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SOLIDWORKS 2017 Basic Tools Paul Tran, 2016-11 SOLIDWORKS 2017 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 have 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 *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 [SOLIDWORKS 2018 Basic Tools](#) Paul Tran,2017-10

SOLIDWORKS 2018 Basic Tools is the first book in a three part series It introduces new users to the SOLIDWORKS interface SOLIDWORKS tools and basic modeling techniques It provides you with a strong understanding of SOLIDWORKS and covers the creation of parts assemblies and drawings Every lesson and exercise in this book was created based on real world projects Each of these projects has been broken down and developed into easy and comprehensible steps Furthermore at the end of every chapter there are self test questionnaires to ensure that you have gained sufficient knowledge from each section before moving on to more advanced lessons This book takes the approach that in order to understand SOLIDWORKS inside and out you should create everything from the beginning and take it step by step [Engineering Graphics with SOLIDWORKS 2015 and Video Instruction](#) David Planchard,2015-01-14

Engineering Graphics with SOLIDWORKS 2015 and video instruction is written to assist the technical school two year college four year university instructor student or industry professional that is a beginner or intermediate SOLIDWORKS user The book combines the fundamentals of engineering graphics and dimensioning practices with a step by step project based approach to learning SOLIDWORKS with video instructions Learn by doing not just by reading The book is divided into four sections 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 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 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 provides a section on the Certified Associate Mechanical Design CSWA program with sample exam questions and initial and final SOLIDWORKS models Chapter 11 provides a section on Additive Manufacturing 3D printing and its benefits and features Understand the terms and technology used in low cost 3D printers Review individual features commands and tools using the video instruction and 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 These professionals are directly involved with SOLIDWORKS every day Their responsibilities go far beyond the creation of just a 3D model

Engineering Design with SolidWorks 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 Simulation Xpress 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 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 2024 Basic Tools Paul Tran, 2024 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 2024 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 **Automating SOLIDWORKS 2021 Using Macros** Mike Spens, 2021-03-18 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 2021 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 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

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