

**TUTORIAL BOOKS**



**SOLIDWORKS 2017  
LEARN BY DOING - PART 1**

# Solidworks 2017 Learn By Doing Part Assembly Drawings

**Tutorial Books**



## **Solidworks 2017 Learn By Doing Part Assembly Drawings:**

*Solidworks 2017 Tutorial Books*, 2017-01-30 SOLIDWORKS 2017 Learn by doing introduces new users to mechanical design using SOLIDWORKS and how it can be used to create a variety of models In fourteen tutorial based chapters author guides you through all the necessary commands and options in SOLIDWORKS 2017 from sketching to parametric modeling and finally ending with rendering The commands are presented one step at a time using simple examples The approach used in this book helps you to become a skilled SOLIDWORKS user SOLIDWORKS 2017 Learn by doing begins with introduction basic modeling The later chapters focus on additional modeling top down assemblies sheet metal modeling drafting surface modeling mold tools weldments DimXpert and rendering Table of Contents 1 Getting Started 2 Modeling Basics 3 Assembly Basics 4 Creating Drawings 5 Sketching 6 Additional Modeling Tools 7 Sheet metal Modeling 8 Top Down Assembly 9 Dimensions and Annotations 10 Surface Design 11 Mold Tools 12 Weldments 13 DimXpert 14 Appearances and Rendering

*Learning SOLIDWORKS 2017* Randy Shih, 2017-03 This book will teach you everything you need to know to start using SOLIDWORKS 2017 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 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 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 2020 Learn by Doing - Part 1 Tutorial Books](#), 2020-02-10 SOLIDWORKS 2020 Learn by doing Part 1 introduces new users to mechanical design using SOLIDWORKS and how it can be used to create a variety of models In nine tutorial based chapters the author guides you through all the necessary commands and options in SOLIDWORKS 2020 from sketching to parametric modeling and finally ending with rendering The tools are presented one step at a time using simple examples The approach used in this book helps you to become a skilled SOLIDWORKS user SOLIDWORKS 2020 Learn by doing Part 1 begins with basic to introduction modeling The later chapters focus on additional modeling top down assemblies sheet metal modeling and drafting Table of Contents 1 Getting Started 2 Modeling Basics 3 Assembly Basics 4 Creating Drawings 5 Sketching 6 Additional Modeling Tools 7 Sheet metal Modeling 8 Top Down Assembly 9 Dimensions and Annotations **SOLIDWORKS 2018 Learn by Doing** Tutorial Books, 2018-01-25 SOLIDWORKS 2018 Learn by doing introduces new users to mechanical design using SOLIDWORKS and how it can be used to create a variety of models In fourteen tutorial based chapters author guides you through all the necessary commands and options in SOLIDWORKS 2018 from sketching to parametric modeling and finally ending with rendering The commands are presented one step at a time using simple examples The approach used in this book helps you to become a skilled SOLIDWORKS user SOLIDWORKS 2018 Learn by doing begins with introduction basic modeling The later chapters focus on additional modeling top down assemblies sheet metal modeling drafting surface modeling mold tools weldments DimXpert and rendering Table of Contents 1 Getting Started 2 Modeling Basics 3 Assembly Basics 4 Creating Drawings 5 Sketching 6 Additional Modeling Tools 7 Sheet metal Modeling 8 Top Down Assembly 9

Dimensions and Annotations 10 Surface Design 11 Mold Tools 12 Weldments 13 DimXpert 14 Appearances and Rendering If you are an educator you can request an evaluation copy by sending us an email to online books999 gmail com

