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Training Course

MECANICA SOLUTIONS CATIA V5

ADVANCED DESIGN

OBJECTIVE: Enable users and designers to learn advanced tools of Modeling using complex Solid and Surface features of CATIA V5 to enhance their productivity by reducing modeling time and better control over design modifications. Also learn advanced assembly functions to control complex assembly structures, top to down and down to top assembly concept and modeling in assembly context as per industry standards and practices.

DURATION 5 DAYS

STUDENT PROFILE: CATIA V5 USERS **PRE-REQUISITES: V5 FUNDAMENTALS**

TOPIC & DETAILS	TOPIC DURATION

CATIA Surface Design — 2 Days

- Introduction to Surface Design
- Creating Wireframe Geometry
- Creating Basic and Complex Surfaces
- Performing Operations on the Geometry
- Completing the Geometry in Part Design, Modifying the Geometry
- Draft Analysis and Surface Curvature Analysis

Part Design Expert — 1.5 Days

- Advance Sketch-Based Features
- Using 3D Elements to Create a Part
- Advance Dress-Up Features
- Part Geometry Analysis, Part Manipulations, Annotating Models

Assembly Design Expert — 1.25 Days

- Designing and Managing Contextual Parts
- Creating and Using Published Geometry
- Flexible Sub- Assemblies
- Measuring, Sectioning, Clash
- Managing Scenes, Working with Large Assemblies
- Generating Reports, Numbering & Annotations

Drawing Generation Review — 0.25 Days

- Advance Drawing Generation Techniques
- Managing Large Assembly Drawings

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FUNDAMENTALS

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OBJECTIVE: The students will be introduced to the CATIA V5 fundamental concepts and interface. Students will learn the concept of sketch-based features, the management of parts through an assembly and how to generate standard views from this assembly. The part creation in this course is mainly focused on the creation of parts based on 2D profiles (sketches), and on the assembly of existing components.

DURATION 5 DAYS

STUDENT PROFILE: CATIA V5 USERS **PRE-REQUISITES: NONE**

TOPIC & DETAILS	TOPIC DURATION
Sketcher • Getting Started with CATIA V5 • Sketcher Geometry Management • Sketcher Constraints Management • Sketch Analysis Tools	1 Day
Part Modeling • Sketch-Based Features • Dressing-Up Features • Transforming Features	2 Days
Assembly • Assembling Components • Positioning Components using Constraints • Editing Parts in an Assembly • Designing in the Context of an Assembly • Working with Components	1 Day
Drafting & Drawing Generation Review • Starting a Drawing and View Generation • Additional View Generation • Dimensioning and Annotating • Editing View's Layout and Properties • Finalizing the Drawing and Printing	1 Day

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Training Course

MECANICA SOLUTIONS 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 4 DAYS

STUDENT PROFILE: CATIA V5 USERS

PRE-REQUISITES: CATIA V5 FUNDAMENTALS & GENERAL MACHINING KNOWLEDGE

TOPIC & DETAILS	TOPIC DURATION

Numerical Control Infrastructure

2 Days

- 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 Mgmt
- The different Process Views, Geometry wizard (edge, surface)

CATIA V5 3- Axis Surfacing Machining

2 Days

- Assembling Components
- Positioning Components using Constraints
- Editing Parts in an Assembly
- Designing in the Context of an Assembly
- Working with Components

