

Course Catalog

V5-6R2014 to V5-6R2018

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3DEXPERIENCE®

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CATIA

CATIA Analysis V5

CATIA V5 Analysis (V5A)	
Course Code	CAT-en-V5A-F-V5R24
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers and Structural Analysts
Description	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.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create a Finite Element Analysis model - Prepare a solid or a 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
Prerequisites	Students attending this course should have followed the CATIA V5 Fundamentals course.
Available Online	Yes

ELFINI Structural Analysis (EST)	
Course Code	CAT-en-EST-F-V5R27
Available Release	V5-6R2017
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to use the advanced functionalities provided by the ELFINI Solver. It will also guide you on how to use these functionalities in the overall FEA process. Additionally, you will learn about new analysis types such as Buckling Case, MultiLoads Case, Transfer of Loads and Solution, etc. You will also learn how to use advanced capabilities for post-processing of results.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Use the advanced pre-processing capabilities (such as loads and boundary conditions) to create more realistic FE models - Visualize the images of objects used for pre-processing (which is not possible in GPS) - Use advanced case features (such as Buckling, MultiLoads, Envelope) in addition to the standard Static and Frequency cases - Create your own result image templates, customize the post-processing result images, group selective entities in the result images, and create advanced reports
Prerequisites	Students attending this course should be familiar with Generative Part Structural Analysis Fundamentals, Generative Assembly Structural Analysis, FEM Surface Meshing.

ELFINI Structural Analysis (EST)

Available Online

Yes

FEM Solid (FMD)	
Course Code	CAT-en-FMD-F-V5R27
Available Release	V5-6R2017
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	<p>This course introduces you to the various functionalities available with the FEM Solid (FMD) license in CATIA. You will learn how to create a 3D mesh from existing 2D mesh parts with the help of functionalities such as Tetrahedron Filler, Sweep3D, mesh part transformations, and mesh part extrusion. You will also learn how to directly generate a 3D mesh using OCTREE Tetrahedron Mesher. You will learn how to analyze the generated 3D mesh using the available mesh quality criteria, and how to import/export the meshes into/from CATIA.</p>
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Use different solid meshers such as Tetrahedron Filler, OCTREE Tetrahedron Mesher, Sweep3D Mesher - Create a solid mesh using mesh part transformations like Translation, Rotation, and Symmetry on 3D mesh parts - Analyze the solid meshes using the available mesh quality criteria - Import/Export the meshes into/from CATIA
Prerequisites	<p>Generative Part Structural Analysis Fundamentals, Generative Part Structural Analysis Expert, Generative Assembly Structural Analysis, FEM Surface Meshing</p>

FEM Solid (FMD)

Available Online

Yes

FEM Surface (FMS)	
Course Code	CAT-en-FMS-F-V5R27
Available Release	V5-6R2017
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to create and edit meshes using Beam Mesher, OCTREE Triangular Mesher, Surface Mesher and Advanced Surface Mesher. You will learn how to create meshes from existing meshes using mesh transformations. You will also learn how to create different types of welding meshes. This course will cover how to make use of knowledgeware parameters while creating surface meshes. Additionally, you will learn how to analyze the mesh quality using the available mesh quality checks.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Use various meshers such as surface mesher, advanced surface mesher, beam mesher, and OCTREE triangle mesher - Create a new mesh using existing mesh with the help of mesh transformation tools and mesh operators - Edit the mesh using the mesh edition tools available within the above mentioned meshers - Analyze the mesh quality using various mesh quality checks and tools like Free Edges, Duplicate Elements, Duplicate Nodes, Mesh Interference Checks

FEM Surface (FMS)	
Prerequisites	Students attending this course must be familiar with the fundamentals of CATIA Generative Part Structural Analysis and Generative Assembly Structural Analysis.
Available Online	Yes

Generative Assembly Structural Analysis (GAS)

Course Code	CAT-en-GAS-F-V5R27
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to conduct a Finite Element Analysis using an existing assembly. You will learn how to create connections between assembly components and how to assign appropriate connection properties. You will also learn how to create an analysis assembly from the existing meshed parts.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Interpret the various types of hypotheses that are used for creating an assembly analysis and differentiate between them - Define analysis connections between assembly components - Use existing assembly constraints to automatically create analysis connections - Assign a connection property to the appropriate analysis connection - Compute a static analysis for an assembly - Create and manage an analysis assembly model using existing meshed parts
Prerequisites	Students attending this course should have completed the CATIA V5 Fundamentals and Generative Part Structural Analysis Fundamentals courses

