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# Introduction Solid Modeling Using Solidworks

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Via SolidWorks

Introduction to Solid Modeling Using SolidWorks 2015

Mastering Surface Modeling with SOLIDWORKS 2021

SOLIDWORKS 2019 Tutorial

Learn SOLIDWORKS 2020

Parametric Modeling with SolidWorks 2013

Basic through Advanced Techniques

Introduction to Finite Element Analysis Using SOLIDWORKS Simulation 2015

Space Modeling with SolidWorks and NX

Introduction to Solid Modeling Using SolidWorks 2016

Mastering SolidWorks

Parametric Modeling with SOLIDWORKS 2021

Introduction to Solid Modeling Using Solidworks 2012

Introduction to Solid Modeling Using SolidWorks 2016

Product Design Modeling using CAD/CAE

SOLIDWORKS 2020 Tutorial

Introduction to Solid Modeling Using SolidWorks 2015

Introduction to Solid Modeling Using SOLIDWORKS® 2017

Introduction to Solid Modeling Using SolidWorks 2008

Finite Element Analysis Concepts

Introduction to Solid Modeling Using Solidworks 2018 14e

Introduction to Solid Modeling Using SOLIDWORKS 2019

Introduction to Solid Modeling Using SOLIDWORKS 2020

Introduction to Solid Modeling Using SolidWorks 2017

SolidWorks 2014 Tutorial with Video Instruction

ISE EBook Online Access for Introduction to Solid Modeling Using SolidWorks 2021

Mastering Surface Modeling with SOLIDWORKS 2020

Machining Simulation Using SOLIDWORKS CAM 2020

The Computer Aided Engineering Design Series

Introduction to Solid Modeling Using SolidWorks 2011

Cad/cam With Creo Parametric: Step-by-step Tutorial For Versions 4.0, 5.0, And 6.0

Principles and Practice An Integrated Approach to Engineering Graphics and

AutoCAD 2020

Introduction to Solid Modeling Using SolidWorks 2015

Engineering Design with SOLIDWORKS 2021

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## **ZIMMERMAN BRYNN**

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**Via SolidWorks** SDC  
Publications  
SOLIDWORKS 2019  
Tutorial is written to assist  
students, designers,  
engineers and  
professionals who are new  
to SOLIDWORKS. The text  
provides a step-by-step,  
project based learning  
approach. It also contains

information and examples  
on the five categories in  
the CSWA exam. The book  
is divided into four  
sections. Chapters 1 - 5  
explore the SOLIDWORKS  
User Interface and  
CommandManager,  
Document and System  
properties, simple and  
complex parts and  
assemblies, proper design  
intent, design tables,  
configurations, multi-  
sheet, multi-view  
drawings, BOMs, and

Revision tables using  
basic and advanced  
features. In chapter 6 you  
will create the final robot  
assembly. The physical  
components and  
corresponding Science,  
Technology, Engineering  
and Math (STEM)  
curriculum are available  
from Gears Educational  
Systems. All assemblies  
and components for the  
final robot assembly are  
provided. Chapters 7 - 10  
prepare you for the

Certified Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs from subtractive manufacturing, and its features. You will also learn the terms and technology used in low cost 3D printers. Follow the step-by-step instructions and develop

multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, apply proper design intent, design tables and configurations. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each chapter.

Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

**Introduction to Solid Modeling Using SolidWorks 2015**  
McGraw-Hill Education  
Introduction to Solid Modeling using SolidWorks primarily consists of chapter-long

tutorials, which introduce both basic concepts in solid modeling (such as part modeling, drawing creation, and assembly modeling) and more advanced applications of solid modeling in engineering analysis and design (such as mechanism modeling, mold creation, sheet metal bending, and rapid prototyping). Each tutorial is organized as “keystroke-level” instructions, designed to teach the use of the software.

#### Mastering Surface

Modeling with SOLIDWORKS 2021  
Springer  
SOLIDWORKS 2018 Tutorial with video instruction is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The text provides a step-by-step, project based learning approach. It also contains information and examples on the five categories, to take and understand the Certified Associate - Mechanical Design (CSWA) exam. The book is divided into four sections.

