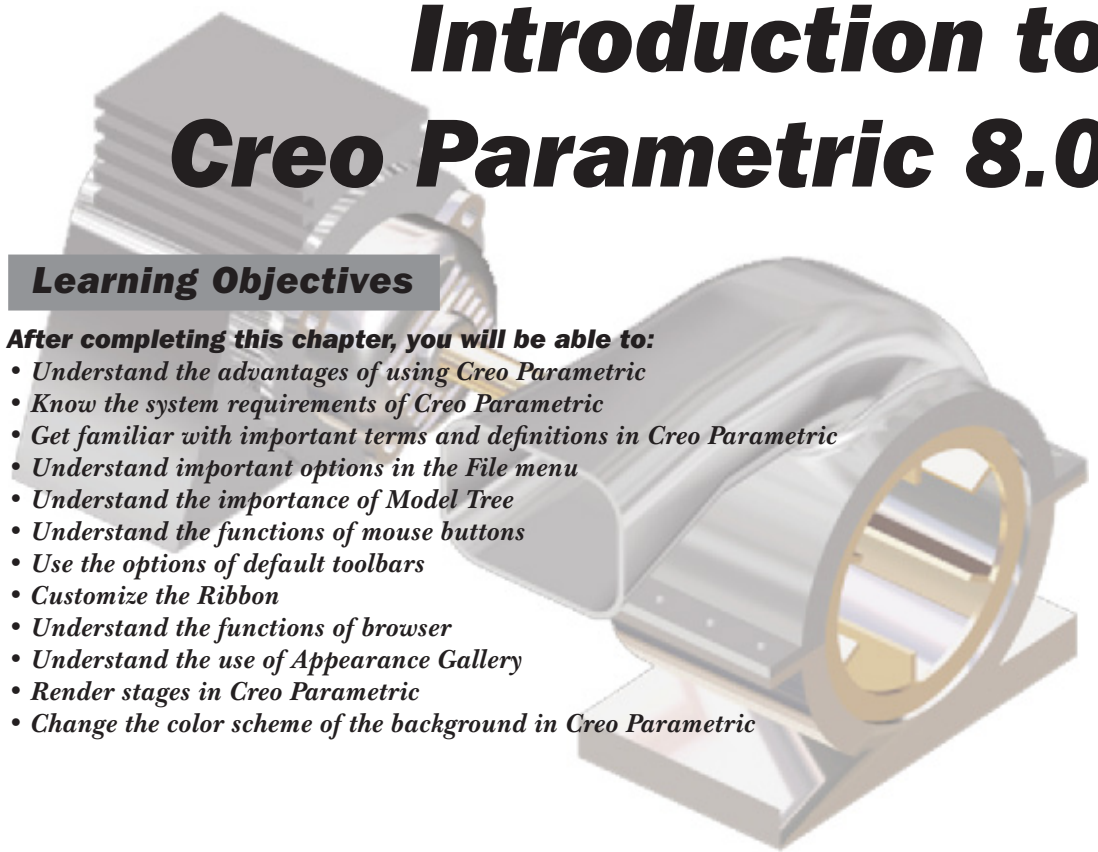
Chapter 1

Introduction to Creo Parametric 8.0



Learning Objectives

After completing this chapter, you will be able to:

- *Understand the advantages of using Creo Parametric*
- *Know the system requirements of Creo Parametric*
- *Get familiar with important terms and definitions in Creo Parametric*
- *Understand important options in the File menu*
- *Understand the importance of Model Tree*
- *Understand the functions of mouse buttons*
- *Use the options of default toolbars*
- *Customize the Ribbon*
- *Understand the functions of browser*
- *Understand the use of Appearance Gallery*
- *Render stages in Creo Parametric*
- *Change the color scheme of the background in Creo Parametric*

INTRODUCTION TO CREO PARAMETRIC 8.0

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 modeling tool.

To make the designing process simple and quick, the designing process have been divided into different modules in this software package. 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 only changes its appearance.

FEATURES OF CREO PARAMETRIC

Different features of the software are discussed next.

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 non feature-based software package, you will have to delete the entire part and then create a new part based on 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.

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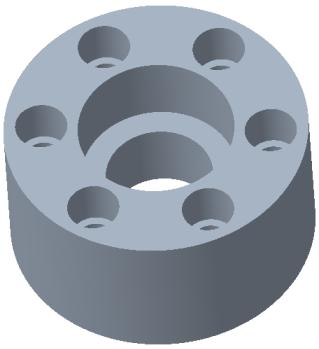


Figure 1-1 *Model with counterbore holes*

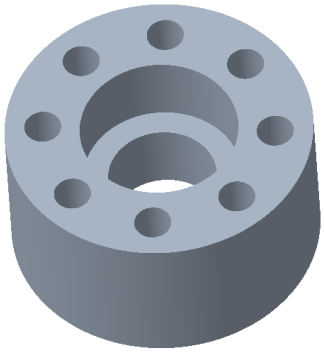


Figure 1-2 *Model after making modifications*

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 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.

Now, when the part is modified in the **Part** mode, as shown in Figure 1-2, 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.

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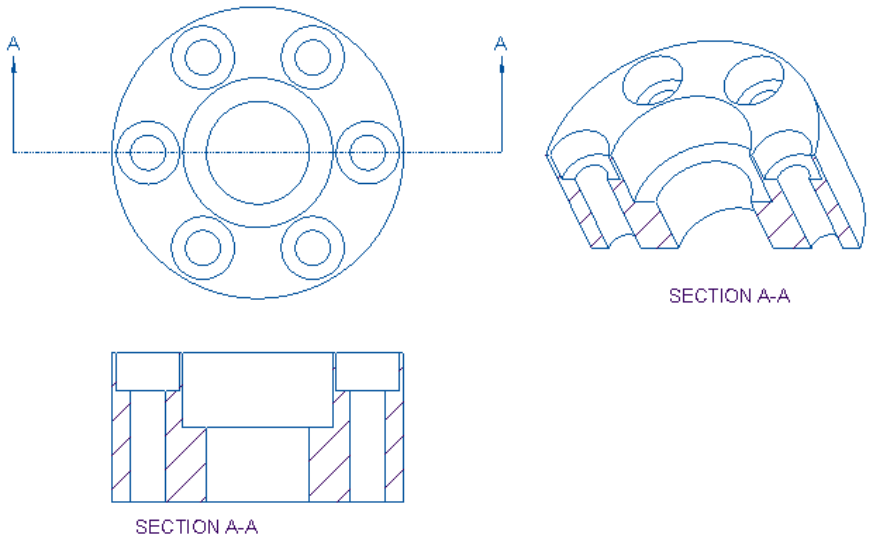


Figure 1-3 *Drawing views of the model before modifications*

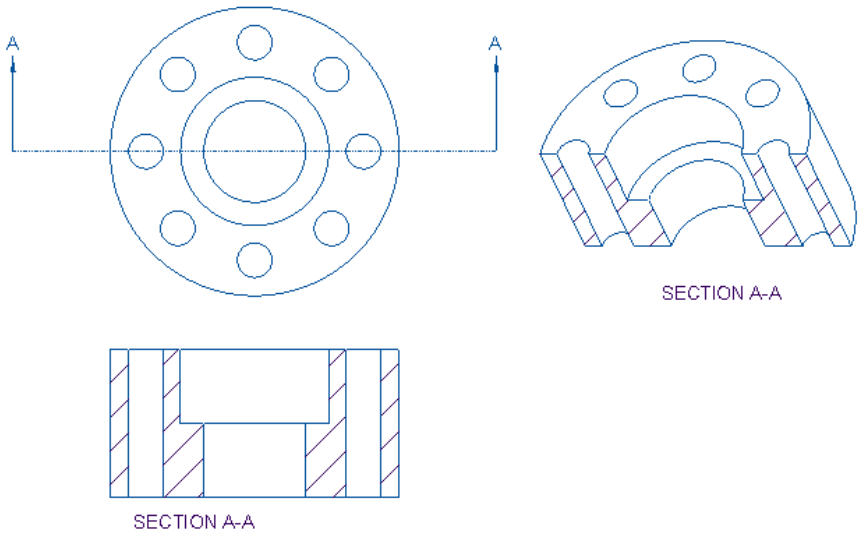


Figure 1-4 *Drawing views of the model after modifications*

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 black rounded rectangle). In an ideal case, the diameter of the hole should be equal to the diameter of the bolt.

