

CATIA LEVEL 1

- Introduction
- Sketcher
- Part Design (Pad – pocket)
- Using Pad and pocket to create Complex geometry
- Part Design (Shaft- Groove – Hole)
- Tip & tricks to create Complex part
- Parametric design – Introduction
- Advanced Part Feature (Rib & Slot)
- Parametric design (f(x))
- Assembly – Introduction
- Assembly Design
- Assembly management
- Parametric design in assembly
- Tip & trick in part design
- DOF in assembly
- DMU Kinematics
- Assembly Analysis
- Advanced Parametric design
- Creating parametric Gear
- Drafting
- CATIA Programming (VBA) – Introduction

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