

Course Objectives :

Modular in design, the CATIA product line adapts to all customer businesses including style and form design, mechanical design, systems and equipment engineering, managing digital mock-ups, numerical control, simulation and analysis, using an open and scalable V5 architecture.

course	Duration hrs
CATIA	48

Outline :

1. Introducing CATIA.
2. CATIA Basics and the User Interface.
3. Introduction to Sketching.
4. Basic Part Modeling.
5. Using Visualizing Techniques.
6. Selecting Features.
7. Patterning.
8. Revolved Features.
9. Shelling and Ribs.
10. Using Equations.& formula.
11. Importing parts from any other CAD program to CATIA

12. Building efficient assembly.
13. Getting more from mates.
14. Assembly configuration and display states.
15. Component patterns.
16. Working with library features.
17. Automating drawing.
18. Working with drawing views.
19. Dimensioning, tables and annotations.
20. Making kinematics simulation for the Assembly.
21. Study velocity and Accelerations.



ENGO SOFT