

## TABLE OF CONTENTS

<b>Introduction</b>	<b>I-1</b>
About the Author	I-3
Acknowledgements	I-4
Contact the Author	I-4
Note to Instructors	I-4
Trademarks, Disclaimers, and Copyrighted Material	I-4
References	I-5
Table of Contents	I-6
Overview of Chapters	I-11
Chapter 1: Overview of SolidWorks and the User Interface	I-11
Chapter 2: 2D Sketching, Features and Parts	I-11
Chapter 3: Assembly Modeling - Bottom up	I-12
Chapter 4: Design Modifications	I-12
Chapter 5: Drawing Fundamentals	I-13
Book Layout	I-14
Windows Terminology in SolidWorks	I-15
<b>Chapter 1 - Overview of SolidWorks and the User Interface</b>	<b>1-1</b>
Chapter Overview	1-1
Chapter Objective	1-3
Start a SolidWorks Session	1-3
SolidWorks UI and CommandManager	1-4
Menu bar toolbar	1-4
Menu bar menu	1-5
Drop-down menu	1-5
Create a new Part	1-6
Novice Mode	1-7
Advanced Mode	1-7
Graphic Interface	1-8
Open a Part	1-9
FeatureManager	1-10
Rollback Bar	1-10
Heads-up View toolbar	1-12
Zoom to Fit	1-12
Zoom to Area	1-12
Zoom in	1-12
Rotate	1-12
Standard Views	1-13
SolidWorks Help	1-13
SolidWorks Tutorials	1-14
Additional User Interface Tools	1-14
Right-click Context toolbar	1-15
Consolidated toolbar	1-15
System feedback icons	1-15
Confirmation Corner	1-16
Heads-up View toolbar	1-16
CommandManager	1-19

Part (default tab)	1-19
Drawing (default tab)	1-20
Assembly (default tab)	1-21
Float/Dock	1-22
FeatureManager Design Tree	1-23
Fly-out FeatureManager	1-25
Task Pane	1-26
SolidWorks Forum	1-26
SolidWorks Resources	1-27
Design Library	1-27
File Explorer	1-28
Search	1-28
View Palette	1-28
Appearances, Scenes and Decals	1-29
Custom Properties	1-29
Document Recovery	1-29
Motion Study tab	1-30
Mouse Movements	1-31
Summary	1-31
<b>Chapter 2 - 2D Sketching, Features and Parts</b>	<b>2-1</b>
Chapter Overview	2-1
Chapter Objective	2-3
Start a SolidWorks Session	2-3
Create a new Part Document	2-4
Set Document Properties	2-5
Drafting Standard	2-5
Units	2-5
Precision	2-5
2D Sketching - Identify the Correct Sketch Plane	2-6
Sketch States	2-6
Under Defined	2-6
Fully Defined	2-6
Over Defined	2-6
Wheel Part - Base Sketch	2-7
Origin	2-7
Geometric Relations	2-8
Sketch Dimensions	2-9
Wheel Part - Sketch1: Circle, Geometric relations and Dimensions	2-7
Wheel Part - First Feature (Extruded Base)	2-9
Design Intent	2-10
Edit Base Sketch	2-12
Edit Sketch Plane	2-12
Wheel Part - Sketch2: Centerline, Line and Mirror Entities	2-13
Wheel Part - Second Feature (Revolved Boss)	2-13
Wheel Part - Sketch3: Centerpoint Straight Slot, Circle and Construction geometry	2-19
Wheel Part - Third Feature (Extruded Cut)	2-19
Wheel Part - Fourth Feature (Circular Pattern)	2-23
Wheel Part - Fifth Feature (Hole Wizard)	2-24
Wheel Part - Sixth Feature (Fillet)	2-25
Wheel Part - Add Material (6061 Alloy)	2-26

