Visit the following websites to learn more about this book:
# Table of Contents

## PART I: Introductory Engineering Design Principles with SolidWorks

<table>
<thead>
<tr>
<th>1. Introduction</th>
<th>1-1</th>
</tr>
</thead>
<tbody>
<tr>
<td>SolidWorks within the Context of CAE Framework</td>
<td>1-1</td>
</tr>
<tr>
<td>Three-dimensional (3D) Wireframes</td>
<td>1-2</td>
</tr>
<tr>
<td>Constructive Solid Geometry (CSG)</td>
<td>1-2</td>
</tr>
<tr>
<td>Boundary Representation (B-Rep) Scheme</td>
<td>1-3</td>
</tr>
<tr>
<td>Feature-based Parametric Modeling</td>
<td>1-4</td>
</tr>
<tr>
<td>Background of SolidWorks</td>
<td>1-4</td>
</tr>
<tr>
<td>Starting a SolidWorks Session</td>
<td>1-5</td>
</tr>
<tr>
<td>SolidWorks User Interface</td>
<td>1-5</td>
</tr>
<tr>
<td>Menu Bar Toolbar</td>
<td>1-6</td>
</tr>
<tr>
<td>Menu Bar Menu</td>
<td>1-6</td>
</tr>
<tr>
<td>Task Pane</td>
<td>1-7</td>
</tr>
<tr>
<td>CommandManager</td>
<td>1-8</td>
</tr>
<tr>
<td>FeatureManager Design Tree</td>
<td>1-10</td>
</tr>
<tr>
<td>Head-up View Toolbar</td>
<td>1-10</td>
</tr>
<tr>
<td>Drop-down Menu</td>
<td>1-11</td>
</tr>
<tr>
<td>Right-click</td>
<td>1-12</td>
</tr>
<tr>
<td>Consolidated Toolbar</td>
<td>1-12</td>
</tr>
<tr>
<td>System Feedback</td>
<td>1-13</td>
</tr>
<tr>
<td>Setting the Document Options</td>
<td>1-13</td>
</tr>
<tr>
<td>System Options</td>
<td>1-13</td>
</tr>
<tr>
<td>Document Properties</td>
<td>1-14</td>
</tr>
<tr>
<td>File Management</td>
<td>1-16</td>
</tr>
<tr>
<td>Caution Needed During SolidWorks Sessions</td>
<td>1-16</td>
</tr>
<tr>
<td>Starting a New Document in SolidWorks</td>
<td>1-16</td>
</tr>
<tr>
<td>My First Part</td>
<td>1-17</td>
</tr>
<tr>
<td>Useful SolidWorks Resources</td>
<td>1-21</td>
</tr>
<tr>
<td>Compatibility of SolidWorks with other Software</td>
<td>1-22</td>
</tr>
<tr>
<td>Summary</td>
<td>1-22</td>
</tr>
<tr>
<td>Exercises</td>
<td>1-23</td>
</tr>
</tbody>
</table>

## 2. Geometric Construction Tools

| 2-1 |
| Introduction | 2-1 |
| Sketch Entities | 2-1 |
| Line | 2-2 |
| Rectangle | 2-3 |
| Parallelogram | 2-3 |
| Slot | 2-4 |
| Polygon | 2-4 |
Circle 2-5
Arc 2-5
Ellipse 2-6
Parabola 2-7
Spline 2-8
Sketch Tools 2-9
Fillet 2-10
Chamfer 2-10
Offset 2-11
Convert Entities 2-12
Trim 2-12
Extend 2-13
Split Entities 2-13
Construction Geometry 2-13
Mirror 2-14
Stretch Entities 2-14
Move Entities 2-15
Rotate Entities 2-16
Scale Entities 2-16
Copy Entities 2-17
Pattern 2-18
Summary 2-19
Exercises 2-20

