BLACKBIRD FOR TRAINING

SOLIDWORKS Essentials: 3D CAD Modeling and Drawings Course

5 – 9 July 2027
Kuala Lumpur (Malaysia)

UK Traininig
PARTNER



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Ref: 4_238845 **Date:** 5 – 9 July 2027 **Location:** Kuala Lumpur (Malaysia) **Fees:** 4900 GBP

Introduction:

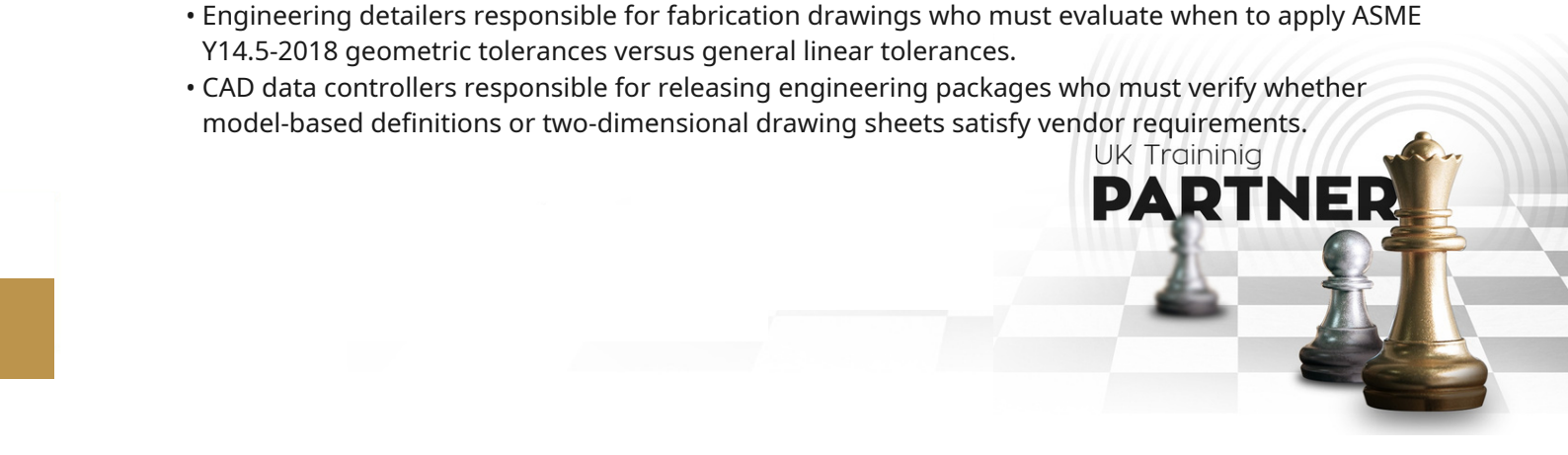
SOLIDWORKS Essentials training in 3D CAD modeling and engineering drawings provides the foundational methods for generating parametric solid models, assemblies, and production drawings within a unified environment. This one-week course prepares engineering draftspersons and product development technicians to construct a verified mechanical assembly and production-ready drawing package. Taught at practitioner depth through an applied lab mode, the syllabus incorporates the ASME Y14.5-2018 standard for dimensioning and tolerancing to govern geometric definitions. Delegates complete this SOLIDWORKS Essentials training with Blackbird Training Centre to standardise engineering workflows.

Course Objectives:

- Define parametric feature relationships, sketch constraints, and parent-child dependencies to maintain stable part history trees.
- Distinguish between extruded boss features and revolved boss features when generating foundational mechanical solid bodies.
- Compare bottom-up assembly techniques against top-down contextual modeling to establish robust multi-component mechanisms without circular references.
- Interpret ASME Y14.5-2018 datum reference frames and geometric feature control frames to ensure precise fabrication tolerances.
- Apply ASME Y14.41 digital product definition data practices to annotate three-dimensional CAD models for paperless manufacturing workflows.
- Select appropriate drawing sheet scales, section cuts, and projection views under ASME Y14.100 to produce unambiguous workshop fabrication sheets.

Target Audience:

- Mechanical drafting personnel responsible for creating manufacturing documentation who must decide which projection views and section planes communicate intent clearly.
- Product development technicians responsible for solid component authoring who must choose between sketch relations and numerical dimensions to stabilise parametric geometry.
- Tooling and fixture modellers responsible for physical alignment who must determine whether standard or mechanical mates best constrain moving assemblies.
- Engineering detailers responsible for fabrication drawings who must evaluate when to apply ASME Y14.5-2018 geometric tolerances versus general linear tolerances.
- CAD data controllers responsible for releasing engineering packages who must verify whether model-based definitions or two-dimensional drawing sheets satisfy vendor requirements.



