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ARMbedded CADD CENTRE



A Unit Of ARMbedded Electronics Pvt. Ltd.



Opp. Indraprastha Colony G. N. 1, Sonipat Road, Rohtak (HR)

JAS-ANZ



AN ISO 9001:2008
CERTIFIED COMPANY

SOLIDWORKS - SYLLABUS

SolidWorks Syllabus		Total Duration : 90 Hour
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, Hardware and Software requirements, SolidWorks Task Scheduler, SolidWorks Rx.</p>	
Session 2	<p><u>SKETCHER</u></p> <p>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,</p>	

Session 3	<p>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</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. Sketch Diagnosis, SketchXpert, 3D Sketching, Rapid Sketch</p>
Session 4	<p><u>PART MODELING</u></p> <p>Part Modeling Tools 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, 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 - 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><u>ASSEMBLY MODELING</u></p> <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>Applying Advanced Mates – Symmetric, Width, Path Mate, Linear/Linear Coupler, Limit Mate.</p> <p>Applying Mechanical Mates – Cam, Hinge, Gear, Rack Pinion, Screw, Universal Joint.</p> <p>Applying Smart mates</p> <p>Applying Mate reference</p>

<p>Session 11</p>	<p>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</p>
<p>Session 12</p>	<p><u>SURFACE MODELING</u></p> <p>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</p>
<p>Session 13</p>	<p><u>DRAFTING</u></p> <p>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</p>
<p>Session 14</p>	<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>

<p>Session 15</p>	<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>Session 16</p>	<p><u>SHEET METAL</u></p> <p>Sheet Metal Design Concepts in Sheet metal design bend allowance bend deduction, Kfactor</p> <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 Trim</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>
<p>Session 17</p>	<p><u>PRODUCT DATA MANAGEMENT</u></p> <p>Product Data Management Introduction to PDM, LAN, WAN, Server, client, user, administrator, advantages of using PDM vault admin. How to log in to PDM Vault Admin? About vault settings, Creating new group, user and accessing rights, PDM data storage, setting up revision schemes</p>

<p>Session 18</p>	<p>Creating new project, Check In/Check Out of a new document, viewing the configurations, How to Check In associated files, viewing document information, Check In documents to vault from local disk, adding notes to the document. How to change document to other group? Archive/Restore a document, Delete/Rollback a document.</p>
<p>Session 19</p>	<p>Attaching non SolidWorks document, How to switch vault to other user? How to customize display of a Vault? How to do bulk Check In? How to make a project archive? How to copy a project? How to assign lifecycle status to a component? How to add property to a document? How to add standard libraries to a vault? About global settings</p>
<p>Session 20</p>	<p><u>PHOTOWORKS</u> PhotoWorks Introduction to PhotoWorks, How to render model in PhotoWorks Studio? How to render specified area? How to save image to a specified file? How to edit scenery? How to apply background and base? Scene editor, How to apply material and decal? How to set camera and lights? Render options</p>
<p>Session 21</p>	<p><u>WELDMENT DESIGN</u> 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</p>

Session 22	<p><u>MOLD DESIGN</u> MOLD DESIGN</p> <p>Introducion of Mold, type of mold design, how to used draft analysis, undercut analysis, parting line analysis, parting line, shut off, parting surface, tool split, core.</p>
Session 23	<p><u>DATA MIGRATION</u></p>
Session 24	<p><u>DIRECT EDITING</u></p>
Session 25	<p><u>SIMULATION EXPRESS</u></p> <p>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. SimulationXpress 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.</p>

	<p>Type of analysis is performed</p> <ul style="list-style-type: none"> • Linear Static Analysis • Frequency Analysis • Dynamic Analysis • Linearized Buckling Analysis • Thermal Analysis • Nonlinear Analysis • Drop Test Analysis • Fatigue Analysis • Design Studies • Pressure Vessel Design • Beams and Trusses <p>All type of support, all type of loads is covered</p>
<p>Session 26</p>	<p><u>FLOW XPRESS</u></p> <p>SolidWorks FloXpress is a fluid dynamics application that calculates how fluid flows through part or assembly models. Based on the calculated velocity field, you can find problem areas in your design and improve them before you manufacture any parts.</p>
<p>Session 27</p>	<p><u>SOLIDWORKS ADVANCE</u></p> <p><u>Evaluate</u></p> <p><u>Dimexert</u></p> <p><u>Solidworks routing</u></p>

Solidworks Toolbox

Advance Modeling

Advance Assembly

Solidworks Motion

Tracing Method

Import & Export

Advance Mold Design

Industrial Components Design