Chapter 1

Introduction to NX

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*

INTRODUCTION TO NX

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

NX provides improved capabilities for convergent modeling, CAD design, drafting, tooling design, automation, design simulation, programming, data translation, validation, mold design, and much more. Active workspace in NX is built based on these items to provide seamless access to PLM capabilities right within NX.

NX, a product of SIEMENS Corp., is a completely re-engineered, next-generation family of CAD/CAM/CAE/PLM 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 specific area, consisting of a set of tools which allows the user to perform specific design tasks in that 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 Sheet Metal 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. In NX, you can create a sketch by using two methods: Direct sketch and Sketch in Task Environment. In the Direct sketch method, you can create a sketch, as required, directly in the Modeling environment by invoking the sketching tools such as **Line**, **Arc/Circle**, **Point** and so on from the **Base** group of the **Curve** tab of the **Ribbon**. The sketch can also be drawn in the Sketch in Task Environment or sketching environment. To create the sketch in this environment, you need to invoke the sketching environment by using the **Sketch** tool from the **Construction** group of the **Home** tab of the **Ribbon**. While drawing a sketch, various applicable constraints and dimensions are automatically applied to it. Additional constraints and dimensions can also be applied manually. After drawing the sketch, you need to convert the sketch into a feature using

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the tools available in the Modeling environment. You can create placed features such as fillets, chamfers, taper, and so on and can also assign materials to the models in this 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 and 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** tool 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 application provides an environment for the design of sheet metal parts used in machinery, enclosures, brackets, and other parts normally manufactured with a brake press. The Sheet Metal application is intended mainly for designing parts with cylindrical bend regions but conical and curved bend regions are also possible. Generally, the sheet metal components are created to generate the flat pattern of a sheet, study the design of the dies and punches,

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study the process plan for designing, and the tools needed for manufacturing the sheet metal components.

SYSTEM REQUIREMENTS

System requirements that ensure the smooth running of NX are as follows:

- 64-bit - Windows 10 or later Operating System.
- 16GB of RAM is the minimum requirement but it is recommended to have 32GB or more RAM for all the applications to run smoothly.
- Java version - 1.8.0 or higher
- True Color (32-bit) or 16 million colors (24-bit)
- Screen Resolution: 1280 x 1024 or higher, widescreen format.

GETTING STARTED WITH NX

Install NX on your system and then start it by double-clicking on its shortcut icon on the desktop of your computer. After the system has loaded all the required files to start NX, the initial interface will be displayed, as shown in Figure 1-1.



Note

When you install NX, by default its shortcut icon is not created on the desktop of your computer. So to create the shortcut icon, click start to display a menu. Drag NX icon from the Siemens NX folder to the desktop.

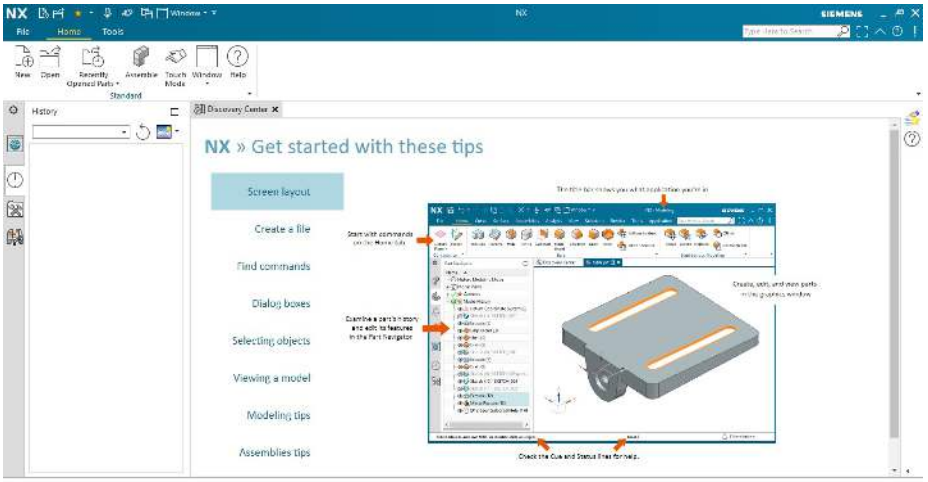


Figure 1-1 *The initial interface that appears after starting NX*

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Tip

1. In this release, by default, the user interface theme is set to ***Light****. But, you can change the interface theme as per your requirement. To do so, choose* ***Menu > Preferences > User Interface*** *from the* ***Top Border Bar****; the* ***User Interface Preferences*** *dialog box will be displayed. Next, select the required option from the* ***Type*** *drop-down list available in the* ***NX Theme*** *group of the* ***Theme*** *node and choose the* ***OK*** *button.*

