Chapter *1*

Introduction to NX 9.0

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.*
• *Understand the use of various hot keys.*
• *Modify the color scheme in NX.*
• *Know about dialog boxes in NX.*

INTRODUCTION TO NX 9.0

Welcome to NX 9.0 (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 9.0, 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 by allowing flexibility in the use of feature-based and parametric designs.

The subject of interpretability offered by NX includes receiving legacy data from 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, Sheet metal 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 and you can draw the sketch directly in the Modeling environment by using the tools available in the **Direct Sketch** group of the **Home** tab. The sketch can also be drawn in the Sketch environment. The Sketch environment can be invoked by choosing the **Sketch** tool from the **Direct Sketch** group of the **Home** tab or by choosing the **Sketch in Task Environment** tool from the **Curve** tab. While drawing a sketch, various applicable constraints and dimensions are automatically applied. Additional constraints and dimensions can also be applied 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 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.

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.

In the Assembly environment you can also assemble an existing assembly with the current assembly. The perform analysis provides the facility to check the interference and clearance 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.

Sheet Metal Environment

The Sheet Metal environment is used for the designing the sheet metal components. Generally, the sheet metal components are created to generate the flat pattern of a sheet, study the design of the dies and punches, study the process plan for designing, and the tools needed for manufacturing the sheet metal components.

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, Windows 8 or UNIX.
- **Memory**: 2 GB of RAM is the minimum requirement for all the applications and 4GB of RAM is recommended for DMU applications.
- **Disk drive**: 10 GB Disk Drive space (Minimum recommended size)
- **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 9.0 on the desktop of your computer. After the system has loaded all the required files to start NX, the initial screen will be displayed, as shown in Figure 1-1.


