

Chapter 1

Introduction to Creo Parametric 2.0

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the advantages of using Creo Parametric.*
- *Understand the bidirectional associative property.*
- *Know the system requirements of Creo Parametric.*
- *Learn various important terms and definitions in Creo Parametric.*
- *Learn various important options in the File menu.*
- *Understand the importance of Model Tree.*
- *Understand the functions of mouse buttons.*
- *Learn the use of default toolbars.*
- *Customize the Ribbon.*
- *Understand the functions of browser.*
- *Understand the use of Appearance Gallery.*
- *Understand the rendering stages in Creo Parametric.*
- *Change the color scheme of the background in Creo Parametric.*

INTRODUCTION TO Creo Parametric 2.0

Welcome to Creo Parametric. If you are a new user of Creo Parametric 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 because of the tremendous improvement in this latest release such as Flexible modeling, Freestyle modeling, and so on. Also, the interface of Creo Parametric is very user friendly. You will find a tremendous reduction in the time taken to complete a design using this solid modeling tool.

Creo Parametric is a powerful software used to create complex designs with great precision. The design intent of a three-dimensional (3D) model or an assembly is defined by its specification and its use. You can use the powerful tools of Creo Parametric to capture the design intent of a complex model by incorporating intelligence into the design. Once you understand the feature-based, associative, and parametric nature of Creo Parametric, 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 Creo Parametric 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. Creo Parametric allows you to work in a 3D environment and calculates the mass properties directly from the created geometry. 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 only appearance of the model. The solid models rendering in Creo Parametric is of much better quality than earlier releases.

FEATURE-BASED NATURE

Creo Parametric is a feature-based solid modeling tool. A feature is defined as the smallest building block and a solid model created in Creo Parametric is an integration of a number of these building blocks. Each feature can be edited individually to bring in the desired change in the solid model. The use of feature-based property provides greater flexibility to the parts created. For example, consider the part shown in Figure 1-1. It consists of one counterbore hole at the center and six counterbore holes around the 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, Creo Parametric 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 Creo Parametric are a combination of various features that can be modified individually at any time.

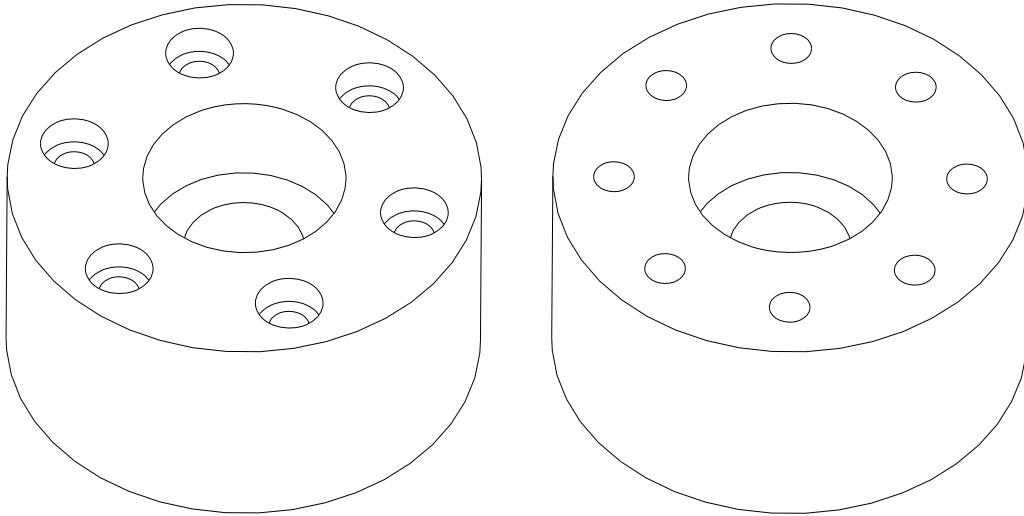


Figure 1-1 Model displaying the counterbore holes Figure 1-2 Model after making the modifications

BIDIRECTIONAL ASSOCIATIVE PROPERTY

There is a bidirectional associativity between all modes of Creo Parametric. 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, then those 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 a 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 Creo Parametric. 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 a modification is made to one mode, it changes the output of all the other modes related to the model. This bidirectional associative nature relates the various modes in Creo Parametric.

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.

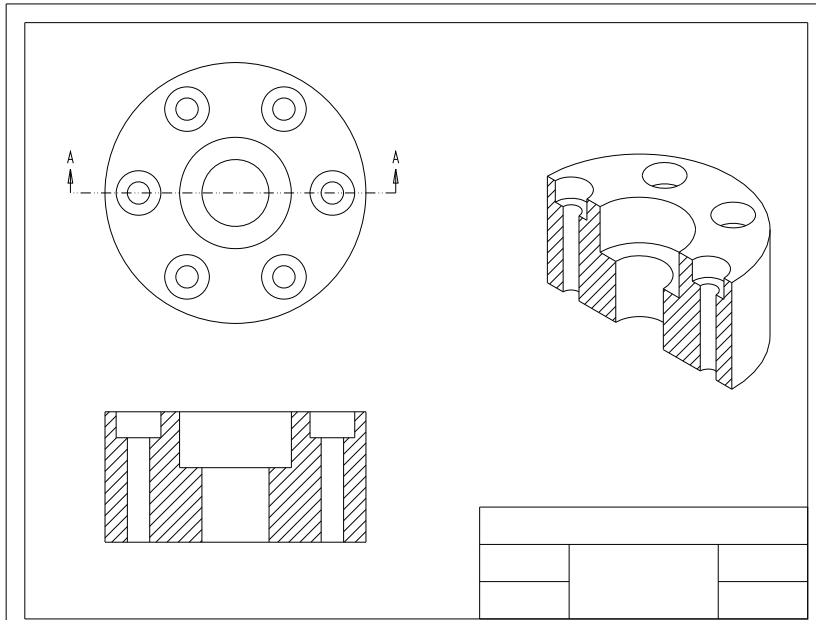


Figure 1-3 Drawing views of the model before modifications

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 1-5 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 Creo Parametric.

Since all modes of Creo Parametric are interrelated, it becomes very easy to modify your model at any time.

PARAMETRIC NATURE

Creo Parametric 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.

