



Chapter 1

Introduction

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the different environments in NX.*
- *Understand the system requirements for NX.*
- *Understand how to start a new file in NX.*
- *Understand the important terms and definitions.*
- *Understand the 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.*

INTRODUCTION TO NX 4

Welcome to **NX 4** (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 the tremendous improvement in this latest release.

NX, developed by UGS 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 state-of-the-art user interface, NX delivers innovative technologies for maximum productivity and creativity, from 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 is 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 in the **Gateway** application. It allows you to examine the geometry and drawing views that have been created. In the **Gateway** application, 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 **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 this is a sketch. The sketch for the features is drawn in the **Sketcher Task** environment that can be invoked within the **Modeling** environment by choosing the **Sketch** button from the **Form Feature** toolbar. You can draw the sketch using the tools in this environment. While drawing a sketch, various applicable constraints are automatically applied to it. You can also apply additional constraints and dimensions manually. After drawing the sketch, exit the **Sketcher Task** environment and convert it into a feature. The tools in the **Modeling** environment can be used to convert the sketch into a feature. You are also provided with other tools to apply the placed features, such as fillets, chamfers, taper, and so on. 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.

Assemblies Environment

The **Assemblies** environment is used to assemble the components using the assembly constraints available in this environment. There are two type of assembly design approaches:

1. Bottom-up
2. 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 in the **Assemblies** environment.

You can also assemble an existing assembly with the current assembly. The **Check Clearance Analysis** provides the interference check 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 bidirectional associative in nature. You can also generate the Bill of Material (BOM) and balloons to 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 on your system:

- System unit: An Intel Pentium III or Pentium 4 based workstation running Microsoft 2000 Professional Edition or Windows XP Professional Edition.
- Memory: 256 MB of RAM is the minimum recommended for all applications. 512 MB of RAM is recommended for DMU applications.
- Disk drive: 4 GB Disk Drive space (Minimum recommended size)
- Internal/External drives: A CD-ROM drive is required for the program installation.

- Display: A graphic color display compatible with the selected platform-specific graphic adapter. 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** on the desktop of your computer. You can also choose **Start > All Programs > UGS NX 4.0 > NX 4.0** from the taskbar menu, as shown in Figure 1-1.

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

Choose **File > New** from the menu bar; the **New Part File** dialog box will be displayed. Enter the name of file in the **File Name** edit box and choose the **OK** button; the **Gateway** application will be displayed on the screen, as shown in Figure 1-3.

In this text book, the **Master Model Template** is used to create a new file. The procedure for starting a new file using the Master Model Template is discussed in the next chapter.

