

Autodesk Inventor Introduction to Solid Modelling - Virtual Training

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*

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.

A connection to our Virtual Training Room, using a Microsoft Windows based PC with a stable Internet connection with a minimum 2-4mb/s download speed.

Course Duration and Running Times

This four day course is scheduled to run one day a week for four weeks. Each day will follow a similar format and is split into four training sessions. A typical day will look like this, although the times may vary depending on the course and delegates.

- 09:00 - Introductions or Recap, Hardware and Session Overview
- 09:30 - Course Module
- 11:00 - Break
- 11:30 - Course Module
- 13:00 - Lunch Break
- 14:00 - Course Module
- 15:30 - Break
- 16:00 - Course Module
- 17:30 - Close

Next Steps

Autodesk Inventor Sheet Metal Design