



TECHHUB SOLUTIONS

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SOLIDWORKS - SYLLABUS

Course Content
<u>SKETCHER</u> Sketch Entities – Inference line, Centerline 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, Sketch Tools - Fillet, Chamfer, Offset, Convert entities, Intersection curve, Face curve, Trim, Extend, Split, Jog Line, Construction Geometry, Mirror, Dynamic Mirror, Move, Copy, Rotate, Scale, Stretch, Sketch pattern, Polygon, Make path, Close Sketch To Model, Sketch picture, Check Sketch for Feature, Area hatch/Fill Blocks – Make block, Edit block, Insert block, Add/Remove Entities, Rebuild, Save, Explode Relations - Adding Sketch Relation, Automatic relations, Dimensioning - Smart, Horizontal, Vertical, Ordinate, Horizontal ordinate, Vertical ordinate, Align ordinate, Fully define sketch. Sketch Diagnosis, Sketch Xpert, 3D Sketching, Rapid Sketch
<u>PART MODELING</u> Part Modeling Tools Creating reference planes Creating Extrude features – Direction1, Direction2, From option, Thin feature, Applying draft, Selecting contours Creating Revolve features – Selecting Axis, Thin features, Selecting contours Creating Swept features -Selecting, Profile and Path, Orientation/twist type, Path Alignment, Guide Curves, Start/End tangency, Thin feature
Creating Loft features – Selecting Profiles, Guide curves, Start/End Constraints, Centerline parameters, Sketch tools, Close loft. Selecting geometries – Selection Manager, Multiple Body concepts Creating Reference - points, axis, coordinates
Creating curves - Split curve, Project curve, Composite curve, Curve through points, Helix and Spiral Creating Fillet features Inserting Hole types CREATING REFERENCE GEOMETRIES Reference, Geometry Reference Planes Creating New Planes Creating Reference Axes, Creating Reference Points, Creating Reference Coordinate Systems
Creating Chamfer, Shell, Rib Creating Pattern - Linear pattern, Circular pattern, Sketch driven pattern, Curve driven pattern, Table driven pattern, Fill pattern, mirror Advanced Modeling Tools - Dome, Free form, Shape feature, Deform, indent, Flex
<u>EQUATIONS, CONFIGURATIONS, AND LIBRARY FEATURES</u> Inserting Fastening features - Mounting boss, snap hook, Snap hook groove, Vent Environment & Utilities - Working with views and manipulating views, Trouble shooting Inserting Library feature, Adding Configuration, Inserting Design table, System options, Measuring Geometries, Calculating Mass, Properties, Feature Statistics, Working With

Equations
<u>ASSEMBLY MODELING</u> Introduction to Assembly Modeling & Approaches – Top down and Bottom up approach Applying Standard Mates - Coincident, Parallel, Perpendicular, Tangent, Concentric, Lock, Distance, Angle.
Applying Advanced Mates – Symmetric, Width, Path Mate, Linear/Linear Coupler, and Limit Mate. Applying Mechanical Mates – Cam, Hinge, Gear, Rack Pinion, Screw, and Universal Joint. Applying Smart mates, Applying Mate reference
Manipulating Components - Replacing Components, Rotating Components, Move Components, Collision Detection, Physical Dynamics, Dynamic Clearance, Detecting Interference, Creating Pattern - Assembly Pattern, Mirror Creating Explode Views Top Down Design – Layout Sketch, Work Part In the Context of an Assembly Smart Components, Smart Fasteners, Physical Simulation
<u>SURFACE MODELING</u> Surface Modeling tools Creating Extrude, Revolve, Swept, loft, Boundary surface. Inserting Planar Surface, Offset Surface, Radiate Surface. Extending a surface, Surface fill, Ruled Surface, Trimming Surface, Mid surface, Replace Face, Delete face, Untrim surface, Knit surface, Thickening a Surface, Move Face
<u>DRAFTING</u> Generating Drawing Views, Introduction To Angle Of Projection Generating Views - Generating Model View, Projected Views, Inserting Standard 3 View 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
Creating Dimensions – Smart, Horizontal, Vertical, Baseline, Ordinate, Horizontal Ordinate, Vertical Ordinate, Chamfer, Attach Dimensions, Align Collinear/Radial, Align Parallel/Concentric, Model Dimensions, Auto dimension, Dim Xpert, Annotations, Spell check 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.
<u>SHEET METAL</u> Sheet Metal Design Concepts in Sheet metal design bend allowance bend deduction, K- factor Inserting Base Flange, Sheet Metal Tab, Edge Flange, Miter Flange, Hem, Jog. Creating Break Corner/Corner Trim, Closed Corners, Rip. Inserting Sketched Bend, Fold/Unfold, Forming Tools. Inserting Cross Break, Welded Corner. Adding Corner Trim, Lofted Trim Conversion Of Solid Body To Sheet Metal. Working with import data - Importing In SolidWorks, Editing Imported Features, Feature Recognition, 2d To 3d Conversion
<u>SIMULATION EXPRESS</u> Stress or static analysis calculates the displacements, strains, and stresses in a part based on material, fixtures, and loads. A material fails when the stress reaches a certain level. Different materials fail at different stress levels. Simulation Xpress uses linear static analysis, based on the Finite Element Method, to calculate stresses. Linear static analysis makes several assumptions to calculate stresses in the part.

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