

Table of Contents

Dedication	iii
Preface	xvii

Chapter 1: Introduction to NX 10.0

Introduction to NX 10.0	1-2
System Requirements	1-4
Getting Started with NX <i>Enhanced</i>	1-4
Important Terms and Definitions	1-6
Understanding the Functions of the Mouse Buttons	1-10
Quick Access Toolbar <i>Enhanced</i>	1-11
Ribbon	1-12
Status Bar	1-16
Hot Keys	1-16
Color Scheme	1-17
Dialog Boxes in NX	1-17
Selecting Objects	1-18
Deselecting Objects	1-18
Selecting Objects Using the QuickPick Dialog Box	1-19
Self-Evaluation Test	1-19

Chapter 2: Drawing Sketches for Solid Models

Introduction	2-2
Starting NX <i>Enhanced</i>	2-3
Starting a New Document in NX	2-4
Invoking Different NX Environments	2-6
Creating Three Fixed Datum Planes (XC-YC, YC-ZC, XC-ZC)	2-7
Displaying the WCS (Work Coordinate System)	2-8
Creating Sketches	2-8
Creating Sketches in the Modeling Environment	2-8
Creating Sketches in the Sketching Environment	2-14
Sketching Tools	2-14
Drawing Sketches Using the Profile Tool	2-14
Using Help Lines to Locate Points	2-17
Drawing Individual Lines	2-18
Drawing Arcs	2-18
Drawing Circles	2-20
Drawing Rectangles	2-21
Placing Points	2-23
Drawing Ellipses or Elliptical Arcs	2-26

Drawing Conics	2-28
Drawing Studio Splines	2-29
Filleting Sketched Entities	2-31
The Drawing Display Tools	2-33
Fitting Entities in the Current Display	2-33
Zooming an Area	2-33
Dynamic Zooming	2-34
Panning Drawings	2-34
Fitting View to Selection	2-34
Restoring the Original Orientation of the Sketching Plane	2-34
Setting Selection Filters in the Sketch in Task Environment	2-34
Selecting Objects	2-36
Deselecting Objects	2-36
Using Snap Points Options While Sketching	2-36
Deleting Sketched Entities	2-37
Exiting the Sketch Environment	2-37
Tutorial 1	2-38
Tutorial 2	2-42
Tutorial 3	2-47
Self-Evaluation Test	2-50
Review Questions	2-51
Exercise 1	2-52
Exercise 2	2-53

Chapter 3: Adding Geometric and Dimensional Constraints to Sketches

Constraining Sketches	3-2
Concept of Constrained Sketches	3-2
Under-Constrain	3-2
Fully-Constrain	3-2
Over-Constrain	3-2
Degree of Freedom Arrows	3-3
Dimensioning Sketches	3-4
Locking the Automatically Applied Dimensions	3-4
Applying Dimensions by Using the Rapid Dimension Tool	3-5
Applying Linear Dimensions	3-5
Applying Radial Dimensions	3-8
Applying Angular Dimensions	3-8
Applying Perimeter Dimensions	3-9
Editing the Dimension Value and Other Parameters	3-9
Animating a Fully-Constrained Sketch	3-9
Measuring the Distance Value between Objects in a Sketch	3-10
Measuring the Distance between Two Objects in a Sketch	3-11
Measuring the Projected Distance between Two Objects	3-11
Measuring the Screen Distance between Two Objects	3-12
Measuring the Length of an Arc or a Line	3-12
Measuring the Angle between Entities	3-13

Measuring the Angle Value Using the By Objects Option	3-13
Measuring the Angle Value Using the By 3 Points Option	3-14
Measuring the Angle Value Using the By Screen Points Option	3-14
Geometric Constraints	3-15
Applying Additional Constraints Individually	3-15
Applying Symmetry Constraint	3-20
Applying Automatic Constraints to a Sketch	3-20
Controlling Inferred Constraints Settings	3-22
Showing All Constraints in a Sketch	3-22
Showing/Removing Constraints	3-22
Converting a Sketch Entity or Dimension into a Reference Entity or Reference Dimension	3-24
Tutorial 1	3-25
Tutorial 2	3-30
Tutorial 3	3-33
Self-Evaluation Test	3-35
Review Questions	3-36
Exercise 1	3-37
Exercise 2	3-38

