

ZENUS INFOTECH INDIA PVT. LTD.
CURRICULUM OF SolidWorks

Course Duration: 6/8 Weeks

Session 1	<p>Introduction to CAD, CAE, PDM</p> <p>Features of SolidWorks,</p> <p>Various products available in SolidWorks for Product Design, Simulation, Communication</p> <p>SolidWorks Graphical User Interface - Feature manager design tree, Callouts, Handles, Confirmation corner, mouse buttons, keyboard shortcuts, Command Manager,</p> <p>Hardware and Software requirements, SolidWorks Task Scheduler, SolidWorks Rx.</p>
Session 2	<p>Sketch Entities – Inference line, Centreline line, Line, Circle, Arc, Ellipse, Rectangle, Slots, Polygon, Parabola, Ellipse, Partial Ellipse, Spline, Spline tools, Spline on surface, Equation driven curve, Points, Text, Construction geometry, Snap, grid,</p>
Session 3	<p>Sketch Tools - Fillet, Chamfer, Offset, Convert entities, Trim, Extend, Split, Jog, Mirror, Dynamic Mirror, Move, Copy, Rotate, Scale, Stretch, Sketch pattern</p> <p>Make path, Close Sketch To Model, Sketch picture, Check Sketch for Feature, Area hatch/Fill</p> <p>Blocks – Make block, Edit block, Insert block, Add/Remove Entities, Rebuild, Save, Explode</p> <p>Relations - Adding Sketch Relation, Automatic relations,</p> <p>Dimensioning - Smart, Horizontal, Vertical, Ordinate, Horizontal ordinate, Vertical ordinate, Align ordinate, Fully define sketch.</p> <p>Sketch Diagnosis, Sketch pert, 3D Sketching, Rapid Sketch</p>
Session 4	<p>Part Modeling Tools</p> <p>Creating reference planes</p> <p>Creating Extrude features – Direction1, Direction2, From option, Thin feature, Applying draft, Selecting contours</p> <p>Creating Revolve features – Selecting Axis, Thin features, Selecting contours</p> <p>Creating Swept features-Selecting, Profile and Path, Orientation/twist type, Path Alignment, Guide Curves, Start/End tangency, Thin feature</p>
Session 5	<p>Creating Loft features – Selecting Profiles, Guide curves, Start/End Constraints,</p>

	<p>Centerline parameters, Sketch tools, Close loft.</p> <p>Selecting geometries – Selection Manager, Multiple Body concepts</p> <p>Creating Reference - points, axis, coordinates</p>
Session 6	<p>Creating curves -</p> <p>Split curve, Project curve, Composite curve, Curve through points, Helix and Spiral</p> <p>Creating Fillet features</p> <p>Inserting Hole types</p>
Session 7	<p>Creating Chamfer</p> <p>Creating Shell</p> <p>Creating Rib</p> <p>Creating Pattern - Linear pattern, Circular pattern, Sketch driven pattern, Curve driven pattern, Table driven pattern, Fill pattern, mirror</p> <p>Advanced Modeling Tools- Dome, Free form, Shape feature, Deform, indent, Flex</p>
Session 8	<p>Inserting Fastening features- Mounting boss, snap hook, Snap hook groove, Vent</p> <p>Environment & Utilities - Working with views and manipulating views, Trouble shooting</p> <p>Inserting Library feature, Adding Configuration, Inserting Design table, System options, Measuring Geometries, Calculating Mass Properties, Feature Statistics, Working With Equations</p>
Session 9	<p>Assembly Modeling Tools</p> <p>Introduction to Assembly Modeling & Approaches – Top down and Bottom up approach</p> <p>Applying Standard Mates- Coincident, Parallel, Perpendicular, Tangent, Concentric, Lock, Distance, Angle.</p>
Session 10	<p>Manipulating Components - Replacing Components, Rotating Components, Move Components, Collision Detection, Physical Dynamics, Dynamic Clearance, Detecting Interference</p> <p>Creating Pattern - Assembly Pattern, Mirror</p> <p>Creating Explode Views</p> <p>Top-Down Design – Layout Sketch, Work Part In the Context of an assembly.</p> <p>Smart Components, Smart Fasteners, Physical Simulation</p>
Session 11	<p>Surface Modeling tools</p> <p>Creating Extrude, Revolve, Swept, loft, Boundary surface.</p>

	<p>Inserting Planar Surface, Offset Surface, Radiate Surface.</p> <p>Extending a surface, Surface fill, Ruled Surface, Trimming Surface, Mid surface, Replace Face, Delete face, Untrim surface, Knit surface, Thickening a Surface, Move Face</p>
Session 12	<p>Generating Drawing Views</p> <p>Introduction To Angle Of Projection</p> <p>Generating Views - Generating Model View, Projected Views, Inserting Standard 3 View</p> <p>View creation relative to model, Inserting predefined views, empty views, Auxiliary Views, Detailed Views, Crop view, Broken –Out Section, Broken Views, Section View, Aligned Section View, Alternate Position View, Working assembly specific view, Drawing properties, Manipulating views</p>
Session 13	<p>Creating Dimensions – Smart, Horizontal, Vertical, Baseline, Ordinate, Horizontal Ordinate, Vertical Ordinate, Chamfer, Attach Dimensions, Align Collinear/Radial, Align Parallel/Concentric, Model Dimensions, Auto dimension, DimXpert, Annotations, Spell check</p>
Session 14	<p>Inserting Annotations - Datum Features, Geometric Tolerance, Surface Finish, Jog Leaders, Hole Callout, Datum Target, Dowel Pins, Area Hatch, Cosmetic Thread, Balloon, Centre Mark, Centre Lines, Layers, Working With Tables, Bill Of Materials, Hole Table, Sheets And Templates, Sheet Format.</p> <p>Sheet Metal Design</p> <p>Concepts in Sheet metal design bend allowance bend deduction, K-factor</p>
Session 15	<p>Inserting Base Flange, Sheet Metal Tab, Edge Flange, Miter Flange, Hem, Jog.</p> <p>Creating Break Corner/Corner Trim, Closed Corners, Rip.</p> <p>Inserting Sketched Bend, Fold/Unfold, Forming Tools.</p> <p>Inserting Cross Break, Welded Corner.</p> <p>Adding Corner Trim, Lofted bend</p> <p>Conversion Of Solid Body To Sheet Metal.</p> <p>Working with import data - Importing In SolidWorks, Editing Imported Features, Feature Recognition, 2d To 3d Conversion</p>