CATIA V5 Training - 3 Axis Surface Machining

WHO SHOULD ATTEND

CAM Programmer and Machinist

DURATION

3 days

METHODOLOGY

Practical hands-on by using CATIA software

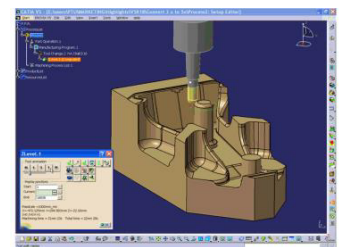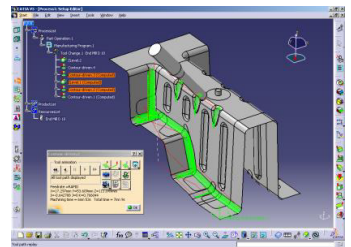## PREREQUISITE

CATIA V5 Fundamentals

INTRODUCTION:

This course will teach you how to use various functionalities common across all the Machining workbenches in CATIA. It will teach you the fundamentals of creating and simulating a Manufacturing Program. You will also learn how to define and manage NC programs dedicated to machining parts that are designed with Surface or Solid geometry. You will learn how to define 3-Axis Roughing, Semi finishing and finishing operations. The course will also help you to improve productivity in mould and die machining using various functionalities of 3-Axis Surface Machining.

OBJECTIVES:

- Identify and use Surface Machining Workbench Tools
- Define 3 Axis Surface Machining Operation in CATIA V5
- Create Machining Area Before performing Operations
- Define Rework Area
- Analyze and modify tool Path

COURSE OUTLINE:

- **Numerical Control Infrastructure (1st day – 2nd day)**

    - Identify and use Manufacturing workbench's Tools
    - Create a Manufacturing Program
    - Simulate Manufacturing Program
    - Manage Tools and Tool Catalogs
    - Define and verify the tool Path
    - Generate NC data using and integrated Post Processor
    - Create shop floor documentation
    - Manage design changes

- **3 Axis Surface Machining Operations (2nd day – 3rd day)**

    - What is 3-Axis Surface Machining
    - Create a geometrical Element Set
    - Create Machining Feature
    - Create 3-Axis Surface Operation:
      - Sweep Roughing
      - Roughing
      - Plunge
      - 4-Axis Curve Sweeping
      - Pencil
      - Z-level
      - Counter-Drive
      - Isoparametric Operations
      - Spiral Milling

- **Profile Contouring**

- **3/5 Axis Converter**

- **Analysis and modifying Tool Path**