Chapter 2

Introduction to NX Nastran

Learning Objectives

After completing this chapter, you will be able to:

- *Understand basic concepts of NX Nastran*
- *Understand different environments in NX Nastran*
- *Understand the system requirements for NX Nastran*
- *Start NX Nastran*
- *Understand important terms and definitions in NX Nastran*
- *Understand the functions of mouse buttons*
- *Understand default tabs in different environments*
- *Use dialog boxes in NX Nastran*
- *Use the NX Nastran Help*

INTRODUCTION TO NX Nastran

NX Nastran is a premium computer-aided engineering (CAE) tool that manufacturers use worldwide for their critical engineering computing needs. For over 30 years, NX Nastran has been the analysis solution in almost every major industry including aerospace, defense, automotive, shipbuilding, heavy machinery, medical, and consumer products.

In CAE, often you can work with data imported from different CAD softwares. You can easily modify a design using NX CAD software and then use the modification as a design variable to arrive at an optimal design.

NX Nastran has the following analysis capabilities:

1. Linear statics (including inertia relief)
2. Normal modes and buckling
3. Heat transfer (steady-state and transient)
4. Transient response
5. Frequency response
6. Response spectrum and random response
7. Geometric nonlinear static and transient response
8. Material nonlinear static and transient response
9. Design optimization and sensitivity (including dynamic and shape optimization)
10. Composite materials
11. Acoustic response
12. Aeroelasticity
13. Superelements
14. Complex eigen analysis
15. Axisymmetric analysis
16. Cyclic symmetry

In NX Nastran you can analyze the basic and as well as complex geometries using different environments. Each environment consists of a set of tools that allows the user to perform specific analysis tasks in a particular area. After starting a new part file, you can invoke the required environment of NX.

BASIC ENVIRONMENTS IN NX

The basic environments in NX are Modeling, Shape Studio, Drafting, Assembly, Sheet metal, and the Manufacturing. Some of 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 which can be drawn directly in this environment by using the tools available in the **Direct Sketch** group of the **Home** tab. The sketch can also be drawn in the Sketching environment. The Sketching 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. When a sketch is drawn, various applicable constraints and dimensions get automatically applied to it. Additional constraints and dimensions can also be manually applied. After drawing the

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

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 the other tools available in this environment. These features are called 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 with the difference that the tools in this environment are used to create surfaces, both basic and advanced. This environment also provides surface editing tools that are used to manipulate surfaces to obtain required shape. This environment is useful for conceptual and industrial designs.

Assembly Environment

The Assembly environment is used to assemble 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, you can assemble the components that were created earlier whereas 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 performed analysis in this environment provides the facility to check the interference and clearance between the components in an assembly.

SYSTEM REQUIREMENTS

The following are the minimum system requirements to ensure smooth functioning of NX Nastran on your system:

- Operating System: Windows 64-bit (Windows XP 64 SP2, Windows Vista 64 SP1, Windows 7, Windows 8, Windows HPC Server 2008 R2)
- Platform: Intel Pentium class, Intel 64 or AMD 64
- Memory: 4 GB or more
- DVD drive: For installing the software
- Graphics adapter: should be capable of supporting 1024x768 High Color (16-bit)
- Microsoft Internet Explorer 6.0 or higher

STARTING NX NASTRAN 9.0

To start NX Nastran 9.0, double-click on the shortcut icon of NX 9.0 displayed on the desktop of your computer. After the necessary files are loaded and the licenses are verified, the initial screen of NX 9 will be displayed.

The default initial screen of NX 9.0 is shown in Figure 2-1. This screen displays information about NX 9.0 which helps you learn more about NX 9.0. You can also view other related information by moving the cursor over the topics displayed on the left of the NX 9.0 screen.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Figure 2-1 *The initial screen that appears after starting NX 9.0*

STARTING A NEW DOCUMENT IN NX NASTRAN 9.0

Ribbon:	Home > Standard > New
Menu:	File > New

To invoke the Advanced Simulation environment of NX Nastran, choose **Menu > File > New** from the **Top Border Bar**; the **New** dialog box will be displayed with the **Model** tab chosen, as shown in Figure 2-2. The **New** dialog box contains tabs to switch between different working environments like Modeling, Drawing, Simulation, Manufacturing, and so on.

