

About The Author Solidworks

The only continuous, step-by-step tutorial for SolidWorks SolidWorks is a 3D CAD manufacturing software package that has been used to design everything from aerospace robotics to bicycles. This book teaches beginners to use SolidWorks through a step-by-step tutorial, letting you build, document, and present a project while you learn. Tools and functionality are explained in the context of professional, real-world tasks and workflows. You will learn the essential functions and gain the skills to use the software at once. SolidWorks is a popular design software for manufacturing, and this book introduces it in the context of actually creating an object Begins with an overview of SolidWorks conventions and the interface Explains how to create models and drawings, create a revolved part and subassembly, and model parts within a subassembly Explores modification capabilities and drawing and Bill of Materials templates Moves on to top-level assembly models and drawings, Toolbox components and the Design Library, mates, export and printing capabilities, and creating renderings Includes a glossary, a foreword from the SolidWorks product manager, and downloadable tutorial files SolidWorks 2010: No Experience Required quickly turns beginners into confident users of SolidWorks. It includes sample design sessions that explore both

applications, command and function cross-references, discussions on data interchangeability, and more.

Engineers working with SOLIDWORKS are often faced with tedious, repetitive work that can consume a lot of time, but it doesn't have to be this way. One of the most exciting aspects of SOLIDWORKS is its robust programming interface or API. The SOLIDWORKS API allows you to write code that can perform almost any series of actions for you.

SOLIDWORKS was built from the ground up to automate, and in this book, you will learn how to take advantage of these powerful tools to speed up your work. Automating SOLIDWORKS 2019 Using Macros is designed as a tutorial to help beginner to intermediate programmers develop macros for SOLIDWORKS. Experience with programming isn't required. The book starts with a new chapter on the fundamentals of Visual Basic.NET and the SOLIDWORKS API to make the learning process easier for beginners. The rest of the book introduces you to developing macros using the SOLIDWORKS API. The book concludes with a chapter dedicated to some of the author's favorite source code for you to use as the basis for typical automation procedures. The focus of this book is primarily on the Visual Studio Tools for Applications (VSTA) macro interface. It covers many of the major API functions through practical use cases. It will teach you the

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fundamentals of Visual Basic.NET as well as SOLIDWORKS, SOLIDWORKS PDM Professional, SOLIDWORKS Document Manager and Excel API functions. Author Mike Spens has been professionally developing macros for SOLIDWORKS for more than a decade. He has helped numerous companies develop their own programs and streamline their workflows. If you want to learn how to develop your own macros for SOLIDWORKS, following best practices and using well written code, then this is the perfect book for you.

Engineering Graphics with SOLIDWORKS 2021 is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The book combines the fundamentals of engineering graphics and dimensioning practices with a step-by-step project based approach to learning SOLIDWORKS. The book is divided into four sections with 11 Chapters. Chapters 1 - 3: Explore the history of engineering graphics, manual sketching techniques, orthographic projection, Third vs. First angle projection, multi-view drawings, dimensioning practices (ASME Y14.5-2009 standard), line type, fit type, tolerance, fasteners in general, general thread notes and the history of CAD leading to the development of SOLIDWORKS. Chapters 4 - 9: Comprehend the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple machine parts, simple

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and complex assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. Follow the step-by-step instructions in over 80 activities to develop eight parts, four sub-assemblies, three drawings and six document templates. Chapter 10: Prepare for the Certified SOLIDWORKS Associate (CSWA) exam. Understand the curriculum and categories of the CSWA exam and the required model knowledge needed to successfully take the exam. Chapter 11: Provide a basic understanding between Additive vs. Subtractive manufacturing. Discuss Fused Filament Fabrication (FFF), STereoLithography (SLA), and Selective Laser Sintering (SLS) printer technology. Select suitable filament material. Comprehend 3D printer terminology. Knowledge of preparing, saving, and printing a model on a Fused Filament Fabrication 3D printer. Information on the Certified SOLIDWORKS Additive Manufacturing (CSWA-AM) exam. Review individual features, commands, and tools using SOLIDWORKS Help. The chapter exercises analyze and examine usage competencies based on the chapter objectives. The book is designed to complement the SOLIDWORKS Tutorials located in the SOLIDWORKS Help menu. Desired outcomes and usage competencies are listed for each project. Know your objectives up front. Follow the step-by step procedures to achieve your

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design goals. Work between multiple documents, features, commands, and properties that represent how engineers and designers utilize SOLIDWORKS in industry. The author developed the industry scenarios by combining his own industry experience with the knowledge of engineers, department managers, vendors and manufacturers.

