

# Autodesk Inventor Introduction to Solid Modelling

## COURSE DESCRIPTION

This course teaches new Inventor users how to approach parametric design using Autodesk Inventor. The course provides a hands-on learning experience to help you 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
- 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



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## OTHER INFORMATION

### Prerequisites

A background in drafting of 3D parts is recommended.

### Course Duration

4 Days (Split into 2 + 2)

### Next Steps

Autodesk Inventor Sheet Metal Design