Generative Assembly Structural Analysis (GAS)

Available Online

Yes

Generative Dynamic Response Analysis (GDY)	
Course Code	CAT-en-GDY-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Structural Analysts
Description	This course will teach you how to perform a harmonic or transient analysis on a single part using finite elements. You will learn how to generate and visualize 2D graphical results and how to export the resulting data in Text or Excel format.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Understand and differentiate between harmonic and transient analyses - Define the load and restraint excitations - Define the correct prerequisites for an excitation case - Visualize and animate the 3D images of the analysis results - Generate the translation, velocity, and acceleration graphs - Export the results data in Text or Excel format
Prerequisites	Students attending this course should have taken the CATIA V5 Fundamentals and Generative Part Structural Analysis Fundamentals courses
Available Online	Yes

Generative Part Structural Analysis Expert (GPE)	
Course Code	CAT-en-GPE-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to use advanced Finite Element Analysis pre-processing techniques and post-processing tools, including the concept of defining virtual parts to avoid excessive geometric modeling. You will learn how to perform frequency analysis on a single part, and how to use adaptive meshing to achieve pre-defined accuracy.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Define and customize the material properties of the parts to be analyzed - Apply pressure, acceleration, and force density loads - Define virtual parts to simplify the analysis - Apply pivot, ball-joint, and user-defined restraints - Compute the frequency analysis for a single part - Create planar sections to visualize the internal result values - Compute and refine a mesh using adaptive meshing in order to achieve the pre-defined accuracy
Prerequisites	Students attending this course should have taken the CATIA V5 Fundamentals and Generative Part Structural Analysis Fundamentals courses

Generative Part Structural Analysis Expert (GPE)

Available Online

Yes

Generative Part Structural Analysis Fundamentals (GPF)	
Course Code	CAT-en-GPF-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you the basic concepts of Finite Element Analysis and the general analysis process. You will learn how to perform a simple static analysis on a single part using finite elements, and how to produce the final report of the analysis results.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Understand why, when, and how to use Finite Element Analysis - Use different element types and shapes to mesh a part - Apply clamp, slider, and iso-static restraints - Apply force, moment, and displacement loads - Compute the static analysis for a single part - Visualize the images of the analysis results and produce the analysis reports - Refine existing meshes to produce more accurate results
Prerequisites	Students attending this course should be familiar with the fundamentals of CATIA V5
Available Online	Yes

CATIA

CATIA Equipment and Systems Engineering V5

CATIA V5 for Electrical Designers (V5VE)	
Course Code	CAT-en-V5VE--V5R27
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	48 hours
Course Material	English
Level	Fundamental
Audience	Automotive Electrical Harness Designers and new CATIA Electrical Designers.
Description	<p>This course will introduce you to the fundamental concepts of CATIA V5. You will learn how to design parts and assemblies and create simple drawings. Once you are familiar with the fundamentals, you will be introduced to the Electrical Library products. You will learn how to create the Electrical Harness for an automotive assembly. You will also learn how to map the functional specifications of the Harness system to the digital mock-up created in CATIA V5 and create the harness documentation.</p>
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Understand the CATIA V5 interface - Design an automotive wire harness - Route the signals and create the wires - Flatten and synchronize the harness - Design and manage parts in the context of an assembly - Generate harness documentation - Produce simple drawings
Prerequisites	Students attending this course should be familiar with Mechanical Design and the Windows Operating System.

CATIA V5 for Electrical Designers (V5VE)

Available Online

Yes

Electrical 3D Design and Documentation (EC1)	
Course Code	CAT-en-EC1-F-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Electrical Harness Designers
Description	This course will teach you how to use the Electrical 3D Design Part and Assembly workbenches with a focus on the Consumer Goods industry. You will learn how to create electrical assemblies, route multi-branchables, route wires, and create drawings of the electrical harness assemblies.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create harnesses suitable for consumer goods - Design wire harnesses with appliances - Route signals and create wires - Flatten and synchronize an electrical or a geometrical harness - Generate the harness documentation - Produce simple drawings
Prerequisites	Students attending this course should be familiar with the fundamentals of Mechanical Design. They should also know Part Design, the Assembly Design and the Drafting workbenches.
Available Online	Yes

