

## Autodesk Inventor Fundamentals: Introduction to Solid Modeling

The Autodesk® Inventor®: Introduction to Solid Modeling training 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:

- 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
- Placing and constraining/connecting parts in assemblies
- Manipulating the display of components in an assembly
- Duplicating components in an assembly
- Obtaining model measurements and property information
- Creating Presentation files (Exploded views and Animations)
- Modifying and analyzing the components in an assembly
- Simulating motion in an assembly
- Creating parts and features in assemblies
- Creating and editing an assembly Bill of Materials
- Working with projects
- Creating and annotating drawings and views
- Customizing the Autodesk Inventor environment

### Prerequisites:

As an introductory training guide, Autodesk Inventor: 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.