Table of Contents

Dedication iii
Preface xvii

Chapter 1: Introduction to NX

Introduction to NX 1-2
System Requirements 1-4
Getting Started with NX 1-4
Important Terms and Definitions 1-6
Understanding the Functions of the Mouse Buttons 1-10
Quick Access Toolbar 1-11
Ribbon 1-14
Status Bar 1-18
Hot Keys 1-19
Color Scheme 1-19
Dialog Boxes in NX 1-20
Selecting Objects 1-20
Deselecting Objects 1-21
Selecting Objects Using the QuickPick Dialog Box 1-21
Self-Evaluation Test 1-21

Chapter 2: Drawing Sketches for Solid Models

Introduction 2-2
Starting NX 2-2
Starting a New Document in NX 2-3
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 2-13
Sketching Tools 2-13
    Drawing Sketches Using the Profile Tool 2-14
    Using Help Lines to Locate Points 2-17
    Drawing Individual Lines 2-17
    Drawing Arcs 2-17
    Drawing Circles 2-19
    Drawing Rectangles 2-20
    Placing Points 2-21
    Drawing Ellipses or Elliptical Arcs 2-26

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

Drawing Conics 2-27
Drawing Studio Splines 2-28
Filleting Sketched Entities 2-30
The Drawing Display Tools 2-31
Fitting Entities in the Current Display 2-32
Zooming an Area 2-32
Dynamic Zooming 2-32
Panning Drawings 2-33
Fitting View to Selection 2-33
Restoring the Original Orientation of the Sketching Plane 2-33
Setting Selection Filters in the Sketch in Task Environment 2-33
Selecting Objects 2-35
Deselecting Objects 2-35
Using Snap Points Options While Sketching 2-36
Deleting Sketched Entities 2-36
Exiting the Sketch Environment 2-36
Tutorial 1 2-37
Tutorial 2 2-41
Tutorial 3 2-46
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
Degrees 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 3-9
Applying Perimeter Dimensions 3-9
Editing the Dimension Value and Other Parameters 3-10
Animating a Fully-Constrained Sketch 3-10
Geometric Constraints 3-11
Applying Additional Constraints Individually 3-11
Applying Symmetry Constraint 3-18
Applying Automatic Constraints to a Sketch 3-18
Controlling Inferred Constraints Settings 3-20
Showing All Constraints in a Sketch 3-20

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

Sketch Relations Browser 3-21
Converting a Sketch Entity or Dimension into a Reference Entity or Reference Dimension 3-22
Creating Alignment Constraints 3-23
Tutorial 1 3-25
Tutorial 2 3-30
Tutorial 3 3-33
Self-Evaluation Test 3-37
Review Questions 3-37
Exercise 1 3-38
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
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
Trim Recipe Curve 4-20
Scale Curve 4-21
Transforming Sketched Entities 4-23
Editing Sketched Entities by Dragging 4-26
Exiting the Sketch Environment 4-27
Changing the View of the Sketch 4-27
Creating Base Features by Extruding 4-27
Extrude Dialog Box Options 4-28
Creating Solid Revolved Bodies 4-34
Copying, Moving, and Rotating Objects 4-39
Hiding Entities 4-45
Showing Hidden Entities 4-46
Hiding All Entities Using a Single Tool 4-46
Rotating the View of a Model in 3D Space 4-46
Setting Display Modes 4-47
Tutorial 1 4-47
Tutorial 2 4-50
Tutorial 3 4-54
Self-Evaluation Test 4-57
Review Questions 4-58

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

Exercise 1 4-59
Exercise 2 4-59

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 5-10
Creating Fixed and Relative Datum Axes 5-14
Other Extrusion Options 5-17
Specifying the Boolean Operation 5-17
Other Extrusion Termination Options 5-19
Projecting External Elements 5-21
Tutorial 1 5-23
Tutorial 2 5-28
Tutorial 3 5-32
Self-Evaluation Test 5-36
Review Questions 5-37
Exercise 1 5-38
Exercise 2 5-39
Exercise 3 5-41

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 6-7
Creating Rectangular Grooves 6-7
Creating Ball End Grooves 6-9
Creating U Grooves 6-10
Creating Slots 6-11
Creating Rectangular Slots 6-12
Creating Ball-End Slots 6-14
Creating U-Slots 6-16
Creating T-Slots 6-17
Creating Dove-Tail Slots 6-19
Creating Ribs 6-20
Creating Chamfers 6-23
Creating a Chamfer Feature Using the Symmetric Method 6-23
Creating a Chamfer Feature Using the Asymmetric Method 6-24
Creating a Chamfer Feature Using the Offset and Angle Method 6-25

