



CATIA V5 Generative Structural Analysis

Course Code	EDU-CAT-en-V5A-F
Brand & Release	CATIA V5R21, V5R23 (V5-6R2013)
Duration	2 days
Level	Fundamentals
Prerequisites	CATIA V5 Fundamentals

Objectives:

This course will introduce the concepts and benefits of Finite Element Analysis and the general analysis process. It will teach you how to prepare a model for analysis, create 1D, 2D and 3D FE models, and compute a simple static analysis for a single part or an assembly.

Class Structure:

- Create a Finite Element Analysis model
- Prepare a solid or surface model for analysis
- Create 1D, 2D and 3D meshes for beam, surface, and solid models
- Assign properties, loads and constraints, and define assembly connections
- Compute an analysis for a part or an assembly
- Generate and display analysis results

Class Lessons:

Lesson 1: Introduction to Finite Element Analysis

- Step 1: Finite Element Analysis Process
- Step 2: Introduction to GPS workbench

Lesson 2: Pre-processing

- Step 1: What is Pre-processing
- Step 2: Applying Material
- Step 3: Generating Mesh
- Step 4: Assigning Physical Property
- Step 5: Applying Restraints and Loads
- Step 6: Checking the Finite Element Model

Lesson 3: Computation

- Step 1: Introduction to Computation
- Step 2: Solving Methods
- Step 3: Computing a Static Case

Lesson 4: Post-processing

- Step 1: What is Post-processing
- Step 2: Visualization Images
- Step 3: Sensors
- Step 4: Report Generation

Lesson 5: Mesh Refinement

- Step 1: What is Mesh Refinement
- Step 2: Adaptive Mesh Refinement

Lesson 6: Assembly Structural Analysis

- Step 1: Assembly Structural Analysis
- Step 2: Analysis Assembly
- Step 3: Analysis Connections and Connection Properties
- Step 4: Virtual Parts
- Step 5: Distributed Mass