Chapter 4: Editing, Extruding, and Revolving Sketches

Editing Sketches	4-2
Trimming Sketched Entities	4-2
Extending Sketched Entities	4-3
Creating a Corner between Sketched Entities	4-4
Moving Sketched Entities by Using the Move Curve Tool	4-5
Offsetting Sketched Entities by Using Offset Move Curve	4-10
Modifying Entities by Using the Resize Curve Tool	4-11
Modifying Chamfer in Sketched Entities by Using Resize Chamfer Curve Tool	4-13
Deleting Sketched Entities by Using Delete Curve Tool	4-13
Offsetting Sketched Entities	4-14
Mirroring Sketched Entities	4-16
Creating a Linear Sketch Pattern	4-17
Creating a Circular Sketch Pattern	4-18
Creating a General Sketch Pattern	4-19
Transforming Sketched Entities	4-19
Editing Sketched Entities by Dragging	4-22
Exiting the Sketch Environment	4-23
Changing the View of the Sketch	4-23
Creating Base Features by Extruding	4-24
Extrude Dialog Box Options	4-24
Creating Solid Revolved Bodies	4-31
Copying, Moving, and Rotating Objects	4-35
Hiding Entities	4-40
Showing Hidden Entities	4-40
Hiding All Entities Using a Single Tool	4-40
Rotating the View of a Model in 3D Space	4-41

Setting Display Modes	4-42
Tutorial 1	4-42
Tutorial 2	4-44
Tutorial 3	4-48
Self-Evaluation Test	4-51
Review Questions	4-51
Exercise 1	4-52
Exercise 2	4-53

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

Additional Sketching and Reference Planes	5-2
Types of Datum Planes	5-3
Creating Three Fixed (Principle) Datum Planes	5-3
Creating Relative Datum Planes	5-3
Creating Datum Coordinate Systems	5-9
Creating Fixed and Relative Datum Axes	5-13
Other Extrusion Options	5-16
Specifying the Boolean Operation	5-16
Specifying Other Extrusion Termination Options	5-18
Projecting External Elements	5-20
Tutorial 1	5-22
Tutorial 2	5-26
Tutorial 3	5-30
Self-Evaluation Test	5-34
Review Questions	5-35
Exercise 1	5-36
Exercise 2	5-37
Exercise 3	5-38

Chapter 6: Advanced Modeling Tools-I

Advanced Modeling Tools	6-2
Creating Holes by Using the Hole Tool	6-2
Creating General Holes	6-2
Creating Drill Size Hole	6-4
Creating Screw Clearance Hole	6-5
Creating Threaded Hole	6-5
Creating Hole Series	6-6
Creating Grooves	6-6
Creating Rectangular Grooves	6-7
Creating Ball End Grooves	6-8
Creating U Grooves	6-9
Creating Slots	6-11
Creating Rectangular Slots	6-11
Creating Ball-End Slots	6-13
Creating U-Slots	6-15
Creating T-Slots	6-16

Creating Dove-Tail Slots	6-18
Creating Ribs	6-19
Creating Chamfers	6-22
Creating a Chamfer Feature Using the Symmetric Method	6-22
Creating a Chamfer Feature Using the Asymmetric Method	6-23
Creating a Chamfer Feature Using the Offset and Angle Method	6-24
Creating an Edge Blend	6-25
Tutorial 1	6-31
Tutorial 2	6-37
Self-Evaluation Test	6-44
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
Pattern Feature Tool	7-2
Creating a Linear Pattern	7-2
Creating a Circular Pattern	7-7
Creating a Polygon Pattern	7-10
Creating a Spiral Pattern	7-11
Creating a Pattern Along a Curve	7-12
Creating a General Pattern	7-14
Creating a Reference Pattern	7-14
Creating a Helix Pattern	7-15
Creating a Fill Pattern	7-17
Mirror Feature Tool	7-19
Mirror Face Tool	7-20
Mirror Geometry Tool	7-21
Sweeping Sketches Along the Guide Curves	7-21
Creating Swept Features	7-23
Creating Tubes or Cables	7-25
Creating Threads	7-27
Creating Symbolic Threads	7-27
Creating Detailed Threads	7-29
Creating Shell Features	7-30
Shelling the Entire Solid Body	7-31
Tutorial 1	7-32
Tutorial 2	7-35
Tutorial 3	7-38
Tutorial 4	7-42
Self-Evaluation Test	7-48
Review Questions	7-49
Exercise 1	7-50
Exercise 2	7-50

Chapter 8: Editing Features and Advanced Modeling Tools-III

Editing Features	8-2
Editing a Hole Feature	8-2
Editing the Positioning of a Groove Feature	8-2
Editing the Positioning of a Slot Feature	8-3
Editing the Parameters of Features	8-3
Editing the Parameters of Features with Rollback	8-3
Editing Sketches of the Sketch-based Features	8-3
Reordering Features	8-4
Advanced Modeling Tools	8-4
Creating Boss Features	8-4
Creating Pocket Features	8-5
Creating Pad Features	8-8
Creating Drafts	8-11
Tutorial 1	8-14
Tutorial 2	8-18
Tutorial 3	8-26
Self-Evaluation Test	8-33
Review Questions	8-34
Exercise 1	8-35
Exercise 2	8-36
Exercise 3	8-36