To enter in NX Nastran, choose the **Simulation** tab and select **NX Nastran** from the **Name** column for the **FEM** type from the **Template** rollout, if not selected by default, refer to Figure 2-3.

The rollouts available in the **Simulation** tab for invoking NX Nastran are discussed next.

Templates Rollout

In this rollout, different FEM and SIM type templates are available for different solvers. You can also select the **Millimeters** or **Inches** as a unit from the **Units** drop-down list available in the **Filters** sub-rollout.

New File Name Rollout

In this rollout, the **Name** and **Folder** edit boxes are available. You can enter the name of FEM or SIM in the **Name** edit box and the location to save the file in the **Folder** edit box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Figure 2-2 *The* ***New*** *dialog box with the* ***Model*** *tab chosen*

Figure 2-3 *The* ***New*** *dialog box with the* ***Simulation*** *tab chosen*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

After defining the required parameters in these rollouts, choose the **OK** button; the **New FEM** dialog box with the FEM environment will be displayed, refer to Figure 2-4.

Figure 2-4 *The **New FEM** dialog box with the FEM environment*

For importing any NX model in the FEM environment, select the **Associate to Master Part** check box from the **CAD Part** rollout of the **New FEM** dialog box; the **CAD Part** rollout will be modified and the **Open Part** button will be displayed, as shown in Figure 2-5. Choose this button; the **Open** dialog box will be displayed. Select the part to import and choose the **OK** button from the **Open** dialog box. Next, choose the **OK** button from the **New FEM** dialog box; the part will get imported in the FEM environment, as shown in Figure 2-6.

Note

*1. If you choose a file type other than NX such as ***.igs**, ***.stp**, you need to leave the **Associate to Master Part** check box cleared and then choose the **OK** button from the dialog box; the FEM environment window will be displayed, refer to Figure 2-6. Now, open the part from this window to import the non-NX model in the FEM environment.*

You will learn more about the importing and exporting of the CAD model in NX in the later chapters of this textbook.

*2. If you want to create an Idealized Part, then select the **Create Idealized Part** check box from the **Idealized Part** sub-rollout in the **CAD Part** rollout of the **New FEM** dialog box.*

You will learn more about creating an Idealized Part later in the textbook.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Figure 2-5 *The modified* ***New FEM*** *dialog box*

Figure 2-6 *The part imported in the FEM environment*

In the FEM environment, you can define materials, meshing, and other physical properties of the model. The model after defining materials and applying mesh is shown in Figure 2-7.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Figure 2-7 *Meshed model in the FEM environment*

For applying loads, constraints, and other boundary conditions, you need to switch to the Simulation environment. To do so, choose the **New Simulation** button from the **Change Window Drop-down** of the **Home** tab; the **New Part File** window will be displayed. Select the **NX Nastran** template with the **Sim** type from the **Template** rollout if not selected by default, as shown in Figure 2-8.

Figure 2-8 *The **New Part File** window*

Choose the **OK** button from the **New Part File** window; the **New Simulation** dialog box with the Simulation environment will be displayed. Choose the **OK** button from the **New Simulation**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

dialog box; the **Solution** dialog box will be displayed. In this dialog box, you can describe the analysis details. Choose the **OK** button from the dialog box; you will enter in the Simulation environment. The Simulation environment with the meshed model is shown in Figure 2-9. In this environment, you can apply boundary conditions, loads, and solve the analysis.

Figure 2-9 *The Simulation environment with meshed model*

Note

You will learn more about the Simulation environment and its dialog boxes later in the textbook.

IMPORTANT TERMS AND DEFINITIONS

Some important terms used in NX Nastran are discussed next.

Advanced Simulation

The Advanced Simulation is a comprehensive finite element modeling and results visualization application. It includes pre- and post-processing and supports a broad range of product performance evaluation solutions. Advanced Simulation provides numerous additional features that support advanced analysis processes.

1. Advanced Simulation provides separate simulation and FEM files that help in the development of FE models across a distributed work environment.