A fan of the SolidWorks Bible, but want more detail on assemblies? Here you go. SolidWorks fans have long sought more detail on SolidWorks topics, and now you have it. We took our popular SolidWorks Bible, divided it into two books (SolidWorks 2011 Assemblies Bible and SolidWorks 2011 Parts Bible) and packed each new book with a host of items from your wish lists, such as more extensive coverage of the basics, additional tutorials, and expanded coverage of topics largely ignored by other books. This SolidWorks 2011 Assemblies Bible shows you how to organize parts data to create assemblies or subassemblies using the latest version of the 3D solid modeling program, SolidWorks. Thoroughly describes best practices and beginning-to-advanced techniques using both video and text. Explains and thoroughly covers every assembly function and is written in a way that enables the reader to make better decisions while using the software. Written by well-known and well-respected SolidWorks guru Matt Lombard. Can stand alone or also with the SolidWorks 2011 Parts Bible.

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for a complete SolidWorks reference set Keep both the SolidWorks 2011 Assemblies Bible and the SolidWorks 2011 Parts Bible on your desk, and you'll have the best resource set out there on SolidWorks.

Beginner's Guide to SOLIDWORKS 2021 – Level II starts where Beginner's Guide – Level I ends, following the same easy to read style and companion video instruction, but this time covering advanced topics and techniques. The purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create components in the context of an assembly and reference other components (Top-down design), propagate design changes with SOLIDWORKS' parametric capabilities, mold design, welded structures and more while explaining the basic concepts of each trade to allow you to understand the how and why of each operation. The author uses simple examples to allow you to better understand each command and environment, as well as to make it easier to explain the purpose of each step, maximizing the learning time by focusing on one task at a time. This book is focused on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. At the end of this book, you will have acquired enough skills to be highly competitive when it comes to designing with SOLIDWORKS, and

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

This book will teach you everything you need to know to start using SOLIDWORKS 2019 with easy to understand, step-by-step tutorials. This book features a simple robot design used as a project throughout the book. You will learn to model parts, create assemblies, run simulations and even create animations of your robot design. No previous experience with Computer Aided Design (CAD) is needed since this book starts at an introductory level. The author begins by getting you familiar with the SOLIDWORKS interface and its basic tools right

away. You will start by learning to model simple robot parts and before long you will graduate to creating more complex parts and multi-view drawings. Along the way you will learn the fundamentals of parametric modeling through the use of geometric constraints and relationships. You will also become familiar with many of SOLIDWORKS's powerful tools and commands that enable you to easily construct complex features in your models. Also included is coverage of gears, gear trains and spur gear creation using SOLIDWORKS. This book continues by examining the different mechanisms commonly used in walking robots. You will learn the basic types of planar four-bar linkages commonly used in mechanical designs and how to use the GeoGebra Dynamic Geometry software to simulate and analyze 2D linkages. Using the knowledge you gained about linkages and mechanisms, you will learn how to modify your robot and change its behavior by modifying or creating new parts. In the second to last chapter of this book you learn how to combine all the robot parts into assemblies and then run motion analysis. You will finish off your project by creating 3D animations of your robot in action. Finally, in the last chapter, the author introduces you to 3D printing. You will learn the general principles of 3D printing including a brief history of 3D printing, the types of 3D printing technologies, commonly used filaments, and the

basic procedure for printing a 3D model. Being able to turn your designs into physical objects will open up a whole new world of possibilities to you. There are many books that show you how to perform individual tasks with SOLIDWORKS, but this book takes you through an entire project and shows you the complete engineering process. By the end of this book you will have modeled and assembled nearly all the parts that make up the TAMIYA® Mechanical Tiger and can start building your own robot.