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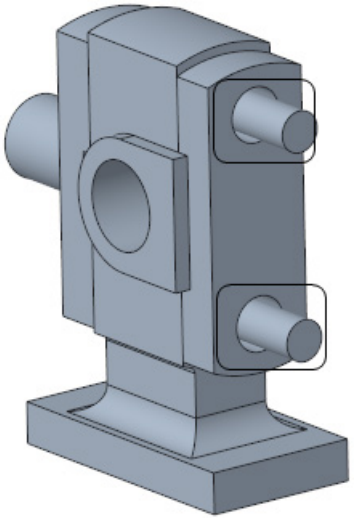


Figure 1-5 *Crosshead assembly illustrating difference in diameter of the hole and the bolt*

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.

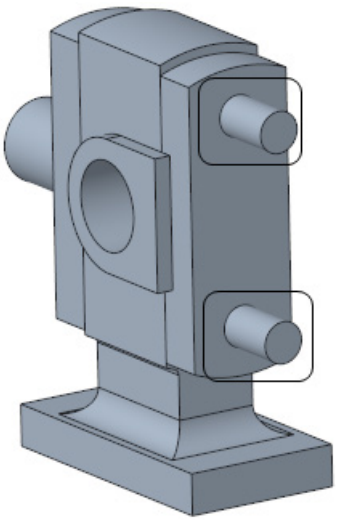


Figure 1-6 *Assembly after modifying the diameter of the hole*

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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.

SYSTEM REQUIREMENTS

The preferred system requirements for Creo Parametric are given below.

1. Operating System: Windows 7 64-bit Edition (Professional, Ultimate, and Enterprise), Windows 8.1 64-bit Edition or Windows 8.1 Pro 64-bit Edition, Windows 10 64-bit Edition (Professional, Enterprise).

2. Monitor: 1280 x 1024 (or higher) resolution support with 32-bit color.

3. Processor: 2.5 GHz minimum (Core i5 or higher).

4. Memory: 4 GB or higher.

5. Hard disk space: 5 GB minimum.

6. Microsoft TCP/IP Ethernet Network Adapter.

7. Microsoft approved 3-button mouse.

8. Microsoft Internet Explorer 6.0 or later.

9. A certified and supported graphics card.

GETTING STARTED WITH Creo Parametric

Install the Creo Parametric on your system; the shortcut icon of Creo Parametric 8.0.0.0 will automatically be created on the desktop. You can start it by double-clicking on its shortcut icon on the desktop.

Figure 1-7 shows the initial screen and Figure 1-8 shows the **Resource Centre** window of Creo Parametric. Both windows appear when you start Creo Parametric.

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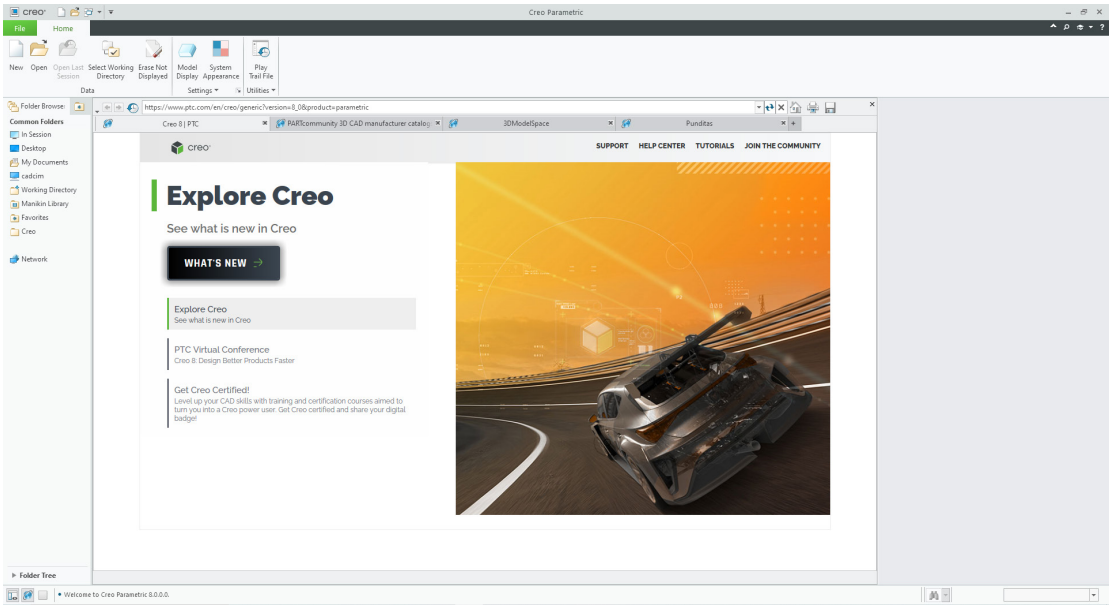


Figure 1-7 *Initial screen of Creo Parametric*

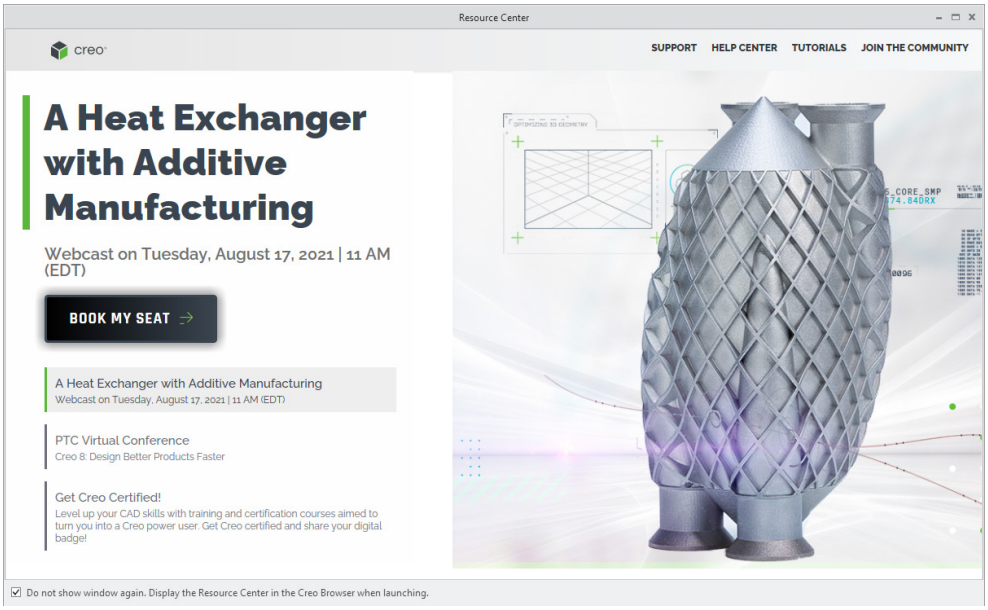


Figure 1-8 *The* ***Resource Center*** *window displayed on starting Creo Parametric*

IMPORTANT TERMS AND DEFINITIONS

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

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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.

