Table of Contents

Dedication iii

Preface xxi

Chapter 1: Introduction to SOLIDWORKS 2019

Introduction to SOLIDWORKS 2019 1-2
- Part Mode 1-2
- Assembly Mode 1-3
- Drawing Mode 1-3

System Requirements 1-4
Getting Started with SOLIDWORKS 1-4
Menu Bar and SOLIDWORKS Menus 1-7
CommandManager 1-8
- Part Mode CommandManagers 1-8
- Assembly Mode CommandManagers 1-10
- Drawing Mode CommandManagers 1-10
- Customized CommandManager 1-11

Toolbar 1-12
- Pop-up Toolbar 1-12
- View (Heads-Up) Toolbar 1-12
- Shortcut Bar 1-13
- Mouse Gestures 1-13

Dimensioning Standards and Units 1-14
Important Terms and Their Definitions 1-14
- Feature-based Modeling 1-14
- Parametric Modeling 1-14
- Bidirectional Associativity 1-15
- Windows Functionality 1-16
- SWIFT Technology 1-16
- Geometric Relations 1-16
- Blocks 1-18
- Library Feature 1-18
- Design Table 1-18
- Equations 1-18
- Collision Detection 1-19
- What's Wrong Functionality 1-19
- SimulationXpress 1-19
- Physical Dynamics 1-19
- Physical Simulation 1-20
- Seed Feature 1-20
- FeatureManager Design Tree 1-20
- Absorbed Features 1-20
- Child Features 1-20
- Dependent Features 1-20

Auto-Backup Option 1-20

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

Selecting Hidden Entities 1-21
Hot Keys 1-21
Color Scheme 1-23
Self-Evaluation Test 1-23
Review Questions 1-24

Chapter 2: Drawing Sketches for Solid Models

The Sketching Environment 2-2
Starting a New Session of SOLIDWORKS 2019 2-3
Task Panes 2-3
SOLIDWORKS Resources Task Pane 2-3
Design Library Task Pane 2-5
File Explorer Task Pane 2-5
View Palette Task Pane 2-5
Appearances, Scenes, and Decals Task Pane 2-5
Custom Properties Task Pane 2-6
Starting a New Document in SOLIDWORKS 2019 2-7
Part 2-7
Assembly 2-7
Drawing 2-7
Understanding the Sketching Environment 2-8
Setting the Document Options 2-10
Modifying the Drafting Standards 2-11
Modifying the Linear and Angular Units 2-11
Modifying the Snap and Grid Settings 2-13
Learning Sketcher Terms 2-14
Origin 2-14
Inferencing Lines 2-15
Select Tool 2-15
Invert Selection Tool 2-16
Drawing Lines 2-17
Orientation Rollout 2-17
Options Rollout 2-18
Drawing Continuous Lines 2-18
Drawing Individual Lines 2-19
Line Cursor Parameters 2-20
Drawing Tangent or Normal Arcs Using the Line Tool 2-21
Drawing Construction Lines or Centerlines 2-22
Drawing Midpoint Line 2-22
Drawing the Lines of Infinite Length 2-22
Drawing Circles 2-23
Drawing Circles by Defining their Center Points 2-24
Drawing Circles by Defining Three Points 2-24
Drawing Construction Circles 2-25
Drawing Arcs 2-25
Drawing Tangent/Normal Arcs 2-25
Drawing Centerpoint Arcs 2-27

