAGER

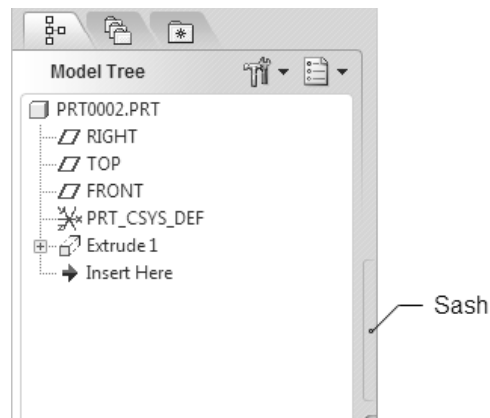
The **Menu Manager** contains a set of menus and submenus cascaded in it. It is displayed for some particular tasks only. For example, if you choose **Edit > Feature Operations** from the menu bar, the **Menu Manager** will be displayed with options to reorder, copy, move, or mirror features. Note that the display of the menus depends on the task chosen.

While using the **Menu Manager**, always complete the option selected by choosing **Done** or **Done Sel** after the current task is over. This is important when you are in the **Drawing** mode of Pro/ENGINEER. If you are directly selecting one option after another, then it is easy to lose track of commands or options in the **Menu Manager**.

## MODEL TREE

The **Model Tree** stores and displays all features in a chronicle. You can select any desired feature of a model or an assembly from the **Model Tree** and apply different operations on the selected feature. You can also select the feature by right-clicking on it; a shortcut menu will be displayed. Move the cursor on the shortcut menu and choose the required option from it by using the left mouse button.

When you create a new object file, the **Model Tree** appears and is attached to the drawing area by default. But in this case, the **Model Tree** is attached to the drawing area, and so the drawing area becomes small. You can hide the **Model Tree** by clicking the sash on the right edge of the navigator, as shown in Figure 1-23. The **Model Tree** can slide in or slide out, thus increasing or decreasing the area on the drawing area. It can also be stretched horizontally to cover the drawing area.



**Figure 1-23** Partial view of the **Model Tree**



**Tip:** By looking at the **Model Tree**, you can understand the method and approach used to create the model. Using the **Model Tree**, you can modify the features of a model. Generally, when you import a model from a different file format in Pro/ENGINEER, you do not get the features of the model in the **Model Tree** and therefore, you will not be able to modify it.



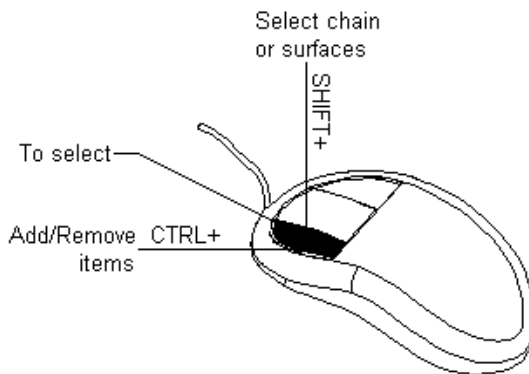
### Note

In Pro/ENGINEER, most of the features are created using the **Dashboard**. The **Dashboard** is displayed below the message area and it contains all options to complete the selected operation on the model.

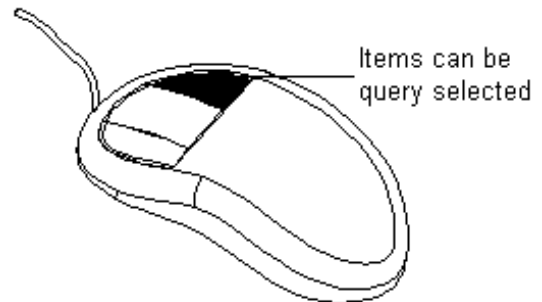
## UNDERSTANDING THE FUNCTIONS OF THE MOUSE BUTTONS

While working with Pro/ENGINEER Wildfire 5.0, it is important to understand the function of the three buttons of the mouse to make efficient use of this device. The various combinations of the keys and three buttons of the mouse are listed below:

1. Figure 1-24 shows the functions of the left mouse button. The left mouse button is used to make a selection. Using CTRL+left mouse button, you can add or remove items from the selection set.
2. Figure 1-25 shows the functions of the right mouse button. The right mouse button is used to invoke the shortcut menus and to query select the items. When you bring the cursor on an item, it is highlighted in cyan color. Now if you hold down the right mouse button, a shortcut menu is displayed. Choose the **Pick From List** option from the shortcut menu; the **Pick From List** dialog box will be displayed. You can make selections from this dialog box.



