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Engineering graphics with SOLIDWORKS 2020 are written to help students, designers, engineers and professionals who are new to SOLIDWORKS. The book combines the basics of engineering graphics and measurement techniques with a step-by-step project based on the SOLIDWORKS learning approach. The book is divided into four sections with 11 Chapters. Chapters 1 - 3: Explore the history of engineering graphics, manual sketch methods, orthographic projection, third vs. first-corner projection, multidisciplinary drawings, measurement practices (ASME Y14.5-2009 standard), type of line, fit type, tolerance, fastening in general, common flow notes and CAD leads to the development of SOLIDWORKS. Chapters 4 - 9: Understand solidWORKS user interface and CommandManager, document and system properties, simple machine parts, simple and complex assemblies, proper design intent, table design, configuration, multi-sheet, multifunctional drawings, BOMs, and revision tables using basic and advanced features. Follow step-by-step instructions in more than 80 eight-part development events, four sub-collections, three drawings, and six document templates. Chapter 10: Preparing for the SolidWORKS Associate (CSWA) exam. Understanding the curriculum and categories of the CSWA exam and the required model knowledge required to pass the exam successfully. Chapter 11: Provide a basic understanding between additive and subtraction production. Discuss fusion filater technology (FFF), STereoLithography (SLA) and selective laser caking technology (SLS). Choose the right thread material. Understand the terminology of a 3D printer. Knowledge of the preparation, preservation and printing of the model on the 3D printer Fused Filament Fabrication. SolidWORKS (CSWA-AM) Certified Additive Manufacturing Information. See individual features, commands, and tools with SOLIDWORKS help. Chapter exercises analyze and examine the competencies of using the goals of the chapter. The book is designed in addition to solidWORKS tutorials on the SOLIDWORKS help menu. Each project lists the desired results and competencies for use. Know your goals beyond your goal. Follow the step-by-step procedure to achieve your design goals. Work between multiple documents, features, teams and properties that represent how engineers and designers use SOLIDWORKS in the industry. The author developed industry scenarios, combining his own industry experience with the knowledge of engineers, department managers, suppliers and manufacturers. Comprehensive Handbook for SOLIDWORKS 2020 - Contains 260 plus standalone tutorials - Begins with a basic review of SOLIDWORKS 2020 and its new features - Tutorials written for each theme with new and intermediate users in mind, includes access to the original and final state of each tutorial - Contains You to 3D printing SOLIDWORKS 2020 Reference Guide presents a comprehensive guide written to help the newcomer intermediate user SOLIDWORKS 2020. SOLIDWORKS is a huge software package, and no book can cover all topics for all users. This book provides a centralized reference space for many of the tools FEATURES and techniques SOLIDWORKS 2020. This book covers the following: System and Document Properties - FeatureManagers - PropertyManagers - ConfigurationManagers - RenderManagers - 2D and 3D Sketch Tools - Sketch Entities - 3D Tools - Motion Research - Sheet Metal - Motion Research - SOLIDWORKS - PhotoView 360 - Pack and Go - 3D PDF Files - Intelligent Modeling Techniques - 3D Printing Terminology and More Chapter 1 Provides Basic Overview of Concepts and Terminology used in this book using SOLIDWORKS 2020 software. If you're completely new to SOLIDWORKS, you should read Chapter 1 in detail and complete Lesson 1, Lesson 2 and Lesson 3 in SOLIDWORKS Tutorials. If you're familiar with the earlier release of SOLIDWORKS, you can still skim Chapter 1 to familiarize yourself with some of the commands, menus and features you haven't used; or you can just go to any section in any chapter. Each chapter contains detailed Information PropertyManager on key topics with individual standalone short tutorials to reinforce and demonstrate the functionality and simplicity of the SOLIDWORKS tool or function. The book provides access to more than 260 models, their solutions and additional supporting materials. Learn by doing, not just by reading. Formulate your skills to create, modify, and edit sketches and solid features. Learn how to reuse features, parts, and builds with symmetry, templates, copied components, design tables, configurations, and more. The book is designed in addition to online tutorials and online references contained in SOLIDWORKS 2020. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful projects. The author developed the textbooks, combining his own experience in the industry with the knowledge of engineers, department managers, professors, suppliers and manufacturers. He is directly associated with SOLIDWORKS every day, and his responsibilities go far beyond creating just a 3D model. Using step-by-step, a project based on tutorials designed for beginners or intermediate users - Will prepare you for a certified SOLIDWORKS Associate Exam - includes a chapter introducing you to a 3D-printed SOLIDWORKS 2020 Tutorial written to help students, designers, engineers and professionals who are new to SOLIDWORKS. In the text a step-by-step, project-based approach to learning. It also contains information and examples in five categories of the CSWA exam. Book Book divided into four sections. Chapters 1 - 5 explore the SOLIDWORKS user interface and CommandManager properties, documents and systems, simple and complex parts and assemblies, correct design intent, design tables, configurations, many leafy, multi-species drawings, BOMs and Revision tables using basic and advanced features. In Chapter 6, you'll create the ultimate robot assembly. The Physical Components and Relevant Science, Technology, Engineering and Mathematics (STEM) curriculum is available from Gears Educational Systems. All assemblies and components for the final assembly of the robot are provided. Chapters 7 - 10 prepare you for a certified associate-mechanical design (CSWA) exam. Certification points to the basis and learner knowledge of 3D CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing) as it differs from subtraction production and its features. You will also learn the conditions and technologies used in low-cost 3D printers. Follow the step-by-step instructions and develop several assemblies that combine more than 100 extruded parts and components. Formulate your skills to create, modify, and edit sketches and solid features. Learn how to reuse functions, parts, and builds using symmetry, templates, copied components, applying appropriate design intentions, design tables, and configurations. Learn by doing, not just by reading. Each chapter lists the desired results and competencies for use. Know your goal behind its 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 use SOLIDWORKS in the industry. SOLIDWORKS 2020 tutorial is written to help students, designers, engineers and professionals who are new to SOLIDWORKS. The text contains a project-based, project-based approach to learning. It also contains information and examples in five categories of the CSWA exam. The book is divided into four sections. 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Follow the step-by-step instructions and develop several assemblies that combine more than 100 extruded parts and components. Formulate your skills to create, modify, and edit sketches and solid features. Learn how to reuse functions, parts, and builds using symmetry, templates, copied components, applying appropriate design intentions, design tables, and configurations. Learn by doing, not just by reading. Each chapter lists the desired results and competencies for use. Know your goal behind its 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 use SOLIDWORKS in the industry. David Planchard is the founder and president of D and M Education LLC, an associate professor at Worcester Polytechnic Institute and a partner with SolidWorks Solution Partner since 1999. David has written numerous books on SolidWorks, many with a multimedia CD aimed at beginners and intermediate users. His books have been the top selling for the last nine years and are used by the best engineering schools around the world! Prior to his studies at DMM, David spent more than 27 years in industry and academia, holding various engineering, marketing and teaching positions and degrees. It has five U.S. patents and one international patent. He has published and is the author of numerous works on machine design, product design, materials mechanics, and solid modeling. He is an active member of the SolidWorks user group and the Cisco Regional Academy user group. Dave's goal is to write clear, concise and well-illustrated Step-by-Step SolidWorks books for professionals, colleges, universities, technical institutes, and secondary schools. Learn by doing, not just by reading! David is certified by BSME, MSM and Certified SolidWorks Professional (CSWP). David is a partner at SolidWorks Solution and an adjunct at Worcester Polytechnic Institute. Institute.

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