Chapter 9: Assembly Modeling-I

The Assembly Environment	9-2
Invoking the Assembly Environment	9-2
Invoking the Assembly Environment Using the New Dialog Box	9-2
Invoking the Assembly Environment in the Current Part File	9-2
Types of Assembly Design Approaches	9-3
Creating Bottom-up Assemblies	9-4
Placing Components in the Assembly Environment	9-4
Changing the Reference Set of a Component	9-6
Applying Assembly Constraints to Components	9-7
Points to Remember while Assembling Components	9-16
Creating a Pattern Component in an Assembly	9-17
Replacing a Component in an Assembly	9-19
Moving a Component in an Assembly	9-20
Mirroring a Component in an Assembly	9-24
Modifying a Component in the Assembly File	9-26
Tutorial 1	9-27
Tutorial 2	9-40
Self-Evaluation Test	9-52
Review Questions	9-53
Exercise 1	9-54

Exercise 2	9-56
Exercise 3	9-59
Exercise 4	9-62

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	10-5
Checking the Interference between the Components of an Assembly	10-6
Checking Interference Using the Simple Interference Tool	10-6
Checking Interference Between the Assembly Components	10-7
Checking Interference and Clearance, and Analyzing Cross-sections of Components Using the View Section Tool	10-11
Creating Exploded Views of an Assembly	10-16
Exploding Views Automatically	10-17
Exploding Views Manually	10-19
Tutorial 1	10-20
Tutorial 2	10-24
Tutorial 3	10-29
Tutorial 4	10-35
Self-Evaluation Test	10-40
Review Questions	10-40
Exercise 1	10-41
Exercise 2	10-46
Exercise 3	10-53

Chapter 11: Surface Modeling

Introduction to Surface Modeling	11-2
Invoking the Shape Studio Environment	11-2
Creating an Extruded Surface	11-2
Creating a Revolved Surface	11-3
Creating a Ruled Surface	11-4
Creating a Surface Using the Through Curves Tool	11-6
Creating a Surface Using the Through Curve Mesh Tool	11-7
Creating a Surface Using the Four Point Surface Tool	11-9
Creating a Swoop Surface	11-10
Creating Planar Surfaces from 2D Sketches and Edges of Solid or Surface	11-11
Creating a Transition Surface Using the Transition Tool	11-12
Creating an N-Sided Surface	11-14
Creating a Silhouette Flange Surface	11-17
Extending a Surface Using the Law Extension Tool	11-21
Creating a Surface Offset Using the Offset Surface Tool	11-24
Trimming and Extending a Surface Using the Trim and Extend Tool <i>Enhanced</i>	11-25
Trimming a Sheet by Using the Trimmed Sheet Tool	11-26
Creating a Surface Using the Studio Surface Tool	11-28
Creating a Surface between Two Walls Using the Styled Blend Tool	11-31

Creating Surfaces Using the Styled Sweep Tool	11-35
Sewing Individual Surfaces into a Single Surface	11-37
Adding Thickness to a Surface	11-37
Adding a Draft	11-38
Tutorial 1	11-40
Tutorial 2	11-45
Self-Evaluation Test	11-53
Review Questions	11-54
Exercise 1	11-55
Exercise 2	11-56

Chapter 12: Advanced Surface Modeling

Creating Curves from Bodies	12-2
Creating Intersection Curves	12-2
Creating Section Curves	12-3
Creating Extract Curves	12-8
Creating Isoparametric Curves	12-9
Projecting Curves	12-11
Advanced Surface Modeling Tools	12-14
Creating Dart Features	12-14
Creating Emboss Body on a Sheet or Solid Body	12-16
Creating Face Blend Features <i>Enhanced</i>	12-17
Creating Bridge Features	12-23
Tutorial 1	12-25
Tutorial 2	12-35
Tutorial 3	12-44
Self-Evaluation Test	12-55
Review Questions	12-56
Exercise 1	12-56
Exercise 2	12-57

Chapter 13: Generating, Editing, and Dimensioning the Drawing Views

The Drafting Environment	13-2
Invoking the Drafting Environment	13-2
Invoking the Drafting Environment Using the Drawing Template from the New Dialog Box	13-2
Invoking the Drafting Environment in the Current Part File	13-3
Editing the Drawing Sheet Parameters in the Drafting Environment	13-6
Invoking the Drafting Tools	13-6
Types of Drawing Views in NX	13-6
Base View	13-6
Projected View	13-6
Detail View	13-6
Section View	13-6
Auxiliary View	13-7
Half-Section View	13-7
Revolved Section View	13-7

