

Read Free Beginners Guide To
Solidworks 2017 Level I

Beginners Guide To Solidworks 2017 Level I

SOLIDWORKS 2017 Basic Tools is the first book in a three part series. It introduces new users to the SOLIDWORKS interface,

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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

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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

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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

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take it step by step.

- 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

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- Covers the creation of parts, assemblies and drawings • Features a quick reference guide and a Certified SOLIDWORKS Associate practice exam • The first book of a three book series SOLIDWORKS 2021 Basic Tools is

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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

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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.

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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

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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

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ins.

"The most complete resource for SolidWorks on the market. Matt Lombard's in-depth knowledge plus his snappy wit and wisdom make SolidWorks accessible to users at all levels." -- Mike

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Sabocheck, Territory Technical Manager, SolidWorks Corporation
The most comprehensive single reference on SolidWorks Whether you're a new, intermediate, or professional user, you'll find the in-depth coverage you need to

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succeed with SolidWorks 2007 in this comprehensive reference. From customizing the interface to exploring best practices to reinforcing your knowledge with step-by-step tutorials, the techniques and shortcuts in this

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detailed book will help you accomplish tasks, avoid the time-consuming pitfalls of parametric design, and get a firm handle on one of the leading 3D CAD programs on the market. *
Customize the user interface and

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connect hotkeys to macros * Create sketches, parts, assemblies, and drawings * Build intelligence into parts * Work with patterns, equations, and configurations * Learn multibody, surface, and master model techniques * Write,

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record, and edit Visual Basic(r) macros Design with advanced 3D features Increase speed and efficiency with subassemblies Use multibody models to their full potential What's on the CD-ROM? The CD includes all the parts,

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assemblies, drawings, and examples you need to follow the tutorials in each chapter. You'll also find finished models, templates, and more. See the CD appendix for details and complete system requirements

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SOLIDWORKS Sheet Metal Design 2021 textbook has been designed for instructor-led courses as well as self-paced learning. It is intended to help engineers and designers interested in learning SOLIDWORKS for creating real-

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world sheet metal components. This textbook is a great help for SOLIDWORKS users new to sheet metal design. It consists of total 132 pages covering the sheet metal design environment of SOLIDWORKS. It teaches users to

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use SOLIDWORKS mechanical design software for creating parametric 3D sheet metal components. This textbook not only focuses on the usage of the tools and commands of SOLIDWORKS for creating sheet

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metal components but also on the concept of design. It contains Tutorials followed by theory that provide users with step-by-step instructions for creating sheet metal components. Moreover, it ends with Hands-on Test Drives

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which allow users to experience the user friendly and technical capabilities of SOLIDWORKS.

A Power Guide for Beginners and Intermediate Users

Beginner's Guide to SOLIDWORKS 2017 - Level I

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SOLIDWORKS 2017 in 5 Hours
with Video Instruction

SOLIDWORKS 2017 Basic Tools
Engineering Design with
SOLIDWORKS 2017 and Video
Instruction

Full Color edition

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SOLIDWORKS Simulation 2019: A Power Guide for Beginners and Intermediate Users textbook is designed for instructor-led courses as well as for self-paced learning. It is intended to help engineers and designers

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interested in learning finite element analysis (FEA) using SOLIDWORKS Simulation. This textbook benefits new SOLIDWORKS Simulation users and is a great teaching aid in classroom training. It

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*consists of 10 chapters,
total 394 pages covering
various types of finite
element analysis (FEA) such
as Linear Static Analysis,
Buckling Analysis, Fatigue
Analysis, Frequency
Analysis, Drop Test*

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Analysis, and Non-linear Static Analysis. This textbook covers important concepts and methods used in finite element analysis (FEA) such as Preparing Geometry, Boundary Conditions (load and

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fixture), Element Types, Contacts, Connectors, Meshing, Mesh Controls, Mesh Quality Check (Jacobian Check and Aspect Ratio), Adaptive Meshing (H-Adaptive and P-Adaptive), Iterative Methods (Newton-Raphson

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Scheme and Modified Newton-Raphson Scheme), Incremental Methods (Force, Displacement, or Arc Length), and so on. This textbook not only focuses on the usages of the tools of SOLIDWORKS Simulation but

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also on the fundamentals of finite element analysis (FEA) through various real-world case studies. The case studies used in this textbook allow users to solve various real-world engineering problems, step-

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by-step. Moreover, the Hands-on test drives are given at the end of the chapters which allow users to experience the user friendly and technical capabilities of SOLIDWORKS Simulation. Every chapter begins with

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learning objectives related to the topics covered in that chapter. Moreover, every chapter ends with a summary which lists the topics learned in that chapter followed by questions to assess the

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knowledge. Table of Contents: Chapter 1. Introduction to FEA and SOLIDWORKS Simulation Chapter 2. Introduction to Analysis Tools and Static Analysis Chapter 3. Case Studies of Static Analysis

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Chapter 4. Contacts and Connectors Chapter 5.