The SolidWorks CAM 2021 Black Book (Colored) is the 2nd edition of our series on SolidWorks CAM. The book is written to help professionals as well as learners get familiar with functionality of the software. The book follows a step by step methodology. In this book, we have tried to give real-world examples with real challenges in manufacturing design. We have tried to reduce the gap between university use and industrial use of SolidWorks CAM. The book covers almost all the information required by a learner to master SolidWorks CAM. Some of the salient features of this book are: In-Depth explanation of concepts Every new topic of this book starts with the explanation of the basic concepts. In this way, the user becomes capable of relating the things with real world. Topics Covered Every chapter starts with a list of topics being covered in that chapter. In this way, the user can easy find the topic of his/her

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interest easily. Instruction through illustration The instructions to perform any action are provided by maximum number of illustrations so that the user can perform the actions discussed in the book easily and effectively. There are about 400 small and large illustrations that make the learning process effective. Tutorial point of view At the end of concept's explanation, the tutorial make the understanding of users firm and long lasting. Most of the tools in this book are discussed in the form of tutorials. Project Free projects and exercises are provided to students for practicing. For Faculty If you are a faculty member, then you can ask for video tutorials on any of the topic, exercise, tutorial, or concept.

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rest of the book introduces you to developing macros using the SOLIDWORKS API. The book concludes with a chapter dedicated to some of the author's favorite source code for you to use as the basis for typical automation procedures. The focus of this book is primarily on the Visual Studio Tools for Applications (VSTA) macro interface. It covers many of the major API functions through practical use cases. It will teach you the fundamentals of Visual Basic.NET as well as SOLIDWORKS, SOLIDWORKS PDM Professional, SOLIDWORKS Document Manager and Excel API functions. Author Mike Spens has been professionally developing macros for SOLIDWORKS for more than a decade. He has helped numerous companies develop their own programs and streamline their workflows. If you want to learn how to develop your own macros for SOLIDWORKS, following best practices and using well written code, then this is the perfect book for you.

This senior undergraduate level textbook is written for Advanced Manufacturing, Additive Manufacturing, as well as CAD/CAM courses. Its goal is to assist students in colleges and universities, designers, engineers, and professionals interested in using SolidWorks as the design and 3D printing tool for emerging manufacturing technology for practical applications. This textbook will bring a new dimension to SolidWorks by introducing readers to the role of SolidWorks in the relatively new manufacturing paradigm shift, known as 3D-Printing which is based on Additive Manufacturing (AM) technology. This new textbook: Features modeling of complex parts and surfaces Provides a step-by-step tutorial type approach with pictures showing how to model using SolidWorks Offers a user-Friendly approach for the design of parts, assemblies, and drawings, motion-analysis, and FEA topics Includes clarification of connections between SolidWorks and 3D-Printing based on Additive Manufacturing

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Discusses a clear presentation of Additive Manufacturing for Designers using SolidWorks CAD software "Introduction to SolidWorks: A Comprehensive Guide with Applications in 3D Printing" is written using a hands-on approach which includes a significant number of pictorial descriptions of the steps that a student should follow to model parts, assemble parts, and produce drawings.

The complete SolidWorks reference-tutorial for beginner to advanced techniques Mastering SolidWorks is the reference-tutorial for all users. Packed with step-by-step instructions, video tutorials for over 40 chapters, and coverage of little-known techniques, this book takes you from novice to power user with clear instruction that goes beyond the basics.

Fundamental techniques are detailed with real-world examples for hands-on learning, and the companion website provides tutorial files for all exercises. Even veteran users will find value in new techniques that make familiar tasks faster, easier, and more organized, including advanced file management tools that simplify and streamline pre-flight checks. SolidWorks is the leading 3D CAD program, and is an essential tool for engineers, mechanical designers, industrial designers, and drafters around the world. User friendly features such as drag-and-drop, point-and-click, and cut-and-paste tools belie the software's powerful capabilities that can help you create cleaner, more precise, more polished designs in a fraction of the time. This book is the comprehensive reference every SolidWorks user needs, with tutorials, background, and more for beginner to advanced techniques. Get a grasp on fundamental SolidWorks 2D and 3D tasks using realistic examples with text-based tutorials Delve into advanced functionality and capabilities not commonly covered by how-to guides Incorporate improved search, Pack-and-Go and other file management tools into your workflow Adopt best practices and exclusive techniques

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you won't find anywhere else Work through this book beginning-to-end as a complete SolidWorks course, or dip in as needed to learn new techniques and time-saving tricks on-demand. Organized for efficiency and designed for practicality, these tips will remain useful at any stage of expertise. With exclusive coverage and informative detail, Mastering SolidWorks is the tutorial-reference for users at every level of expertise.