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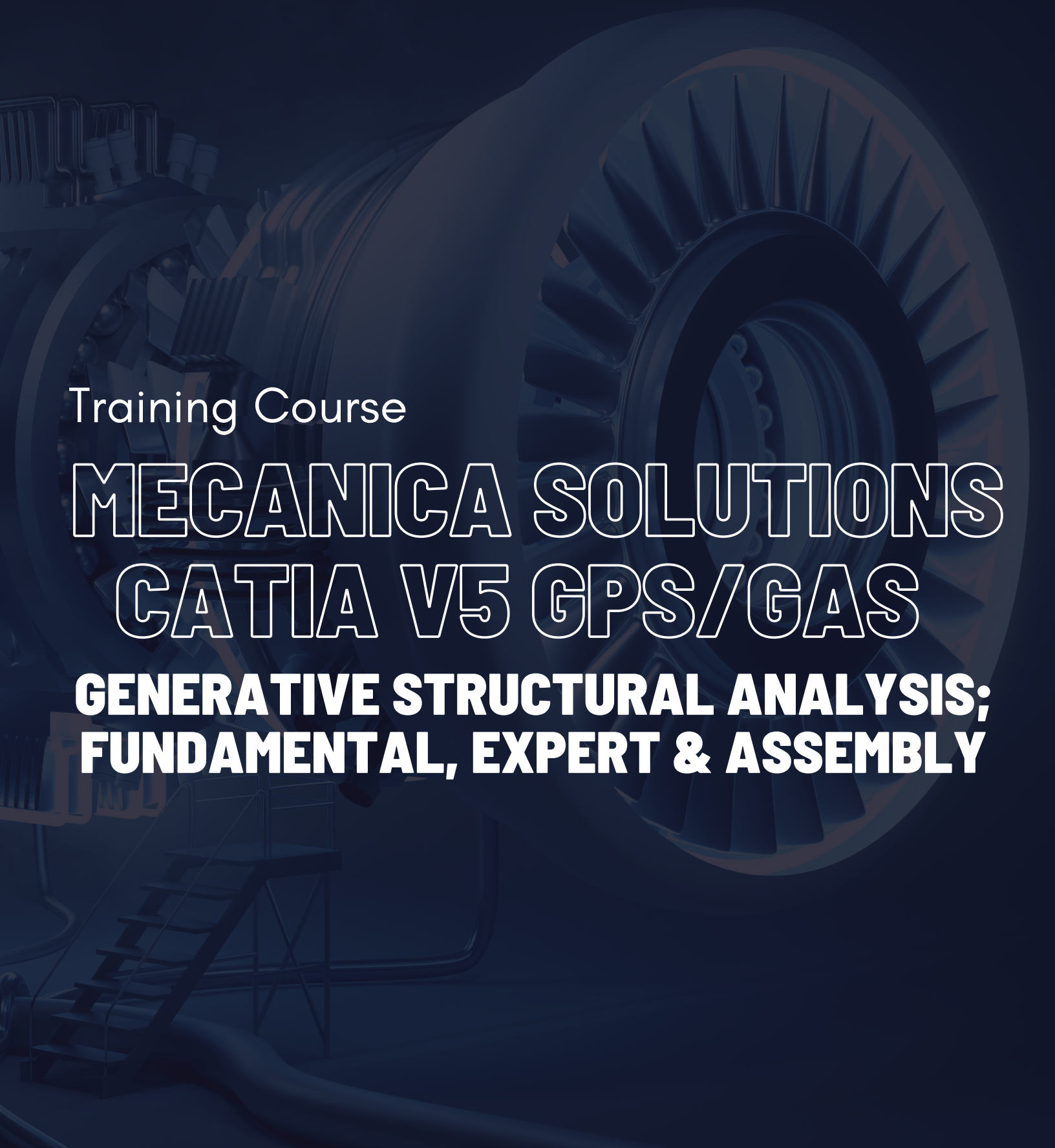

Training Course

MECANICA SOLUTIONS CATIA V5 GPS/GAS

GENERATIVE STRUCTURAL ANALYSIS; FUNDAMENTAL, EXPERT & ASSEMBLY

OBJECTIVE: This course covers the main tools for Structural Analysis on a single part. Throughout this course, you will learn how to perform a basic static analysis using the finite elements method. Upon completion of this course you will be able to: Define and customize material properties, Apply pressure, acceleration and force density loads; define virtual parts, Apply pivot, ball-joint, and user-defined restraints, Compute a frequency analysis for a single part, Create planar sections with which to visualize internal result values, Compute and refine a mesh using adaptive meshing in order to achieve a pre-defined accuracy.

