# mecanica

# 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 3-Axis machining techniques and being aware of all capabilities in terms of strategies, parameters and transition paths.

| DURATION<br>4 Days | STUDENT PROFILE<br>CATIA V5 Users | PRE-REQUISITES<br>V5 Fundamentals &<br>General Machining<br>Knowledge |
|---|---|---|
| **TOPIC** | **DETAILS** | **DURATION** |
| **Numerical Control Infrastructure** | ▪ Workbench Presentation, Define Setup (Part Operation)<br>▪ Machining Operation, Macros, Tools Presentation<br>▪ Tool Path Verification & Simulation<br>▪ Auto sequence, Auxiliary Operations<br>▪ Output generation, Axial Operations<br>▪ Hole selection - Machining Pattern Management<br>▪ The different Process Views, Geometry wizard (edge, surface) | 2 DAYS |
| **CATIA VS 3- Axis Surfacing Machining** | ▪ Assembling Components<br>▪ Positioning Components using Constraints<br>▪ Editing Parts in an Assembly<br>▪ Designing in the Context of an Assembly<br>▪ Working with Components | 2 DAYS |