

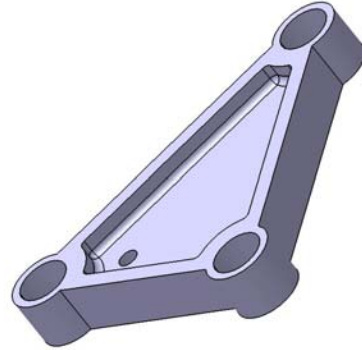
Mechanical Design Application Using CATIA® V5

CATIA is recognized as a leading PLM and high-end feature based parametric solid modeling design software. CATIA V5 utilizes an intuitive windows user interface that puts powerful design tools in the hands of users operating in the familiar Windows® environment.

This training program has been tailored for manufacturing, design, and engineering professionals who need to develop skill and speed in using CATIA V5 to create and evaluate complex 3-D solid models and prepare fully dimensioned engineering drawings. This is a 5-hour class that meets one day per week for 16 consecutive weeks. Our computer lab is state-of-the-art with computers that have been properly equipped for feature based solid modeling CAD work. This will be a challenging class that will expose the participants to real-world modeling and design problems utilizing some of the more powerful tools offered in CATIA V5.

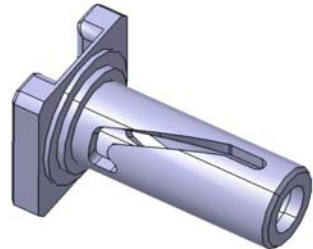
Fundamentals of CATIA and Feature Based Modeling

- Principles of feature based solid modeling
- Parametric and associative application
- Design intent
- Working in the Windows environment
- Screen layout and workbenches
- Working with and viewing models
- Customizing and managing CATIA V5
- Sketching theory and technique
- Application of geometric and dimensional constraints
- Creating Pad, Shaft, Pocket, and Groove features
- Applying fillet, chamfer, draft, and shell dress-up features
- Working with multi-profile sketches and features
- Creating hole features



Advanced solid modeling techniques

- Creating reference planes, points, and lines
- Advanced sketching tools and sketch analysis
- Advanced ribs, slots, and multi-section features
- Reusing designs with transformation tools and duplication techniques
- Part design tables and catalogs
- Extracting and interrogation of part/design information
- Creating multiple bodies and Boolean operations
- Application project



Creating Assemblies

- The assembly/product workbench
- Positioning and constraining components
- Assembly technique and extracting information
- Clearance analysis and Bill of Materials
- Application project

Creating Engineering Drawings and Detailing

- The drafting workbench
- Setting up the drawing frame and title block
- Creating standard and detail auxiliary views
- Dimensioning and annotations
- Inserting a Bill of Material (BOM)
- Product detailing
- Application project