- Starts at an introductory level, designed for beginners •
- Comprehensive coverage of beginning tools and techniques •
- Uses a step by step, tutorial approach with real world projects •
- Covers the creation of parts, assemblies and drawings •

Features a quick reference guide and a Certified SOLIDWORKS Associate practice exam • The first book of a three book series SOLIDWORKS 2021 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. Who this book is for This book is for the beginner who is not familiar with the SOLIDWORKS program and its add ins.

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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.

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GeoGebra Dynamic Geometry software to simulate and analyze 2D linkages. Using the knowledge you gained about linkages and mechanisms, you will learn how to modify your robot and change its behavior by modifying or creating new parts. In the second to last chapter of this book you learn how to combine all the robot parts into assemblies and then run motion analysis. You will finish off your project by creating 3D animations of your robot in action. Finally, in the last chapter, the author introduces you to 3D printing. You will learn the general principles of 3D printing including a brief history of 3D printing, the types of 3D printing technologies, commonly used filaments, and the basic procedure for printing a 3D model. Being able to turn your designs into physical objects will open up a whole new world of possibilities to you. There are many books that show you how to perform individual tasks with SOLIDWORKS, but this book takes you through an entire project and shows you the complete engineering process. By the end of this book you will have modeled and assembled nearly all the parts that make up the TAMIYA® Mechanical Tiger and can start building your own robot.

SOLIDWORKS 2017 and Engineering Graphics: An Integrated Approach combines an introduction to SOLIDWORKS 2017 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 2017 and Engineering Graphics: An Integrated Approach is to introduce the aspects of Engineering Graphics with the use of modern Computer Aided Design package – SOLIDWORKS

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2017. 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 sixteen chapters, with detailed step-by-step tutorial style lessons, designed to introduce beginning CAD users to the graphics language used in all branches of technical industry. This book does not attempt to cover all of SOLIDWORKS 2017'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. SOLIDWORKS 2020 Quick Start introduces new users to the basics of using SOLIDWORKS 3D CAD software in five easy lessons. This book is intended for the student or designer who needs to learn SOLIDWORKS quickly and effectively. This book is perfect for engineers in industry who are expected to have SOLIDWORKS skills for their company's next project or students who need to learn SOLIDWORKS without taking a comprehensive CAD course. Based on years of teaching SOLIDWORKS to engineering students, SOLIDWORKS 2020 Quick Start concentrates on the areas where new users can improve 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. This book begins with an overview of SOLIDWORKS and the User Interface (UI), its menus, toolbars and commands. With a

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quick pace, you learn the essentials of 2D sketching, part and assembly creation, perform motion study, develop detailed part and assembly drawings and much more. Throughout this book you develop a mini Stirling Engine and investigate the proper design intent and constraints. SOLIDWORKS 2018 Tutorial with video instruction is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The text provides a step-by-step, project based learning approach. It also contains information and examples on the five categories, to take and understand the Certified Associate - Mechanical Design (CSWA) exam. The book is divided into four sections. Chapters 1 - 5 explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. In chapter 6 you will create the final robot assembly. The physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and components for the final robot assembly are provided. Chapters 7 - 10 prepare you for the Certified Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs from subtractive manufacturing, and its features. You will also learn the terms and technology used in low cost 3D

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printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, apply proper design intent, design tables and configurations. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each chapter. Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

SOLIDWORKS 2016: A Power Guide for Beginners and Intermediate Users textbook is designed for instructor-led courses as well as for self-paced learning. This textbook is intended to help engineers and designers who are interested in learning SOLIDWORKS for creating 3D mechanical designs. It will be a great starting point for new SOLIDWORKS users and a great teaching aid in classroom training. This textbook contains 13 chapters which consist of 758 pages covering major environments of SOLIDWORKS: Part, Assembly, and Drawing, which teaches you how to use the SOLIDWORKS mechanical design software to build parametric models and assemblies, and how to make drawings of parts and assemblies. Every chapter of this textbook contains tutorials which intend to help users to experience how things can be done in SOLIDWORKS