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

Drawing 3 Point Arcs 2-28
Drawing Rectangles 2-29
Drawing Rectangles by Specifying their Corners 2-29
Drawing Rectangles by Specifying the Center and a Corner 2-30
Drawing Rectangles at an Angle 2-30
Drawing Centerpoint Rectangles at an Angle 2-31
Drawing Parallelograms 2-31
Drawing Polygons 2-32
Drawing Splines 2-34
Drawing Slots 2-35
Creating a Straight Slot 2-35
Creating a Centerpoint Straight Slot 2-35
Creating a 3 Point Arc Slot 2-36
Creating a Centerpoint Arc Slot 2-36
Placing Sketched Points 2-37
Drawing Ellipses 2-37
Drawing Elliptical Arcs 2-38
Drawing Parabolic Curves 2-39
Drawing Conic Curves 2-40
Drawing Display Tools 2-41
Zoom to Fit 2-41
Zoom to Area 2-41
Zoom In/Out 2-42
Zoom to Selection 2-42
Pan 2-42
Previous View 2-42
Redraw 2-43
Shaded Sketch Contours 2-43
Deleting Sketched Entities 2-43
Tutorial 1 2-43
Tutorial 2 2-49
Tutorial 3 2-52
Tutorial 4 2-56
Self-Evaluation Test 2-63
Review Questions 2-64
Exercise 1 2-65
Exercise 2 2-66
Exercise 3 2-66
Exercise 4 2-67

Chapter 3: Editing and Modifying Sketches

Editing Sketched Entities 3-2
Trimming Sketched Entities Enhanced 3-2
Extending Sketched Entities 3-6
Convert Entities 3-6
Intersection Curves 3-7
Filleting Sketched Entities 3-8

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

Chamfering Sketched Entities 3-9
Offsetting Sketched Entities 3-11
Offsetting Edges or Face of a Model 3-12
Mirroring Sketched Entities 3-14
Mirroring Entities Dynamically 3-15
Moving Sketched Entities 3-16
Rotating Sketched Entities 3-17
Scaling Sketched Entities 3-19
Stretching Sketched Entities 3-19
Copying and Pasting Sketched Entities 3-21
Creating Patterns 3-21
Creating Linear Sketch Patterns 3-22
Creating Circular Sketch Patterns 3-25
Editing Patterns 3-27
Writing Text in the Sketching Environment 3-28
Modifying Sketched Entities 3-29
Modifying a Sketched Line 3-29
Modifying a Sketched Circle 3-29
Modifying a Sketched Arc 3-29
Modifying a Sketched Polygon 3-29
Modifying a Spline 3-30
Modifying the Coordinates of a Point 3-32
Modifying an Ellipse or an Elliptical Arc 3-32
Modifying a Parabola 3-32
Dynamically Modifying and Copying Sketched Entities 3-32
Splitting Sketched Entities 3-33
Creating Segments in Sketched Entities 3-33
Tutorial 1 3-34
Tutorial 2 3-39
Tutorial 3 3-43
Self-Evaluation Test 3-48
Review Questions 3-49
Exercise 1 3-50
Exercise 2 3-50
Exercise 3 3-51
Exercise 4 3-51
Exercise 5 3-52
Exercise 6 3-52

Chapter 4: Adding Relations and Dimensions to Sketches

Applying Geometric Relations to Sketches 4-2
Applying Relations Using the Add Relations PropertyManager 4-2
Automatic Relations 4-7
Design Intent 4-8
Dimensioning a Sketch 4-9
Horizontal/Vertical Dimensioning 4-11
Aligned Dimensioning 4-23

Angular Dimensioning 4-24
Diametric Dimensioning 4-26
Radial Dimensioning 4-26
Linear Diametric Dimensioning 4-27
Ordinate Dimensioning 4-28
Path Length Dimension 4-29
Auto Insert Dimension 4-30
Concept of a Fully Defined Sketch 4-30
Fully Defined 4-30
Overdefined 4-30
Underdefined 4-30
Dangling 4-31
No Solution Found 4-31
Invalid Solution Found 4-31
Sketch Dimension or Relation Status 4-31
Deleting Overdefined Dimensions 4-32
Displaying and Deleting Relations 4-35
Opening an Existing File 4-38
Tutorial 1 4-41
Tutorial 2 4-50
Tutorial 3 4-56
Self-Evaluation Test 4-64
Review Questions 4-64
Exercise 1 4-65
Exercise 2 4-66
Exercise 3 4-66

