Chapter 1

Introduction to NX 8.5

Learning Objectives

After completing this chapter, you will be able to:

- *Understand different environments in NX.*
- *Understand the system requirements for NX.*
- *Start a new file in NX.*
- *Understand the important terms and definitions used in NX.*
- *Understand functions of the mouse buttons.*
- *Identify different types of toolbars in NX.*
- *Understand the use of various hot keys.*
- *Modify the color scheme in NX.*
- *Learn about various dialog boxes in NX.*

INTRODUCTION TO NX 8.5

Welcome to NX 8.5 (commonly referred to as NX). As a new user of this software package, you will join hands with thousands of users of this high-end CAD/CAM/CAE tool. If already familiar with the previous releases, you can upgrade your designing skills with tremendous improvement in this latest release.

NX 8.5, a product of SIEMENS Corp., is a completely re-engineered, next-generation family of CAD/CAM/CAE software solutions for Product Life Cycle Management. Through its exceptionally easy-to-use and state-of-the-art user interface, NX delivers innovative technologies for maximum productivity and creativity, from the basic concept to the final product. NX reduces the learning curve, as it allows the flexibility of using feature-based and parametric designs.

The subject of interpretability offered by NX includes receiving legacy data from the other CAD systems and even between its own product data management modules. The real benefit is that the links remain associative. As a result, any changes made to this external data are notified to you and the model can be updated quickly.

When you open an old file or start a new file in NX, you will enter the Gateway environment. It allows you to examine the geometry and drawing views that have been created. In the Gateway environment, you can invoke any environment of NX.

NX serves the basic design tasks by providing different environments. An environment is defined as a specified environment, consisting of a set of tools, which allows the user to perform specific design tasks in a particular area. You need to start the required environment after starting a new part file. As a result, you can invoke any environment of NX in the same working part file. The basic environments in NX are the Modeling environment, Shape Studio environment, Drafting environment, Assembly environment, and the Manufacturing environment. These environments are discussed next.

Modeling Environment

The Modeling environment is a parametric and feature-based environment, in which you can create solid models. The basic requirement for creating solid models in this environment is a sketch. This sketch is drawn in the Sketch task environment. The Sketch task environment can be invoked by choosing the **Sketch in Task** button from the **Feature** toolbar. You can draw the sketch using various tools of the Sketch task environment. In NX, you can also draw a sketch directly in the Modeling environment by using the tools available in the **Direct Sketch** toolbar that is displayed at the bottom of the screen in the Modeling environment. While drawing a sketch, various applicable constraints and dimensions are automatically applied to it. You can also apply additional constraints and dimensions manually. After drawing the sketch, you need to convert the sketch into a feature. The tools to convert a sketch into a feature are available in the Modeling environment. You can also create placed features such as fillets, chamfers, taper, and so on by using other tools available in this environment. These features are called the placed features. You can also assign materials to the model in the Modeling environment.

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Shape Studio Environment

The Shape Studio environment is also a parametric and feature-based environment, in which you can create surface models. The tools in this environment are similar to those in the Modeling environment. The only difference is that the tools in this environment are used to create basic and advanced surfaces. You are also provided with the surface editing tools, which are used to manipulate the surfaces to obtain the required shape. This environment is useful for conceptual and industrial design.

Assembly Environment

The Assembly environment is used to assemble the components using the assembly constraints available in this environment. There are two types of assembly design approaches in NX, Bottom-up and Top-down.

In the bottom-up approach of the assembly, the previously created components are assembled together to maintain their design intent. In the top-down approach, components are created in the Assembly environment.

You can also assemble an existing assembly with the current assembly. The Check Clearance analysis provides the facility to check the interference between the components in an assembly.

Drafting Environment

The Drafting environment is used for the documentation of the parts or assemblies created earlier in the form of drawing views and their detailing. There are two types of drafting techniques, Generative drafting and Interactive drafting.

The generative drafting technique is used to automatically generate the drawing views of the parts and assemblies. The parametric dimensions added to the component in the Modeling environment during its creation can also be generated and displayed automatically in the drawing views. The generative drafting is bidirectionally associative in nature. If you modify the dimensions in the Drafting environment, the model will automatically update in the Modeling environment and vice-versa. You can also generate the Bill of Material (BOM) and balloons in the drawing views.

In interactive drafting, you need to create the drawing views by sketching them using the normal sketching tools and then adding the dimensions.

SYSTEM REQUIREMENTS

The following are the system requirements to ensure the smooth running of NX:

- **Operating System**: Windows Vista, Windows Vista X 64 bit (Professional, Ultimate, and Enterprise Editions), Windows 7 or UNIX.
- **Memory**: 2 GB of RAM is the minimum requirement for all applications and 4GB of RAM is recommended for DMU applications.
- **Disk drive**: 10 GB Disk Drive space (Minimum recommended size)

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- **Internal/External drives**: A DVD-ROM drive is required for the program installation.
- **Display**: A graphic color display compatible with the selected platform-specific graphic adapter is required. The minimum recommended monitor size is 17 inches.
- **Graphics adapter**: A graphics adapter with a 3D OpenGL accelerator is required with a minimum resolution of 1024x768 for Microsoft Windows workstations and 1280x1024 for UNIX workstations.

GETTING STARTED WITH NX

Install NX on your system and then start it by double-clicking on the shortcut icon of NX 8.5 on the desktop of your computer. Alternatively, you can choose **Start > All Programs > Siemens NX 8.5 > NX 8.5** from the taskbar to start NX, as shown in Figure 1-1.