**Figure 1-24** Functions of the left mouse button



**Figure 1-25** Functions of the right mouse button

3. Figure 1-26 shows the functions of the middle mouse button in the 3D mode. The middle mouse button is used to spin the model in the drawing area and view it from different directions.

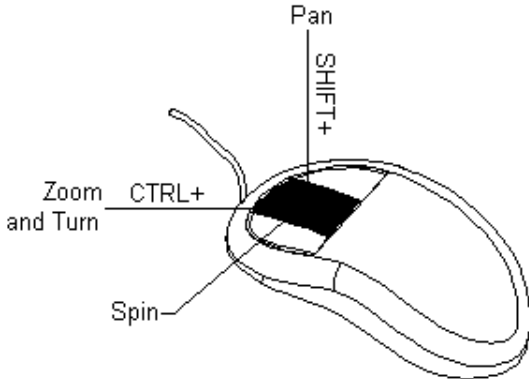
CTRL+middle mouse button is used to dynamically zoom in and out by moving the mouse. When you press and hold the CTRL key and move the mouse up, the view is reduced, and so you zoom out. When the mouse is moved down, the view is enlarged, and so you zoom in.

When you use CTRL+middle mouse button and move the mouse horizontally, the model is rotated about a point that you specified as center.

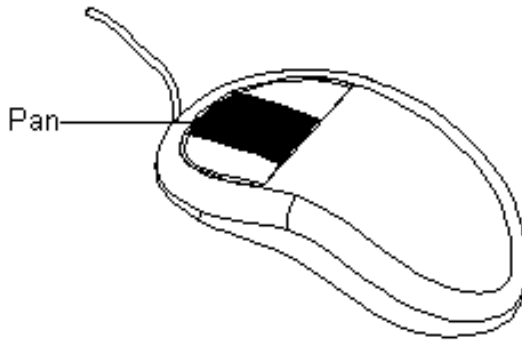
SHIFT+middle mouse button is used to pan the object on the screen.

4. Figure 1-27 shows the functions of the middle mouse button in the 2D mode (sketcher environment and **Drawing** mode). The middle mouse button is used to place dimensions in the drawing area. It is also used to confirm an option or to abort the creation of an entity.

The middle mouse button is used to pan in the **Sketch** mode and the **Drawing** mode.



**Figure 1-26** Functions of the middle mouse button in the 3D mode



**Figure 1-27** Functions of the middle mouse button in the **Sketch** mode



**Note**

When you spin the model with the **Spin Center** on, the model is rotated about the spin center. If the spin center is turned off, then the model is rotated about the cursor.

## TOOLBARS

Before starting work on Pro/ENGINEER Wildfire 5.0, it is very important for you to understand the default toolbars and buttons in the main window. Figure 1-28 shows various default screen components in Pro/ENGINEER Wildfire 5.0.

In Pro/ENGINEER Wildfire 5.0, the toolbars are at two locations, the **Top Toolchest** and the **Right Toolchest**. The area on the top of the drawing area, where the toolbars are present, is called the **Top Toolchest** and the area on the right of the drawing area, where the toolbars are present, is called the **Right Toolchest**. The toolbars that initially appear on the screen are shown in the Figure 1-28. You will notice that all toolbar buttons are not available. These buttons

are available after you open a new or an existing file. Only those buttons that are required for the current session are activated. As you proceed to enter one of the modes provided by Pro/ENGINEER Wildfire 5.0, you will notice that the toolbar buttons required by that mode are activated. To make the designing easy and user-friendly, this software package provides you with a number of toolbars. Different modes of Pro/ENGINEER Wildfire 5.0 display different toolbars. Some of the frequently used toolbars are shown in Figure 1-28.