DURATION 3 DAYS

STUDENT PROFILE: CATIA V5 MECHANICAL DESIGNERS **PRE-REQUISITES: V5 FUNDAMENTALS**

TOPIC & DETAILS	TOPIC DURATION
Generative Part Structural Analysis Fundamental	1 Day

Introduction to Finite Element Analysis

- What is Finite Element Analysis,
- Why to Use Finite Element Analysis
- Application of Finite Element Analysis

Introduction to GPS Analysis

- Accessing the Generative Part Structural Analysis Workbench
- The Generative Part Structural Analysis Interface
- The GPS General Process
- The Generative Part Structural Analysis Tree Structure

GPS Pre-Processing

- Managing Mesh-Part,
- Defining Restraints, Defining Loads

Computation

- Specifying the External Storage
- Computing a Static Case

TOPIC & DETAILS	TOPIC DURATION

Generative Part Structural Analysis Fundamental (cont'd)

GPS Post-Processing

- Results Visualization
- Results Management
- Refinement

Managing Analysis

- About Saving an Analysis Document,
- About Save As
- How to Use Save Management

SAVING DOCUMENT USING 'SEND TO' MECHANISM, USER SETTINGS

Generative Part Structural Analysis Expert

1 Day

This course will focus on advanced Finite Element Analysis pre-processing techniques and post-processing tools, including the concept of virtual parts to avoid excessive geometric modeling. It will teach you how to perform a frequency analysis on a single part, and the use of adaptive meshing to achieve pre-defined accuracy.

GPS Advanced Pre-Processing Tools

- Advanced Pre-Processing Tools
- Frequency Analysis

Computation

- Computing a Frequency Case
- Computing with Adaptivity
- Historic of Computation

GPS Advanced post-Processing Tools

- Results Visualization
- Results Management

REFINEMENT

TOPIC & DETAILS	TOPIC DURATION

Generative Assembly Structural Analysis

1 Day

This course will focus on advanced Finite Element Analysis pre-processing techniques and post-processing tools, including the concept of virtual parts to avoid excessive geometric modeling. It will teach you how to perform a frequency analysis on a single part, and the use of adaptive meshing to achieve pre-defined accuracy.

Introduction to GAS

- Generative Assembly Structural Analysis Overview
- Hypotheses Used for Analysis

Analysis Connections

- Analysis Connection using Assembly Constraints
- General Analysis Connection
- Defining Line Analysis Connections
- Defining Point Analysis Connections, Defining Surface Analysis
- Connections Points to Points Analysis Connection
- Set of Analysis Connections

GAS Connection Properties

- Face to Face Connection Properties
- Distant Connection Properties
- Welding Connection Properties
- Nodes to Nodes Connection Property

Compute a Static Analysis for an AssemblyAnalysis Assembly Management

CREATE AND MANAGE AN ANALYSIS ASSEMBLY MODEL USING EXISTING MESHED PARTS

Training Course

MECANICA SOLUTIONS CATIA V5

SURFACE AND ADVANCED SURFACE

OBJECTIVE: This course covers tools for surface design included in the Generative Shape Design Workbench that are not present in the Wireframe and Surface Design Workbench. At the end of the course, the student will be able to model complex fillets and analyze surface quality.

DURATION 4 DAYS

STUDENT PROFILE: CATIA DESIGNER **PRE-REQUISITES: CATIA V5 FUNDAMENTALS**

TOPIC & DETAILS	TOPIC DURATION
CATIA Surface Design • Introduction to Surface Design • Creating Wireframe Geometry, Creating Basic and Complex Surfaces • Performing Operations on the Geometry • Completing the Geometry in Part Design, Modifying the Geometry Draft • Analysis and Surface Curvature Analysis	2 Days
CATIA Surface Design Expert • Surface Design Overview • Review of tools covered in a Surface Design course • Extremums, Creating a Connect Curve • Extracting Multiple Edges from a Sketch • Wireframe Analysis and Repair Curve Connect Checker • Porcupine Curvature Analysis • Smoothing Curves, Common Tools for Swept Surfaces • Creating a Spine Curve, Laws, Creating Advanced Swept Surfaces, Creating a Line Type Swept Surface • Creating a Circle Type Swept Surface, Creating an Adaptative Swept Surface • Creating Advanced Blend Surfaces, Creating a Fillet using Hold Curve and Spine • Blending Vertex when Making Fillets • Inverting Orientation • Creating a Conic Blend Surface, Creating a Blend Surface • Surface Analysis and Repair Connect Checker • Healing Surfaces, Draft Analysis, Mapping Analysis • Managing Features and Open Bodies, Managing the Geometry and Open Bodies • Hybrid Design, Working with Hybrid Part	2 Days

