

Solidworks Commands Guide

• 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.

The Commands Guide Tutorial for SolidWorks 2010 is a comprehensive reference book written to assist beginner to intermediate users of SolidWorks. SolidWorks is an immense software package, and no one book can cover all topics for all users. The book provides a centralized reference location to address many of the System and Document properties, FeatureManagers, PropertyManagers, ConfigurationManagers and RenderManagers along with 2D and 3D Sketch tools, Sketch entities, 3D Feature tools, Motion Study, SustainabilityXpress, DFMXpress, SimulationXpress, Sheet Metal, PhotoView 360 and more. Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2010 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 (17 total) 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. All models for the 230 plus tutorials are located on the enclosed CD with their solution (initial and final). 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 compliment the Online Tutorials and Online Help contained in SolidWorks 2010. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The authors developed the tutorials 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.

The SOLIDWORKS 2016 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2016. 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 2016. This book covers the following: System and Document propertiesFeatureManagersPropertyManagersConfigurationManagersRenderManagers2D and 3D Sketch toolsSketch entities3D Feature toolsMotion StudySheet MetalMotion StudySolidWorks SimulationPhotoView 360Pack and Go3D PDFsIntelligent Modeling techniques3D printing terminology and more

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Commands Guide Tutorial for SolidWorks 2013SDC Publications

Beginner's Guide to SOLIDWORKS 2019 – 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.

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.

Beginner's Guide to SOLIDWORKS 2016 – 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 the commands required in the Certified SOLIDWORKS Associate test as listed on the SOLIDWORKS website, and some, as well as several more.

SOLIDWORKS 2018 Basic Tools is the first book in a three part series. It introduces new users to the SOLIDWORKS interface, SOLIDWORKS tools and basic modeling techniques. It provides you with a strong understanding of SOLIDWORKS and covers the creation of parts, assemblies and drawings. 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.

Learn to effectively create, modify, and edit sketches and solid features using the latest release of one of the most popular software programs in the industry, SolidWorks 2008. A Commands Guide for SolidWorks 2008 offers readers a comprehensive reference guide that begins with an overview of the software's concepts and terminology. Subsequent chapters address System and Document properties, FeatureManagers, PropertyManagers, ConfigurationManagers, and RenderManagers as well as 2D and 3D sketch tools, sketch entities, and more. This thorough coverage provides readers with a solid understanding of the techniques for reusing features, parts, and assemblies through symmetry, patterns, copied components, design tables, and configurations. The perfect resource for users with beginner to intermediate-level experience with SolidWorks, this book illustrates clearly and carefully how multiple design situations and systematic steps combine to produce successful designs.

SolidWorks 2012 Tutorial with Video Instruction is target towards a technical school, two year college, four year university or industry professional that is a beginner or intermediate CAD user. The text provides a student who is looking for a step-by-step project based approach to learning SolidWorks with an enclosed 1.5 hour video instruction DVD, SolidWorks model files, and preparation for the CSWA exam. The book is divided into two sections. Chapters 1 - 7 explore the SolidWorks User Interface and CommandManager, Document and System properties, simple machine parts, simple and complex assemblies, design tables, configurations, multi-sheet, multi-view drawings, BOMs, Revision tables using basic and advanced features along with Intelligent Modeling Techniques, SustainabilityXpress, SimulationXpress and DFMXpress. Chapters 8 - 11 prepare you for the new Certified SolidWorks Associate Exam (CSWA). The CSWA certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. 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, 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.

This book is written to assist you with passing the SOLIDWORKS associate level exams. It provides you with detailed information and exercises that will aid you in passing the following exams: Certified SOLIDWORKS Associate (CSWA), Certified SOLIDWORKS Associate Sustainable Design (CSWA-SD), Certified SOLIDWORKS Associate Simulation (CSWSA-S) and the Certified SOLIDWORKS Associate Additive Manufacturing (CSWA-AM) exam. There are three goals for this book. The primary goal of this book is not only to help you pass the CSWA, CSWA-SD, CSWSA-S and CSWA-AM exams, but also to ensure that you understand and comprehend the concepts and implementation details of the four certification processes. The second goal is to provide the most comprehensive coverage of CSWA, CSWA-SD, CSWSA-S and CSWA-AM exam related topics available, without too much coverage of topics not on the exam. The third and ultimate goal is to get you from where you are today to the point that you can confidently pass the CSWA, CSWA-SD, CSWSA-S and CSWA-AM exams.