3. Features 3-1
Introduction 3-2
Extruded Boss/Base 3-3
  Draft, Dome, Rib 3-3
  Extruded Cut 3-6
  Revolved Boss/Base 3-7
  Revolved Cut 3-10
  Lofted Boss/Base 3-11
  Lofted Cut 3-14
  Swept Boss/Base 3-15
  Swept Cut 3-17
  Hole Wizard 3-18
  Shell 3-21
  Fillet Tool 3-22
  Chamfer Tool 3-25
  Linear Pattern 3-26
  Circular Pattern 3-28
  Mirror 3-29
  Editing Features 3-29
  Tutorials 3-31
    Tutorial 1 3-31
A Comprehensive Introduction to SolidWorks 2013

Tutorial 2
Tutorial 3
Tutorial 4
Tutorial 5
Tutorial 6
Tutorial 7
Tutorial 8
Tutorial 9
Tutorial 10
Patterns—Advanced Methods
Sketch Driven Pattern
Curve Driven Pattern
Table Driven Pattern
Reference Planes
Tutorial 1 for Planes
Tutorial 2 for Planes
Tutorial 3 for Planes
Tutorial 4 for Planes
Creating Patterns
Summary
Exercises

4. Part Modeling—CSWA Preparations

Introduction
Tutorials
Tutorial 4-1a Widget
Tutorial 4-1b Widget
Tutorial 4-2
Tutorial 4-3a
Tutorial 4-3b
Tutorial 4-4
Tutorial 4-5a
Tutorial 4-5b
Tutorial 4-6a
Tutorial 4-6b
Tutorial 4-7a
Tutorial 4-7b
Summary
Exercises

5. Advanced Part Modeling—CSWA Preparations

Introduction
Advanced Part Modeling Tutorials
Tutorial 1: Block with hook
Tutorial 2: Bracket 1
Tutorial 3: Bracket 2

vii
A Comprehensive Introduction to SolidWorks 2013

Tutorial 4: Inclined Block 5-15
Tutorial 5: Inclined Block 5-22
Tutorial 6: Inclined Block 5-33
Tutorial 7: Bent Plate 5-42
Tutorial 8: Inclined Block 5-49
Tutorial 9: Model with Notched Offset Section View 5-62
Summary 5-70
Exercises 5-71

6. Revolved, Swept, and Lofted Parts 6-1
Revolved Boss/Base 6-1
- Practical Examples 6-3
  - Engine Cylinder 6-3
  - Pulley 6-5
Swept Boss/Base 6-8
- Practical Examples 6-10
  - Spring 6-10
  - O-Ring 6-12
  - Threaded Cap 6-15
Lofted Boss/Base 6-19
- Practical Examples 6-22
  - Impeller 6-22
  - Aircraft Wing 6-27
Practical Swept Feature: Elbow Casting 6-30
Practical Swept Feature: Lathe Tailstock 6-37
Summary 6-42
Exercises 6-43

7. Part Modeling with Equation Driven Curves 7-1
Introduction 7-1
Equation Driven Curves 7-1
Problem 1 7-2
Integral Calculus Solution for Area 7-2
SolidWorks Solution for Area 7-3
Part Design Using the SolidWorks Equation Driven Curves 7-5
Effect of Changing the Axis of Rotation 7-7
Problem 2 7-8
Integral Calculus Solution for Area 7-8
SolidWorks Solution for Area 7-8
Part Design Using the SolidWorks Equation Driven Curves 7-10
Problem 3 7-11
Integral Calculus Solution for Volume 7-12
SolidWorks Solution for Area 7-12
Part Design Using the SolidWorks Equation Driven Curves  7-13
Problem 4  7-14
Integral Calculus Solution for Area  7-14
SolidWorks Solution for Area  7-15
Problem 5  7-17
Integral Calculus Solution for Area  7-17
SolidWorks Solution for Area  7-17
Part Design Using the SolidWorks Equation Driven Curves  7-19
Summary  7-21
Exercises  7-22

8. Assembly Modeling —CSWA Preparations  8-1
Introduction  8-1
Starting the Assembly Mode of SolidWorks  8-2
Inserting Components in the Assembly Document  8-3
Mates  8-5
Assembly Modeling Methodology  8-8
Project  8-10
Planning the Assembly  8-10
Starting the Assembly Mode of SolidWorks  8-10
Inserting Components in the Assembly Document  8-12
Mates  8-14
Assembling the Components  8-14
Assembly Analysis  8-19
Exploded View  8-20
Animated Exploded View  8-22
Tutorials  8-23
  Tutorial 8-1  8-23
  Tutorial 8-2  8-26
  Tutorial 8-3  8-29
Large Assemblies  8-31
Summary  8-34