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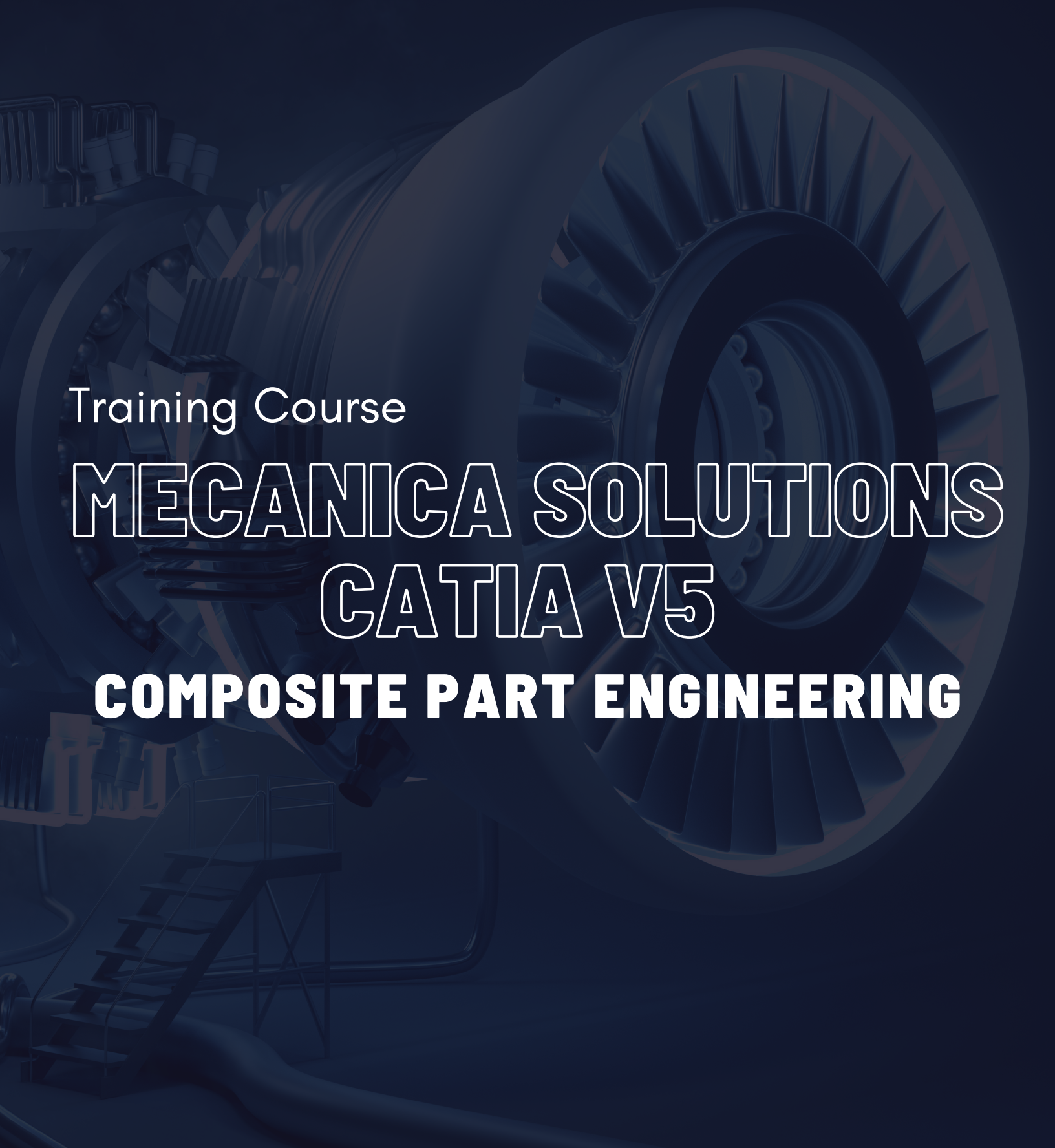

Training Course

MECANICA SOLUTIONS CATIA V5

COMPOSITE PART ENGINEERING

OBJECTIVE: The students will be introduced to how to design simple Composites Parts using a Manual approach. Then learn how to use a Zone-based approach to complete the preliminary design and then the detail ed design. Users will learn the advance Grid based approach of composite modeling. They will also learn how to generate plies automatically, use the analysis tools and simulate fiber behaviour. Finally, the students will learn how to generate exact solids, create composites drawings, Import / Export Ply data and create Ply Book.

