

Autodesk Inventor Essentials

Overview

The course will provide students with the best usage approaches for parametric design philosophy through a hands-on, practice-intensive curriculum. Students acquire the knowledge needed to complete the process of designing models from conceptual sketching, through to solid modeling, assembly design, and drawing production.

Prerequisites

It is recommended that delegates have a basic working knowledge of one or more of the following:

- Drafting, design or mechanical engineering principles.
 - Microsoft Windows XP, Microsoft Windows Vista, Microsoft Windows 7 or Microsoft Windows 8.
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Available Exams and Certifications

- Autodesk Certified User
 - Autodesk Certified Professional
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Course Accreditations

4 Credits (By the South African Institute of Mechanical Engineers -SAIMechE)

Course Outline

Inventor in a nutshell

- A brief interactive session to provide a feel for the products capability and enthuse the delegates from the outset

Introducing Inventor as a BIM tool

- Political positioning of Inventor within the wider BIM context, this is a lecture only and will often be taken out and delivered as part of an executive overview
- What is BIM and what does it mean?
- The benefits of BIM
- What will BIM deliver?
- Introducing levels of BIM
- Industry drivers Introducing Levels of BIM Implications on team and workflow; fee and deliverables; contract and insurance issues

UI Tour

- Introduction to the Autodesk Inventor interface, project setup, and the concept of parametric 3D design; as well as exploring the general workflow of modelling parts, creating detailed drawings and assembly of the parts, and then detailing the assembly

Parametric Sketching Options

- This module explores the principles of creating parameter-driven sketches for use in modelling features and parts. It looks at creating part features using basic 2D and 3D sketches, including the tools and settings that govern their creation, along with using AutoCAD data to create sketches
- This exercise covers best practices for working with sketches, it provides a number of practical examples on sketching, from new and existing sketches, dynamic prompting, dimensions, annotation, constraints, projecting geometry and utilising 3D geometry.

Basic Part Modelling

- This module focuses on the configuration of the general options and settings associated with Inventor and in particular part files. It continues by taking a closer look at the tools and techniques involved with the creation of a 3D parametric part
- This exercise provides a number of practical examples on using the extrude functionality and the revolve tools to create parts, together with using work planes, axes and points to create helper geometry. The exercise continues with using fillets, threads and holes to ultimately define a part.

Reusing Parts and Features

- This module introduces iParts and iFeatures and looks at improving efficiencies through the re-use of already created parts, it examines the different workflows involved for reusing parts and features such as link parameters for maximum consistency and design efficiency
- This exercise begins with labelling parameters prior to creating an iPart using the iProperties table, it introduces Sheet Metal iParts, colour, iFeatures to create a part. The exercise concludes with placing components into an assembly prior to publishing the component.

Advanced Modelling Techniques

- This module examines some of the primary tools and the workflows used for multi-body part creation, from simple extrusion to complex combinations of features such as curvy modelling techniques using 2D paths, sweeps and lofts, it further explores tolerances, parameters and iProperties.
- This exercise provides a practical example of creating paths using the sweep tool and the path and guide rails options prior to using the loft, shell and sculpt tools. The exercise continues with multiple body part sculpting, creating derived components and applying patterns.

Assembly Design Workflows

- This module provides an insight into this key concept of Autodesk Inventor design, how assembly relationships using constraints and joint tools are created, how to organise designs using structured sub-assemblies, using adaptive components, managing bills of materials, how to substitute parts for sub-assemblies, working with assembly design accelerators and generators as well as design calculations
- This exercise begins with creating and then applying a number of the common constraints before using driving relationships to simulate motion, using a number of joint tools are explored prior to considering efficient assembly workflows. The exercise concludes with a practical look at bills of materials, iAssembly and the bolted connection generator.

Creating Drawings and Annotations

- This module introduces the drawing manager and the creation of templates and styles, part and assembly drawings, annotation, iParts and iAssembly drawings as well as how to share the drawings outside of the workgroup in a variety of formats such as DWF and PDF
- This exercise begins with the creation of a drawing template, define a sheet size and create a title block prior to linking the model and iProperties. Drawing views, dimensions and annotation properties are used to produce different views for sheet metal and weldment. The exercise concludes with view drawing view references to iParts and iAssembly.

Presentations and Exploded Views

- This module explores the presentation and storyboard tools available and how these are used to create exploded views of assemblies as well creating animated assembly instructions, repair manuals and part replacement diagrams. It looks at 3D views and how to publish the animated assemblies as videos or animated DWF files
- This exercise begins with creating a presentation file from a template before selecting an assembly to create a storyboard, it continues with the view and performing some linear tweaks to produce a stunning representation of an assembly.

Large Assemblies

- This module considers large assemblies and how these are managed and opened quickly using the Express mode tools. Although departments may take a different view on what constitutes a large assembly, this module will explore the tools, tips, settings and LOD strategies available to improve best practice performance with operational benefits
- This exercise looks at using subassemblies within upper-level assemblies to reduce assembly constraints, this eliminates the number of redundant calculations Inventor must make to solve the model, and improved organisation and the re-use of parts with the demote option.

Sheet Metal Parts

- This is an extremely powerful toolset of Inventor that is focused on productivity and capturing manufacturing requirements. This module looks at how to create accurate sheet-metal models and flat patterns as well as how to create documentation and set up sheet-metal styles and templates
- This exercise begins with extruding the contour flange tool before revolving the contour roll option prior to using the folded part tool. It continues with applying the flange and hem tools together with options for cut, punch, chamfer, bend, seam, rip and unfold. The exercise concludes with sheet metal rules, bend tables and parts lists.

The Frame Generator

- The frame generator comprises a number of tools that combine skeletons and sub-assemblies to automate frame modelling. This module looks at member location, member insertion and different treatments to frame ends. It looks at how to get the most out of this utility when creating structural frames from the Autodesk Inventor library of common shapes and how to publish profiles and Bills of Materials (BOM's)
- This exercise provides a practical example of creating a frame design from scratch, it takes an assembly file containing two parts and a frame sub-assembly, one of the parts contains an unconsumed sketch, this is used to embed a monument base to support a frame that has already been trimmed and mitred.

Weldment Design

- This module builds upon the understanding of parts and assemblies, it explore the Autodesk Inventor weldment modelling environment and the weldment design and documentation tools. Beginning with weldment workflows this module provides tips and tricks as it looks at applying weld preparations, weld beads, welding symbols together with how to document and quantify weldment design
- This exercise begins with creating a weld preparation, it continues with cosmetic welds, fillet welds, groove welds and machining before applying weld symbols and defining parts in a Bill of Material (BOM)