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step by step. Moreover, every chapter ends with hands-on test drives which allow users to experience themselves the ease-of-use and powerful 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 Assemblies - I Chapter 12. Working with Assemblies - II Chapter 13. Working with Drawing

Explore a practical and example-driven approach to understanding SOLIDWORKS 2020 and achieving CSWA and CSWP certification Key Features Gain comprehensive insights into the core aspects of mechanical part modeling Get up to speed with generating assembly designs with both standard and advanced mates Focus on design practices for both 2D as well as 3D modeling and prepare to achieve CWSP and CWSA certification Book Description SOLIDWORKS is the leading choice for 3D engineering and product design applications across industries such as aviation, automobiles, and consumer product design. This book takes a practical approach to getting you up and running with SOLIDWORKS 2020. You'll start with the basics, exploring the software interface and working with drawing files. The book then guides you through topics

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such as sketching, building complex 3D models, generating dynamic and static assemblies, and generating 2D engineering drawings to equip you for mechanical design projects. You'll also do practical exercises to get hands-on with creating sketches, 3D part models, assemblies, and drawings. To reinforce your understanding of SOLIDWORKS, the book is supplemented by downloadable files that will help you follow up with the concepts and exercises found in the book. By the end of this book, you'll have gained the skills you need to create professional 3D mechanical models using SOLIDWORKS, and you'll be able to prepare effectively for the Certified SOLIDWORKS Associate (CSWA) and Certified SOLIDWORKS Professional (CSWP) exams. What you will learn

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

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

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The book starts with sketching and ends at advanced topics like Mold Design, Sheetmetal, Weldment, and SolidWorks MBD. In this edition of book, we have included many new features of SolidWorks like 3D printing, SolidWorks MBD (Model Based Definition), Magnetic Mates, Asset Builder, 3D PDFs, and so on. Mastering SolidWorks: The Design Approach, Second Edition is entirely updated for SolidWorks 2014 and presents SolidWorks as a design system rather than a software program, using design, modeling, and drafting concepts as the building blocks, instead of focusing on menus and commands. It describes design approaches, methodologies, and techniques to help CAD designers/engineers and draftspersons achieve their engineering tasks in the fastest, easiest, and most effective way. It develops command sequences to achieve CAD and modeling tasks, providing SolidWorks syntax and details. Starting with a CAD task to accomplish, the book then goes about how to accomplish it, motivating students to learn more than simply going through layers of menus and commands. Intended for design courses, the book uses a minimal amount of mathematical concepts, covering basic math in Chapter 8 (Curves), Chapter 9 (Surfaces), and Chapter 13 (Analysis Tools). Intended for design courses, the book uses a minimal amount of mathematical concepts, covering basic math in Chapter 8 (Curves), Chapter 9 (Surfaces), and Chapter 13 (Analysis Tools). * Shows concepts to those who are curious about how CAD/CAM systems work "under the hood." * Broadens the book appeal to many students,

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professors, and readers. * The coverage of math in chapters 8, 9, and 13 may be ignored without affecting the continuity of the material in those chapters. Step-by-Step instructions help students learn SolidWorks as a design system rather than a software program. * Ample illustrations guide students as they learn. Tutorials offer comprehensive coverage of a full design task. * Each tutorial ends with a hands-on exercise that both challenges the student's understanding and extends it. Examples with Solutions cover a single concept in detail. * Each example offers a hands-on exercise that builds on the previous example, ensuring the student has gone through each example. Each chapter includes challenging modeling and design examples and problems. * The book's unique approach covers the theoretical concepts behind the various functions of SolidWorks. * This sheds light about why things work the way they do, as well as explains their limitations and uses.

If you want to gain proficiency and expertise with SolidWorks surface modeling, this is the resource for you. You'll learn how to apply concepts, utilize tools, and combine techniques and strategies in hands-on tutorials. This Bible covers the range from sketching splines and shelling to modeling blends and decorative features. Complete with professional tips and real-world examples, this inclusive guide enables you to coax more out of SolidWorks surfacing tools.