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

Creating an Edge Blend 6-26
Tutorial 1 6-32
Tutorial 2 6-39
Self-Evaluation Test 6-46
Review Questions 6-47
Exercise 1 6-48
Exercise 2 6-49
Exercise 3 6-51

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 7-15
- Creating a Helix Pattern 7-15
- Creating Patterns by Defining Boundary 7-16
- 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 7-22
Creating Swept Features 7-23
Creating Tubes or Cables 7-26
Creating Threads 7-27
- Creating Symbolic Threads 7-28
- Creating Detailed Threads 7-30

Creating Shell Features 7-31
Editing Features 7-33
- Editing a Hole Feature 7-33
- Editing the Positioning of a Groove Feature 7-33
- Editing the Parameters of Features 7-34
- Editing the Parameters of Features with Rollback 7-34
- Editing Sketches of the Sketch-based Features 7-34
- Reordering Features 7-35
- Creating Drafts 7-35
- Creating Lattice Enhanced 7-38

Measuring Objects New 7-41
Tutorial 1 7-43
Tutorial 2 7-47
Tutorial 3 7-51
Tutorial 4 7-55
Tutorial 5 7-61

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

Self-Evaluation Test 7-65
Review Questions 7-66
Exercise 1 7-67
Exercise 2 7-68
Exercise 3 7-69

Chapter 8: Assembly Modeling-I

The Assembly Environment 8-2
Invoking the Assembly Environment 8-2
Invoking the Assembly Environment Using the New Dialog Box 8-2
Invoking the Assembly Environment in the Current Part File 8-2
Types of Assembly Design Approaches 8-3
Creating Bottom-up Assemblies 8-3
Placing Components in the Assembly Environment 8-4
Changing the Reference Set of a Component 8-6
Applying Assembly Constraints to Components Enhanced 8-7
Points to Remember while Assembling Components 8-17
Creating a Pattern Component in an Assembly 8-17
Replacing a Component in an Assembly 8-19
Moving a Component in an Assembly 8-21
Mirroring a Component in an Assembly 8-24
Modifying a Component in the Assembly File 8-27
Tutorial 1 8-28
Tutorial 2 8-41
Self-Evaluation Test 8-55
Review Questions 8-56
Exercise 1 8-57
Exercise 2 8-63
Exercise 3 8-67
Exercise 4 8-71

Chapter 9: Assembly Modeling-II

The Top-down Assembly Design Approach 9-2
Creating Components Using the Top-down Assembly Design Approach 9-2
Creating Subassemblies 9-4
Editing Assembly Constraints 9-5
Checking the Interference between the Components of an Assembly 9-5
Checking Interference Using the Simple Interference Tool 9-6
Checking Interference Between the Assembly Components 9-7
Checking Interference and Clearance, and Analyzing Cross-sections of Components Using the View Section Tool 9-12
Creating Exploded Views of an Assembly 9-17
Exploding Views Automatically 9-18
Exploding Views Manually 9-20
Digital Mock Up Application (DMU) 9-21
Tutorial 1 9-23

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

Tutorial 2 9-28
Tutorial 3 9-32
Tutorial 4 9-40
Self-Evaluation Test 9-47
Review Questions 9-48
Exercise 1 9-48
Exercise 2 9-55
Exercise 3 9-61

Chapter 10: Surface Modeling

Introduction to Surface Modeling 10-2
Invoking the Shape Studio Environment 10-2
- Creating an Extruded Surface 10-3
- Creating a Revolved Surface 10-3
- Creating a Combined Surface 10-4
- Creating a Surface Using the Through Curves Tool 10-6
- Creating a Surface Using the Through Curve Mesh Tool 10-8
- Creating a Surface Using the Four Point Surface Tool 10-9
- Creating Planar Surfaces from 2D Sketches and Edges of Solid or Surface 10-10
- Editing Surface Using the X-Form Tool New 10-11
- Creating an N-Sided Surface 10-12
- Creating a Silhouette Flange Surface 10-15
- Extending a Surface Using the Law Extension Tool 10-20
- Creating a Surface Offset Using the Offset Surface Tool 10-24
- Trimming and Extending a Surface Using the Trim and Extend Tool 10-24
- Trimming a Sheet by Using the Trimmed Sheet Tool 10-26
- Creating a Surface Using the Studio Surface Tool 10-28
- Creating a Surface between Two Walls Using the Styled Blend Tool 10-31
- Creating Surfaces Using the Styled Sweep Tool 10-36
- Sewing Individual Surfaces into a Single Surface 10-37
- Adding Thickness to a Surface 10-38
- Adding a Draft 10-39