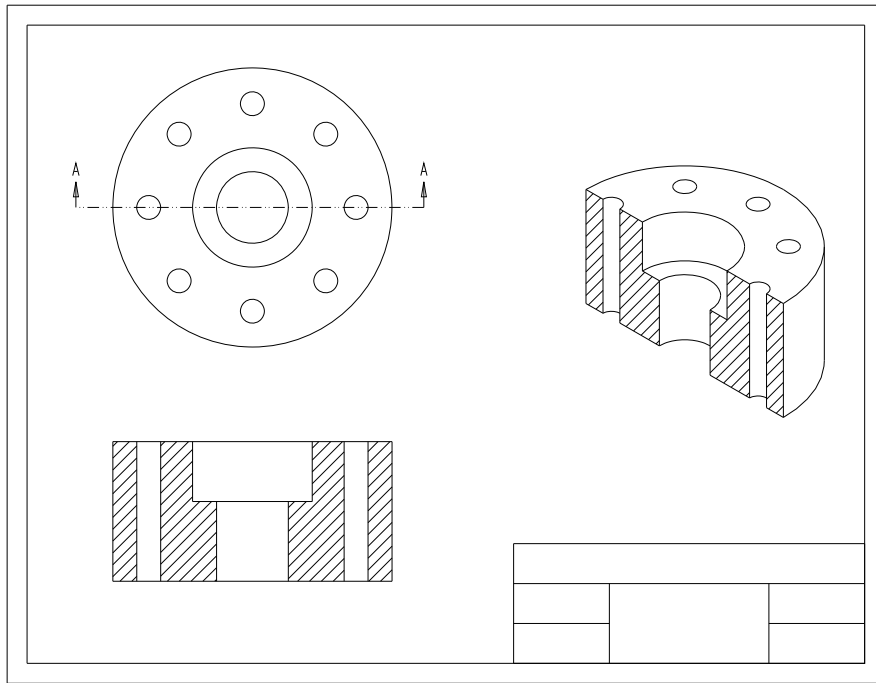


Figure 1-4 Drawing views of the model after modifications

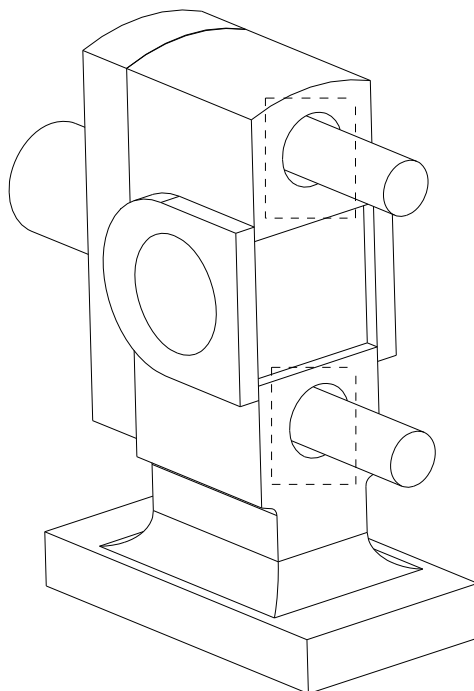


Figure 1-5 Diameter of the hole and the bolt in the Crosshead assembly

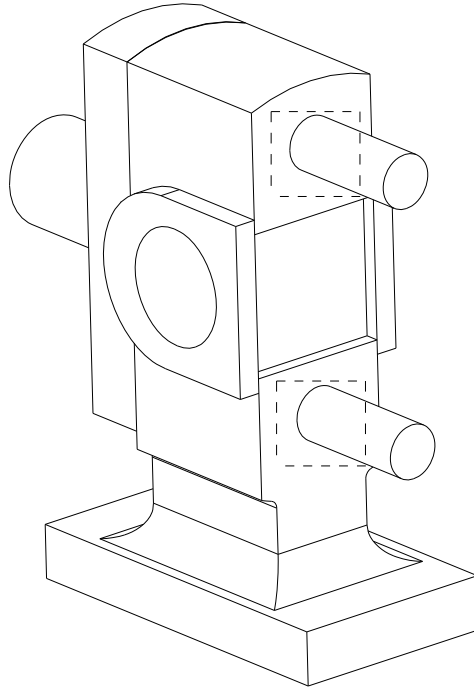


Figure 1-6 Model after modifying the diameter of the hole

SYSTEM REQUIREMENTS

The system requirements for Creo Parametric are given below.

1. Operating System: Windows XP Professional Edition, XP Professional x64 Edition, Windows 7 (Professional, Ultimate, and Enterprise Editions) or later.
2. Monitor: 1280 x 1024 (or higher) resolution support with 32-bit or more.
3. Processor: 3.0 GHz minimum (Core 2 Duo or higher, as recommended).
4. Memory: 2GB RAM minimum (3GB RAM or higher, as recommended).
5. Hard disk space: 3.0GB minimum (4.0GB or higher, as recommended).
6. An ethernet adapter interface card or network card.
7. Microsoft approved 3-button mouse.
8. Microsoft Internet Explorer 8.0 or later.
9. A certified and supported graphics card.

GETTING STARTED WITH Creo Parametric

Once you have installed Creo Parametric 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 **All Programs > PTC Creo > Creo Parametric 2.0**, as shown in Figure 1-7.

The second option to start Creo Parametric 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.

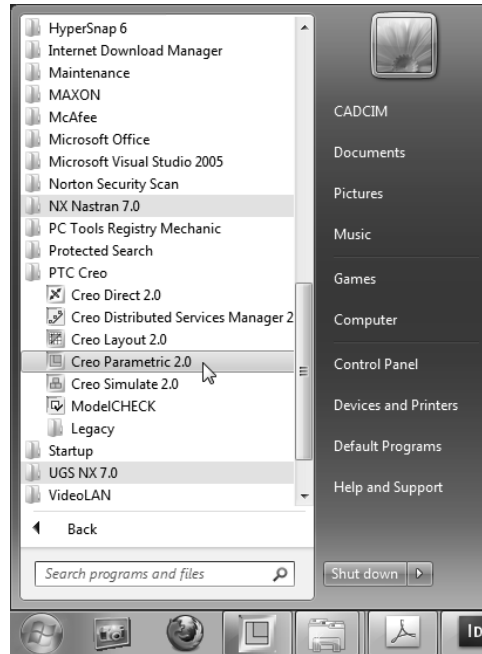


Figure 1-7 Starting Creo Parametric from the Start menu

Figure 1-8 shows the screen that appears when you start Creo Parametric.

IMPORTANT TERMS AND DEFINITIONS

Some important terms that will be used in this book while working with Creo Parametric are discussed next.

Entity

An element of the 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 are 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.

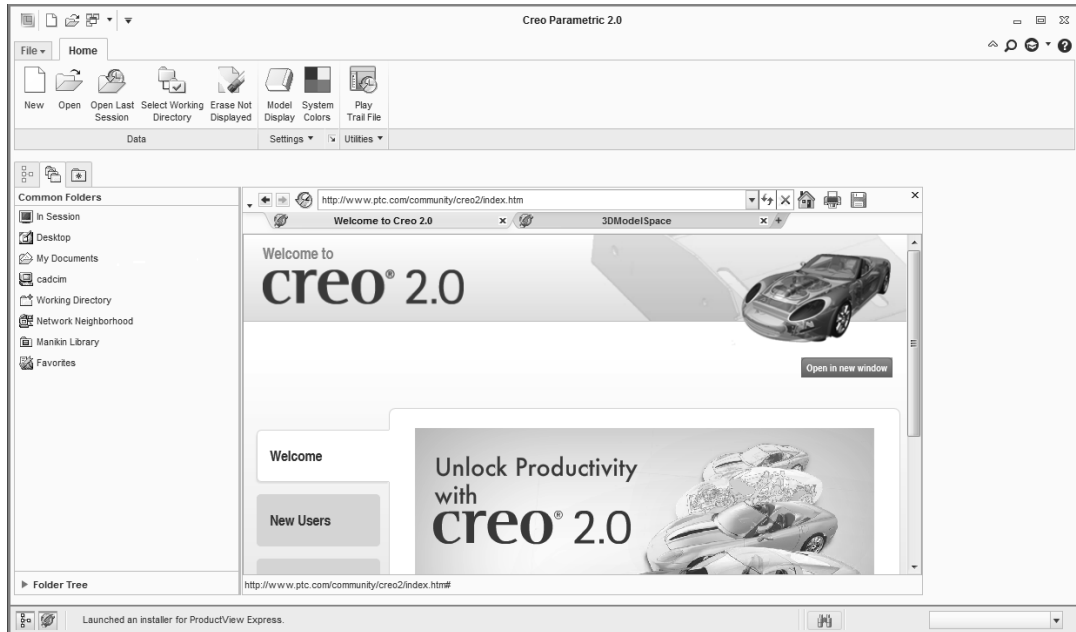


Figure 1-8 Initial screen appearance after starting Creo Parametric

Parameter

It is defined as a numeric value or a word 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 light blue color. These are automatically applied to the sketch. 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 dark blue 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, Creo Parametric makes the constraints and dimensions appear in blue box, 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.

Select Working Directory

A working directory is a directory on your system where you can save the work done in the current session of Creo Parametric. You can set any directory existing on your system as the working directory. Before starting the work in Creo Parametric, 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 Creo Parametric. If the working directory is selected before saving the object files that you create, it becomes easy to organize them. In Creo Parametric, the working directory can be set in two ways:

Using the Navigator

When you start a Creo Parametric 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. To do so, click on the **Folder Tree** node displayed at the bottom of the navigator; the expanded **Folder Tree** area will be displayed. Browse to the required location using the nodes available with the folders and select the desired folder. The selected folder will become the working directory for the current session. Alternatively, 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.

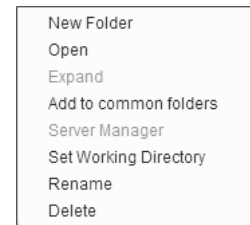


Figure 1-9 Shortcut menu

Using the Select Working Directory Dialog Box

To specify a working directory, choose **File > Manage Session > 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.

Choose the arrow at the upper left corner of the **Select Working Directory** dialog box; a flyout will be displayed; as shown in Figure 1-11. This flyout displays some of the drives present on your computer along with the **Favorites** folder. 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.

