

# Training Agenda for Design and Detailing using CATIA V5

Introduction of machine drawing with Geometrical Dimensioning & Tolerance	
Basic features of CATIA	Introduction to CATIA
	Basic Sketches
	Sketch Based Features
	Dress Up Features
	Transformation Features
	Editing, Repairs
Basic features of part and assembly modelling	Editing, design changes
	Bottom-Up Assembly modelling
	Publication
	Publication Using Assemblies
Detailing	Drawing Sheets and Views
	Dimensions
	Annotations
	Sheet Formats and Templates
	Assembly Drawing Views
	Bill of Materials and Tables
	Settings in drawings
Advanced part modelling	Multi section Solids
	Rib
	Solid Combine
	Boolean Operation
	Other Advanced Tools (Surface based features & Annotations in part modelling)
Advanced assembly modelling	Top-Down Assembly modelling
	Assembly Editing
	Assembly Display States
	Large Assemblies
Sheet metal	Modelling Sheet Metal Parts
	Sheet Metal Forming Tools

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Sheet metal	Additional Sheet Metal Features
	Converting Parts to Sheet Metal
Surfaces	Wire Frame
	Basic Surface Features
	Advanced Surface Features
	Surface Operation Features
	Transformation Features
	Converting Surface to Solid
File management-CATIA	CATIA- File Structure
	Save Management & Send to option
	Downward Compatibility
Queries resolution, Tool test for the topics covered	

First half will be theory session and second half will be hands on,  
for the topics covered on that particular day.