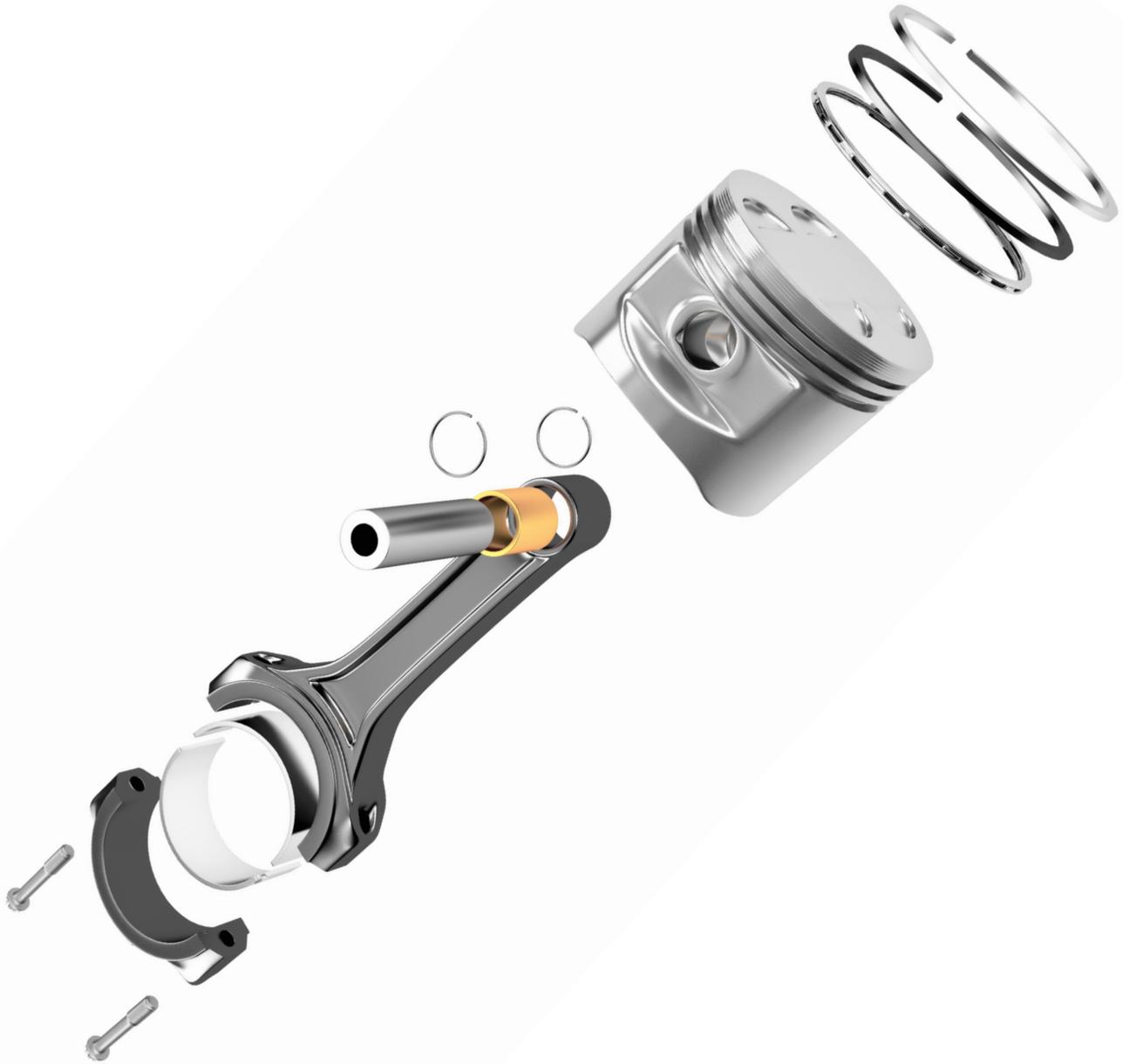


SOLIDWORKS® 2021 Quick Start



David C. Planchard, CSWP,
SOLIDWORKS Accredited Educator

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

Introduction	I-1
About the Author	I-3
Acknowledgements	I-4
Contact the Author	I-5
Note to Instructors	I-5
Trademarks, Disclaimer, and Copyrighted Material	I-5
References	I-6
Table of Contents	I-7
Overview of Chapters	I-15
Chapter 1: Overview of SOLIDWORKS and the User Interface	I-15
Chapter 2: 2D Sketching, Features and Parts	I-15
Chapter 3: Assembly Modeling - Bottom up Method	I-16
Chapter 4: Design Modifications	I-16
Chapter 5: Drawing and Dimensioning Fundamentals	I-17
Chapter 6: Certified Associate - Mechanical Design (CSWA) exam	I-17
Chapter 7: Additive Manufacturing - 3D Printing	I-18
Book Layout	I-20
Windows Terminology in SOLIDWORKS	I-21
Chapter 1 - Overview of SOLIDWORKS and the User Interface	I-1
Chapter Objective	I-3
What is SOLIDWORKS?	I-3
Basic concepts in SOLIDWORKS	I-3
Start a SOLIDWORKS Session	I-4
<i>Tutorial: Start a SOLIDWORKS Session</i>	I-4
Welcome dialog box	I-4
Home Tab	I-5
Recent Tab	I-5
Learn Tab	I-5
Alerts Tab	I-6
SOLIDWORKS User Interface (UI) and CommandManager	I-7
Menu Bar toolbar	I-7
Menu Bar menu	I-8
Drop-down menu	I-8
Create a New Part Document	I-9
Novice Mode	I-10
Advanced Mode	I-10
Graphic Window (Default)	I-11