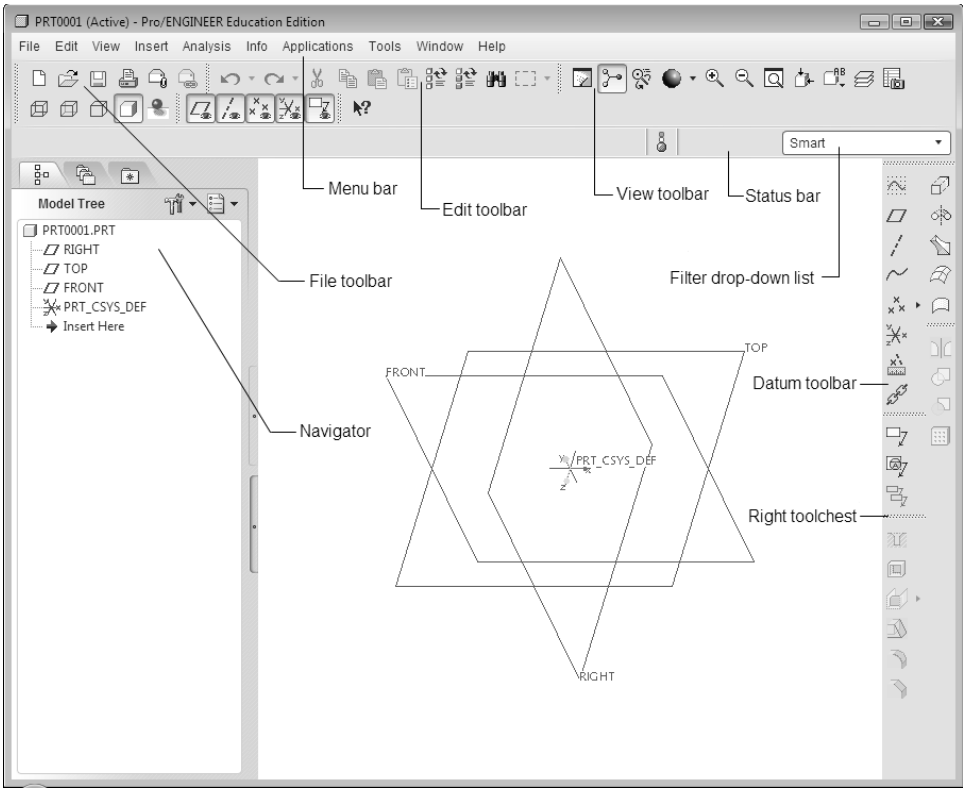


Figure 1-28 Items in the main window

Datum Display Toolbar

Figure 1-29 shows the **Datum Display** toolbar. It has five buttons that are used to control the visibility of datum features like the planes, axes, and points. You can also use this toolbar to control the visibility of the coordinate systems and annotation element.

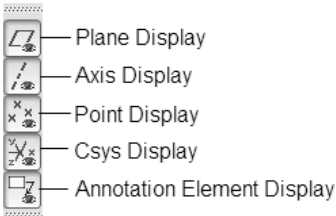


Figure 1-29 The Datum Display toolbar



**Tip:** To add buttons to the toolbar and add toolbars to the **Toolchest**, choose **Tools > Customize Screen** from the menu bar; the **Customize** dialog box is displayed. This dialog box is used to customize the main window of Pro/ENGINEER Wildfire 5.0.

Right-click on the **Right Toolchest** to invoke a shortcut menu that has all the names of the toolbar, which are available for the current session. Select any toolbar to display it in the selected **Right Toolchest**.

## File Toolbar

The **File** toolbar has six buttons, as shown in Figure 1-30. These buttons are used to save a file, print the current file, open a new file, or open an existing file. The functions of the **Mail Recipient (as Attachment)** and **Mail Recipient (as Link)** buttons of this toolbar are discussed next.

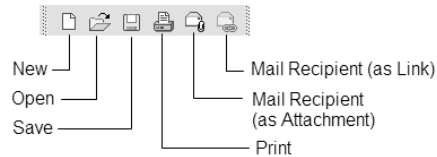


Figure 1-30 The File toolbar

The **Mail Recipient (as Attachment)** button is used to send an email with the current model attached with the mail. When you choose this button, the **Send As Attachment** dialog box will be displayed, as shown in Figure 1-31. Next, choose the **OK** button to send the attachment. You can zip an attachment by selecting the **Create a ZIP file containing the attachment** check box.

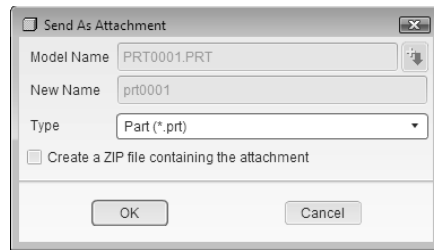


Figure 1-31 The Send As Attachment dialog box

The **Mail Recipient (as Link)** button is used to send an email with a link to the current model. This button is available to the users of Windchill.