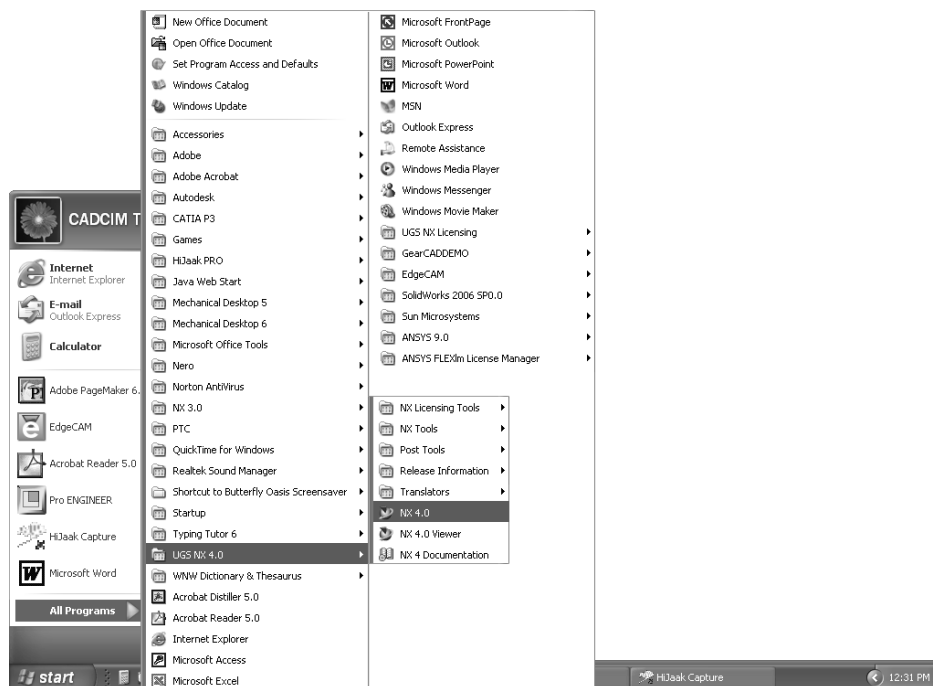


Figure 1-1 Starting NX 4 using the taskbar shortcuts

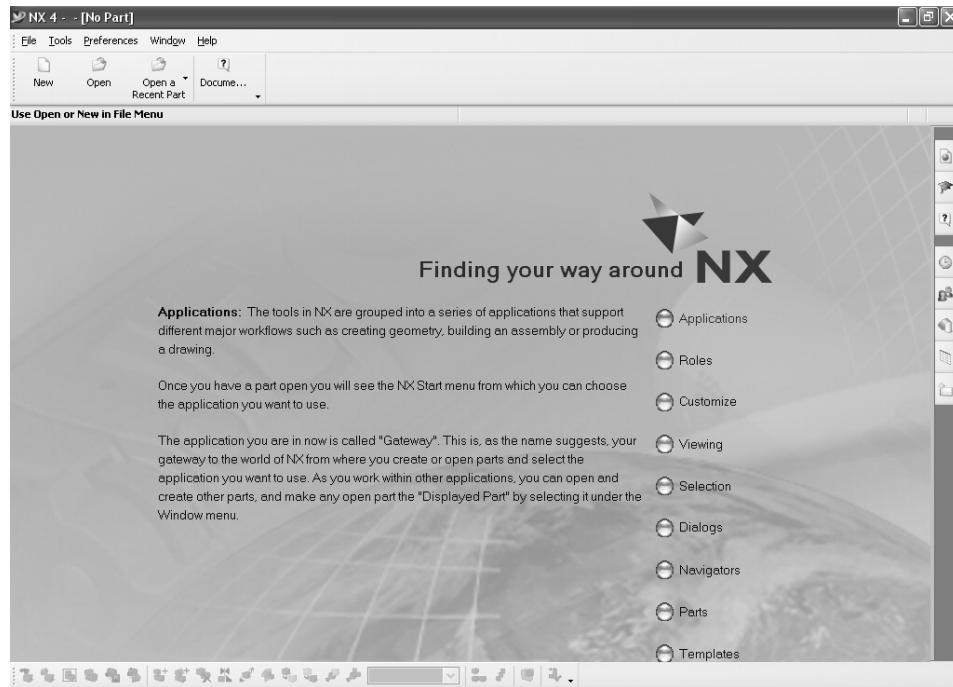


Figure 1-2 The initial screen that appears after starting NX 4

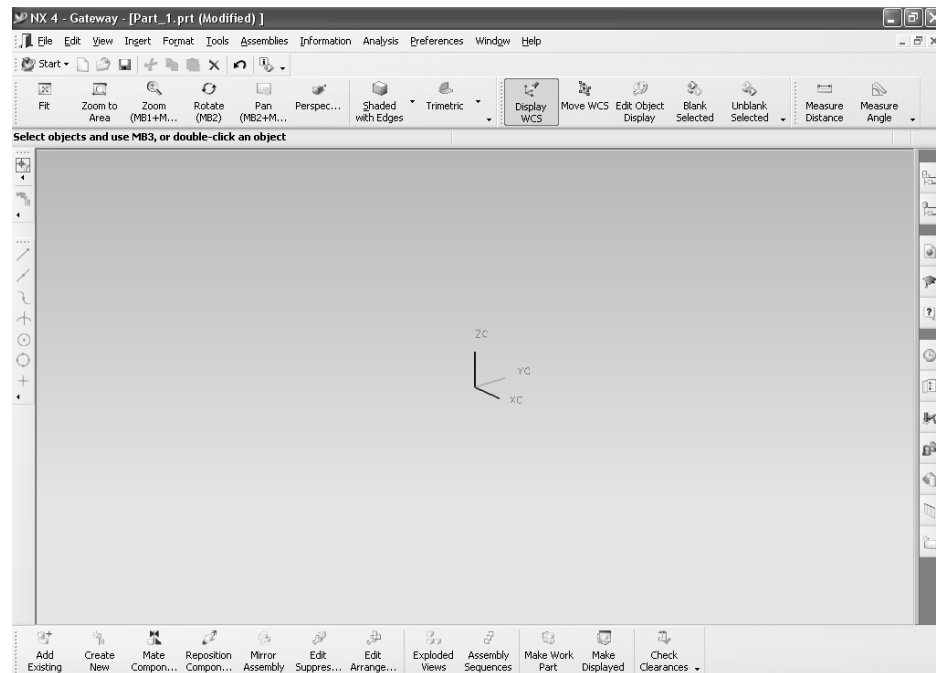


Figure 1-3 The screen that appears after creating a new part file

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. These features understand their fit and function properly and, therefore, can be modified any time during the design process. 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 greater 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 design process. This property makes the designing process very easy. For example, consider the design of the body of a pipe housing shown in Figure 1-4.

To change the design by modifying the diameter of the holes and their number on the front, top, and the bottom face, you have 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.

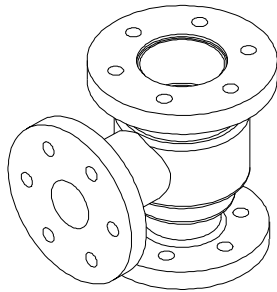


Figure 1-4 Body of a pipe housing

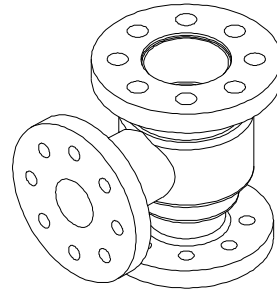


Figure 1-5 Modified body of pipe housing

Bidirectional Associativity

As mentioned earlier, NX has different environments such as the **Modeling** environment, **Assemblies** environment and the **Drafting** environment. The bidirectional associativity that exists between all these environments ensures that any modification made in the model in any one 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 **Assemblies** and the **Drawing** environments. 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 **Assemblies** 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.

