



PREREQUISITES

For online training, students should have access to a machine with Inventor installed and activated. Having a dual monitor setup is highly recommended.

INCLUSIONS

6 half days of Online Training and Tuition, Cadgroup Certificate of Completion, and a Inventor Fundamentals eBook.

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AUTODESK® INVENTOR

Fundamentals

The Autodesk® Inventor Introduction to Solid Modelling training course provides you with an understanding of the parametric design philosophy through hands-on, practice intensive curriculum. You will learn the key skills and knowledge needed to create design models using Autodesk® Inventor, starting with sketching, through to solid modelling, assembly design and drawing production.

COURSE OUTLINE

- Introduction to Parametric Modelling with Inventor:
Understanding the Inventor interface; viewing and navigating the 3D modelling; 3D model view and display modes.
- Creating, constraining and dimensioning 2D sketches:
Using the sketching tools; establishing geometric constraints; creating dimensional constraints; sketch editing tool.
- Creating & Editing 3D Base Features:
Creating extruded and revolved features; creating sweet and lofted features; creating and using work features.
- Creating & Editing Secondary Solid Features:
Creating chamfers and fillets; creating holes and threads; creating other feature types.
- Creating Equations and Working with Parameters
- Manipulating the Display of the Model:
Reordering and suppressing features; using section views and design views; dimensioning using model based definition annotation.
- Managing and Checking Features
- Duplicating Geometry in the Model:
Creating rectangle and circular feature patterns; creating sketch driven patterns; creating mirrored features.
- Placing & Constraining/Connecting Parts in Assemblies
- Manipulating the Display of Components in an Assembly
- Using Assembly Tools
- Obtaining and Adding Model Information
- Simulating Motion in an Assembly:
Driving constraints and joints; using contact solver and interference tools.
- Creating Parts & Features in an Assembly
- Creating & Editing Assembly Bill of Materials
- Creating Presentation Files (Exploded Views)
- Creating & Annotating Drawings & Views
- Customising the Inventor Environment