2. To change the ***Default Presentation of Dialog Content*** *of any dialog box, choose* ***Menu > Preferences > User interface*** *from the* ***Top Border Bar****; the* ***User Interface Preferences*** *dialog box will be displayed. Next, select the* ***Dialog and Precision*** *node available on the left in the dialog box; the* ***Dialog Boxes*** *group will be displayed. Now, select the* ***More*** *radio button in the* ***Default Presentation of Dialog Content*** *area and choose the* ***OK*** *button.*



Note

The ***Top Border Bar*** *is not displayed by default in the initial screen of the new part file. To make it visible, right-click in the* ***Ribbon*** *area; a shortcut menu will be displayed. Next, select the* ***Top Border Bar*** *option from it.*

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.

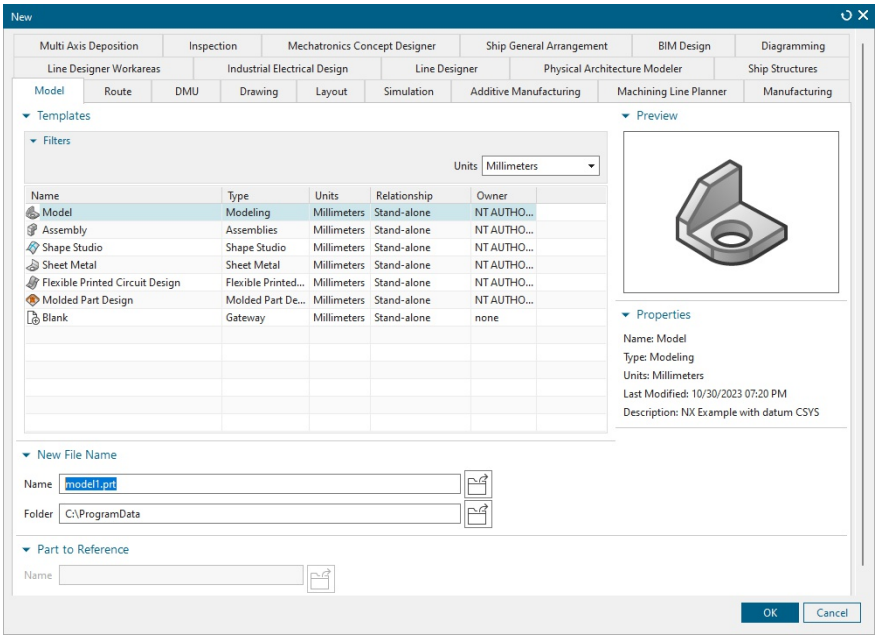


Figure 1-2 *The* ***New*** *dialog box*

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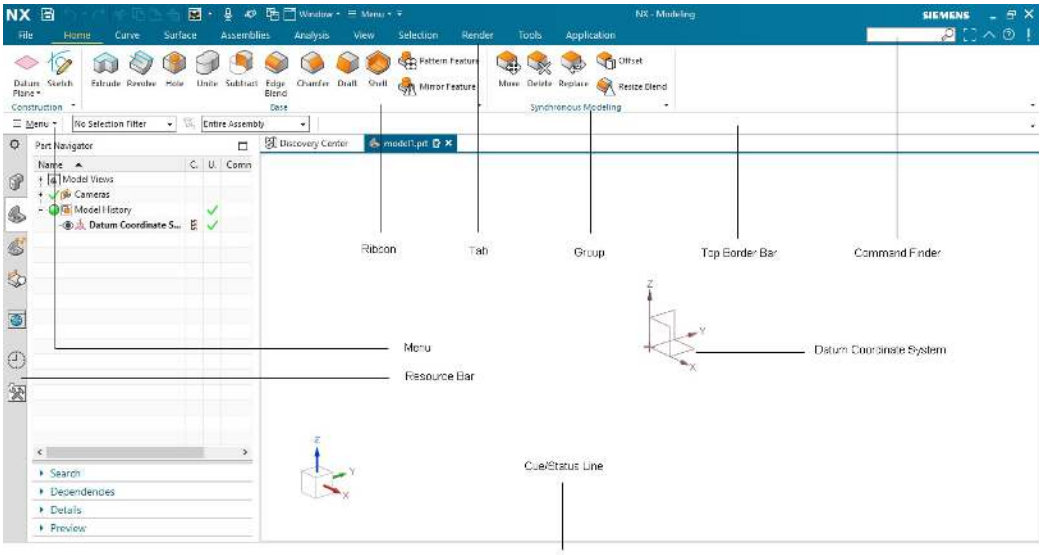


