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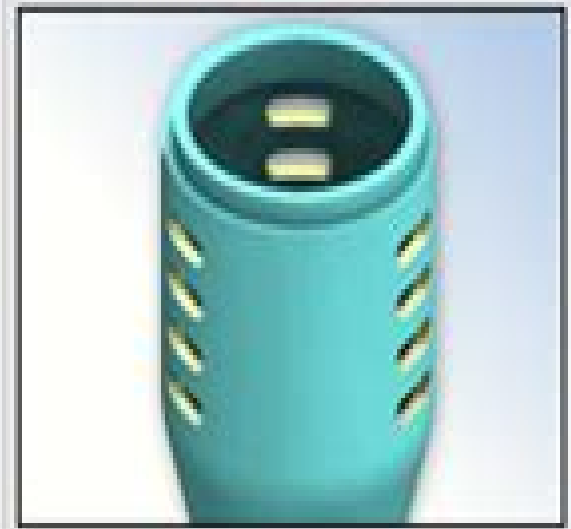


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Solidworks Tutorials Guide 2010

David C. Planchard, Marie P. Planchard

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Planchard, 2010-04-06 Drawing and Detailing with SolidWorks 2010 is written to educate and assist students designers engineers and professionals in the drawing and detailing tools of SolidWorks Explore the learning process through a series of design situations industry scenarios projects and objectives targeted towards the beginning to intermediate SolidWorks user Work through numerous activities to create multiple view multiple sheet detailed drawings and assembly drawings Develop Drawing templates Sheet formats and Custom Properties Construct drawings that incorporate part configurations assembly configurations and design tables Manipulate annotations in parts drawings assemblies Revision tables Bills of Materials and more Apply your drawing and detailing knowledge to over thirty exercises The exercises test your usage competency as well as explore additional topics with industry examples Advanced exercises require the ability to create parts and assemblies Drawing and Detailing with SolidWorks 2010 is not a reference book for all drafting and drawing techniques The book

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SOLIDWORKS 2019 Tutorial David Planchard, 2018-12-21 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

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SolidWorks 2012 Tutorial David C. Planchard, Marie P. Planchard, 2012 SolidWorks 2012 Tutorial with Video Instruction is target towards a technical school two year college four year university or industry professional that is a beginner or intermediate CAD user The text provides a student who is looking for a step by step project based approach to learning SolidWorks with an enclosed 1 5 hour video instruction DVD SolidWorks model files and preparation for the CSWA exam The book is divided into two sections Chapters 1 7 explore the SolidWorks User Interface and CommandManager Document and System properties simple machine parts simple and complex assemblies design tables configurations multi sheet multi view drawings BOMs Revision tables using basic and advanced features along with Intelligent Modeling Techniques SustainabilityXpress SimulationXpress and DFMXpress Chapters 8 11 prepare you for the new Certified SolidWorks Associate Exam CSWA The CSWA certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles 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 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

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