Tutorial 1 10-40
Tutorial 2 10-46
Self-Evaluation Test 10-54
Review Questions 10-55
Exercise 1 10-55
Exercise 2 10-57

Chapter 11: Advanced Surface Modeling

Creating Curves from Bodies 11-2
- Creating Intersection Curves 11-2
- Creating Section Curves 11-3
- Creating Isoparametric Curves 11-8
- Projecting Curves 11-10

Advanced Surface Modeling Tools 11-13
- Creating Emboss Body on a Sheet or Solid Body 11-13

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

Creating Face Blend Features 11-15
Creating Bridge Features 11-21
Tutorial 1 11-23
Tutorial 2 11-35
Tutorial 3 11-42
Self-Evaluation Test 11-55
Review Questions 11-55
Exercise 1 11-56
Exercise 2 11-57

Chapter 12: Generating, Editing, and Dimensioning the Drawing Views

The Drafting Environment 12-2
Invoking the Drafting Environment 12-2
Invoking the Drafting Environment Using the Drawing Template from the New Dialog Box 12-2
Invoking the Drafting Environment in the Current Part File 12-3
Editing the Drawing Sheet Parameters in the Drafting Environment 12-6
Invoking the Drafting Tools 12-6
Types of Drawing Views in NX 12-6
Base View 12-6
Projected View 12-6
Detail View 12-6
Section View 12-7
Auxiliary View 12-7
Half-Section View 12-7
Revolved Section View 12-7
Break-Out Section View 12-7
Broken View 12-7
Generating Drawing Views 12-7
Generating Views Using the View Creation Wizard Tool 12-8
Generating the Base View 12-12
Generating the Orthographic Drawing Views Using the Projected View Tool 12-14
Generating the Detail View Using the Detail View Tool 12-16
Creating a Section Line 12-18
Generating Section Views Using the Section View Tool 12-20
Generating the Break-Out Section View 12-25
Generating the Broken View 12-27
Manipulating the Drawing Views 12-29
Moving the Drawing Views Using the Move/Copy View Tool 12-29
Aligning the Drawing Views Using the Align View Tool 12-30
View Boundary 12-33
Section in View Tool 12-33
Inserting a Drawing Sheet Using the New Sheet Tool 12-35
Modifying the Properties of the Generated Drawing View 12-35
Modifying the Scale Value of the Drawing View 12-35
Adding Dimensions to the Drawing Views 12-37

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

Generating Exploded Views of an Assembly 12-42
Creating Parts List and Associative Balloons 12-42
Creating a Parts List for an Assembly 12-42
Creating Associative Balloons 12-43
Creating a Tabular Note (Title Block) 12-44
Adding Multiline Text to a Drawing Sheet 12-46
Printing Tools 12-47
Print 12-47
Plot 12-47
Tutorial 1 12-48
Tutorial 2 12-54
Tutorial 3 12-58
Self-Evaluation Test 12-63
Review Questions 12-64
Exercise 1 12-65
Exercise 2 12-65

Chapter 13: Synchronous Modeling

Introduction 13-2
Synchronous Modeling Tools 13-2
Move Face 13-2
Move Edge 13-5
Pull Face 13-7
Offset Region 13-8
Offset Edge 13-10
Replace Face 13-11
Resize Blend 13-12
Reorder Blends 13-13
Resize Chamfer 13-13
Label Chamfer 13-14
Label Notch Blend 13-14
Resize Face 13-15
Delete Face 13-16
Copy Face 13-18
Cut Face 13-20
Paste Face 13-20
Mirror Face 13-21
Pattern Face 13-22
Make Coplanar 13-23
Make Coaxial 13-24
Make Tangent 13-26
Make Symmetric 13-27
Make Parallel 13-28
Make Perpendicular 13-30
Make Offset 13-31
Linear Dimension 13-32
Angular Dimension 13-34

