Autodesk Inventor Introduction to Solid Modelling

This course teaches new Inventor users how to approach parametric design using Autodesk Inventor. The course provides a hands-on learning experience ensuring that you will acquire the knowledge needed to design models from conceptual sketching, through to solid modelling, assembly design and drawing production.

Course Modules

- Creating, Constraining and Dimensioning 2D Sketches
- Creating and Editing the Solid Base of 3D Features 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 Assembly Parts
- Manipulating Display of Components in an Assembly
- Obtaining Model Measurements
- Obtaining Property Information
- Creating Presentations (Exploded Views)
- Modifying and Analysing Components in an Assembly
- Simulating Motion in an Assembly
- Creating Parts and Features in an Assembly
- Creating and Editing an Assembly
- Bill of Materials
- Working with Projects
- Creating and Annotating Drawings and Views
- Customising the Autodesk Inventor Environment

Prerequisites

Inventor Introduction to Solid Modelling does not assume prior knowledge of any 3D modelling or CAD software, although a background in drafting of 3D parts is recommended.

Course Duration

2 + 2 days

Next Steps

Autodesk Inventor Sheet Metal Design