Adaptive Mesh Methods

Chapter 6. Buckling Analysis

Chapter 7. Fatigue Analysis

Chapter 8. Frequency

Analysis Chapter 9. Drop

Test Analysis Chapter 10.

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*Non-Linear Static Analysis
Main Features of the
Textbook Comprehensive
coverage of tools Step-by-
step real-world case studies
Hands-on test drives to
enhance the skills at the
end of chapters Additional*

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notes and tips Customized content for faculty (PowerPoint Presentations) Free learning resources for students and faculty Technical support for the book: info@cadartifex.com SOLIDWORKS Surface Design

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2021 for Beginners and Intermediate Users textbook has been designed for instructor-led courses as well as self-paced learning. It is intended to help engineers and designers interested in learning

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SOLIDWORKS for creating real-world surface models. This textbook is a great help for SOLIDWORKS users new to surface design. It consists of total 106 pages covering the surface design environment of SOLIDWORKS.

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It teaches users to use SOLIDWORKS mechanical design software for creating parametric complex shape surface models that are not possible to create with solid modeling due to its limitations. This textbook

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not only focuses on the usage of the tools and commands of SOLIDWORKS for creating surface models but also on the concept of design. It contains Tutorials followed by theory that provide users with step-

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by-step instructions for creating surface designs. Moreover, it ends with Hands-on Test Drives which allow users to experience the user friendly and technical capabilities of SOLIDWORKS. Main Features of the

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Textbook:

- *Comprehensive coverage of tools*
- *Step-by-step real-world tutorials with every chapter*
- *Hands-on test drives to enhance the skills at the end of every chapter*
- *Additional notes and tips*
- *Customized*

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*content for faculty
(PowerPoint Presentations) •
Free learning resources for
faculty and students •
Technical support for the
book by contacting
info@cadartifex.com
SOLIDWORKS 2019 Tutorial is*

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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

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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

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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.

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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

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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

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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

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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

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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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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.

Beginner's Guide to

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SOLIDWORKS 2021 - Level II starts where Beginner's Guide - Level I ends, following the same easy to read style and companion video instruction, but this time covering advanced topics and techniques. The

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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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changes with SOLIDWORKS' parametric capabilities, mold design, welded structures and more while explaining the basic concepts of each trade to allow you to understand the how and why of each

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operation. The author uses simple examples to allow you to better understand each command and environment, as well as to make it easier to explain the purpose of each step, maximizing the learning time by focusing on

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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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learn. At the end of this book, you will have acquired enough skills to be highly competitive when it comes to designing with SOLIDWORKS, and while there are many less frequently used commands and options

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available that will not be covered in this book, rest assured that those covered are most of the commands used every day by SOLIDWORKS designers. The author strived hard to include many of the commands required in

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the Certified SOLIDWORKS Professional Advanced and Expert exams as listed on the SOLIDWORKS website. Includes Video Instruction Each copy of this book includes access to video instruction. In these videos

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the author provides a clear presentation of tutorials found in the book. The videos reinforce the steps described in the book by allowing you to watch the exact steps the author uses to complete the exercises

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while he provides additional details along the way.

Captioned versions of these videos are also available for customers who want or need video captions.

Creo Parametric 7.0: A Power Guide for Beginners and

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Intermediate Users

*SOLIDWORKS 2021 Basic Tools
AutoCAD 2021 for
Architectural Design: A
Power Guide for Beginners
and Intermediate Users
Solidworks 2018*

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Parametric Modeling with SOLIDWORKS 2017 contains a series of seventeen tutorial style lessons designed to introduce SOLIDWORKS 2017, solid modeling and parametric modeling techniques and concepts.

