
Solidworks Instruction Manual

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Engineering Graphics
with SOLIDWORKS 2017
and Video Instruction
SDC Publications
Engineering Design
with SOLIDWORKS 2018
and video instruction

is written to assist students, designers, engineers and professionals. The book provides a solid foundation in SOLIDWORKS by utilizing projects with step-by-step instructions for the beginner to intermediate SOLIDWORKS user featuring machined, plastic and sheet metal components.

Desired outcomes and usage competencies are listed for each project. The book is divided into five sections with 11 projects. Project 1 - Project 6: Explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. Additional techniques include the edit and reuse of features, parts, and assemblies through symmetry, patterns, configurations, SOLIDWORKS 3D ContentCentral and the SOLIDWORKS Toolbox. Project 7: Understand

Top-Down assembly modeling and Sheet Metal parts. Develop components In-Context with InPlace Mates, along with the ability to import parts using the Top-Down assembly method. Convert a solid part into a Sheet Metal part and insert and apply various Sheet Metal features. Project 8 - Project 9: Recognize SOLIDWORKS Simulation and Intelligent Modeling techniques. Understand a general overview of SOLIDWORKS Simulation and the type of questions that are on the SOLIDWORKS Simulation Associate - Finite Element Analysis (CSWSA-FEA) exam. Apply design intent and intelligent modeling techniques in a sketch, feature, part, plane, assembly and drawing. Project 10: Comprehend the

differences between additive and subtractive manufacturing.

Understand 3D printer terminology along with a working knowledge of preparing, saving, and printing CAD models on a low cost printer.

Project 11: Review the Certified Associate - Mechanical Design (CSWA) program.

Understand the curriculum and categories of the CSWA exam and the required model knowledge needed to successfully take the exam. The author developed the industry scenarios by combining his own industry experience with the knowledge of engineers, department managers, vendors and manufacturers. These professionals are directly involved with SOLIDWORKS every day. Their responsibilities

go far beyond the creation of just a 3D model.

[SOLIDWORKS 2016 Tutorial with Video Instruction](#) SDC Publications

A comprehensive resource packed with information for both beginners and advanced users

SolidWorks is the leading 3D solid modeling software used in computer-aided design. It's powerful but not simple. This complete guide introduces beginners to the software but then goes far beyond, covering numerous details that advanced users have requested. Beginners will learn not only how the software works but why, while more experienced users will learn all about search criteria, Pack-and-Go, other file management concepts, and much more. A valuable companion website contains before and after real-world parts and assemblies along with many example files used in the text. Additionally, the text of the book is augmented by video tutorials with author voice-over which can be found on the website. SolidWorks is the leading

3D CAD program, and previous editions of this book have sold more than 33,000 copies. Covers necessary information to give beginners a solid foundation in the software, including part and assembly modeling and 2D drawing techniques. Addresses a wide range of advanced topics not treated in other books, including best practices, search criteria, Pack-and-Go, and other file management concepts. Includes tutorials on both beginning and advanced topics, with videos; sample part, assembly, and drawing files; and before-and-after example files available on the companion website. SolidWorks 2013 Bible is the ultimate resource on SolidWorks 2013, the book beginners can start with and advanced users will want to keep close at hand.

SOLIDWORKS Exercises - Learn by Practicing (3rd Edition)
SDC Publications
SOLIDWORKS 2019

Tutorial is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The text

provides a step-by-step, project based learning approach. It also contains information and examples on the five categories in the CSWA exam. The book is divided into four sections. Chapters 1 - 5 explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. In chapter 6 you will create the final robot assembly. The physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and components for the final robot assembly are provided. Chapters 7 - 10 prepare you for the Certified

Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs from subtractive manufacturing, and its features. You will also learn the terms and technology used in low cost 3D printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, apply proper design intent, design tables and configurations. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each chapter. Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

SOLIDWORKS 2020 Learn by Doing - Part 1 CRC Press

A comprehensive introduction to SOLIDWORKS using tutorial style, step-by-step instructions Designed for beginning or intermediate SOLIDWORKS users Learn to create parts and assemblies using machined, plastic and sheet metal components Also covers Simulation, Sustainability, and Intelligent Modeling techniques Includes bonus chapters on the CSWA exam and 3D printing

