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This is the second part of

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SOLIDWORKS 2017 Learn by doing. The chapters in this book introduce you to surface design, mold design, and weldments in SOLIDWORKS 2017. The topics covered in this book are: Surface design •Creating Basic

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Drawings 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.

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Drawings, DimXpert, and rendering. Table of Contents
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Appearances and Rendering

This book will teach you
everything you need to know

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Drawings 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,

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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

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Drawing 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

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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

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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

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Drawings 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

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Drawings 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

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Drawings 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

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Drawings
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

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Drawings 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

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Drawings 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

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Drawings 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 Tutorial

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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

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Drawings 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

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Drawings and
CommandManager, Document and
System properties, simple
machine parts, simple and
complex assemblies, proper
design intent, design
tables, configurations,
equations, multi-sheet,

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Drawings 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

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Drawings 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

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Drawings 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

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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

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Drawings 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.

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Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

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Drawings 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-

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Drawings 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

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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

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Drawings 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

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Drawings 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

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Drawings 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

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Drawings. 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

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Drawings 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

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Drawings 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

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Drawings. 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

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Drawings 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

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Drawings
Beyond the creation of just a 3D model. The book is designed to complement the SOLIDWORKS Tutorials contained in SOLIDWORKS 2017.

The SOLIDWORKS 2017

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Drawings 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

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Drawings 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 propertiesFeatu

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reManagersPropertyManagersCo
nfigurationManagersRenderMan
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and Go3D 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

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Drawings 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

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Drawings 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

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Drawings 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

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Drawings 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

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Drawings sketches and solid features.
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reuse features, parts and
assemblies through symmetry,
patterns, copied components,
design tables,
configurations and more. The
book is designed to

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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.

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Drawings
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

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SOLIDWORKS every day and his responsibilities go far beyond the creation of just a 3D model.

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Drawings 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

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modeling, drafting, surface
modeling, mold tools,
weldments, Model-based
dimensioning, Appearances,
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Intermediate User textbook is designed for instructor-led courses as well as for self-paced learning. It is intended to help engineers and designers interested in learning SOLIDWORKS for creating 3D mechanical

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Drawings Taken together, this textbook can be a great starting point for new SOLIDWORKS users and a great teaching aid in classroom training. This textbook consists of 14 chapters, total 768 pages covering

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major environments of
DRAWINGS: Sketching
environment, Part modeling
environment, Assembly
environment, and Drawing
environment, which teach you
how to use the SOLIDWORKS
mechanical design software

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Drawings to build parametric models and assemblies, and how to make drawings of those parts and assemblies. Moreover, this textbook includes the topic of Configurations.

This textbook not only focuses on the usages of the

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tools/commands of SOLIDWORKS but also on the concept of design. Every chapter of this textbook contains tutorials which instruct users how things can be done in SOLIDWORKS step by step. Moreover, every chapter ends

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with hands-on test drives
which allow users to
experience themselves the
ease-of-use and powerful
capabilities of SOLIDWORKS.
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Dimensions Chapter 5.
Creating First/Base Feature
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Chapter 8. Advanced Modeling

- II Chapter 9. Patterning

and Mirroring Chapter 10.

Advanced Modeling - III

Chapter 11. Working with

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Chapter 13. Working with
Assemblies - II Chapter 14.
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Additional notes and tips
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students Additional student
and faculty projects

Technical support for the
book: info@cadartifex.com

This book is intended to
help new users learn the

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Drawings 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

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Drawings 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

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Drawings 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

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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

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Drawings 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

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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

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Drawings not be covered in this book. This is meant to be a starting point to help new users to learn the basic and most frequently used commands.

Beginner's Guide to

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SOLIDWORKS 2017 - 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

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Drawings 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

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Drawings 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

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Drawings. 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

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Drawings 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

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Drawings 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

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Drawings 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

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