

## WHO SHOULD ATTEND

Engineers, Tool Designers,  
Drawing Maker, Inspection  
Personnel

## DURATION

3 days

## METHODOLOGY

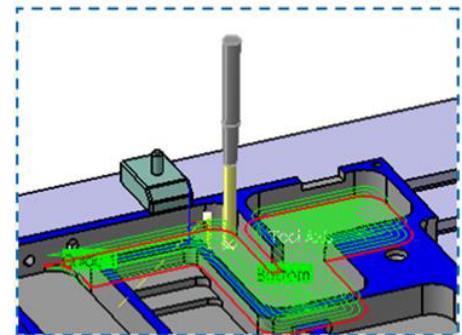
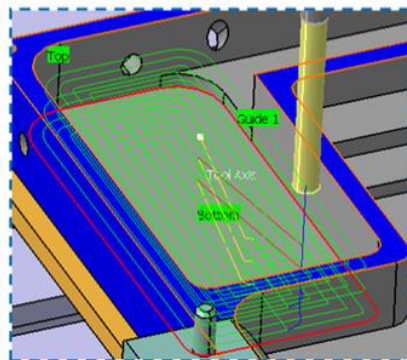
Instructor-led  
Presentation, exercises  
and discussion

## INTRODUCTION:

This course will teach you how to define and manage NC programs dedicated to machining parts using Prismatic Machining techniques in the Prismatic Machining (PMG) workbench. You will learn to create 2.5 Axis Milling operations. You will also learn to use the PMG functionalities for creating Prismatic Machining and Rework Areas.

## OBJECTIVES:

- Identify and use the Prismatic Machining workbench tools.
- Define Prismatic Machining operations (2.5-Axis Milling) in CATIA V5.
- Create Prismatic Machining Area and Rework Area.
- Define and modify NC Macros.



## CONTACT

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## CATIA V5 Training - Prismatic Machining

### KEY TOPICS

#### DAY 1

- Identify and use the Manufacturing workbenches' tools
- Create a Manufacturing Program
- Simulate a Manufacturing Program
- Manage Tools and Tool Catalogs
- Define and verify the Tool Path

#### DAY 2

- Generate NC data using an integrated Post Processor
- Create shop floor documentation
- Manage design changes
- Import V4 data
- Facing
- Pocketing
- 4-Axis Pocketing

#### DAY 3

- Prismatic Roughing
- Profile contouring
- Groove milling
- Point to point
- Curve following
- Prismatic rework area
- Prismatic machining area
- Transition Paths (macros)

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