Electrical Harness Flattening (EHF)	
Course Code	CAT-en-EHF-F-V5R24
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Electrical Designers
Description	<p>This course introduces you to the Electrical Harness Flattening workbench. You will learn how to flatten and synchronize an electrical / geometrical harness integrated within the Digital Mock-Up. You will also learn how to modify the bundle segments of a harness. Additionally, the course teaches you how to define and generate a report. It teaches you how to create a 2D drawing of a 3D harness. You will also learn how to create a Catalog Text Template for annotations and dimensions.</p>
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Flatten and synchronize the electrical or the geometrical harnesses - Modify the bundle segments of a harness to fit your drawing - Define and generate reports - Create 2D drawings of 3D harnesses - Create Text Templates Catalog
Prerequisites	<p>Students attending this course should be familiar with the Catalog Editor, CATIA V5 Electrical Harness Installation and Assembly, and Electrical Wire Routing workbenches of CATIA V5.</p>

Electrical Harness Flattening (EHF)

Available Online

Yes

Electrical Librarian and Harness Installation (ELI)	
Course Code	CAT-en-ELI-F-V5R28
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Electrical Harness Designers
Description	This course will teach you how to create electrical devices and their catalog. You will learn how to connect devices, design harnesses and create bundle segments. You will also learn how to manage the branch points, protections and links, and how to perform knowledge checks.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Build and manage an Electrical Components Catalog - Design a harness that is integrated within the Digital Mock-Up - Connect bundle segments to electrical components
Prerequisites	Students attending this course should be familiar with the fundamentals of CATIA V5. They should know how to use the Part Design and the Catalog Editor workbenches.
Available Online	Yes

Electrical Wire Routing (EWR)	
Course Code	CAT-en-EWR-F-V5R28
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Electrical Designers
Description	This course will teach you how to use the Electrical Wire Routing workbench to route signals and create wires using a harness and a functional definition. You will also learn how to integrate external tools with CATIA's electrical products.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Route signals - Create wires - Integrate external tools with CATIA's electrical products
Prerequisites	Students attending this course should be familiar with the fundamentals of CATIA V5. They should know how to use the Electrical Librarian and the Electrical Harness Installation workbenches.
Available Online	Yes

CATIA

CATIA Infrastructure V5

V5 Administration (ADM)	
Course Code	CAT-en-ADM-F-V5R27
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	Administrators of CATIA V5
Description	This course will teach you how to install CATIA V5 and its service packs. You will learn to use different tools to manage licenses, environments and standards. You will also learn to use tools available in batch mode and how to manage V4 and V5 data.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Install CATIA V5 and service packs - Manage CATIA licenses and environments - Manage CATIA settings and standards - Use CATIA V5 data management tools - Manage CATIA V4 data in V5 and CATIA V5 data in V4
Prerequisites	Students attending this course should be familiar with system administration.
Available Online	Yes

CATIA

CATIA Machining V5

Advanced Part Machining (AMG)	
Course Code	CAT-en-AMG-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	12 hours
Course Material	English
Level	Fundamental
Audience	NC Programmers
Description	This course teaches you how to generate high quality NC programs for machining complex 3D parts and free-form shapes using advanced machining techniques. You will learn how to perform 2.5 to 5-Axis machining operations and Axial Machining.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Define a Multi-Axis Flank Contouring operation - Define a Multi-Axis Helix Machining operation - Define a Cavities Roughing operation
Prerequisites	Students attending this course must be familiar with the NCI, PMG, SMG, and MMG workbenches.
Available Online	Yes

Multi-Axis Surface Machining (MMG)	
Course Code	CAT-en-MMG-F-V5R27
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	NC Programmers
Description	This course teaches you how to create high quality NC programs for machining complex 3D parts and free-form shapes using Multi-Axis machining techniques. The course also teaches you to define 5-Axis machining operations.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Identify and use the multi-axis surface machining workbench tools - Define various 5-axis machining operations
Prerequisites	Students attending this course should be familiar with CATIA V5 Surface Machining (SMG) Fundamentals
Available Online	Yes

Numerical Control Infrastructure (NCI)	
Course Code	CAT-en-NCI-F-V5R27
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	NC Programmers
Description	This course will teach you how to use various functionalities common across all the Machining workbenches in CATIA. It will teach you the fundamentals of creating and simulating a Manufacturing Program.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Identify and use the Manufacturing workbenches' tools - Create a Manufacturing Program - Simulate a Manufacturing Program - Manage Tools and Tool Catalogs - Compute and verify the Tool Path - Generate NC data using an integrated Post Processor - Create shop floor documentation - Manage design changes - Import V4 data
Prerequisites	Students attending this course should be familiar with CATIA V5 fundamentals
Available Online	Yes

