CURRICULUM OF SOLIDWORKS

Duration 6 Weeks

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

SOLIDWORKS

- 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 SKETCHES

- 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
<table>
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<th>SURFACE MODELING</th>
<th>SHEET METAL MODELING</th>
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<td>● Fundamentals Concepts Of Sheet Metal</td>
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<td>● Creating Extrude Features</td>
<td>● Using Sheet Metal Tools</td>
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<td>● Creating Revolved Surface</td>
<td>● Creating Base Flange</td>
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<td>● Creating Swept Surface</td>
<td>● Creating Edge Flange</td>
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<td>● Creating Lofted Feature</td>
<td>● Creating Miter Flange</td>
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<td>● Boundary Surface</td>
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<td>● Creating Planar Surface</td>
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<td>● Creating Offset Surface</td>
<td>● Creating Break Corner/Corner-Trim</td>
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<td>● Creating A Radiate Surface</td>
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<td>● Extending A Surface</td>
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<td>● Creating Surface Fill</td>
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<tr>
<td>● Creating Ruled Surfaces</td>
<td>● Creating Sketched Bend</td>
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<tr>
<td>● Trimming Surface</td>
<td>● Creating Unfold/Fold</td>
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<td>● Creating Mid- Surface</td>
<td>● Flattening Sheet Metal Bends</td>
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<td>● Replacing Faces</td>
<td>● Forming Tools</td>
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<td>● Deleting Faces</td>
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<tr>
<td>● Un-Trimming Surface</td>
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<td>● Creating Knit Surface</td>
<td>● Inserting Corner Trim</td>
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<tr>
<td>● Thickening A Surface</td>
<td>● Creating Lofted Bends</td>
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<tr>
<td>● Move Faces</td>
<td>● Creating Sheet Metal Part By Converting A Solid Body</td>
</tr>
</tbody>
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