

Table of Contents

Dedication	iii
Preface	xv

Chapter 1: Introduction

Introduction to NX 4	1-2
System Requirements	1-3
Getting Started With NX	1-4
Important Terms and Definitions	1-6
Understanding the Functions of the Mouse Buttons	1-12
Toolbars	1-12
Hot Keys	1-18
Color Scheme	1-19
Self-Evaluation Test	1-20

Chapter 2: Drawing Sketches for Solid Models

The Sketcher Task Environment	2-2
Starting NX 4	2-3
Starting a New Document in NX 4	2-4
Starting Different NX Environments	2-7
Invoking the Sketcher Task Environment	2-8
The Drawing Display Tools	2-8
Fitting all Entities in the Current Display	2-9
Zooming to an Area	2-9
Dynamic Zooming	2-10
Panning Drawings	2-10
Fitting View to Selection	2-10
Restoring the Original Orientation of the Sketching Plane	2-10
Setting Selection Filters in the Sketching Environment	2-10
Selecting Sketch Objects	2-11
Selecting General Objects	2-11
Selecting Constraints	2-11
Selecting Objects	2-11
Deselecting Objects	2-11
Sketching Tools	2-11
Drawing Sketches Using the Profile Tool	2-12
Drawing Individual Lines	2-15
Drawing Arcs	2-16
Drawing Circles	2-17
Drawing Rectangles	2-18
Placing Points	2-20

Drawing Ellipses or Elliptical Arcs	2-23
Drawing General Conics	2-24
Drawing Studio Splines	2-24
Filleting Sketched Entities	2-27
Using Snap Points While Sketching	2-28
Deleting the Sketched Entities	2-29
Exiting the Sketcher Task Environment	2-29
Tutorial 1	2-29
Tutorial 2	2-36
Tutorial 3	2-40
Self-Evaluation Test	2-44
Review Questions	2-44
Exercise 1	2-45
Exercise 2	2-46

Chapter 3: Adding Constraints and Dimensions to Sketches

Geometric Constraints	3-2
Applying Additional Constraints Individually	3-2
Applying Automatic Constraints to the Sketch	3-6
Controlling Infer Constraints Settings	3-7
Showing All Constraints in the Sketch	3-8
Turning Off the Display of All Constraints in the Sketch	3-9
Showing/Removing Constraints	3-9
Converting a Sketch Entity to a Reference Entity	3-11
Animating a Fully Constrained Sketch	3-12
Dimensioning Sketched Objects	3-12
Applying Horizontal Dimensions	3-13
Editing the Dimension Value and Other Parameters	3-13
Applying Vertical Dimensions	3-14
Applying Parallel Dimensions	3-15
Applying Perpendicular Dimensions	3-16
Applying Angular Dimensions	3-16
Applying Diameter Dimensions	3-17
Applying Radius Dimensions	3-17
Applying Perimeter Dimensions	3-18
Applying Dimensions Using the Inferred Dimensions Tool	3-18
Measuring the Distance Value between the Objects in a Sketch	3-19
Measuring the Distance Between Two Objects in a Sketch	3-19
Measuring the Projected Distance Between Two Objects	3-19
Measuring the Screen Distance Between Two Objects	3-20
Measuring the Length of an Arc or a Line	3-21
Measuring the Angle Between the Entities	3-22
Measuring the Angle Value Using the By 3 Points Tool	3-22
Measuring the Angle Value Using the By Objects Tool	3-23
Measuring the Angle Value Using the By Screen Points Tool	3-23
Tutorial 1	3-24

Tutorial 2	3-29
Tutorial 3	3-33
Self-Evaluation Test	3-37
Review Questions	3-38
Exercise 1	3-39
Exercise 2	3-39

Chapter 4: Editing, Extruding, and Revolving Sketches

Editing Sketches	4-2
Trimming Sketched Entities	4-2
Extending Sketched Entities	4-3
Mirroring Sketched Entities Using the Mirror Tool	4-4
Transforming Sketched Entities	4-5
Editing Sketched Entities by Dragging	4-10
Exiting the Sketching Environment	4-10
Changing the View of the Sketch	4-11
Creating Base Features by Extruding	4-11
Creating Thin Base Features by Extruding	4-16
More Options in the Extrude Dialog Box	4-19
Creating Solid Revolved Bodies	4-19
Creating Thin Revolved Bodies	4-22
Hiding Entities	4-23
Rotating the View of a Model in 3D Space	4-24
Setting Display Modes	4-25
Tutorial 1	4-25
Tutorial 2	4-31
Tutorial 3	4-34
Self-Evaluation Test	4-37
Review Questions	4-38
Exercise 1	4-39
Exercise 2	4-39

Chapter 5: Working with Fixed and Relative Datum Planes, Coordinate Systems, and Datum Axes