Prismatic Machining (PMG)	
Course Code	CAT-en-PMG-F-V5R27
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	NC Programmers
Description	This course will teach you how to define and manage NC programs to machine parts using prismatic machining techniques in the prismatic machining (PMG) workbench. You will learn to create 2.5 axis milling operations. You will also learn to use the PMG functionalities to create prismatic machining and rework areas.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Define prismatic machining operations - Create a prismatic machining area and a rework area - Define and modify NC macros
Prerequisites	Students attending this course should have completed the CATIA V5 Fundamentals course and the Numerical Control Infrastructure course.
Available Online	Yes

Surface Machining (SMG)	
Course Code	CAT-en-SMG-F-V5R24
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	NC Programmers
Description	This course will teach you how to define and manage NC programs dedicated to machining parts that are designed with Surface or Solid geometry. You will learn how to define 3-Axis Roughing, Semi-finishing and Finishing operations. The course will also help you to improve productivity in mould and die machining using various functionalities of 3-Axis Surface Machining.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Define 3-Axis Surface Machining operations - Define Probing Operations - Create a Machining Area before performing the operations - Define a Rework Area - Analyze and modify the Tool Path
Prerequisites	Students attending this course should be familiar with the CATIA V5 Fundamentals course and the Numerical Control Infrastructure workbench
Available Online	Yes

CATIA

CATIA Mechanical Design V5

2D Layout for 3D Design (LO1)	
Course Code	CAT-en-LO1-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to create 2D layout views in a 3D model and use them to design the part in 3D environment.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create 2D layout views in a 3D environment - Export 2D geometry into a 3D environment - Create drawings using the 2D layout views
Prerequisites	Students attending this course should be familiar with CATIA V5 fundamentals
Available Online	Yes

3D Functional Tolerancing and Annotation (FTA)	
Course Code	CAT-en-FTA-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to annotate a 3D part. You will learn how to create annotation planes and how to add and manage 3D annotations on these planes. You will also learn how to create 3D views and use them to create 2D drawing views.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create and manage annotation planes and views - Manage and position annotations - Manage 3D geometry associated with the 3D annotations
Prerequisites	Students attending this course should be familiar with basic solid and surface creation functions and Knowledgeware.
Available Online	Yes

CATIA Composites Laser Projection Fundamentals (CLJ)	
Course Code	CAT-en-CLJ-F-V5R27
Available Release	V5-6R2017
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Composite Laser Projection Operators
Description	This course will teach you how to create laser projectors, specify the laser projection parameters and simulate the laser projection behavior. You will learn how to compute and export the laser projection data. You will also learn how to import and read the laser projection files which are exported earlier.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Define a laser projector - Compute and export laser projection data - Create additional geometries and stacking texts for export - Read the laser projection files
Prerequisites	Students attending this course should have basic Composite Design and Manufacturing skills.
Available Online	Yes

CATIA Detail Drafting (DDR)	
Course Code	CAT-en-DDR-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	16 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Draftsmen
Description	This course will teach you how to use the Drafting workbench tools to create interactive product views. You will also learn how to use advanced tools to dress-up and annotate the views. Additionally, you will learn how to customize the Drafting workbench to suit your needs.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create an interactive view and draw a sketch on it - Add annotations to dress-up the view - Use advanced dimensioning tools - Perform 2D-3D links management - Customize the Drafting workbench in accordance with your requirements
Prerequisites	Students attending this course should know how to create 2D views in CATIA V5
Available Online	Yes

CATIA Generative Drafting Fundamentals (ANSI) (GDRA)	
Course Code	CAT-en-GDRA-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Draftsmen
Description	This course will teach you how to use the Drafting workbench of CATIA V5 to create drawings. You will learn how to produce a drawing of a 3D model by creating projection and section views, and how to add basic dimensions to it.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create simple projection views and section views of 3D parts - Position the views on a drawing sheet - Add dimensions to the views - Manage the graphic properties of the drawing sheet - Finalize the drawing sheet by adding a title block
Prerequisites	Students attending this course should be familiar with the basics of CATIA V5.
Available Online	Yes

