



AUTODESK FUSION 360 TRAINING

Objectives

This course will cover principles of 3D parametric part design, assembly design and creating production-ready part, and assembly drawing by using Autodesk Fusion 360. After completing this course user will be able to:

- ✓ Understand the design process in Autodesk Fusion 360
- ✓ Able to design model from conceptual sketching through solid modelling
- ✓ Create multiple designs using several of tools.
- ✓ Understand how to assembly parts.
- ✓ Understand how to use work plane on X, Y, Z axis.

Pre-requisites

No previous CAD experience is necessary. It is recommended that you have a working knowledge of:

- Microsoft® Windows® 7, Microsoft® Windows® 8 or Microsoft® Windows® 10.

TRAINING PROGRAMME DAY 1

Chapter 1: Getting Started

- Autodesk Fusion 360 User Interface
- Data Panel Interface
- Navigating the model
- Understanding workspace
- Design history

Chapter 3: Sculpting

- Creating a T-Spline form
- Modifying a T-Spline form
- Add details to T-Spline form
- Editing T-Spline form

Chapter 2: Intermediate Sketching

- Creating a new sketch
- Creating geometry in sketch
- Understanding sketch constraints
 - Creating Circles
 - Sketching Rectangles
 - Creating Arcs
 - Sketching Polygons
- Projecting Geometry
- Editing Sketch Geometry

TRAINING PROGRAMME DAY 2

Chapter 4: Solid Modelling

- Using Quick Shape Creation
- Using the Extrude Feature
- Using the Revolve Feature
- Using the Sweep Feature
- Using the Rib Feature
- Using the Press/Pull Feature
- Using Thicken to Create a Solid from Surfaces
- Working with Parameters

Chapter 5: Editing Your Model

- Editing a Solid - Fillet and Chamfer
- Editing a Solid - Rule Fillets
- Editing a Solid - Hole
- Editing a Solid - Shell
- Editing a Solid - Draft
- Editing a Solid - Combine
- Editing a Solid - Split
- Editing a Solid - Delete
- Editing a Solid - Move
- Editing a Solid - Replace Face
- Editing a Solid - Align
- Duplicating a Solid - Mirror
- Duplicating a Solid - Pattern

TRAINING PROGRAMME DAY 3

Chapter 6: Model Assembly

- Understanding Components in Fusion 360
- Creating Components
- Reusing Components
- Positioning Components
- Working with Components in the Browser
- Introducing Joints - Using the Rigid Joint Type
- Working with the Different Types of Joints
- Creating Slider and Planar Joints
- Understanding Joint Origins
- Creating Rigid Groups
- Understanding Contact Sets
- Enabling Motion Linking
- Driving Joints

Chapter 5: Drawing

- About Fusion 360 Drawings
- Creating a Drawing of a model
- Creating Views
- Create & Edit Annotations
- Drawing Settings and Preferences
Output the Drawing