

Download SOLIDWORKS 2018 Tutorial with Video Instruction free David C. Planchard CSWP (PDF, ePub, Mobi)

SOLIDWORKS 2018 Tutorial with Video Instruction

A Step-by-Step Project Based Approach Utilizing 3D Solid Modeling



David C. Planchard, CSWP,
SOLIDWORKS Accredited Educator

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- Uses step-by-step, project based tutorials designed for beginning or intermediate users
- Comes with training videos demonstrating all the basic tools needed to start using SOLIDWORKS
- Will prepare you for the Certified SOLIDWORKS Associate Exam
- Includes a chapter introducing you to 3D printing
- This edition features an expanded section on 3D printing

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.

New in the 2018 Edition

Chapter 11 on additive manufacturing was updated with new vendor and technology changes.

The videos cover the following topics:

- Start a SOLIDWORKS 2018 session
- Understand the SOLIDWORKS 2018 Interface
- Create 2D Sketching, Sketch Planes and use Sketch tools
- Create 3D Features and apply Design Intent
- Create an Assembly
- Create fundamental Drawings Part 1 & Part 2

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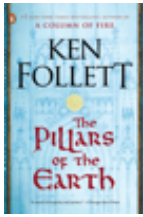
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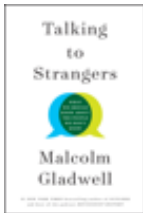
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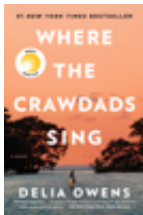
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