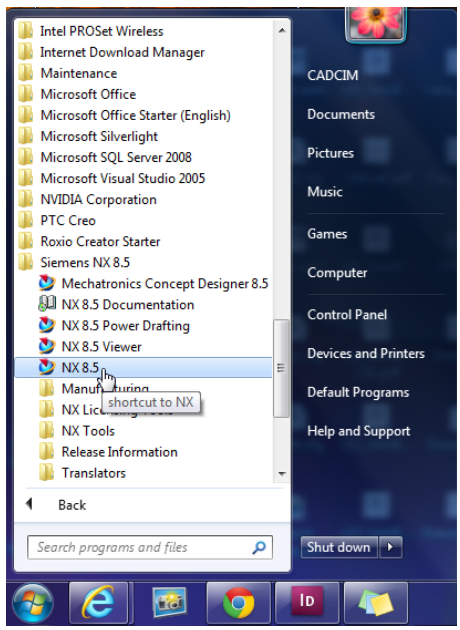

Figure 1-1 *Starting NX 8.5 using the taskbar*

After the system has loaded all the required files to start NX, the initial screen will be displayed, as shown in Figure 1-2.

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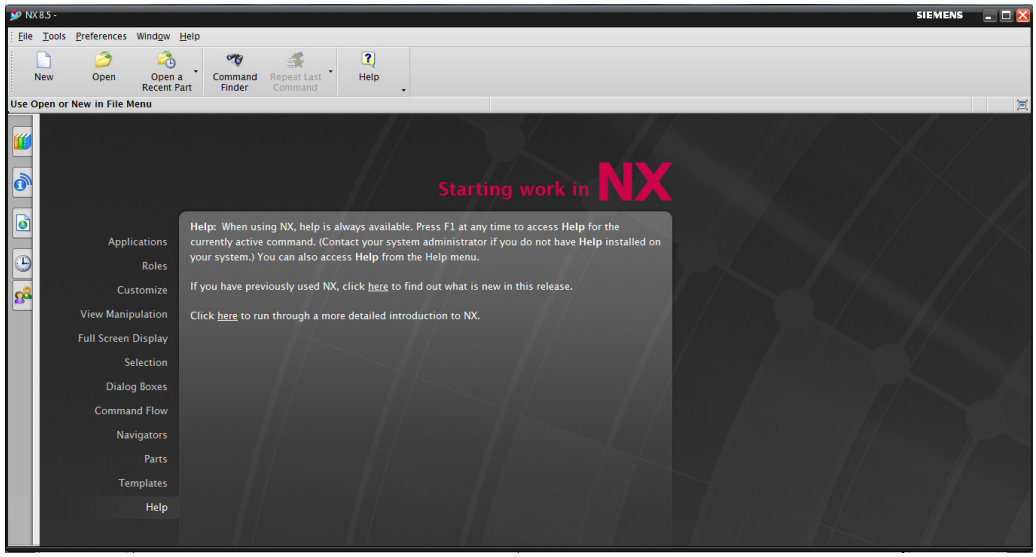

Figure 1-2 *The initial screen that appears after starting NX 8.5*

Choose **File > New** from the menu bar; the **New** dialog box will be displayed. Make sure that **Model** template is selected in the **Templates** rollout of the dialog box. Next, enter the name of the file in the **Name** edit box and choose the **OK** button; the Modeling environment will be displayed on the screen, refer to Figure 1-3.

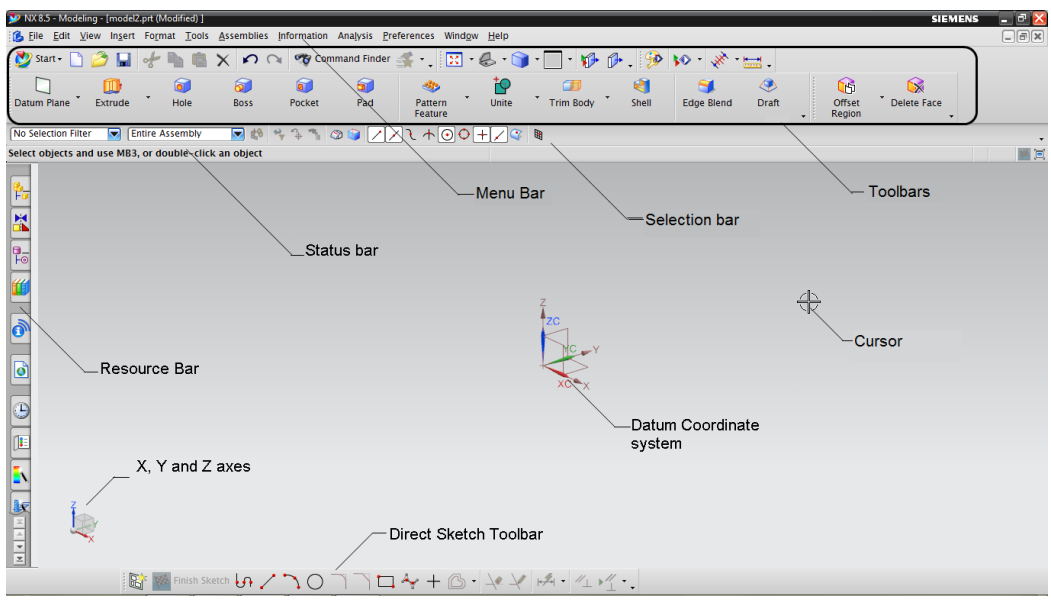

Figure 1-3 *The Modeling environment displayed on screen*

In this textbook, the Model template has been used to create a new file. The procedure for starting a new file using the Model template is discussed in the next chapter.

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IMPORTANT TERMS AND DEFINITIONS

Some important terms and definitions of NX are discussed next.