Chapters 1 - 5 explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. In chapter 6 you will create the final robot assembly. The physical components and corresponding Science, Technology, Engineering

and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and components for the final robot assembly are provided. Chapters 7 - 10 prepare you for the Certified Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs

from subtractive manufacturing, and its features. You will also learn the terms and technology used in low cost 3D printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns,

copied components, apply proper design intent, design tables and configurations. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each chapter. Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

SOLIDWORKS 2019 Tutorial SDC Publications Introduction to Solid Modeling Using SolidWorks® 2011 presents "keystroke-level" tutorials, providing users new to the SolidWorks® program with all the detail they need to become confident using the software. Topics are illustrated and infused with examples from the real world such as flanges, brackets, helical springs, and more. Additionally, this easy-to-use guide has modular chapters, allowing for

flexible organization of a course or self-study. Accessible and updated for the newest version of software, Introduction to Solid Modeling Using SolidWorks® 2011 by Howard and Musto relates solid modeling exercises to engineering concepts in a way that introduces the engineering design process while simultaneously building student proficiency with a state-of-the-art software tool. *Learn SOLIDWORKS 2020* Springer The book discusses the

theoretical fundamentals of CAD graphics to enhance readers' understanding of surface modeling and free-form design by demonstrating how to use mathematical equations to define curves and surfaces in CAD modelers. Additionally, it explains and describes the main approaches to creating CAD models out of 3D scans of physical objects. All CAD approaches are demonstrated with guided examples and supported with comprehensive engineering explanations.

Furthermore, each approach includes exercises for independent consolidation of advanced CAD skills. This book is intended for engineers and designers who are already familiar with the basics of modern CAD tools, e.g. feature based and solid based modeling in 3D space, and would like to improve and expand their knowledge and experience. It is also an easy-to use guide and excellent teaching and research aid for academics and practitioners alike.

*Parametric Modeling with SolidWorks 2013* SDC Publications  
Parametric Modeling with SolidWorks 2013 contains a series of sixteen tutorial style lessons designed to introduce SolidWorks 2013, solid modeling and parametric modeling techniques and concepts. This book introduces SolidWorks 2013 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 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 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 SolidWorks 2013 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 are included on the Certified SolidWorks Associate (CSWA) Examination. Reference guides located at the front of the book and in each chapter show where these performance tasks are covered.

*Basic through Advanced Techniques* SDC Publications

This book will teach you all the important concepts and steps used to conduct

machining simulations using SOLIDWORKS CAM. SOLIDWORKS CAM is a parametric, feature-based machining simulation software offered as an add-in to SOLIDWORKS. It integrates design and manufacturing in one application, connecting design and manufacturing teams through a common software tool that facilitates product design using 3D solid models. By carrying out machining simulation, the machining process can be defined and verified early in the product design stage.

Some, if not all, of the less desirable design features of part manufacturing can be detected and addressed while the product design is still being finalized. In addition, machining-related problems can be detected and eliminated before mounting a stock on a CNC machine, and manufacturing cost can be estimated using the machining time estimated in the machining simulation. This book is intentionally kept simple. It's written to help you become familiar with the

practical applications of conducting machining simulations in SOLIDWORKS CAM. This book provides you with the basic concepts and steps needed to use the software, as well as a discussion of the G-codes generated. After completing this book, you should have a clear understanding of how to use SOLIDWORKS CAM for machining simulations and should be able to apply this knowledge to carry out machining assignments on your own product designs. In order

to provide you with a more comprehensive understanding of machining simulations, the book discusses NC (numerical control) part programming and verification, as well as introduces applications that involve bringing the G-code post processed by SOLIDWORKS CAM to a HAAS CNC mill and lathe to physically cut parts. This book points out important, practical factors when transitioning from virtual to physical machining. Since the machining capabilities

offered in the 2020 version of SOLIDWORKS CAM are somewhat limited, this book introduces third-party CAM modules that are seamlessly integrated into SOLIDWORKS, including CAMWorks, HSMWorks, and Mastercam for SOLIDWORKS. This book covers basic concepts, frequently used commands and options required for you to advance from a novice to an intermediate level SOLIDWORKS CAM user. Basic concepts and commands introduced

include extracting machinable features (such as 2.5 axis features), selecting a machine and cutting tools, defining machining parameters (such as feed rate, spindle speed, depth of cut, and so on), generating and simulating toolpaths, and post processing CL data to output G-code for support of physical machining. The concepts and commands are introduced in a tutorial style presentation using simple but realistic examples. Both milling and turning operations are included.

One of the unique features of this book is the incorporation of the CL data verification by reviewing the G-code generated from the toolpaths. This helps you understand how the G-code is generated by using the respective post processors, which is an important step and an excellent way to confirm that the toolpaths and G-code generated are accurate and useful.

