



Computer Aided Design using SolidWorks

Day 1 Basic Solid Modeling

Introduction & Sketcher Tools

- a) CAD Tools and Applications: CAD - CAM - CAE
- b) Parametric Feature Based Modelling and Parent-Child Relation
- c) Design Intent and Associativity between 3 Modes
- d) SolidWorks - Getting Started & Graphical User Interface
- e) Sketch Entities and Tools
- f) Dimensioning and Adding Relations to define the Sketch

Sketched Features (Boss / Base and Cut)

- a) Base Features –
- b) Extrude & Revolve
- c) Reference Geometry, Curves & 3D Sketch
- d) Sweep & Loft

Editing & Refining Model

- a) Editing Sketch, Sketch Plane and Editing Feature
- b) Suppress / Un-Suppress Feature and Reordering Feature

Day 2 Advance Features

Applied Features

- a) Patterns & Mirror
- b) Fillet/Round & Chamfer
- c) Hole & Hole Wizard
- d) Draft, Shell, Rib and Scale
- e) Dome, Flex and Wrap

Multi Body

- a) Indent Tool
- b) Combine Bodies – Boolean Operations
- c) Split, Move/Copy and Delete Bodies

Other Tools & Options

- a) Design Table and Configurations
- b) Adding Equations and Link Values
- c) Tools - Measure and Mass Properties
- d) Appearance - Edit Material, Colour and Texture
- e) Options - System and Document Properties

Day 3 Surfacing Techniques

Basic Surface Creations

- a) Extrude & Revolve
- b) Sweep & Loft
- c) Boundary Surface
- d) Planar Surface

Other Derived Techniques

- a) Offset Surface
- b) Radiate Surface
- c) Ruled Surface
- d) Fill Surface
- e) Mid Surface

Modify / Edit Surfaces

- a) Fillet/Round
- b) Extend
- c) Trim & Untrim
- d) Knit Surfaces
- e) Delete and Patch

Surfaces for Hybrid Modelling

- a) Thicken – Boss / Base and Cut
- b) Replace face
- c) End condition for Sketched feature - Up to Surface or Offset from Surface.
- d) Solid body from closed surfaces

Day 4 Assembly & Mechanisms

Bottom up Assembly Approach

- e) Inserting Components/Sub-Assemblies
- f) Adding Mates - Standard & Advance
- g) Editing Mates, Part and Replacing Components

Top down Approach & Mechanisms

- a) Inserting New Part to Existing Assembly
- b) Use of Layout Sketching
- c) External References - In-context and Out-of-context, Locked and Broken

Assembly Features

- a) Component Patterns & Mirrors
- b) Cuts & Holes
- c) Belt/Chain and Weld Bead

Representations of Assembly Components

- a) Light Weight, Suppressed and Resolved

- b) Hide, Transparency and Isolate
- c) Exploded View

Assembly Check

- a) Interference Detection,
- b) Collision Detection and Physical Dynamics

Motion Study

- c) Assembly Motion & Physical Simulation
- d) Animation Wizard & Save as AVI file
- e) Mechanism Analysis – Plot Displacement, Velocity and Acceleration Diagram

Day 5 Detailed Drafting

Introduction to Engineering Drawings

- a) General Procedure for Drafting & Detailing –
- b) Inserting Drawing Views, Dimensioning and Adding Annotations
- c) Drawing Templates & Sheet Format
- d) Setting Options

Drawing Views

- a) Model View & Standard 3 View
- b) Projected View & Auxiliary View
- c) Section & Aligned Section View
- d) Detail View, Broken-out Section and Crop View.

Dimensioning

- a) Standards, Rules and Guidelines
- b) Dimension Insertion/Creation - Insert Model Items & Dimension tool

Annotations

- a) Notes & Holes Callout
- b) Datum & Geometric Tolerances
- c) Surface Finish & Weld Symbols
- d) Centre Mark & Centre line
- e) BOM Balloon & Bill of Material

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