Additional Sketching and Reference Planes	5-2
Types of Datum Planes	5-3
Creating the Fixed (Principle) Datum Planes	5-4
Creating Relative Datum Planes	5-4
Creating Associative Datum Coordinate Systems	5-10
Creating Fixed and Relative Datum Axes	5-14
Other Extrusion Options	5-18
Specifying the Boolean Operation	5-18
Specifying Other Extrusion Termination Options	5-19
Projecting External Elements	5-22

Tutorial 1	5-24
Tutorial 2	5-29
Tutorial 3	5-34
Self-Evaluation Test	5-38
Review Questions	5-39
Exercise 1	5-40
Exercise 2	5-41

Chapter 6: Advanced Modeling Tools-I

Advanced Modeling Tools	6-2
Creating Holes	6-2
Creating Simple Holes	6-2
Creating Counterbore Holes	6-4
Creating Countersink Holes	6-6
Creating Grooves	6-7
Creating Rectangular Grooves	6-8
Creating Ball End Grooves	6-10
Creating U Grooves	6-11
Creating Slots	6-13
Creating Rectangular Slots	6-13
Creating Ball-End Slots	6-15
Creating U Slots	6-18
Creating T Slots	6-20
Creating Dove-Tail Slots	6-22
Creating Chamfers	6-23
Creating an Edge Chamfer Feature Using the Symmetric Offsets Option	6-24
Creating an Edge Chamfer Feature Using the Asymmetric Offsets Option	6-25
Creating an Edge Chamfer Feature Using the Offset Angle Option	6-26
Creating an Edge Blend	6-26
Creating a Constant Radius Edge Blend	6-27
Creating a Variable Radius Edge Blend	6-27
Creating an Edge Blend Using the Stop Short Tool	6-29
Creating a Corner Blend Using the Setback Tool	6-30
Tutorial 1	6-31
Tutorial 2	6-37
Self-Evaluation Test	6-45
Review Questions	6-45
Exercise 1	6-46
Exercise 2	6-47
Exercise 3	6-48

Chapter 7: Advanced Modeling Tools-II

Advanced Modeling Tools	7-2
Working with the Instance Feature Tool	7-2
Creating Rectangular Arrays Using the Instance Feature Tool	7-2

Creating Circular Arrays Using the Instance Feature Tool	7-4
Mirroring Selected Features Using the Instance Feature Tool	7-7
Mirroring Bodies Using the Instance Feature Tool	7-8
Using the Pattern Face Option of the Instance Feature Tool	7-9
Sweeping Sketches Along Guide Curves	7-10
Creating Swept Features	7-11
Other Swept Options	7-13
Creating Tubes or Cables	7-14
Creating Threads	7-15
Creating Symbolic Threads	7-16
Creating Detailed Threads	7-19
Creating Shell Features	7-20
Shell Dialog Box Options	7-20
Shelling the Entire Body	7-22
Tutorial 1	7-22
Tutorial 2	7-26
Tutorial 3	7-31
Self-Evaluation Test	7-34
Review Questions	7-35
Exercise 1	7-36
Exercise 2	7-36

Chapter 8: Editing Features and Advanced Modeling Tools-III

Editing Features	8-2
Editing a Hole Feature	8-2
Editing the Positioning a Hole Feature	8-3
Editing the Positioning of a Groove Feature	8-4
Editing the Positioning of a Slot Feature	8-4
Editing the Parameters of Features	8-5
Editing the Parameters of Features with the Rollback	8-5
Editing Sketches of the Sketch-based Features	8-5
Reordering Features	8-5
Advanced Modeling Tools	8-6
Creating Boss Features	8-6
Creating Pocket Features	8-7
Creating Pad Features	8-12
Creating Drafts	8-15
Tutorial 1	8-19
Tutorial 2	8-24
Tutorial 3	8-32
Self-Evaluation Test	8-39
Review Questions	8-40
Exercise 1	8-41
Exercise 2	8-42

Chapter 9: Assembly Modeling-I

The Assembly Application	9-2
Starting a New File in the Assembly Application of NX	9-2
Types of Assembly Design Approaches	9-3
Creating Bottom-up Assemblies	9-3
Placing Components in the Assembly	9-4
Changing the Reference Set of the Component	9-6
Applying Assembly Constraints to Components	9-6
Using the Options in the Mating Conditions Dialog Box	9-13
Points to Remember while Assembling Components	9-14
Creating a Component Array in an Assembly	9-14
Replacing a Component in an Assembly	9-16
Repositioning a Component in an Assembly	9-17
Mirroring a Component in an Assembly	9-20
Modifying a Component in the Assembly File	9-23
Tutorial 1	9-24
Tutorial 2	9-38
Self-Evaluation Test	9-52
Review Questions	9-53
Exercise 1	9-53