Feature-based Modeling

A feature is defined as the smallest building block that can be modified individually. A model created in NX is a combination of a number of individual features and each feature is related to the other directly or indirectly. If a proper design intent is maintained while creating the model, then these features automatically adjust their values to any change in their surroundings. This provides a great flexibility to the design.

Parametric Modeling

The parametric nature of a software package is defined as its ability to use the standard properties or parameters in defining the shape and size of a geometry. The main function of this property is to derive the selected geometry to a new size or shape without considering its original dimensions. You can change or modify the shape and size of any feature at any stage of the designing process. This property makes the designing process an easy task. For example, consider the design of the body of a pipe housing, as shown in Figure 1-4.

To change the design by modifying the diameter of the holes and their number on the front, top, and bottom face, you need to select the feature and change the diameter and the number of instances in the pattern. The modified design is shown in Figure 1-5.

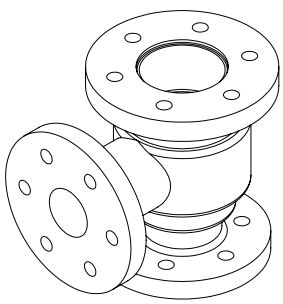

Figure 1-4 *Body of a pipe housing*

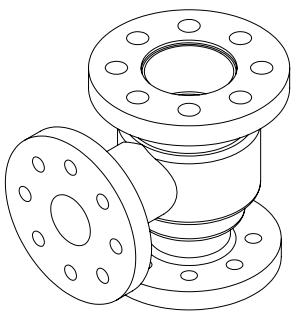

Figure 1-5 *Modified body of the pipe housing*

Bidirectional Associativity

As mentioned earlier, NX has different environments such as the Modeling environment, Assembly environment, and the Drafting environment. The bidirectional associativity that exists between all these environments ensures that any modification made in the model in any of the environments of NX, is automatically reflected in the other environments immediately. For example, if you modify the dimension of a part in the Modeling environment, the change will be reflected in the Assembly and the Drawing environments as well. Similarly, if you modify the dimensions of a part in the drawing views generated in the Drafting environment, the changes will be reflected in the Modeling and Assembly environments. Consider the drawing views of the pipe housing shown in Figure 1-6. When you modify the model in the Modeling environment, the changes will be reflected in the Drafting environment automatically. Figure 1-7

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shows the drawing views of the pipe housing after increasing the diameter and the number of holes.

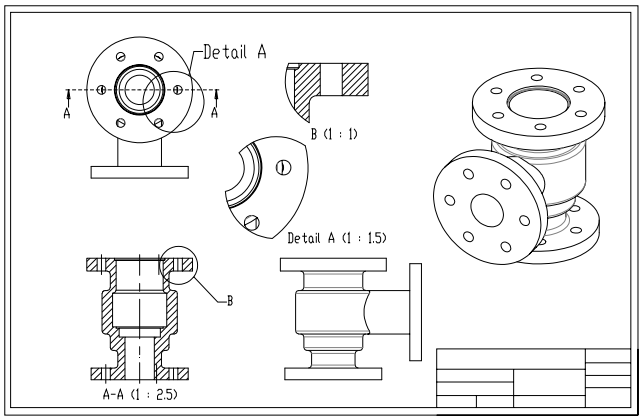

Figure 1-6 *The drawing views of a pipe housing*

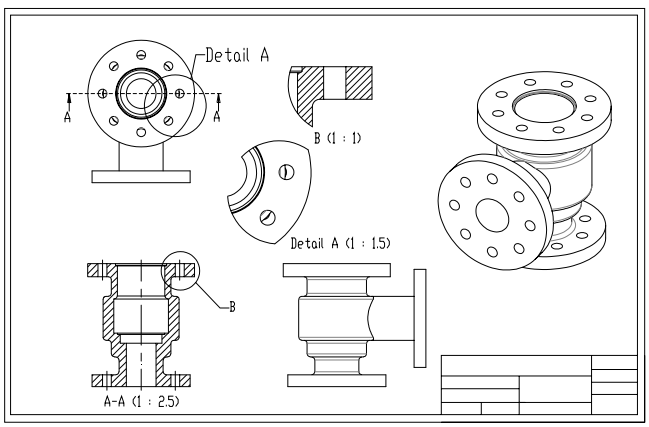

Figure 1-7 *The drawing views of pipe housing after making the modifications*

*.prt

*.prt is a file extension associated with all the files that are created in the Modeling, Shape Studio, Assembly, Sheet Metal, and Drafting environments of NX.

Resource Bar

The **Resource Bar** combines all navigator windows, a history palette, an integrated web browser, and a parts template in one common place for a better user interface. By default, the **Resource Bar** is located on the left side of the NX window.

Roles

Roles are a set of system customized tools and toolbars used for different applications. In NX, you have different roles for different industrial applications. The **Roles** tab in the **Resource Bar**

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is used to activate the required role. In this book, the **Essentials with full menus** role has been used, as it contains all the required tools. To activate this role, choose the **Roles** tab from the **Resource Bar** and click on the **System Defaults** option, if it is not expanded already; a flyout will be displayed. Click on the **Essentials with full menus** icon to activate that role. Figure 1-8 shows the **Roles** navigator that appears when you choose the **Roles** tab in the **Resource Bar**.

Part Navigator

The **Part Navigator** keeps a track of all the operations that are carried out on the part. Figure 1-9 shows the part navigator that appears when you choose the **Part Navigator** tab in the **Resource Bar**.