2. Advanced Simulation supports a complete range of element types (0D, 1D, 2D, and 3D) to control specific part meshing.

3. Advanced Simulation includes a number of geometry modification tools to manipulate any CAD geometry for making its analysis easier and faster.

fem

The *fem* file used in NX Nastran contains information about the nodes and elements in the

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

finite element model. It also includes physical properties, materials, and beam element sections of the model. The file contains the idealized (modified) model that you create from the master CAD part.

sim

The *sim* (Simulation) file contains the simulation data such as loads, constraints, and all motion objects (such as links, joints, and connectors). You can create multiple simulation files for a single *fem* file.

Note

If any of the properties is missing in the fem or sim file while solving the analysis, the ***Information*** *window will be displayed with errors and warnings, refer to Figure 2-10.*

You will learn more about solving an analysis later in the textbook.

Figure 2-10 *The* ***Information*** *window*

Resource Bar

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

Simulation Navigator

The **Simulation Navigator** is located at top of the Resource Bar. It displays the following files and steps associated with analysis.

1. FEM file
2. Associated CAD geometry
3. Mesh collectors and meshes
4. Loads, constraints, and objects that define boundary conditions
5. Solution and results

Using the **Simulation Navigator**, you can review the structure, content, and status of your analysis. You can also show and hide the geometry, meshes, and boundary conditions of the model. A partial view of the **Simulation Navigator** is shown in Figure 2-11.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Figure 2-11 Partial view of the ***Simulation Navigator***

Simulation File View

The **Simulation File View** is located below the **Simulation Navigator**. You can use the **Simulation File View** in the following ways:

1. View all the loaded parts as well as all FEM and Simulation files in sequential order, as shown in Figure 2-12.

2. Create Idealized part without manipulating the master part.

3. Create multiple FEM and Simulation files for any design or idealized part.

You can switch to any simulation environment by choosing any option from the **Simulation File View**. To do so, right-click on a simulation step; a shortcut menu will be displayed, refer to Figure 2-13. Choose the **Make Displayed Part** option from the shortcut menu; the simulation environment will be invoked. You can also switch to simulation environment by double-clicking.

Figure 2-12 The ***Simulation File View***

Figure 2-13 Shortcut menu displayed

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Post Processing Navigator

The **Post Processing Navigator** is available in the **Resource Bar** below the **Simulation Navigator.** The **Post Processing Navigator** is used to manage, view, and interrogate the results of an analysis. Using the **Post Processing Navigator**, you can:

1. Create nodal and elemental contour plots of results on the model and also import results.
2. Create cross sections in post views to show internal variation in the larger portion of part.
3. Annotate results to show maximum and minimum variation points.
4. Identify the values of selected nodes and elements and export them for further analysis.
5. Create and display graphs of result data across selected nodes.
6. Quickly generate a contour plot of the selected results or result components.
7. Animate post views.
8. Control the display of results based on meshes or groups.

You can open the **Post Processing Navigator** to get the current simulation results. To do so, right-click on the **Results** step in the **Simulation Navigator**; a shortcut menu will be displayed, refer to Figure 2-14. From this shortcut menu, choose the **Open** option from the menu; the **Post Processing Navigator** will be displayed with the current simulation results, refer to Figure 2-15.

Figure 2-14 *Shortcut menu displayed on choosing the* ***Results*** *step of* ***Simulation Navigator***

Note

You can also import an existing results file in the ***Post Processing Navigator*** *using the* ***Imported Results*** *option shown in the navigator. You will learn more about importing results in the* ***Post Processing Navigator*** *later in the textbook.*

Part Navigator

The **Part Navigator** located in the Resource Bar keeps a track of all the operations related to model preparation. Figure 2-16 shows the part navigator that appears when you choose the **Part Navigator** tab in the Resource Bar.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Figure 2-15 The ***Post Processing Navigator***

Figure 2-16 The ***Part Navigator***

UNDERSTANDING THE FUNCTIONS OF THE MOUSE BUTTONS

To work in NX Nastran environments, it is necessary that you understand the functions of mouse buttons. The efficient use of the mouse buttons along with the CTRL key can reduce the time required to complete any design task. Different combinations of the CTRL key and the mouse buttons are described next:

1. The left mouse button helps you make a selection by simply selecting a face, surface, loads, or constraints from the geometry area or from the Navigators. For multiple selections, press the CTRL key along with the left mouse button.

