CETPA INFOTECH PVT. LTD.
CURRICULUM OF SOLIDWORKS

SOLIDWORKS

INTRODUCTION OF DESIGN CONCEPT AND PROCEDURE

- Detailed Concept Of CAD
- Need & Importance Of CAD
- Overview About Actual Designing In Industries, Fundamentals Of Design And Its Implementation Methods
- All Characteristics Of Solidworks To User Friendly Atmosphere
- Superiority Of Solidworks With Its Use And Demand In Industries

---

TAKING THE SOLIDWORKS TOUR

- Introduction To Solidworks.
- System Requirements
- Starting Solidworks In Windows
- The Workbench Concept
- Workbenches In Solidworks
- Adjusting The Solidworks Interface
- Creating And Managing Workspace
- Graphic User Interface Of Solidworks
- Menu And Toolbars
- Opening Files
- Creating New Files
- Keyboard Shortcuts
- Selecting/Moving Objects With Mouse
- Working With Planes
- Properties Toolbar
- Changing The Properties
- Changing The Interface From 3d Modeling To 2d Sketching And Vice-Versa
- Uses & Description About Feature Manager Design Tree
- Working With Respect To Ucs.
- Setting Up The Document Options

SKETCHER

- Getting Started With Sketch
- Creating Centerlines
- Constructing Lines

- Constructing Ellipse
- Constructing a Circle
- Constructing an Arc
- Creating Slots
- Creating Polygon
- Creating a Parabola
- Creating a Spline
- Equation Driven Curve
- Point
- Creating Text
- Creating Construction Geometry
- Rapid Sketch

EDITING SKETECHES

- Sketch Fillet
- Sketch Chamfer
- Offsetting Entities
- Converting Entities
- Trim
- Extending Entities
- Jog
- Mirror
- Moving Sketch Entities
- Moving Sketch Entities
- Copying The Sketch Entities
- Rotating Sketch Entities
- Scaling Sketch Entities
- Stretching Sketch Entities
- Modify Sketch
- Close Sketch Of Model
- Sketch Picture
- Area Hatch / Fill
- Sketch Patterns
- Blocks
- Relations
- Automatic Relations
- Conflicts In Relations
- Dimensioning
- Dimension Property Manager
- Exiting The Sketch
- 3d Sketching

PART MODELING

- Terminologies Used In Part Modeling Environment
- Entering The Part Module
- Choosing The Sketch Plane
- Extruding Boss / Base Features
- Revolving Boss / Base Features
- Creating Sweep Features
- Creating Loft Features
- Creating Cut Features
- Selecting Geometrics In Solid Works

REFERENCE GEOMETRY

- Reference Planes
- Creating New Planes
- Creating Reference Axes
- Creating Reference Points
- Creating Reference Coordinate Systems
- Editing Reference Geometries
- Creating Curves

PLACED FEATURES

- Creating Simple Holes
- Creating Standard Holes Using The Hole Wizard
- Creating Fillets
- Creating Chamfers.
- Creating Shell Features
- Creating Rib Feathers
- Creating Draft Feature
- Creating Pattern

ASSEMBLY MODELING

- Types Of Assembly Design Approaches
- Working With Solid Works Assembly Bottom-Up Approach
- Positioning The Components In Assembly
- Assembly Mates
- Standard Mates
- Advanced Mates
- Mechanical Mates
- Smart Mates
- Mate Reference
- Replacing The Assembly Components
- Rotating A Component
- Moving Components
- Detecting Interference

- Assembly Pattern
- Assembly Mirror
- Creating Exploded View
- Physical Simulation
- Top Down Design
- Assembly Performance
- Configuration In Assembly
- Smart Components
- Smart Fasteners

DRAWING VIEWS & DETAILING

- Introduction Of Drawing
- Need & Importance Of Drawing
- Starting The Drawing Workbench
- Defining The Sheet & Sizes
- Adjusting Of Drawing Sheet According To Object/Assembly
- Types Of Projection
- Using Predefined Drawing Styles
- Scaling The Drafted View

DRAWING VIEWS

- Creating Drawing From Part Or Assembly
- Creating A New Drawing Document
- Generating Standard Views
- Derived Views
- Creating Broken Views
- Working With Assembly Specific View
- Drawing View Properties
- Manipulating Views

DETAILING

- Creating Dimensions
- Creating Model Dimension
- Creating Auto Dimension
- Dim-Xpert
- Creating Smart Dimension
- Creating Ordinate Dimension
- Creating Chamfer Dimension

DATA EXCHANGE

- Converting Files For Transferring
- Converting Into IGES, STEP,PARASOLID Etc.
- Convert Into Jpeg, Mpeg, Tiff, Pdf Files

SURFACE MODELING

- Creating Surface Features
- Creating Extrude Features
- Creating Revolved Surface
- Creating Swept Surface
- Creating Lofted Feature
- Boundary Surface
- Creating Planar Surface
- Creating Offset Surface
- Creating A Radiate Surface
- Extending A Surface
- Creating Surface Fill
- Creating Ruled Surfaces
- Trimming Surface
- Creating Mid- Surface
- Replacing Faces
- Deleting Faces
- Un-Trimming Surface
- Creating Knit Surface
- Thickening A Surface
- Move Faces

SHEET METAL MODELING

- Fundamentals Concepts Of Sheet Metal
- Using Sheet Metal Tools
- Creating Base Flange
- Creating Edge Flange
- Creating Miter Flange
- Creating Hem
- Creating Log
- Creating Break Corner/Corner-Trim
- Creating Closed Corners
- Creating Rip
- Creating Sketched Bend
- Creating Unfold/Fold
- Flattening Sheet Metal Bends
- Forming Tools
- Cross Break
- Welded Corner
- Inserting Corner Trim
- Creating Lofted Bends
- Creating Sheet Metal Part By Converting A Solid Body