Course Outline:

Day 1: Parametric Sketching and Solid Feature Foundations

- Extruded Boss/Base vs Revolved Boss/Base: Selecting Primary Volumetric Creation Methods for Prismatic or Cylindrical Geometries
- Geometric Sketch Relations: Constraining 2D Profiles to Maintain Intent During Subsequent Dimensional Revisions
- Parametric Dimensioning Schemes: Establishing Driving Numerical Parameters Across Early Stage Part Sketches
- Thin-Walled Shell Operations: Applying Uniform Wall Thicknesses Across Open-Cavity Plastic and Cast Components
- Instantaneous Design Intent: Evaluating Parent-Child Sketch Dependencies to Prevent Downstream Feature Failures

Day 2: Technical Standards and Intermediate Part Architecture

- ASME Y14.5-2018 vs ISO 1101: Comparing Standardized Geometric Tolerancing Frameworks for Global Manufacturing Operations
- Swept Features: Guiding Planar Profiles Along Complex Three-Dimensional Spline Trajectories for Ducting
- Multi-Body Solid Modelling: Combining Independent Solid Bodies Within Single Files for Welded Frames
- Material Property Assignment: Applying Physical Density and Thermal Parameters to Evaluate Mass Properties
- Draft and Rib Features: Incorporating Injection Moulding Angles to Facilitate Tool Ejection Requirements

Day 3: Mechanism Assembly and Component Interaction

- Bottom-Up Assembly vs Top-Down Assembly: Balancing Component Isolation Against In-Context Parametric Associations
- Standard Geometric Mates: Constraining Spatial Degrees of Freedom Between Adjacent Machine Components
- Mechanical Mate Application: Simulating Functional Linkages Using Slot, Cam-Follower, and Gear Relationships
- Subassembly Hierarchies: Grouping Functional Mechanical Modules to Optimise System Rebuild Performance and Organisation
- Bills of Materials Generation: Compiling Automated Component Quantities and Standard Part Properties for Procurement



Day 4: Engineering Drawing Generation and Digital Product Definition

- ASME Y14.41 MBD vs ASME Y14.100 Drafting: Evaluating 3D Annotations Against Traditional Orthographic Documentation
- Orthographic View Projection: Laying Out Standard Third-Angle Multi-View Drawings for Fabrication Workshops
- Section and Detail Views: Exposing Internal Assembly Features and Fine Geometric Tolerancing Profiles
- DimXpert Annotation Engine: Applying Semi-Automated Geometric Feature Control Frames to Machined Features
- Revision Table Management: Tracking Engineering Changes and Approval Workflows Across Technical Drawing Sheets

Day 5: Practical Assembly Modeling and Drawing Production Lab

- Lab Assembly Alignment: Applying Earlier Mate Techniques to Construct a Functional Mechanical Mechanism
- Interference Detection vs Clearance Verification: Resolving Clash Violations and Kinematic Overlaps in the Lab Workspace
- Drawing Sheet Configuration: Applying ASME Y14.100 Standards to Format Title Blocks and Scaling Rules
- Automated BOM Insertion: Embedding Linked Assembly Component Tables Onto Final Production Drawing Sheets
- Final Deliverable Completion: Finalising the Constrained Mechanical Assembly and Production-Ready Drawing Package

Skills You Will Gain:

- Parametric Sketch Constraining
- 3D Solid Feature Modeling
- Mechanical Mate Configuration
- Subassembly Hierarchy Structuring
- Geometric Dimensioning and Tolerancing
- Orthographic View Layout
- Automated Bill of Materials Extraction
- Model-Based Definition Implementation



Why Attend This Course:

- From struggling with fragile sketches and frequent rebuild errors to establishing stable geometric relationships and intentional parametric dependencies.
- From wrestling with mismatched assembly components to implementing precise standard and mechanical mates that reflect physical mechanism movement.
- From creating ambiguous manual 2D drawings to publishing standardized manufacturing sheets adhering to ASME Y14.5-2018 and ASME Y14.100.
- From uncoordinated design documentation to delivering a verified mechanical assembly and production-ready drawing package with geometric dimensioning and tolerancing.

Conclusion:

Parametric computer-aided authoring forms the technical backbone of modern mechanical product engineering. This course clarifies the practical distinctions between bottom-up and top-down assembly strategies, model-based definition and traditional two-dimensional drafting, and geometric feature control frames versus basic linear tolerances. Designed for mechanical draftsmen, detailing technicians, and design modellers seeking disciplined CAD competencies, this practitioner-level course provides the applied methods required to build robust solid parts, verify kinematic mechanisms, and generate precise fabrication drawings that eliminate ambiguity on the shop floor.

Frequently Asked Questions FAQ:

What is SOLIDWORKS Essentials training?

SOLIDWORKS Essentials training is a foundational engineering curriculum that teaches professionals how to build parametric 3D parts, assemble multi-component machinery, and generate standard technical manufacturing drawings. It establishes core modeling habits, parametric sketch constraints, mechanism mates, and automated documentation extraction for product development environments.

What is the difference between bottom-up and top-down assembly in SOLIDWORKS?

Bottom-up assembly constructs individual parts in separate files before inserting and mating them into an assembly, whereas top-down assembly builds parts directly inside the assembly context. Bottom-up modeling ensures reusable independent components, while top-down modeling allows geometry to adapt dynamically to adjacent component dimensions.

Which technical standards govern SOLIDWORKS engineering drawings?

ASME Y14.5-2018 governs geometric dimensioning and tolerancing, while ASME Y14.100 regulates technical drawing practices and projection rules. Additionally, ASME Y14.41 establishes digital product definition standards for model-based annotations, alongside international drafting conventions outlined in ISO 128 and ISO 1101.



Is this SOLIDWORKS Essentials course suitable for operational drafting technicians?

Yes, this course is tailored for operational CAD detailers and drafting technicians who require reliable modeling and drafting skills. Delegates master sketch constraining, solid feature creation, interference detection, and standardized drawing production without requiring previous advanced parametric design experience.