Wheel Part - View Mass Properties	2-28
Wheel Part - Modify the Number of Instances in the Circular Pattern	2-29
Wheel Part - View the new Mass Properties	2-29
Wheel Part - Return to the original Number of Instances	2-29
Wheel Part - Apply Appearance	2-30
Summary	2-30
Exercises	2-32
<b>Chapter 3 - Assembly Modeling</b>	<b>3-1</b>
Chapter Overview	3-1
Chapter Objective	3-3
Start a SolidWorks Session	3-3
Create a new Assembly Document	3-4
Set Document Properties	3-5
Drafting Standard	3-5
Units	3-5
Precision	3-5
Assembly Modeling Approach	3-6
Linear Motion and Rotational Motion	3-6
Create the Fly Wheel Assembly	3-7
Insert the First Component - Bracket (Fixed to the origin)	3-7
Mate Types	3-9
Standard Mates	3-9
Advanced Mates	3-10
Mechanical Mates	3-11
Quick Mates	3-11
Insert the Second Component - Bushing	3-12
Insert a Concentric and Coincident Mate	3-13
Insert the Third Component - Axle	3-14
Insert a Concentric and Distance Mate	3-15
Insert the Fourth Component - Wheel	3-15
Insert a Concentric and Distance Mate	3-17
Insert the Fifth Component - Collar	3-18
Insert a Concentric and Coincident Mate	3-19
Insert the Sixth Component - 2 MM Set Screw	3-20
Insert a Concentric, Tangent and Coincident Mate	3-22
Create an Exploded View of the Fly Wheel Assembly	3-23
Create the Stirling Engine Assembly	3-26
Hide Component	3-27
Insert the Fly Wheel Assembly	3-28
Rotate Component	3-28
Insert a Concentric Mate	3-29
Insert a second Concentric Mate	3-30
Apply the Measure tool	3-30
Modify the Axle Component Length	3-31
Make the Fly Wheel Assembly Flexible	3-32
Insert a Coincident Mate	3-32
Show Components	3-33
Pack and Go the Assembly	3-34
Summary	3-36
Exercises	3-37

<b>Chapter 4 - Design Modifications</b>	<b>4-1</b>
Chapter Overview	4-1
Chapter Objective	4-3
Start a SolidWorks Session	4-3
Open an Existing Assembly	4-4
Stirling Engine Modified Assembly	4-4
Verify Collision between Components	4-5
Apply the Move Component tool	4-5
Set Collision Detection	4-5
Apply the Interference Detection tool	4-6
Calculate the Interference -Note there is interference	4-8
Modify the Assembly (Connection Rod Mate)	4-9
Verify the Modification - Measure tool	4-10
Apply the Interference Detection tool - check Solution	4-11
Calculate the Interference - No interference	4-12
Locate the Center of Mass	4-13
Display the Center of Mass	4-14
Create a new Coordinate System	4-15
Display the Mass Properties - New Coordinate System	4-16
Apply Assembly Visualization	4-16
Sort Assembly Components by Mass	4-17
Create a Motion Study	4-19
Create and Save an AVI file	4-19
Summary	4-21
Exercises	4-22
<b>Chapter 5 - Drawing Fundamentals</b>	<b>5-1</b>
Chapter Overview	5-1
Chapter Objective	5-3
Start a SolidWorks Session	5-3
Create a new Drawing Document	5-4
Set Sheet Properties	5-5
Set Document Properties	5-6
Drafting Standard	5-6
Units	5-6
Precision	5-6
Title Block	5-6
Create the Fly Wheel Assembly Drawing	5-8
View Palette	5-8
Isometric Exploded View	5-8
Sheet Scale	5-8
Modify Display Mode	5-9
Auto Balloons	5-9
Bill of Materials	5-11
Set Custom Properties	5-13
Create the Bushing Part Drawing	5-14
View Palette	5-15
Front, Top, Right and Isometric View	5-16
Import Dimensions (Model Items tool)	5-16
Move Dimensions	5-17
Hide Dimensions	5-18

Insert Dimension Text	5-19
Modify Display Mode	5-19
Create Dimension Extension Line Gaps	5-20
Insert Dimensions (Smart Dimension tool)	5-20
Insert Annotation	5-21
Hide a View	5-22
Modify the Sheet Scale	5-23
Summary	5-24
Exercises	5-25
<b>Appendix</b>	
ECO Form	A-1
Types of Decimal Dimensions (ASME Y14.5)	A-2
SolidWorks Keyboard Shortcuts	A-3
Windows Shortcuts	A-3
Helpful On-Line information	A-5
SolidWorks Document types	A-6
<b>Index</b>	<b>I-1</b>

View the provided videos for each section of the book to enhance the user experience.

- SolidWorks Interface.
- 2D Sketching, Sketch Planes and Sketch tools.
- 3D Features and Design Intent.
- Creating an Assembly.
- Fundamentals in Drawings Part 1 & Part2.

