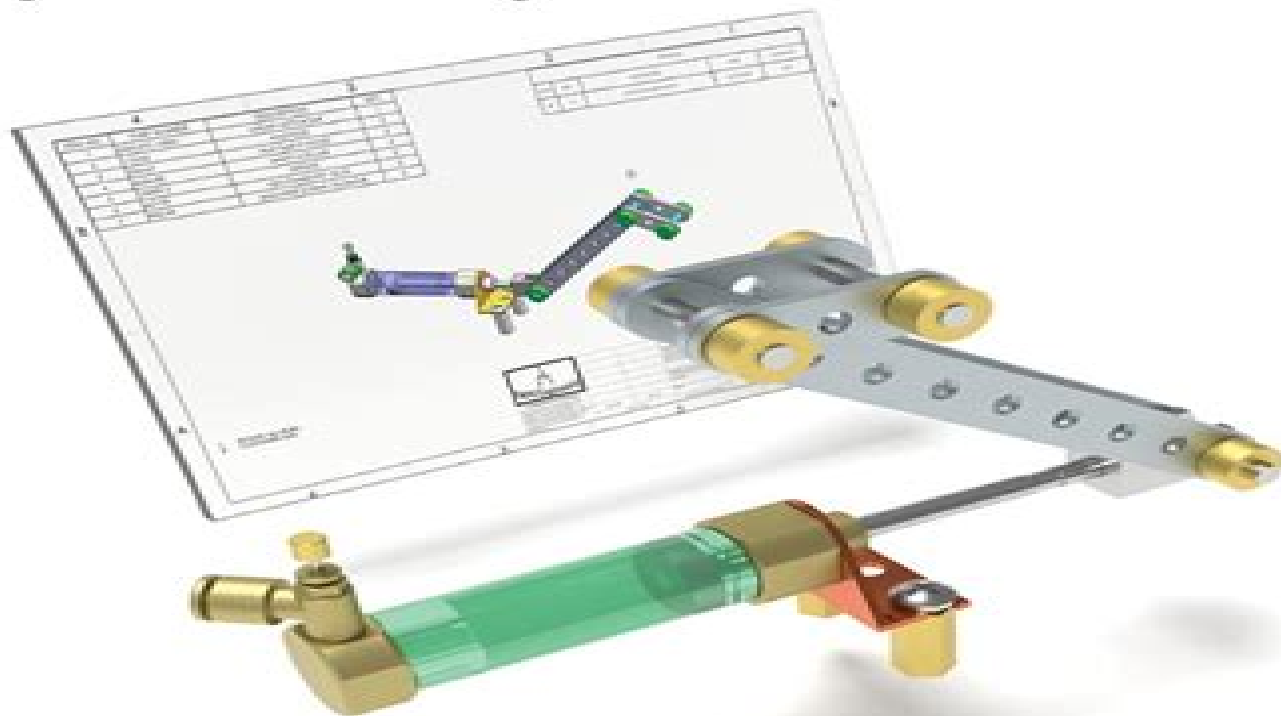


SOLIDWORKS 2025 Tutorial

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Utilizing 3D Solid Modeling

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Solidworks 2015 Tutorials Guide

David Planchard

A red circular graphic with a gradient, appearing as a partial circle or a thick arc, located to the right of the author's name.

Solidworks 2015 Tutorials Guide:

Beginner's Guide to SolidWorks 2015 - Level I Alejandro Reyes, 2015-01-14 This book is intended to help new users learn the basic concepts of SolidWorks and good solid modeling techniques in an easy to follow guide that includes video instruction It is a great starting point for those new to SolidWorks or as a teaching aid in classroom training to become familiar with the software's interface basic commands and strategies as users complete a series of models while learning different ways to accomplish a particular task At the end of this book you will have a fairly good understanding of the SolidWorks interface and the most commonly used commands for part modeling assembly and detailing after completing a series of components and their 2D drawings complete with Bill of Materials The book focuses on the processes to complete the modeling of a part instead of focusing on individual software commands or operations which are generally simple enough to learn The author strived hard to include the commands required in the Certified SolidWorks Associate test as listed on the SolidWorks website as well as several more SolidWorks is an easy to use CAD software that includes many time saving tools that will enable new and experienced users to complete design tasks faster than before Most commands covered in this book have advanced options which may not be covered in this book This is meant to be a starting point to help new users to learn the basic and most frequently used commands