CATIA Generative Drafting Fundamentals (ISO) (GDRI)	
Course Code	CAT-en-GDRI-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Draftsmen
Description	This course will teach you how to use the Drafting workbench of CATIA V5 to create drawings. You will learn how to produce a drawing of a 3D model by creating projection and section views section views, and how to add basic dimensions to it.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create simple projection and section views of 3D parts - Position the views on a drawing sheet - Add dimensions to the views - Finalize the drawing sheet by adding a title block
Prerequisites	Students attending this course should be familiar with the basics of CATIA V5.
Available Online	Yes

CATIA Generative Sheetmetal Design (SMD)	
Course Code	CAT-de-SMD-F-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Sheetmetal Designers
Description	This course will teach you how to design a sheetmetal part using associative feature-based modeling. You will learn how to integrate both standard and user-defined stamped features into your designs and calculate the resulting flat patterns in accordance with either the standard bend allowances or your company's bend allowance tables.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Understand the terminology and the design process for creating a sheetmetal part - Define and manage the sheetmetal part parameters - Design walls, bends, and flanges - Add features such as cutouts, holes, corners, and chamfers - Create standard and user-defined stamped features - Manage folded and unfolded views and export a finished flat pattern
Prerequisites	Students attending this course should be familiar with CATIA V5 Fundamentals
Available Online	Yes

CATIA Mechanical Design V5R19 to V5-6R2014 Update (UMD94)

Course Code	CAT-de-UMD94-U-V5R24
Available Release	V5-6R2014
Duration	8 hours
Course Materials	English , German
Level	Update
Audience	Mechanical Designers
Description	The Mechanical Update course will allow you to update your skills from CATIA V5R19 to CATIA V5-6R2014 and take advantage of the new and enhanced tools in the Sketcher, Part Design and Drafting workbenches.
Objectives	<p>Upon completion of this course you will be able to take advantage of the new and enhanced tools in CATIA V5-6R2014 for the following workbenches:</p> <ul style="list-style-type: none"> - Sketcher - Part Design - Drafting
Prerequisites	Students attending this course should be familiar with the V5R19 CATIA Mechanical Design workbenches.
Available Online	Yes

CATIA Part Design (PDG)	
Course Code	CAT-en-PDG-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to use the CATIA Part Design workbench to design 3D mechanical parts from 2D sketches. You will learn how to create and modify solid features in order to prepare 3D parts for manufacturing.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Design 3D mechanical parts using basic features - Create 3D solid features based on 2D sketches - Apply Dress-Up features to the 3D parts - Duplicate and move the 3D features - Modify a 3D part
Prerequisites	Students attending this course must have completed the CATIA V5 Fundamentals and CATIA Sketcher courses.
Available Online	Yes

CATIA Part Design Added Exercises (PDG)

Course Code	CAT-en-PDG-X-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	12 hours
Course Materials	English , French , German , Japanese
Level	Exercise
Audience	Mechanical Designers
Description	This course provides you with an exercise database for additional practice on CATIA Part Design. The exercises have been arranged in increasing order of difficulty. The fundamental exercises will check and refresh your basic Part Design skills before you move on to more complex topics. The advanced exercises will make you practice recommended design methodologies using realistic parts.
Objectives	These exercises will allow you to put your Mechanical skills into practice on selected scenarios. You will apply the recommended methodology in various situations and thus enhance your understanding and usage of the Mechanical workbenches.
Prerequisites	Students attending this course must have completed the CATIA Part Design and CATIA Knowledge Fundamentals courses.
Available Online	Yes

CATIA Part Design Expert (PDG)	
Course Code	CAT-en-PDG-A-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	12 hours
Course Materials	English , French , German , Japanese
Level	Advanced
Audience	Mechanical Designers
Description	This course will teach you how to design complex 3D mechanical parts using the Boolean approach. You will learn how to work in a Multi-Model Environment and maintain links between your 3D models. You will also learn how to analyze your designs in order to optimize them.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create a part using 3D reference elements - Create advanced Sketch-Based Features - Apply advanced Dress-Up Features - Design 3D parts using Boolean operations - Work in a Multi-Model Environment and share your designs with others - Analyze parts and optimize them - Annotate the parts for review
Prerequisites	Students attending this course should have completed the CATIA V5 Fundamentals, Getting started with CATIA V5, CATIA Sketcher, and CATIA Part Design Fundamentals courses.
Available Online	Yes