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This book introduces SOLIDWORKS 2017 on a step-by-step basis, starting with constructing basic shapes, all the way through to the creation of assembly drawings and motion analysis. This book takes a

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hands on, exercise intensive approach to all the important parametric modeling techniques and concepts. Each lesson introduces a new set of commands and concepts, building on previous

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lessons. The lessons guide the user from constructing basic shapes to building intelligent solid models, assemblies and creating multi-view drawings. This book also covers some of the more advanced features of

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SOLIDWORKS 2017, including how to use the SOLIDWORKS Design Library, basic motion analysis, collision detection and analysis with SimulationXpress. The exercises in this book cover the performance tasks that

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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

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covered. This book also introduces you to the general principles of 3D printing including a brief history of 3D printing, the types of 3D printing technologies, commonly used filaments, and the basic

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procedure for printing a 3D model. 3D printing makes it easier than ever for anyone to start turning their designs into physical objects and by the end of this book you will be ready to start printing out your

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own designs.

Creo Parametric 7.0: A Power Guide for Beginners and Intermediate Users textbook is designed for instructor-led courses as well as self-paced learning. It is intended to help engineers

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and designers interested in learning Creo Parametric for creating 3D mechanical design. This textbook benefits new Creo users and is a great teaching aid in classroom training. It consists of 12 chapters,

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with a total of 736 pages covering the major modes of Creo Parametric such as the Sketch, Part, Assembly, and Drawing modes. The textbook teaches users to use Creo Parametric mechanical design software for building

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parametric 3D solid components, assemblies, and 2D drawings. This textbook not only focuses on the usage of the tools/commands of Creo Parametric but also on the concept of design. Each chapter of this

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textbook contains tutorials which help users to easily operate Creo Parametric step-by-step. Moreover, each chapter ends with hands-on test drives which allow users to experience the user friendly and technical

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capabilities of Creo Parametric. Table of Contents: Chapter 1. Introduction to Creo Parametric Chapter 2. Drawing Sketches and Applying Dimensions Chapter 3. Editing and Modifying

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Sketches Chapter 4. Creating Base Feature of a Solid Model Chapter 5. Creating Datum Geometries Chapter 6. Advanced Modeling - I Chapter 7. Advanced Modeling - II Chapter 8. Patterning and Mirroring Chapter 9.

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Advanced Modeling - III

Chapter 10. Working with

Assemblies - I Chapter 11.

Working with Assemblies - II

Chapter 12. Working with

Drawings

The purpose of Creo

Parametric 4.0 Advanced

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Tutorial is to introduce you to some of the more advanced features, commands, and functions in Creo Parametric. Each lesson concentrates on a few of the major topics and the text attempts to explain the

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“why’ s” of the commands in addition to a concise step-by-step description of new command sequences. This book is suitable for a second course in Creo Parametric and for users who understand the features already covered

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in Roger Toogood's Creo Parametric Tutorial. The style and approach of the previous tutorial have been maintained from the previous book and the text picks up right where the last tutorial left off. The material

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covered in this tutorial represents an overview of what is felt to be the most commonly used and important functions. These include customization of the working environment, advanced feature creation (sweeps,

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round sets, draft and tweaks, UDF's, patterns and family tables), layers, Pro/PROGRAM, and advanced drawing and assembly functions. Creo Parametric 4.0 Advanced Tutorial consists of eight lessons. A

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continuing theme throughout the lessons is the creation of parts for a medium-sized modeling project. The project consists of a small three-wheeled utility cart. Project parts are given at the end of each lesson that

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utilize functions presented earlier in that lesson.

Final assembly is performed in the last lesson.

AutoCAD 2021 for
Architectural Design: A
Power Guide for Beginners
and Intermediate Users

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textbook is designed for instructor-led courses as well as for self-paced learning. It is intended to help architects, designers, and CAD operators interested in learning AutoCAD for creating 2D architectural

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drawings. This textbook is a great help for new AutoCAD users and a great teaching aid for classroom training. This textbook consists of 12 chapters, and a total of 488 pages covering tools and commands of the Drafting &

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Annotation workspace of AutoCAD. The textbook teaches you to use AutoCAD software for creating, editing, plotting, and managing real world 2D architectural drawings.