9. Part and Assembly Drawings—CSWA Preparations  9-1
Introduction  9-1
Orthographic Projection  9-1
Creating a SolidWorks Drawing Template  9-3
  Document Properties  9-3
  Sheet Properties  9-6
  Title Block  9-7
  Saving the Template  9-8
Part Drawing of Tool Post  9-9
Screw: Standard Views  9-10
A Comprehensive Introduction to SolidWorks 2013

Post: Standard Views 9-11
Wedge: Standard Views 9-13
Ring: Standard Views 9-15
Block: Standard Views 9-17
Assigning Properties through File > Properties 9-19
Assigning Properties through the Configuration Manager 9-21
Tool Post Assembly Drawing 9-22
Exploded View 9-22
Balloons 9-25
Bill of Material (BOM) 9-26
Inserting Section Views 9-28
Half Section 9-31
Notched Offset Section View 9-34
Aligned Section View 9-37
Full Section 9-40
Summary 9-42
Exercises 9-42

PART II: Intermediate Engineering Design Principles with SolidWorks

10. Reverse Engineering Using Auto Trace & FeatureWorks 10-1
Reverse Engineering 10-1
SolidWorks Reverse Engineering Tools 10-3
Create Auto Trace Tool 10-3
Methodology for Importing and Extracting Image Features 10-6
FeatureWorks Tool 10-13
FeatureWorks Product Overview 10-13
FeatureWorks Add-In 10-16
FeatureWorks Options 10-17
FeatureWorks PropertyManager 10-18
Problem Description 10-18
Automatic Feature Recognition Methodology 10-19
Import Geometry 10-19
Import Diagnostics 10-21
Automatic Feature Recognition 10-23
Interactive Feature Recognition Methodology 10-24
Summary 10-34
Exercises 10-34
References 10-34

11. Top-Down Design 11-1
Designing From Layout 11-1
Layout of the Cabinet 11-1
Microcontroller-Insert Component 11-8
Cooling Unit-Insert Component 11-12
Power Supply-Insert Component 11-13
Housing-Insert Component 11-15
Designing from Part Outline 11-19
Part Model 11-19
Modeling In-Context 11-19
Mold Design Using Top-Down Approach 11-27
Summary 11-34
Exercises 11-34

12. Surface Modeling 12-1
Generalized Methodology for Freeform Surface Design 12-1
Control Polygon 12-1
Lofting B-Splines Using a Control Polygon 12-3
Freeform Surface Design 12-5
Effect of Modifying a Control Polygon on Freeform Surface 12-6
Extruded Surface: Type I 12-8
Revolved Surface: Type I 12-9
Knitting Multiple Surfaces 12-10
Thicken Feature 12-10
Fill Surface 12-11
Extruded Surface 12-12
Revolved Surface 12-14
Swept Surface 12-15
Loft Surface 12-19
Freeform Surface Design: Boundary Surface 12-20
Summary 12-31
Exercises 12-32

13. Toolboxes and Design Libraries 13-1
Introduction 13-1
SolidWorks Toolbox Add-Ins 13-1
How to Use the SolidWorks Design Library & Toolbox 13-2
Features Available in the Toolbox via CommandManager 13-2
Features Available in the Toolbox via Design Library 13-3
Structural Steel 13-3
Beam Calculator 13-12
Grooves 13-15
O-Ring Grooves: Retaining Ring Grooves 13-17
Creating Standard Parts Using Design Library & Toolbox 13-18
Adding Set Screws to the Collar of a Shaft 13-19
Gear Design Using SolidWorks 13-21
Design Methodology for Gears 13-21
SolidWorks Solution Procedure 13-22
Spur Gear Design 13-23
Problem Description 13-23
SolidWorks Solution Procedure 13-23
Animation 13-31
Helical Gear Design 13-33
Bevel Gear Design 13-41
Summary 13-47
Exercises 13-47

