

# A Comprehensive Introduction to SolidWorks® 2013



Godfrey Onwubolu, Ph.D.

Visit the following websites to learn more about this book:



[amazon.com](https://www.amazon.com)

[Google books](https://books.google.com)

[BARNES & NOBLE](https://www.barnesandnoble.com)

# Table of Contents

## PART I: Introductory Engineering Design Principles with SolidWorks

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

Tutorial 2	3-33
Tutorial 3	3-34
Tutorial 4	3-36
Tutorial 5	3-37
Tutorial 6	3-38
Tutorial 7	3-39
Tutorial 8	3-42
Tutorial 9	3-43
Tutorial 10	3-46
Patterns—Advanced Methods	3-49
Sketch Driven Pattern	3-49
Curve Driven Pattern	3-51
Table Driven Pattern	3-56
Reference Planes	3-57
Tutorial 1 for Planes	3-58
Tutorial 2 for Planes	3-58
Tutorial 3 for Planes	3-59
Tutorial 4 for Planes	3-60
Creating Patterns	3-61
Summary	3-66
Exercises	3-67
<b>4. Part Modeling—CSWA Preparations</b>	<b>4-1</b>
Introduction	4-1
Tutorials	4-1
Tutorial 4-1a Widget	4-1
Tutorial 4-1b Widget	4-4
Tutorial 4-2	4-5
Tutorial 4-3a	4-7
Tutorial 4-3b	4-11
Tutorial 4-4	4-12
Tutorial 4-5a	4-12
Tutorial 4-5b	4-17
Tutorial 4-6a	4-18
Tutorial 4-6b	4-23
Tutorial 4-7a	4-24
Tutorial 4-7b	4-29
Summary	4-30
Exercises	4-30
<b>5. Advanced Part Modeling—CSWA Preparations</b>	<b>5-1</b>
Introduction	5-1
Advanced Part Modeling Tutorials	5-2
Tutorial 1: Block with hook	5-2
Tutorial 2: Bracket 1	5-6
Tutorial 3: Bracket 2	5-11

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
<b>6. Revolved, Swept, and Lofted Parts</b>	<b>6-1</b>
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
<b>7. Part Modeling with Equation Driven Curves</b>	<b>7-1</b>
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
<b>8. Assembly Modeling —CSWA Preparations</b>	<b>8-1</b>
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
<b>9. Part and Assembly Drawings—CSWA Preparations</b>	<b>9-1</b>
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

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
<b>14. Animation with Basic Motion</b>	<b>14-1</b>
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
<b>15. Animation with SolidWorks Motion</b>	<b>15-1</b>
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 <i>Add-In</i>	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
<b>16. Rendering</b>	<b>16-1</b>
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**

<b>17. Mold Design</b>	<b>17-1</b>
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

<b>18. Sheet Metal Parts-I</b>	<b>18-1</b>
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

<b>19. Sheet Metal Parts-II</b>	<b>19-1</b>
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

<b>20. Weldments</b>	<b>20-1</b>
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
<b>21. Routings in Piping and Tubing</b>	<b>21-1</b>
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
<b>22. Power Transmission Elements</b>	<b>22-1</b>
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

Belt and Pulley Assembly Modeling Using SolidWorks	22-21
Animation	22-27
Chain Drive: Chains and Sprockets	22-28
Problem Description	22-28
Chain and Sprocket Assembly Modeling Using SolidWorks	22-29
Animation	22-35
Bevel Gear Box Design	22-36
Problem Description	22-38
Assembly Modeling	22-39
Summary	22-43
Exercises	22-43
<b>23. Cam Design</b>	<b>23-1</b>
Introduction	23-1
Types of Cams	23-1
Types of Followers	23-1
Creating Cams Using the Traditional Method	23-1
Creating Cams in SolidWorks	23-2
Problem Definition	23-2
SolidWorks Toolbox Add-ins	23-2
To Access the SolidWorks Cams Tool	23-3
Cam-Circular Setup	23-3
Cam-Circular Motion	23-5
Cam-Circular Creation	23-7
Cam Model Modification	23-8
Creating a Hub	23-9
Creating a Hub Using the Cam-Circular Dialog	23-10
Creating a Hole for a Key Using the Hole Wizard	23-11
Cam Shaft Assembly	23-12
Spring	23-12
Cam Follower Bracket	23-13
Cam Bracket	23-14
Roller	23-15
Cam Shaft	23-15
Handle	23-16
Pin	23-16
Assembly of Cam Shaft Components	23-16
Exercises	23-17
<b>24. Mechanism Design Using Blocks</b>	<b>24-1</b>
Introduction	24-1
Blocks Toolbar	24-1
Problem Description	24-2
Creating Sketches of a Mechanism	24-3

