Course: Product Design using Solidworks

Course Description

This course is designed for both students and professionals covering all major environments of SOLIDWORKS. SOLIDWORKS is a feature-based 3D parametric solid modeling software. After completing this course, the trainees/students will be able to create solid components, sheet metal components, assemblies, drawing views with bill of materials (BOM) as well as apply direct modeling techniques to facilitate rapid design prototyping. Also, the trainees/students will learn the editing techniques that are essential for making a successful design.

Class and Lab hours:

40 Hours

Prerequisite:

Need to be an Engineer/Technologist with knowledge of design concepts.

Course Objectives

Upon completion of the course, trainees/students will be able to:

Understand the parametric design concepts
Create 3DModels
Edit 3D Models and Sketches
Assemble the Components
Understand Surface Modeling
Create Sheet Metal Components
Generate Drawing Views

Major Instructional Areas:

Introduction to SOLIDWORKS
Working with Solid Models
Assembly Modeling
Working with Surfaces
Working with Sheet Metal Components
Generating Drawing Views

Detailed Course Outline

Unit Heading	Unit Outcomes	Unit Topics
1.Introduction to SOLIDWORKS 2015	Upon completion of this unit, the students are expected to: • Understand how to start SOLIDWORKS • Understand various modes and interface of SOLIDWORKS • Understand about Command Manager of SOLIDWORKS • Understand various important terms in SOLIDWORKS • Understand about hot keys and color scheme of SOLIDWORKS	• Introduction to SOLIDWORKS 2015 • Getting Started with SOLIDWORKS • Menu Bar and SOLIDWORKS Menus • Command Manager • Toolbar • Dimensioning Standard and Units • Important Terms and Their Definitions • Hot Keys • Color Scheme
2. Drawing Sketches for Solid Models	• Understand sketching environment of SOLIDWORKS • How to open a new document in part, assembly, and drawing mode • Understand various terms used in the sketching environment • Use various sketching tools • Delete sketched entities	• The Sketching Environment • Starting a New Session of SOLIDWORKS 2015 • Task Panes • Starting a New Document in SOLIDWORKS 2015 • Understanding the Sketching Environment • Setting the Document Options • Learning Sketcher Terms • Drawing Sketch Entities • Drawing Display Tools • Deleting Sketched Entities
3. Editing and Modifying Sketches	• Edit Sketches using various editing tools • Create rectangular and circular patterns of sketched entities • Write text in the sketching environment • Modify sketched entities using options in their Property Manager • Modify sketches dynamically by dragging	• Editing Sketched Entities • Creating Patterns • Editing Patterns • Writing Text in the Sketching Environment • Modifying Sketched Entities
4. Adding Relations and Dimensions to Sketches	• Understand need of applying relations and dimensions • Dimensioning sketches • Modify the dimensions of sketches • Understand the concept of under defined, fully	• Applying Geometric Relations to Sketches • Design Intent • Dimension a Sketch • Concept of a Fully Defined Sketch • Deleting Over defined Dimensions • Opening an Existing File

	• *defined and over defined sketches* • *View and examine the relations applies to sketches* • *Open an existing file*	
5. Advanced Dimensioning Techniques and Base Feature Options	• *Fully define a sketch using advanced dimensioning technique* • *Dimension the true length of an arc* • *Measure distances and view section properties* • *Create base feature by extrusion or revolve of sketch* • *Dynamically rotate the view of the model* • *Modify the orientation of the view* • *Change the display modes of solid models* • *Apply materials to models* • *Change the appearances of models*	• *Advanced Dimensioning Techniques* • *Measuring Distances and Viewing Section Properties* • *Creating Base Features by Extruding Sketches* • *Creating Base Features by Revolving Sketches* • *Determining the Mass Properties of Parts* • *Dynamically Rotating the View of a Model* • *Modifying the View Orientation* • *Restoring the Previous View* • *Displaying the Drawing Area in Viewports* • *Display Modes of a Model* • *Additional Display Modes* • *Assigning Materials and Textures to Models*
6. Creating Reference Geometries	• *Create a reference plane* • *Create a reference axis* • *Create reference points* • *Create a reference coordinate system* • *Create a model using the advanced Boss/Base options* • *Create a model using the contour selection technique* • *Create a cut feature* • *Create multiple disjoint bodies*	• *Importance of Sketching Planes* • *Reference Geometry* • *Advanced Boss/Base Options* • *Modeling Using the Contour Selection Method* • *Creating Cut Features* • *Concept of Feature Scope*
7. Advanced Modeling Tools-I	• *Create holes using the Simple Hole option* • *Create standard holes using the Hole Wizard tool* • *Create standard external threads* • *Apply simple and advanced fillets* • *Understand various selection methods for filleting*	• *Creating Simple Holes* • *Creating Standard Holes Using the Hole Wizard* • *Adding External Cosmetic Threads* • *Creating Fillets* • *Selection Options* • *Creating Fillets Using the Fillet Xpert* • *Creating Chamfers* • *Creating Shell Features* • *Creating Wrap Features*