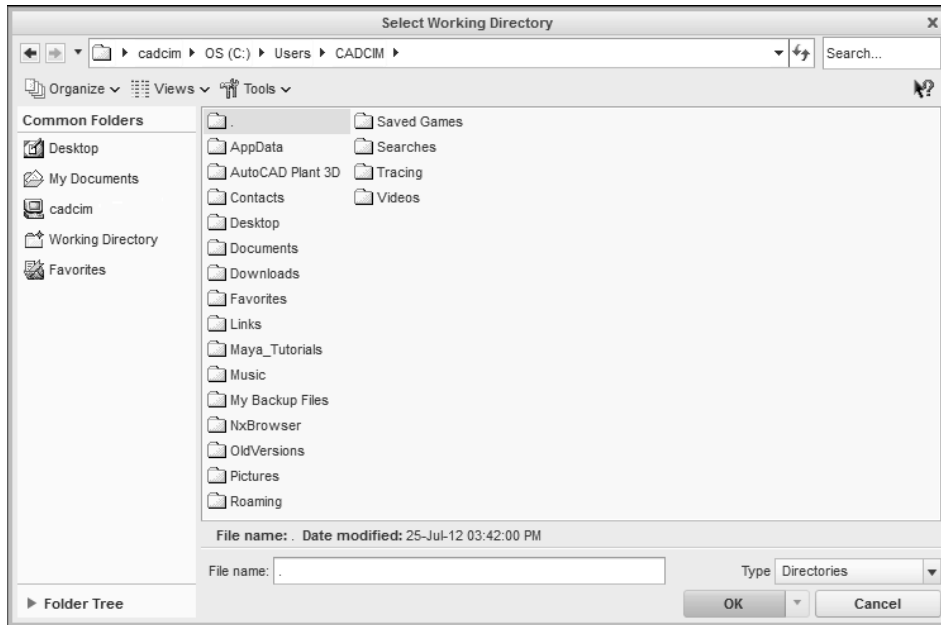


Figure 1-10 The Select Working Directory dialog box

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 by entering 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 get listed, and if you select the **All Files (*)** option, then all files along with the directories are listed in the dialog box.

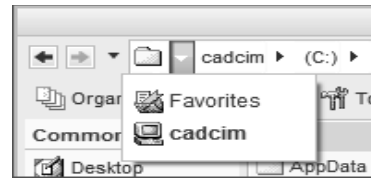


Figure 1-11 The flyout with some drives

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.



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 folder in this dialog box. Alternatively, you can create a new directory using this dialog box.

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 displays the name, size, and date on which it was last modified.

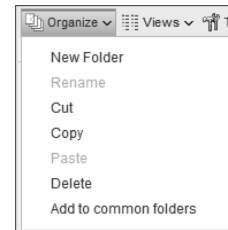


Figure 1-12 The **Organize** flyout

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.

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** folder selected by default. Similarly, you can set the **Pro/Library** as the working directory.

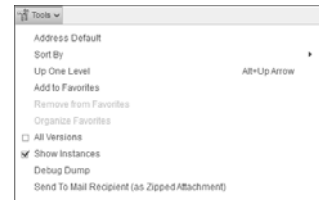


Figure 1-13 The **Tools** flyout

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 or BACKSPACE key 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.

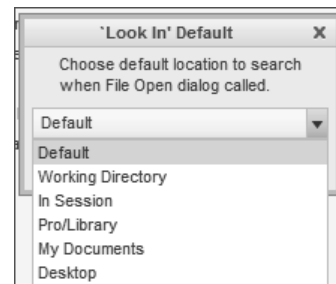


Figure 1-14 The **'Look In' Default** dialog box with options in the drop-down list

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 **Organize** flyout.

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 node 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 Creo Parametric 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 the **New** tool from the **Data** group of the **Home** tab in the **Ribbon** or choose the **New** tool from the **Quick Access** toolbar; the **New** dialog box will be displayed, as shown in Figure 1-15. This dialog box displays various modes available in Creo Parametric. In this dialog box, by default, the **Part** mode radio button is selected and the default name of the object file is displayed in the **Name** edit box. You can also enter a new name for the object file. Note that the name must not contain a special character.

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 default template will be loaded. To load a template other than the default one, clear the **Use default template** check box and then choose 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 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.

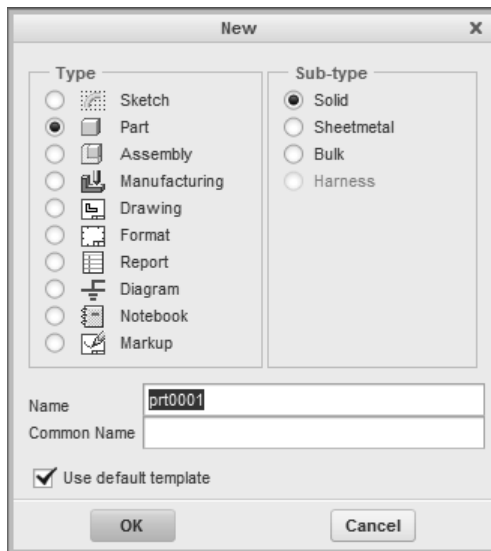


Figure 1-15 The New dialog box

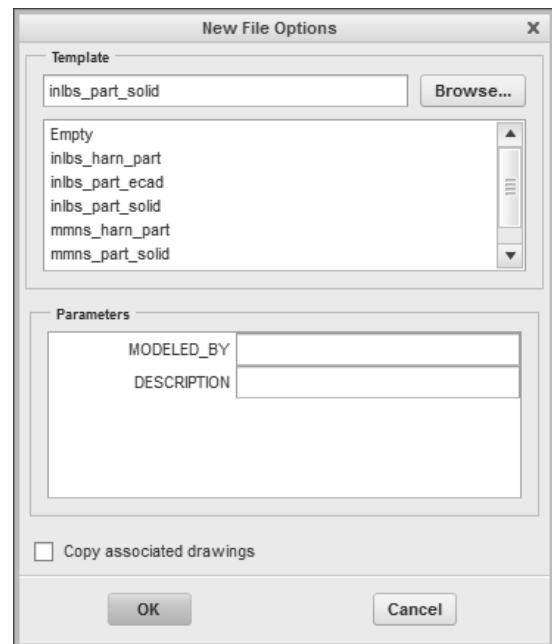


Figure 1-16 The New File Options dialog box

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.

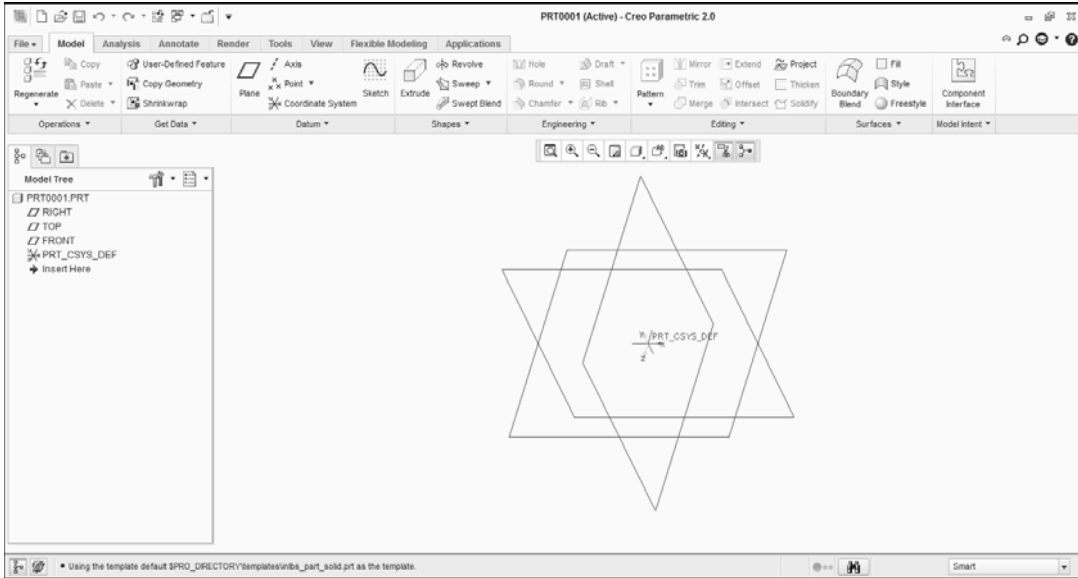


Figure 1-17 The initial screen appearance after entering the **Part** mode

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 **Quick Access** toolbar, 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 same as discussed in the **Select Working Directory** dialog box. The rest of the options in this dialog box are discussed next.

Tools

On choosing this button, a flyout will be displayed. The options available in this flyout are 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 Creo Parametric, 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.

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** flyout; 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 Creo Parametric. It also contains many other file formats that can be imported in Creo Parametric. These file formats include IGES, SET, STEP, DWG/DWF, Medusa, Inventor, Parasolid, Rhino, and so on.

By default, the Creo Files option is selected in this drop-down list. As a result, you can open the files created in any mode of Creo Parametric. 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.

