

CATIA V5 MANUFACTURING

OBJECTIVE: This course enables users to define and manage NC programs dedicated to machining parts designed with Surface or Solid Geometry using prismatic (2.5 Axis) operations, 3-Axis machining techniques, 4 and 5 Axis machining operations and being aware of all capabilities in terms of strategies, parameters and transition paths.

DURATION 5 Days	STUDENT PROFILE CATIA V5 Users	PRE-REQUISITES V5 Fundamentals & General Machining Knowledge
TOPIC	DETAILS	DURATION
Numerical Control Infrastructure	▪ Workbench Presentation ▪ Define Setup (Part Operation) ▪ Machining Operation, Macros ▪ Tools Presentation ▪ Tool Path Verification & Simulation ▪ Auto sequence ▪ Auxiliary Operations ▪ Output generation ▪ Axial Operations ▪ Hole selection - Machining Pattern Management ▪ The different Process Views, Geometry wizard (edge, surface)	1 DAY
CATIA V5 Prismatic Machining	▪ Facing Operation ▪ Pocketing Operation ▪ Prismatic Roughing Operation ▪ Profile Contouring Operation ▪ Curve Following Operation ▪ Groove Milling Operations ▪ Creating a Machining Feature ▪ Prismatic Machining Area Creation ▪ Prismatic Rework Area Creation	1 DAY
CATIA V5 3- Axis Surfacing Machining	▪ Geometrical Zone ▪ Machining/ Slope Area Creation ▪ Rework Area Creation, Offset Group Creation ▪ Creating a 3-Axis Surface Machining Operation ▪ Sweep Roughing Operation ▪ Roughing Operation ▪ Sweeping Operation	1 DAY

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Information contained within is subject to change.

All classes are dependent on minimum enrollment

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	▪ Pencil Operation ▪ Z-Level Operation ▪ Contour Driven Operation ▪ Iso Parametric Machining Operation ▪ Spiral Milling Operation, Profile Contouring Operation	
Multi-Axis Machining	▪ Handling Machining Axis Changes ▪ Machine Rotation ▪ Multi-Axis Sweeping ▪ Multi-Axis Contour Driven ▪ Multi-Axis Curve Machining ▪ Multi-Axis Isoparametric Machining Operation ▪ Multi-Axis Drilling	1 DAY
Advanced Machining	▪ Multi Axis Flank Contouring ▪ Multi Axis Helix machining ▪ Cavities Roughing	1 DAY



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