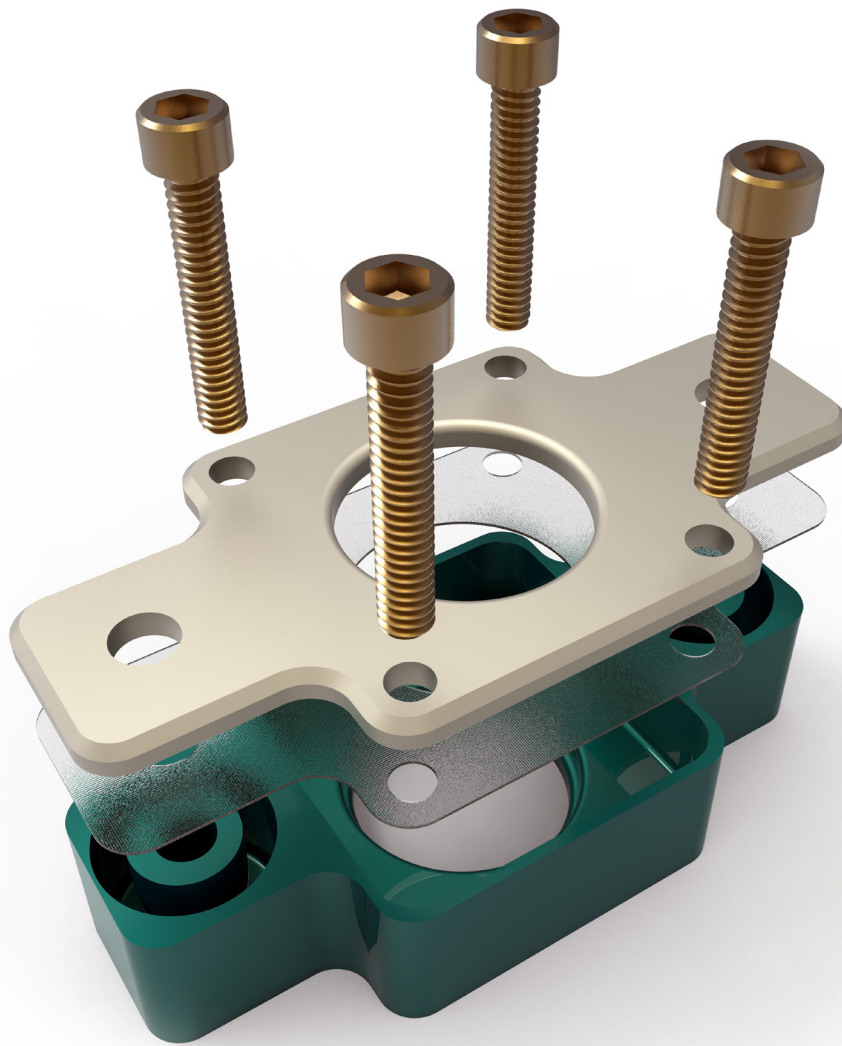


SOLIDWORKS® 2019 Quick Start



David C. Planchard, CSWP,
SOLIDWORKS Accredited Educator



Better Textbooks. Lower Prices.
www.SDCpublications.com



ACCESS CODE
UNIQUE CODE INSIDE

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) program	I-17
Chapter 7: Additive Manufacturing - 3D Printing	I-18
Book Layout	I-19
Windows Terminology in SOLIDWORKS	I-21
Chapter 1 - Overview of SOLIDWORKS and the User Interface	1-1
Project Objective	1-3
What is SOLIDWORKS?	1-3
Basic concepts in SOLIDWORKS	1-3
Start a SOLIDWORKS Session	1-4
<i>Activity: Start a SOLIDWORKS Session</i>	1-4
Welcome dialog box	1-4
Home Tab	1-5
Recent Tab	1-5
Learn Tab	1-5
Alerts Tab	1-6
SOLIDWORKS User Interface (UI) and CommandManager	1-7
Menu Bar toolbar	1-8
Menu Bar menu	1-8
Drop-down menu	1-9
Create a New Part Document	1-9
Novice Mode	1-10
Advanced Mode	1-10
Graphic Window (Default)	1-11
View Default Sketch Planes	1-12
Open a Part	1-12
Part FeatureManager	1-13
FeatureManager Rollback Bar	1-13
Heads-up View toolbar	1-15
Dynamic Annotation Views	1-15
Zoom to Fit	1-15
Zoom to Area	1-15
Window-Select	1-15