## Help Button

This button, as shown in Figure 1-32, is also known as the **What's This?** button. When you choose this button, a question mark symbol (?) is attached to the cursor. Depending on the location in the main window where you click, the **Pro/ENGINEER Help** window with the help topics is displayed.



Figure 1-32 The Help button

## Model Display Toolbar

Figure 1-33 shows the **Model Display** toolbar. This toolbar has five buttons that are used to set the display mode for viewing a model. The **Wireframe** button is used to display the model as a wireframe model. The visible and the invisible edges of the model will be displayed in this mode. The **Hidden line** button is used to display the visible lines of the model in bright color and the hidden lines in dull

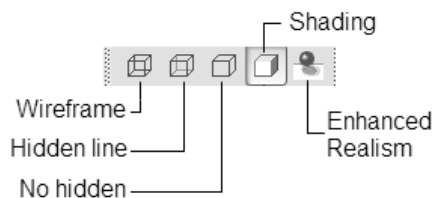


Figure 1-33 The Model Display toolbar



color. However, if you plot the model in this display mode, you will notice that the hidden lines are shown as dashed lines in the plot. The **No hidden** button is used to display only the visible edges of the model. When the **Shading** display mode is chosen, the model is displayed as shaded using a default color. However, you have the flexibility to alter the color of your model according to your needs. The last button available in this toolbar is **Enhanced Realism**. This is a toggle button, which enables or disables the realtime rendering as per requirement. The realtime rendering includes the effect of reflections, shadow, and light. You will learn about them later in the chapter.



### Note

The **Enhanced Realism** button will be available only when you are working on a 3d mode such as the **Part**, **Assembly** or **Manufacturing**.

## View Toolbar

The **View** toolbar, shown in Figure 1-34, has eleven buttons. The first button is **Repaint**, which is used for repainting the screen. This means you can remove any temporary information from the drawing area. The second button is **Spin Center**, which is used to toggle on/off the visibility of the spin center. The third button is **Orient Mode**. This button, when chosen, turns on or off the visibility of the datums and allows you to spin the model using the middle mouse button. The fourth button, **Appearance Gallery**, is newly added in this release and is used to set the appearances of the model. The fifth and sixth buttons are used to zoom in and zoom out the object, respectively. These buttons will be discussed in Chapter 2. The **Refit** button is used to fit the model on the screen. The **Reorient** button is used to orient the model. The next button is **Named View List**, is used to display the saved views. The **Layers** button displays the layers in the navigation area. The **View Manager** button is used to save, retrieve, or delete a view.

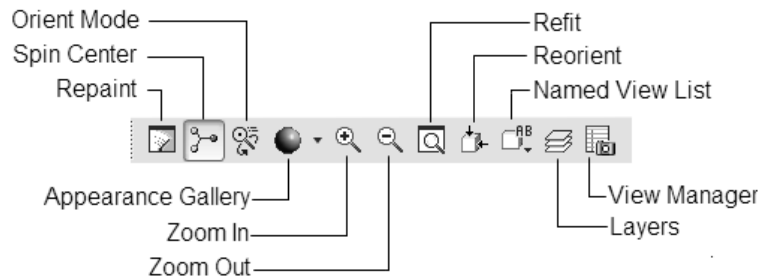


Figure 1-34 The View toolbar



### Note

If you want to change the predefined unit system, choose **File > Properties** from the menu bar to display the **Model Properties** dialog box. In this dialog box, click on the **change** link on the right of the **Units** option under the **Materials** head; the **Units Manager** dialog box will be displayed with the **System of Units** tab chosen. Next, select the desired unit system from the list box and then choose the **Set** button; the **Changing Model Units** message box will be displayed. Choose the **OK** button message box; the new unit system will be set and displayed with the red arrow on the left.

## NAVIGATOR

The navigator is present on the left of the drawing area and can slide in or out on the drawing area. To make the navigator slide in or out, you need to select the sash present on its right edge. A partial view of the navigator is shown in Figure 1-35. It has the following functions:

1. When you browse files using the navigator, the browser expands and the files in the selected folder are displayed in the browser.
2. When you open a model, the **Model Tree** is displayed in the navigation area.
3. The buttons on the top of the navigator are used to display the different items in the navigation area. The **Model Tree** button is used to display the **Model Tree** in the navigation area. This button is available only when a model is opened. The **Folder Browser** button is used to display the folders that are in the local system. The **Favorites** button is used to display the contents of the **Personal Favorites** folder.
4. Any other locations, if available on your system, can also be accessed by using the navigator.