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

Radial Dimension 13-36
Group Face 13-37
Edit Cross Section 13-38
Tutorial 1 13-38
Tutorial 2 13-44
Self-Evaluation Test 13-52
Review Questions 13-53
Exercise 1 13-54
Exercise 2 13-54

Chapter 14: Sheet Metal Design

The Sheet Metal Module 14-2
Setting the Sheet Metal Part Properties 14-4
Creating the Base Feature 14-8
Adding Flanges to a Sheet Metal Part Enhanced 14-9
Creating Contour Flanges 14-16
Creating Lofted Flanges 14-17
Adding a Jog to the Sheet 14-18
Bending the Sheet Metal Part 14-19
Unbending the Sheet Metal Part 14-19
Rebending the Sheet Metal Part 14-20
Filleting or Chamfering Corners 14-20
Closing the Corners of a Sheet Metal Part 14-21
Creating Dimples in a Sheet Metal Part 14-24
Creating Louvers in a Sheet Metal Part 14-27
Creating Drawn Cutouts in a Sheet Metal Part 14-29
Creating Beads in a Sheet Metal Part 14-30
Adding Gussets to a Sheet Metal part 14-33
Adding Hems 14-36
Creating a Sheet Metal Part Using Solid Body 14-38
Converting a Solid Part into a Sheet Metal Part 14-39
Ripping the Corners of a Solid Part 14-40
Creating the Flat Pattern of a Sheet Metal Part 14-40
Creating the Flat Pattern 14-40
Creating the Flat Solid 14-41
Exporting a Flat Pattern 14-42
Renewing Features 14-43
Tutorial 1 14-44
Tutorial 2 14-55
Self-Evaluation Test 14-61
Review Questions 14-62
Exercise 1 14-63
Exercise 2 14-64

CHAPTERS AVAILABLE FOR FREE DOWNLOAD

In this textbook, two chapters have been given for free download. You can download these chapters from our website *www.cadcim.com*. To download these chapters, use the following path: *Textbooks > CAD/CAM > NX > Siemens NX 2019 for Designers, 12th Edition > Chapters for Free Download* and then select the chapter name from the **Chapters for Free Download** drop-down. Click the **Download** button to download the chapter in the PDF format.

Chapter 15: Introduction to Injection Mold Design

Introduction to Injection Mold Design 15-2
Invoking the Mold Environment 15-2
Various Methods For Creating Mold Design 15-4
- Manual Method 15-4
- Semi Automatic 15-4
- Automatic 15-4

Analysis of Part 15-4
- Mold Design Validation 15-5
- Check Wall Thickness 15-6
- Easy Fill Advanced 15-7

Mold Wizard 15-15
- Initialize Project 15-15
- Mold CSYS 15-16
- Shrinkage 15-17
- Check Regions 15-18
- Patch Surface 15-19
- Define Regions 15-20
- Design Parting Surface 15-20
- Edit Parting and Patch Surface 15-21
- Workpiece 15-21
- Cavity Layout 15-22
- Define Cavity and Core 15-25
- Mold Base Library 15-26
- Standard Part Library 15-26
- Design Fill 15-27
- Runner 15-28
- Slider and Lifter Library 15-29
- Slider and Lifter Library 15-30
- Design Ejector Pin 15-31
- Ejector Pin Post Processing 15-33
- Sub Insert Library 15-33
- Pocket 15-34
- View Manager 15-34
- Cooling 15-35
- Assembly Drawings 15-39
- Component Drawing 15-40

Tutorial 1 15-41
Self-Evaluation Test 15-76

Review Questions 15-77
Exercise 1 15-78

Chapter 16: Concepts of Geometric Dimensioning and Tolerancing

History of Tolerances and Allowances 16-2
Methods of Tolerancing 16-2
Limit Dimensioning 16-3
Plus and Minus Tolerancing 16-4
Geometric Tolerances 16-4
Form Tolerances 16-10
Profile Tolerances 16-19
Orientation Tolerances 16-22
Location Tolerances 16-25
Runout Tolerances 16-29
Fits 16-33
Hole Basis System 16-33
Shaft Basis System 16-34
Standards of Fits 16-39
Standard Inch Fits 16-39
Standard Metric Fits 16-42
Tutorial 1 16-46
Self-Evaluation Test 16-52
Review Questions 16-53
Exercise 1 16-53
Index **I-1**

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