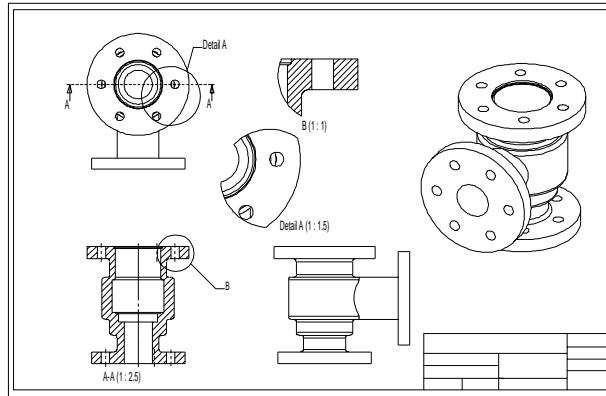


Figure 1-6 The drawing views of the body part before making the modifications

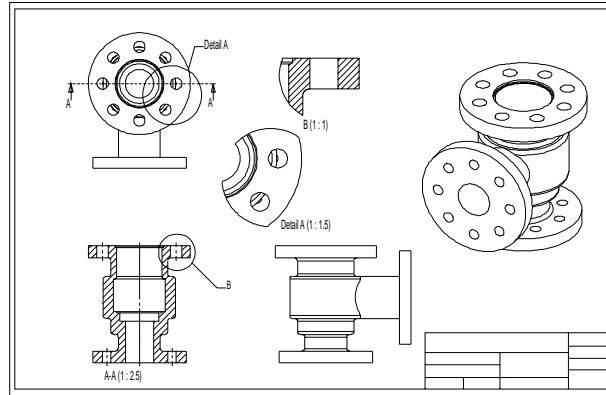


Figure 1-7 The drawing views after making the modifications

*.prt

*.prt is a file extension associated with all the files that are created in the **Sketcher**, **Modeling**, **Shape Studio**, **Assemblies**, 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 right 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 from the **Resource Bar** is used to activate the required role. To activate this role, choose the **Roles** palette from the **Resource Bar** and click on the **System Defaults** option; the flyout menu will be displayed. Click on the **Essential with full menus** file to activate role. Note that this book will follow the **Essential with full menus** role. The Figure 1-8 shows the **Roles** navigator that appears when you choose the **Roles** tab in the **Resource Bar**.



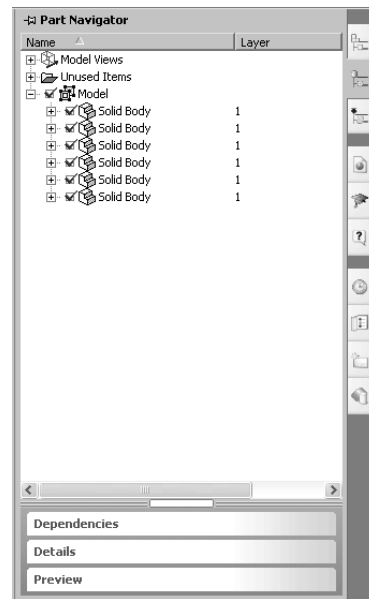
Figure 1-8 The Roles navigator

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



*Figure 1-9 The **Part** navigator*

size and location with respect to the other elements or reference geometries. There are two types of constraints in NX. The constraints in the **Sketcher** environment are called geometric constraints and 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 **Assemblies** 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 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.

The constraints in the **Sketcher** environment are discussed next.

Fixed

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

Coincidence

This constraint is used to make two points, two lines, a point and a line, or a point and a curve, coincident.

Concentric

This constraint is used to make two circles, arc, an arc and a circle, a point and a circle, or a point and an arc concentric.

Colinear

This constraint is used to make two lines pass through the same straight line.

Point on Curve

This constraint is used to make the point lie on a curve.

Point on String

This constraint is used to make the point lie on an extended string.

Midpoint

This constraint forces a selected point to be placed at the midpoint of the selected line or arc.

Horizontal

The **Horizontal** constraint forces the selected line segment to become horizontal.

Vertical

The **Vertical** constraint forces the selected line segment to become vertical.

Parallel

The **Parallel** constraint is used to force any two selected line segments to become parallel to each other. The selected segments can also be an ellipses.

Perpendicular

The **Perpendicular** constraint is used to force any two selected line segments to become perpendicular to each other. The selected segments can also be an ellipses.

Tangent

Tangent constraint is used to force the selected line segments or curves to become tangent to another curve.

Equal Length

The **Equal Length** constraint is used to force two or more selected line segments to become equal in length.

Equal Radius

The **Equal Radius** constraint is used to force two or more selected arc segments to become equal in length.

Constant Length

This constraint is used to make the length of the line constant. You can not modify the length of this line by adding dimensions.

Constant Angle

This constraint is used to define a line as having a constant angle.

Mirror

This constraint is used to define two objects such that they are mirror images of each other.

Assembly Constraints

The constraints in the **Assemblies** 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. The constraints in this environment are discussed next.

Mate

This constraint is used to force two selected entities to create a contact with each other. These entities can be the two faces, two planes, or a plane and a face.

Align

This constraint is used to force two selected entities to be coplanar. These entities can be the two faces, two planes, or a plane and a face.

Angle

This constraint is used to place two selected entities at an angle with respect to each other. These entities can be the central axes of circular components, two faces, two planes, or a combination of the axis and face, a plane and a face, or an axis and a plane.

Parallel

This constraint is used to force two entities to become parallel.

Perpendicular

This constraint is used to force two selected entities to become perpendicular.

Center

This constraint is used to force two selected components to become concentric. The components must have their own center axes.

Distance

This constraint is used to place two selected faces at a distance with respect to each other.

