



General Course Outline

SOLIDWORKS – SYLLABUS:

Introduction to CAD, PDM Features of Solid Works,

Various products available in Solid Works for Product Design.

Solid Works Graphical User Interface:

Feature manager design tree, Callouts, Handles, Confirmation corner, mouse buttons, keyboard shortcuts, Command Manager, Hardware and Software requirements.

SKETCHER:

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.

PART MODELING:

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 Chamfer

Creating Shell

Creating Rib

Creating Pattern - Linear pattern, Circular pattern, Sketch driven pattern, Curve driven pattern, Table driven pattern, Fill pattern, mirror.

Environment & Utilities - Inserting Library feature, Adding Configuration, Inserting Design table, System options, Measuring Geometries, Calculating Mass Properties, Feature Statistics, Working With Equations.

Assembly Modeling Tools:

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, Limit Mate.

Manipulating Components - - Replacing Components, Rotating Components, Move Components, Collision Detection.

Creating Pattern - Assembly Pattern, Mirror.

Creating Explode Views

SURFACE MODELING:

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.

DRAFTING:

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, DimXpert, 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.

SHEET METAL:

Sheet Metal Design

Concepts in Sheet metal design bend allowance bend deduction, Kfactor

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.

WELDMENT DESIGN:

Weldment Design

Introduction to Weldment, 3D sketch, How to create user defined

profile for structural member? How to insert structural member? How

to apply gusset and fillet bead? How to insert end cap? Extruded

Boss/Base, Trim/Extend, mirror, placing holes, using different

structural members, Weldment cut list.