**SOLIDWORKS 2016 Learn by Doing** Tutorial Books,2016-02-10 SOLIDWORKS 2016 Learn by doing introduces new users to mechanical design using SOLIDWORKS and how it can be used to create a variety of models In fourteen tutorial based chapters author guides you through all the necessary commands and options in SOLIDWORKS 2016 from sketching to parametric modeling and finally ending with rendering The commands are presented one step at a time using simple examples The approach used in this book helps you to become a skilled SOLIDWORKS user SOLIDWORKS 2016 Learn by doing begins with introduction basic modeling The later chapters focus on additional modeling top down assemblies sheet metal modeling drafting surface modeling mold tools weldments DimXpert and rendering Table of Contents 1 Getting Started 2 Modeling Basics 3 Assembly Basics 4 Creating Drawings 5 Sketching 6 Additional Modeling Tools 7 Sheet metal Modeling 8 Top Down Assembly 9 Dimensions and Annotations 10 Surface Design 11 Mold Tools 12 Weldments 13 DimXpert 14 Appearances and Rendering *Engineering Design with SOLIDWORKS 2017 and Video Instruction* David Planchard,2017 Engineering Design with SOLIDWORKS 2017 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 He is directly

involved with SOLIDWORKS every day His responsibilities go far beyond the creation of just a 3D model The book is designed to complement the SOLIDWORKS Tutorials contained in SOLIDWORKS 2017 **SOLIDWORKS 2017 Tutorial with Video Instruction** David Planchard,2017 SOLIDWORKS 2017 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 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 equations multi sheet multi view drawings BOMs and 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 View 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 **The Flipped Classroom** Carl Reidsema, Lydia Kavanagh, Roger Hadgraft, Neville Smith,2017-02-27 Teaching and learning within higher education continues to evolve with innovative and new practices such as flipped teaching This book contributes to the literature by developing a much deeper understanding of the complex phenomenon of flipped classroom approaches within higher education It also serves as a practical guide to implementing flipped classroom teaching in academic practice across different higher educational institutions and disciplines Part 1 of this book Practice describes the considerations involved in flipped classroom teaching including the challenges faced in transforming teaching and learning within higher education Further it reviews the educational concepts on which the flipped classroom is based including a selected history of similar innovations in the past The final sections of Part 1 explore the tools needed for flipping the design steps assessment methods and the role of reflective practice within flipped teaching environments p Part 2 of the book Practices provides a range of case studies from higher educational institutions in different countries and disciplines to demonstrate the many shapes and sizes of flipped classrooms Many of the challenges such as engaging students in their own learning and shifting them from spectators in the learning process to active participants prove to be universal *SolidWorks 2015 Learn by Doing (Part, Assembly, Drawings, Sheet Metal, Surface*

*Design, Mold Tools, Weldments, DimXpert, and Rendering*) Tutorial Books,2015-02-12 The objective of this book is to help you learn SOLIDWORKS 2015 by practically using its various features The fourteen lessons in this tutorial introduce you to the designing documentation and presentation in SOLIDWORKS 2015 The topics covered in this tutorial are part and assembly design sketch 3D part assembly 2D drawing sheetmetal surface design mold tools weldments DimXpert and rendering

**SOLIDWORKS 2025 Learn By Doing** Tutorial Books,2025-04-08 uhb Unlock the power of SOLIDWORKS 2025 with this hands on tutorial based guide Whether you re a beginner or an experienced user SOLIDWORKS 2025 Learn by Doing provides a step by step approach to mastering essential design concepts from sketching and parametric modeling to assemblies and sheet metal Why Choose This SOLIDWORKS 2025 Guide Comprehensive Coverage Learn essential commands options and workflows Step by Step Tutorials Clear practical examples make learning easy 14 In Depth Chapters Progress smoothly from basics to advanced techniques CSWA CSWP Exam Prep Includes sample questions for certification success Perfect for All Skill Levels Ideal for students professionals and engineers With a structured approach this book helps you build confidence in mechanical design ensuring you re prepared for real world projects and SOLIDWORKS certification Start your SOLIDWORKS 2025 journey today Get your copy of SOLIDWORKS 2025 Learn by Doing and take your skills to the next level