Tangent

This constraint is used to place two selected components tangent to each other.

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 make using the solid feature. 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 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 the following two types:

1. Sketch-based Features
2. Placed-features

The sketch-based features are ones that 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 the Absolute Coordinate System origin, which is (0,0,0).

UNDERSTANDING THE FUNCTIONS OF THE MOUSE BUTTONS

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

1. The left mouse button is used to make a selection by simply selecting a face, surface, sketch, or object from the geometry area or from the **Part Navigator**. For a multiple selection, 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 mouse button and then click the right mouse button 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 pressing and holding 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 press and hold the middle mouse button to invoke the **Zoom** tool. Alternatively, press and hold the right 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 button mouse in performing the zoom functions.

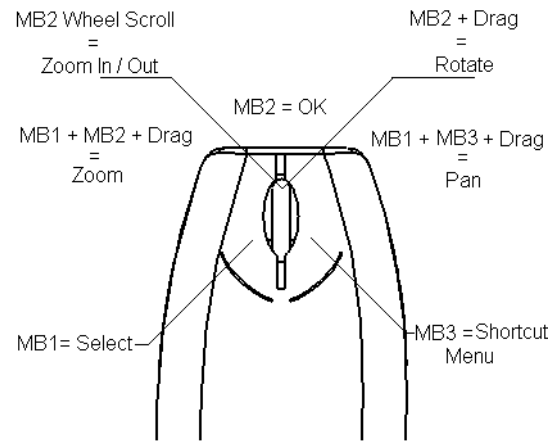


Figure 1-10 Functions of the mouse buttons

TOOLBARS

NX offers a user-friendly design environment by providing specific toolbars to each environment. Therefore, it is important that you get acquainted with the various standard toolbars and buttons that appear in the 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. Invoke the different environments such as **Modeling** environment, **Assemblies** environment, and **Drafting** environment and so on to complete the design.

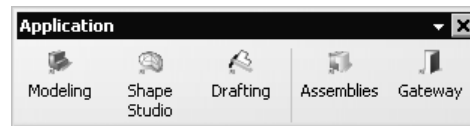


Figure 1-11 The Application toolbar

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 to 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, and invoke the help topics.

Status Bar

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

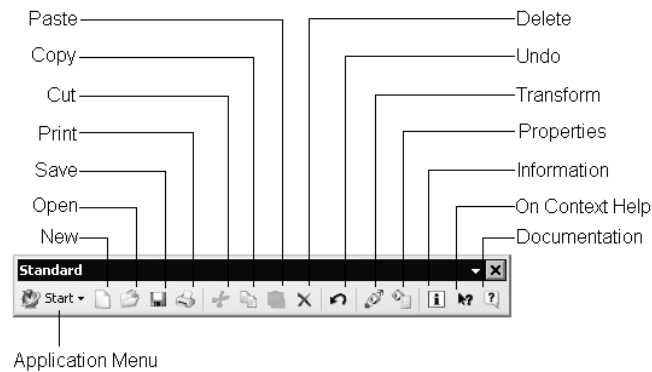


Figure 1-12 The Standard toolbar



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

It gives information about the operations being done. It also gives information about the picked entity.

Modeling Environment Toolbars

You can invoke the **Modeling** environment by choosing the **Modeling** button from the **Application** toolbar. Alternatively, you can choose **Application > Modeling** from the menu bar. The toolbars in the **Modeling** environment are discussed next.

View Toolbar

The tools in the **View** toolbar, shown in Figure 1-14, are used for manipulating the views of the model. The **View** toolbar is available in all the environments.

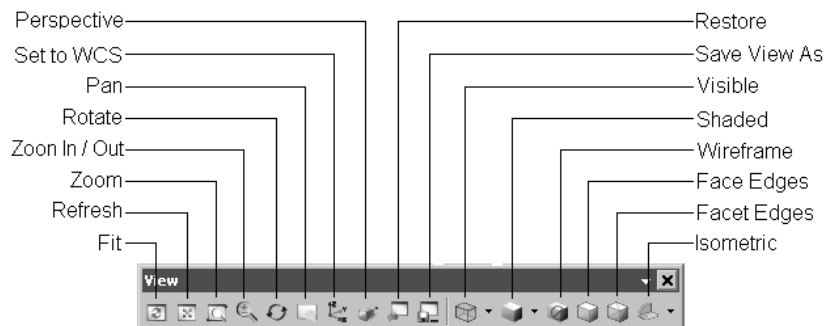


Figure 1-14 The View toolbar

**Note**

Some of the buttons are not available in the **Drafting** environment.