CATIA Product Design (ASM)	
Course Code	CAT-en-ASM-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to create a simple product structure and how to add existing components and position them correctly. You will learn how to add new parts and design them in the context of a product. You will also learn how to analyze assemblies and ensure design coherence.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create a new product and add components to it - Move the components within a product by positioning them using assembly constraints - Modify an existing product structure - Design new parts in the context of a product - Check the mechanical properties of a product and analyze its degrees of freedom - Analyze interferences between parts and perform measurements
Prerequisites	Students attending this course should be familiar with CATIA Part Design
Available Online	Yes

CATIA Product Design Added Exercises (ASM)	
Course Code	CAT-en-ASM-X-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Exercise
Audience	Mechanical Designers
Description	This course provides you with additional exercises to practice the concepts that you have learnt in the CATIA Product Design course. These exercises represent typical industrial scenarios and demonstrate how CATIA Product Design helps you to achieve your design objectives.
Objectives	These exercises will allow you to put your Mechanical skills into practice on selected scenarios. You will apply the recommended methodology in various situations and thus enhance your understanding and usage of the Mechanical workbenches.
Prerequisites	Students attending this course should have attended the CATIA Product Design course and the CATIA Product Design Expert course
Available Online	Yes

CATIA Product Design Expert (ASM)	
Course Code	CAT-en-ASM-A-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	16 hours
Course Materials	English , French , German , Japanese
Level	Advanced
Audience	Mechanical Designers
Description	This course will teach you how to design parts in the context of a complex product structure using collaborative engineering methods. You will learn how to optimize CATIA's performance when working with large and complex designs. You will also learn how to generate annotations and bills of material for your assembly drawings.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Optimize performance for large and complex designs - Manage contextual links between product documents using publications - Create and use parameters to drive a product design - Create sections to visualize the internal product structure - Create scenes and explode views of a product - Generate annotations and bills of material for assembly drawings
Prerequisites	Students attending this course should be familiar with CATIA Product Design and CATIA Part Design
Available Online	Yes

CATIA Sketcher (SKE)	
Course Code	CAT-en-SKE-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to use the CATIA Sketcher workbench. You will learn how to create two-dimensional sketches by drawing and constraining the various geometric elements. You will also learn how to analyze the sketches and edit them.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Work in the CATIA Sketcher environment - Create 2D sketch geometry - Analyze the sketched geometry - Edit existing 2D profiles - Dimension the sketch and modify it using constraints - Manage sketches within a 3D environment
Prerequisites	Students attending this course must have completed the CATIA V5 Fundamentals course
Available Online	Yes

CATIA Surface Design (GS1)	
Course Code	CAT-en-GS1-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Mechanical Surface Designers
Description	This course will teach you how to use the Generative Shape Design tools. You will learn how to create wireframes and surfaces. You will also learn about the concept of hybrid design and how to use it while creating wireframes and surfaces. This course covers only those Generative Shape Design tools that are available with a MD2 license.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create a clean topology from a set of surfaces and smooth sharp edges - Detect and correct the discontinuities on curves and surfaces - Create solids from surfaces
Prerequisites	Students attending this course should be familiar with CATIA V5 Fundamentals.
Available Online	Yes

CATIA Surface Design Added Exercises (GS1)	
Course Code	CAT-en-GS1-X-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Exercise
Audience	Mechanical Surface Designers
Description	This course provides you with an exercise database for additional practice on CATIA Surface Design. The exercises have been created based on Industry practices. You will get to practice skills such as creating wireframes and surfaces, creating surfacic shells and solid parts, and working with multiple parts that are referencing a common part.
Objectives	These exercises will allow you to put your Shape skills into practice on selected scenarios. You will apply the recommended methodology in various situations. You will enhance your understanding and usage of the Shape workbenches.
Prerequisites	Students attending this course should be familiar with CATIA V5 Surface Design.
Available Online	Yes

CATIA Tools For Proficient Users (PRO)	
Course Code	CAT-en-PRO-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Advanced CATIA V5 Users
Description	This course will teach you how to use advanced CATIA functions such as Catalog Edition, Powercopy Management, and User Defined Feature Management.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create advanced replication features like Power Copies - Store components and Power Copies into a catalog and reuse them in a new context - Analyze and migrate CATIA V4 models to CATIA V5
Prerequisites	Students attending this course should be familiar with CATIA Fundamentals and CATIA Part Design
Available Online	Yes