Table of Contents: Chapter

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- 1. Introduction to AutoCAD
- Chapter 2. Creating Drawings - I
- Chapter 3. Working with Drawing Aids and Layers
- Chapter 4. Creating Drawings - II
- Chapter 5. Modifying and Editing Drawings - I
- Chapter 6. Working with

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Blocks and Xrefs Chapter 7.
Working with Dimensions and
Dimensions Style Chapter 8.
Editing Dimensions and
Adding Text Chapter 9.
Modifying and Editing
Drawings - II Chapter 10.
Hatching and Gradients

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Chapter 11. Working with
Layouts Chapter 12. Printing
and Plotting

SOLIDWORKS Simulation 2019:
a Power Guide for Beginners
and Intermediate Users

AutoCAD 2022: A Power Guide
for Beginners and

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Intermediate Users

AutoCAD 2021: A Power Guide
for Beginners and
Intermediate Users

Getting started with Parts,
Assemblies and Drawings
Engineering Design and
Graphics with Solidworks

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2016

AutoCAD 2021: A Power Guide for Beginners and Intermediate Users textbook is designed for instructor-led courses as well as for self-paced learning. It is intended to help engineers, designers, and CAD

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operators interested in learning AutoCAD for creating 2D engineering drawings as well as 3D Models. This textbook is a great help for new AutoCAD users and a great teaching aid for classroom training. This textbook consists of

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13 chapters, and a total of 556 pages covering major workspaces of AutoCAD such as Drafting & Annotation and 3D Modeling. This textbook teaches you to use AutoCAD software for creating, editing, plotting, and managing

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real world 2D engineering drawings and 3D Models. This textbook not only focuses on the usage of the tools/commands of AutoCAD but also on the concept of design. Every chapter of this textbook contains tutorials that

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provide users with step-by-step instructions on how to create mechanical designs and drawings with ease. Moreover, every chapter ends with hands-on test drives which allow users to experience themselves the user friendly and

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powerful capabilities of AutoCAD.

Table of Contents: Chapter 1.

Introduction to AutoCAD Chapter

2. Creating Drawings - I Chapter 3.

Working with Drawing Aids and

Layers Chapter 4. Creating

Drawings - II Chapter 5. Modifying

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and Editing Drawings - I Chapter 6. Working with Dimensions and Dimensions Style Chapter 7. Editing Dimensions and Adding Text Chapter 8. Modifying and Editing Drawings - II Chapter 9. Hatching and Gradients Chapter

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10. Working with Blocks and Xrefs

Chapter 11. Working with Layouts

Chapter 12. Printing and Plotting

Chapter 13. Introducing 3D Basics and Creating 3D Models

This book is intended to help new users learn the basic concepts of

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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

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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

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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

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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

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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

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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

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point to help new users to learn the basic and most frequently used commands.

SOLIDWORKS 2017 Advanced Techniques picks up where SOLIDWORKS 2017 Intermediate Skills leaves off. Its aim is to take

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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

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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

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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

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on to more advanced lessons. This book takes the approach that in order to understand SOLIDWORKS, inside and out, you should create everything from the beginning and take it step by step.

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Beginner's Guide to SOLIDWORKS 2022 – 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

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techniques. The purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create components in the context of an assembly and reference other components (Top-down design),

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propagate design changes with SOLIDWORKS' parametric capabilities, mold design, welded structures and more while explaining the basic concepts of each trade to allow you to understand the how and why of

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each operation. The author uses simple examples to allow you to better understand each command and environment, as well as to make it easier to explain the purpose of each step, maximizing the learning time by focusing on

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one task at a time. This book is focused on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. At the end of this book,

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you will have acquired enough skills to be highly competitive when it comes to designing with SOLIDWORKS, and while there are many less frequently used commands and options available that will not be covered in this

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book, rest assured that those covered are most of the commands used every day by SOLIDWORKS designers. The author strived hard to include many of the commands required in the Certified SOLIDWORKS Professional

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Advanced and Expert exams as listed on the SOLIDWORKS website. Includes Video Instruction Each copy of this book includes access to video instruction. In these videos the author provides a clear presentation of tutorials found in

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the book. The videos reinforce the steps described in the book by allowing you to watch the exact steps the author uses to complete the exercises while he provides additional details along the way. Captioned versions of these videos

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are also available for customers who want or need video captions.

SolidWorks 2007 Bible

Beginner's Guide to

SOLIDWORKS 2019 - Level II

Beginner's Guide to

SOLIDWORKS 2021 - Level I

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Solidworks 2017

SOLIDWORKS Sheet Metal

Design 2021

*Solidworks 2017A Power Guide for
Beginners and Intermediate*

*UsersCreatespace Independent
Publishing Platform*

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

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software but then goes far beyond, covering numerous details that advanced users have requested.