DURATION 5 DAYS

STUDENT PROFILE:
CATIA V5 COMPOSITE DESIGNERS

PRE-REQUISITES:
V5 ADVANCED DESIGN

TOPIC & DETAILS	TOPIC DURATION

Introduction to Composites Design

1 Day

[Common for Design & Manufacturing Users]

- Getting Started with CATIA V5 CPE
- Familiarizing with Composites Design approaches and terminologies
- Design using manual approach
- Create, modify and analyze the piles with manual approach

Creating Zone Based Design

0.75 Day

[Only for Composite Design Users]

- Design using Classic zone approach
- Design using Solid zone approach
- Create & Modify Piles with zone approach
- Create & Modify Piles using solid approach

Composite modeling using Grid Approach

1.5 Days

[Only for Composite Design Users]

- Understand the concept of Grid approach in Composites Design
- Defining composite parameter, Grid Panel and Virtual Stacking
- Generate plies using the Grid approach
- Modify the Ply geometry
- Create a solid or a top surface using the ply geometry
- Create and modify a composite part using the Composites Grid Design approach

TOPIC & DETAILS	TOPIC DURATION
Reviewing Composite Entities [Common for Design & Manufacturing Users] • Getting Started with CATIA V5 CPE • Familiarizing with Composites Design approaches and terminologies • Design using manual approach • Create, modify and analyze the piles with manual approach	0.5 Day
Generating Production Data [Only for Manufacturing Users] • Apply the manufacturing constraints like Producibility • Generate a Manufacturing Stacking from an Engineering Stacking • Synchronize the link between the Manufacturing and Engineering parts • Creating Dart • Create Multi-Splice and No-Splice on the Plies • Compute Flattening	0.75 Day
Creating Production Documents [Common for Design & Manufacturing Users] • Exporting the Composite data & core samples results Create a drawing from design • Create Ply book, review Ply table and perform inspections on design	0.5 Day