Whether it's your first venture into 3D technical drawing software or you're switching to SolidWorks from something else, you're probably excited about what this

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CAD program has to offer. Chances are, you figure it's going to take awhile to get the hang of it before you can begin cranking out those perfectly precise 3D designs. SolidWorks For Dummies, 2nd Edition, can help you dramatically shorten that get-acquainted period! SolidWorks For Dummies, 2nd Edition will help you get up and running quickly on the leading 3D technical drawing software. You'll see how to set up SolidWorks to create the type of drawings your industry requires and how to take full advantage of its legendary 3D features. You'll discover how to:

- Work with virtual prototypes
- Understand the user interface
- Use templates and sketch, assemble, and create drawings
- Automate the drawing process
- Review drawings and collaborate with other team members
- Define and edit sketches
- Create dimensions and annotations
- Print or plot your drawings
- Leverage existing designs

Sample files on the bonus CD-ROM show you how to apply the latest version of SolidWorks and accomplish specific tasks. Even if you're brand-new to CAD software, SolidWorks For Dummies, 2nd Edition will have you feeling like a pro in no time. You'll find you've entered a whole new dimension. Note: CD-ROM/DVD and other supplementary materials are not included as part of eBook file.

The SolidWorks 2015 Black Book is second edition of our series on SolidWorks. With lots of additions and thorough review, we present a book to help professionals as well as learners in creating some of the most complex solid models. The book follows a

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step by step methodology. In this book, we have tried to give real-world examples with real challenges in designing. We have tried to reduce the gap between university use of SolidWorks and industrial use of SolidWorks. The book covers almost all the information required by a learner to master the SolidWorks. The book starts with sketching and ends at advanced topics like mold design, sheetmetal, and Weldment. Some of the salient features of this book are:

- In-Depth explanation of concepts Every new topic of this book starts with the explanation of the basic concepts. In this way, the user becomes capable of relating the things with real world.
- Topics Covered Every chapter starts with a list of topics being covered in that chapter. In this way, the user can easy find the topic of his/her interest easily.
- Instruction through illustration The instructions to perform any action are provided by maximum number of illustrations so that the user can perform the actions discussed in the book easily and effectively. There are about 900 illustrations that make the learning process effective.
- Tutorial point of view At the end of concept's explanation, the tutorial make the understanding of users firm and long lasting. Almost each chapter of the book has tutorials that are real world projects. Project Free projects and exercises are provided to students for practicing.
- For Faculty If you are a faculty member, then you can ask for video tutorials on any of the topic,

exercise, tutorial, or concept.

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

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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.

The SolidWorks Simulation 2020 Black Book, 7th edition is written for professionals and students of Finite Element Analysis field. The book starts with basics of FEA, goes through all the simulation tools and ends up with practical examples of analysis with explanation of Solver selection, iteration methods and integration techniques.

A comprehensive resource packed with information for both beginners and advanced users SolidWorks is the leading 3D solid modeling software used in computer-aided design. It's powerful but not simple. This complete guide introduces beginners to the software but then goes far beyond, covering

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numerous details that advanced users have requested. Beginners will learn not only how the software works but why, while more experienced users will learn all about search criteria, Pack-and-Go, other file management concepts, and much more. A valuable companion website contains before and after real-world parts and assemblies along with many example files used in the text. Additionally, the text of the book is augmented by video tutorials with author voice-over which can be found on the website. SolidWorks is the leading 3D CAD program, and previous editions of this book have sold more than 33,000 copies Covers necessary information to give beginners a solid foundation in the software, including part and assembly modeling and 2D drawing techniques Addresses a wide range of advanced topics not treated in other books, including best practices, search criteria, Pack-and-Go, and other file management concepts Includes tutorials on both beginning and advanced topics, with videos; sample part, assembly, and drawing files; and before-and-after example files available on the companion website SolidWorks 2013 Bible is the ultimate resource on SolidWorks 2013, the book beginners can start with and advanced users will want to keep close at hand.

Parametric Modeling with SOLIDWORKS 2020 contains a series of seventeen tutorial style lessons designed to introduce SOLIDWORKS 2020, solid

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modeling and parametric modeling techniques and concepts. This book introduces SOLIDWORKS 2020 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 2020, 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. This book also introduces you to the general principles of 3D printing including a brief history of 3D printing, the types of 3D printing technologies, commonly used filaments, and the basic procedure for printing a 3D model. 3D printing makes it easier than ever for anyone to start turning their designs into physical objects and by the end of this book you will be ready

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to start printing out your own designs.