**Introduction to Finite Element Analysis Using SOLIDWORKS Simulation 2015** SDC

Publications Introduction to Solid Modeling using SolidWorks primarily consists of chapter-long tutorials, which introduce both basic concepts in solid modeling (such as part modeling, drawing creation, and assembly modeling) and more advanced applications of solid modeling in engineering analysis and design (such as mechanism modeling, mold creation, sheet metal bending, and rapid prototyping). Each tutorial is organized as

"keystroke-level" instructions, designed to teach the use of the software. This new edition has been fully updated for the SolidWorks 2018 software package. All tutorials and figures have been modified for the new version of the software. Additional resources are available online at [www.mhhe.com/howard2018](http://www.mhhe.com/howard2018). Included on the website are tutorials for three popular SolidWorks Add-Ins, SolidWorks® Simulation, SolidWorks® Motion™ and PhotoView360. Instructors

can also access PowerPoint files for each chapter, the book figures in PowerPoint format, model files for all tutorials, and end-of-chapter problems, as well as a teaching guide. What's New: -Video tutorials accompany several chapters and introduce the chapter's content by showing visual examples -Fully updated text to reflect newest version of SOLIDWORKS -Tutorials and figures have been updated for the new version of the software *Space Modeling with*

*SolidWorks and NX* SDC Publications  
Introduction to Solid Modeling Using SolidWorks McGraw-Hill College  
*Introduction to Solid Modeling Using SolidWorks 2016* Academic Press  
Young engineers are often required to utilize commercial finite element software without having had a course on finite element theory. That can lead to computer-aided design errors. This book outlines the basic theory, with a minimum of

mathematics, and how its phases are structured within a typical software. The importance of estimating a solution, or verifying the results, by other means is emphasized and illustrated. The book also demonstrates the common processes for utilizing the typical graphical icon interfaces in commercial codes. In particular, the book uses and covers the widely utilized SolidWorks solid modeling and simulation system to demonstrate applications in heat

transfer, stress analysis, vibrations, buckling, and other fields. The book, with its detailed applications, will appeal to upper-level undergraduates as well as engineers new to industry.

Mastering SolidWorks SDC Publications  
Principles and Practices An Integrated Approach to Engineering Graphics and AutoCAD 2020 combines an introduction to AutoCAD 2020 with a comprehensive coverage of engineering graphics principles. By adopting

this textbook, you will no longer need to adopt separate CAD and engineering graphics books for your course. Not only will this unified approach give your course a smoother flow, your students will also save money on their textbooks. What's more, the tutorial exercises in this text have been expanded to cover the performance tasks found on the AutoCAD 2020 Certified User Examination. The primary goal of Principles and Practices An Integrated Approach to Engineering

Graphics and AutoCAD 2020 is to introduce the aspects of engineering graphics with the use of modern Computer Aided Design/Drafting software - AutoCAD 2020. This text is intended to be used as a training guide for students and professionals. The chapters in the text proceed in a pedagogical fashion to guide you from constructing basic shapes to making complete sets of engineering drawings. This text takes a hands-on, exercise-intensive approach to all the

important concepts of Engineering Graphics, as well as in depth discussions of CAD techniques. This textbook contains a series of thirteen chapters, with detailed step-by-step tutorial-style lessons designed to introduce beginning CAD users to the graphic language used in all branches of technical industry. The CAD techniques and concepts discussed in the text are also designed to serve as the foundation to the more advanced parametric feature-based

CAD packages, such as Autodesk Inventor. After completing this text your students will be prepared to pass the AutoCAD Certified User Examination. Certified User Reference Guides located at the front of the book and in each chapter show where these performance tasks are covered. McGraw-Hill College The purpose of this book is to introduce the reader to 3D CAD/CAM modelling using Creo™ Parametric (Creo) software. This concise textbook consists

of ten lessons covering the basics in Part and Assembly Modelling, Mould Design, NC Simulation, and Engineering Drawings. Each lesson provides essential knowledge and guides the user through the process of performing a practical exercise or task. The modelling philosophy, implementation of corresponding features, and commands behind each exercise are explained and presented in a step-by-step manner. The material is richly

illustrated with screenshots and icons from the software interface to facilitate the learning process. Suitable for beginners and intermediate users, CAD/CAM with Creo Parametric enables the reader to make a quick start in learning how to use complex 3D CAD/CAM software such as Creo in engineering design and manufacturing. The aim is to develop an understanding of the main modelling principles and software tools as a basis for independent learning

