



# CATIA V5 Fundamentals

## 5 Days Course



**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.

<b>Price per Student</b>		<b>\$ 2,000.00 CDN</b>
<b>Duration:</b> 5 Days		
<b>TOPIC</b>	<b>DETAILS</b>	
<b>Sketcher</b>	Getting Started with CATIA V5 Sketcher Geometry Management Sketcher Constraints Management Sketch Analysis Tools	
<b>Part Modeling</b>	Sketch Based Features Dressing-Up Features Transforming Features	
<b>Assembly</b>	Assembling Components Positioning Components using Constraints Editing Parts in an Assembly Designing in Context of an Assembly Working with Components	
<b>Drafting &amp; Drawing Generation</b>	Starting a Drawing and View Generation Additional View Generation Dimensioning and Annotating Editing View's Layout and Properties Finalizing the Drawing and Printing	



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