Figure 1-1 *The initial screen that appears after starting NX 9.0*

Choose **File > New** from the **Ribbon**; the **New** dialog box will be displayed as shown in Figure 1-2. 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.
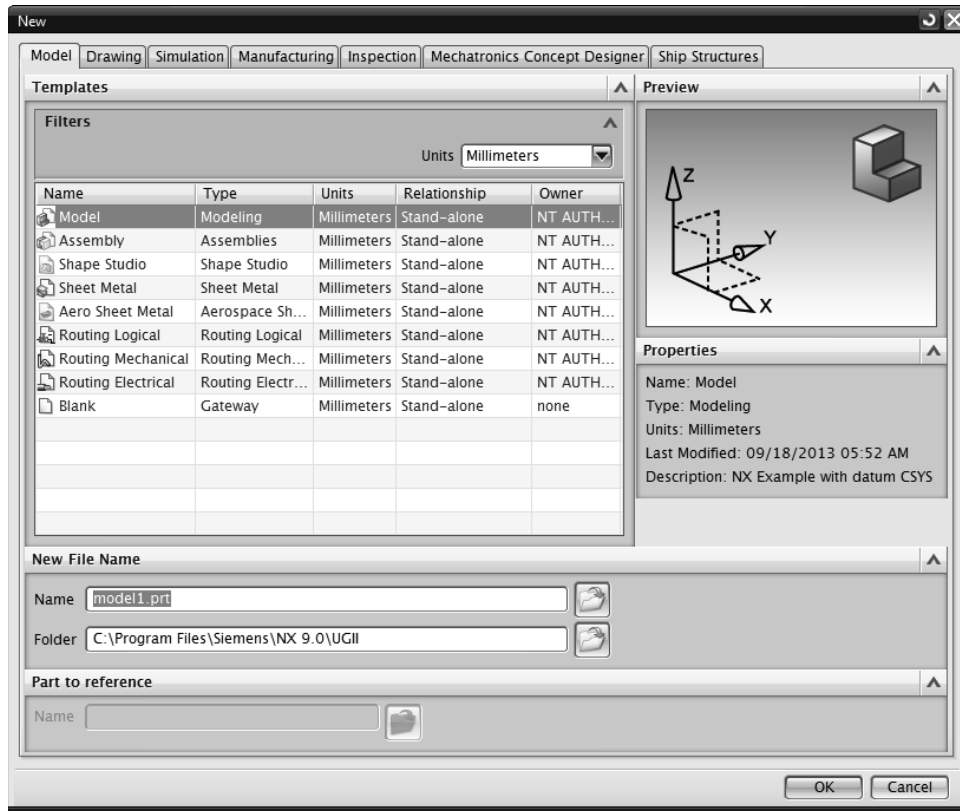
Figure 1-2 *The New dialog box*
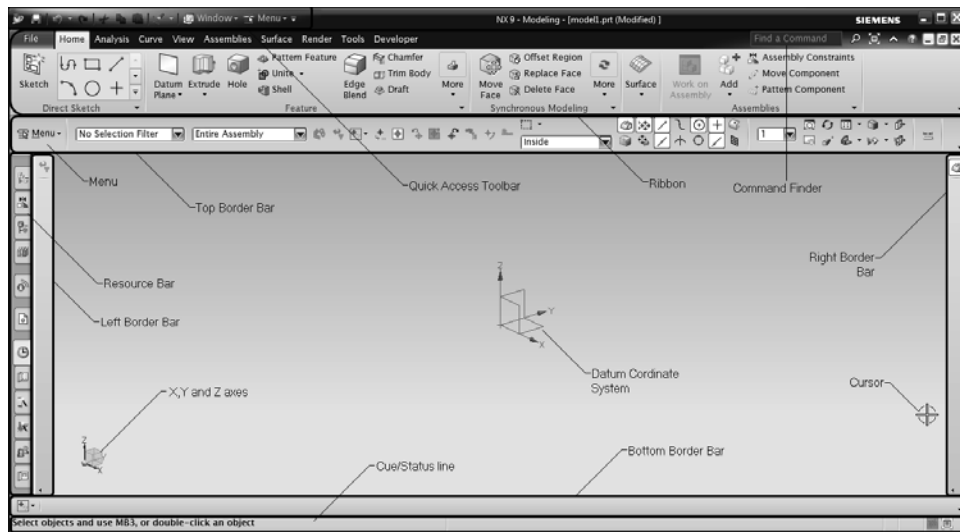
Figure 1-3 *The Modeling environment displayed on screen*

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.
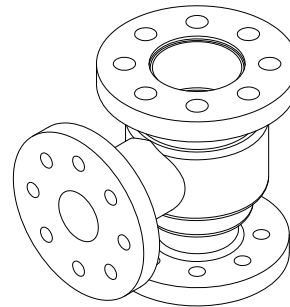
Parametric Modeling

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.
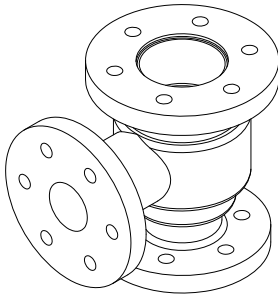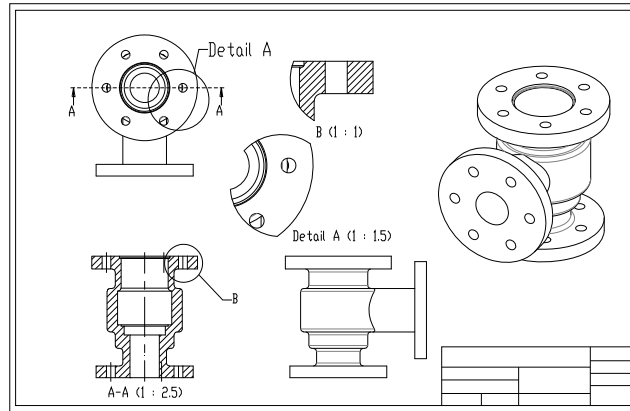

*Figure 1-4 Body of a pipe housing*   *Figure 1-5 Modified body of the pipe housing*