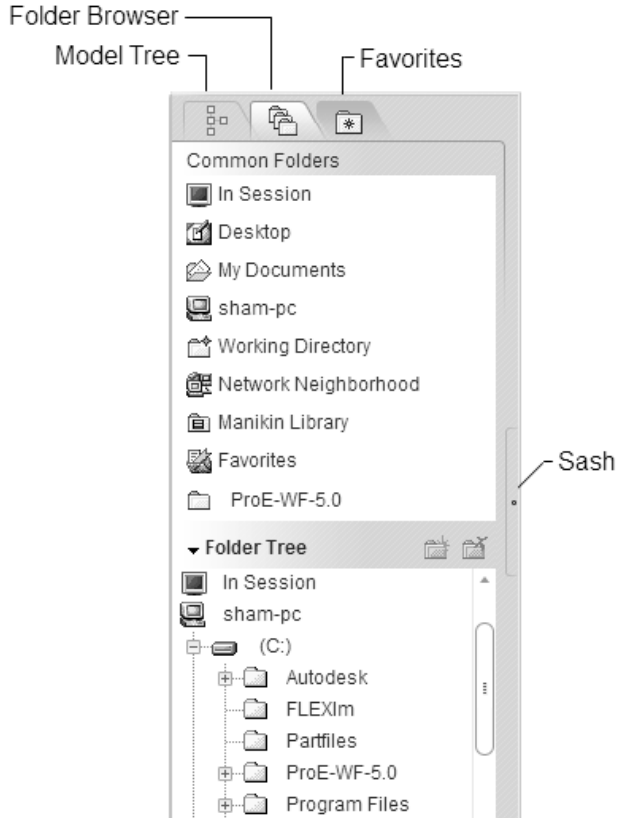


Figure 1-35 Partial view of the navigator

## BROWSER

The Browser is present on the right of the navigator. It can slide in and out of the navigator and can also be stretched horizontally. When the Pro/ENGINEER Wildfire 5.0 session starts, the browser is displayed on the screen. The browser has two tabs. The homepage is linked to *ptc.com*, by default, and the other tab is linked to the PTC's online resources. You can add more tabs in it, switch between tabs, and view the browsing history similar to any other internet explorer. The main advantage of using this browser is that you can work on the Pro/ENGINEER files and navigate through the browser, simultaneously. Figure 1-36 shows the browser with some part files displayed in it. You can also browse to the required folder, change the display of files to thumbnails for previewing the file, dynamically view the models in a pop-up window, or drag and drop the selected file into the Pro/ENGINEER window,





*Figure 1-36 Pro/ENGINEER Wildfire 5.0 browser*

The functions of the browser are listed next.

1. It is used to preview the Pro/ENGINEER Wildfire 5.0 files and browse the file system.
2. A Pro/ENGINEER file can be opened by using the browser by double-clicking on the file or by dragging it to the drawing area. When you open a file using the browser, the model is displayed on the screen and the browser is closed automatically.
3. You can access the Pro/ENGINEER user community site.
4. You can connect to your client's computer and jointly work with them using the Browser.

## APPEARANCE GALLERY

A new library of appearances and real world material have been added in Pro/ENGINEER Wildfire 5.0. You can also add shadows and reflections to the background. This is useful while rendering with a background image. This gives a realistic view of the model.

The **Appearance Gallery** button is available in the **View** toolbar of the **Top Toolchest**. It is used to assign different colors to the model and change its appearances. To change the appearance of the model, click on the down arrow beside the **Appearance Gallery** button;



a palette will be displayed, as shown in Figure 1-37. Select the required appearance from the palette; the **Select** box will be displayed. Select the entities whose appearance you want to change; the appearance of the entities will change. You can assign appearance to the entire model or to the individual face of the model, or entire assembly or individual component of an assembly. Various areas and options in the palette are discussed next.

## Search

This option is used for searching any specific appearance you need. To search an appearance, type a keyword related to the appearance to be searched in the **Search** text box and press ENTER; all the appearances matching with the keyword will be searched from all the system libraries and then displayed in the palette. For example, if you type **glass** in the text box, then **ptc-glass** will be displayed in the **My Appearances** area and the **Glass** library is displayed in the **Library** area.

