

Commands Guide Tutorial for SolidWorks® 2013

A comprehensive reference guide
with over 240 tutorials

David C. Planchard CSWP
Marie P. Planchard CSWP



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Commands Guide Tutorial For Solidworks 2013

David C. Planchard, Marie P. Planchard

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Commands Guide Tutorial for SolidWorks 2013 David C. Planchard, Marie P. Planchard, 2012-12-27 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 designed 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.

Engineering Graphics with SolidWorks 2013 and Video Instruction David Planchard, Marie Planchard, 2013-02-18 Engineering Graphics with SolidWorks 2013 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 15 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 complement 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 every day Their responsibilities go far beyond the creation of just a 3D model

Engineering Design with SolidWorks 2013 and Video Instruction David C. Planchard, Marie P. Planchard, 2013
Engineering Design with SolidWorks 2013 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: SimulationXpress, SustainabilityXpress, and DFMXpress, and Intelligent Modeling techniques. 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 Project 1-8 to achieve the design goals. 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 enclosed Video Instruction DVD. 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 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 every day Their responsibilities go far beyond the creation of just a 3D model The book is design to compliment the SolidWorks Tutorials contained in SolidWorks 2013 There are over 2 5 hours of video instructions on the enclosed DVD *SolidWorks 2014 in 5 Hours with Video Instruction* David Planchard,2014-03-07 SolidWorks 2014 in 5 Hours with video instruction introduces the new user to the basics of using SolidWorks 3D CAD software in five easy lessons This book is intended for the student or designer that needs to learn SolidWorks quickly and effectively for senior capstone machine design kinematics dynamics and other engineering and technology projects that use SolidWorks as a tool Engineers in industry are expected to have SolidWorks skills for their company s next project Students need to learn SolidWorks without taking a formal CAD course Based on years of teaching SolidWorks to engineering students SolidWorks 2014 in 5 Hours concentrates on the areas where the new user improves efficiency in the design modeling process By learning the correct SolidWorks skills and file management techniques you gain the most knowledge in the shortest period of time You develop a mini Stirling Engine and investigate the proper design intent and constraints The mini Stirling Engine is based on the external combustion closed cycle engine of Scottish inventor Robert Stirling In addition to 3D modeling the engine can be used to teach and connect many engineering and physics principles You begin with an overview of SolidWorks and the User Interface UI its menus toolbars and commands With a quick pace you learn the essentials of 2D sketching part and assembly creation preform motion study develop detailed part and assembly drawings and much more View the provided videos for each section of the book to enhance your experience SolidWorks Interface 2D Sketching Sketch Planes and Sketch tools 3D Features and Design Intent Creating an Assembly Fundamentals in Drawings Part 1

Fundamentals in Drawings Part 2 **Engineering Design with SolidWorks 2014 and Video Instruction** David Planchard,2014 Engineering Design with SolidWorks 2014 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 SimulationXpress Sustainability SustainabilityXpress and DFMXpress and Intelligent Modeling techniques 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 Project 1 8 to achieve the design goals 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 design to compliment the SolidWorks Tutorials contained in SolidWorks 2014

Commands Guide Tutorial for SolidWorks 2012 David C. Planchard, Marie P. Planchard, 2011-12-18 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 **Product Design Modeling using CAD/CAE** Kuang-Hua Chang,2014-01-20 Product Design Modeling using CAD CAE is the third part of a four part series It is the first book to integrate discussion of computer design tools throughout the design process Through this book you will Understand basic design principles and all digital design paradigms Understand computer aided design engineering and manufacturing CAD CAE CAM tools available for various design related tasks Understand how to put an integrated system together to conduct all digital design ADD Provides a comprehensive and thorough coverage of essential elements for product modeling using the virtual engineering paradigm Covers CAD CAE in product design including solid modeling mechanical assembly parameterization product data management and data exchange in CAD Case studies and tutorial examples at the end of each chapter provide hands on practice in implementing off the shelf computer design tools Provides two projects showing the use of Pro ENGINEER and SolidWorks to implement concepts discussed in the book **SolidWorks 2013 Tutorial** David C. Planchard,Marie P. Planchard,2013 SolidWorks 2013 Tutorial with Video Instruction is targeted 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 **Beginner's Guide to Solidworks 2013** Alejandro Reyes (Electrical Engineer),2013-03-18 Beginner s Guide to SolidWorks 2013 Level II starts where Beginner s Guide Level I ends following the same easy to read style 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.