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

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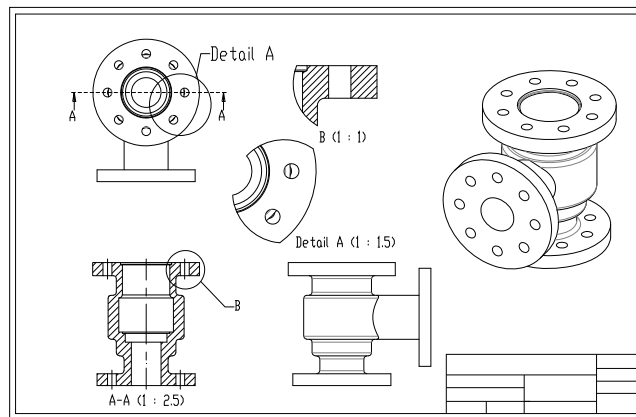
Figure 1-7 *The drawing views of pipe housing after making the modifications*

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

Resource Bar
The **Resource Bar** combines all the navigator windows, the history palette, the integrated web browser, and the parts template at 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**

is used to activate the required role. In this book, the **Essentials** 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** 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.
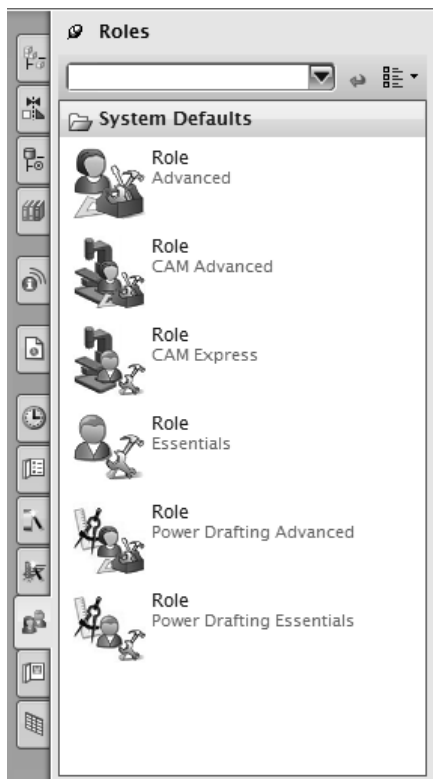
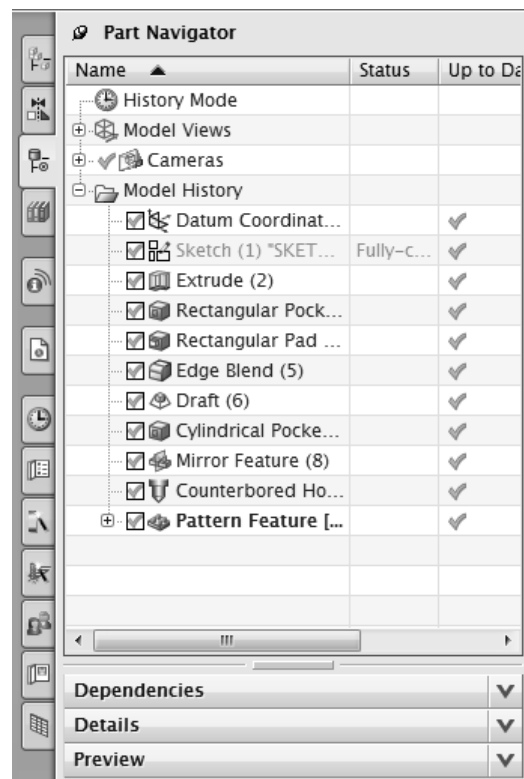*Figure 1-8* The **Roles** navigator

Figure 1-9 The **Part Navigator**

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. Linear Dimensions
2. Radial Dimensions
3. Angular Dimensions
4. Perimeter Dimensions

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

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 origin of the Datum Coordinate System, which is (0,0,0). By default, the display of WCS is turned on. To turn off the display of WCS, choose the **Display WCS** tool from the **Tools > Utilities > More > WCS** in the **Ribbon**; the WCS will be turned off from the drawing window. Note that it is a toggle button.