CATIA V5-6R2014 to V5-6R2016 Update for Mechanical Designers (UMD46)

Course Code	CAT-en-UMD46-U-V5R26
Available Release	V5-6R2016
Duration	6 hours
Course Materials	English , German
Level	Update
Audience	Mechanical Designers
Description	This course will teach you how to use the enhanced functionalities in CATIA V5-6R2016. You will see how to preview fillets and chamfers in the Sketcher workbench and how to limit a tritangent fillet using multiple elements in the Part Design workbench. You will also see how to create a Moving Datum Target annotation in the Drafting workbench.
Objectives	<p>Upon completion of this course you will be able to take advantage of the new and enhanced tools in CATIA V5-6R2016 for the following workbenches:</p> <ul style="list-style-type: none"> - Sketcher - Part Design - Drafting - Generative Wireframe and Surface
Prerequisites	Student attending this course should be familiar with the V5-6R2014 CATIA Mechanical Design workbenches.
Available Online	Yes

CATIA V5-6R2014 Update for Designers (UMSD24)	
Course Code	CAT-de-UMSD24-U-V5R24
Available Release	V5-6R2014
Duration	4 hours
Course Materials	English , French , German , Japanese
Level	Update
Audience	Mechanical Designers
Description	This course will teach you how to use the enhanced functionalities in the CATIA V5-6R2014. You will see how to align different points in the Sketcher workbench and how to create mirrored extent in rectangular patterns in the Part Design workbench. You will also see a different way to create the helix in the Generative Shape Design workbench and how to align 2 views in the Drafting workbench.
Objectives	<p>Upon completion of this course you will be able to take advantage of the new and enhanced tools in CATIA V5-6R2014 for the following workbenches:</p> <ul style="list-style-type: none"> - Sketcher - Part Design - Generative Shape Design - Drafting
Prerequisites	Student attending this course should be familiar with the V5-6R2013 CATIA Mechanical Design workbenches.
Available Online	Yes

CATIA V5-6R2015 Update for Mechanical Designers (UMD25)	
Course Code	CAT-en-UMD25-U-V5R25
Available Release	V5-6R2015
Duration	3 hours
Course Materials	English , French , German , Japanese
Level	Update
Audience	Mechanical Designers
Description	This course will teach you how to use the enhanced functionalities in CATIA V5-6R2015. You will see how to create a polygon in the Sketcher workbench and how to create pattern instances at points and axis systems in the Part Design workbench. You will also see how to manage dimension systems in the Drafting workbench.
Objectives	<p>Upon completion of this course you will be able to take advantage of the new and enhanced tools in CATIA V5-6R2015 for the following workbenches:</p> <ul style="list-style-type: none"> - Sketcher - Part Design - Drafting
Prerequisites	Student attending this course should be familiar with the V5-6R2014 CATIA Mechanical Design workbenches.
Available Online	Yes

CATIA V5-6R2016 Update for Mechanical Designers (UMD26)

Course Code	CAT-en-UMD26-U-V5R26
Available Release	V5-6R2016
Duration	6 hours
Course Materials	English , French , German , Japanese
Level	Update
Audience	Mechanical Designers
Description	This course will teach you how to use the enhanced functionalities in CATIA V5-6R2016. You will see how to preview fillets and chamfers in the Sketcher workbench and how to limit a tritangent fillet using multiple elements in the Part Design workbench. You will also see how to create a Moving Datum Target annotation in the Drafting workbench.
Objectives	Upon completion of this course you will be able to take advantage of the new and enhanced tools in CATIA V5-6R2016 for the following workbenches: <ul style="list-style-type: none"> - Sketcher - Part Design - Drafting - Generative Wireframe and Surface
Prerequisites	Student attending this course should be familiar with the V5-6R2015 CATIA Mechanical Design workbenches.
Available Online	Yes

CATIA V5-6R2017 Update for Mechanical Designers (UMD27)	
Course Code	CAT-en-UMD27-U-V5R27
Available Release	V5-6R2017
Duration	2 hours
Course Materials	English , French , German
Level	Update
Audience	Mechanical Designers
Description	Upon completion of this course you will be able to effectively use the new and enhanced tools in CATIA V5-6R2017 for the Sketcher, Part Design, Drafting and Generative Shape Design workbenches.
Objectives	<p>Upon completion of this course you will be able to effectively use the new and enhanced tools in CATIA V5-6R2017 for the following workbenches.</p> <ul style="list-style-type: none"> - Sketcher - Part Design - Drafting - Generative Shape Design
Prerequisites	Students attending this course should be familiar with the V5-6R2016 CATIA Mechanical Design workbenches.
Available Online	Yes