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.



Note

In case there are conflicting dimensions and constraints, 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

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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 the following 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 next to the folders and then 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.

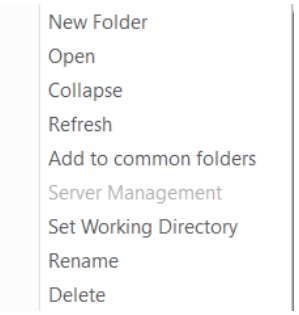


Figure 1-9 *Shortcut menu*

Using the Select Working Directory Dialog Box

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

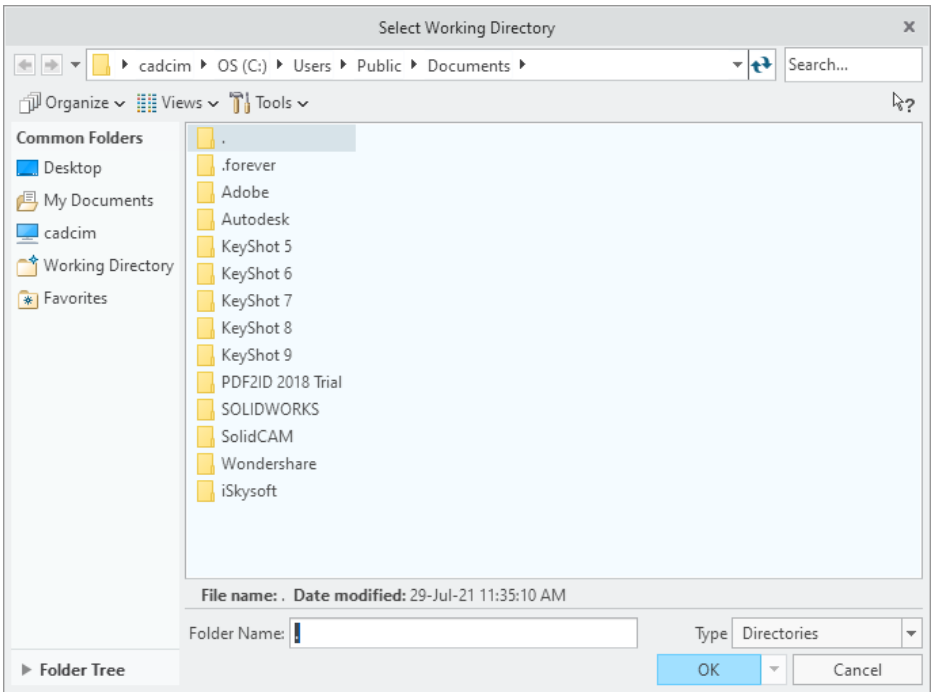


Figure 1-10 *The* ***Select Working Directory*** *dialog box*

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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.

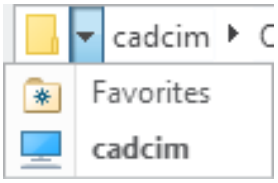


Figure 1-11 *The flyout displayed*

Folder Name

This 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

This 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.

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.

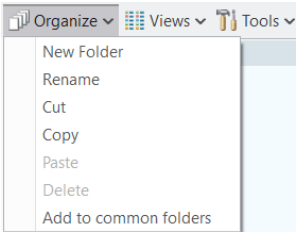


Figure 1-12 *The **Organize** flyout*

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** radio button 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** radio button 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.



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. Also, you can create a new directory using this dialog box.*

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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.

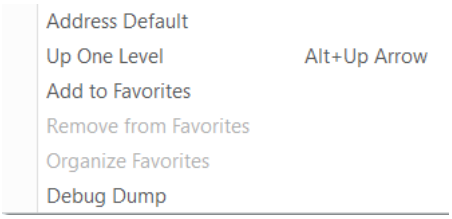


Figure 1-13 *The* ***Tools*** *flyout*

Address Default: When you choose 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.

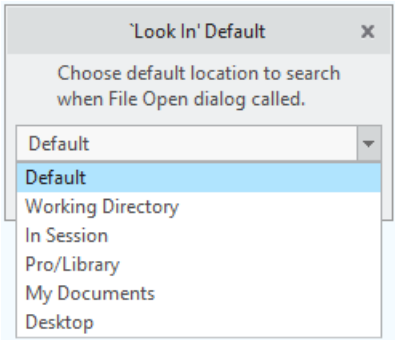


Figure 1-14 *The* ***'Look In' Default*** *dialog box with options in the drop-down list*

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 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.

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

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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 radio buttons, **Model Name** and **Markup/Instance Name**. If you select the **Model Name** radio button, the file list will be sorted out alphabetically by the model name in the **Select Working Directory** dialog box. The **Markup/Instance Name** radio button 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.

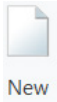
If you want to select one of the favorite working directories, then select the **Favorites** folder from the **Common Folders**; the list of all directories that were saved as favorites will be displayed in the list box. Select the required favorite directory and choose the **OK** button; the selected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



Note

*In Creo Parametric, an object can be created in different modes such as **Part**, **Drawing**, **Sketch**, and then saved as a file.*

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

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and the default name of the object file is displayed in the **File name** edit box. You can also enter a new name for the object file. Note that the name must not contain any special character and a space between characters.

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.

If the measuring units for creating models is millimeters, select **mmns_part_solid_abs** or **solid_part_mmks_abs** from the list in the **Template** area and then choose the **OK** button 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.