14. Animation with Basic Motion 14-1
Different Types of Motion Studies 14-1
Modeling of Linkages 14-2
Assembly Modeling of Linkages 14-3
MotionManager Interface 14-7
Summary 14-10

15. Animation with SolidWorks Motion 15-1
Types of Motion Studies 15-1
Deciding Which Type of Study to Use 15-2
Animation, Basic Motion & Motion Analysis 15-2
SolidWorks Motion Overview 15-3
SolidWorks Motion Add-In 15-4
Four-bar Linkage Mechanism and Slider-Crank Mechanism 15-5
Problem Description 15-6
SolidWorks Parts and Assembly 15-6
Using SolidWorks Motion 15-7
Methodology for Motion Analysis Using SolidWorks Motion 15-7
Defining Bodies 15-7
Driving Joints 15-8
Turning off Gravity 15-10
Running Motion Analysis 15-12
Kinematic Analysis 15-13
Interference Check 15-16
Dynamic Analysis 15-17
Validating the Results 15-22
Summary 15-22
Exercises 15-23

16. Rendering 16-1
Introduction 16-1
Surface Model 16-1
Zebra Stripes 16-2
Inserting the Zebra Stripes Tool 16-2
Including Zebra Stripes Effects on a Part 16-4
Removing Zebra Stripes Effects 16-4
Curvature 16-5
Inserting the Curvature Tool 16-5
PhotoWorks Toolbar 16-6
Inserting the PhotoWorks Tool 16-6
Summary 16-8
Exercises 16-8

PART III: Engineering Design Practice with SolidWorks

17. Mold Design 17-1
Mold Design Background 17-1
Mold Design Tools Overview 17-2
Mold Design Methodology 17-4
Enhancements in SolidWorks for Mold Design 17-4
Summary 17-11
Exercises 17-11

18. Sheet Metal Parts-I 18-1
Sheet Metal Manufacturing Processes 18-1
Sheet Metal Part Design Methodology Using Insert Bends 18-3
Sheet Metal Part Design Methodology Using Base Flange 18-21
Summary 18-24
Exercises 18-24

19. Sheet Metal Parts-II 19-1
Comparing Sheet Metal Design Methods 19-1
Advantages of the Base Flange Approach 19-1
Tutorials on Base Flange Approach for Sheet Metal Design 19-2
The Sheet Metal User Interface 19-2
Tutorial 1: General Sheet Metal Part 19-3
Tutorial 2: P1 19-15
Tutorial 3: P2 19-22
Tutorial 4: Hanger Support 19-30
Tutorial 5: Jogged Sheet Metal Part 19-39
Tutorial 6: Lofted Sheet Metal 19-45
Tutorial 7: P9 19-48
Tutorial 8: General Sheet Metal 19-60
Tutorial 9: Hanger 19-70
Tutorial 10: CSWP-SMTL 19-83
Summary 19-92
Exercises 19-93
20. Weldments  20-1
   Introduction  20-1
   Creating Parts with a 3D Sketch  20-1
   Tutorial 1  20-2
   Tutorial 2  20-4
   Tutorial 3  20-5
   Weldments Toolbar  20-8
      Structural Member  20-8
      Trimming the Structural Members  20-11
      Adding End Caps to Structural Members  20-12
      Adding Gussets to Structural Members  20-13
      Adding Fillet Beads to Structural Members  20-14
   Weldment of Parts  20-15
   Assembly versus Solid Bodies  20-16
   Summary  20-22
   Exercises  20-23

21. Routings in Piping and Tubing  21-1
   Introduction  21-1
   Activating the SolidWorks Routing Add-Ins  21-1
   Background  21-2
   Customizing Routing Templates  21-2
   Adding Parts to the Routing Library  21-3
   Illustration 1  21-3
      Starting a Route  21-4
      Creating a Route  21-6
   Illustration 2  21-7
   Route Drawing  21-10