Figure 1-3 *The Modeling environment of NX displayed on screen*

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.

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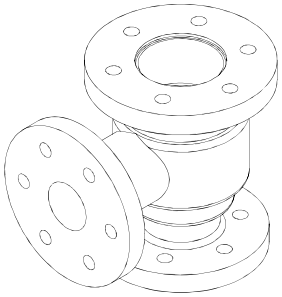


Figure 1-4 Body of a pipe housing

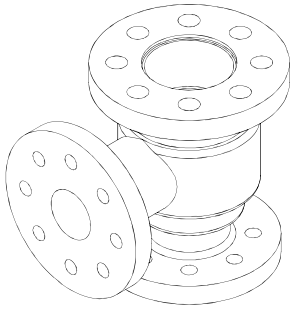


Figure 1-5 Modified body of the pipe housing

Bidirectional Associativity

As mentioned earlier, NX has different environments such as the Modeling environment, the 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 Drafting 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.

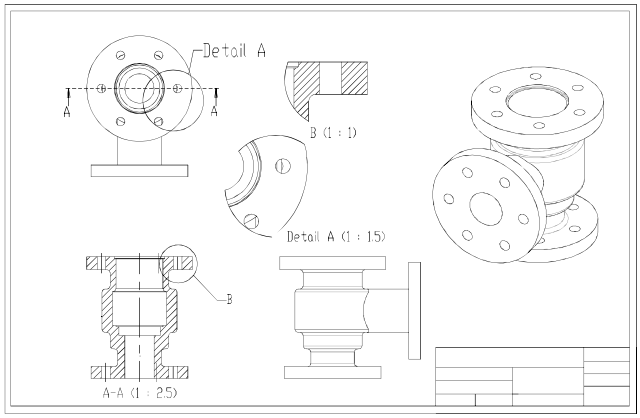


Figure 1-6 The drawing views of a pipe housing

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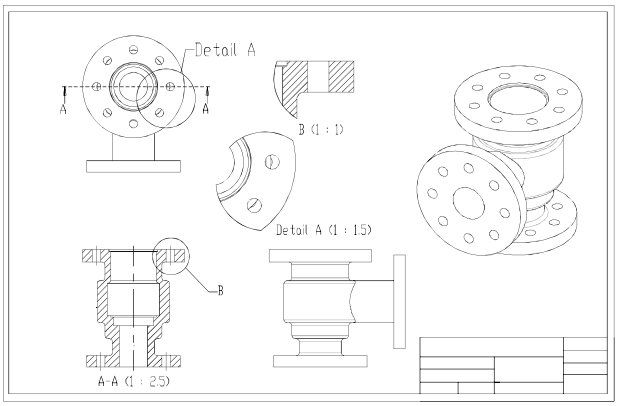


Figure 1-7 *The drawing views of pipe housing after making the modifications*

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, and the integrated web browser 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 sets 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 **Content** option; a flyout will be displayed. Click on the **Essentials** icon to activate that role; the **Load Role** message box will be displayed. Next, choose the **OK** button to close the dialog box. 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**.