Break-Out Section View	13-7
Broken View	13-7
Generating Drawing Views	13-7
Generating Views Using the View Creation Wizard Tool	13-7
Generating the Base View	13-12
Generating the Orthographic Drawing Views Using the Projected View Tool	13-14
Generating the Detail View Using the Detail View Tool	13-15
Creating a Section Line <small>New</small>	13-18
Generating Section Views Using the Section View Tool <small>Enhanced</small>	13-20
Generating the Break-Out Section View	13-25
Generating the Broken View	13-26
Manipulating the Drawing Views	13-28
Moving the Drawing Views Using the Move/Copy View Tool	13-28
Aligning the Drawing Views Using the Align View Tool	13-29
View Boundary	13-31
Section in View Tool	13-32
Displaying the Model Using the Display Sheet Tool	13-33
Inserting a Drawing Sheet Using the New Sheet Tool	13-34
Modifying the Properties of the Generated Drawing View	13-34
Modifying the Scale Value of the Drawing View	13-34
Adding Dimensions to the Drawing Views	13-36
Retrieving Dimensions from the Model	13-36
Adding Dimensions to the Drawing View	13-37
Generating Exploded Views of an Assembly	13-41
Creating Parts List and Associative Balloons	13-41
Creating a Parts List for an Assembly	13-42
Creating Associative Balloons	13-42
Creating a Tabular Note (Title Block)	13-43
Adding Multiline Text to a Drawing Sheet	13-45
Printing Tools	13-46
Print	13-46
Plot	13-46
Tutorial 1	13-47
Tutorial 2	13-52
Tutorial 3	13-58
Self-Evaluation Test	13-65
Review Questions	13-66
Exercise 1	13-67
Exercise 2	13-67

Chapter 14: Synchronous Modeling

Introduction	14-2
Move Face	14-2
Move Edge	14-5
Pull Face	14-7
Offset Region	14-8

Offset Edge	14-9
Replace Face	14-10
Resize Blend	14-11
Reorder Blends	14-12
Resize Chamfer	14-13
Label Chamfer	14-14
Label Notch Blend	14-14
Resize Face	14-14
Delete Face	14-15
Copy Face	14-17
Cut Face	14-18
Paste Face	14-19
Mirror Face	14-20
Pattern Face	14-20
Make Coplanar	14-21
Make Coaxial	14-22
Make Tangent	14-23
Make Symmetric	14-24
Make Parallel	14-25
Make Perpendicular	14-26
Make Fixed	14-27
Make Offset	14-28
Show Related Face	14-29
Linear Dimension	14-30
Angular Dimension	14-31
Radial Dimension	14-33
Shell Body	14-34
Shell Face	14-35
Change Shell Thickness	14-36
Group Face	14-37
Edit Cross Section in History Free Mode	14-38
Edit Cross Section in History Mode	14-39
Tutorial 1	14-40
Tutorial 2	14-46
Self-Evaluation Test	14-55
Review Questions	14-56
Exercise 1	14-57
Exercise 2	14-57

Chapter 15: Sheet Metal Design

The Sheet Metal Module	15-2
Setting the Sheet Metal Part Properties	15-3
Creating the Base Feature	15-7
Adding Flanges to a Sheet Metal Part	15-8
Creating Contour Flanges	15-15
Creating Lofted Flanges	15-16
Adding a Jog to the Sheet	15-17

Bending the Sheet Metal Part	15-18
Unbending the Sheet Metal Part	15-19
Rebending the Sheet Metal Part	15-19
Filleting or Chamfering Corners	15-19
Closing the Corners of a Sheet Metal Part	15-20
Creating Dimples in a Sheet Metal Part	15-23
Creating Louvers in a Sheet Metal Part	15-26
Creating Drawn Cutouts in a Sheet Metal Part	15-28
Creating Beads in a Sheet Metal Part	15-28
Adding Gussets to a Sheet Metal part	15-31
Adding Hems	15-34
Creating a Sheet Metal Part Using Solid Body	15-37
Converting a Solid Part into a Sheet Metal Part	15-38
Ripping the Corners of a Solid Part	15-39
Creating the Flat Pattern of a Sheet Metal Part	15-39
Creating the Flat Pattern	15-40
Creating the Flat Solid	15-40
Exporting a Flat Pattern	15-41
Tutorial 1	15-41
Tutorial 2	15-52
Self-Evaluation Test	15-57
Review Questions	15-58
Exercise 1	15-58
Exercise 2	15-60
Student Projects	SP-1
Index	I-1