and solving more complex engineering problems. *Parametric Modeling with SOLIDWORKS 2021* McGraw-Hill Education The new edition of Introduction to Solid Modeling Using SolidWorks 2015 has been fully updated for the SolidWorks 2015 software package. All tutorials and figures have been modified for the new version of the software. The eleventh edition of this text primarily consists of chapter-long tutorials, which introduce both basic concepts in solid

modeling and more advanced applications of solid modeling in engineering analysis and design. Each tutorial is organized as "keystroke-level" instructions, designed to teach the use of the software. While these tutorials offer a level of detail appropriate for new professional users, this text was developed to be used as part of an introductory engineering course, taught around the use of solid modeling as an integrated engineering design and analysis tool.

Features such as: Design Intent Boxes and Future Study Boxes, help to integrate the concepts learned in solid modeling into the overall study of engineering. Additional resources are also available with this text at [www.mhhe.com/howard2015](http://www.mhhe.com/howard2015). Included on the website are tutorials for three popular SolidWorks Add-Ins, SolidWorks Simulation, SolidWorks Motion and PhotoView 360, and the book figures in PowerPoint format. Instructors can also access PowerPoint files for

each chapter and model files for all tutorials and end-of-chapter problems as well as a teaching guide.

[Introduction to Solid Modeling Using Solidworks 2012](#) SDC Publications

The primary goal of [Introduction to Finite Element Analysis Using SOLIDWORKS Simulation 2015](#) is to introduce the aspects of Finite Element Analysis (FEA) that are important to engineers and designers. Theoretical aspects of FEA are also introduced as they are needed to help better



understand the operation. The primary emphasis of the text is placed on the practical concepts and procedures needed to use SOLIDWORKS Simulation in performing Linear Static Stress Analysis and basic Modal Analysis. This text covers SOLIDWORKS Simulation and the lessons proceed in a pedagogical fashion to guide you from constructing basic truss elements to generating three-dimensional solid elements from solid models. This text takes a hands-on, exercise-

intensive approach to all the important FEA techniques and concepts. This textbook contains a series of fourteen tutorial style lessons designed to introduce beginning FEA users to SOLIDWORKS Simulation. The basic premise of this book is that the more designs you create using SOLIDWORKS Simulation, the better you learn the software. With this in mind, each lesson introduces a new set of commands and concepts, building on previous lessons.  
SDC Publications

Introduction to Solid Modeling using SolidWorks primarily consists of chapter-long tutorials, which introduce both basic concepts in solid modeling (such as part modeling, drawing creation, and assembly modeling) and more advanced applications of solid modeling in engineering analysis and design (such as mechanism modeling, mold creation, sheet metal bending, and rapid prototyping). Each tutorial is organized as “keystroke-level”

instructions, designed to teach the use of the software. This new edition has been fully updated for the SolidWorks 2016 software package. All tutorials and figures have been modified for the new version of the software. Additional resources are available online at [www.mhhe.com/howard2016](http://www.mhhe.com/howard2016). Included on the website are tutorials for three popular SolidWorks Add-Ins, SolidWorks® Simulation, SolidWorks® Motion™ and PhotoView360, and the book figures in PowerPoint

format. Instructors can also access PowerPoint files for each chapter, model files for all tutorials, and end-of-chapter problems, as well as a teaching guide. [Introduction to Solid Modeling Using SolidWorks 2016](#) McGraw-Hill Higher Education

- Teaches SOLIDWORKS users advanced surface modeling skills
- Includes tips and techniques for hybrid modeling
- Uses clear, step-by-step instructions to help you create real-world projects
- Covers how to make

molded parts and repair and patch surfaces

Mastering Surface Modeling with SOLIDWORKS 2020 focuses on surfacing tools, an important aspect of SOLIDWORKS' design capabilities that fills in the gaps that might be left by using solid modeling alone. If you are a SOLIDWORKS user currently relying on solid modeling for designs, or are just not familiar with surface modeling techniques, this book will add these skills to your repertoire to help you

create the highest-quality models. For instructors teaching this advanced skillset, this book's proven techniques, practical examples and training files will give students a broad understanding of the procedures needed to build freeform shapes and place them well on their way to creating sophisticated surface designs of their own. This manual is one of only a few on the market completely dedicated to mastering surfacing tools. Each of the ten chapters has clean, clear