Chapter 5: Advanced Dimensioning Techniques and Base Feature Options

Advanced Dimensioning Techniques 5-2
Fully Defining the Sketches 5-2
Dimensioning the True Length of an Arc 5-4
Measuring Distances and Viewing Section Properties 5-5
Measuring Distances 5-5
Determining the Section Properties of Closed Sketches 5-7
Creating Base Features by Extruding Sketches 5-9
Creating Solid Extruded Features 5-9
Creating Thin Extruded Features 5-12
Creating Base Features by Revolving Sketches 5-14
Creating Solid Revolved Features 5-15
Creating Thin Revolved Features 5-16
Determining the Mass Properties of Parts 5-18
Dynamically Rotating the View of a Model 5-19
Rotating the View Freely in 3D Space 5-19
Rotating the View around a Selected Vertex, Edge, or Face 5-19
Modifying the View Orientation 5-20
Changing the Orientation Using the Reference Triad 5-22

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

Restoring the Previous View 5-22
Displaying the Drawing Area in Viewports 5-22
Displaying the Drawing Area in Two Horizontal Viewports 5-23
Displaying the Drawing Area in Two Vertical Viewports 5-23
Displaying the Drawing Area in Four Viewports 5-24
Display Modes of a Model 5-24
Wireframe 5-24
Hidden Lines Visible 5-25
Hidden Lines Removed 5-25
Shaded With Edges 5-25
Shaded 5-25
Additional Display Modes 5-25
Shadows in Shaded Mode 5-25
Perspective 5-26
Assigning Materials and Textures to Models 5-26
Assigning Materials to a Model 5-27
Changing the Appearance of the Model 5-27
Editing the Appearances 5-29
Tutorial 1 5-30
Tutorial 2 5-34
Tutorial 3 5-39
Self-Evaluation Test 5-43
Review Questions 5-44
Exercise 1 5-45
Exercise 2 5-46
Exercise 3 5-46

Chapter 6: Creating Reference Geometries

Importance of Sketching Planes 6-2
Creating Reference Geometry 6-2
Reference Planes 6-3
Creating New Planes 6-4
Creating Reference Axes 6-10
Creating Reference Points 6-13
Creating Reference Coordinate Systems 6-16
Creating Center of Mass 6-16
Creating a Bounding Box 6-17
Advanced Boss/Base Options 6-18
Modeling Using the Contour Selection Method 6-25
Creating Cut Features 6-28
Creating Extruded Cuts 6-29
Creating Multiple Bodies in the Cut Feature 6-32
Creating Revolved Cuts 6-34
Concept of the Feature Scope 6-34
Tutorial 1 6-35
Tutorial 2 6-39
Tutorial 3 6-44

Self-Evaluation Test 6-50
Review Questions 6-51
Exercise 1 6-52
Exercise 2 6-53
Exercise 3 6-54

Chapter 7: Advanced Modeling Tools-I

Advanced Modeling Tools 7-2
Creating Simple Holes 7-2
Creating Standard Holes Using the Hole Wizard 7-3
Creating Advanced Holes 7-9
Creating Threads 7-12
Adding External Cosmetic Threads 7-15
Creating Fillets Enhanced 7-16
Selection Options 7-32
Creating Fillets Using the FilletXpert 7-35
Creating Chamfers 7-36
Creating Shell Features 7-42
Creating Wrap Features 7-45
Tutorial 1 7-47
Tutorial 2 7-54
Tutorial 3 7-61
Self-Evaluation Test 7-69
Review Questions 7-70
Exercise 1 7-71
Exercise 2 7-72
Exercise 3 7-73