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 the three buttons of the mouse, 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.
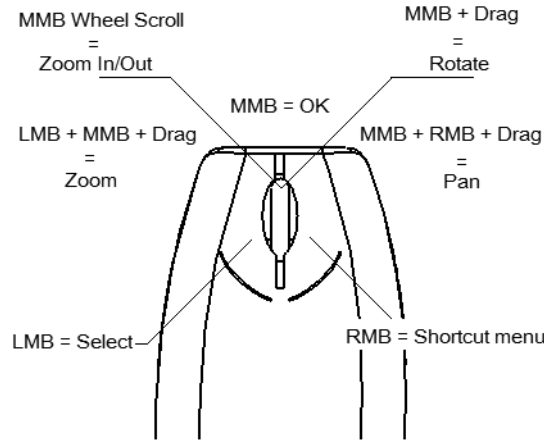
Figure 1-10 *Functions of the mouse buttons*

Various screen components of NX are discussed next.

QUICK ACCESS TOOLBAR

This toolbar is common to all the environments of NX. Figure 1-11 shows the **Quick Access** toolbar. The buttons in this toolbar are used to start a new file, open an existing file, save a file of the current document, 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.

Figure 1-11 *The **Quick Access** toolbar*

RIBBON

NX offers a user-friendly design interface by providing the **Ribbon**. The **Ribbon** comprises a series of tabs. In tabs, the various tools and options are grouped together based on their functionality in different groups and galleries. The display of these tabs and their groups depends upon the environment invoked. The different environments and some of their respective tabs and groups are discussed next.

Modeling Environment

The Modeling environment can be invoked by selecting the Modeling template from the **New** dialog box. You can also invoke the Modeling environment from any other opened environment. To do so, choose **File > Applications > Modeling** from the **Ribbon**. Some of the tabs of the Modeling environment are discussed next.

Home Tab

The **Home** tab consist of a series of groups and galleries and are discussed next.
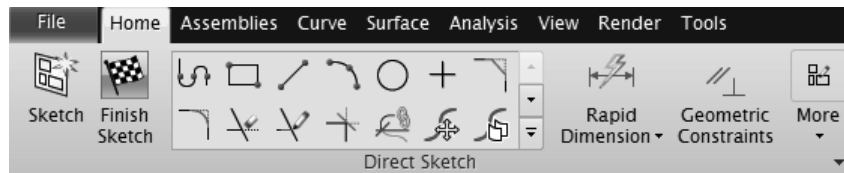
Direct Sketch Group

It is one of the most important groups of the **Home** tab. The tools available in this group are used to draw and edit sketches. You can apply constraints to geometric entities and assign dimension to a sketch using the tools of this group. The sketching tools available in this group are grouped together in the **Sketch Curve** gallery, refer to Figure 1-12. Note that by default, the **Sketch Curve** gallery is in collapse form. As a result, some of the tools are not visible by default. To expand this gallery, click on the lower down arrow available in front of this gallery, see Figure 1-13. Note that some of the tools such as **Rapid Dimension**, **Geometric Constraints**, **Fillet**, **Chamfer**, **Quick Trim**, and **Quick Extend** are still not available in the expanded **Sketch Curve** gallery. To access all the tools of the **Sketch Curve** gallery, choose the **Sketch** tool available in the **Direct Sketch** group; the **Create Sketch** dialog box will be displayed. The options available in this dialog box are discussed in later chapters. Select the required sketching plane and then choose the **OK** button, the **Sketch Curve** gallery expanded such that it contains all the sketching tools including dimensioning and geometric constraints, see Figure 1-14.


