iKUSASA

CNC TRAINING CENTRE

WE ARE THE *FUTURE*

SOLIDWORKS
ESSENTIALS

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SOLIDWORKS
Essentials

Requirements: Understanding of technical drawings, Mechanical design experience, Computer literacy, and any previous CAD/CAM Experience is to your advantage.

Duration: 5 Days

Time: 08:30 - 16:00

Inclusions: Catering & Refreshments (full time in-house training), relevant stationary, competency certificate.

OVERVIEW

Unlock the power of 3D design with confidence. Our SOLIDWORKS Essentials course provides a comprehensive introduction to the core concepts and features of SOLIDWORKS, equipping learners with the practical skills needed to create professional-quality 3D models and 2D drawings.
Designed for beginners and users looking to formalise their experience, this instructor-led course focuses on the fundamental tools of part modelling, assembly creation, and detailed drawings. By the end of the course, delegates will be able to confidently navigate the SOLIDWORKS interface, produce intelligent parametric models, and streamline the design process using industry-standard practices.
Whether you're starting your design career or upskilling for new opportunities, this course sets a solid foundation for mastering SOLIDWORKS and progressing into more advanced training.

The course consists of 5 days in-class where we cover the complete basics in terms of:

MODULE 1:

- What is SOLIDWORKS?
- What is Design Intent?
- File References (Object Linking & Embedding)
- SOLIDWORKS User Interface
- Using the Command Manager

MODULE 2:

- 2D Sketching
- Default Planes
- Sketch Geometry
- Mechanics of Sketching
- Sketch Guidelines
- Sketch Feedback
- Desired Design Intent
- Automatic Sketch Relations
- Added Sketch Relations
- Selecting Multiple Objects
- Dimensioning

MODULE 3:

- Choosing the Best Profile
- Selecting Sketch Planes
- Boss Extrusions
- Cuts
- Fillets
- Auto-transition between lines and fillets
- Using the Hole Wizard
- Fillet Propagations
- Rollback Bar
- Driving Dimensions
- Driven Dimensions
- Associativity between the Model & Drawing
- Rebuilding the Model

MODULE 4:

- Design Intent
- Boss Feature with Draft
- Symmetry in the Sketch
- Sketching inside the model
- View options

- Using model edges in a sketch
- Creating trimmed sketch geometry
- Copy & Paste features

MODULE 5:

- Why use patterns?
- Linear Patterns
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Using pattern seed only
- Up to reference
- Sketch driven patterns

MODULE 6:

- Using the revolve feature
- Thin Features
- Splines
- Editing Materials
- Mass Properties
- File Properties

MODULE 7:

- Analysing and adding draft
- Shelling
- Rib Features
- Full Round Fillets

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MODULE 8 - 9:

- Part Editing
- Design Changes
- Information from a model
- Rebuilding tools
- Replace sketch entities
- Sketch contours
- Supressing features

MODULE 10:

- Using configurations
- Other methods to create configurations
- Renaming features and dimensions
- Design rules using global variables and equations
- Equations
- Using operators and functions
- Modelling strategies for configurations
- Editing parts that have configurations
- Design library

MODULE 11:

- Using global variables & equations
- Renaming features & dimensions
- Design intent when using global variables
- Equations
- Using operators & functions

MODULE 12:

- Section View
- Model Views
- Broken View
- Detail Views
- Drawing sheets & sheet formats

- Projected Views
- Model Items
- Annotations

MODULE 13:

- Creating a new assembly
- Positioning the base component
- FeatureManager design tree & symbols
- Adding components
- Mating components
- Using part configurations in assemblies
- Sub-assemblies
- Smart mates
- Interting sub-assemblies
- Pack & Go

MODULE 14:

- Analysing the assembly
- Clearance detection
- Collision detection
- Exploded assemblies
- Explode line sketch
- Rollback and Reorder
- Bill of Materials
- Assembly Drawings