SolidWorks 2015 Reference Guide David Planchard, 2014-11-02 The SolidWorks 2015 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2015 SolidWorks is an immense software package and no one book can cover all topics for all users This book provides a centralized reference location to address many of the tools features and techniques of SolidWorks 2015 This book covers the following System and Document

propertiesFeatureManagersPropertyManagersConfigurationManagersRenderManagers2D and 3D Sketch toolsSketch entities3D Feature toolsMotion StudySheet MetalMotion StudySolidWorks SimulationPhotoView 360Pack and Go3D PDFsIntelligent Modeling techniques3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2015 software If you are completely new to SolidWorks you should read Chapter 1 in detail and complete Lesson 1 Lesson 2 and Lesson 3 in the SolidWorks Tutorials If you are familiar with an earlier release of SolidWorks you still might want to skim Chapter 1 to become acquainted with some of the commands menus and features that you have not used or you can simply jump to any section in any chapter Each chapter provides detailed PropertyManager information on key topics with individual stand alone short tutorials to reinforce and demonstrate the functionality and ease of the SolidWorks tool or feature The book provides access to over 240 models their solutions and additional support materials Learn by doing not just by reading Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components design tables configurations and more The book is designed to compliment the Online Tutorials and Online Help

contained in SolidWorks 2015 The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs The author developed the tutorials by combining his own industry experience with the knowledge of engineers department managers professors vendors and manufacturers He is directly involved with SolidWorks every day and his responsibilities go far beyond the creation of just a 3D model

SolidWorks 2015 Tutorial with Video

Instruction David Planchard,2014-12 SolidWorks 2015 Tutorial with video instruction is target towards a technical school two year college four year university or industry professional that is a beginner or intermediate CAD user The text provides a student who is looking for a step by step project based approach to learning SolidWorks with video instruction SolidWorks model files and preparation for the Certified Associate Mechanical Design CSWA exam The book is divided into three sections Chapters 1 6 explore 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 Revision tables using basic and advanced features 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 Review Chapter 11 on Additive Manufacturing 3D printing and its benefits and features Understand the terms and technology used in low cost 3D printers Follow the step by step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components apply proper design intent design tables and configurations Learn by doing not just by reading Desired outcomes and usage competencies are listed for each chapter Know your objective up front Follow the steps in each chapter to achieve your design goals Work between multiple documents features commands custom properties and document properties that represent how engineers and designers utilize SolidWorks in industry

SOLIDWORKS 2015 in 5 Hours with Video Instruction David Planchard,2015-02 OLIDWORKS 2015 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 2015 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 SOLIDWORKS 2019 Tutorial David Planchard, 2018-12-21 SOLIDWORKS 2019 Tutorial is written to assist students designers engineers and professionals who are new to SOLIDWORKS The text provides a step by step project based learning approach It also contains information and examples on the five categories in the CSWA exam The book is divided into four sections Chapters 1 5 explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple and complex parts and assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and Revision tables using basic and advanced features In chapter 6 you will create the final robot assembly The physical components and corresponding Science Technology Engineering and Math STEM curriculum are available from Gears Educational Systems All assemblies and components for the final robot assembly are provided Chapters 7 10 prepare you for the Certified Associate Mechanical Design CSWA exam The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles Chapter 11 covers the benefits of additive manufacturing 3D printing how it differs from subtractive manufacturing and its features You will also learn the terms and technology used in low cost 3D printers Follow the step by step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components apply proper design intent design tables and configurations Learn by doing not just by reading Desired outcomes and usage competencies are listed for each chapter Know your objective up front Follow the steps in each chapter to achieve your design goals Work between multiple documents features commands custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry *Official Guide to Certified SOLIDWORKS Associate Exams: CSWA, CSWA-SD, CSWA-S, CSWA-AM (SOLIDWORKS 2019 - 2021)* David Planchard, 2020-11 This book is written to assist you with passing the SOLIDWORKS associate level exams It provides you with detailed information and exercises that will aid you in passing the following exams Certified SOLIDWORKS Associate CSWA Certified SOLIDWORKS Associate Sustainable Design CSWA SD Certified SOLIDWORKS Associate Simulation CSWA S and the Certified SOLIDWORKS Associate Additive Manufacturing CSWA AM exam There are three goals for this book The primary goal of this book is not only to help you pass the CSWA CSWA SD CSWA S and CSWA AM exams but also to ensure that you understand and comprehend the concepts and implementation details of the four certification processes The second goal is to provide the most comprehensive coverage of

CSWA CSWA SD CSWA S and CSWA AM exam related topics available without too much coverage of topics not on the exam

The third and ultimate goal is to get you from where you are today to the point that you can confidently pass the CSWA