Constraints

Constraints are the logical operations that are performed on the selected element to define its size and location with respect to the other elements or reference geometries. There are three types of constraints in NX: Geometric, Dimensional, and Assembly. The constraints in the sketch are called geometric and dimensional constraints and these are used to precisely define the size and position of the sketched elements with respect to the surroundings. The assembly constraints are available in the Assembly environment and are used to define the precise position of the components in the assembly. These constraints are discussed next.

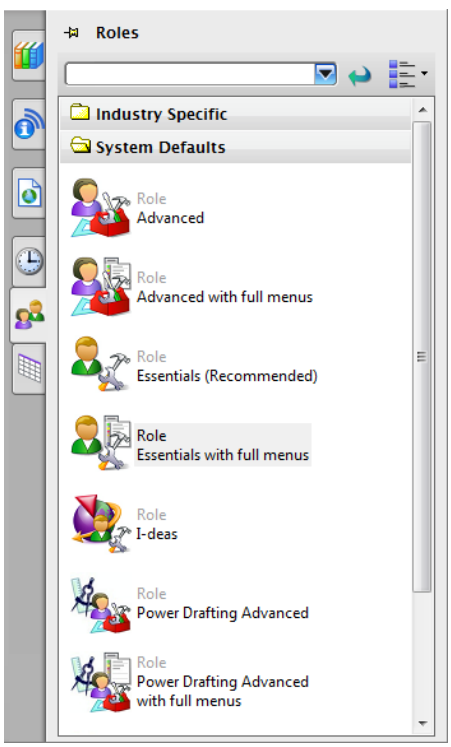

*Figure 1-8 The **Roles** navigator*

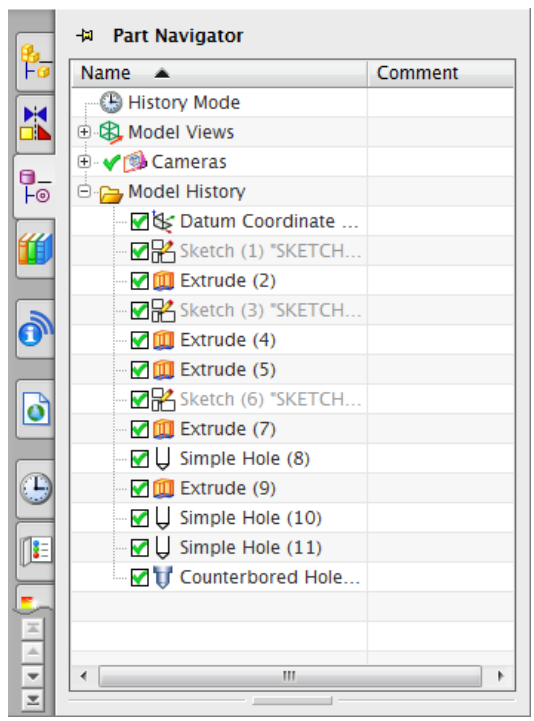

*Figure 1-9 The **Part Navigator***

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Geometric Constraints

These are the logical operations performed on the sketched elements to define their size and position with respect to the other elements. Geometric constraints are applied using two methods, automatic constraining and manual constraining. While drawing the sketch, some constraints are automatically applied to it.

Dimensional Constraints

After creating the sketch, you need to apply different types of dimensional constraints to it. Various types of dimensions in NX are:

1. Horizontal Dimensions
2. Vertical Dimensions
3. Parallel Dimensions
4. Perpendicular Dimensions
5. Angular Dimensions
6. Diameter Dimensions
7. Radius Dimensions
8. Perimeter Dimensions

NX is a parametric software and therefore, you can modify the dimensions of sketch at any time. You will learn more about modifying dimensions of the sketch in later chapters.

Assembly Constraints

The constraints in the Assembly environment are the logical operations performed to restrict the degrees of freedom of the component and to define its precise location and position with respect to the other components of the assembly.

Solid Body

The solid body contains all the features such as extrude, pad, pocket, hole, and so on.

Sheet Body or Surfaces

Surfaces are geometric features that have no thickness. They are used to create complex shapes that are difficult to be created using the solid features. After creating the surface, you can assign a thickness to it in order to convert it into a solid body. Surfaces are created in the Modeling environment. No separate environment is required to create the surfaces.

Features

A feature is defined as the basic building block of a solid model. The combination of various features results in a solid body. In the Modeling environment of NX, the features are of two types:

1. Sketch-based features
2. Placed-features

The sketch-based features are the ones that require a sketch for their creation. The placed-features do not require a sketch to create them.

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WCS (Work Coordinate System)

The WCS is a local coordinate system and can be repositioned to a convenient location while making a model. The XC-YC plane of the WCS is used to perform many operations. When you create a new file, by default, the WCS is positioned at the Datum Coordinate System origin, which is (0,0,0). By default, the display of WCS is turned off. To turn on the display of WCS, choose the **Display WCS** button from the **Utility** toolbar; the WCS will be displayed at the origin of the drawing window.

UNDERSTANDING THE FUNCTIONS OF THE MOUSE BUTTONS

To work in the NX environment, it is necessary that you understand the functions of the mouse buttons. The efficient use of these three buttons, along with the CTRL key, can reduce the time required to complete the design task. The different combinations of the CTRL key and the mouse buttons are listed below:

1. The left mouse button is to make a selection by simply selecting a face, surface, sketch, or an object from the geometry area or from the **Part Navigator**. For multiple selections, select the entities using the left mouse button.

2. The right mouse button is used to invoke the shortcut menu, which has different options such as **Zoom**, **Fit**, **Rotate**, **Pan**, and so on.

3. Press and hold the middle and the right mouse buttons to invoke the **Pan** tool. Next, drag the mouse to pan the model. You can also invoke the **Pan** tool by first pressing and holding the SHIFT key and then the middle mouse button. Figure 1-10 shows the use of a three button mouse in performing the pan functions.