Chapter 10: Assembly Modeling-II

The Top-down Assembly Design Approach	10-2
Creating Components Using the Top-down Assembly Design Approach	10-2
Creating Subassemblies	10-4
Editing Assembly Constraints	10-4
Modifying Angle and Distance Values	10-5
Replacing Assembly Constraints	10-5
Checking for Interference between the Components in an Assembly	10-7
Interference and Clearance Check Using the Check Clearance Analysis	10-7
Interference Check Using the Assembly Clearance Method	10-9
Creating Exploded Views of an Assembly	10-14
Automatic Exploded Views	10-15
Manual Exploded Views	10-17
Tutorial 1	10-19
Tutorial 2	10-23
Tutorial 3	10-28
Tutorial 4	10-34
Self-Evaluation Test	10-43
Review Questions	10-44
Exercise 1	10-44

Chapter 11: Generating, Editing, and Dimensioning the Drawing Views

The Drafting Application	11-2
Editing Drawing Sheet Parameters in the Drawing Environment	11-4
Invoking the Drafting Tools	11-4
Types of Drawing Views in NX	11-4
Base View	11-4
Part View	11-4
Projected View	11-4
Detail View	11-5
Section View	11-5
Auxiliary View	11-5
Half Section View	11-5
Revolved Section View	11-5
Break-Out Section View	11-5
Broken View	11-5
Generating Drawing Views	11-5
Generating the Base View	11-6
Generating the Drawing View using the Part View Tool	11-7
Generating the Orthographic Drawing Views using the Projected View Tool	11-7
Generating the Detail View using the Detail View Tool	11-8
Generating Section Views using the Section View Tool	11-10
Generating the Half Section View using the Half Section View Tool	11-13
Adding the Revolved Section View	11-14
Generating the Break-Out Section for a Drawing View	11-14
Generating the Broken View	11-15
Manipulating the Drawing Views	11-18
Aligning the Drawing Views using the Align View Tool	11-18
View Boundary	11-20
Displaying the Model using the Display Sheet Tool	11-21
Inserting a Drawing Sheet using the New Sheet Tool	11-21
Modifying the Properties of the Generated Drawing View	11-21
Modifying the Scale Value of the Drawing View	11-21
Displaying the Hidden Lines in the Drawing View	11-22
Adding Dimensions to the Drawing Views	11-23
Retrieving Dimensions from the Model	11-23
Adding Dimensions to the Drawing View	11-23
Generating Exploded Views of the Assembly	11-26
Creating the Parts List and Associative Balloons	11-27
Creating the Parts List for an Assembly	11-27
Creating Associative Balloons	11-27
Creating a Tabular Note (Title Block)	11-28
Adding Text to the Drawing Sheet	11-29
Editing the Multiline Text	11-30
Tutorial 1	11-31

Tutorial 2	11-36
Tutorial 3	11-41
Self-Evaluation Test	11-48
Review Questions	11-49
Exercise 1	11-50
Exercise 2	11-50

Chapter 12: Surface Modeling

Introduction to Surface Modeling	12-2
Invoking the Sheet Modeling Environment	12-2
Creating an Extruded Surface	12-2
Creating a Revolved Surface	12-3
Creating a Ruled Surface	12-3
Creating a Surface Using the Through Curve Tool	12-5
Creating a Surface Using the Through Curve Mesh Tool	12-7
Creating a Surface by 4 Points	12-8
Creating a Swoop Surface	12-8
Creating the Planar Surfaces from 2D Sketches and Solid Edges	12-10
Creating a Transition Surface Using the Transition Tool	12-11
Creating an N-Sided Surface	12-13
Creating a Silhouette Flange Surface	12-15
Extending a Surface Using the Law Extension Tool	12-17
Creating a Surface Offset Using the Offset Surface Tool	12-19
Trimming and Extending a Surface using the Trim and Extend Tool	12-19
Trimming a Sheet by using the Trimmed Sheet Tool	12-20
Creating a Surface Using the Studio Surface Tools	12-22
Creating a Surface Between Two Walls Using the Styled Blend Tool	12-28
Creating Surfaces Using the Styled Sweep Tool	12-30
Sewing Individual Surfaces Into a Single Surface	12-32
Adding Thickness to a Surface	12-33
Tutorial 1	12-34
Tutorial 2	12-40
Self-Evaluation Test	12-46
Review Questions	12-47
Exercise 1	12-48
Exercise 2	12-49

Chapter 13: Advanced Surface Modeling

Creating Curves from Bodies	13-2
Creating Intersection Curves	13-2
Creating Section Curves	13-3
Creating Extract Curves	13-8
Advanced Surface Modeling Tools	13-11
Creating Dart Features	13-11
Creating Emboss Sheet Features	13-13

Creating Face Blend Features	13-14
Creating Soft Blend Features	13-18
Creating Fillet Features	13-20
Creating Bridge Features	13-24
Tutorial 1	13-29
Tutorial 2	13-39
Self-Evaluation Test	13-45
Review Questions	13-46
Exercise 1	13-46
Exercise 2	13-47
Student Projects	1
Index	1