Beginners will learn not only how the software works but why, while more experienced users will learn all about search criteria, Pack-and-Go, other

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file management concepts, and much more. A valuable companion website contains before and after real-world parts and assemblies along with many example files used in the text.

Additionally, the text of the book is augmented by video tutorials with

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author voice-over which can be found on the website. SolidWorks is the leading 3D CAD program, and previous editions of this book have sold more than 33,000 copies Covers necessary information to give beginners a solid foundation in the

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*software, including part and assembly modeling and 2D drawing techniques
Addresses a wide range of advanced topics not treated in other books, including best practices, search criteria, Pack-and-Go, and other file management concepts Includes*

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tutorials on both beginning and advanced topics, with videos; sample part, assembly, and drawing files; and before-and-after example files available on the companion website SolidWorks 2013 Bible is the ultimate resource on SolidWorks 2013, the book

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beginners can start with and advanced users will want to keep close at hand.

SOLIDWORKS 2018 Tutorial with video instruction is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The text provides a

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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

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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

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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

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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

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engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs from subtractive manufacturing, and its features. You will also learn the terms and technology used in low cost 3D

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printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to

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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

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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

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SOLIDWORKS in industry.

Autodesk Fusion 360: A Power Guide for Beginners and Intermediate Users (4th Edition) textbook has been designed for instructor-led courses as well as self-paced learning. It is intended to help engineers and

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designers, interested in learning Fusion 360, to create 3D mechanical designs. This textbook is a great help for new Fusion 360 users and a great teaching aid for classroom training. This textbook consists of 14 chapters, a total of 750 pages covering major

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workspaces of Fusion 360 such as DESIGN, ANIMATION, and DRAWING. The textbook teaches you to use Fusion 360 mechanical design software for building parametric 3D solid components and assemblies as well as creating animations and 2D

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drawings. This edition of textbook has been developed using Autodesk Fusion 360 software version: 2.0.9313 (November 2020 Product Update). This textbook not only focuses on the usages of the tools/commands of Fusion 360 but also on the concept of design.

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Every chapter in this textbook contains tutorials that provide users with step-by-step instructions for creating mechanical designs and drawings with ease. Moreover, every chapter ends with hands-on test drives that allow users to experience for themselves the

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user friendly and powerful capacities of Fusion 360. Table of Contents:

Chapter 1. Introducing Fusion 360

Chapter 2. Drawing Sketches with

Autodesk Fusion 360 Chapter 3.

Editing and Modifying Sketches

Chapter 4. Applying Constraints and

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Dimensions Chapter 5. Creating Base Feature of Solid Models Chapter 6. Creating Construction Geometries Chapter 7. Advanced Modeling - I Chapter 8. Advanced Modeling - II Chapter 9. Patterning and Mirroring Chapter 10. Editing and Modifying 3D

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Models Chapter 11. Working with Assemblies - I Chapter 12. Working with Assemblies - II Chapter 13. Creating Animation of a Design Chapter 14. Working with Drawings
SOLIDWORKS 2017 Advanced Techniques

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*Beginner's Guide to SOLIDWORKS
2022 - Level II*

*Beginner's Guide to SOLIDWORKS
2020 - Level II*

*Beginner's Guide to SOLIDWORKS
2018 - Level I*

Part, Assembly, Drawings, Sheet

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Metal, and Surfacing

The complete SolidWorks reference-tutorial for beginner to advanced techniques Mastering SolidWorks is the reference-tutorial for all users.

Packed with step-by-step instructions, video tutorials for over

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40 chapters, and coverage of little-known techniques, this book takes you from novice to power user with clear instruction that goes beyond the basics. Fundamental techniques are detailed with real-world examples for hands-on learning, and

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the companion website provides tutorial files for all exercises. Even veteran users will find value in new techniques that make familiar tasks faster, easier, and more organized, including advanced file management tools that simplify and streamline

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pre-flight checks. SolidWorks is the leading 3D CAD program, and is an essential tool for engineers, mechanical designers, industrial designers, and drafters around the world. User friendly features such as drag-and-drop, point-and-click, and

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cut-and-paste tools belie the software's powerful capabilities that can help you create cleaner, more precise, more polished designs in a fraction of the time. This book is the comprehensive reference every SolidWorks user needs, with

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tutorials, background, and more for beginner to advanced techniques.