**SOLIDWORKS 2019 Learn by Doing** Tutorial Books,2019-02-20 SOLIDWORKS 2019 Learn by doing introduces new users to mechanical design using SOLIDWORKS and how it can be used to create a variety of models In fourteen tutorial based chapters author guides you through all the necessary commands and options in SOLIDWORKS 2019 from sketching to parametric modeling and finally ending with rendering The commands are presented one step at a time using simple examples The approach used in this book helps you to become a skilled SOLIDWORKS user SOLIDWORKS 2019 Learn by doing begins with introduction basic modeling The later chapters focus on additional modeling top down assemblies sheet metal modeling drafting surface modeling mold tools weldments MBD Dimensions and rendering Table of Contents1 Getting Started2 Modeling Basics3 Assembly Basics4 Creating Drawings5 Sketching6 Additional Modeling Tools7 Sheet metal Modeling8 Top Down Assembly9 Dimensions and Annotations10 Surface Design11 Mold Tools12 Weldments13 MBD Dimensions14 Appearances and Rendering

**Beginner's Guide to SOLIDWORKS 2017 - Level I** Alejandro Reyes,2017-01-17 This book is intended to help new users learn the basic concepts of SOLIDWORKS and good solid modeling techniques in an easy to follow guide that includes video instruction It is a great starting point for those new to SOLIDWORKS or as a teaching aid in classroom training to become familiar with the software s interface basic commands and strategies as users complete a series of models while learning different ways to accomplish a particular task At the end of this book you will have a fairly good understanding of the SOLIDWORKS interface and the most commonly used commands for part modeling assembly and detailing after completing a series of components and their 2D drawings complete with Bill of Materials The book focuses on the processes to complete the modeling of a part instead of focusing on individual software

commands or operations which are generally simple enough to learn The author strived hard to include the commands required in the Certified SOLIDWORKS Associate and Certified SOLIDWORKS Professional Exams as listed on the SOLIDWORKS website SOLIDWORKS is an easy to use CAD software that includes many time saving tools that will enable new and experienced users to complete design tasks faster than before Most commands covered in this book have advanced options which may not be covered in this book This is meant to be a starting point to help new users to learn the basic and most frequently used commands

SOLIDWORKS 2020 Learn by Doing Tutorial Books,2020-02-22 SOLIDWORKS 2020 Learn by doing introduces new users to mechanical design using SOLIDWORKS and how it can be used to create a variety of models In fourteen tutorial based chapters the author guides you through all the necessary commands and options in SOLIDWORKS 2019 from sketching to parametric modeling and finally ending with rendering The commands are presented one step at a time using simple examples The approach used in this book helps you to become a skilled SOLIDWORKS user

SOLIDWORKS 2020 Learn by doing begins with introduction to basic modeling The later chapters focus on additional modeling top down assemblies sheet metal modeling drafting surface modeling mold tools weldments Model based dimensioning Appearances and SimulationXpress

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Learn SOLIDWORKS 2020 Tayseer Almattar,2019-12-02 Explore a practical and example driven approach to understanding SOLIDWORKS 2020 and achieving CSWA and CSWP certification Key FeaturesGain comprehensive insights into the core aspects of mechanical part modelingGet up to speed with generating assembly designs with both standard and advanced matesFocus on design practices for both 2D as well as 3D modeling and prepare to achieve CWSP and CWSA certificationBook 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 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 learnUnderstand the fundamentals of SOLIDWORKS and parametric modelingCreate 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