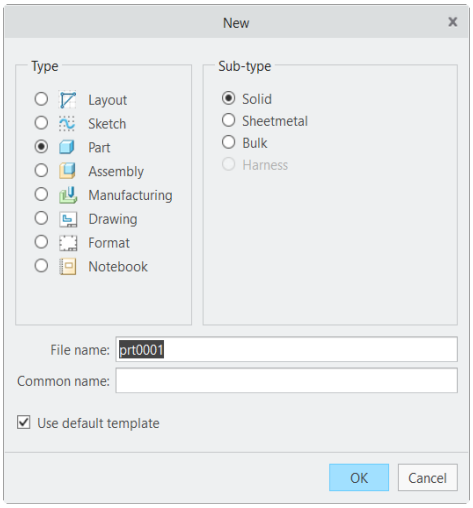


Figure 1-15 *The* ***New*** *dialog box*

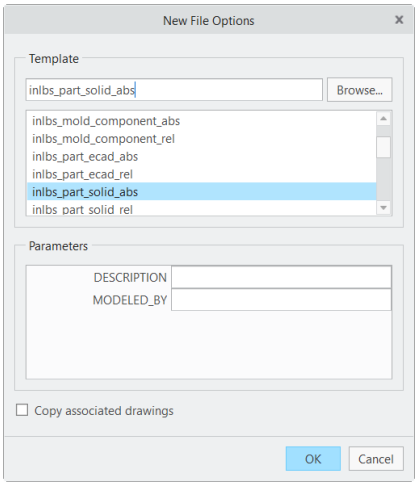


Figure 1-16 *The* ***New File Options*** *dialog box*

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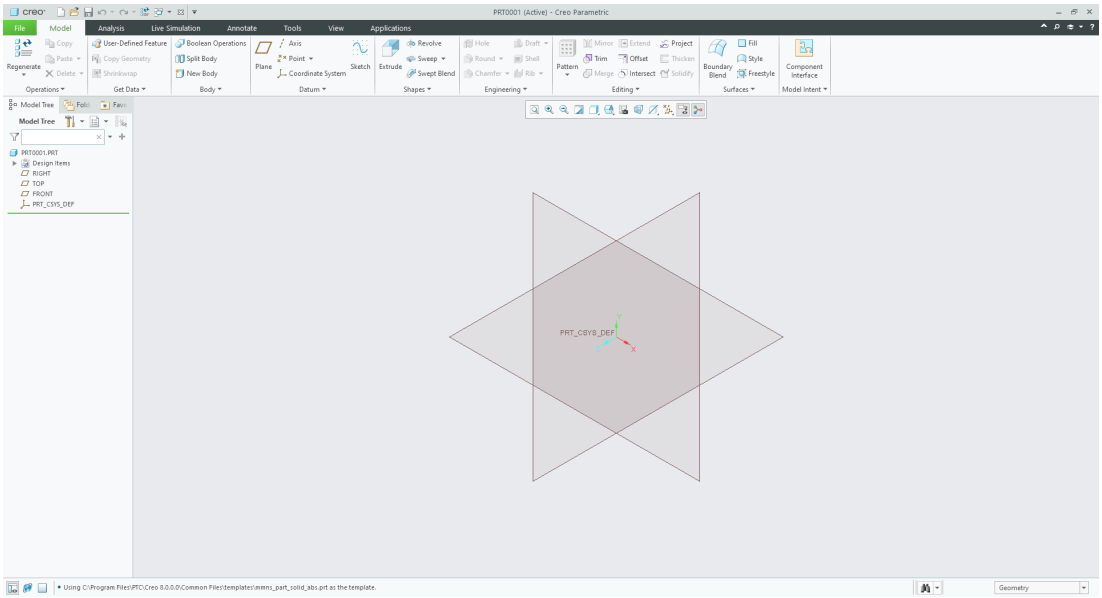


Figure 1-17 *The initial screen appearance in the* ***Part*** *mode*

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.



Note

In Creo Parametric, the name of the tools in some of the workbenches will be displayed with them only when the system screen is set to high resolution, such as 1900x1200 or 2880x1800 and so on. Otherwise only the tool icon will be displayed.

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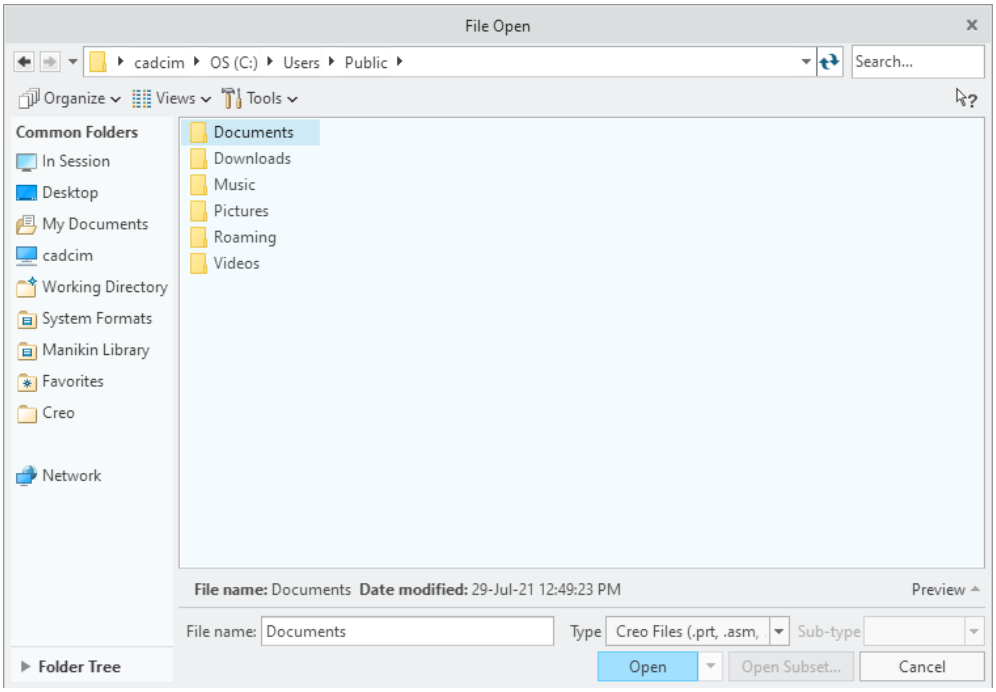


Figure 1-18 *The* ***File Open*** *dialog box*

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 chosen, displays all versions of an object file. In Creo Parametric, once a file is saved, its new version is generated with an extension that is incremented by 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 chosen, displays all instances of the object file. Select the required file and then choose the **Show Instances** option from the **Tools** flyout; all the instances 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.

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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** from 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 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 by right-clicking in the preview area.



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.

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Tip

Suppose you open an assembly that has a component named Nut. Close the assembly and now open another assembly that also 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 such type of errors, 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.

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 files.

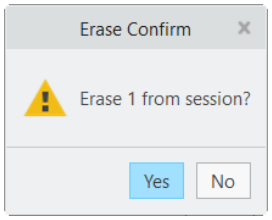


***Figure 1-19** The **Erase Confirm** message box*

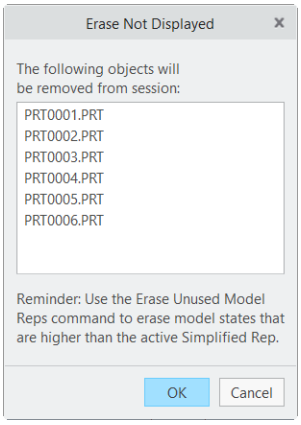


***Figure 1-20** The **Erase Not Displayed** dialog box*

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.

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Delete Old Versions

This option is used to delete all old versions of the current file. When you choose the **Delete Old Versions** option, the **Delete Old Versions** message box is displayed confirming to delete all old versions files. Choose **Yes** from the message box. All versions of that file will be deleted from the hard disk except the latest version.



Note

To delete the older versions of the model, set the directory containing the model as the working directory.

Delete All Versions

This option is used to delete all versions including the current file from the hard disk. When you choose the **Delete 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.

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 (for example, *C:\Users\Public\Documents*). The file *trail.txt* contains all the working steps used in the last session.

Trail files are editable text (.txt) files. To retrieve the data, open the file *trail.txt* in Notepad, refer to Figure 1-21. If there is more than one such file in the directory, open the latest one and save this file by choosing **File > Save As** with another name like *crash.txt* and exit.