CSWA Exam The CSWA certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles. The intended audience for this section of the book is anyone trying to take and pass the CSWA exam with a minimum of 6 - 9 months of SOLIDWORKS experience and basic knowledge of engineering fundamentals and practices. SOLIDWORKS recommends that you review their SOLIDWORKS Tutorials on Parts, Assemblies and Drawings as a prerequisite and have at least 45 hours of classroom time learning SOLIDWORKS or using SOLIDWORKS with basic engineering design principles and practices.

CSWA-SD Exam The Certified SOLIDWORKS Associate Sustainable Design (CSWA-SD) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of environmental assessment and sustainable design. This section of the book is intended for anyone interested in Sustainable design as well as life cycle assessment and trying to take and pass the CSWA-SD exam. Although no hands-on usage of SOLIDWORKS is required for the CSWA-SD certification exam, it is a good idea to review the SOLIDWORKS SustainabilityXpress and SOLIDWORKS Sustainability tutorials inside of SOLIDWORKS to better understand the actual workflow. The CSWA-SD is based off the SOLIDWORKS Sustainable Design Guide that incorporates concepts including sustainability, environmental assessment and life cycle impact assessment.

CSWSA-S Exam The Certified SOLIDWORKS Associate Simulation (CSWSA-S) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of stress analysis and the Finite Element Method (FEM). The CSWSA-S section of the book is for anyone trying to take and pass the CSWSA-S with a minimum of 6 - 9 months of SOLIDWORKS experience and knowledge in the following areas: Engineering Mechanics - Statics, Strength of Materials, Finite Element Method/Finite Element Analysis Theory, Applied concepts in SOLIDWORKS Simulation: namely Static Analysis, Solid, Shell, and Beam elements, Connections and Applying loads and boundary conditions and interpreting results. The purpose of this section in the book is NOT to educate a new or intermediate user on SOLIDWORKS Simulation, but to cover and to inform you on the types of questions, layout and what to expect when taking the CSWSA-S exam.

CSWA-AM Exam The Certified SOLIDWORKS Associate Additive Manufacturing (CSWA-AM) certification indicates a foundation in and apprentice knowledge of today's 3D printing technology and market. The intended audience for this section of the book is anyone trying to take and pass the CSWA-AM exam and an interest in Additive Manufacturing. The CSWA-AM exam is meant to be taken after the completion of the 10-part learning path located on MySOLIDWORKS.com. The CSWA-AM exam fundamentally covers two 3D printing technologies: Fused Filament Fabrication (FFF) and STereoLithography (SLA). There are a few questions on Selective Laser Sintering (SLS) technology and available software-based printing aids.

The SOLIDWORKS 2017 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2017. 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 2017. This book covers the following: System and Document

propertiesFeatureManagersPropertyManagersConfigurationManagersRenderManagers2D and 3D Sketch toolsSketch entities3D Feature toolsMotion StudySheet MetalMotion StudySOLIDWORKS SimulationPhotoView 360Pack and Go3D PDFsIntelligent Modeling techniques3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS 2017 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 250 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 compliment the Online Tutorials and Online Help contained in SolidWorks 2017. 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.

The Official Guide to Certified SOLIDWORKS Associate Exams: CSWA, CSDA, CSWSA-FEA is written to assist the SOLIDWORKS user to pass the associate level exams.