Mastering SolidWorks: The Design Approach, Second Edition is entirely updated for SolidWorks 2014 and presents SolidWorks as a design system rather than a software program, using design, modeling, and drafting concepts as the building blocks, instead of focusing on menus and commands. It describes design approaches, methodologies, and techniques to help CAD designers/engineers and draftspersons achieve their engineering tasks in the fastest, easiest, and most effective way. It develops command sequences to achieve CAD and modeling tasks, providing SolidWorks syntax and details. Starting with a CAD task to accomplish, the book then goes about how to accomplish it, motivating students to learn more than simply going through layers of menus and commands. Intended for design courses, the book uses a minimal amount of mathematical concepts, covering basic math in Chapter 8 (Curves), Chapter 9 (Surfaces), and Chapter 13 (Analysis Tools). Intended for design courses, the book uses a minimal amount of mathematical concepts, covering basic math in Chapter 8 (Curves), Chapter 9 (Surfaces), and Chapter 13 (Analysis Tools). • Shows concepts to those who are curious about how CAD/CAM systems work "under the hood." • Broadens the book appeal to many students, professors, and readers. • The coverage of math in

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chapters 8, 9, and 13 may be ignored without affecting the continuity of the material in those chapters. Step-by-Step instructions help students learn SolidWorks as a design system rather than a software program. • Ample illustrations guide students as they learn. Tutorials offer comprehensive coverage of a full design task. • Each tutorial ends with a hands-on exercise that both challenges the student's understanding and extends it. Examples with Solutions cover a single concept in detail. • Each example offers a hands-on exercise that builds on the previous example, ensuring the student has gone through each example. Each chapter includes challenging modeling and design examples and problems. • The book's unique approach covers the theoretical concepts behind the various functions of SolidWorks. • This sheds light about why things work the way they do, as well as explains their limitations and uses.

Engineering Graphics with SOLIDWORKS 2020 is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The book combines the fundamentals of engineering graphics and dimensioning practices with a step-by-step project based approach to learning SOLIDWORKS. The book is divided into four sections with 11 Chapters. Chapters 1 - 3: Explore the history of engineering graphics, manual

sketching techniques, orthographic projection, Third vs. First angle projection, multi-view drawings, dimensioning practices (ASME Y14.5-2009 standard), line type, fit type, tolerance, fasteners in general, general thread notes and the history of CAD leading to the development of SOLIDWORKS.

Chapters 4 - 9: Comprehend the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple machine parts, simple and complex assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. Follow the step-by-step instructions in over 80 activities to develop eight parts, four sub-assemblies, three drawings and six document templates. Chapter 10: Prepare for the Certified SOLIDWORKS Associate (CSWA) exam. Understand the curriculum and categories of the CSWA exam and the required model knowledge needed to successfully take the exam. Chapter 11: Provide a basic understanding between Additive vs. Subtractive manufacturing. Discuss Fused Filament Fabrication (FFF), STereoLithography (SLA), and Selective Laser Sintering (SLS) printer technology. Select suitable filament material. Comprehend 3D printer terminology. Knowledge of preparing, saving, and printing a model on a Fused Filament Fabrication 3D printer. Information on the Certified SOLIDWORKS Additive Manufacturing (CSWA-AM)

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exam. Review individual features, commands, and tools using SOLIDWORKS Help. The chapter exercises analyze and examine usage competencies based on the chapter objectives. The book is designed to complement the SOLIDWORKS Tutorials located in the SOLIDWORKS Help menu. 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 author developed the industry scenarios by combining his own industry experience with the knowledge of engineers, department managers, vendors and manufacturers.

The SolidWorks Electrical 2021 Black Book is, 7th edition of SolidWorks Electrical Black Book, written to help professionals as well as learners in performing various tedious jobs in Electrical control designing. The book follows the best proven step by step methodology. This book is more concentrated on making you able to use tools at right places. The book starts with basics of Electrical Designing, goes through all the Electrical controls related tools and ends up with practical examples of electrical schematics. Chapters also cover Reports that make you comfortable in creating and editing electrical component reports. There are two annexures added

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to explain basic concepts of control panel designing.

Some of the salient features of this book are: In-Depth explanation of concepts Every new topic of this book starts with the explanation of the basic concepts. In this way, the user becomes capable of relating the things with real world. Topics Covered Every chapter starts with a list of topics being covered in that chapter. In this way, the user can easily find the topic of his/her interest easily.

