

Mechanical Design  
Solutions

## V5F: CATIA V5 Fundamentals

Course Code EDU-CAT-E-V5F-F

Brand & Release CATIA V5R19

Duration 5 days

Level Fundamentals

### Objectives

Understand the CATIA V5 interface  
 Plan the construction of a part in order to properly convey its visual and functional aspects  
 Create simple parts in CATIA V5  
 Understand how to manage parts in the context of an assembly  
 Produce simple drawings and assembly layouts

### Participants' Profile

New CATIA V5 Mechanical Designers

### Prerequisites

Mechanical design experience.  
 Experience with the Windows operating system.

### Content

This course will introduce you to CATIA V5. It will teach you how to build simple parts and assemblies in CATIA, and how to make simple drawings of those parts and assemblies.

- Lesson 1: Introduction to CATIA
- Lesson 2: Profile Creation
- Lesson 3: Basic Features
- Lesson 4: Additional Features
- Lesson 5: Dress-Up Features
- Lesson 6: Reusing Data
- Lesson 7: Finalizing Design Intent
- Lesson 8: Assembly Design
- Lesson 9: Design in Context
- Lesson 10: Drafting ISO
- Lesson 10: Drafting ANSI



## **TEMARIO CATIA V5 FUNDAMENTALS (Desglosado)**

### **DAY 1**

#### **LESSON 1 – INTRODUCTION TO CATIA**

- 1.- Understand the CATIA software.
- 2.- Open CATIA
- 3.- Understand the CATIA interface.

#### **LESSON 2 – PROFILE CREATION**

- 1.- Create a new part.
- 2.- Select an appropriate sketch support.
- 3.- Create sketched geometry.
- 4.- Constrain the sketch.
- 5.- Create the pad feature.
- 6.- Save and close the document.

#### **LESSON 3 – BASIC FEATURES**

- 1.- Determine a suitable base feature.
- 2.- Create pad and pocket features.
- 3.- Create holes.
- 4.- Create fillets and chamfers.
- 5.- Hide reference planes and deactivate holes.

### **DAY 2**

#### **LESSON 4 – ADDITIONAL FEATURES**

- 1.- Create feature profiles.
- 2.- Create multiple-profile sketch features.
- 3.- Create reference geometry.
- 4.- Create shaft and groove features.
- 5.- Shell the model.

#### **LESSON 5 – DRESS-UP FEATURES**

- 1.- Apply a draft.
- 2.- Create a stiffener.
- 3.- Create threads and taps.
- 4.- Edit features.

## **DAY 3**

### **LESSON 6 – REUSING DATA**

- 1.- Duplicate features.
- 2.- Transform body.
- 3.- Copy and paste data.
- 4.- Insert data from a catalog.

### **LESSON 7 – FINALIZIN DESIGN INTENT**

- 1.- Apply material properties.
- 2.- Analyze the model.
- 3.- Create formulas.

## **DAY 4**

### **LESSON 8 – ASSEMBLY DESIGN**

- 1.- Create a new CATProduct.
- 2.- Assemble the base component.
- 3.- Manipulate the position of the component and fix it.
- 4.- Assemble other components and fully constrain their placement.
- 5.- Save the assembly.

### **LESSON 9 – DESIGN IN CONTEXT**

- 1.- Open and existing assembly.
- 2.- Insert a new model.
- 3.- Create a sketch with external references.
- 4.- Create assembly – level features:
  - a.- Assembly – level hole feature
  - b.- Remove Boolean operation

## **DAY 5**

### **LESSON 10 – DRAFTING**

- 1.- Start a new drawing.
- 2.- Apply a title block.
- 3.- Create views.
- 4.- Create dimensions and annotations.
- 5.- Save the drawing.
- 6.- Print the drawing.

### **MASTER PROJECT**

- 1.- Create a part.
  - \* Create the support used in the Drill Press assembly. The part uses features learned in this course.
- 2.- Finalize the part.
  - \*Apply material and take measurements of the model.
- 3.- Create the Drill Press Assembly.
  - \*Using the models you have built in case studies, as well as additional models, create the Drill Press Assembly.
- 4.- Modify a part in context of the assembly.
  - \*Makes changes to the support within the context of the assembly.
- 5.- Create and assembly drawing.
  - \*Create a simple assembly drawing of the Drill Press Assembly.