instructions with plentiful diagrams to lead you through carefully selected exercises based on the author's own work experience and techniques. You are guided from a review of surfacing basics, to advanced surface modeling of real-world objects, to an explanation and example of hybrid modeling, to surface repairs and patches. Peruse the table of contents and pick and choose the chapters you are interested in or complete all chapters

consecutively to give you an in-depth understanding of all the tools and procedures needed to create surface designs. The projects you will work on in this book include a shoehorn, computer mouse, phone case, a modem housing, and stents. Woven into each of these are procedures, approaches and solutions for possible issues that might arise when you are using surfacing tools. These can be applied to any project you create. Each project touches on a variety of frequently used

commands such as extrude, loft, boundary, and sweep; surface revolved, filled, split, and knit; using deform and configurations; mirroring bodies; creating an axis, curve driven and circular patterns, fillets, and molded parts. Look for the post-it notes next to commands for helpful tips and definitions. Throughout the book, you will learn techniques of hybrid modeling, the combination of surface and solid modeling. The last part of the book takes it one step further.

Chapter 8 examines hybrid modeling in-depth, guiding you step-by-step from a 2D sketch to the final product, a handle housing. The last two chapters focus on molded parts, creating and saving visual properties of models and how to repair faulty surfaces. The advanced surfacing tools and techniques in this book give you the confidence to tackle projects using hybrid modeling. It is the best method to take full advantage of SOLIDWORKS' modeling

power and create more complex designs. *Product Design Modeling using CAD/CAE* Introduction to Solid Modeling Using SolidWorks Through a series of step-by-step tutorials and numerous hands-on exercises, this book aims to equip the reader with both a good understanding of the importance of space in the abstract world of engineers and the ability to create a model of a product in virtual space – a skill essential for any

designer or engineer who needs to present ideas concerning a particular product within a professional environment. The exercises progress logically from the simple to the more complex; while Solid Works or NX is the software used, the underlying philosophy is applicable to all modeling software. In each case, the explanation covers the entire procedure from the basic idea and production capabilities through to the real model; the conversion from 3D model to 2D

manufacturing drawing is also clearly explained. Topics covered include modeling of prism, axisymmetric, symmetric and sophisticated shapes; digitization of physical models using modeling software; creation of a CAD model starting from a physical model; free form surface modeling; modeling of product assemblies following bottom-up and top-down principles; and the presentation of a product in accordance with the rules of technical documentation. This book,

which includes more than 500 figures, will be ideal for students wishing to gain a sound grasp of space modeling techniques. Academics and professionals will find it to be an excellent teaching and research aid, and an easy-to-use guide.

### **SOLIDWORKS 2020**

**Tutorial** SDC Publications Product Design Modeling using CAD/CAE is the third part of a four-part series. It is the first book to integrate discussion of computer design tools throughout the design

process. Through this book, you will: Understand basic design principles and all digital design paradigms Understand computer-aided design, engineering, and manufacturing (CAD/CAE/CAM) tools available for various design-related tasks Understand how to put an integrated system together to conduct all-digital design (ADD) Provides a comprehensive and thorough coverage of essential elements for product modeling using the virtual engineering

paradigm Covers CAD/CAE in product design, including solid modeling, mechanical assembly, parameterization, product data management, and data exchange in CAD Case studies and tutorial examples at the end of each chapter provide hands-on practice in implementing off-the-shelf computer design tools Provides two projects showing the use of Pro/ENGINEER and SolidWorks to implement concepts discussed in the book

Introduction to Solid Modeling Using SolidWorks 2015 World Scientific

This text presents a tutorial-based introduction to solid modeling and the SOLIDWORKS software. Although the tutorials can be followed by anyone interested in learning the software, it is geared toward freshman engineering students or high school students interested in engineering. Accordingly, the examples and problems are based on the authors'

experience with teaching engineering students. This text primarily consists of chapter-long tutorials, which introduce both basic concepts in solid modeling (such as part modeling, drawing creation, and assembly modeling) and more advanced applications of solid modeling in engineering analysis and design (such as mechanism modeling, mold creation, sheet metal bending, and rapid prototyping). Each tutorial is organized as “keystroke-level”

instructions, designed to teach the use of the software.  
Introduction to Solid Modeling Using SOLIDWORKS® 2017  
McGraw-Hill  
Engineering Design with SOLIDWORKS 2021 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.

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