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
Selection Enhancements	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	1-32
Custom Properties	1-32
SOLIDWORKS Forum	1-32
User Interface for Scaling High Resolution Screens	1-32
Motion Study tab	1-33
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	2-8
Sketch Dimensions	2-9
Wheel Part - Sketch1: Circle, Geometric relations and Dimensions	2-9
Wheel Part - First Feature (Extruded Base)	2-10
Design Intent	2-10
Edit Base Sketch	2-12
Edit Sketch Plane	2-12
Display Modes, View Modes and View tools	2-13
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	2-33
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	3-13
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	3-24
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	4-16
Sort Assembly Components by Mass	4-17
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	5-20
Import Dimensions (Model Items tool)	5-22
Move Dimensions	5-23
Hide Dimensions	5-23
Insert Dimension Text	5-24
Modify Display Mode	5-25
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
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

Exclusive Online Bonus Content

Instructions for download on inside front cover of book

Chapter 6 – Introduction to the Certified SOLIDWORKS Associate (CSWA) 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	6-4
Part 2 of the Exam	6-7
Introduction and Drafting Competencies	6-7
Advanced Part Creation and Modification	6-7
Assembly Creation and Modification	6-9
Intended Audience	6-10
During the Exam	6-11
Drafting Competencies	6-12
Example 1	6-12
Example 2	6-12
Example 3	6-13
Example 4	6-13
Example 5	6-13
Example 6	6-13
Basic Part Creation and Modification, Intermediate Part Creation and Modification	6-14
Example 1	6-15
Example 2	6-16
Example 3	6-17
Example 4	6-18
Example 5	6-19
Example 6	6-20
Example 6A	6-22
Example 6B	6-22
Advanced Part Creation and Modification	6-23
Example 1	6-23
Example 2	6-25
Example 3	6-26
Example 4	6-27
Example 5	6-28
Example 6	6-30
Example 6A	6-31
Assembly Creation and Modification	6-32
Example 1	6-33
Chapter 7 - Additive Manufacturing	7-1
Chapter Objective	7-3
Additive vs. Subtractive Manufacturing	7-4
3D Printer Technology	7-5
Fused Filament Fabrication (FFF)	7-5
StereoLithography (SLA)	7-8
Selective Laser Sintering (SLS)	7-10
Select the Correct Filament Material for FFF	7-11

PLA (Polylactic Acid)	7-12
Flex/Soft PLA	7-12
PLA Storage	7-13
PLA Part Accuracy	7-13
ABS (Acrylonitrile-Butadiene-Styrene)	7-13
ABS Storage	7-14
ABS Part Accuracy	7-14
Nylon	7-15
Nylon 618	7-15
Nylon 645	7-15
Nylon Storage	7-16
Nylon Part Accuracy	7-16
PVA (Polyvinyl Alcohol)	7-16
STereoLithography (*.stl) file	7-17
Save an STL (*.stl) file	7-17
Additive Manufacturing (*.amf) file	7-18
Save an Additive Manufacturing (*.amf) file	7-18
3D Manufacturing Format (*.3mf) file	7-19
Save a 3D Manufacturing Format (*.3mf) file	7-19
What is a Slicer?	7-20
How does a Slicer Work?	7-20
Slicer Parameters	7-20
Layer Height	7-20
Shell (Wall) Thickness	7-21
Infill Density/Overlap	7-21
Infill Patterns	7-21
Print Speed	7-22
Support Types	7-22
Touching Buildplate	7-22
Everywhere	7-23
Bed Platform Adhesion	7-23
Raft	7-23
Skirt	7-23
Brim	7-23
Part Orientation	7-24
Example 1	7-24
Example 2	7-25
Optimize Print Direction	7-25
Thin Region	7-25
Area of Overhang	7-25
Amount of needed Support	7-25
Remove Model from the Build Plate	7-27
Non-heated Build Plate	7-27
Heated Build Plate	7-27
Know the Printer's Limitations	7-28
Tolerance for Interlocking Parts	7-28
General Printing Tips	7-28
Reduce Infill/Overlap	7-28
Control Build Area Temperature	7-29
Add Pads	7-30
Safe Zone Rule	7-30

First Layer Not Sticking	7-30
Level Build Platform	7-31
Minimize Internal Support	7-31
Design a Water Tight Mesh	7-31
Clearance	7-31
In General	7-32
Print Directly from SOLIDWORKS	7-33
Add-in	7-33
SOLIDWORKS Additive Manufacturing Certification (CSWA-AM)	7-34
Summary	7-35