Engineering Design with SOLIDWORKS 2022 is written to assist students, designers, engineers and professionals. The book provides a solid foundation in SOLIDWORKS by utilizing

projects with step-by-step instructions for the beginner to intermediate SOLIDWORKS user featuring machined, plastic and sheet metal components. Desired outcomes and usage competencies are listed for each project. The book is divided into five sections with 11 projects. Project 1 - Project 6: Explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. Additional techniques include the edit and reuse of features, parts, and assemblies through symmetry, patterns, configurations, SOLIDWORKS 3D ContentCentral and the SOLIDWORKS Toolbox. Project 7: Understand Top-Down assembly modeling and Sheet Metal parts. Develop components In-Context with InPlace Mates, along with the ability to import parts using the Top-Down assembly method. Convert a solid

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knowledge of engineers, department managers, vendors and manufacturers. These professionals are directly involved with SOLIDWORKS every day. Their responsibilities go far beyond the creation of just a 3D model.

Project Based
SOLIDWORKS 2021
SDC Publications
Engineering Graphics with SolidWorks 2012 and Video Instruction DVD is written to assist technical school, two year college, four year university instructor/student or industry professional that is a beginner or intermediate SolidWorks user. The book combines the fundamentals of engineering graphics and dimensioning practices with a step-by-step project based

approach to learning SolidWorks with the enclosed 1.5 hour Video Instruction DVD. Learn by doing, not just by reading! The book is divided into two parts: Engineering Graphics and SolidWorks 3D CAD software. In Chapter 1 through Chapter 3, you explore the history of engineering graphics, manual sketching techniques, orthographic projection, isometric projection, multi-view drawings, dimensioning practices and the history of CAD leading to the development of SolidWorks. In Chapter 4 through Chapter 8, you apply engineering graphics fundamentals and learn the

SolidWorks User Interface, Document and System properties, simple parts, simple and complex assemblies, design tables, configurations, multi-sheet, multi-view drawings, Bill of Materials, Revision tables, basic and advanced features. Follow the step-by-step instructions in over 70 activities to develop eight parts, four sub-assemblies, three drawings, and six document templates. Formulate the skills to create and modify solid features to model a 3D FLASHLIGHT assembly. Chapter 9 provides a bonus section on the Certified SolidWorks Associate CSWA program with

sample exam questions and initial and final SolidWorks models. Passing the CSWA exam proves to employers that you have the necessary fundamental engineering graphics and SolidWorks competencies. Review individual features, commands, and tools for each project with the book 's 1.5 hour Video Instruction DVD and SolidWorks Help. The chapter exercises analyze and examine usage competencies based on the project objectives. The book is designed to compliment the SolidWorks Tutorials located in the SolidWorks Help menu. Each section explores the SolidWorks Online

User ' s Guide to build your working knowledge of SolidWorks. Desired outcomes and usage competencies are listed for each project. Know your objectives up front. Follow the step-by step procedures to achieve your design goals. Work between multiple documents, features, commands, and properties that represent how engineers and designers utilize SolidWorks in industry. The authors developed the industry scenarios by combining their own industry experience with the knowledge of engineers, department managers, vendors, and manufacturers. These professionals are

directly involved with SolidWorks everyday. Their responsibilities go far beyond the creation of just a 3D model.

Beginner's Guide to SOLIDWORKS 2022 - Level I SDC Publications Project Based SOLIDWORKS is specifically designed to complement an engineering graphics course. It covers how to apply engineering graphics concepts, such as part prints, section views, assembly drawings, tolerancing and fasteners. It also extends these topics into the world of design. Project Based SOLIDWORKS takes a specific part or assembly and teaches you how to model each part and its configurations, create part prints including assembly drawings if appropriate, and takes it one step further and teaches concepts such as FEA,

tolerancing, and parametric design. This book comes with instructional videos showing you how to perform each of the tutorials. It also comes with instructional videos showing how to complete each problem in the book. The exception to this is when a problem is open ended and each student will get different results. After completing all the tutorials in this book, you will be able to design moderately difficult parts and assemblies in a realistic manner. This book is perfect for a freshman design class that wishes to include realistic design problems within their curriculum. Structure Project Based SOLIDWORKS is arranged in projects. For example, Chapter two deals with the modeling of a Connecting Rod, Chapter three continues with the connecting rod to introduce the concept of

configurations, Chapter four creates a part print of the connecting rod, and Chapter five wraps up the project by performing a static FEA on the connecting rod. At the beginning of each chapter a list of prerequisite tutorials or knowledge is listed. You do not necessarily need to complete the tutorials within the book in order, but make sure that you have the prerequisite knowledge before you begin. Topics covered The following topics are covered in this book. • Part modeling • Part configurations • Assembly • Static FEA • Part Prints • Assembly drawings • Fasteners • Tolerancing • Parametric Modeling • 3D-sketches Videos Project Based SOLIDWORKS comes with over ten hours of video instruction. These videos follow along with the book and complement the text perfectly. Videos for each tutorial and problem in

the book are provided.