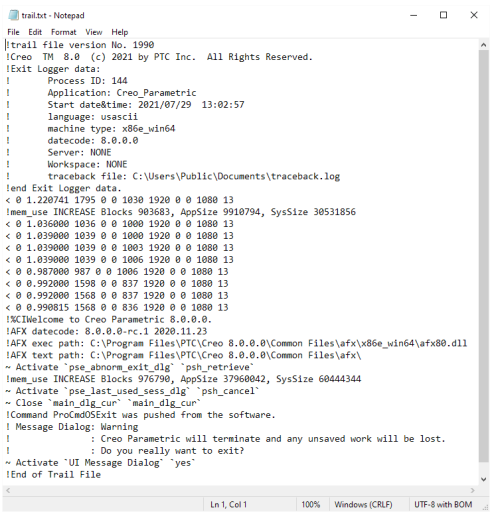


Figure 1-21 *The* ***trail.txt*** *file modified in Notepad*

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.

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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 **File 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 **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.

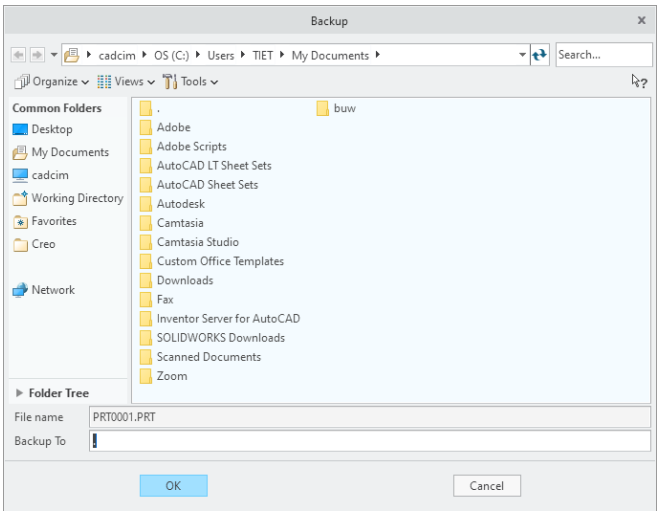


***Figure 1-22** The **Backup** dialog box*

In the **File 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.

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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 file name** edit box and choose the **OK** button.

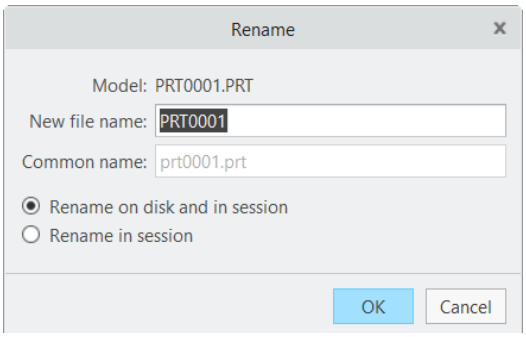


***Figure 1-23** The **Rename** dialog box*

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 file which is currently being used 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 are old versions and 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 can be done by using the **File > Manage File > Delete Old Versions** option from the menu bar.*

MENU MANAGER

The **Menu Manager** consist of a cascading menu in which a set of menus and submenus are embedded. The display of the menus depends on the task chosen.

While using the **Menu Manager**, you need to choose the **Done** or **Done Sel** option to execute and complete the selected option. 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 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.

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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 **Show Navigator** 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.

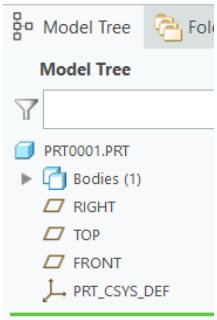


Figure 1-24 *Partial view of the* ***Model Tree***

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

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 in a different file format in Creo Parametric, the features of the model are not displayed 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 click the cursor on an item, it is highlighted in green 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.

3. Figure 1-27 shows the functions of the scroll wheel of the mouse in the 3D mode. The scroll wheel is used to spin the model in the drawing area and view it from different directions. The CTRL+scroll wheel is used to dynamically zoom in and out the view. When you press and hold the CTRL+scroll wheel and move the cursor up, the view is zoom out. When the mouse is moved down, the view is zoom in.

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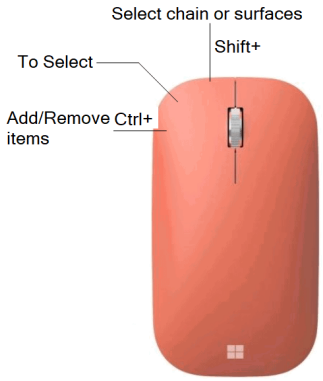


Figure 1-25 *Functions of the left mouse button*



Figure 1-26 *Functions of the right mouse button*

When you press and hold the CTRL+scroll wheel and move the mouse horizontally, the model is rotated about a point that is specified as center.

The SHIFT+scroll wheel is used to pan the object on the screen.

4. Figure 1-28 shows the functions of the scroll wheel 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.

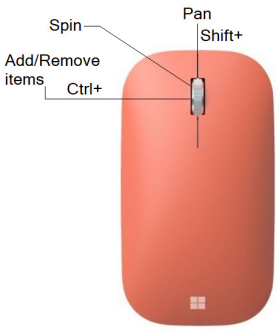


Figure 1-27 *Functions of the scroll wheel in the 3D mode*



Figure 1-28 *Functions of the scroll wheel in the **Sketch** mode*

The scroll wheel is used to pan view in the **Sketch** mode and the **Drawing** mode.



Note

*When you spin the model with the **Spin Center** button, which is located in the Graphics bar, turned on, the model rotates about the spin center origin. If this button is turned off, then the model rotates about the specified point.*

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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 interface 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 tools will be enabled only after you create a part or open an existing file. However, 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.

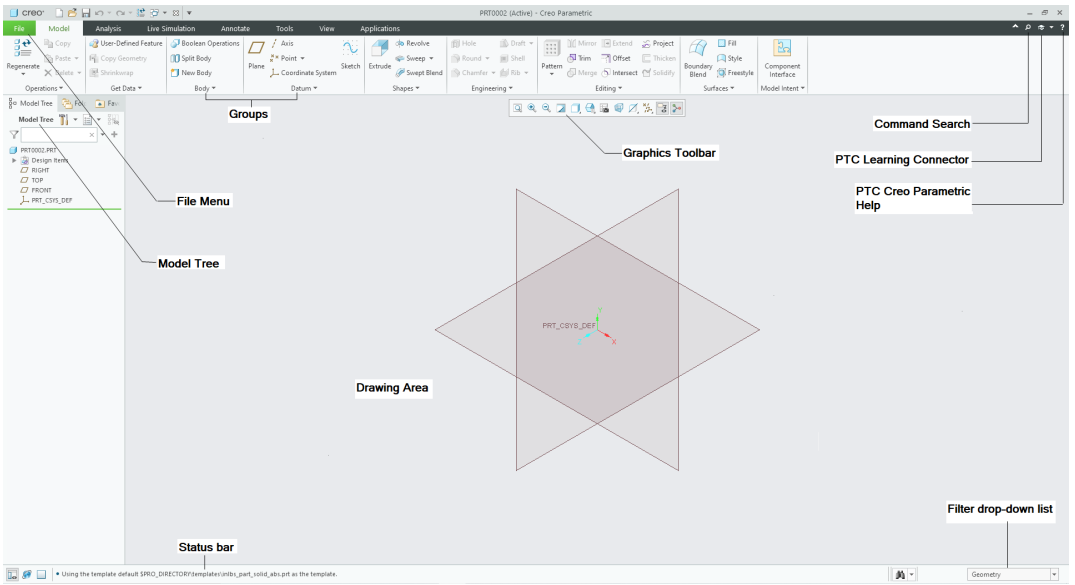


Figure 1-29 *Default interface components*

TOOLBARS

In Creo Parametric, there are two toolbars, **Quick Access** and **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 **Graphics** toolbar.

