Course Length: 5 days

Using the CATIA V5-6R2018: Introduction to Modeling training class, you learn the process of designing models with CATIA V5 from conceptual sketching, through to solid modeling, assembly design, and drawing production. Upon completion of this training course, you will have acquired the skills to confidently work with CATIA V5, and gained an understanding of the parametric design philosophy of CATIA V5. It is expected that all new users of CATIA V5 need to complete this training course.

This guide was developed using CATIA V5-6R2018, Service Pack 1.

Course Topics:
- Overview of Parametric Design Process
- Customization of CATIA V5 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 Catalogs and Save Management
- Working with Multi-Body Models
- Effective Modeling Tips and Techniques

Prerequisites: Experience in mechanical design and drawing production is recommended.