Additional videos on specific, difficult topics are also included. Captioned versions of these videos are also available for customers who want or need video captions.

[Beginner's Guide to SOLIDWORKS 2023 - Level I](#) SDC Publications SOLIDWORKS 2022

Advanced Techniques picks up where SOLIDWORKS 2022 Intermediate Skills leaves off. Its aim is to take you from an intermediate user with a basic understanding of SOLIDWORKS and modeling techniques to an advanced user capable of creating complex models and able to use the advanced tools provided by SOLIDWORKS. The text covers parts, surfaces, SimulationXpress, sheet metal, top-down assemblies and core and cavity molds. Every lesson and exercise in this book was created based on real

world projects. Each of these projects has been broken down and developed into easy and comprehensible steps.

Furthermore, at the end of every chapter there are self test questionnaires to ensure that you have gained sufficient knowledge from each section before moving on to more advanced lessons. This book takes the approach that in order to understand SOLIDWORKS, inside and out, you should create everything from the beginning and take it step by step. Who this book is for This book is for the intermediate to advanced user who has already completed the SOLIDWORKS Basic Tools book and may have also completed the SOLIDWORKS Intermediate Skills book. People who are very familiar with SOLIDWORKS and its add ins will also find this book to be a valuable resource.

Commands Guide
Tutorial for SolidWorks
2012 SDC Publications
SOLIDWORKS 2023: A
Power Guide for
Beginners and
Intermediate Users
textbook has been
designed for instructor-
led courses as well as
self-paced learning. It
is intended to help
engineers and
designers interested in
learning SOLIDWORKS
for creating 3D
mechanical designs.
This textbook is a
great help for new
SOLIDWORKS users
and a great teaching
aid in classroom
training. This textbook
consists of 14
chapters, with a total
of 780 pages covering
the major
environments of

SOLIDWORKS such as
Sketching environment,
Part modeling
environment, Assembly
environment, and
Drawing environment.
This textbook teaches
users to use
SOLIDWORKS
mechanical design
software for creating
parametric 3D solid
components,
assemblies, and 2D
drawings. This
textbook also includes
a chapter on creating
multiple configurations
of a design. This
textbook not only
focuses on the usage of
the tools and commands
of SOLIDWORKS but
also on the concept of
design. Every chapter
in this textbook
contains tutorials that
provide users with step-

by-step instructions for creating mechanical designs and drawings with ease. Moreover, every chapter ends with hands-on test drives which allow users to experience the user friendly and technical capabilities of SOLIDWORKS.

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Chapter 5. Creating Base Feature of Solid Models	Chapter 13. Working with Assemblies - I
Chapter 6. Creating Reference Geometries	Chapter 14. Working with Assemblies - II
Chapter 7. Advanced Modeling - I	Chapter 15. Working with Drawings
Chapter 8. Advanced Modeling - II	Introduction to SolidWorks SDC Publications

This book will teach you everything you need to know to start using SOLIDWORKS 2021 with easy to understand, step-by-step tutorials. This book features a simple robot design used as a project throughout the book. You will learn to model parts, create assemblies, run simulations and even create animations of

your robot design. No previous experience with Computer Aided Design (CAD) is needed since this book starts at an introductory level. The author begins by getting you familiar with the SOLIDWORKS interface and its basic tools right away. You will start by learning to model simple robot parts and before long you will graduate to creating more complex parts and multi-view drawings. Along the way you will learn the fundamentals of parametric modeling through the use of geometric constraints and relationships. You will also become familiar with many of SOLIDWORKS's powerful tools and commands that enable you to easily construct complex features in your models.