22. Power Transmission Elements  22-1
   Gears and Power Transmission  22-1
   Spur Gears  22-2
      Creating Gears Using SolidWorks  22-2
      Problem Description  22-3
      Support Plate Sizing  22-4
      Gear Assembly Modeling Using SolidWorks  22-4
      Assembly of the Support Plate, Pin and Gears  22-6
      Animation  22-10
   Rack and Pinion Gears  22-11
      Problem Description  22-11
      Gear Assembly Modeling Using SolidWorks  22-12
      Animation  22-18
   Belts and Pulleys  22-20
      Problem Description  22-20
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Belt and Pulley Assembly Modeling Using SolidWorks</td>
<td>22-21</td>
</tr>
<tr>
<td>Animation</td>
<td>22-27</td>
</tr>
<tr>
<td>Chain Drive: Chains and Sprockets</td>
<td>22-28</td>
</tr>
<tr>
<td>Problem Description</td>
<td>22-28</td>
</tr>
<tr>
<td>Chain and Sprocket Assembly Modeling Using SolidWorks</td>
<td>22-29</td>
</tr>
<tr>
<td>Animation</td>
<td>22-35</td>
</tr>
<tr>
<td>Bevel Gear Box Design</td>
<td>22-36</td>
</tr>
<tr>
<td>Problem Description</td>
<td>22-38</td>
</tr>
<tr>
<td>Assembly Modeling</td>
<td>22-39</td>
</tr>
<tr>
<td>Summary</td>
<td>22-43</td>
</tr>
<tr>
<td>Exercises</td>
<td>22-43</td>
</tr>
</tbody>
</table>

**23. Cam Design**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>23-1</td>
</tr>
<tr>
<td>Types of Cams</td>
<td>23-1</td>
</tr>
<tr>
<td>Types of Followers</td>
<td>23-1</td>
</tr>
<tr>
<td>Creating Cams Using the Traditional Method</td>
<td>23-1</td>
</tr>
<tr>
<td>Creating Cams in SolidWorks</td>
<td>23-2</td>
</tr>
<tr>
<td>Problem Definition</td>
<td>23-2</td>
</tr>
<tr>
<td>SolidWorks Toolbox Add-ins</td>
<td>23-2</td>
</tr>
<tr>
<td>To Access the SolidWorks Cams Tool</td>
<td>23-3</td>
</tr>
<tr>
<td>Cam-Circular Setup</td>
<td>23-3</td>
</tr>
<tr>
<td>Cam-Circular Motion</td>
<td>23-5</td>
</tr>
<tr>
<td>Cam-Circular Creation</td>
<td>23-7</td>
</tr>
<tr>
<td>Cam Model Modification</td>
<td>23-8</td>
</tr>
<tr>
<td>Creating a Hub</td>
<td>23-9</td>
</tr>
<tr>
<td>Creating a Hub Using the Cam-Circular Dialog</td>
<td>23-10</td>
</tr>
<tr>
<td>Creating a Hole for a Key Using the Hole Wizard</td>
<td>23-11</td>
</tr>
<tr>
<td>Cam Shaft Assembly</td>
<td>23-12</td>
</tr>
<tr>
<td>Spring</td>
<td>23-12</td>
</tr>
<tr>
<td>Cam Follower Bracket</td>
<td>23-13</td>
</tr>
<tr>
<td>Cam Bracket</td>
<td>23-14</td>
</tr>
<tr>
<td>Roller</td>
<td>23-15</td>
</tr>
<tr>
<td>Cam Shaft</td>
<td>23-15</td>
</tr>
<tr>
<td>Handle</td>
<td>23-16</td>
</tr>
<tr>
<td>Pin</td>
<td>23-16</td>
</tr>
<tr>
<td>Assembly of Cam Shaft Components</td>
<td>23-16</td>
</tr>
<tr>
<td>Exercises</td>
<td>23-17</td>
</tr>
</tbody>
</table>

**24. Mechanism Design Using Blocks**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>24-1</td>
</tr>
<tr>
<td>Blocks Toolbar</td>
<td>24-1</td>
</tr>
<tr>
<td>Problem Description</td>
<td>24-2</td>
</tr>
<tr>
<td>Creating Sketches of a Mechanism</td>
<td>24-3</td>
</tr>
</tbody>
</table>
A Comprehensive Introduction to SolidWorks 2013