Form Feature Toolbar

The tools in this toolbar, shown in Figure 1-15, are used to convert a sketch drawn in the **Sketcher** 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.

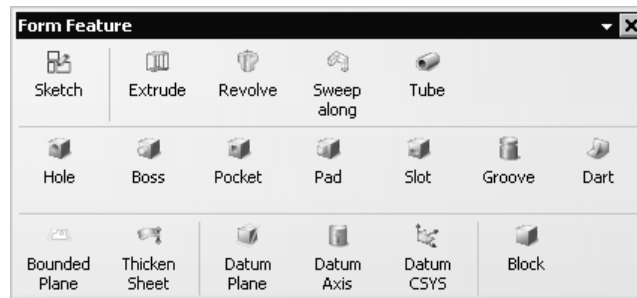


Figure 1-15 The **Form Feature** toolbar

Sketcher Environment Toolbar

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

Sketcher Toolbar

The **Finish** button from the **Sketcher** toolbar are used to switch back to the **Modeling** environment, where you can convert the sketch into a feature. Figure 1-16 shows the buttons that are available in it.

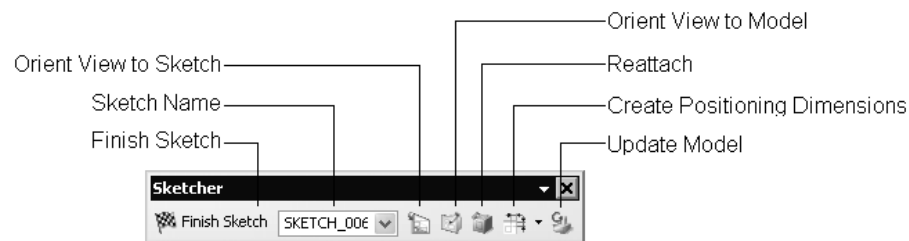


Figure 1-16 The **Sketcher** toolbar

Sketch Curve Toolbar

The tools in the **Sketch Curve** toolbar are used to draw the sketches. It is one of the most important toolbars in the **Sketcher** environment. Figure 1-17 shows the buttons in it.

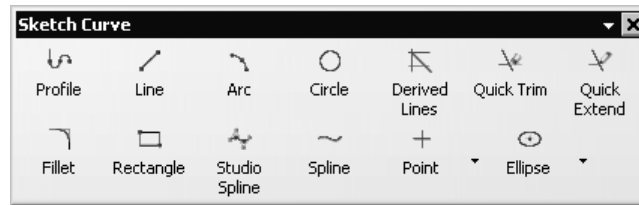


Figure 1-17 The Sketch Curve toolbar

Sketch Constraints Toolbar

The tools in the **Sketch Constraint** toolbar are used to apply constraints to the geometric entities and assign dimensions to a drawn sketch. You can make a sketch fully defined using the tools in this toolbar. Figure 1-18 shows the buttons in the **Sketch Constraints** toolbar.

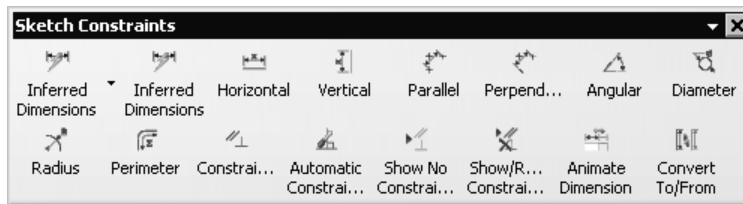


Figure 1-18 The Sketch Constraints toolbar

Sketch Operations Toolbar

The tools in the **Sketch Operations** toolbar are used to edit the drawn sketches. Figure 1-19 shows the buttons in the **Operation** toolbar.