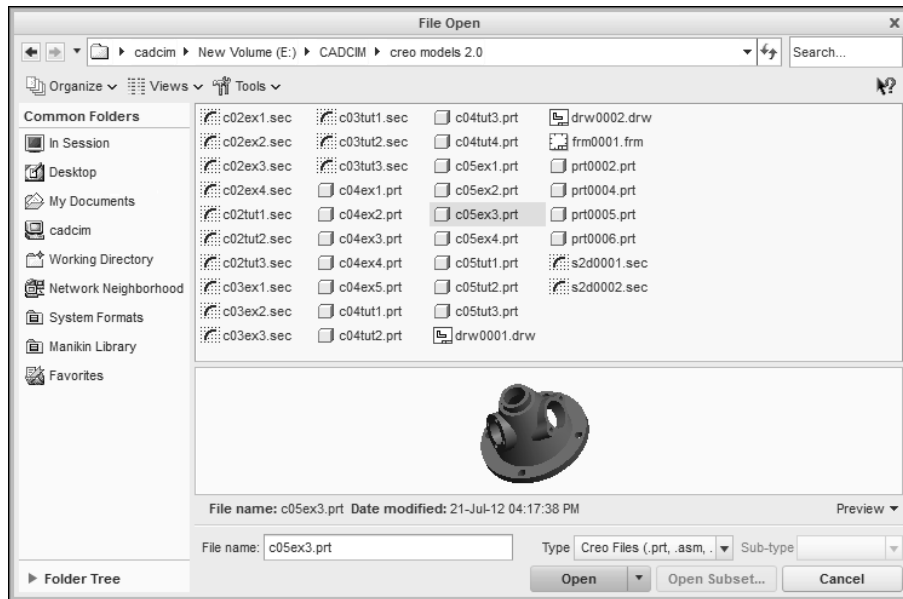


Figure 1-18 The **File Open** dialog box with the Preview area

Preview

The **Preview** button is used to preview the model before opening it. On choosing this button, you can preview the selected model in the **File Open** dialog box. 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 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 Creo Parametric. However, you are provided with prompts in the message area. Whenever you have to enter a 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 Creo Parametric in the current session are stored in its temporary memory. This temporary memory is stored in a folder named **In Session**. Once you exit Creo Parametric, the contents of this folder are deleted automatically. However, the original files are not removed from their actual location.

Erase

As discussed earlier, all files opened in a session of Creo Parametric are saved in the temporary memory. There are three options to erase objects in the temporary memory: **Erase Current**, **Erase Not Displayed**, and **Erase Unused Model Reps**. These options are available in the **Manage Session** flyout in the **File** menu. The options that are displayed in this flyout are discussed next.



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 Creo Parametric (in session).

To avoid this error of assemblies, you should erase the files in the current session of Creo Parametric before opening the next assembly.

Erase Current

The **Erase 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.

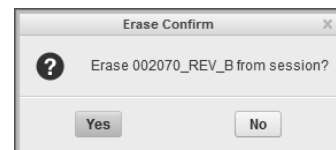


Figure 1-19 The **Erase Confirm** message box

Erase Not Displayed

The **Erase** option is used to delete the files stored in the temporary memory. To do so, choose **File > Manage Session > Erase Not Displayed** from the menu bar; the **Erase Not Displayed** dialog box will be displayed, as shown in Figure 1-20. The files that are not open in the current session will be displayed in this dialog box. Choose the **OK** button from this dialog box to remove these.

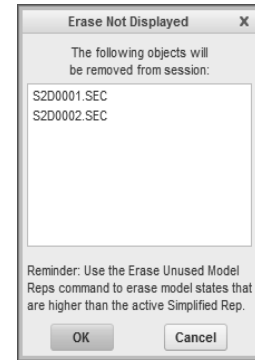


Figure 1-20 The **Erase Not Displayed** dialog box

Erase Unused Model Reps

This option is used to remove the unused simplified representations from the **In Session** folder. When you choose this option from the **File** menu, a message box will be displayed with the message that all the objects which were not displayed have been erased.

Delete

There are two options to delete files permanently from the hard disk. These options are available in the **Manage File** flyout in the **File** menu. The options in this flyout are discussed next.

Delete 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.

Delete 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.



Note

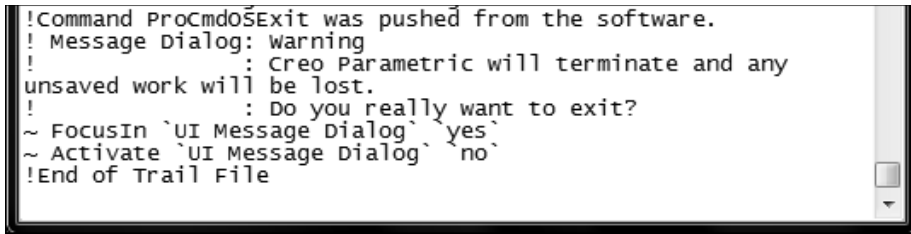
*If there are simplified representations in the memory, then **Object Erase** menu will be displayed. Choose the **Current Obj** option if you want to delete object files only or you can choose the **Simpfld Reps** option if you want to delete the representations only.*

Play Trail File

The **Play Trail File** option is used to recover data in case of a program crash or if you exit the session without saving the data. While working in Creo Parametric, a file called *trail.txt* is created in its default working directory that would be similar to *C:\Users\Public\Documents*. The file *trail.txt* contains all the working steps used in the last session.

To retrieve the data, open the file *trail.txt* in Notepad. If there is more than one such file in the directory open the latest one. Scroll to the end of this file and change the **Activate 'UI Message**

Dialog from **'yes'** to **'no'**, as shown in Figure 1-21. Save this file by choosing **File>Save As** with another name like *crash.txt* and exit.



```
!Command ProcMdo$Exit was pushed from the software.
! Message Dialog: Warning
!                  : Creo Parametric will terminate and any
unsaved work will be lost.
!                  : Do you really want to exit?
~ FocusIn `UI Message Dialog` `yes`
~ Activate `UI Message Dialog` `no`
!End of Trail File
```

Figure 1-21 Modified *trail.txt* file using Notepad

Now, restart Creo Parametric and choose the **Play Trail File** option from **Manage Session**. Select the file *crash.txt* from the **Open** dialog box and then choose the **Open** button; Creo Parametric will repeat every step you made in the last session and restore the data.

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 from the **Quick Access** 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.

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 > Save As**, the **Save a Copy** dialog box will be displayed. Now, 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 in the **Save a Copy** dialog box.

Save a Backup

The **Save a Backup** option in the **Save As** flyout is used to create a backup copy of an object file in the memory. When you choose this option, the **Backup** dialog box will be displayed, as shown in Figure 1-22.

In the **Model Name** edit box of the **Backup** dialog box, the name of the file for which you want to create 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, Creo Parametric will save all its dependent files in the specified directory.

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.

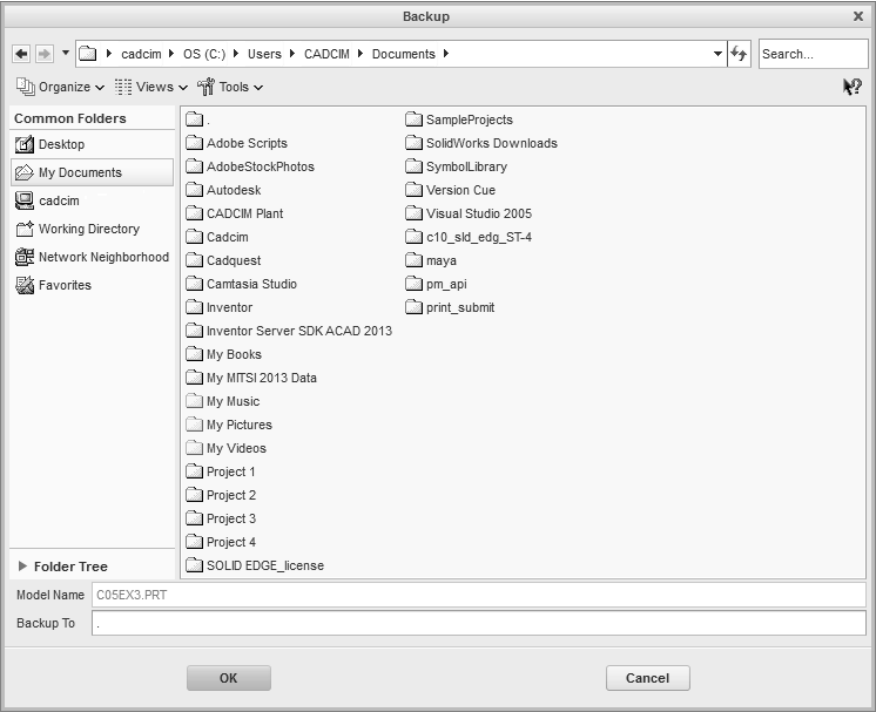


Figure 1-22 The *Backup* dialog box

Rename

The **Rename** option in the **Manage File** flyout of 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 1-23. Specify the new name of the object in the **New Name** edit box and choose **OK**.