Beginner's Guide to Solidworks 2013 Alejandro Reyes, 2013-03-05 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.

Commands Guide Tutorial for SolidWorks 2011 David C. Planchard, Marie P. Planchard, 2010 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

Chapter1 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

Commands Guide Tutorial for Solidworks 2010 David C. Planchard,Marie P. Planchard,2010 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 **SolidWorks 2013 Part II - Advanced Techniques** Paul Tran,2013 SolidWorks 2013 Part II Advanced Techniques picks up where SolidWorks 2013 Part I Basic Tools 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 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. **A Commands Guide Tutorial for SolidWorks 2007** David C. Planchard, Marie P. Planchard, 2007

Parametric Modeling with SolidWorks 2013 Randy Shih, Paul Schilling, 2013-02-20. **Parametric Modeling with SolidWorks 2013** contains a series of sixteen tutorial style lessons designed to introduce SolidWorks 2013 solid modeling and parametric modeling techniques and concepts. This book introduces SolidWorks 2013 on a step by step basis starting with constructing basic shapes all the way through to the creation of assembly drawings and motion analysis. This book takes a hands on exercise intensive approach to all the important Parametric Modeling techniques and concepts. Each lesson introduces a new set of commands and concepts building on previous lessons. The lessons guide the user from constructing basic shapes to building intelligent solid models, assemblies and creating multi view drawings. This book also covers some of the more advanced features of SolidWorks 2013 including how to use the SolidWorks Design Library, basic motion analysis, collision detection and analysis with SimulationXpress. The exercises in this book cover the performance tasks that are included on the Certified SolidWorks Associate CSWA Examination. Reference guides located at the front of the book and in each chapter show where these performance tasks are covered.

Solidworks 2013 and Engineering Graphics Randy Shih, 2013-04-27. **SolidWorks 2013 and Engineering Graphics An Integrated Approach** combines an introduction to SolidWorks 2013 with a comprehensive coverage of engineering graphics principles. Not only will this unified approach give your course a smoother flow, your students will also save money on their textbooks. What's more, the exercises in this book cover the performance tasks that are included on the Certified SolidWorks Associate CSWA Examination. Reference guides located at the front of the book and in each chapter show where these performance tasks are covered. The primary goal of **SolidWorks 2013 and Engineering Graphics An Integrated Approach** is to introduce the aspects of Engineering Graphics with the use of modern Computer Aided Design package SolidWorks 2013. This text is intended to be used as a training guide for students and professionals. The chapters in this text proceed in a pedagogical fashion to guide you from constructing basic shapes to making complete sets of engineering drawings. This text takes a hands on exercise intensive approach to all the important concepts of Engineering Graphics as well as in depth discussions of parametric feature based CAD techniques. This textbook contains a series of fifteen chapters with detailed step by step tutorial style lessons designed to introduce beginning CAD users to the graphic language used in all branches of technical industry. This

book does not attempt to cover all of SolidWorks 2013's features only to provide an introduction to the software. It is intended to help you establish a good basis for exploring and growing in the exciting field of Computer Aided Engineering.

Learning SolidWorks 2013 Randy H. Shih, 2012-11-30 This book will teach you everything you need to know to start using SolidWorks 2013 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 Drafting 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 final 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. 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.

Beginner's Guide to SolidWorks 2014 - Level I Alejandro Reyes, 2014-02-26

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 **Beginner's Guide to SOLIDWORKS 2021 - Level I** Alejandro Reyes,2021-01-29 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 SOLIDWORKS 2020 Reference Guide David Planchard,2019-12 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.

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