



J.S. University, Shikohabad
Faculty of Mechanical engineering

Value Added Course

AY: 2021-22

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Value Added Courses

Faculty of Mechanical Engineering



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SOLID WORKS ESSENTIALS

Learning Objective:

This Course will provide knowledge of Solid Works

Duration: 30 Hours. (Theory and Practical)

Course Outcomes: -

After completion of the course the student shall be able to:-

1. To understand the relationship of art and science to design.
2. To develop proficiency in design skills and methodologies.
3. To gain first-hand experience of the design process in the context of a 'real', open-ended multi disciplinary design project
4. To work effectively and professionally in a team while executing a design project
5. To apply engineering analysis tools in the design process

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Syllabus

Module I: Introduction

CAD, CAE, PDM Features of Solid Works, Various products available in Solid Works for Product Design, Simulation, Communication Solid Works Graphical.

Module 2: Sketch entities, Sketch Tools and Blocks

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 and other tools.

Blocks – Make block, Edit block, Insert block, Add/Remove Entities, Rebuild, Save, Explode Relations - Adding Sketch.

Module 3: Relations

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.

Module 4: Part Modelling Tools, Swept ,Loft features

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

Module 5: Creating Curves

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.

References:-

1. Solid Works 2019 Training Guide Paperback – 1 January 2019 by Linkan Sagar (Author)
Publisher : BPB (1 January 2019)
2. SolidWorks 2022 Black Book (Colored) Hardcover – Import, 24 December 2021 by Gaurav Verma (Author), Matt Weber (Author) ,Publisher : Cadcamcae Works(24December 2021)
3. Learn SOLIDWORKS 2020: A hands-on guide to becoming an accomplished SOLIDWORKS Associate and Professional Paperback – Import, 2 December 2019 by Tayseer Almattar (Author) Publisher : Packt Publishing Limited (2 December 2019)
4. SolidWorks 2021 - Step-By-Step Guide: Part, Assembly, Drawings, Sheet Metal, & Surfacing Paperback – Import, 1 Decmber 2020 by Amit Bhatt (Author), Mark Wiley (Author), Cadfolks (Author) ,Publisher : Independently Published(1December 2020)

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