Information is provided to aid a person to pass the Certified SOLIDWORKS Associate (CSWA), Certified SOLIDWORKS Sustainable Design Associate (CSDA) and the Certified SOLIDWORKS Simulation Associate Finite Element Analysis (CSWSA FEA) exam. There are three goals for this book. The primary goal is not only to help you pass the CSWA, CSDA and CSWSA-FEA exams, but also to ensure that you understand and comprehend the concepts and implementation details of the three certification processes. The second goal is to provide the most comprehensive coverage of CSWA, CSDA and CSWSA-FEA exam related topics available, without too much coverage of topics not on the exam. The third and ultimate goal is to get you from where you are today to the point that you can confidently pass the CSWA, CSDA and the CSWSA-FEA exam. The Certified SOLIDWORKS Associate (CSWA) certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles. Passing this exam provides students the chance to prove their knowledge and expertise and to be part of a worldwide industry certification standard. The Certified SOLIDWORKS Sustainable Design Associate (CSDA) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of environmental assessment and sustainable design. The Certified SOLIDWORKS Simulation Associate - Finite Element Analysis (CSWSA-FEA) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of stress analysis and the Finite Element Method (FEM).

The SolidWorks 2014 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2014. 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 2014. Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2014 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. Videos are provided to introduce the new user to the basics of using SolidWorks 3D CAD software. 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 (18 total) provides detailed PropertyManager information on key topics with individual standalone short tutorials to reinforce and demonstrate the functionality and ease of the SolidWorks tool or feature. All models for the 240 plus tutorials are located on the enclosed book CD with their solution (initial and final). 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 compliment the Online Tutorials and Online Help contained in SolidWorks 2014. 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.

- Uses step-by-step, project based tutorials designed for beginning or intermediate users
- Will prepare you for the Certified SOLIDWORKS Associate Exam
- Includes a chapter introducing you to 3D printing

SOLIDWORKS 2020 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.

This book is intended to help new users to learn the basic concepts of SolidWorks and good solid modeling techniques in an easy to follow guide. It will be 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 the user completes 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 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. The author strived hard to include the commands required in the Certified SolidWorks Associate test as listed on the SolidWorks website, as well as several more.

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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.

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This book, along with the companion disc that accompanies it, will provide you with a wealth of information about the three segments of the CSWP CORE exam. The intended audience for this book is a person who has passed the CSWA exam and who has eight or more months of SolidWorks training and usage. This guide is not intended to teach you how to use SolidWorks, but is written to provide you with CSWP exam tips, hints and information on sample questions and categories that are aligned with the exam. This guide is written to help you take and pass the CSWP exam. This book comes with a companion DVD containing segment videos for you to follow while you use the book. Each segment video provides valuable information, tips and tricks to successfully pass the CSWP CORE exam. SolidWorks model files, in both their initial and final state, are provided on this DVD for both SolidWorks 2012 and 2013. The book is organized into three chapters. Each chapter is focused on a segment of the CSWP CORE exam. This is not intended to be a step-by-step book.

The Commands Guide Tutorial for SolidWorks 2012 is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2012. SolidWorks is an immense software package, and no one book can cover all topics for all users. The book provides a centralized reference location to address many of the tools, features and techniques of SolidWorks 2012. 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 Sustainability Sustainability Xpress FlowXpress PhotoView 360 Pack and Go Intelligent Modeling techniques and more. Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks® 2012 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 (18 total) provides detail 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. All models for the 240 plus tutorials are located on the enclosed book CD with their solution (initial and final). 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 design to compliment the Online Tutorials and Online Help contained in SolidWorks 2012. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The authors developed the tutorials by combining their own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. These professionals are directly involved with SolidWorks everyday. Their responsibilities go far beyond the creation of just a 3D model.

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The SolidWorks 2015 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2015. 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 2015. This book covers the following: System and Document propertiesFeatureManagersPropertyManagersConfigurationManagersRenderManagers2D and 3D Sketch toolsSketch entities3D Feature toolsMotion StudySheet MetalMotion StudySolidWorks SimulationPhotoView 360Pack and Go3D PDFsIntelligent Modeling techniques3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2015 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 240 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 compliment the Online Tutorials and Online Help contained in SolidWorks 2015. 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.

