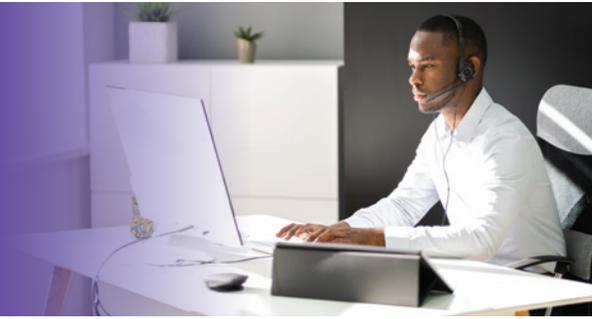


# CATIA V5-6R2018 Training

Introduction for NC & FEA Engineers



## Course Length: 2 days

This training course introduces you to the interface and modeling capabilities of CATIA V5 with a focus on the specific tools required to perform NC and FEA operations.

On completion of this training course, you will have acquired the skills to work with existing model data in CATIA V5 and to create new geometry using wireframe, solid, and surface modeling techniques. This extensive hands-on training course, with numerous practices, focuses on concepts of measurement, analysis, and simple geometry creation.

## Course Topics:

- Overview of Parametric Design Process
- Customization of CATIA V5 Environment
- Feature Management using the Hide/Show, Activate/Deactivate Functions
- Obtaining Part Information
- Assembly Design Workbench and Assembly Creation Techniques
- Creating and Constraining Sketch Geometry
- Adding Material with Pad and Shaft Features
- Introduction to Surfacing
- Creating Wireframe elements
- Creating Surfaces
- Performing Surface Operations

## Complimentary when you purchase this course:

- CATIA Sketcher Workbench Quick Reference Card
- CATIA Part & Assembly Design Workbench Quick Reference Card
- CATIA Generative Shape Design Workbench Quick Reference Card

**Prerequisites:** None



## Learn Where It's Convenient for You

### Attend Classes:

- Online with an instructor
- At one of our training centers
- At your on-site training facility
- At a location of your choice (via our mobile training labs)

Contact us:  
[training@rand.com](mailto:training@rand.com)  
877.726.3243



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Class curriculum is developed by ASCENT, developers of professional training courseware and technical documentation for Autodesk, Dassault Systèmes and PTC software.