	• *Chamfer the edges and vertices of a model* • *Create the shell feature* • *Create the wrap feature*	
8. Advanced Modeling Tools-II	• *Mirror features, faces, and bodies* • *Create linear and circular patterns* • *Create sketch driven patterns* • *Create curve driven patterns* • *Create table driven patterns* • *Create fill and variable patterns* • *Create rib features* • *Display the section view of a model* • *Change the display state of a part*	• *Creating Mirror Features* • *Creating Linear Pattern Features* • *Creating Circular Pattern Features* • *Creating Sketch Driven Patterns* • *Creating Curve Driven Patterns* • *Creating Table Driven Patterns.* • *Creating Fill Patterns* • *Creating Variable Patterns* • *Creating Rib Features* • *Displaying the Section View of a Model* • *Changing the Display States*
9. Editing Features	• *Edit feature and sketch* • *Change existing sketch plane* • *Understand use of Instant 3D tool* • *Cut, Copy and Paste features and sketches Copy using drag and drop* • *Delete features and bodies* • *Suppress and un suppress features* • *Un suppress feature with dependents* • *Hide bodies* • *Move and copy bodies* • *Reorder feature* • *Roll back and rename feature* • *Create folder in Feature Manager design tree* • *Use of what's wrong dialog box*	• *Editing Using the Edit Feature Tool* • *Editing Sketches of the Sketch-based Features* • *Editing the Sketch Plane Using the Edit Sketch Plane Tool* • *Editing Using the Instant3D Tool* • *Editing Features and Sketches by Using the Cut, Copy, and Paste Options* • *Cutting, Copying, and Pasting Features and Sketches from One Document to the Other* • *Copying Features Using Drag and Drop* • *Deleting Features* • *Deleting Bodies* • *Suppressing Features* • *Un suppressing the Suppressed Features* • *Un suppressing Features with Dependents* • *Hiding Bodies* • *Moving and Copying Bodies* • *Reordering the Features* • *Rolling Back the Feature* • *Renaming Features* • *Creating Folders in the Feature Manager Design Tree* • *What's Wrong Functionality*
10. Advanced Modeling Tools-III	• *Create sweep features* • *Create cut-sweep features* • *Create loft features* • *Add a section to a loft feature* • *Create lofted cuts* • *Create 3D Sketches* • *Create grid systems*	• *Creating Sweep Features* • *Creating Cut-Sweep Features* • *Creating Loft Features* • *Adding a Section to a Loft Feature* • *Creating Lofted Cuts* • *Creating 3D Sketches* • *Creating Grid Systems* • *Editing 3D Sketches* • *Creating Curves*

	• *Edit 3D sketches* • *Create curves* • *Extrude a 3D sketch* • *Create draft features*	• *Extruding a 3D Sketch* • *Creating Draft Features*
11. Advanced Modeling Tools-IV	• *Create dome, deform, and flex features* • *Create indents* • *Create mounting boss, snap hooks, snap hook grooves and vents* • *Create Lip/Groove feature* • *Use of Freeform tool* • *Add datum, dimension and tolerances using Dim Xpert* • *Automatic dimensioning of a part*	• *Advanced Modeling Tools* • *Creating Fastening Features* • *Creating Freeform Features* • *Dimensioning a Part Using DimXpert*
12. Assembly Modeling-I	• *Modify drawing standards* • *Create bottom-up assemblies* • *Create top-down assemblies* • *Move and rotate individual components* • *Move and rotate individual components using the triad* • *Assembly visualization*	• *Assembly Modeling* • *Creating Bottom-up Assemblies* • *Creating Top-down Assemblies* • *Moving Individual Components* • *Rotating Individual Components* • *Moving and Rotating Individual Components Using the Triad* • *Assembly Visualization*

Evaluation:

There will be one exam that every trainee/student must pass with at least 75% or more to get a certificate of completion from BIMNCAD.

Suggested Learning Approach

In this course, you will study individually or within a group of your peers. As you work on the course deliverables, you are encouraged to share ideas with your peers and instructor, work collaboratively on projects and team assignments, raise critical questions, and provide constructive feedback.