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Saving the Sketches as Different Block Files</td>
<td>24-4</td>
</tr>
<tr>
<td>Inserting the Block into the Layout Environment</td>
<td>24-4</td>
</tr>
<tr>
<td>Applying Relations to the Blocks</td>
<td>24-5</td>
</tr>
<tr>
<td>Converting Blocks into Parts</td>
<td>24-9</td>
</tr>
<tr>
<td>Extruding the Parts</td>
<td>24-10</td>
</tr>
<tr>
<td>Summary</td>
<td>24-14</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>25. Die Design</td>
<td>25-1</td>
</tr>
<tr>
<td>Scope of Die Design</td>
<td>25-1</td>
</tr>
<tr>
<td>Components of a Die Set</td>
<td>25-1</td>
</tr>
<tr>
<td>Pierce and Blank Die</td>
<td>25-2</td>
</tr>
<tr>
<td>Scrap Strip</td>
<td>25-5</td>
</tr>
<tr>
<td>Design of a Die Holder of a Die Set</td>
<td>25-5</td>
</tr>
<tr>
<td>Design of a Punch Holder of a Die Set</td>
<td>25-8</td>
</tr>
<tr>
<td>Design of a Guide Post</td>
<td>25-10</td>
</tr>
<tr>
<td>Design of Bushing</td>
<td>25-10</td>
</tr>
<tr>
<td>Design of a Die Block</td>
<td>25-11</td>
</tr>
<tr>
<td>Summary</td>
<td>25-12</td>
</tr>
<tr>
<td>Exercises</td>
<td>25-12</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>26. Aluminium Extrusion from Manufacturers’ Websites</td>
<td>26-1</td>
</tr>
<tr>
<td>Accessing Manufacturers’ AutoCAD 2D Aluminium Section</td>
<td>26-1</td>
</tr>
<tr>
<td>Creating a SolidWorks Sketch from AutoCAD 2D Section Profile</td>
<td>26-5</td>
</tr>
<tr>
<td>Creating SolidWorks Structural Elements Using AutoCAD 2D</td>
<td>26-8</td>
</tr>
<tr>
<td>Creating SolidWorks Structural Machine Frame from AutoCAD 2D</td>
<td>26-10</td>
</tr>
<tr>
<td>References</td>
<td>26-10</td>
</tr>
<tr>
<td>Exercises</td>
<td>26-11</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>27. Geneva Wheel Mechanism</td>
<td>27-1</td>
</tr>
<tr>
<td>Historical Background</td>
<td>27-1</td>
</tr>
<tr>
<td>Introduction</td>
<td>27-1</td>
</tr>
<tr>
<td>Advantages and Disadvantages</td>
<td>27-2</td>
</tr>
<tr>
<td>Geometry of the External Geneva Mechanism</td>
<td>27-3</td>
</tr>
<tr>
<td>Kinematics of the External Geneva Drive</td>
<td>27-4</td>
</tr>
<tr>
<td>SolidWorks Modeling and Simulation of Geneva Mechanism</td>
<td>27-5</td>
</tr>
<tr>
<td>Summary</td>
<td>27-11</td>
</tr>
<tr>
<td>References</td>
<td>27-11</td>
</tr>
<tr>
<td>Projects</td>
<td>27-12</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>28. Event-based Motion Analysis</td>
<td>28-1</td>
</tr>
<tr>
<td>Introduction</td>
<td>28-1</td>
</tr>
<tr>
<td>Event-based Motion View</td>
<td>28-1</td>
</tr>
<tr>
<td>Chapter</td>
<td>Title</td>
</tr>
<tr>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>28</td>
<td>Tasks</td>
</tr>
<tr>
<td>28</td>
<td>Triggers</td>
</tr>
<tr>
<td>28</td>
<td>Actions</td>
</tr>
<tr>
<td>28</td>
<td>Time</td>
</tr>
<tr>
<td>28</td>
<td>SolidWorks Event-based Motion Analysis Solution</td>
</tr>
<tr>
<td>28</td>
<td>Summary</td>
</tr>
<tr>
<td>28</td>
<td>Reference</td>
</tr>
<tr>
<td>29</td>
<td>Electrical Routing</td>
</tr>
<tr>
<td>29</td>
<td>Introduction</td>
</tr>