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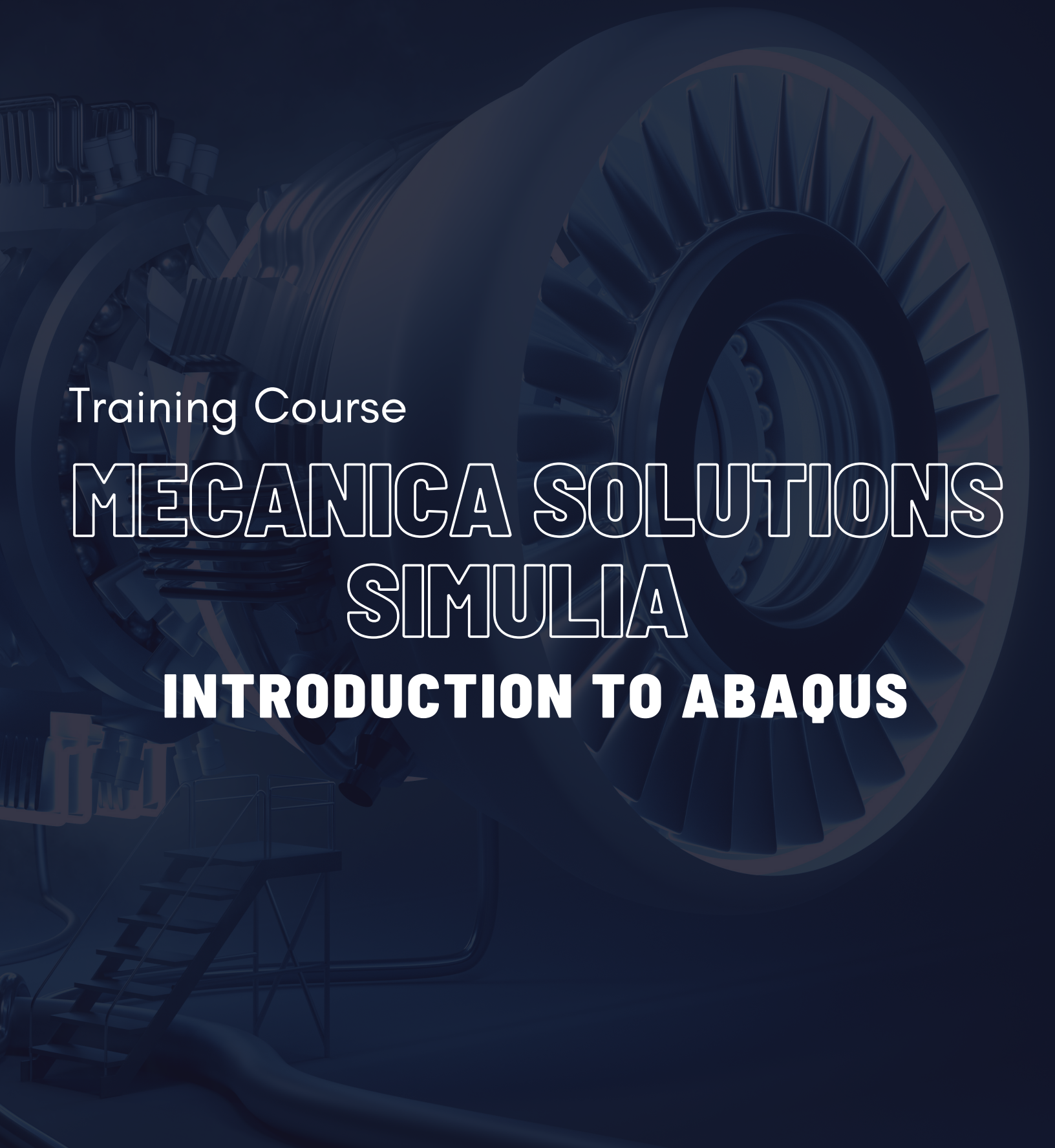

Training Course

MECANICA SOLUTIONS SIMULIA

INTRODUCTION TO ABAQUS

OBJECTIVE: This course is a comprehensive and unified introduction to the modeling and analysis capabilities of Abaqus. It teaches you how to solve linear and nonlinear problems, submit and monitor analysis jobs and view simulation results using the interactive interface of Abaqus.
The following products are covered by this seminar: Abaqus/CAE, Abaqus/Standard and Abaqus/Explicit.
This course is divided into lectures, demonstrations and workshops. The course's workshops are integral to the training. They are designed to reinforce concepts presented during the lectures and demonstrations. They are intended to provide users with the experience of running and trouble-shooting actual Abaqus analyses.

DURATION 4 DAYS TRAINING + 1 DAY MENTORING WORKSHOP

STUDENT PROFILE: SIMULATION/ABAQUS USERS

TOPIC & DETAILS	TOPIC DURATION
Introduction to Abaqus	4 Days

- Linear and nonlinear structural analysis
- Static, dynamic and heat transfer analysis
- Material models: linear elasticity, hyperelasticity, and metal plasticity.
- Loads and constraints
- Modeling contact
- Selecting the appropriate elements for your problem
- Feature-based modeling, parts and assemblies
- Working with CAD geometry and imported meshes
- Mesh generation techniques
- Creating, submitting and monitoring analysis jobs
- Viewing simulation results
- Restarting an analysis