*Figure 1-12 The **Direct Sketch** group*

Figure 1-13 Expanded
***Sketch Curve** gallery*

*Figure 1-14 The expanded **Sketch Curve** gallery of the **Direct Sketch** group after invoking Sketching environment*

Feature Group

The tools in this group are shown in Figure 1-15 and are used to convert a sketch drawn in the Sketching environment into a feature. This group contains sketch-based feature tools and placed-feature tools. You can create datum plane, axis, and points using the tools in this group.
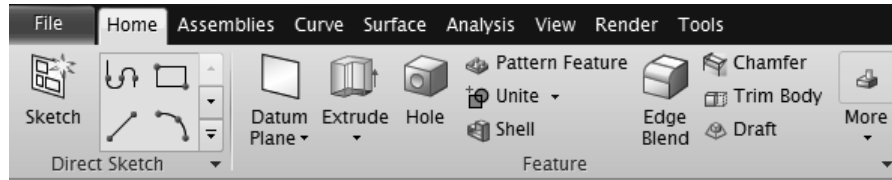
*Figure 1-15 The **Feature** group*

Synchronous Modeling Group
The Synchronous modeling technology is one of the latest enhancements in NX. This technology is used to modify the parts even if the modeling history is not available. The tools available in the **Synchronous Modeling** group are used to modify and improve an existing design in less time. Figure 1-16 shows the **Synchronous Modeling** group.
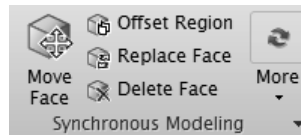
*Figure 1-16 The **Synchronous Modeling** group*

Curve Tab
The **Curve** tab comprises a series of groups and galleries. You can invoke Sketching environment by using the **Sketch in Task Environment** tool of this tab. Figure 1-17 shows the **Curve** tab.
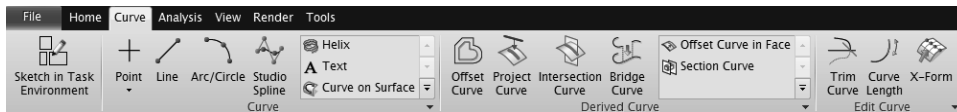

*Figure 1-17 The **Curve** tab*

Surface Tab
You can create surface design in the Modeling environment as well as in the Shape Studio environment. The tools used to create solid bodies are also used to create surface bodies. The **Surface** group under the **Surface** tab, which have tools to create the surface design and is discussed next. Note that this tab is not available by default.

Surface Group
The tools in the **Surface** group are used to create complex surfaces. Figure 1-18 shows the **Surface** group.
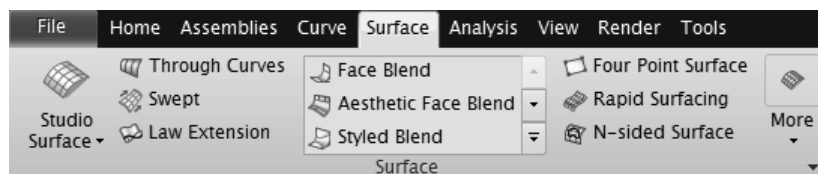
*Figure 1-18 The **Surface** group*

Tip: *By default, all tools are not available in a group. Therefore, you may need to customize the group to add those tools that are not available by default. To customize a group, click on the down arrow at the bottom right corner of the group; a drop-down will be displayed. Click on the tool to be added or removed from the group. Note that a tick mark available on the left of a tool indicates that it is already added to the group.*

*Similarly, you can add or remove groups from the **Ribbon** by using **Ribbon Options** arrow available at the bottom right corner of the **Ribbon**.*

Some of the tabs that are available in the Modeling environment as well as common to other environments of NX are discussed next.