View Default Sketch Planes	I-12
Open a Part	I-12
Part FeatureManager	I-13
FeatureManager Rollback Bar	I-13
Heads-up View toolbar	I-15
Dynamic Annotation Views	I-15
Zoom to Fit	I-15
Zoom to Area	I-15
Window-Select	I-15

Rotate	1-15
Front View	1-16
Right View	1-16
Top View	1-16
Trimetric view	1-16
SOLIDWORKS Help	1-16
SOLIDWORKS Tutorials	1-17
User Interface Tools	1-17
Right-click	1-18
Consolidated toolbar	1-18
System feedback icons	1-18
Confirmation Corner	1-19
Heads-up View toolbar	1-19
CommandManager (Default Part tab)	1-22
CommandManager (Default Drawing tab)	1-23
CommandManager (Default Assembly tab)	1-24
CommandManager (Float/Fit)	1-25
Collapse the CommandManager	1-25
FeatureManager Design Tree	1-26
FeatureManager design tree tab	1-26
PropertyManager tab	1-26
Configuration Manager tab	1-26
DimXpertManager tab	1-26
DisplayManager tab	1-26
CAM tab	1-26
Hide/Show tab	1-26
Sensors tool	1-26
Tags	1-27
Split	1-27
Fly-out FeatureManager	1-28
Task Pane	1-29
SOLIDWORKS Resources	1-29
Design Library	1-30
File Explorer	1-30
Search	1-31
View Palette	1-31
Appearances, Scenes and Decals	1-31
Custom Properties	1-32
SOLIDWORKS Forum	1-32
3DEXPERIENCE MARKETPLACE	1-32
Motion Study tab	1-32
3D Views tab	1-34
Dynamic Reference Visualization	1-34
Mouse Movements	1-35
Single-Click	1-35
Double-Click	1-35
Right-Click	1-35
Scroll Wheel	1-35
Summary	1-36