**SolidWorks 2021 - Step-By-Step Guide** Amit Bhatt, CADFolks, 2020-12 This book starts with SolidWorks 2021 using step by step examples It begins with creating sketches and parts assembling them and then creating print ready drawings This book gives you an idea about how you can design and document various mechanical components and helps you to learn some advanced tools and techniques This book also follows some of the best practices in creating parts In addition to this there are some additional chapters covering sheet metal and surface design Each topic in this book has a brief introduction and a step by step example This will help you to learn SolidWorks 2018 quickly and easily Go through with the User Interface A step by step practice to create sketches and 3D models Teach you about advance Part Modeling tools Learn the procedure to create Multiple body parts Learn to modify components at each step Learn to create assemblies Learn Top down assembly design Learn to create 2D drawings Learn basic tools available in Sheet Metal and Surface Environment Create sheet metal drawings Create complex shapes using surface modeling tools You can download Resource Files from [www.cadfolks.com](http://www.cadfolks.com) Available very soon

**SOLIDWORKS 2017 Reference Guide** David Planchard, 2017 The SOLIDWORKS 2017 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2017 SOLIDWORKS is an immense software package and no one book can cover all topics for all users This book provides a centralized reference location to address many of the tools features and techniques of SOLIDWORKS 2017 This book covers the following System and Document properties FeatureManagersPropertyManagersConfigurationManagersRenderManagers 2D and 3D Sketch tools Sketch entities 3D Feature tools Motion Study Sheet Metal Motion Study SOLIDWORKS Simulation PhotoView 360 Pack and Go 3D PDFs Intelligent Modeling techniques 3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS 2017 software If you are completely new to SOLIDWORKS you should read Chapter 1 in detail and complete Lesson 1 Lesson 2 and Lesson 3 in the SOLIDWORKS Tutorials If you are familiar with an earlier release of SOLIDWORKS you still might want to skim Chapter 1 to become acquainted with some of the commands menus and features that you have not used or you can simply jump to any section in any chapter Each chapter provides detailed PropertyManager information on key topics with individual stand alone short tutorials to reinforce and demonstrate the functionality and ease of the SOLIDWORKS tool or feature The book provides access to over 250 models their solutions and additional support materials Learn by doing not just by reading Formulate the

skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components design tables configurations and more The book is designed to compliment the Online Tutorials and Online Help contained in SolidWorks 2017 The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs The author developed the tutorials by combining his own industry experience with the knowledge of engineers department managers professors vendors and manufacturers He is directly involved with SOLIDWORKS every day and his responsibilities go far beyond the creation of just a 3D model

**Learn SOLIDWORKS** Tayseer Almattar,2022-01-31 Get to grips with leading 3D engineering and product design application to design robust 3D models and achieve CSWA and CSWP certification with SOLIDWORKS Specialist Tayseer Almattar Key Features Gain comprehensive insights into the core aspects of 3D modeling s mechanical parts Learn how to generate assembly designs with both standard and advanced mates Discover design practices for both 2D as well as 3D modeling and prepare to achieve CSWP and CSWA certification Book DescriptionSOLIDWORKS is the leading choice for 3D engineering and product design applications across industries such as aviation automobile and consumer product design This book helps you to get up and running with SOLIDWORKS and understand each new concept and tool with the help of easy to follow exercises You ll begin with the basics exploring the software interface and finding out how to work with drawing files The book then guides you through topics such as sketching building complex 3D models generating dynamic and static assemblies and generating 2D engineering drawings to prepare you to take on any design project You ll also work with practical exercises to get hands on experience 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 to understand the concepts and exercises more easily Finally you ll also work on projects for 3D modeling objects inspired by everyday life By the end of this SOLIDWORKS book you ll have gained the skills you need to create professional 3D mechanical models using SOLIDWORKS and 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 Join different parts together to form static and dynamic assemblies 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 makers draftsmen and hobbyists looking to get started with SOLIDWORKS and explore the software Individuals who are interested in becoming Certified SOLIDWORKS Associates CSWAs or Certified SOLIDWORKS Professionals CSWPs will also find this book useful No specific background is needed to follow the concepts in the book as it starts from the basics of SOLIDWORKS However basic theoretical knowledge of 3D modeling will be helpful to get the most