Graphics Toolbar

The **Graphics** toolbar, shown in Figure 1-30, has twelve 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.The **Perspective View** button toggles the perspective view. Using the **Saved Orientations** drop-down, you can change the view.

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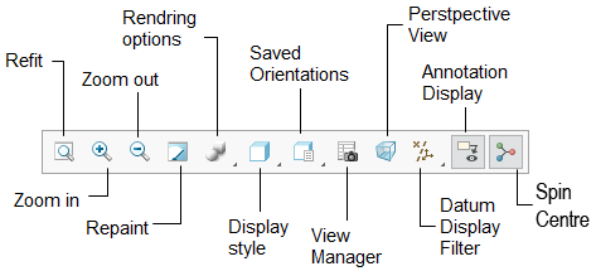


Figure 1-30 *The* ***Graphics*** *toolbar*

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 to on/off.

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 will be displayed. Using this dialog box, you can configure various parameters of Creo Parametric. You can also customize the **Ribbon** interface.

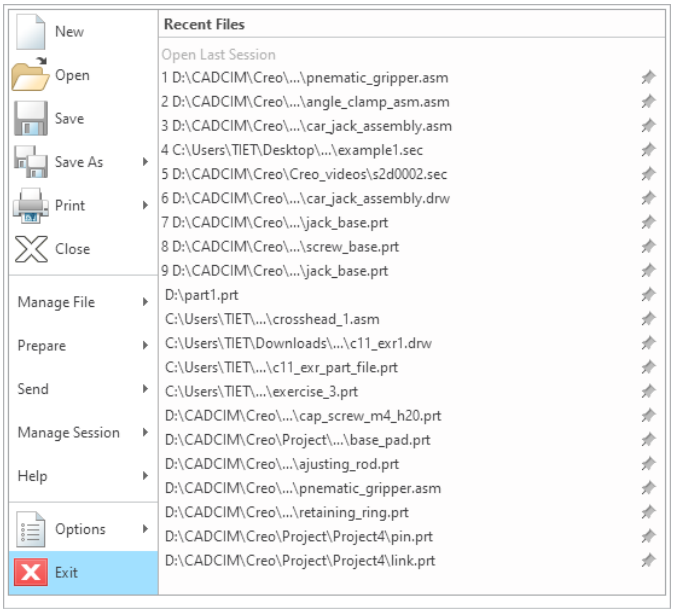


Figure 1-31 *The* ***File*** *menu*

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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. Choose the **New Group** or **New Tab** option, depending on your requirement, from the **New** drop-down list; the new group or tab will be added in the list box on the right side of the dialog box. Now, choose the button to be added in the new group from the list box on the left side of dialog box and then choose the **Adds selected item to the Ribbon** option from the dialog box; the selected button will be added to the group, refer to Figure 1-32.

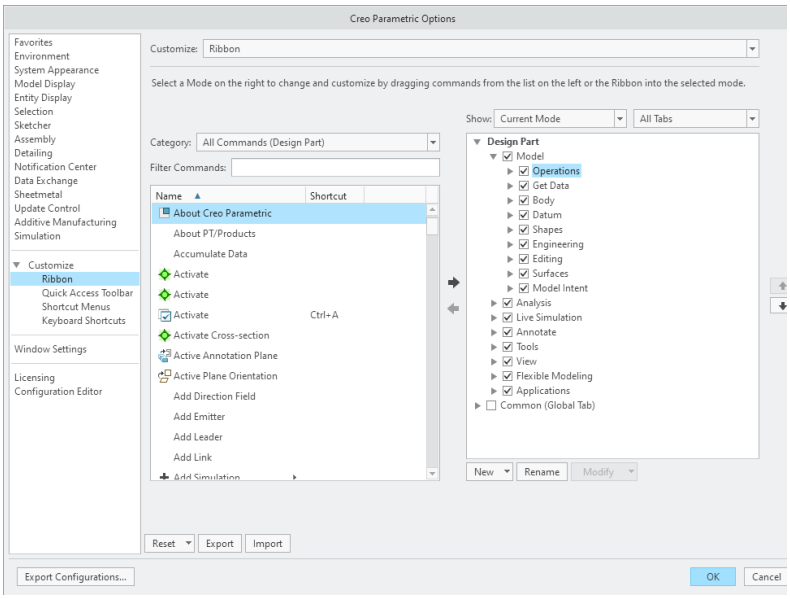


Figure 1-32 *Customizing the* ***Ribbon***

Creo Parametric Help

When you choose the **Creo Parametric Help** button, you are directed to **support.ptc.com** and the online Help page is displayed in your default browser. In this page, you can search for any help topic on Creo. 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, refer to 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*

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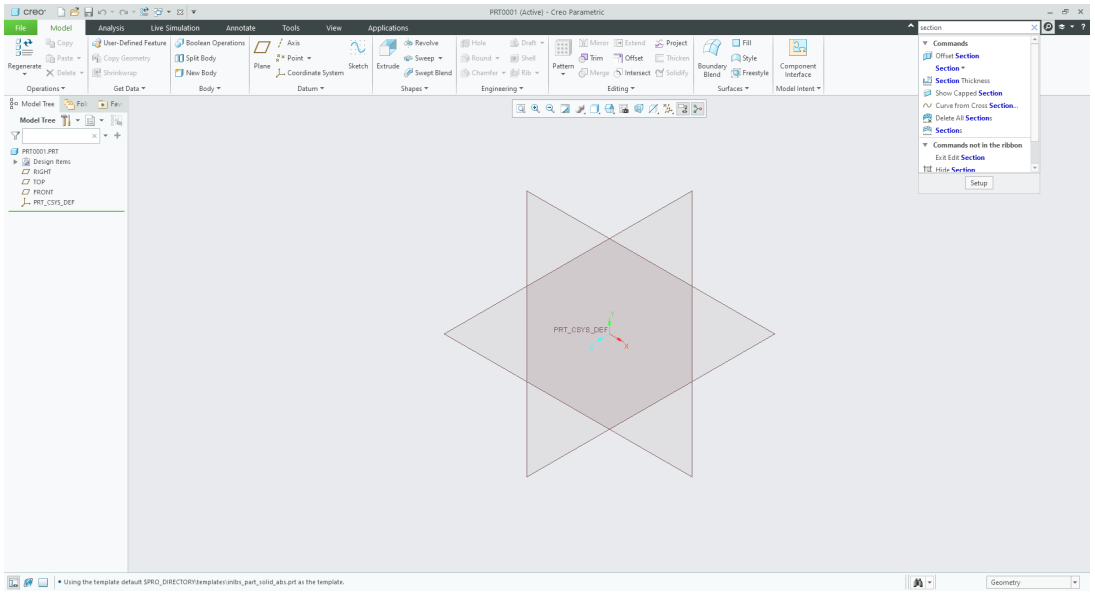