Also included is coverage of gears, gear trains and spur gear creation using SOLIDWORKS. This book continues by examining the different mechanisms commonly used in walking robots. You will learn the basic types of planar four-bar linkages commonly used in mechanical designs and how to use the GeoGebra Dynamic Geometry software to simulate and analyze 2D linkages. Using the knowledge you gained about linkages and mechanisms, you will learn how to modify your robot and change its behavior by modifying or creating new parts. In the second to last chapter of this book you learn how to combine all the robot parts into assemblies and then run motion analysis. You will finish off your project by creating 3D

animations of your robot in action. Finally, in the last chapter, the author introduces you to 3D printing. You will learn the general principles of 3D printing including a brief history of 3D printing, the types of 3D printing technologies, commonly used filaments, and the basic procedure for printing a 3D model. Being able to turn your designs into physical objects will open up a whole new world of possibilities to you.

There are many books that show you how to perform individual tasks with SOLIDWORKS, but this book takes you through an entire project and shows you the complete engineering process. By the end of this book you will have modeled and assembled nearly all the parts that

make up the TAMIYA® Mechanical Tiger and can start building your own robot.

Commands Guide Tutorial for SolidWorks 2013 SDC Publications

This senior undergraduate level textbook is written for Advanced Manufacturing, Additive Manufacturing, as well as CAD/CAM courses. Its goal is to assist students in colleges and universities, designers, engineers, and professionals interested in using SolidWorks as the design and 3D printing tool for emerging manufacturing technology for practical applications. This textbook will bring a new dimension to SolidWorks by introducing readers to the role of SolidWorks in the relatively new manufacturing paradigm shift, known as 3D-Printing which is based on Additive Manufacturing (AM)

technology. This new textbook: Features modeling of complex parts and surfaces Provides a step-by-step tutorial type approach with pictures showing how to model using SolidWorks Offers a user-Friendly approach for the design of parts, assemblies, and drawings, motion-analysis, and FEA topics Includes clarification of connections between SolidWorks and 3D-Printing based on Additive Manufacturing Discusses a clear presentation of Additive Manufacturing for Designers using SolidWorks CAD software "Introduction to SolidWorks: A Comprehensive Guide with Applications in 3D Printing" is written using a hands-on approach which includes a significant number of pictorial descriptions of the steps that a student should follow to model parts, assemble parts, and produce drawings.

Beginner's Guide to SOLIDWORKS 2022 - Level II CADArtifex Beginner ' s Guide to SOLIDWORKS 2021 – Level II starts where Beginner ' s Guide – Level I ends, following the same easy to read style and companion video instruction, but this time covering advanced topics and techniques. The purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create components in the context of an assembly and reference other components (Top-down design), propagate design changes with SOLIDWORKS ' parametric capabilities, mold design, welded structures and more while explaining the basic concepts of each

trade to allow you to understand the how and why of each operation. The author uses simple examples to allow you to better understand each command and environment, as well as to make it easier to explain the purpose of each step, maximizing the learning time by focusing on one task at a time. This book is focused on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. At the end of this book, you will have acquired enough skills to be highly competitive when it comes to designing with SOLIDWORKS, and while there are many less frequently used

commands and options available that will not be covered in this book, rest assured that those covered are most of the commands used every day by SOLIDWORKS designers. The author strived hard to include many of the commands required in the Certified SOLIDWORKS Professional Advanced and Expert exams as listed on the SOLIDWORKS website. Includes Video Instruction Each copy of this book includes access to video instruction. In these videos the author provides a clear presentation of tutorials found in the book. The videos reinforce the steps described in the book by allowing you to watch the exact steps the author uses to complete the exercises while he

provides additional details along the way. Captioned versions of these videos are also available for customers who want or need video captions. Solidworks 2013 Bible SDC Publications SOLIDWORKS 2020 Learn by doing - Part 1 introduces new users to mechanical design using SOLIDWORKS and how it can be used to create a variety of models. In nine tutorial-based chapters, the author guides you through all the necessary commands and options in SOLIDWORKS 2020, from sketching to parametric modeling and finally ending with rendering. The tools are presented one step at a time using simple

examples. The approach used in this book helps you to become a skilled SOLIDWORKS user. SOLIDWORKS 2020 Learn by doing - Part 1 begins with basic to introduction modeling. The later chapters focus on additional modeling, top-down assemblies, sheet metal modeling, and drafting. Table of Contents 1. Getting Started 2. Modeling Basics 3. Assembly Basics 4. Creating Drawings 5. Sketching 6. Additional Modeling Tools 7. Sheet metal Modeling 8. Top-Down Assembly 9. Dimensions and Annotations Beginner's Guide to SOLIDWORKS 2021 -