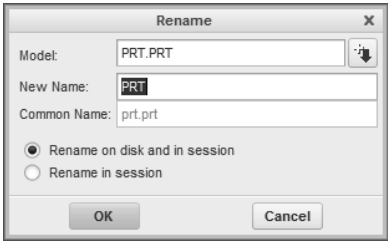


Figure 1-23 The *Rename* dialog box

MANAGING FILES

As discussed earlier, a new file is generated whenever you save an object. The number of files generated are 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 **File > Manage File > Delete Old Versions** option from the menu bar.

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 **Operations > Feature Operations** from the **Ribbon**, 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 Creo Parametric. If you are directly selecting one option after another, then it is easy to loose 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 as shown in Figure 1-24. Therefore, the drawing area becomes small. You can hide the **Model Tree** by choosing the **Toggle the display of navigation area** button available below the **Model Tree**. The **Model Tree** can slide in or slide out, thus increasing or decreasing the drawing area. It can also be stretched horizontally to cover the drawing area.

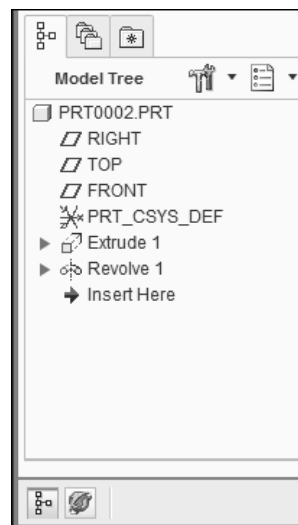


Figure 1-24 Partial view of the **Model Tree**

**Note**

In Creo Parametric, most of the features are created using the **Dashboard**. The **Dashboard** is displayed below the **Quick Access** toolbar and it contains all the options to complete the related operations on the model.



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 Creo Parametric, you do not get features of the model in the **Model Tree** and therefore, you will not be able to modify it.

UNDERSTANDING THE FUNCTIONS OF THE MOUSE BUTTONS

While working with Creo Parametric, it is important to understand the function of the three buttons of the mouse to make an efficient use of this device. The various combinations of the keys and three buttons of the mouse are listed below:

1. Figure 1-25 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 1-26 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 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.

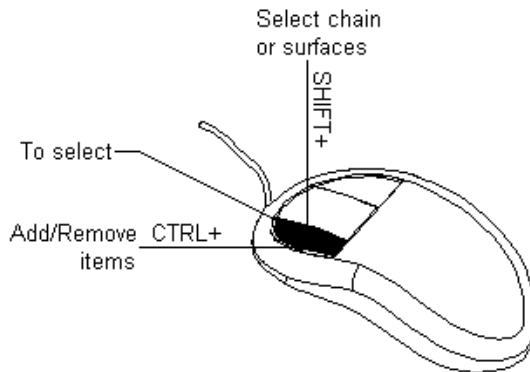


Figure 1-25 Functions of the left mouse button

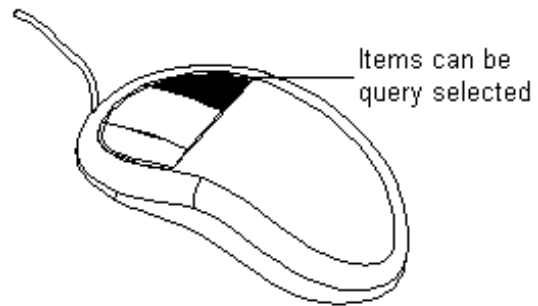


Figure 1-26 Functions of the right mouse button

3. Figure 1-27 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.

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

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

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

4. Figure 1-28 shows the functions of the middle mouse button in the 2D mode (sketcher environment and **Drawing** mode). It 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 view in the **Sketch** mode and the **Drawing** mode.

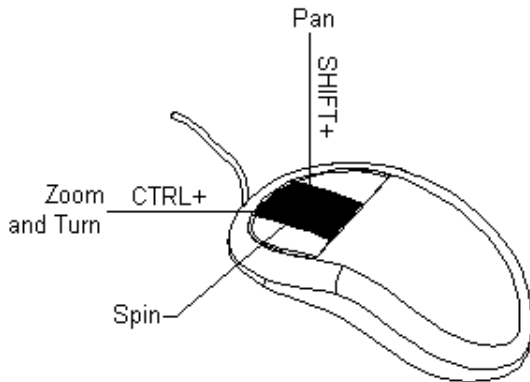


Figure 1-27 Functions of the middle mouse button in the 3D mode

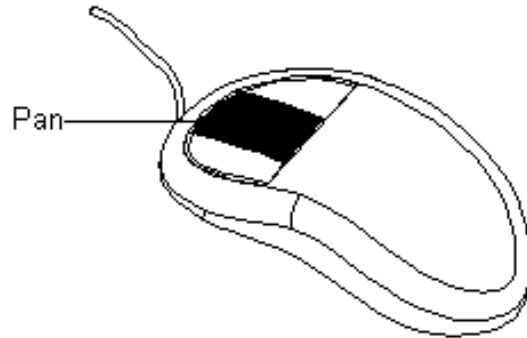


Figure 1-28 Functions of the middle mouse button in the **Sketch** mode



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.

RIBBON

Before you start working on Creo Parametric, it is very important for you to understand the default **Ribbon** and tools in the main window. Figure 1-29 shows various default screen components in Creo Parametric. The **Ribbon** is composed of a series of groups, which are organized into tabs depending on their functionality. The groups in the **Ribbon** that initially appear on the screen are shown in Figure 1-29. You will notice that all the tools in the groups are not enabled. These buttons will be enabled only after you create a part or open an existing file but the tools that are required for the current session are already enabled. As you proceed to enter one of the modes provided by Creo Parametric, you will notice that the tools required by that mode are enabled. Additionally, to make the designing easy and user-friendly, this software package provides you with a number of groups. Different modes of Creo Parametric display different groups. Some of the frequently used groups are shown in Figure 1-29.

TOOLBARS

In Creo Parametric, there are two toolbars, **Quick Access** and **In-graphics**. The toolbar on the top of the window is called the **Quick Access** toolbar and the toolbar on the top of the drawing area is called the **In-graphics** toolbar.

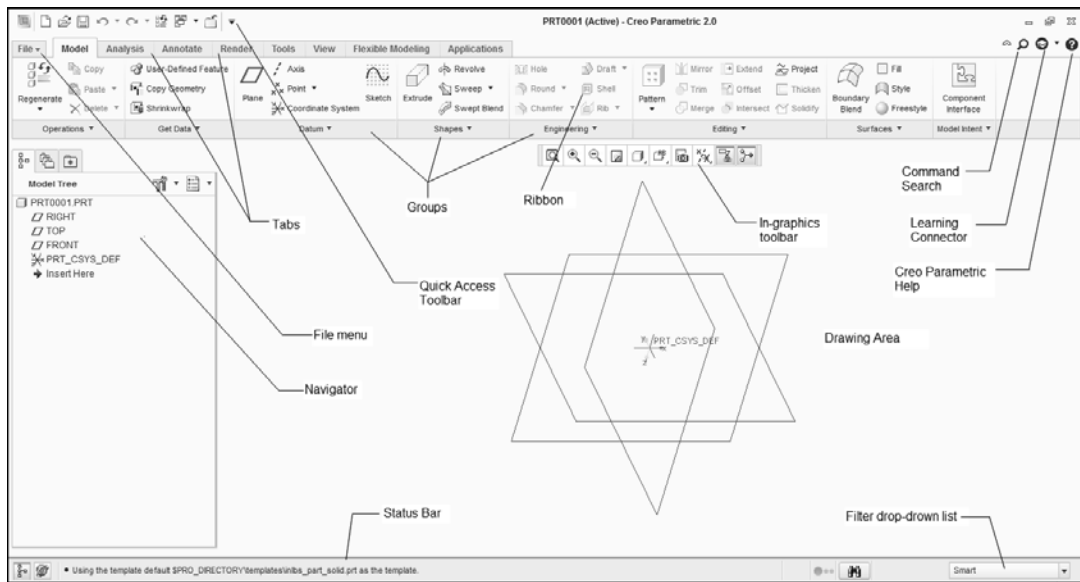


Figure 1-29 Default screen components

In-graphics Toolbar

The **In-graphics** toolbar, shown in Figure 1-30, has ten buttons by default. The first button is **Refit**, which is used to fit a model on the screen. The other buttons are **Zoom In** and **Zoom Out**, which are used to enlarge or diminish the model view, respectively. The **Repaint** button is used for repainting the screen, which helps in removing any temporary information from the drawing area. You can change the display style by using the **Display Style** drop-down. Using the **Named Views** button, you can change the view.