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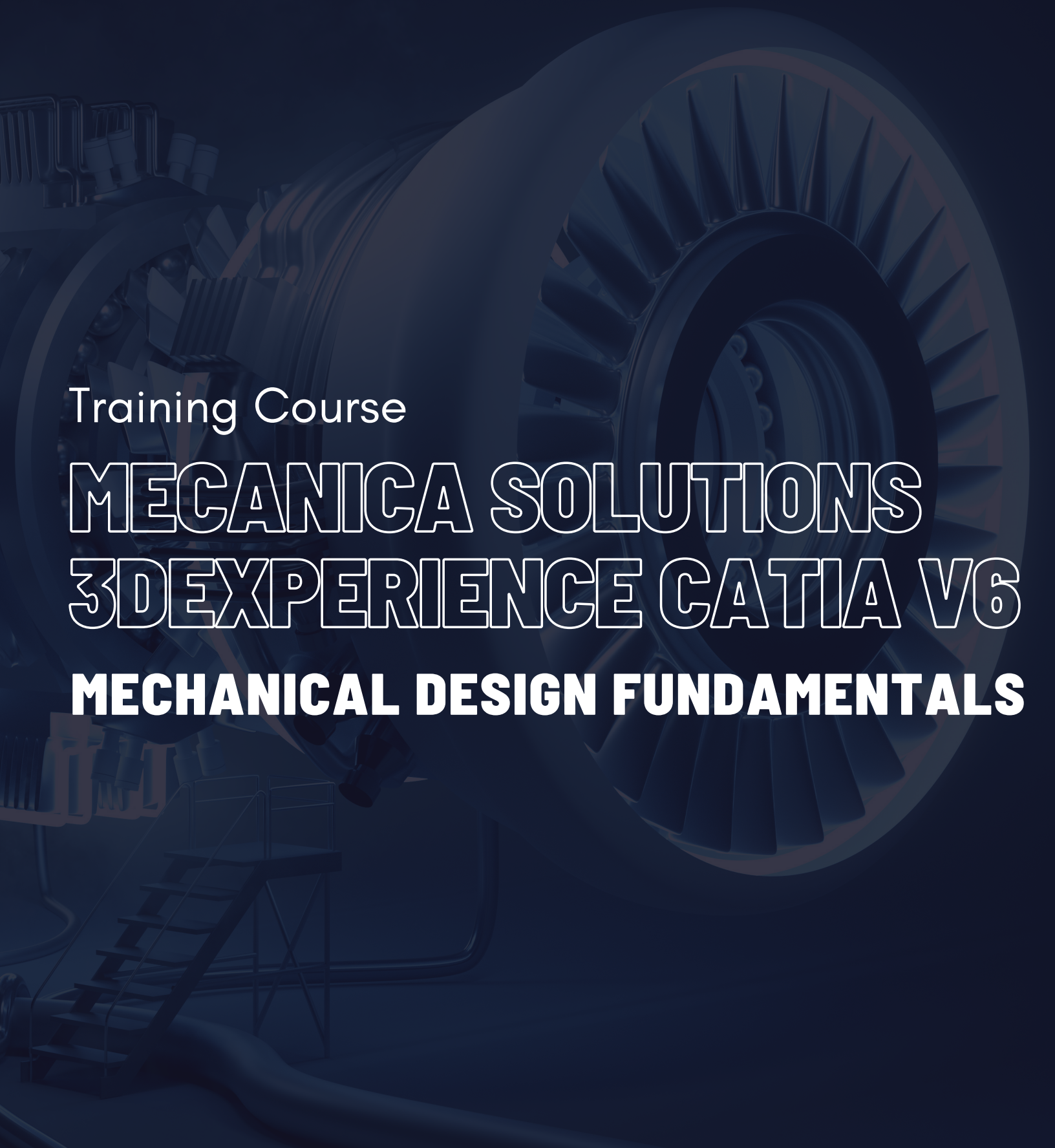

Training Course

MECANICA SOLUTIONS 3DEXPERIENCE CATIA V6

MECHANICAL DESIGN FUNDAMENTALS

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LEARNING EXPERIENCE | COURSE CATALOG