Level I SDC
Publications
SOLIDWORKS
Exercises - Learn by
Practicing (3rd
Edition) book is
designed to help
engineers and
designers interested in
learning SOLIDWORKS
by practicing 100 real-
world mechanical
models. This book
does not simply
provide step-by-step
instructions to design
3D models, instead it is
a practice book that
challenges users to
first analyze the
drawings and then
create the models
using the powerful
toolset of
SOLIDWORKS. This
approach helps users
to enhance their design
skills and take it to the

next level. You can also
access the video
instruction for creating
each exercise of the
book. This book is
written with a wide
range of SOLIDWORKS
users in mind, varying
from beginners to
advanced users. In
addition to
SOLIDWORKS, each
exercise of this book
can also be designed on
any other CAD
software such as
CATIA, Creo
Parametric, NX,
Autodesk Inventor, and
Solid Edge. NOTE: The
exercises/models
available for download
are created in
SOLIDWORKS 2021
and cannot be opened
in the lower version of
SOLIDWORKS.
Learning SOLIDWORKS

2021 Packt Publishing Ltd advanced file

The complete SolidWorks reference-tutorial for beginner to advanced techniques Mastering SolidWorks is the reference-tutorial for all users. Packed with step-by-step instructions, video tutorials for over 40 chapters, and coverage of little-known techniques, this book takes you from novice to power user with clear instruction that goes beyond the basics. Fundamental techniques are detailed with real-world examples for hands-on learning, and the companion website provides tutorial files for all exercises. Even veteran users will find value in new techniques that make familiar tasks faster, easier, and more organized, including

management tools that simplify and streamline pre-flight checks. SolidWorks is the leading 3D CAD program, and is an essential tool for engineers, mechanical designers, industrial designers, and drafters around the world. User friendly features such as drag-and-drop, point-and-click, and cut-and-paste tools belie the software's powerful capabilities that can help you create cleaner, more precise, more polished designs in a fraction of the time. This book is the comprehensive reference every SolidWorks user needs, with tutorials, background, and more for beginner to advanced techniques. Get a grasp on fundamental SolidWorks 2D and 3D tasks using realistic

examples with text-based expertise.

tutorials Delve into advanced functionality and capabilities not commonly covered by how-to guides

Incorporate improved search, Pack-and-Go and other file management tools into your workflow Adopt best practices and exclusive techniques you won ' t find anywhere else

Work through this book beginning-to-end as a complete SolidWorks course, or dip in as needed to learn new techniques and time-saving tricks on-demand. Organized for efficiency and designed for practicality, these tips will remain useful at any stage of expertise. With exclusive coverage and informative detail, Mastering SolidWorks is the tutorial-reference for users at every level of

SOLIDWORKS 2021 Reference Guide Independently Published

- A comprehensive reference book for SOLIDWORKS 2020
- Contains 260 plus standalone tutorials
- Starts with a basic overview of SOLIDWORKS 2020 and its new features
- Tutorials are written for each topic with new and intermediate users in mind
- Includes access to each tutorial ' s initial and final state
- Contains a chapter introducing you to 3D printing

The SOLIDWORKS 2020 Reference Guide is a comprehensive reference book written to assist the beginner

to intermediate user of SOLIDWORKS 2020. SOLIDWORKS is an immense software package, and no one book can cover all topics for all users. This book provides a centralized reference location to address many of the tools, features and techniques of SOLIDWORKS 2020. This book covers the following:

- System and Document properties
- FeatureManagers
- PropertyManagers
- ConfigurationManagers
- RenderManagers
- 2D and 3D Sketch tools
- Sketch entities
- 3D Feature tools
- Motion Study
- Sheet Metal
- Motion Study
- SOLIDWORKS Simulation

- PhotoView 360
- Pack and Go
- 3D PDFs
- Intelligent Modeling techniques
- 3D printing terminology and more

Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS 2020 software. If you are completely new to SOLIDWORKS, you should read Chapter 1 in detail and complete Lesson 1, Lesson 2 and Lesson 3 in the SOLIDWORKS Tutorials. If you are familiar with an earlier release of SOLIDWORKS, you still might want to skim Chapter 1 to become acquainted with some of the commands,

menus and features that you have not used; or you can simply jump to any section in any chapter. Each chapter provides detailed PropertyManager information on key topics with individual stand-alone short tutorials to reinforce and demonstrate the functionality and ease of the SOLIDWORKS tool or feature. The book provides access to over 260 models, their solutions and additional support materials. Learn by doing, not just by reading. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through

symmetry, patterns, copied components, design tables, configurations and more. The book is designed to complement the Online Tutorials and Online Help contained in SOLIDWORKS 2020. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The author developed the tutorials by combining his own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. He is directly involved with SOLIDWORKS every day and his

responsibilities go far beyond the creation of just a 3D model.

Beginner's Guide to SolidWorks 2014 - Level I John Wiley & Sons

Explore a practical and example-driven approach to understanding SOLIDWORKS 2020 and achieving CSWA and CSWP certification

Key Features Gain comprehensive insights into the core aspects of mechanical part modeling

Get up to speed with generating assembly designs with both standard and advanced mates

Focus on design practices for both 2D as well as 3D modeling and prepare to achieve CWSP and CWSA

certification

Book Description

SOLIDWORKS is the leading choice for 3D engineering and product design applications across industries such as aviation, automobiles, and consumer product design. This book takes a practical approach to getting you up and running with SOLIDWORKS 2020. You'll start with the basics, exploring the software interface and working with drawing files. The book then guides you through topics such as sketching, building complex 3D models, generating dynamic and static assemblies, and generating 2D engineering drawings to

equip you for mechanical design projects. You'll also do practical exercises to get hands-on with creating sketches, 3D part models, assemblies, and drawings. To reinforce your understanding of SOLIDWORKS, the book is supplemented by downloadable files that will help you follow up with the concepts and exercises found in the book. By the end of this book, you'll have gained the skills you need to create professional 3D mechanical models using SOLIDWORKS, and you'll be able to prepare effectively for the Certified SOLIDWORKS Associate (CSWA) and

Certified SOLIDWORKS Professional (CSWP) exams. What you will learn

- Understand the fundamentals of SOLIDWORKS and parametric modeling
- Create professional 2D sketches as bases for 3D models using simple and advanced modeling techniques
- Use SOLIDWORKS drawing tools to generate standard engineering drawings
- Evaluate mass properties and materials for designing parts and assemblies
- Understand the objectives and the formats of the CSWA and CSWP exams
- Discover expert tips and tricks to generate different part and assembly

configurations for your mechanical designs. Who this book is for: This book is for aspiring engineers, designers, drafting technicians, or anyone looking to get started with the latest version of SOLIDWORKS. Anyone interested in becoming a Certified SOLIDWORKS Associate (CSWA) or Certified SOLIDWORKS Professional (CSWP) will also find this book useful.

Learn SOLIDWORKS 2020 SDC Publications

- Designed to teach new users the basic concepts of SOLIDWORKS and good solid modeling techniques
- Uses a task oriented approach to learning

- **SOLIDWORKS** Focuses on the processes to complete the modeling of a part, instead of individual commands
- Includes access to extensive video instruction
- Covers commands found on the CSWA exam and includes a practice test
- This edition features expanded content covering the CSWA exam

This book is intended to help new users learn the basic concepts of SOLIDWORKS and good solid modeling techniques in an easy to follow guide that includes video instruction. It is a great starting point for those new to SOLIDWORKS or as a teaching aid in

classroom training to become familiar with the software ' s interface, basic commands and strategies as users complete a series of models while learning different ways to accomplish a particular task. At the end of this book, you will have a fairly good understanding of the SOLIDWORKS interface and the most commonly used commands for part modeling, assembly and detailing after completing a series of components and their 2D drawings complete with Bill of Materials. The book focuses on the processes to complete the modeling of a part, instead of

focusing on individual software commands or operations, which are generally simple enough to learn. Throughout this book the author introduces you to new commands that are required to pass the Certified SOLIDWORKS Associate exam, as listed on the SOLIDWORKS website. A dedicated chapter provides you with details about the exam, as well as a practice test to help you prepare for the actual exam. SOLIDWORKS is an easy to use CAD software that includes many time saving tools that will enable new and experienced users to complete design tasks faster than