Application Tab

Using the **Application** tab, you can invoke any other environment from the currently invoked environment. By default, the **Application** tab is not available in the **Ribbon.** Therefore, you need to customize to add it. To do so, right-click on the **Ribbon** and select the **Application** option from the shortcut menu displayed. You can add this tab to all environments of NX. Figure 1-19 shows the **Application** tab.
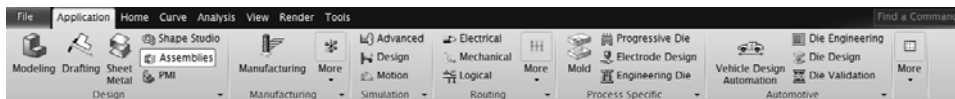

*Figure 1-19   The **Application** tab*

Tip: *Some of the tabs are, by default, available in their respective environments. However, you can add more tabs to an environment. To do so, right-click on the **Ribbon**; 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 option; it will become available as a tab in the **Ribbon**.*

View Tab

The tools in the **View** tab, as shown in Figure 1-20, are used for manipulating the views of the model. The **View** tab is available in all the environments. Some of the tools in the **View** tab are not available in the **Drafting** environment.

*Figure 1-20   The **View** tab*

Assembly Environment

In NX, you can invoke the assembly environment within the Modeling environment and create assemblies by using different assembly tools. The tools for assembling are available in the **Assembly** tab and are discussed next.

Assemblies Tab

The tools that are used to create an assembly are grouped together in the **Assembly** tab. To add this tab, choose **File > Applications > Assemblies** check box from the **Ribbon**. Alternatively, choose the **Assemblies** button from the **Application** tab. The tools of the **Component** group available in this tab 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 group. Figure 1-21 shows the tools in this group.
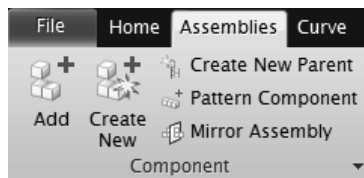

*Figure 1-21 The **Component** group*

Drafting Environment

To invoke the Drafting environment, choose **File > Applications > Drafting** from the **Ribbon**. Alternatively, this environment can be invoked by choosing the **Drafting** tool from the **Application** tab. You can also invoke this environment by using the **Drawing** template of the **New** dialog box. The groups in the Drafting environment are discussed next.

View Group

This group is displayed in the **Home** tab after invoking the Drafting environment. The tools in the **View** group 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 1-22 shows the **View** group.
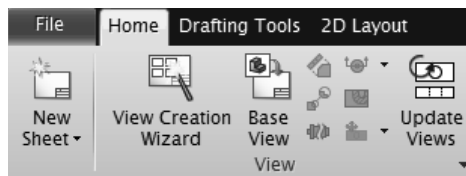

*Figure 1-22 The **View** group*

Dimension Group

The tools in the **Dimension** group are used to generate various dimensions in the drawing views. Figure 1-23 shows the **Dimension** group.

***Figure 1-23*** *The **Dimension** group*

Annotation Group

The tools in the **Annotation** group are used to generate the GDT parameters, annotations, symbols, and so on. Figure 1-24 shows the **Annotation** group.

Figure 1-24 *The **Annotation** group*

Sheet Metal Environment

The tools in the Sheet Metal environment are used to create a sheet metal component. Figure 1-25 shows the groups and tools of Sheet Metal environment.
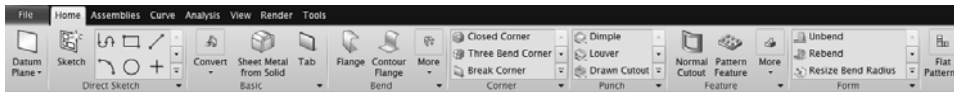
Figure 1-25 *Tools in the Sheet Metal environment*

STATUS BAR

The Status Bar that appears at the bottom of the drawing window comprises of two areas and buttons, as shown in Figure 1-26. These are discussed next.
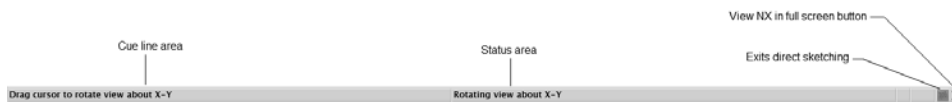
Figure 1-26 *The Status Bar*

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 graphics area will be maximized and will give you a full screen display. For getting the default screen display, you need to choose this button again.

