

CATIA V5 FUNDAMENTALS

OBJECTIVE: The students will be introduced to the CATIA V5 fundamental concepts and interface. Students will learn the concept of sketch-based features, the management of parts through an assembly and how to generate standard views from this assembly. The part creation in this course is mainly focused on the creation of parts based on 2D profiles (sketches), and on the assembly of existing components.

DURATION 5 Days	STUDENT PROFILE CATIA V5 Users	PRE-REQUISITES None
TOPIC	DETAILS	DURATION
Sketcher	<ul style="list-style-type: none"> ▪ Getting Started with CATIA V5 ▪ Sketcher Geometry Management ▪ Sketcher Constraints Management ▪ Sketch Analysis Tools 	1 DAY
Part Modeling	<ul style="list-style-type: none"> ▪ Sketch-Based Features ▪ Dressing-Up Features ▪ Transforming Features 	2 DAYS
Assembly	<ul style="list-style-type: none"> ▪ Assembling Components ▪ Positioning Components using Constraints ▪ Editing Parts in an Assembly ▪ Designing in the Context of an Assembly ▪ Working with Components 	1 DAY
Drafting & Drawing Generation Review	<ul style="list-style-type: none"> ▪ Starting a Drawing and View Generation ▪ Additional View Generation ▪ Dimensioning and Annotating ▪ Editing View's Layout and Properties ▪ Finalizing the Drawing and Printing 	1 DAY

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Information contained within is subject to change.

All classes are dependent on minimum enrollment

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