The next drop-down is **Datum Display Filters**. Using the options in this drop-down, you can toggle the display of datums to on/off. You can toggle the display of annotations to on/off using the **Annotation Display** button. The last button in this toolbar is **Spin Center**, which is used to toggle the visibility of the spin center on/off.

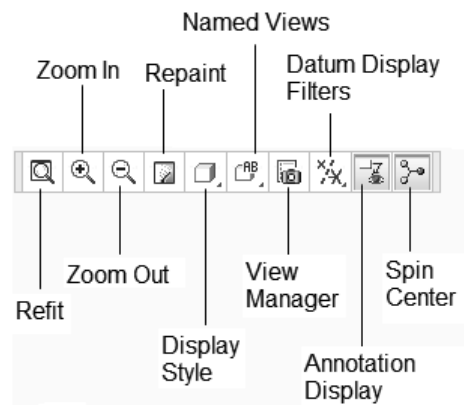


Figure 1-30 The *In-graphics* toolbar

File Menu

The **File** menu contains options shown in Figure 1-31. These options are used to create a new file, save a file, print the current file, or open an existing file. Various other options are also available in the menu to manage the current session and files. The **Options** tool is also

available in the **File** menu. When you choose this tool, the **Creo Parametric Options** dialog box is displayed. Using this dialog box, you can configure various parameters of Creo Parametric. You can also customize the **Ribbon** interface.

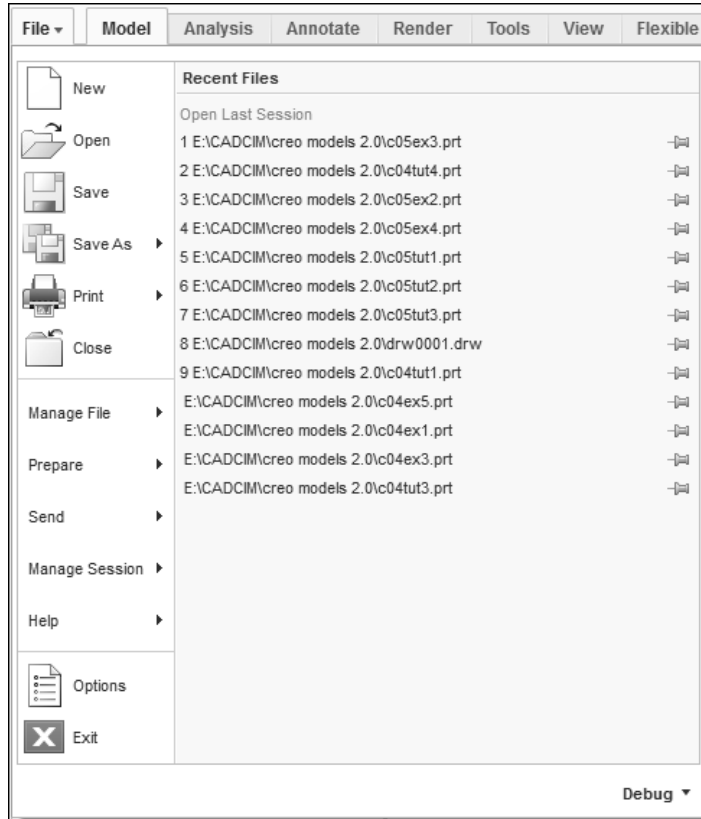


Figure 1-31 The File menu

Customizing the Ribbon

To customize the ribbon, choose **Options** from the **File** menu; the **Creo Parametric Options** dialog box will be displayed. Select the **Customize Ribbon** option from the left area of the dialog box; the dialog box will be modified. Now, choose **Commands Not in the Ribbon** option from the **Choose commands from** drop-down in the right area of the dialog box. Next, choose the **New Group** or **New Tab** button from the dialog box depending on the requirement; the new group or tab will be added in the list box below the **Customize the Ribbon** drop-down in the dialog box. Now, choose the button to be added in the new group from the list box available below the **Choose commands from** drop-down and then choose the **Add>>** button from the dialog box; the selected button will be added to the group, refer to Figure 1-32.

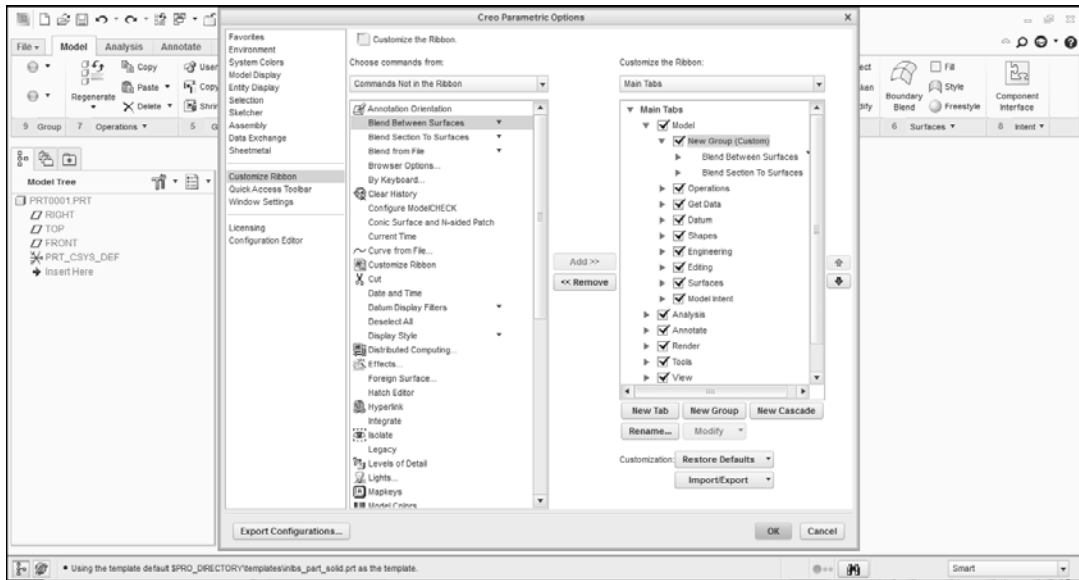


Figure 1-32 Customizing the Ribbon

Creo Parametric Help

When you choose the **Creo Parametric Help** button, the **Creo Parametric Help** dialog box will be displayed and you will be prompted to enter the Username and Password. The help database is accessible only to the users who have customer account with PTC.

Command Search



The **Command Search** button is available at the top right corner of the program window. When you choose this button, the search box is displayed, as shown in Figure 1-33. You need to enter the first few letters of the command to be searched in the box. When you enter the first letter in the search box, a list of commands starting from that letter will be displayed. You can invoke any command by selecting it from the list, or you can enter the next letter to refine your search. On moving the cursor on any of the commands in the list, the command will get highlighted in the application window as shown in Figure 1-34. Also the path of the command will be displayed below the cursor.