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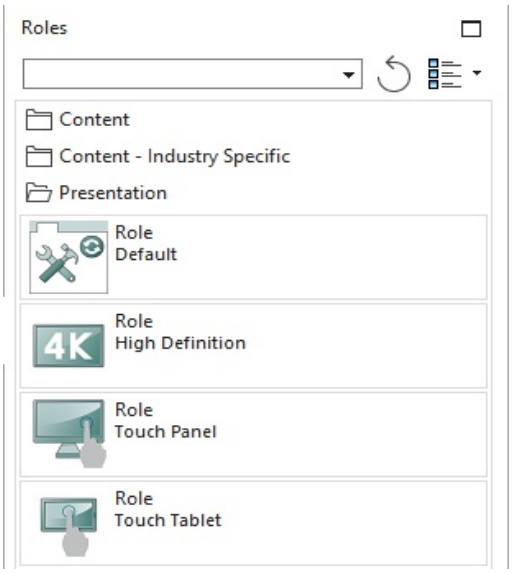


*Figure 1-8 The **Roles** navigator*

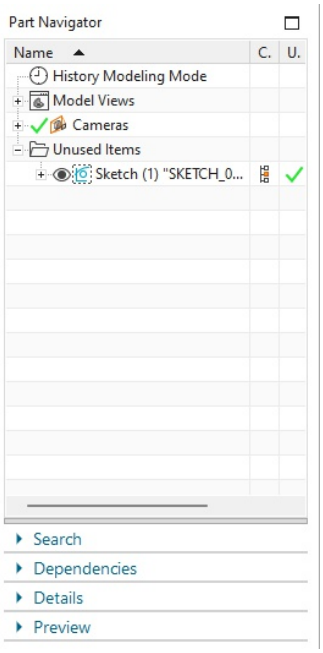


*Figure 1-9 The **Part Navigator***

Constraints

Constraints are the logical operations that are performed on an 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 geometric and dimensional constraints available in the sketching environment 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.

Geometric Constraints

These are the logical operations performed on the sketched elements to define their size and position with respect to 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. You will learn more about applying constraints to the sketch in later chapters of this book.

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 Dimension
2. Radial Dimension
3. Angular Dimension
4. Perimeter Dimension

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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 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 other components of the assembly.

Solid Body

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

Sheet Body or Surfaces

Surfaces are geometric features that have zero thickness and mass. They are used to create complex shapes which 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. You can also invoke the Surface environment by choosing the **Shape Studio** option from the **Templates** rollout of the **New** dialog box.

Features

A feature is defined as a 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 require a sketch for their creation and 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 off. To turn on the display of WCS, choose **Menu > Format > WCS > Display**; the WCS will be displayed in the drawing window. Note that **Display** button 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:

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1. The left mouse button is used 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 by dragging 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.

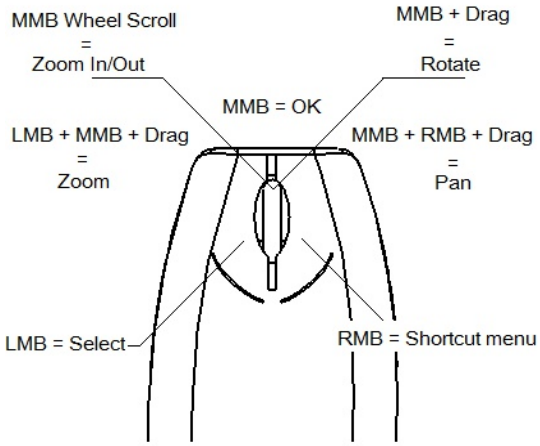


Figure 1-10 *Functions of the mouse buttons*

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, Touch Mode, search a tool, and invoke the help topics.

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In NX, when you have a session running in which several parts are open, you can switch between the running parts using the **Switch Window** button. This button is available next to the **Touch Mode** button in the **Quick Access** toolbar, refer to Figure 1-11.



Figure 1-11 *Partial view of the* ***Quick Access*** *toolbar*



Note

1. You can use the Ctrl+Tab keys to open the panel to show all files that are currently open. You can also move from item to item using the Tab key or hold the Ctrl key and use the mouse wheel to scroll through them. You can also select an item to open it.

2. The ***Switch Window*** *button located next to the* ***Touch Mode*** *button allows you to open the panel manually. By using this button, you will be able to navigate through the displayed parts using the left and right arrowkeys on your keyboard.*

Using Multiple Windows in NX

NX allows you to use multiple windows while sketching. You can view multiple parts at once in separate tabbed windows. The **Window** drop-down list available in the **Quick Access** toolbar contains the options which allow you to view the same part or different parts in two or more separate windows. Figure 1-12 shows the **Window** drop-down list.