4. Press and hold the middle mouse button to invoke the **Rotate** tool. Next, drag the mouse to dynamically rotate the view of the model in the geometry area and view it from different directions. Figure 1-10 shows the use of the three button mouse in performing the rotate operation.

5. Press and hold the CTRL key and then the middle mouse button to invoke the **Zoom** tool. Alternatively, press and hold the left mouse button and then the middle mouse button to invoke the **Zoom** tool. Next, drag the mouse dynamically to zoom in or out the model in the geometry area. Figure 1-10 shows the use of the three mouse buttons in performing the zoom functions.

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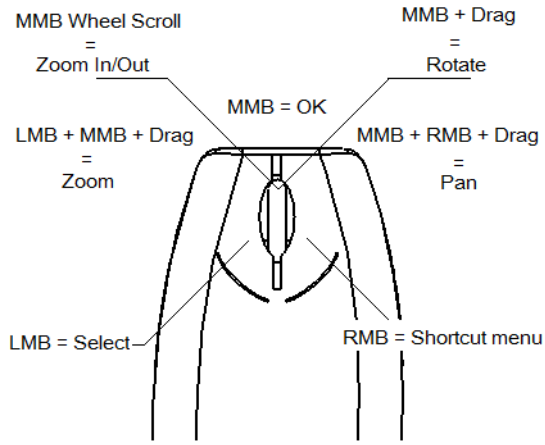

Figure 1-10 *Functions of the mouse buttons*

TOOLBARS

NX offers a user-friendly design environment by providing specific toolbars for each environment. Therefore, it is important that you get acquainted with various standard toolbars and buttons that appear in different environments of NX. These toolbars are discussed next.

Application Toolbar

This toolbar is common to all environments of NX. Figure 1-11 shows the **Application** toolbar. You can invoke any other environment from the currently invoked environment. For example, you can invoke the Assembly and Drafting environments from the Modeling environment using this toolbar to complete the design.



Figure 1-11 *The **Application** toolbar*



Tip: *By default, some of the toolbars are available in their respective environments. However, you can add more toolbars to an environment. To do so, right-click on any toolbar; a shortcut menu will be displayed. You will observe that the tools that are not available in the graphics window are not selected in the shortcut menu. Select any unselected toolbar; it will become available in the graphics window.*

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Standard Toolbar

This toolbar is common to all environments of NX. Figure 1-12 shows the **Standard** toolbar. The buttons in this toolbar are used to start a new file, open an existing file, save a file, and print the current document. These buttons are also used to cut and place the selection on a temporary clipboard, copy a selection, paste the content from the clipboard to a selected location, undo, redo, search a tool, and invoke the help topics. The **Start** button in this toolbar is used to invoke different NX environments.



Figure 1-12 *The* ***Standard*** *toolbar*

Status Bar

The status bar that appears at the top of the drawing window comprises of two areas, as shown in Figure 1-13. These areas are discussed next.

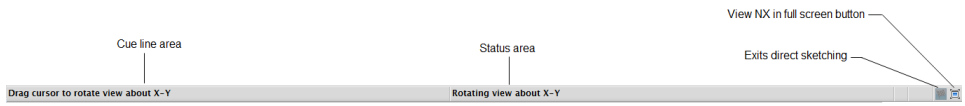

Figure 1-13 *The* ***Status*** *bar*

Cue Line Area

The cue line area is the prompt area. In this area, you will be prompted to select the entities for completing the tool task.

Status Area

This area gives information about the operations that can be carried out.

View NX in Full Screen

If you choose this button, the graphic area will be maximized and it gives you full screen display. For getting the default screen display, you need to choose this button again.

Modeling Environment Toolbars

You can invoke the Modeling environment, if it is not already invoked by choosing **Start > Modeling** from the **Standard** toolbar. The toolbars in the Modeling environment are discussed next.

View Toolbar

The tools in the **View** toolbar, as shown in Figure 1-14, are used for manipulating the views of the model. The **View** toolbar is available in all the environments. Some of the buttons in the **View** toolbar are not available in the **Drafting** environment.



Figure 1-14 *The* ***View*** *toolbar*

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Direct Sketch Toolbar

The tools in this toolbar are shown in Figure 1-15 and are used to create sketch directly in the Modeling environment.



*Figure 1-15 The **Direct Sketch** toolbar*

Feature Toolbar

The tools in this toolbar are shown in Figure 1-16 and are used to convert a sketch drawn in the Sketch task environment into a feature. This toolbar contains sketch-based feature tools and placed feature tools. You can create the datum plane, axis, and points using the tools in this toolbar.

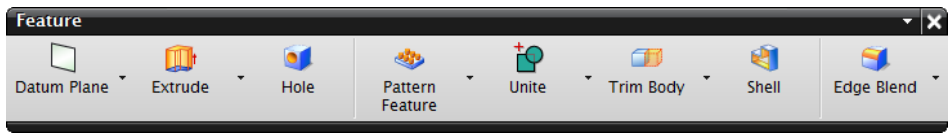

*Figure 1-16 The **Feature** toolbar*

Sketch Task Environment Toolbars

The **Sketch in Task** button in the **Feature** toolbar is used to invoke the Sketch task environment, where you can create a sketch. After choosing the **Sketch in Task** button, select a plane or a planar face to invoke the Sketch task environment. The toolbars in the Sketch task environment are discussed next.

Sketch Toolbar

The **Finish Sketch** button in the **Sketch** toolbar is used to switch back to the Modeling environment, where you can convert the sketch into a feature. Figure 1-17 shows the buttons that are available in the **Sketch** toolbar.