## View Options

You can set the display of the appearance icons in the palette by using the **View Options** flyout. These options will be available when you click on the button at the top-right corner of the palette. The icons can be displayed as small thumbnails, large thumbnails, both names and thumbnails, or only names. To set the display of icons, invoke the **View Options** flyout and then choose the required option from it. Choose **Rendered Samples** from this to view the rendered icons of appearances. In this flyout, the **Show Tooltips** option is chosen by default. As a result, tooltips are always displayed for appearances.

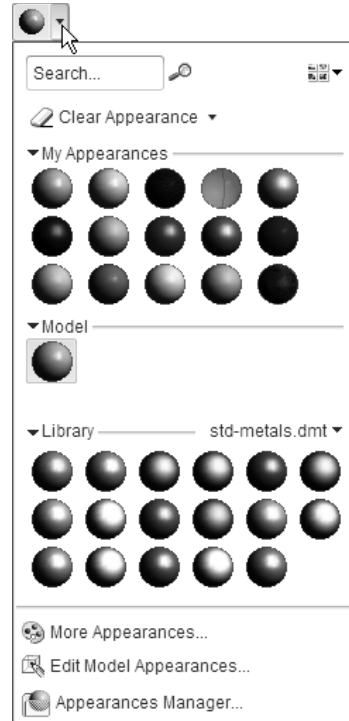
## Clear Appearance

This option is used to remove the selected appearance from the model. To do so, select the appearance and then choose this option. Alternatively, first choose this option and then select the appearance to be removed from the model.

To remove all appearances assigned to a model, click on the black arrow displayed besides the **Clear Appearance** button; a flyout will be displayed. Next, choose the **Clear All Appearances** option from this flyout.

## My Appearance

The **My Appearance** area displays all appearances stored in the startup directory. You can also add new appearances to the palette, edit the existing appearance, or delete the selected appearance in the **Appearance** area. To do so, right-click on an appearance icon and then choose the required option from the shortcut menu displayed. Note that you cannot delete the active appearance.



*Figure 1-37 Appearance Gallery*

## Model

The **Model** area consists of the appearances stored and used in an active model. You can edit the selected appearance, add new appearances, or select the objects to apply the selected appearance by using the shortcut menu as discussed in the **My Appearance** area.

## Library

This area displays the predefined appearances in the current library. The library contains appearances from the System library and the Photolux library. Appearances in the library are grouped in various classes such as metals, glass, wood, and so on, based on their similar characteristics. The name of the current library is displayed on the right of the **Library** head. To assign appearances from other libraries, click on the library name; a flyout will be displayed with a tree list of classes and libraries. Browse through the tree and select the required library; all appearances in the selected library will be listed. From the list displayed, you can assign an appearance to the model as well as create a new appearance, as discussed earlier.

## More Appearances

Choose the **More Appearances** button; the **Appearance Editor** will be displayed, as shown in Figure 1-38. The **Appearance Editor** provides options for editing the appearance and advanced rendering, thereby providing much more realistic images.

Using the **Appearance Editor**, you can edit the name, description, and keywords of the selected appearance, except the default one. The advanced options are given under two tabs, **Basic** and **Map**. The **Basic** tab provides with various classes and sub-classes of appearances. By default, **Generic** is selected in the **Class** drop-down list. However, you can select the required class and their sub-classes. Also, the properties related to selected class such as color, illumination, reflection, transparency, glossiness, and so on are displayed in the **Properties** area. These properties can be modified by adjusting the sliders given next to them.

The **Map** tab allows you to define and apply texture such as wood, fabrics, metals, and so on to the model. You can preview the changes made to the model on changing the settings in the **Appearance Editor**.

## Edit Model Appearances

Choose this button; the **Model Appearance Editor** will be displayed. In this editor, all appearances used in the model will be displayed with their properties. The **Model Appearance Editor** is the same as the **Appearances Editor**. Using this editor, you can modify the properties such as color, texture, shininess, and glossiness.

