

CATIA Syllabus

Module-01: Basic Fundamentals

1. Introduction to CATIA
2. CATIA User Interface
3. Setting Units, Grid & Operations
4. CATIA's Six Work Environments
5. Setting Sketch Workspace and Sketching Methods

Module-02: Sketching Related

6. Profile Toolbar Tools & Sketch Toolbar Tools
7. Operation Toolbar, Standard Toolbar
8. Constraint Toolbar & Dimensional Toolbar
9. View Toolbar & Graph Option Toolbar
10. Measure Toolbar, Select Toolbar, Fit-All-In Tool

Module-03: 3D Solid Modeling

11. Pad, Drafted Filleted Pad, Multi Pad, Pocket, Shaft, Groove, Hole
12. Rib, Slot, Combine, Stiffener, Multi Section Solid, Cut
13. Fillet, Chamfer (5 Types), 3D Draft Operations
14. Shell, Thicken, Remove Face, Replace Face, Thread, Boolean Operations, Extend Reference Elements
15. Advanced Dress-Up Features, All Toolbar Tools, Transformation Toolbar Tools

Module-04: Sheet Metal Modeling

16. Sheet Metal Wall Creating Toolbars (All Tools)
17. Bend Toolbar (All Tools)
18. Rolled Wall Toolbar, All Tools + Views Tools
19. Cutting & Stamping Toolbars (All Tools)

Module-05: Surface Modeling

20. Surface-Based Features Toolbar (All Tools)
21. Operation Toolbar (All Tools)
22. Developed Shape Toolbar (All Tools)
23. Wireframe Surface Toolbar (All Tools)

Module-06: Rendering & Projects

- 24. Apply Material to Any Product/Part
- 25. Rendering Any Part or Product
- 26. Sheetmetal to Project, Solid Model to Project, Surface Model to Project

Module-07: Assembly Related

- 27. Constraining Toolbar Tools
- 28. Move Toolbar Tools
- 29. Assembly Features Toolbar

Module-08: Drawing Sheet Related

- 30. Drawing Sheet Preparation with Detail & BOM (Bill of Materials)