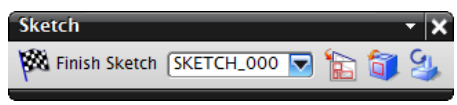

*Figure 1-17 The **Sketch** toolbar*

Sketch Tools Toolbar

It is one of the most important toolbars in the Sketch task environment. The tools in the **Sketch Tools** toolbar are used to draw the sketches as well as edit the drawn sketches. Additionally, you can apply constraints to the geometric entities and assign dimension to a sketch using the tools in this toolbar. You can make a sketch fully defined using these tools. Figure 1-18 shows the buttons that are available in the **Sketch Tools** toolbar.

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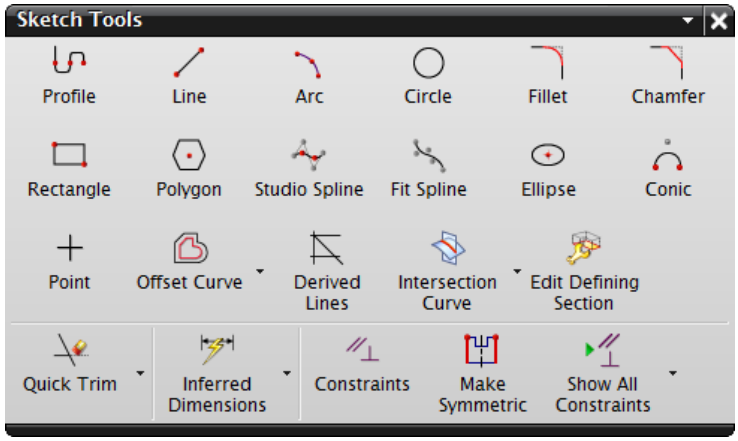

Figure 1-18 *The* ***Sketch Tools*** *toolbar*

Once the basic sketch is complete, you need to convert it into a feature. Choose the **Finish Sketch** button from the **Sketch** toolbar and switch back to the Modeling environment. The remaining toolbars in the Modeling environment are discussed next.

Surface Design Toolbars

You can create the surface design in the Modeling environment as well as in the Shape Studio environment. The tools used to create the solid bodies are also used to create the surface bodies. Some of the toolbars used to create the surface design are discussed next.

Surface Toolbar

The tools in the **Surface** toolbar are used to create complicated surfaces. Figure 1-19 shows the **Surface** toolbar.

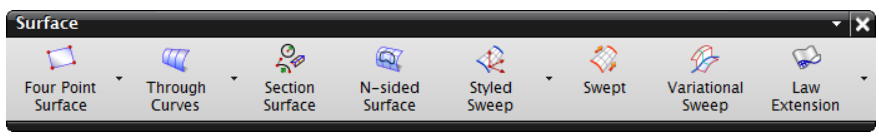

Figure 1-19 *The* ***Surface*** *toolbar*



Tip: *You can add tools in any toolbar as per the requirement. To do so, click on the down arrow in the title bar of a toolbar; a cascading menu will be displayed. Move the cursor over the* ***Add or Remove Buttons*** *option in the cascading menu; another cascading menu will be displayed. Now, move the cursor over the name of the required toolbar; another cascading menu will be displayed containing all the tools related to that toolbar. You will observe that the tools which are not available in the toolbar are unselected. If you select any unselected tool, it will become available in the toolbar.*

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Assembly Environment Toolbars

You can create the assembly in the same Modeling environment. The toolbars that are used to create the assembly can be invoked by choosing **Start > Assemblies** from the **Standard** toolbar. Alternatively, choose the **Assemblies** button from the **Application** toolbar. The toolbars used in the assembly design are discussed next.

Assemblies Toolbar

The tools in the **Assemblies** toolbar are used to insert an existing part or assembly in the current assembly file. You can also create a new component in the assembly file using the tools in this toolbar. Figure 1-20 shows the buttons in the **Assemblies** toolbar.

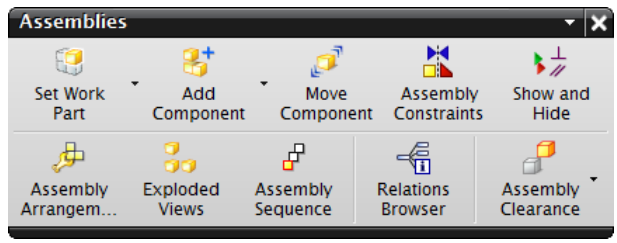

Figure 1-20 *The* ***Assemblies*** *toolbar*

Drafting Environment Toolbars

To invoke the Drafting environment, choose the **Drafting** button from the **Application** toolbar. Alternatively, this environment can be invoked by choosing **Start > Drafting** from the **Standard** toolbar. The toolbars in the Drafting environment are discussed next.

Drawing Toolbar

The tools in the **Drawing** toolbar are used to insert a new sheet, create a new view, generate an orthographic view, section view, and detail view for a solid part or an assembly. Figure 1-21 shows the **Drawing** toolbar.

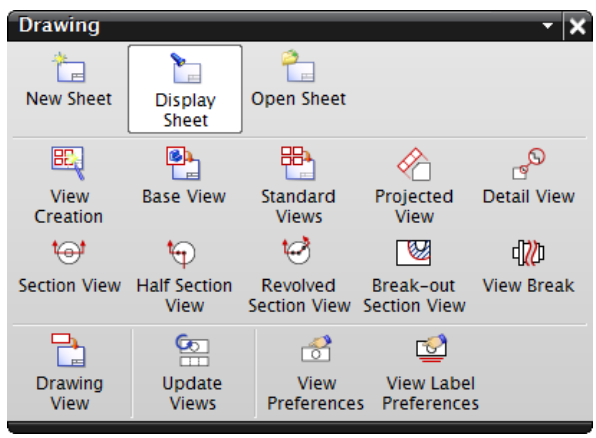

Figure 1-21 *The* ***Drawing*** *toolbar*

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Dimension Toolbar

The tools in the **Dimension** toolbar are used to generate various dimensions in the drawing views. Figure 1-22 shows the **Dimension** toolbar.