Get a grasp on fundamental SolidWorks 2D and 3D tasks using realistic examples with text-based tutorials Delve into advanced functionality and capabilities not

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commonly covered by how-to guides Incorporate improved search, Pack-and-Go and other file management tools into your workflow Adopt best practices and exclusive techniques you won't find anywhere else Work through this

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book beginning-to-end as a complete SolidWorks course, or dip in as needed to learn new techniques and time-saving tricks on-demand. Organized for efficiency and designed for practicality, these tips will remain useful at any stage of

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expertise. With exclusive coverage and informative detail, Mastering SolidWorks is the tutorial-reference for users at every level of expertise. Beginner's Guide to SolidWorks 2014 – Level II starts where Beginner's Guide – Level I ends,

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following the same easy to read style and companion video instruction, but this time covering advanced topics and techniques. The purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create

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components in the context of an assembly and reference other components (Top-down design), propagate design changes with SolidWorks' parametric capabilities, mold design, welded structures, and more while

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explaining the basic concepts of each trade to allow you to understand the how and why of each operation. The author uses simple examples to allow you to better understand each command and environment, as well as to make it

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easier to explain the purpose of each step, maximizing the learning time by focusing on one task at a time. This book is focused on the processes to complete the modeling of a part, instead of focusing on individual software commands or

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operations, which are generally simple enough to learn. At the end of this book, you will have acquired enough skills to be highly competitive when it comes to designing with SolidWorks, and while there are many less frequently

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used commands and options available that will not be covered in this book, rest assured that those covered are most of the commands used every day by SolidWorks designers. The author strived hard to include the commands required in

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the Certified SolidWorks Associate test as listed on the SolidWorks website, and some, as well as several more.

This book is intended to help new users learn the basic concepts of **SOLIDWORKS** and good solid

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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

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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

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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,

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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

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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

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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. Includes Video

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Instruction Each copy of this book includes access to video instruction. In these videos the author provides a visual presentation of tutorials found in the book. The videos reinforce the steps described in the book by allowing you to watch the exact

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steps the author uses to complete the exercises.

Autodesk Fusion 360: Introduction to Surface and T-Spline Modeling textbook has been designed for instructor-led courses as well as self-paced learning. It is intended to help

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engineers and designers interested in learning Autodesk Fusion 360 for creating complex shape real-world models by using surface and T-Spline modeling techniques. This textbook is a great help for Autodesk Fusion 360 users who are

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new to surface and T-Spline modeling. It consists of a total of 232 pages covering the Surface and Form/Sculpt environments of Autodesk Fusion 360. It teaches users to use Autodesk Fusion 360 mechanical design software for

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creating complex shapes, three-dimensional surfaces and T-Spline models of zero thickness. This edition of textbook has been developed using Autodesk Fusion 360 software version: 2.0.10811 (August 2021 Product Update). This

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textbook not only focuses on the usage of the tools and commands of Autodesk Fusion 360 for creating surface and T-Spline models but also on the concept of design. Every chapter in this textbook contains

Tutorials followed by theoretical

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description, that provide users with step-by-step instructions for creating surface designs and sculpting with T-Spline surfaces. Moreover, every chapter ends with Hands-on Test Drives which allow users to experience the user friendly and

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powerful capacities of Autodesk Fusion 360.

Beginner's Guide to SOLIDWORKS 2019 - Level I

Autodesk Fusion 360: A Power Guide for Beginners and Intermediate Users (4th Edition)

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Beginner's Guide to SOLIDWORKS
2021 - Level II

Beginner's Guide to SolidWorks
2014 - Level II

SOLIDWORKS 2018 Basic Tools

• **Starts at an introductory level, designed for beginners** •

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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 •

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Features a quick reference guide and a Certified SOLIDWORKS Associate practice exam • The first book of a three book series SOLIDWORKS 2022 Basic Tools is the first book in a

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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

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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

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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

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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

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for This book is for the beginner who is not familiar with the SOLIDWORKS program and its add ins. SOLIDWORKS 2018 Basic Tools is the first book in a three part series. It introduces

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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

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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.