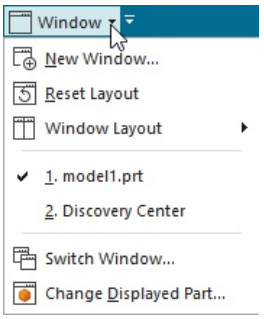


Figure 1-12 *The* ***Window*** *drop-down list*

The options available in this drop-down list are discussed next.

New Window

When you choose this option, a new file tab will be added below the **Top Border Bar**, refer to Figure 1-13. You can switch between the tabs by clicking on the respective tab. An active tab is displayed in light blue color.

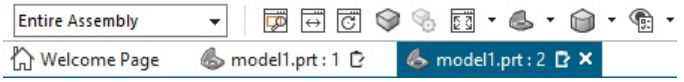


Figure 1-13 *File tab added on choosing the* ***New Window*** *option*

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Window Layout

When you hover the cursor over this option, a cascading window will be displayed, refer to Figure 1-14.

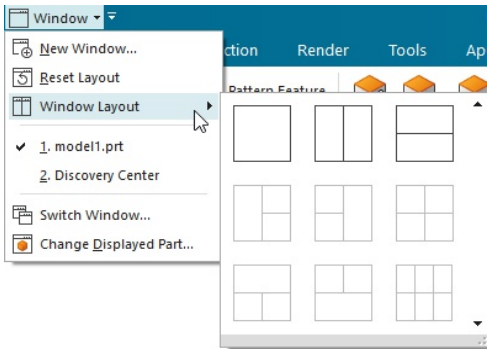


Figure 1-14 *Cascading window displaying the* ***Window Layout*** *options*

You can choose the required layout option from the cascading window. Figure 1-15 shows two tabbed windows displaying the same model. These tabbed windows will appear when you choose the **2 Tabbed Groups** option (second option in the first row) from the cascading window.

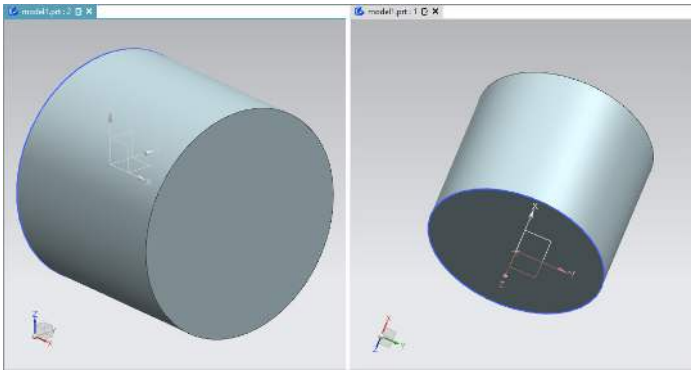


Figure 1-15 *Model displayed in two tabbed windows*



Note

The options in the ***Window Layout*** *cascading window will become active depending upon the number of windows that are active during a session. For instance, if you have a model or sketch open in one window and you choose the* ***New Window*** *option from the* ***Window*** *drop-down list, a new file tab will be added below the* ***Top Border Bar****. Now, you will notice that the first three options in the cascading window become active. Likewise, if you have three or more separate windows having three or more different models, then all the options in the cascading window will become active.*

Reset Layout

When you choose this option, all the non-active tabbed windows will close and only the active window will remain open on the screen.

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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 NX Ribbon gives you the ability to customize the interface for a truly optimized experience. NX-specific extensions such as border bars allow you to add additional commands around the perimeter of the graphics window. 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 **Model** template from the **New** dialog box. You can also invoke the Modeling environment from any other opened environment. To do so, choose **Application > Design > Modeling** from the **Ribbon**. Some of the tabs of the Modeling environment are discussed next.

Home Tab

The **Home** tab consists of a series of groups and galleries and they are discussed next.

Construction 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, create datum planes, datum axis, datum coordinate system, and so on. Figure 1-16 shows the **Construction** group. Figure 1-17 shows the expanded **Construction** group.