Chapter 8: Advanced Modeling Tools-II

Advanced Modeling Tools 8-2
Creating Mirror Features 8-2
Creating Linear Pattern Features 8-6
Creating Circular Pattern Features 8-12
Creating Sketch Driven Patterns 8-14
Creating Curve Driven Patterns 8-16
Creating Table Driven Patterns 8-18
Creating Fill Patterns 8-20
Creating Variable Patterns 8-24
Creating Rib Features 8-25
Displaying the Section View of a Model 8-30
Changing the Display States 8-33
Tutorial 1 8-35
Tutorial 2 8-41
Tutorial 3 8-46
Self-Evaluation Test 8-56
Review Questions 8-57

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

Exercise 1 8-58
Exercise 2 8-58
Exercise 3 8-59
Exercise 4 8-61

Chapter 9: Editing Features

Editing the Features of a Model 9-2
Editing Using the Edit Feature Tool 9-2
Editing Sketches of the Sketch-based Features 9-3
Editing the Sketch Plane Using the Edit Sketch Plane Tool 9-3
Editing Using the Instant3D Tool 9-4
Editing Features and Sketches by Using the Cut, Copy, and Paste Options 9-7
Cutting, Copying, and Pasting Features and Sketches from One Document to Another 9-8
Copying Features using Drag and Drop 9-8
Deleting Features 9-9
Deleting Bodies 9-10
Suppressing Features 9-11
Unsuppressing the Suppressed Features 9-11
Unsuppressing Features with Dependents 9-11
Hiding Bodies 9-11
Moving and Copying Bodies 9-12
Reordering the Features 9-14
Rolling Back the Feature 9-15
Renaming Features 9-16
Creating Folders in the FeatureManager Design Tree 9-16
What's Wrong Functionality 9-16
Tutorial 1 9-18
Tutorial 2 9-24
Tutorial 3 9-28
Self-Evaluation Test 9-34
Review Questions 9-34
Exercise 1 9-35
Exercise 2 9-36

Chapter 10: Advanced Modeling Tools-III

Advanced Modeling Tools 10-2
Creating Sweep Features 10-2
Creating Cut-Sweep Features 10-12
Creating Loft Features 10-13
Adding a Section to a Loft Feature 10-21
Creating Lofted Cuts 10-22
Creating 3D Sketches 10-23
Creating Grid Systems 10-25
Mirroring Sketch Entities in a 3D Sketch 10-26

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

Editing 3D Sketches 10-26
Creating Curves 10-26
Extruding a 3D Sketch 10-37
Creating Draft Features 10-38
Tutorial 1 10-42
Tutorial 2 10-45
Tutorial 3 10-48
Self-Evaluation Test 10-52
Review Questions 10-53
Exercise 1 10-54
Exercise 2 10-55
Exercise 3 10-56

Chapter 11: Advanced Modeling Tools-IV

Advanced Modeling Tools 11-2
Creating Dome Features 11-2
Creating Indents 11-5
Creating Deform Features 11-8
Creating Flex Features 11-12
Creating Fastening Features 11-16
Creating the Mounting Boss 11-16
Creating Snap Hooks 11-19
Creating Snap Hook Grooves 11-22
Creating Vents 11-23
Creating a Lip/Groove Feature 11-27
Creating Freeform Features 11-31
Dimensioning a Part Using MBD Dimension 11-35
Specifying the Datum 11-35
Adding Dimensions 11-38
Specifying the Location of a Feature 11-39
Adding Geometric Tolerance to the Features 11-39
Collecting Pattern Features 11-41
Adding Dimensions Automatically 11-42
Tutorial 1 11-43
Tutorial 2 11-47
Tutorial 3 11-50
Tutorial 4 11-53
Self-Evaluation Test 11-59
Review Questions 11-60
Exercise 1 11-61

Chapter 12: Assembly Modeling-I

Assembly Modeling 12-2
Types of Assembly Design Approach 12-2
Creating Bottom-up Assemblies 12-3
Placing Components in the Assembly Document 12-4