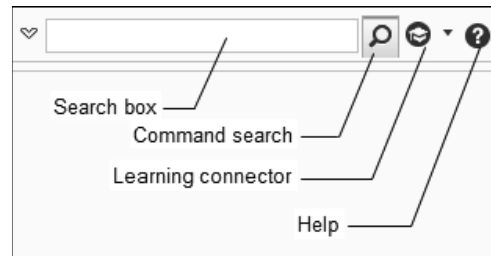


Figure 1-33 The command search box

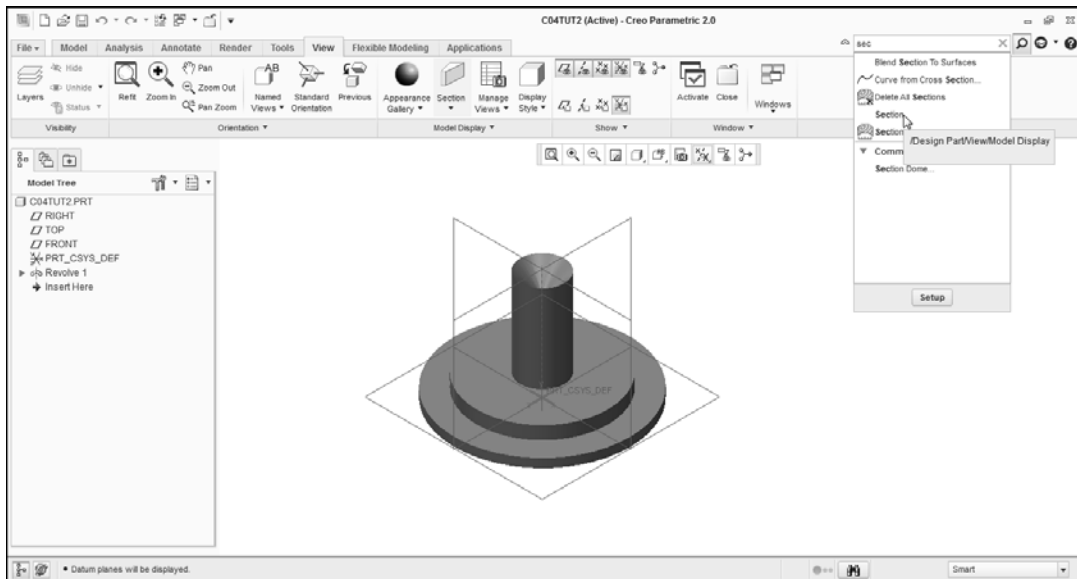


Figure 1-34 Command Search function

LearningConnector Button

The **LearningConnector** button is used to access 200+ video tutorials provided by PTC. When you choose this button, the **LearningConnector** window will be displayed as shown in Figure 1-35. When you invoke any tool from the **Ribbon**, the tutorial corresponding to that tool will be displayed in the **LearningConnector** window.

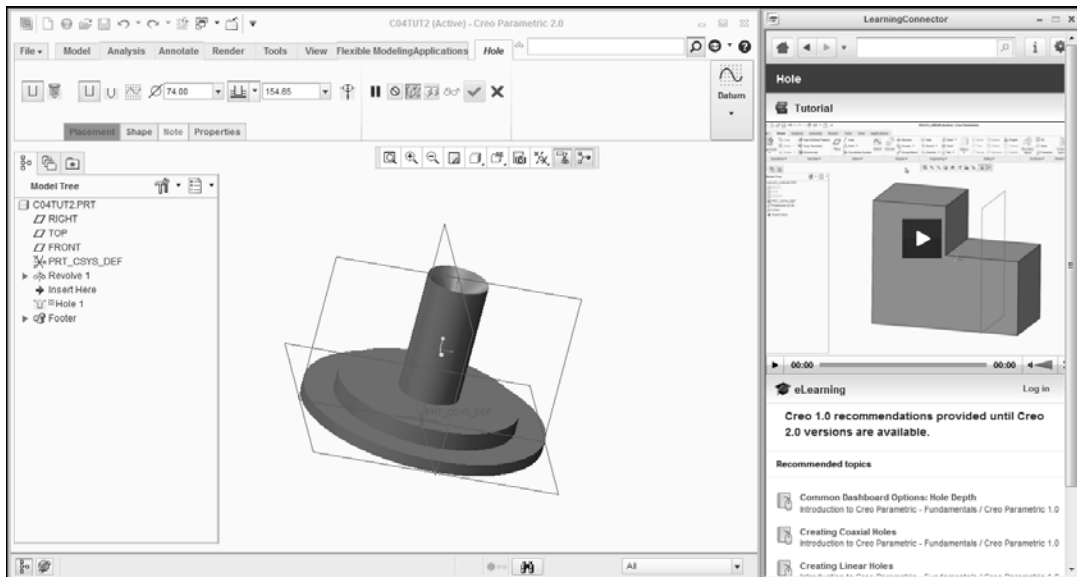


Figure 1-35 Creo Parametric 2.0 screen with **LearningConnector** window displayed



Note

If you want to change the predefined unit system, choose **File > Prepare > Model Properties** from the menu bar to display the **Model Properties** dialog box. In this dialog box, click on the **change** on the right of the **Units** option in 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 from the 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 of the drawing area. To make the navigator slide in or out, you need to select the **Toggle the display of navigation area** button on the bottom left corner of the program window. A partial view of the navigator is shown in Figure 1-36. 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 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 location, if available on your system, can also be accessed by using the navigator.

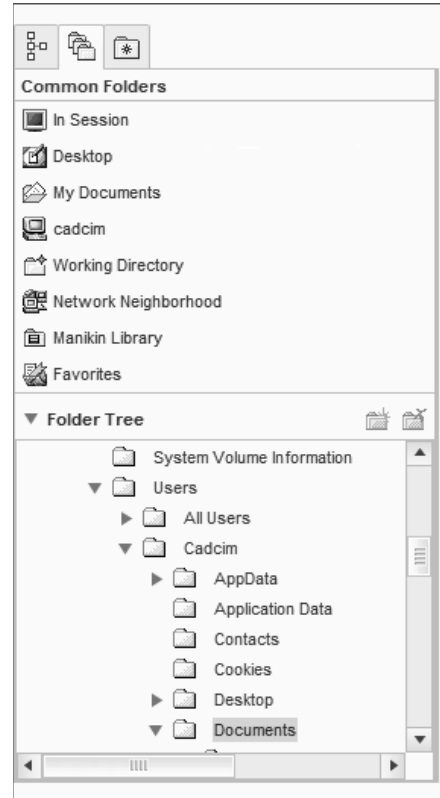


Figure 1-36 Partial view of the navigator

Creo Parametric BROWSER

The Creo Parametric Browser is present on the right of the navigator. It can slide in and out of the navigator by using the **Toggle the display of Creo Parametric browser** button at the bottom left corner of the application window. You can also stretch the browser window horizontally. When the Creo Parametric 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 online resources of PTC. 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 Creo Parametric files and navigate through the browser, simultaneously. Figure 1-37 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 Creo Parametric window.

The functions of the browser are listed next.

1. It is used to preview the Creo Parametric files and browse the file system.
2. A Creo Parametric 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 Creo Parametric user community site.
4. You can connect to your client's computer and jointly work with them using the browser.

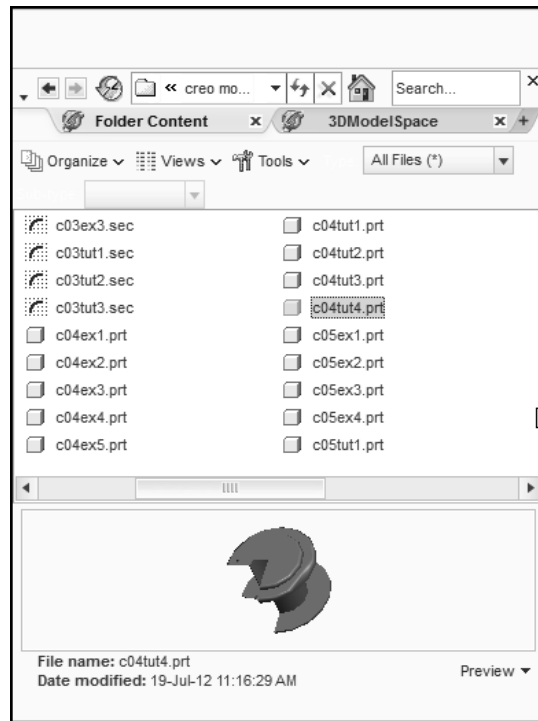


Figure 1-37 Creo Parametric browser

APPEARANCE GALLERY

A new library of appearances and real world materials have been added in Creo Parametric. You can also add shadows and reflections to the background. This is useful while rendering with a background image. This gives a realistic view to the model.

The **Appearance Gallery** button is available in the **Appearance** group of the **Render** tab in the **Ribbon**. 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 1-38. Select the required appearance from the palette; the **Select** box will be displayed. Select the entities from the drawing area whose appearance you want to change; the appearance of the entities will change. You can assign a desired 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 a 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 they will be 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, names and thumbnails, or only names. To do so, invoke the **View Options** flyout and then choose the required option from it. Choose **Rendered Samples** from the flyout 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.

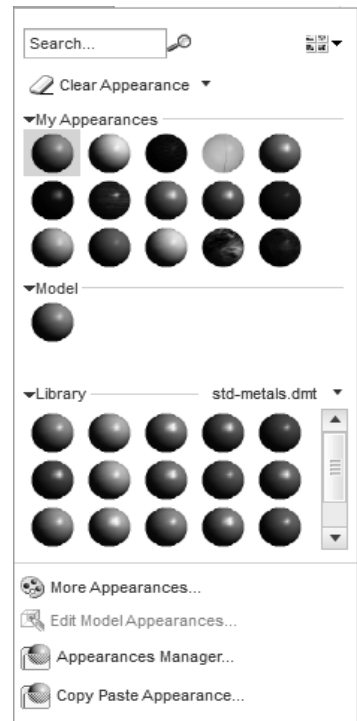


Figure 1-38 Appearance Gallery

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 the appearance icon and then choose the required option from the shortcut menu displayed. Note that you cannot delete the active appearance.

