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