

# CATIA V5 Training

## Numerical Control Manufacturing



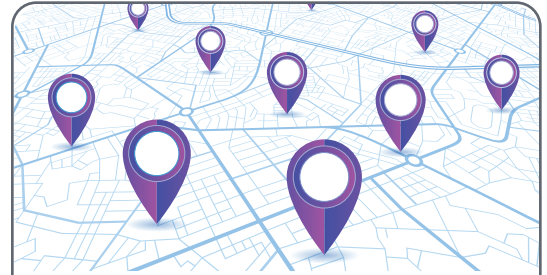
### Course Length: 5 days

In this course, attendees will learn how to define and manage NC programs dedicated to machining parts designed with Surface or Solid geometry using 3-Axis and 5-Axis machining techniques.

### Course Topics:

- NC Review
- Introduction: Workbench Presentation
- Creating a Part Operation
- Import Files
- Tool Path Verification
- Generating Outputs (APT Source - NC Code HTML document)
- Basic Tasks
- Tool Management
- Defining Axial Operations
- Generating Auxiliary Operations in the Part
- Operation
- Managing Design Changes
- Facing
- Pocketing
- Profile contouring
- Curve following
- Point to point
- Transition Paths (macros)
- Creating a Machining Feature
- Geometrical Zone
- Machining/Slope Area Creation
- Rework Area Creation
- Offset Group Creation

**Prerequisites:** CATIA Introduction to Modeling or CATIA Introduction for NC & FEA Engineers



### 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:  
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This curriculum is developed by  
Dassault Systèmes.