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

COLOR SCHEME

NX allows you to use various color schemes as the background screen color and also for displaying solid bodies on the screen. To change the background color scheme, choose **Menu > Preferences > Background** from the **Top Border 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 in general will follow the white background of the NX environment. However, for better understanding and clear visualization, at various places other color schemes are also followed.

DIALOG BOXES IN NX

To create any feature, you need to follow certain steps in an order. These steps are placed in a top-down order in the dialog boxes. This layout of dialog boxes will help you throughout the feature creation operation, refer to Figure 1-27.

In a dialog box, 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 next default action.

The **Reset** button is used to reset the dialog box to its initial settings. The **Hide Collapsed Groups** option from the **Dialog Option** button is used to hide all the collapsed rollouts to simplify the dialog box. To view all the collapsed rollouts, choose the **Show Collapsed Groups** option from the **Dialog Option** button, which will be available only after choosing the **Hide Collapsed Groups** option from the **Dialog Option** button. The **Close** button is used to exit the dialog box.
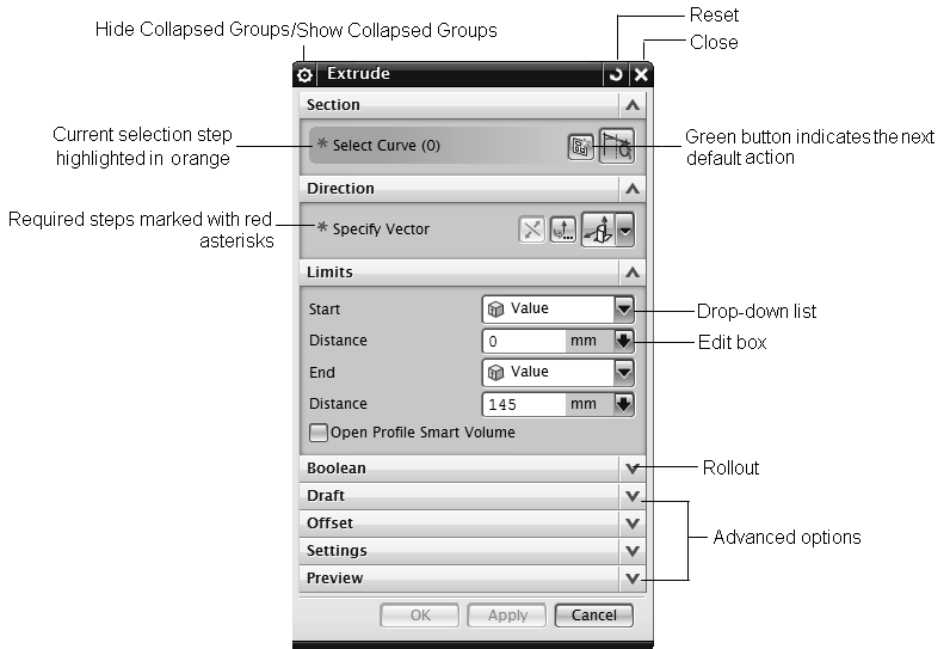
Figure 1-27 The **Extrude** *dialog box*

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 the entities that lie completely inside the box are deselected. Also, you can choose the **Deselect All** button from the **Selection Group** 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 two seconds; the cursor 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 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 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 is used to check the interference and clearance 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 _____group is used to generate the GDT parameters, annotations, and symbols.

Answers to Self-Evaluation Test
1. T, **2.** T, **3.** T, **4.** T, **5.** Perform, **6.** *.prt, **7. Part Navigator**, **8.** Fixed, **9. Rotate**, **10. Annotation**