2. The right mouse button is used to invoke shortcut menu which has different options and tools.

3. Press and hold the middle and 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 2-17 shows the use of a three buttons 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 2-17 shows the use of the three buttons mouse in performing the rotate operation.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Figure 2-17 *Functions of the mouse buttons*

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 2-17 shows the use of three mouse buttons in performing the zoom functions.

QUICK ACCESS TOOLBAR

The **Quick Access Toolbar** toolbar is common to all environments of NX Nastran, refer to Figure 2-18. 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 content from the clipboard to some selected location, undo, redo, search a tool, and invoke the help topics.

Figure 2-18 *The* ***Quick Access Toolbar***

RIBBON

The Ribbon in the NX Nastran interface comprises of a series of tabs which group various tools and options based on their functionality. These tabs and groups are displayed depending upon the environment invoked. The different environments and some of their respective tabs and groups are discussed next.

FEM Environment

In the FEM environment, you can define the type of material, mesh, and other physical properties of the model from the **Home** tab. This tab is discussed next.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Home Tab

The tools in the **Home** tab, as shown in Figure 2-19, are used in the FEM environment to:

1. Entering into the Simulation environment
2. Assign material to the given model
3. Mesh the given model
4. Define connection between two or more parts
5. Refining the model for better mesh quality
6. Check element or mesh quality

Figure 2-19 *Partial view of the* ***Home*** *tab in the FEM environment*

The **Context** group of the **Home** tab is discussed next.

Context Group

This group available in the **Home** tab and is used to enter in the Simulation environment and to create a new simulation file, refer to Figure 2-20. The tools in the **Context** group will be modified in the Simulation environment.

Figure 2-20 *The* ***Context*** *group*

Simulation Environment

The Simulation environment is used for defining loads, constraints, and other boundary conditions. The **Home** tab of this environment is discussed next.

Home Tab

The **Home** tab in the Simulation environment is shown in Figure 2-21. The tools in the **Home** tab are used in the Simulation environment to:

1. Activating the **Simulation** tool to re-enter in the Simulation environment from the FEM environment
2. Managing materials in the model
3. Managing modeling objects
4. Defining loads and constraints
5. Defining simulation object type and its operational region
6. Creating solutions and verify the model analysis steps
7. Solving the analysis and create the analysis report
8. Checking the element or mesh quality

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

*Figure 2-21 Partial view of the **Home** tab in the simulation environment*

The **Context** group of the **Home** tab is discussed next.

Context Group

After entering in the Simulation environment, the **Context** group gets modified, as shown in Figure 2-22. The tools in the **Context** group are used to activate meshing and to again enter in the FEM environment from the Simulation environment.

*Figure 2-22 The **Context** group*

Results Tab

After the analysis of model is performed and results are generated the **Post Processing Navigator** is displayed and the **Results** tab gets available along with the **Post Processing**, **Animation**, **Manipulation** and other groups in the Simulation environment, as shown in Figure 2-23.

*Figure 2-23 The **Results** tab*

The tools in the **Results** tab are used to:

1. Return to the Simulation environment
2. Edit the given view and manipulate the result
3. Render the result
4. Create the annotation
5. Animating results

View Tab

The **View** tab is common to all environments of NX Nastran. The tools in the **View** tab, as shown in Figure 2-24, are used for manipulating the views of the model.

*Figure 2-24 The **View** tab*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

STATUS BAR

The Status Bar appears at the bottom of the drawing window and comprises of two areas and a button, as shown in Figure 2-25. These areas and button are discussed next.

Figure 2-25 *The Status Bar*

Cue Line Area

The cue line area is the prompt area where prompts appear for completing the tool task.

Status Area

This area provides information about the operations to be carried out in the interface.

Enters or exits full screen mode

If you choose this button, the graphics window will be maximized and you can get a full screen display. For getting the default screen display, you need to choose this button again.