Course Code	CAT-en-3DF-F-15-201
Brand	CATIA
Discipline	• 3DEXPERIENCE platform, Cyber-Physical Systems, Digital • Manufacturing, Electrical and Fluids Engineering, Global Product • Development, Mechanical Engineering, Multi-Discipline Engineering
Available Release	3DEXPERIENCE R2020x
Duration	32 Hours
Course Material	English
Level	Fundamental
Audience	Mechanical and Sheet Metal Designers
Description	This course will teach you how to create simple parts, assemblies and drawings. You will learn how to use different feature-based tools to build, review and modify a model. You will also learn how to create and analyze assemblies and how to produce a drawing with different views. Finally, you will learn how to dimension the drawing and annotate the views.
Objectives	Upon completion of this course you will be able to: • Create a new PLM object • Create and constrain 2D sketches • Complete a 3D model using features • Review and edit the features • Create parameters and formulas in the 3D model • Create a new product and add components to it • Move the components within a product by positioning them using assembly constraints • Create simple projection views and section views of 3D parts • Position the views on a drawing sheet • Add dimensions and annotations to the views • Finalize the drawing sheet by adding borders and title blocks
Prerequisites	Students attending this course should have completed the Gateway to the 3DEXPERIENCE platform course.
Available Online	Yes

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Training Course

MECANICA SOLUTIONS 3DEXPERIENCE CATIA V6

TRANSITION TO THE 3DEXPERIENCE PLATFORM FOR MECHANICAL DESIGNERS

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LEARNING EXPERIENCE | COURSE CATALOG

Course Code	CAT-en-3DMT-F-15-201
Brand	CATIA
Discipline	• Styling, Systems Engineering, Mechanical Engineering, Multi- • Discipline Engineering, Electrical and Fluids Engineering, Industrial • Engineering, Multiphysics Simulation
Available Release	3DEXPERIENCE R2020x
Duration	12.0 Hours
Course Material	English
Level	Fundamental
Audience	Designers who need to work with mechanical parts
Description	This course addresses the needs of Mechanical Designers. It will first teach you how to design a new part with the 3DEXPERIENCE platform, insert the part in a product then position and constrain it. You will learn how to assign material properties and compute weight, then complete a simple drawing. Finally, you will learn how to create a new part version, replace the original part and update the product. More advanced topics will also be covered: they will teach you how to manage complex product structures, create product features, manage catalogs and analyze assemblies.
Objectives	Upon completion of this course, you will be able to: • Create new products and parts • Insert a part in a product and position it • Apply materials to parts • Calculate the weight of a product • Insert and complete a drawing • Create a new part version • Replace a part and update a product • Design parts in context • Create assembly features and catalogs • Analyze the assemblies
Prerequisites	Students attending this course should have completed the Gateway to the 3DEXPERIENCE platform course. They should also be familiar with CATIA V5 Mechanical Design.
Available Online	Yes

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Training Course

MECANICA SOLUTIONS

GATEWAY TO THE 3DEXPERIENCE PLATFORM

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LEARNING EXPERIENCE | COURSE CATALOG

Course Code	CAT-en-GTX-F-15-181
Brand	CATIA
Discipline	• Styling, Systems Engineering, Mechanical Engineering, Multi- • Discipline Engineering, Electrical and Fluids Engineering, • Manufacturing Engineering, Industrial Engineering, Multidiscipline • Simulation, Multiphysics Simulation, Product Planning and Program • Management, Strategic Customer Relationship Management, Global • Product Development, IP Classification and Protection, Quality and • Compliance Management, 3DEXPERIENCE platform, Options, • Customization, Deployment and Application Development
Available Release	3DEXPERIENCE R2018x
Duration	4 Hours
Course Material	English , French , German , Japanese
Level	Fundamental
Audience	Users of the 3DEXPERIENCE platform
Description	This course is the entry point to the 3DEXPERIENCE platform. Its purpose is to empower users of the platform by teaching them how to access their work environment, navigate, search, and work on their data, use and manage their dashboard and collaborate with their peers thanks to communities. This course will teach you the new interface and functionalities of the 3DEXPERIENCE platform. You will learn how to connect to the platform, manage your projects, search documents and share content along with knowledge or skills with other users.
Objectives	Upon completion of this course, you will be able to: • Understand the 3DEXPERIENCE interface • Connect to the 3DEXPERIENCE platform • Access your Dashboard • Use the 6WTags for searching content • Share various documents with other users through • 3DSpace • Use standard menus and commands • Explain the functionalities of various apps in the • 3DEXPERIENCE platform • Import new data and export it as 3DXML files • Search for a 3D data using different methods • Explore and open 3D data • Manipulate the tree • Filter data
Prerequisites	There are no prerequisites for this course
Available Online	Yes