Figure 1-19 The Sketch Operations toolbar

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

Feature Operation Toolbar

The tools in the **Feature Operation** toolbar are used to apply the placed features such as taper, edge blend, soft blend, hollow, and so on. Figure 1-20 shows the buttons in the **Feature Operation** toolbar.

Surface Design Toolbars

You can create the surface design in the same **Modeling** environment. A separate environment is not required to create the surface design. The tools used to create the solid bodies are also used to create the surface bodies. Some of the toolbars used to draw the of the surface design are discussed next.

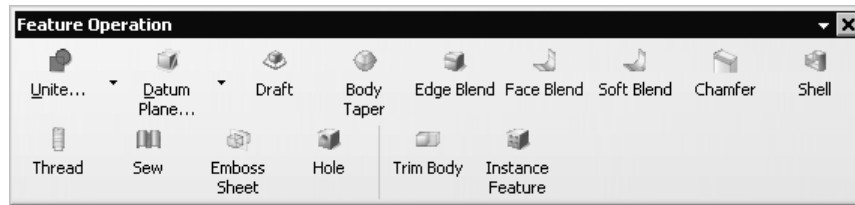


Figure 1-20 The Feature Operation toolbar

Surface Toolbar

The tools in the **Surface** toolbar are used to create complicated surfaces. Figure 1-21 shows the **Surface** toolbar.

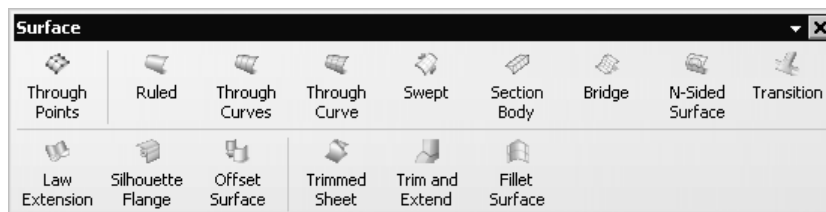


Figure 1-21 The Surface toolbar

Free Form Shape Toolbar

The tools in the **Free Form Shape** toolbar are used to create the surfaces using different options. Figure 1-22 shows the **Free Form Shape** toolbar.



Figure 1-22 The Free Form Shape toolbar

Assemblies Environment Toolbars

You can create the assembly in the same **Modeling** environment. A separate environment is not required to create the assembly design. The toolbars that are used for the assembly can be invoked by choosing **Application > Assemblies** from the menu bar. Alternatively, choose the **Assemblies** button from the **Application** toolbar. The toolbars of 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 1-23 shows the buttons in the **Assemblies** toolbar.

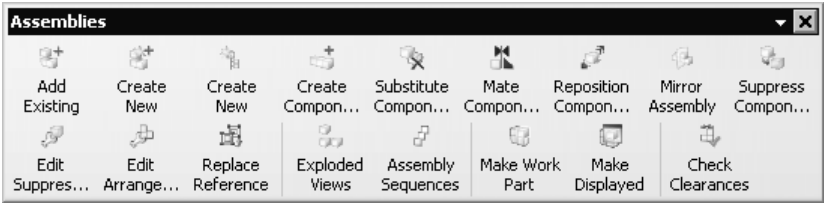


Figure 1-23 The Assemblies toolbar

Drafting Environment Toolbars

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

Drawing Layout Toolbar

The tools in the **Drawing Layout** toolbar are used to insert a new sheet, create a new view, generate an orthographic view, section view, and detail views for a solid part or an assembly. Figure 1-24 shows the **Drawing Layout** toolbar.

