

Autodesk Inventor 2017 (R1): Introduction to Solid Modeling - Part 1: Autodesk Authorized Publisher

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Note: This book is continued in *Autodesk Inventor 2017 (R1): Introduction to Solid Modeling - Part 2*.

The *Autodesk Inventor 2017 (R1): Introduction to Solid Modeling* student guide provides you with an understanding of the parametric design philosophy through a hands-on, practice-intensive curriculum. You will learn the key skills and knowledge needed to design models using Autodesk Inventor, starting with conceptual sketching, through to solid modeling, assembly design, and drawing production.

Topics Covered in Part 1:

- Understanding the Autodesk Inventor software interface
- Creating, constraining, and dimensioning 2D sketches
- Creating and editing the solid base 3D feature from a sketch
- Creating and editing secondary solid features that are sketched and placed
- Creating equations and working with parameters
- Manipulating the display of the model
- Resolving feature failures
- Duplicating geometry in the model

Prerequisites

As an introductory training guide, *Autodesk Inventor 2017 (R1): Introduction to Solid Modeling* does not assume prior knowledge of any 3D modeling or CAD software. Students do need to be experienced with the Windows operating system and a background in drafting of 3D parts is recommended.

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