Chapter 2 - 2D Sketching, Features and Parts	2-1
Chapter Overview	2-1
Chapter Objective	2-3
Start a SOLIDWORKS Session	2-3
Create a new Part Document	2-3
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	2-9
Sketch Dimensions	2-9
Wheel Part - Sketch1: Circle, Geometric relations and Dimensions	2-9
Wheel Part - First Feature (Extruded Base)	2-10
Design Intent	2-11
Edit Base Sketch	2-13
Edit Sketch Plane	2-13
Display Modes, View Modes and View tools	2-14
Wheel Part - Sketch2: Centerline, Line and Mirror Entities	2-14
Wheel Part - Second Feature (Revolved Boss)	2-19
Wheel Part - Sketch3: Centerpoint Straight Slot, Circle and Construction geometry	2-20
Wheel Part - Third Feature (Extruded Cut)	2-22
Wheel Part - Fourth Feature (Circular Pattern)	2-24
Wheel Part - Fifth Feature (Hole Wizard)	2-25
Wheel Part - Sixth Feature (Fillet)	2-26
Wheel Part - Add Material (6061 Alloy)	2-28
Wheel Part - View Mass Properties	2-29
Wheel Part - Modify the Number of Instances in the Circular Pattern	2-30
Wheel Part - View the new Mass Properties	2-30
Wheel Part - Return to the original Number of Instances	2-30
Wheel Part - Apply Appearance	2-31
Summary	2-31
Exercises	2-35
Chapter 3 - Assembly Modeling - Bottom up method	3-1
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 Mate	3-12
Insert the Second Component - Bushing	3-12
Insert a Concentric and Coincident Mate	3-12
Insert the Third Component - Axle	3-14
Insert a Concentric and Distance Mate	3-14
Insert the Fourth Component - Wheel	3-15
Insert a Concentric and Distance Mate	3-16
Insert the Fifth Component - Collar	3-18
Insert a Concentric and Coincident Mate	3-18
Insert the Sixth Component - 2 MM Set Screw	3-20
Insert a Concentric, Tangent and Coincident Mate	3-20
Create an Exploded View of the Fly Wheel Assembly	3-24
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	3-31
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	3-35
Exercises	3-37
Chapter 4 - Design Modifications	4-1
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	4-7
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	4-18
Sort Assembly Components by Mass	4-18
Create a Motion Study	4-20
Create and Save an AVI file	4-20
Summary	4-22
Exercises	4-25
Chapter 5 - Drawing and Dimensioning Fundamentals	5-1
Chapter Overview	5-1
Chapter Objective	5-3
Start a SOLIDWORKS Session	5-3
New Drawing Document	5-4
Sheet Properties	5-5
Document Properties	5-6
Drafting Standard	5-6
Units	5-6
Precision	5-6
Title Block	5-7
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	5-14
Title Block	5-16
Bushing Part Drawing	5-19
View Palette	5-20
Front, Top, Right and Isometric View	5-21
Import Dimensions (Model Items tool)	5-22
Move Dimensions	5-23
Hide Dimensions	5-23
Insert Dimension Text	5-24
Modify Display Mode	5-24
Dimension Extension Line Gaps	5-25
Dimensions (Smart Dimension tool)	5-25
Annotation	5-26
Hide a View	5-27
Modify the Sheet Scale	5-28
Summary	5-29
Exercises	5-31
Chapter 6 - Introduction to the Certified Associated - Mechanical Design Exam	6-1
Chapter Objective	6-3
Introduction	6-3
Part 1 of the Exam	6-4
Basic Part Creation and Modification, Intermediate Part Creation and Modification	6-4
Assembly Creation and Modification	6-6
Part 2 of the Exam	6-7