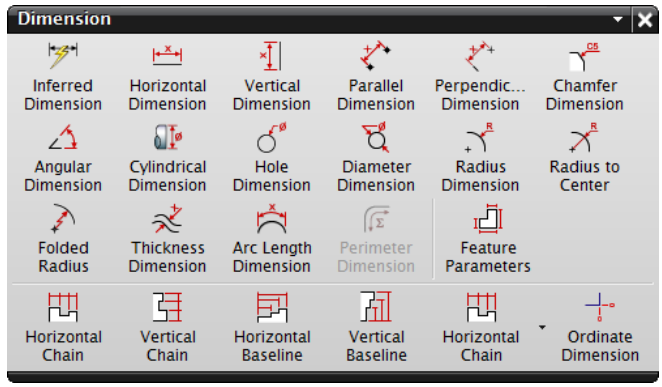

Figure 1-22 *The* ***Dimension*** *toolbar*

Annotation Toolbar

The tools in the **Annotation** toolbar are used to generate the GDT parameters, annotations, symbols, and so on. Figure 1-23 shows the **Annotation** toolbar.

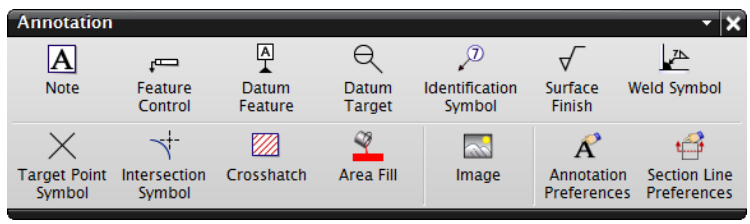

Figure 1-23 *The* ***Annotation*** *toolbar*

Synchronous Modeling Toolbar

The Synchronous Modeling Technology is one of the latest enhancements in NX. The tools available in this toolbar are used to modify and improve an existing design in the shortest period of time. Figure 1-24 shows the **Synchronous Modeling** toolbar.

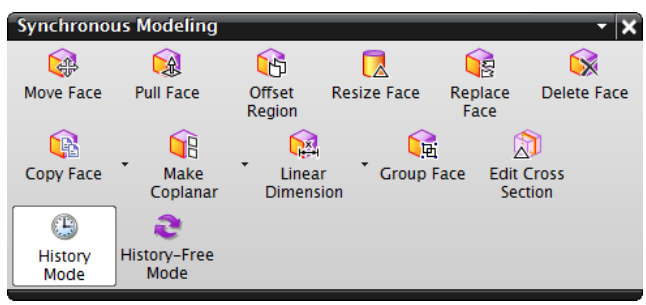

Figure 1-24 *The* ***Synchronous Modeling*** *toolbar*

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NX Sheet Metal Toolbar

The tools in the **NX Sheet Metal** toolbar are used to create a sheet metal component. Figure 1-25 shows the **NX Sheet Metal** toolbar.

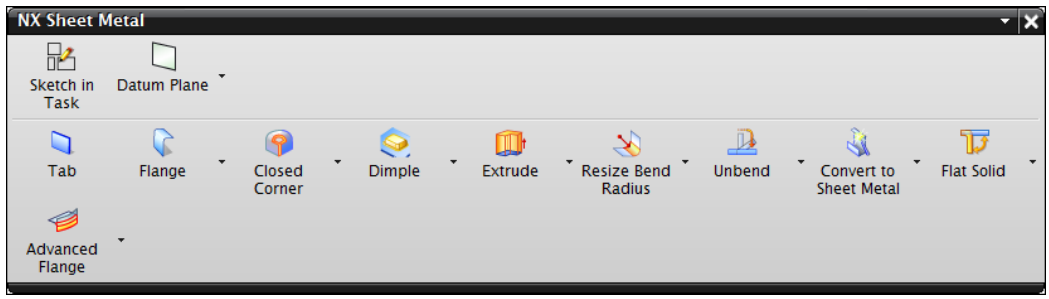

*Figure 1-25 The **NX Sheet Metal** toolbar*

HOT KEYS

NX is more popularly known for its icon driven structure. However, you can still use the keys on the keyboard to invoke some tools. These keys are called hot keys. The hot keys, along with their functions, are listed in the table given next.

Hot Key	Function
CTRL+Z	Invokes the **Undo** tool
CTRL+Y	Invokes the **Repeat** tool
CTRL+S	Saves the current document
F5	Refreshes the **Drawing** window
F1	Invokes the NX **Help** tool
F6	Invokes the **Zoom** tool
F7	Invokes the **Rotate** tool
CTRL+M	Invokes the Modeling environment
CTRL+SHIFT+D	Invokes the Drafting environment

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COLOR SCHEME

NX allows you to use various color schemes as the background screen color and also for displaying the solid bodies on the screen. To change the background color scheme, choose **Preferences > Background** from the menu bar; the **Edit Background** dialog box will be displayed.

Select the **Plain** radio button from the **Shaded Views** and **Wireframe Views** areas. Next, choose the color swatch available on the right side of the **Plain Color** option; the **Color** dialog box will be displayed. Select the **White** color swatch from the **Color** dialog box and choose the **OK** button twice to apply the new color scheme to the NX environment.