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

Assembling Components 12-8
Creating Top-down Assemblies 12-30
Creating Components in the Top-down Assembly 12-30
Moving Individual Components 12-31
Moving Individual Components by Dragging 12-31
Moving Individual Components Using the Move Component Tool 12-31
Rotating Individual Components 12-32
Rotating Individual Components by Dragging 12-32
Rotating Individual Components Using the Rotate Component Tool 12-32
Moving and Rotating Individual Components Using the Triad 12-33
Assembly Visualization 12-34
Tutorial 1 12-35
Tutorial 2 12-49
Self-Evaluation Test 12-55
Review Questions 12-56
Exercise 1 12-57
Exercise 2 12-60
Exercise 3 12-68

Chapter 13: Assembly Modeling-II

Advanced Assembly Mates 13-2
Applying the Profile Center Mate 13-2
Applying the Symmetric Mate 13-3
Applying the Width Mate 13-3
Applying the Path Mate 13-4
Applying the Distance Mate 13-6
Applying the Angle Mate 13-6
Applying the Linear/Linear Coupler Mate 13-6
Mechanical Mates 13-6
Applying the Cam Mate 13-6
Applying the Slot Mate 13-7
Applying the Hinge Mate 13-8
Applying the Gear Mate 13-9
Applying the Rack Pinion Mate 13-10
Applying the Screw Mate 13-10
Applying the Universal Joint Mate 13-11
Creating Sub-assemblies 13-11
Bottom-up Sub-assembly Design Approach 13-11
Top-down Sub-assembly Design Approach 13-12
Inserting a New Sub-assembly 13-12
Deleting Components and Sub-assemblies 13-12
Editing Assembly Mates 13-13
Replacing Mated Entities 13-13
Editing Components 13-14
Editing Sub-assemblies 13-15
Dissolving Sub-assemblies 13-16
Replacing Components 13-16

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

Creating Patterns of Components in an Assembly 13-18
Pattern Driven Component Pattern 13-18
Local Pattern 13-20
Copying and Mirroring Components Enhanced 13-21
Copying a Component along with Mates 13-23
Copying Multiple Components 13-23
Magnetic Mates 13-24
Simplifying Assemblies Using the Visibility Options 13-27
Hiding Components 13-27
Hiding Faces Temporarily 13-27
Suppressing and Unsuppressing the Components 13-28
Changing the Transparency Conditions 13-28
Changing the Display States 13-28
Checking Interferences in an Assembly 13-28
Checking the Hole Alignment 13-30
Creating Assemblies for Mechanism 13-31
Analyzing Collisions Using the Collision Detection Tool 13-32
Creating the Exploded State of an Assembly Enhanced 13-34
Creating the Explode Line Sketch 13-36
Creating Smart Explode Lines 13-38
Tutorial 1 13-39
Tutorial 2 13-51
Self-Evaluation Test 13-56
Review Questions 13-56
Exercise 1 13-57

Chapter 14: Working with Drawing Views-I

The Drawing Mode 14-2
Starting a Drawing Document 14-2
Types of Views 14-5
Model View 14-5
Projected View 14-6
Section View 14-6
Aligned Section View 14-6
Removed Section View 14-6
Auxiliary View 14-6
Detail View 14-6
Break View 14-6
Broken-out Section View 14-6
Crop View 14-6
Alternate Position View 14-7
Generating Standard Drawing Views 14-7
Generating Model Views 14-7
Using the View Palette to Place the Drawing Views 14-9
Generating the Three Standard Views 14-10
Generating Standard Views Using the Relative View Tool 14-12
Generating Standard Views Using the Predefined View Tool 14-14

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

Generating Derived Views Enhanced 14-15
Generating Projected Views 14-16
Generating Section Views 14-17
Generating Broken-out Section Views 14-25
Generating Removed Section Views New 14-27
Generating Auxiliary Views 14-28
Generating Detail Views 14-30
Generating Crop Views 14-33
Generating Broken Views 14-33
Generating Alternate Position Views 14-35
Generating Drawing Views of the Exploded State of an Assembly 14-37
Working with Interactive Drafting in SOLIDWORKS 14-38
Editing and Modifying Drawing Views 14-38
Changing the View Orientation 14-38
Changing the Scale of Drawing Views 14-38
Deleting Drawing Views 14-39
Rotating Drawing Views 14-39
Manipulating the Drawing Views 14-39
Modifying the Hatch Pattern in Section Views 14-40
Properties Rollout 14-40
Layer Rollout 14-41
Options Rollout 14-41
Tutorial 1 14-41
Tutorial 2 14-47
Self-Evaluation Test 14-52
Review Questions 14-53
Exercise 1 14-54