Saving the Sketches as Different Block Files	24-4
Inserting the Block into the Layout Environment	24-4
Applying Relations to the Blocks	24-5
Converting Blocks into Parts	24-9
Extruding the Parts	24-10
Summary	24-14
<b>25. Die Design</b>	<b>25-1</b>
Scope of Die Design	25-1
Components of a Die Set	25-1
Pierce and Blank Die	25-2
Scrap Strip	25-5
Design of a Die Holder of a Die Set	25-5
Design of a Punch Holder of a Die Set	25-8
Design of a Guide Post	25-10
Design of Bushing	25-10
Design of a Die Block	25-11
Summary	25-12
Exercises	25-12
<b>26. Aluminium Extrusion from Manufacturers' Websites</b>	<b>26-1</b>
Accessing Manufacturers' AutoCAD 2D Aluminium Section	26-1
Creating a SolidWorks Sketch from AutoCAD 2D Section Profile	26-5
Creating SolidWorks Structural Elements Using AutoCAD 2D	26-8
Creating SolidWorks Structural Machine Frame from AutoCAD 2D	26-10
References	26-10
Exercises	26-11
<b>27. Geneva Wheel Mechanism</b>	<b>27-1</b>
Historical Background	27-1
Introduction	27-1
Principles of Operation of the Geneva Drive	27-2
Advantages and Disadvantages	27-2
Geometry of the External Geneva Mechanism	27-3
Kinematics of the External Geneva Drive	27-4
SolidWorks Modeling and Simulation of Geneva Mechanism	27-5
Summary	27-11
References	27-11
Projects	27-12
<b>28. Event-based Motion Analysis</b>	<b>28-1</b>
Introduction	28-1
Event-based Motion View	28-1

Tasks	28-1
Triggers	28-2
Actions	28-2
Time	28-4
SolidWorks Event-based Motion Analysis Solution	28-4
Summary	28-11
Reference	28-11
<b>29. Electrical Routing</b>	<b>29-1</b>
Introduction	29-1
Creating the Housing	29-1
Creating the Electrical Harness	29-5
Creating the Route Using the Auto Route	29-7
Reference	29-11
<b>30. Customized Internal and External Threads</b>	<b>30-1</b>
Customized Internal Threads	30-1
Customized External Threads	30-2
Editing Features	30-11
Unified Thread Standard	30-13
External Threads	30-16
Internal Threads	30-17
References	30-19
Exercises	30-19
<b>31. Sustainability Design for Parts</b>	<b>31-1</b>
Introduction	31-1
Activate the Sustainability Application	31-2
Selecting a Material	31-3
Setting the Manufacturing and Use Options	31-3
Comparing Similar Materials	31-4
Summary	31-8
Exercises	31-9
<b>32. Geometric Dimensioning &amp; Tolerancing</b>	<b>32-1</b>
Introduction	32-1
Tolerance Study using SolidWorks	32-2
Geometric Dimensioning and Tolerancing (GD&T)	32-6
Geometric Dimensioning and Tolerancing with SolidWorks	32-12
Using SolidWorks to Define Tolerances in Drawings	32-20
Summary	32-29
Exercises	32-29
<b>33. Evaluating the Cost of Machined Parts</b>	<b>33-1</b>
Introduction	33-1
Model for Automatic Manufacturing Cost Estimation	33-3

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
<b>34. Finite Element Analysis Using SolidWorks</b>	<b>34-1</b>
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