

Autodesk Fusion 360

Target Audience

This course is designed for product designers, engineers, and manufacturing professionals who want to leverage Autodesk Fusion 360 for 3D modeling, simulation, and collaborative product development. It is ideal for beginners and professionals transitioning to cloud-based CAD/CAM/CAE tools.

Course Objective

The objective of this course is to equip participants with the skills to design, simulate, and document 3D models, while utilizing Fusion 360's integrated tools for collaborative product development and digital prototyping.

Course Outcome

- Learn to navigate the Fusion 360 interface and create parametric 3D models with precision.
- Develop skills in assembling components and analyzing motion and interference in designs.
- Understand how to perform simulations for stress, thermal, and motion analysis to optimize designs.
- Gain proficiency in generating technical drawings, rendering models, and preparing designs for manufacturing using CAM tools.

Course Outline: The course comprises **40-hours** of theory and labs and is divided into **14** different chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Chapter 1. Introducing Fusion 360

- Installing Fusion 360
- Getting Started with Fusion 360
- Working with User Interface of Fusion 360
- Invoking a New Design File
- Working with Workspaces
- Managing Data by Using the Data Panel
- Saving a Design File
- Exporting a Design to Other CAD Formats
- Opening an Existing Design File
- Working in the Offline Mode
- Recovering Unsaved Data
- Sharing a Design
- Invoking a Marking Menu
- 3D Printing

Chapter 2. Drawing Sketches with Autodesk Fusion 360

- Invoking a New Design File
- Creating Sketches
- Working with Selection of Planes
- Specifying Units
- Specifying Grids and Snaps Settings
- Drawing a Line Entity
- Drawing a Tangent Arc by Using the Line Tool
- Drawing a Rectangle
- Drawing a Circle
- Drawing an Arc
- Drawing a Polygon
- Drawing an Ellipse
- Drawing a Slot
- Drawing Conic Curves
- Drawing a Spline
- Editing a Spline
- Adding Fit/Control Points in a Spline
- Controlling the Curvature Display of a Spline
- Creating Sketch Points
- Inserting Text into a Sketch

Chapter 3. Editing and Modifying Sketches

- Trimming Sketch Entities
- Extending Sketch Entities
- Offsetting Sketch Entities
- Creating Construction Entities
- Mirroring Sketch Entities
- Patterning Sketch Entities
- Creating a Sketch Fillet
- Creating a Sketch Chamfer
- Scaling Sketch Entities
- Breaking Sketch Entities

Chapter 4. Applying Constraints and Dimensions

- Working with Constraints
- Applying Constraints
- Controlling the Display of Constraints
- Applying Dimensions
- Modifying/Editing Dimensions
- Working with Different States of a Sketch
- Working with SKETCH PALETTE

Chapter 5. Creating Base Features of Solid Models

- Creating an Extrude Feature
- Creating a Revolve Feature
- Navigating a 3D Model in Graphics Area

Chapter 6. Creating Construction Geometries

- Creating a Construction Plane
- Creating a Construction Axis
- Creating a Construction Point

Chapter 7. Advanced Modeling - I

- Using Advanced Options of the Extrude Tool
- Using Advanced Options of the Revolve Tool
- Working with a Sketch having Multiple Profiles
- Projecting Edges onto a Sketching Plane
- Creating 3D Curves

Editing a Feature and its Sketch
Editing the Sketching Plane of a Sketch
Applying Physical Material Properties
Customizing Material Properties
Calculating Mass Properties
Measuring the Distance between Objects

Chapter 8. Advanced Modeling - II

Creating a Sweep Feature
Creating a Loft feature
Creating Rib Features
Creating Web Features
Creating Emboss Features
Creating Holes
Creating a Thread
Creating a Rectangular Box
Creating a Cylinder
Creating a Sphere
Creating a Torus
Creating a Helical and a Spiral Coil
Creating a Pipe
Creating 3D Sketches

Chapter 9. Patterning and Mirroring

Creating a Rectangular Pattern
Creating a Circular Pattern
Creating a Pattern along a Path
Mirroring Features/Faces/Bodies/Components

Chapter 10. Editing and Modifying 3D Models

Working with the Press Pull Tool
Creating Fillets
Creating Chamfers
Creating Shell Features
Adding Drafts
Scaling Objects
Combining Solid Bodies
Offsetting Faces of a Model
Splitting Faces of a Model
Splitting Bodies

Chapter 11. Working with Assemblies - I

- Working with Bottom-up Assembly Approach
- Working with Top-down Assembly Approach
- Creating an Assembly by Using Bottom-up Approach
- Inserting Components in a Design File
- Fixing/Grounding the First Component
- Working with Degrees of Freedom
- Applying Joints
- Editing Joints
- Editing Joint Limits
- Animating a Joint
- Animating the Model
- Locking/Unlocking the Motion of a Joint
- Driving a Joint
- Defining Relative Motion between Two Joints
- Grouping Components Together
- Enabling Contact Sets between Components
- Capturing the Position of Components

Chapter 12. Working with Assemblies - II

- Creating an Assembly by Using Top-down Approach
- Creating Components within a Design File
- Creating Features of an Empty Component
- Fixing/Grounding the First Component
- Applying As-Built Joints
- Defining a Joint Origin on a Component
- Editing Assembly Components

Chapter 13. Creating Animation of a Design

- Invoking the ANIMATION Workspace
- Capturing Views on the Timeline
- Capturing Actions on the Timeline
- Customizing Views and Actions on the Timeline
- Deleting Views and Actions of a Storyboard
- Creating a New Storyboard
- Toggling On or Off Capturing Views
- Playing and Publishing Animation

Chapter 14. Working with Drawings

- Invoking the DRAWING Workspace
- Creating the Base View of a Design
- Creating Projected Views
- Working with the Angle of Projection
- Defining the Angle of Projection
- Defining Drawing Preferences
- Editing Document and Sheet Settings
- Editing and Inserting a New Title Block
- Creating Auxiliary Views
- Creating Section Views
- Creating Detail Views
- Creating Break Views
- Creating an Exploded Drawing View
- Invoking DRAWING Workspace From Animation
- Editing Properties of a Drawing View
- Editing Hatch Properties of a Section View
- Creating a Sketch
- Moving a Drawing View
- Rotating a Drawing View
- Deleting a Drawing View
- Adding Geometries in Drawing Views
- Applying Dimensions
- Editing a Dimension
- Arranging Dimensions
- Breaking Dimension Lines
- Adding Text/Note
- Adding Text/Note With Leader
- Adding the Surface Texture Symbol
- Creating the Bill of Material (BOM)/Part List
- Adding Balloons Manually
- Renumbering Balloons
- Adding Drawing Sheets
- Creating a New Drawing Template
- Exporting a Drawing