Selection Bar

The **Selection** bar, as shown in Figure 2-26, appears below the **Ribbon**. This bar is used for easy selection of the model features like faces or edges.

Figure 2-26 *The* ***Selection*** *bar*

Tip

By default, all the 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 list will be displayed. Click on the tool to be added or removed from the group. Note that the tick mark 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 the* ***Ribbon Options*** *arrow available at its bottom right corner.*

DIALOG BOXES IN NX Nastran

To perform a simulation, you need to follow certain processes in an order. The steps to perform these processes are placed in a top-down order in the corresponding dialog boxes, refer to Figure 2-27. In the dialog box, the current selection step is highlighted in orange. The required steps are marked with asterisks and the completed steps are marked with green check marks.

The **Reset** button is used to reset the dialog box to its initial settings. The **Close** button is used to exit the dialog box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

*Figure 2-27 The **3D Tetrahedral Mesh** dialog box*

USING HELP IN NX Nastran

In NX Nastran, there are different ways to access help. These methods are discussed next.

Note

*For using the NX Nastran help file, you must install the **Documentation** file in your system.*

NX Help

To access help in NX Nastran, choose **Menu > Help > NX Help** from the **Top Border Bar**; the **NX 9.0 Help** menu will open in a browser window, as shown in Figure 2-28.

*Figure 2-28 The **NX 9.0 Help** opened in the browser window*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

The help window is divided into two parts: **Product area** and **NX 9 Help**. The **Product area** provide with hyperlinks of various topics and the **NX 9 Help** provide product description with summary. Through this browser window, NX provides description of each module provided in the software package. You can browse as well as search any description of any related topic. NX also provides video tutorials for the description of topic being searched as well as the whole package.

Apart from the **NX 9.0 Help** browser window, NX offers more ways to access help which are discussed next.

On Context Help

The **On Context Help** option also provides description of the invoked environment. For example, if you are in the Simulation environment and choose **Menu > Help > On Context** from the **Top Border Bar**, the **Advanced Simulation Overview** will open in the browser window, as shown in Figure 2-29.

*Figure 2-29 The **Advanced Simulation Overview** in the browser window*

Command Finder

The **Command Finder** option available in NX Nastran is used to find a particular tool or command. Choose **Menu > Help > Command Finder** from the **Top Border Bar**; the **Command Finder** dialog box will be displayed, refer to Figure 2-30. You can also invoke this dialog box from the **Ribbon**.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Figure 2-30 *The* ***Command Finder*** *dialog box*

To search any tool from the **Command Finder**, enter the name of tool in the **Search** box and choose the **Find Command** button adjacent to the **Search** box. The available tools will be displayed in the dialog box. When you hover the mouse on any option, the software will locate the particular option in the window, as shown in Figure 2-31.

Figure 2-31 *The* ***Command Finder*** *dialog box with the located tool*

Note

In the ***Command Finder*** *dialog box, as you hover the mouse on the tool which is not related to the existing environment, the dialog box will not display the location of the tool. But it will only display the description of the tool.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Self-Evaluation Test

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

1. The _____ environment is used to define boundary conditions of the model.

2. The _____ environment is used to define material for a model.

3. The tools in the _____ tab are used for manipulating the views of the model.

4. In NX Nastran, the FEM environment is used to create mesh in a model. (T/F)

5. The **Command Finder** locates only the tools available in the current working environment. (T/F)

6. The **On Context** is used to provide the description of the current working environment in the browser window. (T/F)

Review Questions

Answer the following questions:

1. Which of the following analysis steps cannot be performed in the FEM environment?

   (a) Meshing (b) Material applying
   (c) Defining loads (d) None of these

2. Which of the following processes cannot be performed using the **Post Processing Navigator**?

   (a) Plotting results (b) Applying boundary conditions
   (c) Creating graphs (d) None of these

3. The tools available in _____ are used to animate the results generated through analysis.

4. Using the **Simulation File View**, you can switch between working environments. (T/F)

5. The **Post Processing Navigator** is used to view and manipulate results. (T/F)

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Answers to Self-Evaluation Test

1. Simulation, **2.** FEM, **3. View**, **4.** T, **5.** F, **6.** T

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com