Chapter 15: Working with Drawing Views-II

Adding Annotations to Drawing Views 15-2
Generating Annotations Using the Model Items Tool 15-2
Adding Reference Annotations 15-4
Aligning the Dimensions 15-19
Editing Annotations 15-19
Adding the Bill of Materials (BOM) to a Drawing 15-20
Table Template Rollout 15-20
Table Position Rollout 15-20
BOM Type Rollout 15-20
Configurations Rollout 15-21
Part Configuration Grouping Rollout 15-22
Keep Missing Item/Row Rollout 15-22
Item Numbers Rollout 15-22
Border Rollout 15-23
Setting Anchor Point for the BOM 15-23
Linking Bill of Materials 15-23
Adding Balloons to the Drawing Views 15-23
Adding Balloons Using the AutoBalloon Tool 15-24

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

Creating Magnetic Lines 15-26
Adding New Sheets to the Drawing Document 15-27
Editing the Sheet Format 15-28
Creating User-Defined Sheet Formats 15-28
Tutorial 1 15-29
Tutorial 2 15-35
Self-Evaluation Test 15-39
Review Questions 15-40
Exercise 1 15-41

Chapter 16: Surface Modeling

Surface Modeling 16-2
Creating an Extruded Surface 16-2
Creating a Revolved Surface 16-4
Creating a Swept Surface 16-4
Creating a Lofted Surface 16-7
Creating a Boundary Surface 16-9
Creating a Planar Surface 16-14
Creating a Fill Surface 16-14
Creating a Radiated Surface 16-19
Offsetting Surfaces 16-20
Trimming Surfaces 16-21
Untrimming Surfaces 16-24
Extending Surfaces 16-27
Knitting Surfaces 16-29
Filleting Surfaces 16-30
Creating a Mid-Surface 16-31
Deleting Holes from Surfaces 16-32
Replacing Faces 16-33
Deleting Faces 16-34
Moving and Copying Surfaces 16-36
Mirroring Surface Bodies 16-36
Adding Thickness to Surface Bodies 16-36
Creating a Thicken Surface Cut 16-37
Creating a Surface Cut Enhanced 16-38
Tutorial 1 16-40
Tutorial 2 16-49
Self-Evaluation Test 16-59
Review Questions 16-59
Exercise 1 16-60
Exercise 2 16-61

Chapter 17: Working with Blocks

Introduction to Blocks 17-2
Blocks Toolbar 17-2
Saving a Sketch as a Block in the Design Library 17-7

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

Creating Mechanisms by Using Blocks 17-7
Creating the Rack and Pinion Mechanism 17-8
Creating the Cam and Follower Mechanism 17-9
Applying Motion to Blocks 17-10
Creating Parts from Blocks 17-12
Selected Blocks 17-12
Block to Part Constraint 17-12
Tutorial 1 17-13
Tutorial 2 17-20
Self-Evaluation Test 17-27
Review Questions 17-27
Exercise 1 17-28