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

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

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

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

Engineering Design with SolidWorks 2015 and Video

Instruction David Planchard,2014-11-28 Engineering Design with SolidWorks 2015 and video instruction is written to assist

students designers engineers and professionals The book provides a solid foundation in SolidWorks by utilizing projects with step by step instructions for the beginner to intermediate SolidWorks user Explore the user interface CommandManager menus toolbars and modeling techniques to create parts assemblies and drawings in an engineering environment Follow the step by step instructions and develop multiple parts and assemblies that combine machined plastic and sheet metal components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components Design Tables Bills of Materials Custom Properties and Configurations Address various SolidWorks analysis tools and Intelligent Modeling techniques along with Additive Manufacturing 3D printing Learn by doing not just by reading Desired outcomes and usage competencies are listed for each project Know your objective up front Follow the steps in Projects 1 9 to achieve the design goals Review Project 10 on Additive Manufacturing 3D printing and its benefits and features Understand the terms and technology used in low cost 3D printers Work between multiple documents features commands and custom properties that represent how engineers and designers utilize SolidWorks in industry Review individual features commands and tools with the Video Instruction The projects contain exercises The exercises analyze and examine usage competencies Collaborate with leading industry suppliers such as SMC Corporation of America Boston Gear and 80 20 Inc Collaborative information translates into numerous formats such as paper drawings electronic files rendered images and animations On line intelligent catalogs guide designers to the product that meets both their geometric requirements and performance functionality The author developed the industry scenarios by combining his own industry experience with the knowledge of engineers department managers vendors and manufacturers These professionals are directly involved with SolidWorks every day Their responsibilities go far beyond the creation of just a 3D model The book is designed to compliment the SolidWorks Tutorials contained in SolidWorks 2015 View the provided videos in the book to enhance the user experience SolidWorks Interface 2D Sketching Sketch Planes and Sketch tools 3D Features and Design Intent Creating an Assembly Fundamentals in Drawings Part 1 Part 2

SolidWorks 2015 Part II - Advanced Techniques Paul Tran, 2014-12-12 SolidWorks 2015 Part II Advanced Techniques picks up where SolidWorks 2015 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 Who this book is for This book is for the intermediate

user who has already completed the Part I Basic Tools book or someone who is very familiar with the SolidWorks Program and its add ins **SOLIDWORKS 2018 Tutorial with Video Instruction** David Planchard,2018 SOLIDWORKS 2018 Tutorial with video instruction is written to assist students designers engineers and professionals who are new to SOLIDWORKS The text provides a step by step project based learning approach It also contains information and examples on the five categories to take and understand the Certified Associate Mechanical Design CSWA exam The book is divided into four sections Chapters 1 5 explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple and complex parts and assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and Revision tables using basic and advanced features In chapter 6 you will create the final robot assembly The physical components and corresponding Science Technology Engineering and Math STEM curriculum are available from Gears Educational Systems All assemblies and components for the final robot assembly are provided Chapters 7 10 prepare you for the Certified Associate Mechanical Design CSWA exam The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles Chapter 11 covers the benefits of additive manufacturing 3D printing how it differs from subtractive manufacturing and its features You will also learn the terms and technology used in low cost 3D printers Follow the step by step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components apply proper design intent design tables and configurations Learn by doing not just by reading Desired outcomes and usage competencies are listed for each chapter Know your objective up front Follow the steps in each chapter to achieve your design goals Work between multiple documents features commands custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry *SOLIDWORKS 2020 Tutorial* David Planchard,2019-12 Uses step by step project based tutorials designed for beginning or intermediate users Will prepare you for the Certified SOLIDWORKS Associate Exam Includes a chapter introducing you to 3D printing SOLIDWORKS 2020 Tutorial is written to assist students designers engineers and professionals who are new to SOLIDWORKS The text provides a step by step project based learning approach It also contains information and examples on the five categories in the CSWA exam The book is divided into four sections Chapters 1 5 explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple and complex parts and assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and Revision tables using basic and advanced features In chapter 6 you will create the final robot assembly The physical components and corresponding Science Technology Engineering and Math STEM curriculum are available from Gears Educational Systems All assemblies and components for the final robot assembly are provided Chapters 7 10 prepare you for the Certified Associate Mechanical Design CSWA exam The certification indicates a foundation in and apprentice