Introduction and Drafting Competencies	6-7
Advanced Part Creating and Modification	6-8
Assembly Creation and Modification	6-10
Intended Audience	6-11
During the Exam	6-12
Drafting Competencies	6-13
Example 1	6-13
Example 2	6-13
Example 3	6-14
Example 4	6-14
Example 5	6-14
Example 6	6-14
Basic Part Creation and Modification, Intermediate Part Creation and Modification	6-15
Example 1	6-16
Example 2	6-17
Example 3	6-18
Example 4	6-19
Example 5	6-20
Example 6	6-21
Example 6A	6-23
Example 6B	6-23
Advanced Part Creation and Modification	6-24
Example 1	6-24
Example 2	6-26
Example 3	6-27
Example 4	6-28
Example 5	6-29
Example 6	6-31
Example 6A	6-32
Assembly Creation and Modification	6-33
Example 1	6-34
Chapter 7 - Additive Manufacturing	7-1
Chapter Objective	7-3
Additive vs. Subtractive Manufacturing	7-4
3D Printer Technology	7-5
Stages of 3D Printing	7-5
Fused Filament Fabrication (FFF)	7-6
StereoLithography (SLA)	7-9
Selective Laser Sintering (SLS)	7-11
Select the Correct Filament Material for FFF	7-12
PLA (Polylactic Acid)	7-13
Flex/Soft PLA	7-13
PLA Storage	7-14
PLA Part Accuracy	7-14
ABS (Acrylonitrile-Butadiene-Styrene)	7-14
ABS Storage	7-15
ABS Part Accuracy	7-15
Nylon	7-16
Nylon 618	7-16
Nylon 645	7-16

Nylon Storage	7-17
Nylon Accuracy	7-17
PVA (Polyvinyl Alcohol)	7-17
STereoLithography (*.stl) file	7-18
Save an STL (*.stl) file	7-18
Additive Manufacturing (*.amf) file	7-19
Save an Additive Manufacturing (*.amf) file	7-19
3D Manufacturing Format (*.3mf) file	7-20
Save a 3D Manufacturing Format (*.3mf) file	7-20
What is a Slicer?	7-21
How does a Slicer Work?	7-21
Slicer Parameters	7-21
Layer Height	7-21
Shell (Wall) Thickness	7-22
Infill Density/Overlap	7-22
Infill Patterns	7-22
Print Speed	7-23
Support Types	7-23
Touching Buildplate	7-23
Everywhere	7-24
Bed Platform Adhesion	7-24
Raft	7-24
Skirt	7-24
Brim	7-24
Part Orientation	7-25
Example 1	7-25
Example 2	7-26
Optimize Print Direction	7-26
Thin Region	7-26
Area of Overhang	7-26
Amount of needed Support	7-26
Remove Model from the Build Plate	7-28
Non-heated Build Plate	7-28
Heated Build Plate	7-28
Know the Printer's Limitations	7-29
Tolerance for Interlocking Parts	7-29
General Printing Tips	7-29
Reduce Infill/Overlap	7-29
Control Build Area Temperature	7-30
Add Pads	7-31
Safe Zone Rule	7-31
First Layer Not Sticking	7-31
Level Build Platform	7-32
Minimize Internal Support	7-32
Design a Water Tight Mesh	7-32
Clearance	7-32
In General	7-33
Print directly from SOLIDWORKS	7-34
Add-in	7-34

SOLIDWORKS Additive Manufacturing Certification (CSWA-AM) Summary	7-35 7-36
Appendix	
SOLIDWORKS Keyboard Shortcuts	A-1
Modeling - Best Practices	A-3
Helpful On-Line Information	A-5
SOLIDWORKS Document Types	A-6
Glossary	G-1
Index	I-1

Download all model files from the SDC Publications website (<https://www.sdcpublications.com/downloads/978-1-63057-381-2>).

The book provides information on creating and storing special Part, Assembly and Drawing templates in the MY-TEMPLATES folder. The MY-TEMPLATES folder is added to the New SOLIDWORKS Document dialog box. Talk to your IT department *before you set* any new locations on a network system. The procedure in the book is designed for your personal computer.



If you do not create the MY-TEMPLATE tab or the special part, drawing, or assembly templates, use the standard SOLIDWORKS default template and apply all of the needed document and custom properties.

