


CATIA

	Name :		Course : CATIA		Mob No.:	
	IMS NO. :	Admission Date :	Start Date :		End Date :	
DATE	TOPICS NAME		DAYS REQUIRED	STUDENT SIGN		FACULTY SIGN
	Introduction to Catia					
	Introduction to UI, Workbenches, Navigation					
	Sketcher Workbench					
	Profile(profile, Rectangle Circle, Polygon)					
	Constraints(Geometrical, Dimensional)					
	Sketch Operation(trim, Extend, Offset, Fillet, Mirror)					
	Part Design Workbench					
	Introduction to Part Design workbench					
	Pad, Pocket, Multi-pad, Multi-Pocket					
	Operations(shaft, Groove, Fillets, Chamfer, Drafts, Shells, Ribs & Slots)					
	Patterns(Rectangular, Circular, User-Defined)					
	Boolean Operations(Union, Subtract, Intersect)					
	Assembly Design					
	Introduction to Assembly, Manipulation in Assembly					
	Assembly Constraints					
	Top-Down Assembly					
	Bottom-Up Assembly					
	Assembly Features					
	Drafting Workbench					
	Generating 2d Drawings from 3D model					
	Creating Views(front, side, isometric)					
	Dimensions , Annotation, Bill of Material					
	Exam					

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