CATIA V5 Fundamentals

Brand	CATIA
Discipline	CATIA Mechanical Design V5
Release	V5R21
Duration	40 hours – 5 Days
Level	Fundamental
Objectives	Upon completion of this course you will be able to: - Understand and use the CATIA V5 interface - Plan the construction of a part in order to convey its visual and functional aspects - Create simple parts in CATIA V5 - Construct an assembly managing the parts - Produce simple drawings and assembly layouts
Course Structure	• Introduction to the sketcher, • Solid modelling using feature based methodology. • CAT Part manipulation and editing. • Drafting workbench. • Introduction to Assembly Design Workbench. • Course meets all the needs for airbus Certification
Participants profile	Mechanical Designers with no CATIA V5 experience
Prerequisites	Students attending this course should have knowledge of Mechanical Design, and Windows Operating System
Description	This course will teach you how to build simple parts and assemblies in CATIA V5. You will learn how to make simple drawings of those parts and assemblies.

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