Instruction through illustration The instructions to perform any action are provided by maximum number of illustrations so that the user can perform the actions discussed in the book easily and effectively. There are about 650 illustrations that make the learning process effective. Tutorial point of view The book explains the concepts through the tutorial to make the understanding of users firm and long lasting. Each chapter of the book has tutorials that are real world projects. Project Free projects and exercises are provided to students for practicing. For Faculty If you are a faculty member, then you can ask for video tutorials on any of the topic, exercise, tutorial, or concept.

This book provides engineering professionals and students with a comprehensive introduction to the popular SolidWorks software. The author takes care to guide readers from beginning through increasingly advanced SolidWorks functionality, and shows how to apply the software to a multitude of widely

divergent manufacturing processes.

The SolidWorks 2020 Black Book is the 7th edition of our series on SolidWorks. With lots of additions and thorough review, we present a book to help professionals as well as learners in creating some of the most complex solid models. The book follows a step by step methodology. In this book, we have tried to give real-world examples with real challenges in designing. We have tried to reduce the gap between university use of SolidWorks and industrial use of SolidWorks. In this edition of book, we have included many new features of SolidWorks like Sketch Ink, Silhouette Entities, 3D Textures, Mesh Modeling, DriveWorksXpress, Markup, SolidWorks Inspection, and so on. New practice questions have been added in this edition. The book covers almost all the information required by a learner to master the SolidWorks. The book starts with sketching and ends at advanced topics like Mold Design, Sheetmetal, Weldment, SolidWorks CAM, Rendering, and MBD. In-Depth explanation of concepts Every new topic of this book starts with the explanation of the basic concepts. In this way, the user becomes capable of relating the things with real world. Topics Covered Every chapter starts with a list of topics being covered in that chapter. In this way, the user can easy find the topic of his/her interest easily. Instruction through illustration The instructions to perform any action are provided by

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maximum number of illustrations so that the user can perform the actions discussed in the book easily and effectively. There are about 1350 illustrations that make the learning process effective. Tutorial point of view At the end of concept's explanation, the tutorial make the understanding of users firm and long lasting. Almost each chapter of the book has tutorials that are real world projects. Moreover most of the tools in this book are discussed in the form of tutorials. Project Free projects and exercises are provided to students for practicing. For Faculty If you are a faculty member, then you can ask for video tutorials on any of the topic, exercise, tutorial, or concept. New Addition If anything is added in this edition but is not available in the previous editions, then it is displayed with New symbol in table of content.

"The most complete resource for SolidWorks on the market. Matt Lombard's in-depth knowledge plus his snappy wit and wisdom make SolidWorks accessible to users at all levels." -- Mike Sabocheck, Territory Technical Manager, SolidWorks Corporation The most comprehensive single reference on SolidWorks Whether you're a new, intermediate, or professional user, you'll find the in-depth coverage you need to succeed with SolidWorks 2007 in this comprehensive reference. From customizing the interface to exploring best practices to reinforcing your knowledge with step-by-step tutorials, the

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techniques and shortcuts in this detailed book will help you accomplish tasks, avoid the time-consuming pitfalls of parametric design, and get a firm handle on one of the leading 3D CAD programs on the market. * Customize the user interface and connect hotkeys to macros * Create sketches, parts, assemblies, and drawings * Build intelligence into parts * Work with patterns, equations, and configurations * Learn multibody, surface, and master model techniques * Write, record, and edit Visual Basic(r) macros Design with advanced 3D features Increase speed and efficiency with subassemblies Use multibody models to their full potential What's on the CD-ROM? The CD includes all the parts, assemblies, drawings, and examples you need to follow the tutorials in each chapter. You'll also find finished models, templates, and more. See the CD appendix for details and complete system requirements

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.

This unique text presents a thorough introduction to SolidWorks for anyone with little or no prior experience with this software. It can be used in virtually any educational - setting from four-year engineering schools to community colleges and voc/tech schools to industrial training centers - and

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will also serve as a reliable reference for on-the-job use or as a self-study manual that can be used with little or no outside help. Unlike other books of its kind, it begins at a very basic level and ends at a fairly advanced level. It's perfect for anyone interested in learning SolidWorks quickly and effectively.

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