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. The author strived hard to include the commands required in the Certified SOLIDWORKS Associate and Certified SOLIDWORKS Professional Exams as listed on the SOLIDWORKS website. 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 visual 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.

This book is intended to help new users to learn the basic concepts of SolidWorks and good solid modeling techniques in an easy to follow guide. It will be 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 the user completes 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. The author strived hard to include the commands required in the Certified SolidWorks Associate test as listed on the SolidWorks website, as well as several more. 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.

This book is written to assist you with passing the SOLIDWORKS associate level exams. It provides you with detailed information and exercises that will aid you in passing the following exams: Certified SOLIDWORKS Associate (CSWA), Certified SOLIDWORKS Associate Sustainable Design (CSWA-SD), Certified SOLIDWORKS Simulation Associate Finite Element Analysis (CSWSA-FEA) and the Certified SOLIDWORKS Associate Additive Manufacturing (CSWA-AM) exam. There are three goals for this book. The primary goal of this book is not only to help you pass the CSWA, CSWA-SD, CSWSA-FEA and CSWA-AM exams, but also to ensure that you understand and comprehend the concepts and implementation details of the four certification processes. The second goal is to provide the most comprehensive coverage of CSWA, CSWA-SD, CSWSA-FEA and CSWA-AM exam related topics available, without too much coverage of topics not on the exam. The third and ultimate goal is to get you from where you are today to the point that you can confidently pass the CSWA, CSWA-SD, CSWSA-FEA and CSWA-AM exams.

SOLIDWORKS 2021: A Power Guide for Beginners and Intermediate User 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 design. 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 798 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. Table of Contents: Chapter 1. Introduction to SOLIDWORKS Chapter 2. Drawing Sketches with SOLIDWORKS Chapter 3. Editing and Modifying Sketches Chapter 4. Applying Geometric Relations and Dimensions Chapter 5. Creating First/Base Feature of Solid Models Chapter 6. Creating Reference Geometries Chapter 7. Advanced Modeling - I Chapter 8. Advanced Modeling - II Chapter 9. Patterning and Mirroring Chapter 10. Advanced Modeling - III Chapter 11. Working with Configurations Chapter 12. Working with Assemblies - I Chapter 13. Working with Assemblies - II Chapter 14. Working with Drawings Main Features of the Textbook Comprehensive coverage of tools Step-by-step real-world tutorials with every chapter Hands-on test drives to enhance the skills at the end of every chapter Additional notes and tips Customized content for faculty (PowerPoint Presentations) Free learning resources for faculty and students Additional student and faculty projects Technical support for the book by contacting info@cadartifex.com

The SOLIDWORKS 2018 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2018. 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 2018. 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 2018 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 250 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 2018. 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.

Describes the basic concepts of SolidWorks 2010 and includes exercises that cover the SolidWorks interface and commands.

The SOLIDWORKS 2019 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2019. 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 2019. 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 2019 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 2019. 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.

The Commands Guide Tutorial for SolidWorks 2011 is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2011. SolidWorks is an immense software package, and no one book can cover all topics for all users. The book provides a centralized reference location to address many of the tools, features and techniques of SolidWorks 2011. 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 Sustainability Sustainability Xpress FlowXpress PhotoView 360 Pack and Go Intelligent Modeling techniques and more. Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2011 software. If you are completely new to SolidWorks, you should read Chapter 1 in detail and complete Tutorial 1, Tutorial 2, and Tutorial 3 in the SolidWorks Tutorials. If you are familiar with an earlier release of SolidWorks, you might still want to skim Chapter 1 to get acquainted with some of the new commands, menus, and features that you haven't used; or you can simply jump to any section in any chapter. Each chapter (18 total) 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. All models for the 240 plus tutorials are provided on the enclosed book CD with their solution (initial and final). Learn by doing, not just reading! Formulate the skills to create, modify and edit sketches and solid features. You will also learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, design tables, configurations and more. The book is designed to compliment the Online Tutorials and Online Help contained in SolidWorks 2011. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs.