Chapter 18: Sheet Metal Design

Sheet Metal Design 18-2
Designing the Sheet Metal Components by Creating the Base Flange 18-2
Creating the Base Flange Enhanced 18-3
Understanding the FeatureManager Design Tree of a Sheet Metal Component 18-6
Creating the Edge Flange 18-8
Creating Tabs 18-16
Creating a Tab and Slot Enhanced 18-17
Creating the Sketched Bend 18-20
Creating the Miter Flange 18-21
Creating Closed Corners 18-23
Creating Welded Corners 18-24
Breaking the Corners 18-25
Creating Corner Relief 18-26
Creating Hems 18-27
Creating the Jog Bend 18-30
Creating the Swept Flange 18-32
Creating Cuts on the Planar Faces of the Sheet Metal Components 18-33
Creating Lofted Bends 18-34
Creating a Flat Pattern View of the Sheet Metal Components 18-35
Creating Sheet Metal Components from a Flat Sheet 18-38
Creating a Sheet Metal Component from a Flat Part 18-39
Converting a Part or a Flat Part into Sheet Metal by Adding Bends 18-39
Adding Bends to the Flattened Sheet Metal Component 18-40
Unbending the Sheet Metal Part Using the No Bends Tool 18-41
Creating a Sheet Metal Component by Designing it as a Part 18-42
Types of Bends 18-42
Converting a Solid Body into a Sheet Metal Part Enhanced 18-44
Designing a Sheet Metal Part from a Solid Shelled Model 18-45
Ripping the Edges 18-46
Creating Cuts in Sheet Metal Components Across the Bends 18-47
Creating Cuts in a Sheet Metal Component Created from a Solid Model 18-47
Creating Cuts in a Sheet Metal Component Created Using the Base Flange 18-49

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

Creating Cylindrical and Conical Sheet Metal Components 18-50
Creating Normal Cuts 18-51
Inserting Forming Tools 18-52
Creating Forming Tools 18-54
Generating the Drawing View of the Flat Pattern of the Sheet Metal Components 18-55
Tutorial 1 18-56
Tutorial 2 18-62
Self-Evaluation Test 18-72
Review Questions 18-72
Exercise 1 18-73

CHAPTERS AVAILABLE FOR FREE DOWNLOAD

In this textbook, three chapters have been given for free download. You can download these chapters from our website *www.cadcim.com*. To download these chapters, follow the path: *Textbooks > CAD/CAM > SOLIDWORKS > SOLIDWORKS 2019 for Designers > 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 19: Equations, Configurations, and Library Features

Working with Equations 19-2
Adding Global Variables 19-4
Suppressing and Unsuppressing Features 19-6
Adding Equations 19-7
Editing Equations 19-8
Deleting/Disabling Equations 19-9
Exporting/Importing Equations 19-9
Working with Configurations 19-9
Creating Configurations Manually 19-9
Editing the Features of a Part with Multiple Configurations 19-11
Creating Configurations by Using Design Tables 19-12
Changing the Suppression State by Using the Design Table 19-16
Editing the Design Table 19-19
Deleting the Design Table 19-20
Changing the Suppression State of a Component without Invoking the Design Table 19-20
Changing the Visibility of Components in Different Configurations of an Assembly 19-21
Library Features 19-22
Creating a Library Feature 19-22
Placing Library Features in a Part 19-23
Editing the Library Features 19-25
Dissolving the Library Features 19-26
Tutorial 1 19-26
Tutorial 2 19-30
Self-Evaluation Test 19-33
Review Questions 19-33

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

Exercise 1 19-34
Exercise 2 19-35
Exercise 3 19-35

Chapter 20: Motion Study

Introduction to Motion Study 20-2
Simulating the Motion of an Assembly 20-2
Timeline 20-2
Timebar 20-2
Keypoint 20-3
MotionManager Design Tree 20-3
Toolbar 20-3
Tutorial 1 20-15
Tutorial 2 20-17
Self-Evaluation Test 20-20
Review Questions 20-20
Exercise 1 20-21
Exercise 2 20-22

Chapter 21: Introduction to Mold Design

Introduction to Mold Design 21-2
Mold Designing Using SOLIDWORKS 21-4
Tutorial 1 21-15
Tutorial 2 21-27
Self-Evaluation Test 21-37
Review Questions 21-38
Exercise 1 21-38

Student Projects SP-1
Index I-1

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