*Figure 1-34 The **Command Search** button*

PTC Learning Connector Button

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

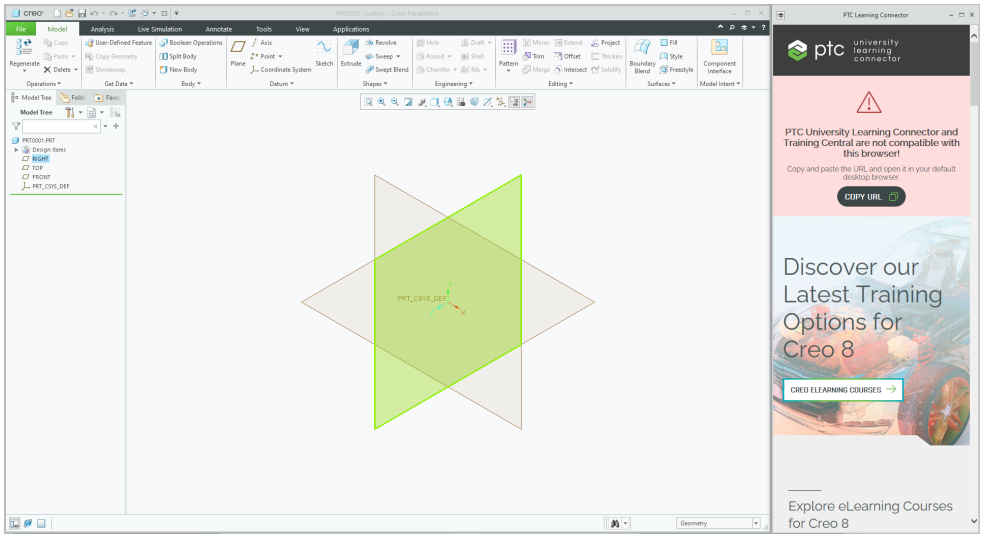


*Figure 1-35 Creo Parametric 8.0 screen with the **PTC Learning Connector** window*

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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 a blue arrow will appear on the left of the unit system selected.*

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 **Show Navigator** 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:

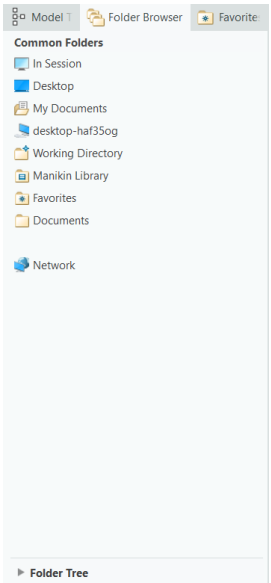


Figure 1-36 *Partial view of the navigator*

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.

Creo Parametric BROWSER

The Creo Parametric browser is present on the right of the navigator. You can also stretch the browser window horizontally. When the Creo Parametric session starts, the browser is displayed on the screen. The browser has four tabs. By default, the homepage is linked to *https://www.ptc.com/en/creo/generic?version=8_0&product=parametric*, the second tab is linked to *ptc.partcommunity.com*, the third tab is linked to *www.3dmodelspace.com/ptc*, and the last one is *home.punditas.com*. 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.

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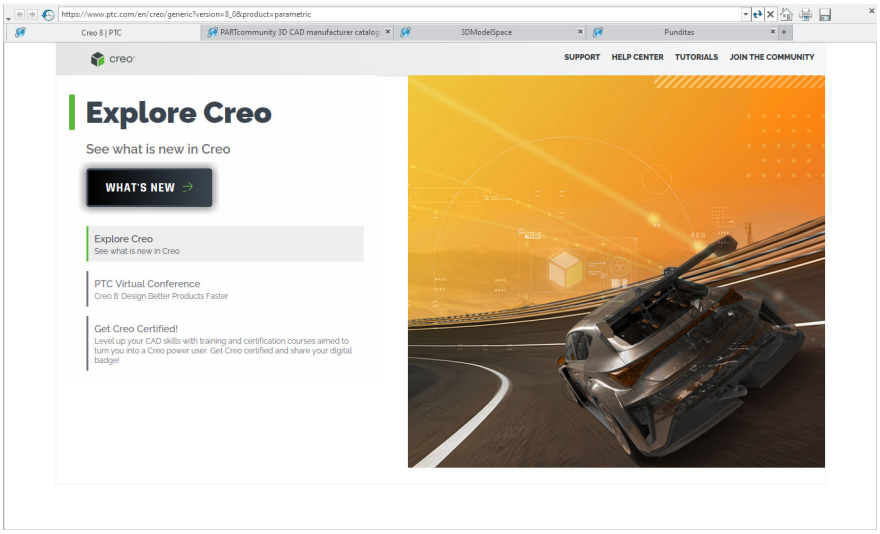


Figure 1-37 *Creo Parametric browser*

The Creo Parametric browser can be used for various purposes.

1. It is used to preview the online Creo Parametric files and browse the file system.

2. Using the browser, you can open a Creo Parametric file by double-clicking on it 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 using the browser.

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 materials has been added in Creo Parametric. You can also add shadows and reflections to the background of the model by using the appearance gallery. 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 **View** 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.

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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.

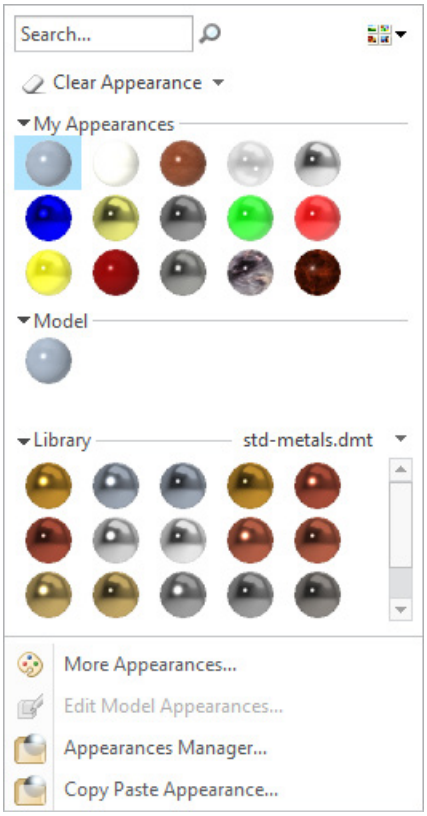


Figure 1-38 *Appearance Gallery*

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. In this flyout, the **Show Tooltips** check box is selected 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; the **Confirm** message box will be displayed. Choose the **Yes** button from the message box to clear appearances.

My Appearance

This 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 **My 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.

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Model

This 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 from the **Appearance Gallery**; the **Appearance Editor** will be displayed, as shown in Figure 1-39. The **Appearance Editor** provides options for editing the appearance as well as for advanced rendering, thereby providing much more realistic images.

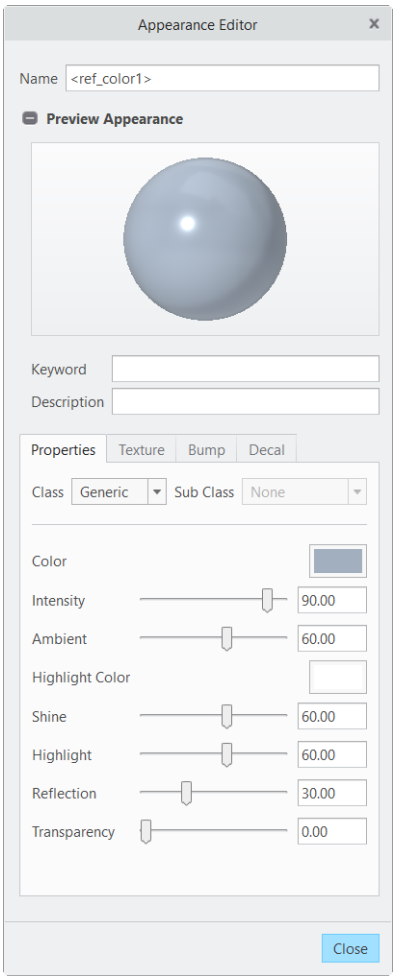


Figure 1-39** The **Appearance Editor

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 the tabs of this dialog box. The **Properties** tab provides 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.