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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

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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 Design with

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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

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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

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engineering environment. Follow the step-by-step instructions and develop multiple parts and assemblies that combine machined, plastic and sheet metal components. Formulate the

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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

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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

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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

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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

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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

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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

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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

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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

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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

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Tutorials contained in SOLIDWORKS 2017. Beginner's Guide to SOLIDWORKS 2019 - Level II starts where Beginner's Guide - Level I ends, following the same easy to read style and

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companion video instruction, but this time covering advanced topics and techniques. The purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create

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components in the context of an assembly and reference other components (Top-down design), propagate design changes with SOLIDWORKS' parametric capabilities, mold design, welded structures and

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more while explaining the basic concepts of each trade to allow you to understand the how and why of each operation. The author uses simple examples to allow you to better understand each

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command and environment, as well as to make it easier to explain the purpose of each step, maximizing the learning time by focusing on one task at a time. This book is focused on the processes to complete the

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modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. At the end of this book, you will have acquired enough skills to be

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highly competitive when it comes to designing with SOLIDWORKS, and while there are many less frequently used commands and options available that will not be covered in this book, rest

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assured that those covered are most of the commands used every day by SOLIDWORKS designers. The author strived hard to include many of the commands required in the Certified SOLIDWORKS

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**Professional Advanced and
Expert exams as listed on the
SOLIDWORKS website.
SOLIDWORKS 2016 Basic
Tools
with Computer Applications
SOLIDWORKS 2018 Tutorial**

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with Video Instruction

**SolidWorks 2021 - Step-By-
Step Guide**

Autodesk Fusion 360:

**Introduction to Surface and T-
Spline Modeling**

Beginner's Guide to

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SOLIDWORKS 2020 – Level II starts where Beginner's Guide – Level I ends, following the same easy to read style and companion videoinstruction, but this time covering advanced

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topics and techniques. The purpose of this book is to teach advanced techniques including sheet metal, surfacing, how to create components in the context of an assembly and

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reference other components (Top-down design), propagate design changes with SOLIDWORKS' parametric capabilities, mold design, welded structures and more while

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explaining the basic concepts of each trade to allow you to understand the how and why of each operation. The author uses simple examples to allow you to better understand

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each command and environment, as well as to make it easier to explain the purpose of each step, maximizing the learning time by focusing on one task at a time. This book

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enough to learn. At the end of this book, you will have acquired enough skills to be highly competitive when it comes to designing with SOLIDWORKS, and while

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there are many less frequently used commands and options available that will not be covered in this book, rest assured that those covered are most of the commands used

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every day by SOLIDWORKS designers. The author strived hard to include many of the commands required in the Certified SOLIDWORKS Professional Advanced and Expert exams

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as listed on the SOLIDWORKS website.

SOLIDWORKS 2017: A Power Guide for Beginners and Intermediate User textbook is designed for instructor-led courses as well as for

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self-paced learning. It is intended to help engineers and designers interested in learning SOLIDWORKS for creating 3D mechanical design. Taken together, this textbook can be a

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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 major

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environments of SOLIDWORKS: Sketching environment, Part modeling environment, Assembly environment, and Drawing environment, which teach you how to use the

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SOLIDWORKS mechanical design software to build parametric models and assemblies, and how to make drawings of those parts and assemblies. Moreover, this textbook

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includes the topic of Configurations. This textbook not only focuses on the usages of the tools/commands of SOLIDWORKS but also on the concept of design. Every

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chapter of this textbook contains tutorials which instruct users how things can be done in SOLIDWORKS step by step. Moreover, every chapter ends with hands-on test drives which

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allow users to experience themselves the ease-of-use and powerful capabilities of SOLIDWORKS. Table of Contents: Chapter 1. Introduction to SOLIDWORKS Chapter 2. Drawing

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Sketches with SOLIDWORKS
Chapter 3. Editing and
Modifying Sketches Chapter
4. Applying Geometric
Relations and Dimensions
Chapter 5. Creating
First/Base Feature of

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***Solid Models Chapter 6.
Creating Reference
Geometries Chapter 7.
Advanced Modeling - I
Chapter 8. Advanced
Modeling - II Chapter 9.
Patterning and Mirroring***

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Chapter 10. Advanced Modeling - III Chapter 11. Working with Configurations Chapter 12. Working with Assemblies - I Chapter 13. Working with Assemblies - II Chapter

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***14. Working with Drawings
Main Features of the
Textbook Comprehensive
coverage of tools Step-by-
step real-world tutorials
with every chapter Hands-
on test drives to enhance***

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the skills at the end of every chapter Additional notes and tips Customized content for faculty (PowerPoint Presentations) Free learning resources for faculty and students

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***Additional student and faculty projects Technical support for the book:
info@cadartifex.com
SOLIDWORKS 2017 in 5 Hours with video instruction introduces the new user to***

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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

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for senior capstone, machine design, kinematics, dynamics, and other engineering and technology projects that use SOLIDWORKS as a tool. Engineers in industry are

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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

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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.