before. Most commands covered in this book have advanced options, which may not be covered in this book. This is meant to be a starting point to help new users to learn the basic and most frequently used commands. Includes Video Instruction Each copy of this book includes access to video instruction. In these videos the author provides a clear presentation of tutorials found in the book. The videos reinforce the steps described in the book by allowing you to watch the exact steps the author uses to complete the exercises while he provides additional details along the way. Captioned

versions of these videos are also available for customers who want or need video captions. Solidworks 2020 SDC Publications Engineering Design with SOLIDWORKS 2016 and video instruction is written to assist students, designers, engineers and professionals. The book provides a solid foundation in SOLIDWORKS by utilizing projects with step-by-step instructions for the beginner to intermediate SOLIDWORKS user. Explore the user interface, CommandManager, menus, toolbars and modeling techniques to create parts, assemblies and drawings in an engineering environment.

Follow the step-by-step instructions and develop multiple parts and assemblies that combine machined, plastic and sheet metal components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, Design Tables, Bills of Materials, Custom Properties and Configurations. Address various SOLIDWORKS analysis tools and Intelligent Modeling techniques along with Additive Manufacturing (3D printing). Learn by doing not just by reading. Desired outcomes and usage competencies are listed for each project. Know your objective up

front. Follow the steps in Projects 1 - 9 to achieve the design goals. Review Project 10 on Additive Manufacturing (3D printing) and its benefits and features. Understand the terms and technology used in low cost 3D printers. Work between multiple documents, features, commands and custom properties that represent how engineers and designers utilize SOLIDWORKS in industry. Review individual features, commands and tools with the Video Instruction. The projects contain exercises. The exercises analyze and examine usage competencies. Collaborate with leading industry suppliers such as SMC Corporation of America, Boston Gear and 80/20 Inc. Collaborative information

translates into numerous formats such as paper drawings, electronic files, rendered images and animations. On-line intelligent catalogs guide designers to the product that meets both their geometric requirements and performance functionality. The author developed the industry scenarios by combining his own industry experience with the knowledge of engineers, department managers, vendors and manufacturers. These professionals are directly involved with SOLIDWORKS every day. Their responsibilities go far beyond the creation of just a 3D model. The book is designed to compliment the SOLIDWORKS Tutorials contained in SOLIDWORKS 2016. Engineering Design with SOLIDWORKS 2016 and Video Instruction SDC Publications SOLIDWORKS 2018 Tutorial with video instruction is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The text provides a step-by-step, project based learning approach. It also contains information and examples on the five categories, to take and understand the Certified Associate - Mechanical Design (CSWA) exam. The book is divided into four sections. Chapters 1 - 5 explore the

SOLIDWORKS User Interface and CommandManager, Document and System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. In chapter 6 you will create the final robot assembly. The physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and components for the final robot assembly are provided. Chapters 7 - 10 prepare you for

the Certified Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs from subtractive manufacturing, and its features. You will also learn the terms and technology used in low cost 3D printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit

sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, apply proper design intent, design tables and configurations. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each chapter. Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in

industry.

Engineering Graphics with SolidWorks 2012

SDC Publications

Beginner ' s Guide to SOLIDWORKS 2022 – Level II starts where Beginner ' s Guide – Level I ends, following the

same easy to read style and companion video instruction, but this time covering advanced topics and techniques. The

purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create components in the

context of an assembly and reference other components (Top-down design), propagate design changes with

SOLIDWORKS ' parametric capabilities, mold design, welded structures and more

while explaining the

basic concepts of each trade to allow you to understand the how and why of each operation. The author uses simple examples to allow you to better understand each command and environment, as well as to make it easier to explain the purpose of each step, maximizing the learning time by focusing on one task at a time. This book is focused on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. At the end of this book, you will have acquired enough skills to be highly competitive when it comes to designing with SOLIDWORKS, and while there are many less

frequently used commands and options available that will not be covered in this book, rest assured that those covered are most of the commands used every day by SOLIDWORKS designers. The author strived hard to include many of the commands required in the Certified SOLIDWORKS Professional Advanced and Expert exams as listed on the SOLIDWORKS website. Includes Video Instruction Each copy of this book includes access to video instruction. In these videos the author provides a clear presentation of tutorials found in the book. The videos reinforce the steps described in the book by allowing you to watch the exact steps the author uses to complete

the exercises while he provides additional details along the way. Captioned versions of these videos are also available for customers who want or need video captions.