

CATIA V5 Training

Introduction to Modeling

Course Length: 5 days

Price: \$2,095 (includes access to ProductivityNOW Knowledge Source Portal (formerly SOLO))

Learn the process of designing models with CATIA V5 from conceptual sketching, through to solid modeling, assembly design, and drawing production. Gain an understanding of the parametric design philosophy of CATIA V5 in this extensive hands-on course. It is expected that all new users of CATIA V5 will require this course.

Upon completion of this course, users will have access to ProductivityNOW Knowledge Source Portal (formerly SOLO) for 30 days. With the combination of this class and ProductivityNOW Knowledge Source Portal (formerly SOLO), users will have acquired the skills to confidently work with CATIA V5.

Course Topics:

- Overview of Parametric Design Process
- Customization of CATIA V5 Environment
- Creating and Constraining Sketch Geometry
- Sketched Feature Techniques
- Adding Material with Pad and Shaft Features
- Removing Material with Pocket and Groove Features
- Fillet, Chamfer, Hole, Draft, and Shell Dress-Up Features
- Multi-Profile Sketched Features
- Pattern, Copy, and Mirror Duplication Features
- Formulas and Parameters
- Obtaining Part Information
- Generative Drafting View Creation
- Generative Drafting Dimensioning and Annotation
- Rib and Slot Features
- Adding and Removing Material with Loft Features
- Feature Management Using the Hide / Show, Activate / Deactivate Functions
- Parent/Child Relationships and Feature Failure Resolution
- Assembly Design Workbench
- Assembly Creation Techniques
- Obtaining Assembly Information
- Standard Parts from Catalogues and Save Management
- Effective Modeling Tips and Techniques

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

TRAINING CENTERS

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Boston, MA
Charlotte, NC
Chicago, IL
Cleveland, OH
Columbus, OH
Dallas, TX
Detroit, MI
Grand Rapids, MI
Albuquerque, NM
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Canada

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