CATIA-V6
Curriculum

- ➢ Overview of Parametric Design Process
- ➢ Customization of CATIA V6 Environment
- ➢ Creating and Constraining Sketch Geometry
- ➢ Sketched Feature Techniques and Formulas
- ➢ Adding Material with Pad and Shaft Features
- ➢ Removing Material with Pocket and Groove Features
- ➢ Creating Reference Elements for construction and measurement
- ➢ Fillet, Chamfer, Hole, Draft, and Shell Dress-Up Features
- ➢ Pattern, Copy, and Mirror Duplication Features
- ➢ Thin Features, Stiffeners
- ➢ Obtaining Part Information
- ➢ Generative Drafting View Creation
- ➢ Generative Drafting Dimensioning and Annotation
- ➢ Rib and Slot Features
- ➢ Multi-sections Solid Features
- ➢ Feature Management Using the Hide / Show, Activate / Deactivate Functions
- ➢ Parent/Child Relationships and Feature Failure Resolution
- ➢ Assembly Design Workbench
- ➢ Constraint creation, assembly management, and PDM considerations
- ➢ Obtaining Assembly Information (Measure, Clash, and Bill of Materials)
- ➢ Standard Parts from Catalogues and Save Management
- ➢ Working with Multi-Body Models
- ➢ Effective Modeling Tips and Technique