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 1-39. 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.

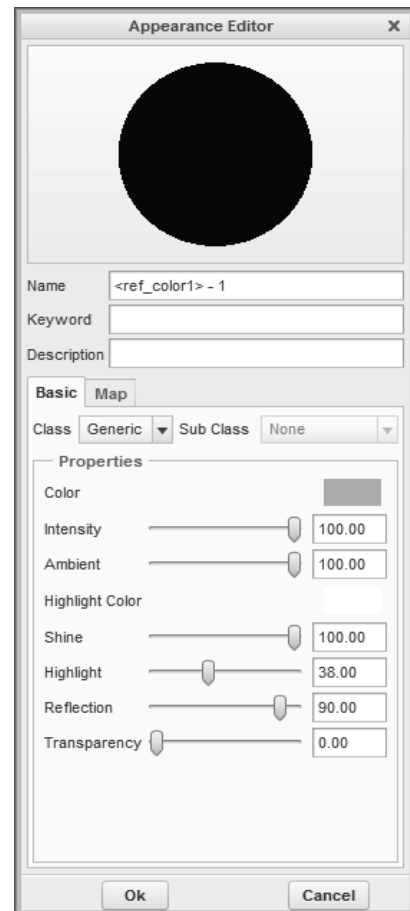


Figure 1-39 The Appearance Editor

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 along with their properties. The **Model Appearance Editor** is same as **Appearances Editor**. Using this editor, you can modify the properties such as color, texture, shine, and gloss of the model.

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 values 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 Creo Parametric

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 to visualize 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 predefined render settings that include lights, rooms, and environmental effects. To set scenes, choose the **Scene** tool from the **Scene** group of the **Render** tab in the **Ribbon**; the **Scenes** dialog box will be displayed, as shown in Figure 1-40. 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.

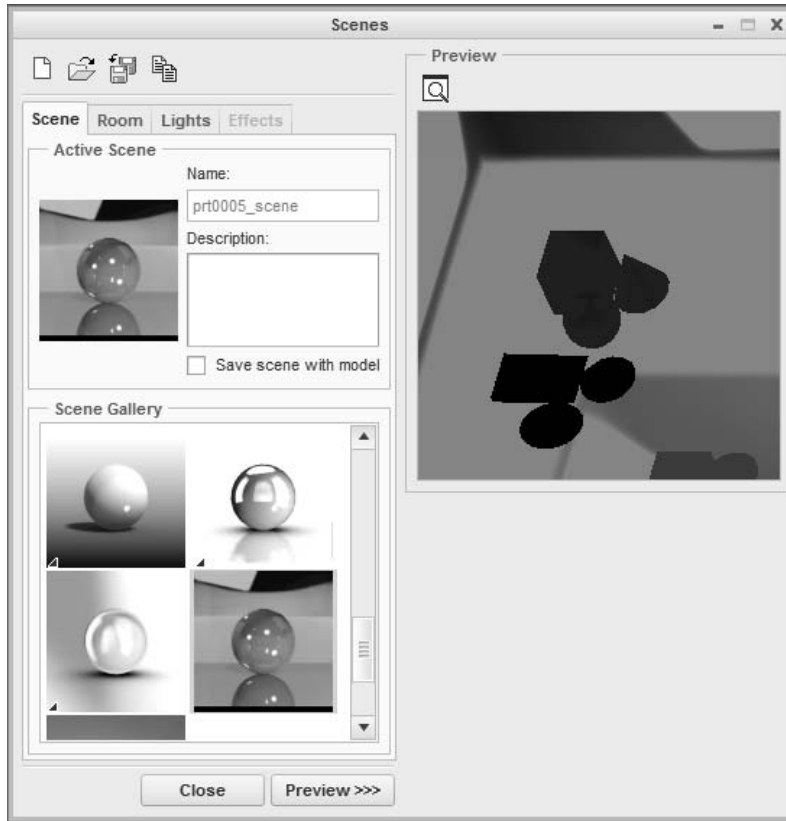


Figure 1-40 The Scenes dialog box

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.

Shadows are 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 the **Perspective view** button from the **Perspective** group in the **Render** tab of the **Ribbon**. To set the desired angle, invoke the **Perspective** dialog box by choosing the **Perspective Settings** button from the **Perspective** group in the **Render** tab of the **Ribbon**. 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.

Rendering the Model

The last step is to render the model. To render the model, choose the **Render Setup** button from the **Setup** group in the **Render** tab of the **Ribbon**; the **Render Setup** window will be displayed. Set the required rendering options in this window. Next, you need to render the model with specified settings. To do so, choose the **Render Window** button from the **Render** group in the **Render** tab of the **Ribbon**. 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 **Shading** button from the **Display Style** drop-down in the **In-graphics** toolbar.*

COLOR SCHEME USED IN THIS BOOK

Creo Parametric allows you to use various color schemes for the background screen color and for displaying the entities on the screen. This textbook will follow the white background of Creo Parametric 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 **File > Options** from the menu bar; the **Creo Parametric Options** window will be displayed. Select the **System Colors** category from the left pane of this window. Next, click on the node adjacent to the **Graphics** category of the **Colors** area in the right pane of this window; different object names will be displayed with color drop-downs adjacent to them, refer to Figure 1-41. Choose white color from the **Background** drop-down in this category and choose the **OK** button to apply the color.

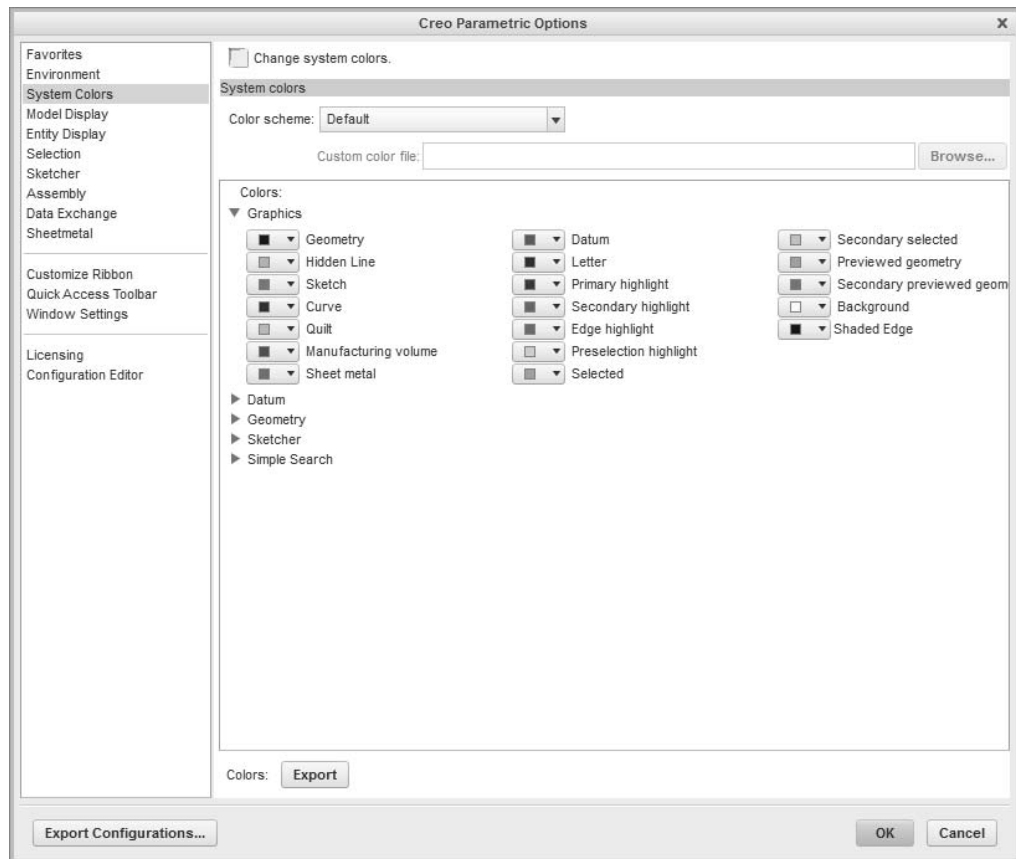


Figure 1-41 The System Colors category in the Creo Parametric Options dialog box