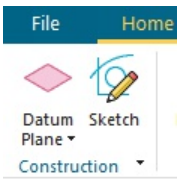


Figure 1-16 *The* ***Construction*** *group*

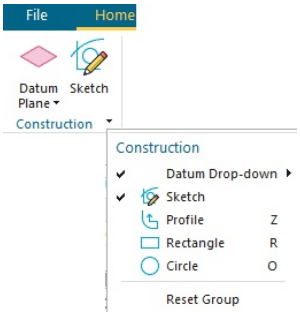


Figure 1-17 *Expanded* ***Construction*** *group*

Note that some of the tools such as **Fillet**, **Chamfer**, **Trim**, and **Extend** are still not available in the expanded **Construction** group. To access all these tools, choose the **Sketch** tool available in the **Construction** 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 sketching environment will become active and you will notice that this environment contains all the sketching tools including dimensioning and geometric constraints arranged in the respective groups, refer to Figure 1-18.

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Figure 1-18 Sketching environment with tools arranged in respective groups

Base Group

The tools in this group are shown in Figure 1-19 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 extrude features, revolved features, rib features, holes, and so on using the tools in this group.

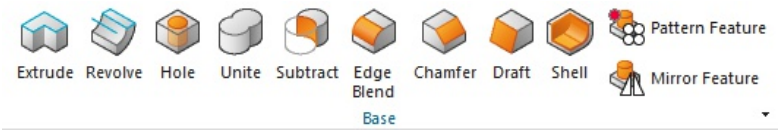


*Figure 1-19 The **Base** group*

Synchronous Modeling Group

The Synchronous modeling technology is one of the significant technologies 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 1-20 shows the **Synchronous Modeling** group.



*Figure 1-20 The **Synchronous Modeling** group*

Curve Tab

The **Curve** tab comprises a series of groups and galleries. You can invoke the sketching environment by using the **Sketch** tool of this tab. Note that you need to customize the group to add the **Sketch** tool to it. Figure 1-21 shows the **Curve** tab**.**



*Figure 1-21 The **Curve** tab*

Surface Tab

You can create surface designs in the Modeling environment as well as in the Shape Studio environment. The **Surface** tab has the tools to create the surface design. Figure 1-22 shows the **Surface** tab.

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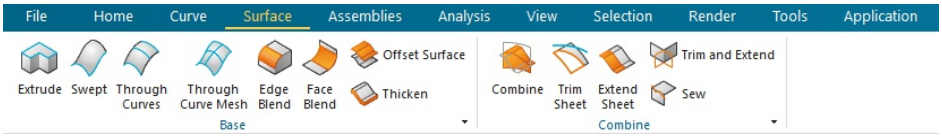


***Figure 1-22** The **Surface** tab*



Tip

*1. 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**.*

2. By default, all tools are not available in a group. Therefore, you may need to customize the group to add those tools. 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 and are 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. Figure 1-23 shows the **Application** tab.



***Figure 1-23** The **Application** tab*

View Tab

The tools in the **View** tab, as shown in Figure 1-24, 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.

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*Figure 1-24 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 components are available in the **Assemblies** tab and are discussed next.

Assemblies Tab

The tools that are used to create an assembly are grouped together in the **Assemblies** tab. The tools of the **Base** 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 1-25 shows the tools in this group.



*Figure 1-25 The **Base** group*

Drafting Environment

To invoke the Drafting environment, choose **Application > Document > Drafting** from the **Ribbon**. You can also invoke this environment by using the templates available in the **Drawing** tab 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 create a new view, generate an orthographic view, section view, and detail view for a solid part or an assembly. Figure 1-26 shows the **View** group.

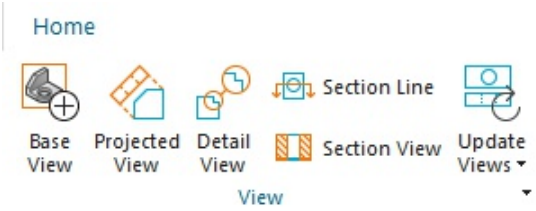


*Figure 1-26 The **View** group*

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Dimension Group

The tools in the **Dimension** group are used to generate various dimensions in the drawing views. Figure 1-27 shows the **Dimension** group.