<tr>
<td>29</td>
<td>Creating the Housing</td>
</tr>
<tr>
<td>29</td>
<td>Creating the Electrical Harness</td>
</tr>
<tr>
<td>29</td>
<td>Creating the Route Using the Auto Route</td>
</tr>
<tr>
<td>29</td>
<td>Reference</td>
</tr>
<tr>
<td>30</td>
<td>Customized Internal and External Threads</td>
</tr>
<tr>
<td>30</td>
<td>Customized Internal Threads</td>
</tr>
<tr>
<td>30</td>
<td>Customized External Threads</td>
</tr>
<tr>
<td>30</td>
<td>Editing Features</td>
</tr>
<tr>
<td>30</td>
<td>Unified Thread Standard</td>
</tr>
<tr>
<td>30</td>
<td>External Threads</td>
</tr>
<tr>
<td>30</td>
<td>Internal Threads</td>
</tr>
<tr>
<td>30</td>
<td>References</td>
</tr>
<tr>
<td>30</td>
<td>Exercises</td>
</tr>
<tr>
<td>31</td>
<td>Sustainability Design for Parts</td>
</tr>
<tr>
<td>31</td>
<td>Introduction</td>
</tr>
<tr>
<td>31</td>
<td>Activate the Sustainability Application</td>
</tr>
<tr>
<td>31</td>
<td>Selecting a Material</td>
</tr>
<tr>
<td>31</td>
<td>Setting the Manufacturing and Use Options</td>
</tr>
<tr>
<td>31</td>
<td>Comparing Similar Materials</td>
</tr>
<tr>
<td>31</td>
<td>Summary</td>
</tr>
<tr>
<td>31</td>
<td>Exercises</td>
</tr>
<tr>
<td>32</td>
<td>Geometric Dimensioning &amp; Tolerancing</td>
</tr>
<tr>
<td>32</td>
<td>Introduction</td>
</tr>
<tr>
<td>32</td>
<td>Tolerance Study using SolidWorks</td>
</tr>
<tr>
<td>32</td>
<td>Geometric Dimensioning and Tolerancing (GD&amp;T)</td>
</tr>
<tr>
<td>32</td>
<td>Geometric Dimensioning and Tolerancing with SolidWorks</td>
</tr>
<tr>
<td>32</td>
<td>Using SolidWorks to Define Tolerances in Drawings</td>
</tr>
<tr>
<td>32</td>
<td>Summary</td>
</tr>
<tr>
<td>32</td>
<td>Exercises</td>
</tr>
<tr>
<td>33</td>
<td>Evaluating the Cost of Machined Parts</td>
</tr>
<tr>
<td>33</td>
<td>Introduction</td>
</tr>
<tr>
<td>33</td>
<td>Model for Automatic Manufacturing Cost Estimation</td>
</tr>
</tbody>
</table>
Automatic Manufacturing Cost Estimation 33-5
Changing Model Geometry 33-7
Adding Material to the Stock Body 33-8
Examining the CostingManager: Setup Folder 33-9
Examining the CostingManager: Mill Operations 33-9
Changing Material Costs 33-10
Changing the Stock Body 33-10
Changing the Quantity of Parts to Manufacture 33-11
Including a Discount 33-11
Adding Custom Operations 33-12
Comparing Material Costs 33-12
Adding Library Features 33-13
Creating a Report 33-13
Summary 33-13
Exercises 33-14
References 33-15

34. Finite Element Analysis Using SolidWorks 34-1
Introduction to COSMOS/SolidWorks Simulation 34-1
  What is SolidWorks Simulation? 34-1
  Product Development Cycle (PDC) 34-2
  What is Finite Element Analysis? 34-3
  How does Finite Element Analysis Work? 34-4
  Types of Engineering Analysis 34-6
  Principles of Finite Element Analysis 34-7
  SolidWorks Simulation Add-ins 34-9
  SolidWorks Simulation CommandManager 34-10
  SolidWorks Simulation Toolbars 34-12
  Starting a New Study in SolidWorks Simulation 34-13
  Basic SolidWorks Simulation Steps 34-14
Finite Element Analysis of a Sheet Metal Plate 34-15
Summary 34-21
Exercises 34-22