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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

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investigate the proper design intent and constraints. The mini Stirling Engine is based on the external combustion, closed cycle engine of Scottish

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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

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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

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motion study, develop detailed part and assembly drawings and much more. Engineering Design and Graphics with SolidWorks 2016 shows students how to use SolidWorks to create

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engineering drawings and designs. The textbook has been updated to cover the new features in SolidWorks 2016. It focuses on the creation of engineering drawings, including

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dimensions and tolerances and the use of standard parts and tools. Each chapter contains step-by-step sample problems that show students how to apply the concepts presented in

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the chapter. Effective pedagogy throughout the text helps students learn and retain concepts: Objectives: Each chapter begins with objectives and an introduction to the

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material. Summaries: Each chapter concludes with a summary and exercise problems. Numerous illustrations: The multitude of illustrations, accompanied

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by explanatory captions, present a visual approach to learning. Students see in the text what they see on the screen with the addition of explanatory text. Practical

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application: The text provides hundreds of exercise projects of varying difficulty (far more than any other computer graphics text). These exercises reinforce

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each chapter's content and help students learn by doing. Flexibility: With the hundreds of problems presented in the book, instructors can assign different problems within

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the same class and from year to year without repeating problems for students. Meets standards: The text teaches ANSI standards for dimensions and tolerances. This helps

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students understand how their designs are defined for production and the importance of proper tolerancing. Step-by-step approach: In presenting the fundamentals of

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engineering drawing using SolidWorks, the text uses a step-by-step approach that allows students to work and learn at their own pace.

SOLIDWORKS 2019 Tutorial

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***Mastering SolidWorks
Sheet Metal, Top Down
Design, Weldments,
Surfacing and Molds
SOLIDWORKS Surface Design
2021 for Beginners and
Intermediate Users***

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Solidworks 2013 Bible

Autodesk Inventor 2022: A Power Guide for Beginners and Intermediate Users textbook has been designed for instructor-led courses as well as self-paced learning. It is intended to help

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engineers and designers, interested in learning Autodesk Inventor, to create 3D mechanical designs. This textbook is an excellent guide for new Inventor users and a great teaching aid for classroom training. It consists of 14 chapters and a

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total of 790 pages covering major environments of Autodesk Inventor such as Sketching environment, Part modeling environment, Assembly environment, Presentation environment, and Drawing environment. The textbook

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teaches you to use Autodesk Inventor mechanical design software for building parametric 3D solid components and assemblies as well as creating animations and 2D drawings. This textbook not only focuses on the usages of the

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tools/commands of Autodesk Inventor but also on the concept of design. Every chapter in this textbook contains Tutorials that provide users with step-by-step instructions for creating mechanical designs and drawings with ease.

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Moreover, every chapter ends with Hands-on Test Drives that allow users to experience for themselves the user friendly and powerful capacities of Autodesk Inventor. This book is intended to help new users learn the basic concepts of

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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

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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

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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

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Materials. The book focuses on the processes to complete the modeling of a part, instead of focusing on individual software commands or operations, which are generally simple enough to learn. Throughout this book the author

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introduces you to new commands that are required to pass the Certified SOLIDWORKS Associate exam, as listed on the SOLIDWORKS website. A dedicated chapter provides you with details about the exam, as well

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as a practice test to help you prepare for the actual exam. SOLIDWORKS is an easy to use CAD software that includes many time saving tools that will enable new and experienced users to complete design tasks faster than

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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.

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Introduction to Mechanism Design: with Computer Applications provides an updated approach to undergraduate Mechanism Design and Kinematics courses/modules for engineering students. The use of web-based simulations, solid

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modeling, and software such as MATLAB and Excel is employed to link the design process with the latest software tools for the design and analysis of mechanisms and machines. While a mechanical engineer might brainstorm with a

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pencil and sketch pad, the final result is developed and communicated through CAD and computational visualizations. This modern approach to mechanical design processes has not been fully integrated in most books, as it is in

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this new text.

SOLIDWORKS 2016 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

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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

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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

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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

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take it step by step.

Creo Parametric 4.0 Advanced Tutorial

Engineering Design with SOLIDWORKS 2018 and Video Instruction

Introduction to Mechanism Design

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Autodesk Inventor 2022: A Power Guide for Beginners and Intermediate Users

Full Color Edition

Engineering Design with SOLIDWORKS 2018 and video instruction is written to

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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

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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

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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,

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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

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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

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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.

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