You can use the **Texture** tab 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

On choosing the **Edit Model Appearances** button; the **Model Appearance Editor** will be displayed. In this editor, all appearances used in the model will be displayed along with their

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properties. The **Model Appearance Editor** is same as the **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 right panel. Also, you can open an existing library file and 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 the solid model or the 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 **Render Studio** tool from the **Rendering** group of the **Applications** tab; the **Render Studio** tab will be invoked. Next, choose the **Scenes** tool from the **Project** group of the **Render Studio** tab; the **Scenes** slide-down panel will be displayed. In the panel, choose the **Edit Scene** option to invoke the **Scene Editor** dialog box, 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 **Gallery** area of the **Scene Editor** dialog box.

Modifying the Scene Environment

The environment of a scene can be modified using the options available in the **Environment** tab of the **Scene Editor** dialog box. You can also apply shadows and reflections to the model by selecting the **Shadow** and **Reflection** check boxes in the **Floor Effects** area of the **Scene Editor** dialog box.

Applying Lights

The next step is to apply the lighting effects on the model. By adding lights to the model, you

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can illuminate the scene from various angles. To add lights, choose the **Lights** tab from the **Scenes Editor** dialog box. You can create a light source as well as position it 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.

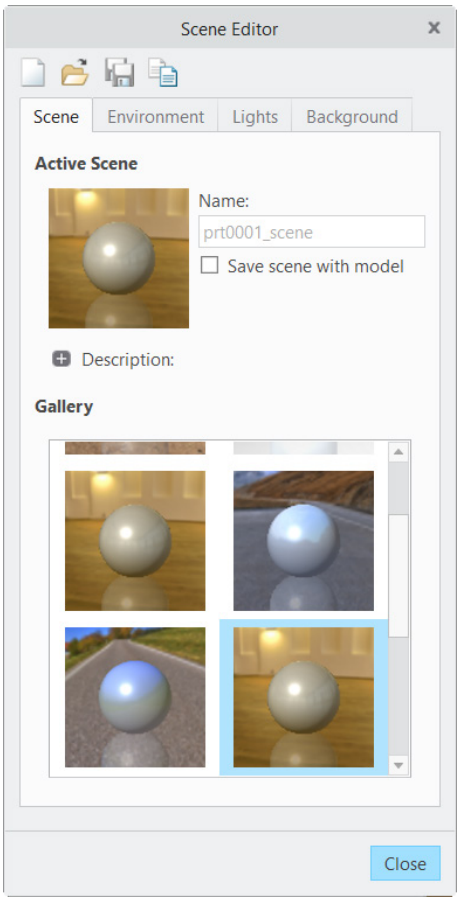


Figure 1-40 *The* ***Scene Editor*** *dialog box*

Adding Effects

Finally, you can add the surrounding effects such as ray bounces, indirect bounces, shadow quality, caustics, illumination, and self shadows. To do so, choose the **Real-Time Settings** option from the **Real-Time** drop-down of the **Render Studio** tab; the **Real-Time Rendering Settings** dialog box will be displayed. Using the options available in this dialog box, you can add desired effects to the model.

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 **Project** group in the **Render Studio** tab.

To modify the settings of the perspective view, invoke the **View** dialog box by choosing the **Reorient** tool from the **Saved Orientations** flyout of the **Graphics** toolbar. Next, choose the **Perspective** tab in the dialog box and specify the settings for the perspective view.

Rendering the Model

The last step is to render the model. To render the model, choose the **Render** button from the **Render Output** group in the **Render Studio** tab; the **Render** dialog box will be displayed. Set the required rendering options in this window and choose the **Render** button to start the rendering process. If required, you can 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 ***Real-Time Rendering*** *button from the* ***Real-Time*** *group of the* ***Render Studio*** *tab.*

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 at some places for the purpose of printing and image clarity. However,

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for better understanding and clear visualization at various places, you can 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** dialog box will be displayed. Select the **System Appearance** category from the left pane of this window. Next, choose the **Light (Previous Creo Default)** option from the **System Colors** drop-down list; different object names will be displayed with color drop-downs adjacent to them, refer to Figure 1-41. Next, choose the **OK** button to apply the color scheme.

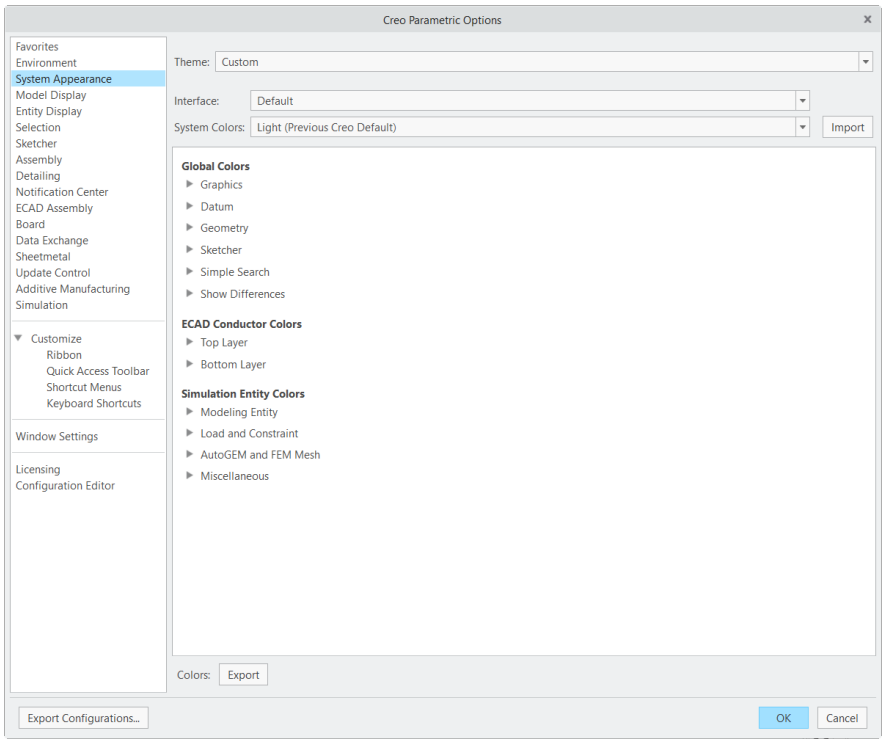


Figure 1-41 *The* ***System Colors*** *category in the* ***Creo Parametric Options*** *dialog box*

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. Which one is the default theme of Creo Parametric?

   a) Light (Previous Creo Default) b) Default
   c) Black on White d) White on Black

2. For which of the following purposes is the **Play Trail File** option used?

   a) Recover Data b) Play Tutorial Video
   c) Browse Files c) None of the above

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3. You can change the system appearance from the_____ dialog box.

4. The _____ stores and display all features of a current model.

5. There is a bidirectional associativity between all modes of Creo Parametric. (T/F)

6. In Creo Parametric, you can use the SHIFT+scroll wheel to pan the object on the screen. (T/F)

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Answers to Self-Evaluation Test

1. b, **2. a**, **3. Creo Parametric Options**, **4. Model Tree**, **5.** T, **6.** T