SOLIDWORKS 2019 Intermediate Skills is part of a three part series which builds on the SOLIDWORKS features learned in SOLIDWORKS 2019 Basic Tools. SOLIDWORKS 2019 Intermediate Skills broadens your SOLIDWORKS knowledge base by covering such features as surveys, lofts and boundaries, the use of multibodies, generating engineering drawings and other SOLIDWORKS functions that are critical for the effective use of this powerful software. This book helps prepare you for the advanced features of SOLIDWORKS which are covered in SOLIDWORKS Advanced Techniques. It uses a step

by step tutorial approach with real world projects. This book also features a Quick-Reference-Guide to the SOLIDWORKS 2019 commands, icons, and customized hotkeys. Who's this book for? This book is for the mid-level user, who is already familiar with the SOLIDWORKS program. It is also a great resource for the more CAD literate individuals who want to expand their knowledge of the different features that SOLIDWORKS 2019 has to offer.

Most CAD professionals today recognize the need to become certified to prove their skills, prepare for new job searches, and to learn new skills while at their existing job. Specifying a Certified SolidWorks Associate (CSWA) certification on your resume is a great way to increase your chances of landing a new job, getting a promotion, or looking more qualified when representing your company on a consulting job. The primary goal of this book is not only to help you pass the CSWA exam, but also to ensure that you understand and comprehend the concepts and implementation details of the CSWA process. The second goal is to provide the most comprehensive coverage of CSWA exam related topics available, without too much coverage of topics not on the exam. The third and ultimate goal is to get you from where you are today to the point that you can confidently pass the CSWA exam. DS SolidWorks Corp. offers various stages of certification. Each stage represents increasing levels of expertise in 3D CAD design as it applies to engineering: Certified SolidWorks Associate CSWA, Certified SolidWorks Professional CSWP and Certified SolidWorks Expert CSWE along with specialty fields in Simulation, Sheet Metal, and Surfacing. The CSWA Certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles. The main requirement for obtaining the CSWA certification is to take and pass the on-line proctored 180 minute exam (minimum of 165 out of 240 points). The new CSWA exam consists of fourteen questions in five categories. Passing this exam provides students the chance to prove their knowledge and expertise and to be part of a worldwide industry certification standard.

SOLIDWORKS 2016 Basic Tools is the first book in a three part series. It introduces new users to the SOLIDWORKS interface, SOLIDWORKS tools and basic modeling techniques. It provides readers with a strong understanding of SOLIDWORKS and covers the creation of parts, assemblies and drawings. Every lesson and exercise in this book was created based on real world projects. Each of these projects have been broken down and developed into easy and comprehensible steps for the reader. Furthermore, at the end of every chapter there are self test questionnaires to ensure that the reader has 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, the reader should create everything from the beginning and take it step by step.

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 the user completes 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. The author strived hard to include the commands required in the Certified SolidWorks Associate test as listed on the SolidWorks website, as well as several more. 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.

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.

Provides an introduction to SolidWorks 2010 through step-by-step tutorials that cover such topics as linkage assembly, front support assembly, the fundamentals of drawing, and pneumatic test module assembly.

The Commands Guide Tutorial for SolidWorks 2013 is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2013. 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 2013. 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 Sustainability Sustainability Xpress FlowXpress PhotoView 360 Pack and Go

Intelligent Modeling techniques and more. Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2013 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 (18 total) 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. All models for the 240 plus tutorials are located on the enclosed book CD with their solution (initial and final). 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 design to compliment the Online Tutorials and Online Help contained in SolidWorks 2013. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The authors developed the tutorials by combining their own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. These professionals are directly involved with SolidWorks everyday. Their responsibilities go far beyond the creation of just a 3D model.

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. The author strived hard to include the commands required in the Certified SOLIDWORKS Associate and Certified SOLIDWORKS Professional Exams as listed on the SOLIDWORKS website. 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.

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