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SOLIDWORKS 2022 Tutorial David Planchard, 2022-04 Uses step by step project based tutorials designed for beginning or intermediate users Will prepare you for the Certified SOLIDWORKS Associate Exam Includes a chapter introducing you to 3D printing SOLIDWORKS 2022 Tutorial is written to assist students designers engineers and professionals who are new to SOLIDWORKS The text provides a step by step project based learning approach It also contains information and examples on the five categories in the CSWA exam The book is divided into four sections Chapters 1 5 explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple and complex parts and assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and Revision tables using basic and advanced features In chapter 6 you will create the final robot assembly The physical components and corresponding Science Technology Engineering and Math STEM curriculum are available from Gears Educational Systems All assemblies and components for the final robot assembly are provided Chapters 7 10 prepare you for the Certified Associate Mechanical Design CSWA exam The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles Chapter 11 covers the benefits of additive manufacturing 3D printing how it differs from subtractive manufacturing and its features You will also learn the terms and technology used in low cost 3D printers Follow the step by step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components apply proper design intent design tables and configurations Learn by doing not just by reading Desired outcomes and usage competencies are listed for each chapter Know your objective up front Follow the steps in each chapter to achieve your design goals Work between multiple documents features commands custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry

SOLIDWORKS 2025 Tutorial David Planchard, Uses step by step project based tutorials designed for beginning or intermediate users Will prepare you for the Certified SOLIDWORKS Associate Exam Includes a chapter introducing you to 3D printing Features a bonus eBook on SOLIDWORKS and the 3DEXPERIENCE

platform Get ready to take your 3D CAD skills to the next level with SOLIDWORKS 2025 Tutorial Whether you re a student designer engineer or professional who s new to SOLIDWORKS this book is the ultimate guide to mastering SOLIDWORKS impressive capabilities And if you re preparing for the Certified SOLIDWORKS Associate Mechanical Design CSWA exam you re in luck because this book has got you covered Featuring a project based learning approach and step by step instructions the first six chapters cover the User Interface CommandManager Document and System properties and beyond with exploration of everything from design intent and design tables to configurations multi sheet drawings BOMs and Revision tables Use basic and advanced features to create simple and complex parts and assemblies And for the grand finale chapter 6 takes you through the creation of a robot assembly complete with all the assemblies and components you ll need Information and examples on the five categories in the CSWA exam are embedded throughout the book but chapters 7 10 specifically focus on preparation for the Certified SOLIDWORKS Associate Mechanical Design CSWA exam which will confirm you have a foundation in and apprentice knowledge of 3D CAD and engineering principles And for those looking to explore the exciting world of additive manufacturing 3D printing chapter 11 presents the benefits of 3D printing how it differs from subtractive manufacturing and the terminology and technology used in low cost 3D printers With clear concise instructions and desired outcomes listed for each chapter of the tutorial you ll know exactly what you re working towards every step of the way Work between multiple documents features and commands like a pro Build multiple assemblies that combine over 100 extruded machined parts and components and develop the skills to create modify and edit sketches and solid features Plus you ll learn how to reuse features parts and assemblies through symmetry patterns copied components and more Start learning by doing and become a 3D CAD expert with SOLIDWORKS 2025 Tutorial Includes a Bonus eBook Covering SOLIDWORKS and 3DEXPERIENCE Platform Included with your purchase of this book is a bonus eBook titled SOLIDWORKS and the 3DEXPERIENCE Platform This eBook is an insightful guide that introduces you to the 3DEXPERIENCE Platform and its integration with SOLIDWORKS This resource simplifies complex concepts allowing users to collaborate efficiently in a single modeling environment accessible through the SOLIDWORKS Task Pane The book features nine detailed step by step tutorials complete with models to practice and understand the tools and advantages of using SOLIDWORKS with the 3DEXPERIENCE platform This guide will help you understand the 3DEXPERIENCE Platform s capabilities demonstrating practical real world applications in educational and professional settings It s an essential resource for anyone looking to leverage the full potential of SOLIDWORKS in conjunction with the 3DEXPERIENCE platform [SOLIDWORKS 2016 Tutorial with Video Instruction](#) David Planchard,2016-01-11 SOLIDWORKS 2016 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 video instruction SOLIDWORKS model files and preparation for the Certified Associate Mechanical Design CSWA exam The book is divided

into three sections Chapters 1 6 explore 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 Revision tables using basic and advanced features 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 Review Chapter 11 on Additive Manufacturing 3D printing and its benefits and features Understand the terms and technology used in low cost 3D printers Follow the step by step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components apply proper design intent design tables and configurations Learn by doing not just by reading Desired outcomes and usage competencies are listed for each chapter Know your objective up front Follow the steps in each chapter to achieve your design goals Work between multiple documents features commands custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry