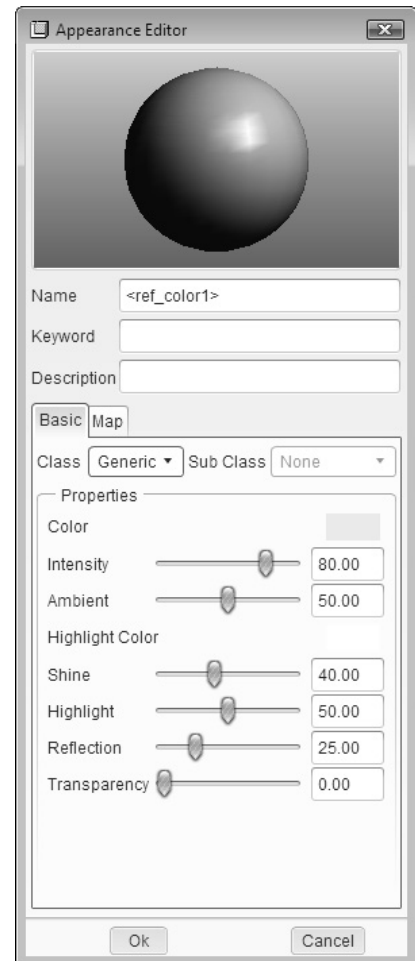


Figure 1-38 The Appearance Editor

## Appearances Manager

Choose this button to display the **Appearances Manager** window. The **Appearances Manager** window displays two panels. The left panel displays the palette with appearances and libraries and the right panel displays the editor with the properties of the active appearance. You can set the required settings in the **Appearances** palette and the **Appearances Editor**. Also, you can open an existing library file, add and save your own textured file.

## RENDERING IN Pro/ENGINEER Wildfire 5.0



Rendering is a process of generating two-dimensional image of a three-dimensional scene or an object to make it more realistic. A rendered image makes it easier for visualizing the shape and size of 3D object as compared to a wireframe or a shaded image. Rendering also helps you express your design intent to other people. You need to alter environments, lights, textures, and so on to get a high quality rendered image.

Generally, the rendering process includes the following stages:

### Loading a File

The first step is to load a solid model or an assembly to be rendered.

### Applying Appearances

Next step is to apply the appearances to the model. You can do so by using the **Appearance Gallery**, which has already been discussed.

### Applying Scenes

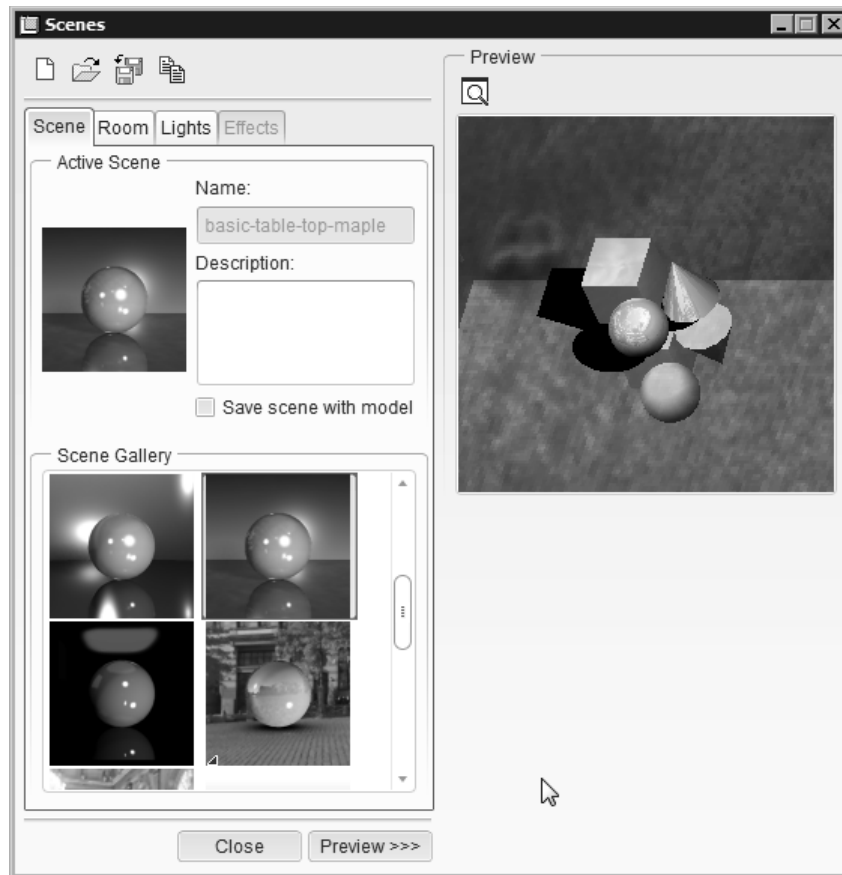
After applying the appearances, you need to apply scenes to the model. Scenes are the predefined render settings that include lights, rooms, and environmental effects. To set scenes, choose **View > Model Setup > Scene** from the menu bar; the **Scenes** window will be displayed, as shown in Figure 1-39. Using this window, you can apply the predefined settings, edit the current settings, create new scenes, copy a scene, and save the scenes for future use. To apply a scene to the model, double-click on the required scene from the **Scene Gallery** area of the **Scenes** window.