Note

For the purpose of printing, this book will follow the white background of the NX environment. However, for a better understanding and also for a clear visualization at various places, this book will follow other color schemes also.

DIALOG BOXES IN NX

To create any feature, you need to follow certain steps in a particular order. These steps are placed in a top-down order in the corresponding dialog boxes. This layout of dialog boxes will help you throughout the feature creation operation, refer to Figure 1-26.

The current selection step will be highlighted in orange. The required steps are marked with red asterisks, and the completed steps are marked with green check marks. The advanced options are collapsed and hidden in the rollouts. The button highlighted in green indicates the next default action.

The **Reset** button is used to reset the dialog box to its initial settings. The **Hide Collapsed Groups** button is used to hide all collapsed rollouts to simplify the dialog box. To view all the collapsed rollouts, choose the **Show Collapsed Groups** button, which will be available only after choosing the **Hide Collapsed Groups** button. You can also click on the arrow symbols displayed at the bottom of the dialog box to show or hide the collapsed rollouts. The **Close** button is used to exit the dialog box.

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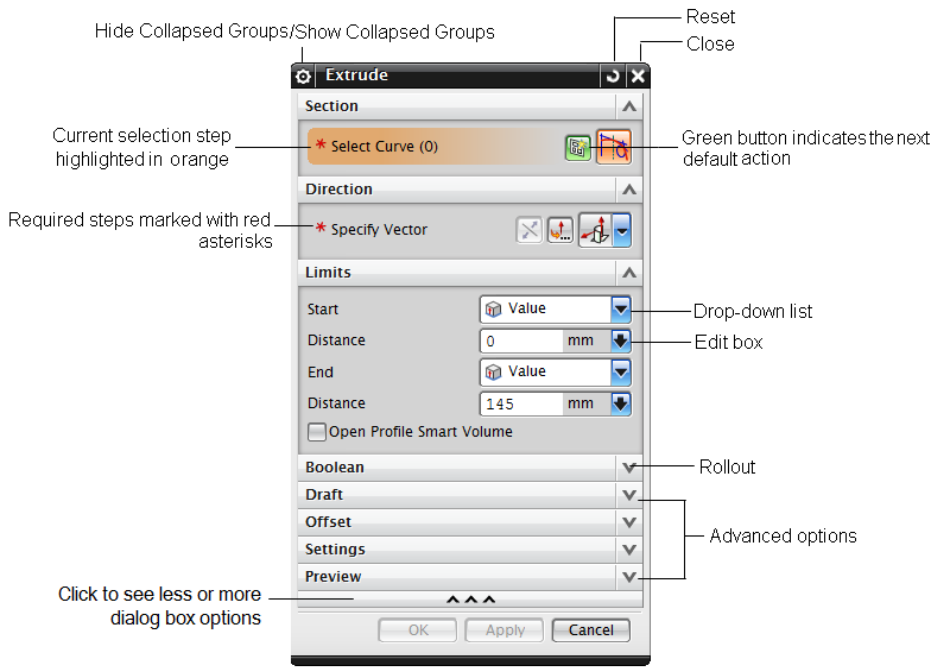

Figure 1-26 *The* ***Extrude*** *dialog box*

SELECTING OBJECTS

When no tool is invoked in the current environment, the select mode will be activated. You can ensure that the select mode is active by pressing the ESC key. In this mode, you can select a wide range of objects from different environments such as individual features, part bodies, surface bodies, planar and non-planar faces, sketched entities, sketch and assembly constraints, and so on by clicking on them. Alternatively, press and hold the left mouse button and drag a box around the objects; all objects that lie completely inside the box are selected.

DESELECTING OBJECTS

By default, the selected objects are displayed in orange color. If you want to deselect any specific object from the selection, press and hold the SHIFT key and click on it; the object will be deselected. If you want to deselect all the selected entities, press the ESC key. Alternatively, press and hold the SHIFT key and drag a box around the entities; all entities that lie completely inside the box are deselected. Also, you can choose the **Deselect All** button from the **Selection** bar to deselect all the selected entities.

SELECTING OBJECTS USING THE QuickPick DIALOG BOX

If objects are close to each other, then it may be difficult to select the required object. In such cases, move the cursor over the object to be selected and wait for three seconds; the cursor

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will be changed to '+' sign with three dots. Next, press the left mouse button; the **QuickPick** dialog box will be displayed. This dialog box will list all the objects near the selected object in the drawing window. Move the cursor over the objects listed; the corresponding objects will be highlighted in magenta color in the drawing window. Select the required object from the **QuickPick** dialog box; the specified object will get selected.

Self-Evaluation Test

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

1. The Modeling environment of NX is a parametric and feature-based environment. (T/F)

2. You can modify an existing design quickly using the **Synchronous Modeling** tools. (T/F)

3. The generative drafting technique is used to automatically generate the drawing views of the parts and assemblies. (T/F)

4. By default, the **Resource Bar** is located on the left side of the NX window. (T/F)

5. The _____ analysis provides the interference check between the components in an assembly.

6. The _____ is a file extension associated with all the files that are created in different environments of NX.

7. The _____keeps a track of all the operations that are carried out on the part.

8. The _____constraint is used to fix a selected entity in terms of its position with respect to the coordinate system of the current sketch.

9. You can invoke the _____ tool by pressing and holding the middle mouse button.

10. The _____toolbar is used to generate the GDT parameters, annotations, and symbols.

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Answers to Self-Evaluation Test

1. T, **2.** T, **3.** T, **4.** T, **5.** Check Clearance, **6.** *.prt, **7. Part Navigator**, **8.** fixed, **9. Rotate**, **10. Annotation**