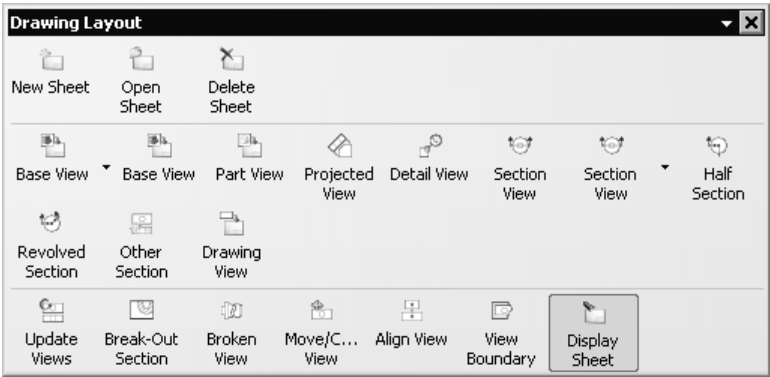


Figure 1-24 The Drawing Layout toolbar

Dimension Toolbar

The tools in the **Dimension** toolbar are used to generate various dimensions to the assembly drawings. Figure 1-25 shows the **Dimension** toolbar.

Drafting Annotation Toolbar

The tools in the **Drafting Annotation** toolbar are used to generate the GDT parameters, annotations, symbols, and so on. Figure 1-26 shows the **Drafting Annotation** 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.

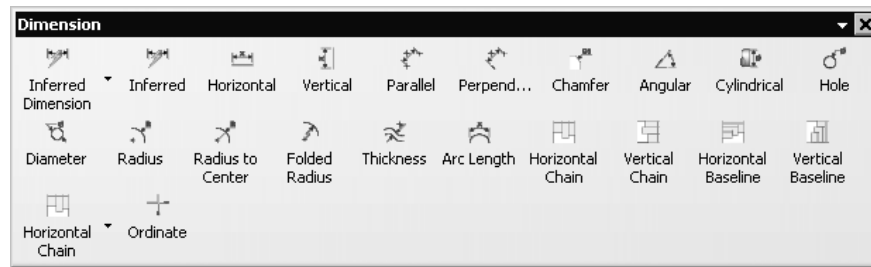


Figure 1-25 The *Dimension* toolbar

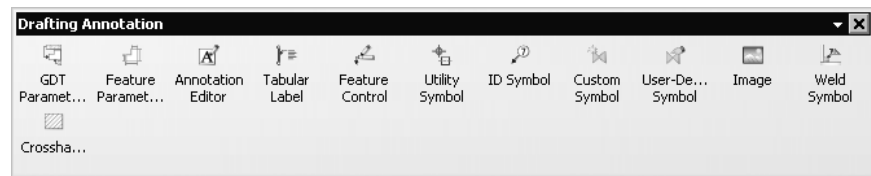


Figure 1-26 The *Drafting Annotation* toolbar

Hot Key	Function
CTRL+Z	Invoke the Undo tool
CTRL+Y	Invoke the Repeat tool
CTRL+S	Save the current document
F5	Refresh the Drawing window
F1	Invoke the NX Help tool
F6	Invoke the Zoom tool
F7	Invoke the Rotate tool
CTRL+M	Invoke the Modeling environment
CTRL+ALT+W	Invoke the Assemblies environment
CTRL+SHIFT+D	Invoke the Drafting environment

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 color scheme, choose **Preferences > Visualization** from the menu bar; the **Visualization Preferences** dialog box will be displayed.

Choose the **Edit Background** button from this dialog box; the **Edit Background** dialog box will be displayed. Select the **Plane** radio button from the **Shaded Views** and **Wireframe Views** areas. Next, choose the color swatch at the right side of the **Plain Color** option; the color dialog box will be displayed. Choose the **White** color from the **Custom Colors** options and choose **OK** to apply the scheme to the NX environment. Note that all the files that you open henceforth will not use this color scheme.

**Note**

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

Self-Evaluation Test

Answer the following questions and then compare your answers with those given at the end of the chapter:

1. The **Modeling** environment of NX is a parametric and feature-based environment. (T/F)
2. In the top-down assembly design approach, the previously created components are assembled together to maintain their design intent. (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 right 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 the 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. Press and hold the middle mouse button to invoke the _____ tool.
10. The _____ toolbar is used to generate the GDT parameters, annotations, and symbols.

Answers to Self-Evaluation Test

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