SOLIDWORKS 2015 and Engineering Graphics Randy Shih, 2015 SOLIDWORKS 2015 and Engineering Graphics An Integrated Approach combines an introduction to SOLIDWORKS 2015 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 2015 and Engineering Graphics An Integrated Approach is to introduce the aspects of Engineering Graphics with the use of modern Computer Aided Design package SOLIDWORKS 2015 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 graphics language used in all branches of technical industry This book does not attempt to cover all of SOLIDWORKS 2015'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 2015 Randy Shih, 2015-02-09 This book will teach you everything you need to know to start using SolidWorks 2015 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 mechanism 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 **Official Certified SOLIDWORKS Professional Certification Guide (SOLIDWORKS 2020 - 2023)** David C. Planchard,2022-11 This book will provide you with a wealth of information about the three segments of the CSWP CORE exam The intended audience for this book is a person who has passed the CSWA exam and who has eight or more months of SOLIDWORKS training and usage This guide is not intended to teach you how to use SOLIDWORKS but is written to provide you with CSWP exam tips hints and information on sample questions and categories that are aligned with the exam This guide is written to help you take and pass the CSWP exam The book is organized into three chapters Each chapter is focused on a segment of the CSWP CORE exam This is not intended to be a step by step book Goals of this book The primary goal is not only to help you pass the CSWP CORE exam but also to ensure that you understand and comprehend the concepts and implementation details of the process The second goal is to provide the most comprehensive coverage of CSWP CORE exam related topics available without too much coverage of topics not on the exam The third and ultimate goal is to get you from where you are today to the point that you can confidently pass all three segments of the CSWP CORE exam Who this book is for The intended audience for this book and the CSWP exam is a person who has passed the CSWA exam and who has eight or more months of SOLIDWORKS training and usage However passing the CSWA exam is not a prerequisite for taking the CSWP exam if you are a commercial user in industry For students that take the CSWP exam through their school you must first pass the CSWA exam *Automating SOLIDWORKS 2015*

Using Macros Mike Spens,2015-02 Automating SOLIDWORKS 2015 Using Macros is designed as a tutorial to help beginner to intermediate programmers develop macros for SOLIDWORKS and SOLIDWORKS Workgroup or Enterprise PDM 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 teaches many Visual Basic NET fundamentals as well as SOLIDWORKS SOLIDWORKS Workgroup or Enterprise PDM and Excel API functions The Author has also added a chapter dedicated to some of his favorite source code for you to use as the basis for typical automation procedures What you ll learn Record macrosControl Custom PropertiesCreate parts and featuresBuild assembliesBatch create drawingsExtract information from PDMCreate many other time saving utilities

Engineering & Computer Graphics Workbook Using SOLIDWORKS 2015 Ronald Barr,Davor Juricic,Thomas Krueger,2015 Engineering Computer Graphics Workbook Using SOLIDWORKS 2015 is an exercise based workbook that uses step by step tutorials to cover the fundamentals of SOLIDWORKS 2015 The intended audience is college undergraduate engineering majors but it could also be used in pre college introductory engineering courses or by self learners The text follows an educational paradigm that was researched and developed by the authors over many years The paradigm is based on the concurrent engineering approach to engineering design in which the 3 D solid model data serves as the central hub for all aspects of the design process The workbook systematically instructs the students to develop 3 D models using the rich tools afforded in SOLIDWORKS The exercises then proceed to instruct the students on applications of the solid model to design analysis using finite elements to assembly modeling and checking to kinematic simulation to rapid prototyping and finally to projecting an engineering drawing The workbook is ideally suited for courses in which a reverse engineering design project is assigned This book contains clear and easy to understand instructions that enable the students to robustly learn the main features of SOLIDWORKS with little or no instructor input

SOLIDWORKS 2017 in 5 Hours with Video Instruction David Planchard,2017-02 SOLIDWORKS 2017 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 2017 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

SolidWorks 2015

Part I - Basic Tools Paul Tran, 2014-12-12 SolidWorks 2015 Part I Basic Tools introduces new users to the SolidWorks interface SolidWorks tools and basic modeling techniques It provides readers with a strong understanding of SolidWorks and covers the creation of parts assemblies and drawings Every lesson and exercise in this book was created based on real world projects Each of these projects have been broken down and developed into easy and comprehensible steps for the reader Furthermore at the end of every chapter there are self test questionnaires to ensure that the reader has gained sufficient knowledge from each section before moving on to more advanced lessons This book takes the approach that in order to understand SolidWorks inside and out the reader should create everything from the beginning and take it step by step Who this book is for This book is for the beginner who is not familiar with the SolidWorks program and its add ins

As recognized, adventure as capably as experience not quite lesson, amusement, as without difficulty as pact can be gotten by just checking out a book **Solidworks 2015 Tutorials Guide** moreover it is not directly done, you could receive even more more or less this life, re the world.

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