### Creating the Rendering Room

A room sets the stage for rendering the model. The room can be cylindrical or rectangular. To create a rendering room, choose the **Room** tab from the **Scenes** window. Using the options in the **Room** tab, you can apply appearances to the wall, ceiling, and floor of the room, set the room orientation with respect to the model, adjust the size of the room and create a background to the room, if required.

### Defining Lights

The next step is to define the lighting effects on the model. Adding lights to the model, illuminates the model from various angles. To add lights, choose the **Lights** tab from the **Scenes** window. You can create a light source as well as position them by using the options in the **Lights** tab.



*Figure 1-39 The Scenes window*

Shadows get enabled as soon as you add lights to the model. The direction of shadows cast by a light bulb, spot light, skylight or environment type of lights is towards the center of the room.

## Adding Effects

Finally, you can add the surrounding effects such as reflection, tone mapping, background, and depth of field. The tone of the image or scene can be set as per the presets such as **Studio Settings**, **Indoor Settings**, and **Outdoor Settings**.

## Setting the Perspective View

You can set the perspective view of the model that can be used for rendering the model. To do so, choose **View > Model Setup > Perspective view** from the menu bar. To set the desired angle, invoke the **Perspective** dialog box by choosing **View > Model Setup > Perspective Settings** from the menu bar. Using this dialog box, you can adjust the perspective view settings such as **Walk Through**, **Fly Through**, **From To**, and **Follow Path**. You can also adjust the amount of perspective for viewing the model by adjusting the eye distance and the focal length.

## Render the Model

The last step is to render the model. To render the model, choose **View > Model Setup > Render Setup** from the menu bar; the **Render Setup** window will be displayed. Set the required rendering options in this dialog box. Next, you need to render the model with the specified settings. To do so, choose **View > Render Window** from the menu bar. If required, modify the render settings and render the model again.



### Note

You can toggle between the display of the rendering effects on the model by choosing the **Enhanced Realism** button in the **Model Display** toolbar.

## COLOR SCHEME USED IN THIS BOOK

Pro/ENGINEER allows you to use various color schemes as the background screen color, and also for displaying the entities on the screen. This textbook will follow the white background of Pro/ENGINEER environment for the purpose of printing. However, for better understanding and clear visualization at various places, this book will follow other color schemes too. To change the color scheme of the background screen, choose **View > Display Settings > System Colors** from the menu bar; the **System Colors** window will be displayed, as shown in Figure 1-40. Choose the **Edit** button on the right of the **Blended Background** check box; the **Blended Color** window will be displayed. Choose the color swatch beside the **Top** option to invoke the **Color Editor**. Next, set the R, G, B colors by using the slider bar to extreme right for changing the color to white and then close the **Color Editor**.

Similarly, change the color of the Bottom to white. Choose **OK** from the **Blended Color** window as well as from the **System Colors** window to accept the changes made and then exit the color scheme.

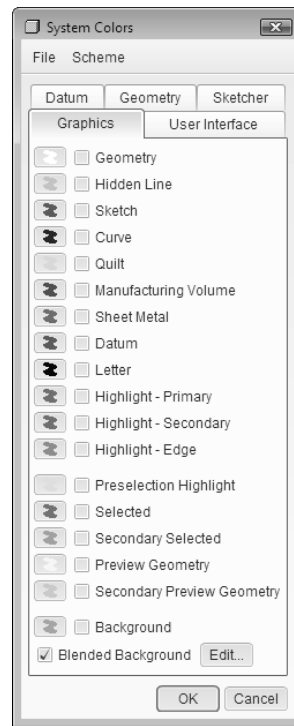


Figure 1-40 The **System Colors** window