out of this book     **Engineering Design with SOLIDWORKS 2019** David Planchard,2019 Engineering Design with SOLIDWORKS 2019 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 featuring machined plastic and sheet metal components Desired outcomes and usage competencies are listed for each project The book is divided into five sections with 11 projects Project 1 Project 6 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 Additional techniques include the edit and reuse of features parts and assemblies through symmetry patterns configurations SOLIDWORKS 3D ContentCentral and the SOLIDWORKS Toolbox Project 7 Understand Top Down assembly modeling and Sheet Metal parts Develop components In Context with InPlace Mates along with the ability to import parts using the Top Down assembly method Convert a solid part into a Sheet Metal part and insert and apply various Sheet Metal features Project 8 Project 9 Recognize SOLIDWORKS Simulation and Intelligent Modeling techniques Understand a general overview of SOLIDWORKS Simulation and the type of questions that are on the SOLIDWORKS Simulation Associate Finite Element Analysis CSWSA FEA exam Apply design intent and intelligent modeling techniques in a sketch feature part plane assembly and drawing Project 10 Comprehend the differences between additive and subtractive manufacturing Understand 3D printer terminology along with a working knowledge of preparing saving and printing CAD models on a low cost printer Project 11 Review the Certified SOLIDWORKS Associate CSWA program Understand the curriculum and categories of the CSWA exam and the required model knowledge needed to successfully take the exam 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     Project Based SOLIDWORKS 2021 Kirstie Plantenberg,2021-02 Project Based SOLIDWORKS is specifically designed to complement an engineering graphics course It covers how to apply engineering graphics concepts such as part prints section views assembly drawings tolerancing and fasteners It also extends these topics into the world of design Project Based SOLIDWORKS takes a specific part or assembly and teaches you how to model each part and its configurations create part prints including assembly drawings if appropriate and takes it one step further and teaches concepts such as FEA tolerancing and parametric design This book comes with instructional videos showing you how to perform each of the tutorials It also comes with instructional videos showing how to complete each problem in the book The exception to this is when a problem is open ended and each student will get different results After completing all the tutorials in this book you will be able to design moderately difficult parts and assemblies in a realistic manner This book is perfect for a freshman design class that wishes to include realistic design problems within their curriculum Structure Project Based

SOLIDWORKS is arranged in projects For example Chapter two deals with the modeling of a Connecting Rod Chapter three continues with the connecting rod to introduce the concept of configurations Chapter four creates a part print of the connecting rod and Chapter five wraps up the project by performing a static FEA on the connecting rod At the beginning of each chapter a list of prerequisite tutorials or knowledge is listed You do not necessarily need to complete the tutorials within the book in order but make sure that you have the prerequisite knowledge before you begin Topics covered The following topics are covered in this book Part modeling Part configurations Assembly Static FEA Part Prints Assembly drawings Fasteners Tolerancing Parametric Modeling 3D sketches Videos Project Based SOLIDWORKS comes with over ten hours of video instruction These videos follow along with the book and complement the text perfectly Videos for each tutorial and problem in the book are provided Additional videos on specific difficult topics are also included Captioned versions of these videos are also available for customers who want or need video captions

## Decoding **Solidworks 2017 Learn By Doing Part Assembly Drawings**: Revealing the Captivating Potential of Verbal Expression

In a period characterized by interconnectedness and an insatiable thirst for knowledge, the captivating potential of verbal expression has emerged as a formidable force. Its power to evoke sentiments, stimulate introspection, and incite profound transformations is genuinely awe-inspiring. Within the pages of "**Solidworks 2017 Learn By Doing Part Assembly Drawings**," a mesmerizing literary creation penned by way of a celebrated wordsmith, readers attempt an enlightening odyssey, unraveling the intricate significance of language and its enduring impact on our lives. In this appraisal, we shall explore the book's central themes, evaluate its distinctive writing style, and gauge its pervasive influence on the hearts and minds of its readership.

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### **Solidworks 2017 Learn By Doing Part Assembly Drawings Introduction**

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