



Course Code	EDU-CAT-E-NCI-F
Brand & Release	CATIA V5R17
Duration	2 days
Language	English
Level	Fundamentals
Method	Companion and ILT

Training Material References

Instructor Foils: EDU-CAT-E-NCI-FI-V5R17 Foils: EDU-CAT-E-NCI-FF-V5R17 Exercises: EDU-CAT-E-NCI-FX-V5R17 Detailed Steps: EDU-CAT-E-NCI-FS-V5R17
--

Objectives

By the end of this course you will be able to create a Manufacturing Program, simulate it and generate outputs. You will be able to perform the following NC Manufacturing Infrastructure functions:
<ul style="list-style-type: none"> <li>- Manage Tools and Tool Catalogs</li> <li>- Define and Verify the Tool Path</li> <li>- Generate NC data using an integrated Post Processor</li> <li>- Create Shop-floor Documentation-</li> <li>- Manage Design Changes</li> <li>- Import V4 Data</li> </ul>

Participants' Profile

Manufacturing Users (NC Programmers)
--------------------------------------

Prerequisites

CATIA V5 Fundamentals
-----------------------

Content

<p>COMMON TOPICS</p> <p>Lesson 1 : Workbench Presentation</p> <p>Lesson 2 : Define Setup (Part Operation)</p> <p>Lesson 3 : Machining Operation, Macros, Tools Presentation</p> <p>Lesson 4 : Tool Path Verification &amp; Simulation</p> <p>lesson 5 : Auto sequence</p> <p>lesson 6 : Auxiliary Operations</p> <p>lesson 7 : Output generation</p> <p>lesson 8 : Axial Operations</p> <p>lesson 9 : Hole selection - Machining Pattern Management</p> <p>lesson 10 : The different Process Views</p> <p>Lesson 11 : Geometry wizard (edge, surface)</p> <p>ADVANCED TOPICS</p> <p>Lesson 12 : Import &amp; Modify Tool Path</p> <p>Lesson 13 : Resource Management</p> <p>Lesson 14 : PP Word Table customization</p> <p>Lesson 15 : V4 NCMill &amp; NCLathe Set Import (optional)</p> <p>Lesson 16 : Design Change Management</p> <p>Lesson 17 : Machining Processes</p> <p>APPENDIX</p> <p>Settings</p>
---

## Exercises

---

- Ex. 1: CATProcess & Workbench Presentation (5min) / All sectors
- Ex. 2: Machining Operation Presentation (15min) / All sectors
- Ex. 3: Replay Simulate and analyze (15min) / All sectors
- Ex. 4: Generate Auxiliary operation (30min) / All sectors
- Ex. 5: Generate Outputs (15min) / All sectors
- Ex. 6: Create Spot drilling and a drilling Operations (15min) / All sectors
- Ex. 7: Import APT source File (30min) / All sectors
- Ex. 8: Manage Resources (30min) / All sectors
- Ex. 9: Auto Complete (30min) / All sectors
- Ex. 10: PP Word Table (30min) / All sectors
- Ex. 11: Import V4 NC data (NCMill / NCLathe) (30min) / All sectors
- Ex. 12: Design Change (30min) / All sectors
- Ex. 13: Machining Process creation & instantiation (60min) / All sectors



Course Code	EDU-CAT-E-AMG-F
Brand & Release	CATIA V5R17
Duration	1.5 days
Language	English
Level	Fundamentals
Method	Companion and ILT

## Training Material References

Instructor Foils: EDU-CAT-E-AMG-FI-V5R17  
Foils: EDU-CAT-E-AMG-FF-V5R17  
Exercises: EDU-CAT-E-AMG-FX-V5R17

## Objectives

In this course, you will learn how to define:

- Multi-Axis Flank Contouring operation
- Multi-Axis Helix machining operation
- Cavities Roughing operation

## Participants' Profile

Manufacturing users knowing how to work with CATIA V5 parts

## Prerequisites

Fundamentals about CATIA V5

## Content

- Multi Axis Flank Contouring operation presentation
- Geometry tab
- Strategy tab
- Multi Axis Helix machining operation presentation
- Geometry tab
- Strategy tab
- Cavities Roughing operation presentation
- Geometry tab
- Strategy tab

## Exercises

- Ex. 1: Multi-Axis Contouring operation (20min) / All sectors
- Ex. 2: Flank finishing (5min) / All sectors
- Ex. 3: Start and End in closed pocket (5min) / All sectors
- Ex. 4: Joggle management (5min) / All sectors
- Ex. 5: Thickness on tool path (5min) / All sectors
- Ex. 6: Curves as part (5min) / All sectors
- Ex. 7: Local modifications (10min) / All sectors
- Ex. 8: Non contiguous drives (10min) / All sectors
- Ex. 9: Non contiguous drives 2 (10min) / All sectors
- Ex. 10: Non contiguous drives in same plane (10min) / All sectors
- Ex. 11: Auxilliary guiding (10min) / All sectors
- Ex. 12: Mx-Helix 01 : Rough body blade / All sectors
- Ex. 13: Mx-Helix 02 : Rough foot blade (10min) / All sectors
- Ex. 14: Mx-Pocket 01 : Rough Frame (10min) / All sectors
- Ex. 15: Mx-Pocket 02 : Rough Zone (10min) / All sectors
- Ex. 16: Mx-Helix 03 : Interpol (10min) / All sectors



Course Code	EDU-CAT-E-MMG-F
Brand & Release	CATIA V5R17
Duration	1 day
Language	English
Level	Fundamentals
Method	ILT

## Training Material References

Instructor Foils: EDU-CAT-E-MMG-FI-V5R17  
Foils: EDU-CAT-E-MMG-FF-V5R17  
Exercises: EDU-CAT-E-MMG-FX-V5R17

## Objectives

In this course, you will learn how to define and manage NC programs to machine Surface or Solid geometry using 5-Axis machining techniques.

## Participants' Profile

Manufacturing users

## Prerequisites

Fundamentals about CATIA V5  
NC Manufacturing Infrastructure  
3-Axis Surface Machining

## Content

- Multi-Axis Sweeping Operation
- Multi-Axis Contour Driven Operation
- Multi-Axis Curve Machining Operation
- Multi-Axis Isoparametric Machining Operation
- Multi-Axis Drilling Operation

## Exercises

- Ex. 1: Fender (90min) / All sectors  
Ex. 2: Hood (40min) / Automotive  
Ex. 3: Pocket (15min) / All sectors  
Ex. 4: Isoparametric and Drill (15min) / All sectors