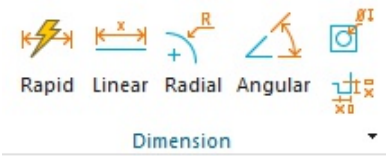


Figure 1-27 *The* ***Dimension*** *group*

Annotation Group

The tools in the **Annotation** group are used to generate the GDT parameters, annotations, symbols, and so on. Figure 1-28 shows the **Annotation** group.

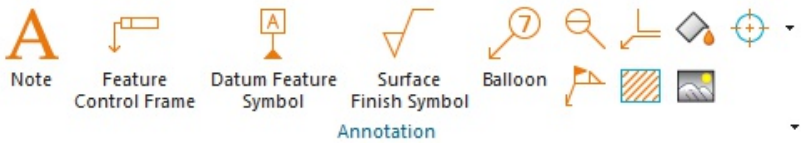


Figure 1-28 *The* ***Annotation*** *group*

Sheet Metal Environment

The tools in the Sheet Metal environment are used to create a sheet metal component. Figure 1-29 shows the groups and tools of Sheet Metal environment.

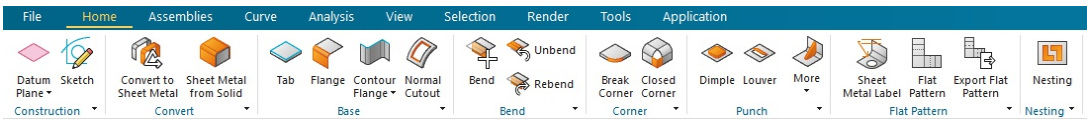


Figure 1-29 *Tools in the Sheet Metal environment*

STATUS BAR

The Status Bar appears at the bottom of the drawing window and comprises of two areas, as shown in Figure 1-30. These areas are discussed next.



Figure 1-30 *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 displays information about the operations that can be carried out.

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.

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 1-31.

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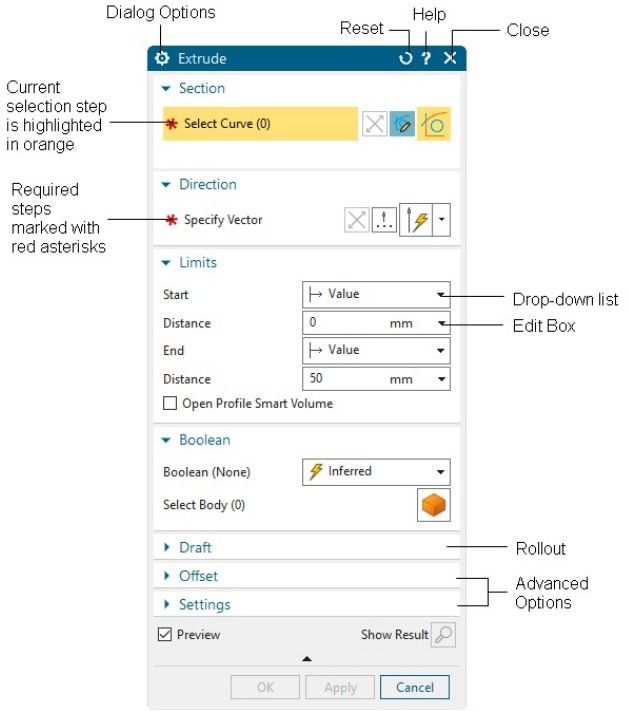


Figure 1-31 *The layout of the* ***Extrude*** *dialog box*

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 **Help** button provides you information about the functions of various options listed in the dialog box. The **Hide Collapsed Groups** option from the **Dialog Options** 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 Options** button, which will be available only after choosing the **Hide Collapsed Groups** option from the **Dialog Options** button. The **Close** button is used to exit the 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.

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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.

Self-Evaluation Test

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

1. The _____ that exists between the environments of NX ensures that any modification made in the model in one environment is automatically reflected in other environments immediately.

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

3. The _____ keeps a track of all the operations that are carried out on the part.

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

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

6. The _____ group is used to generate the GDT parameters, annotations, and symbols.

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

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

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

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

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Answers to Self-Evaluation Test

1. bidirectional associativity, **2.** prt, **3. Part Navigator**, **4.** Fixed, **5. Rotate**, **6. Annotation**, **7.** T, **8.** T, **9.** T, **10.** T

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