CATIA V5-6R2018 Update for Mechanical and Shape Designers (UMSD28)

Course Code	CAT-en-UMSD28-U-V5R28
Available Release	V5-6R2018
Duration	2 hours
Course Material	English
Level	Update
Audience	Mechanical Designers
Description	<p>This course will teach you how to use the enhanced functionalities in CATIA V5-6R2018. You will learn to use the Boolean operations to modify the sketch in the Sketcher workbench and create a chamfer with a hold curve in the Part Design workbench. You will also learn how to add a plane between two reference elements in the Generative Shape Design workbench and add various information about the welding in the Drafting workbench.</p>
Objectives	<p>Upon completion of this course you will be able to effectively use the new and enhanced tools in CATIA V5-6R2018 for the following workbenches:</p> <ul style="list-style-type: none"> - Sketcher - Part Design - Generative Shape Design - Drafting
Prerequisites	<p>Students attending this course should be familiar with the V5-6R2017 CATIA Mechanical Design workbenches.</p>
Available Online	Yes

CATIA V5 Design Essentials	
Course Code	CAT-en-SV5-F-5-27
Available Releases	V5-6R2016 , V5-6R2017
Duration	44 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	<p>Welcome to the CATIA V5 Part Design Essentials course. 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.</p>
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - - Create a new part - - 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

CATIA V5 Design Essentials

Prerequisites

Students attending this course should be familiar with Mechanical Design.

Available Online

Yes

CATIA V5 Foundations for Aerospace Assembly Designers (V5AeA)

Course Code	CAT-en-V5AeA-F-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	Aerospace Structure Designers
Description	This course will introduce you to CATIA V5 assembly design. It will teach you how to manage assembly configurations and how to design and position components within the assembly. In addition you will learn how to create a structured assembly in order to best design parts in an assembly context and how to control and manage the links created between the assembly components.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Understand the terminology used in assembly design - Design structural parts in the context of an assembly - Constrain assembly components - Analyze an assembly - Annotate an assembly
Prerequisites	V5 Foundations for Aerospace Part Designers
Available Online	Yes

CATIA V5 Foundations for Aerospace Part Designers (V5AeD)

Course Code	CAT-en-V5AeD-F-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	40 hours
Course Material	English
Level	Fundamental
Audience	Aerospace Part Designers
Description	This course will introduce you to CATIA V5. It will teach you how to create simple models from 2D sketches, and then the correct techniques for the creation and annotation of complex solid models. It will introduce you to surface design and the concepts of part design in the context of an assembly.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Identify the appropriate CATIA V5 tools used for part design. - Plan the construction of a complex part in order to properly convey its visual and functional aspects. - Annotate parts. - Design simple surface parts. - Modify a part within the context of an assembly.
Prerequisites	Students attending this course should be familiar with Mechanical Design and the Windows Operating System
Available Online	Yes

CATIA V5 Foundations for Aerospace Part Reviewers (V5AeR)

Course Code	CAT-en-V5AeR-F-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Aerospace Part Reviewers
Description	This course will explain how to use CATIA V5 workbenches. It will teach you how to review an existing part by verifying its properties, its coordinates and measurements, and how to add annotations to the parts.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Measure a part with respect to a pre-defined axis system. - Annotate an existing part. - Differentiate between parts and assemblies.
Prerequisites	A Student attending this course should be familiar with Mechanical Design and the Windows Operating System.
Available Online	Yes

CATIA V5 Foundations for Body Designers (V5VB)	
Course Code	CAT-en-V5VB-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	56 hours
Course Materials	English , French , German
Level	Fundamental
Audience	Automotive Body Designers
Description	This course will teach you how to use the fundamental concepts in CATIA V5 to build simple automotive parts and assemblies, and make simple drawings of those parts and assemblies. You will also learn the correct solid and surface modeling methodology necessary for body design.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Describe the CATIA V5 interface - Apply the correct solid and surface modeling methodology for body design - Create an automobile part in order to satisfy its design intent - Design and manage parts in the context of an assembly - Produce simple drawings and assembly layouts
Prerequisites	Students attending this course should be familiar with the fundamentals of Mechanical and Surface Design
Available Online	Yes

CATIA V5 Foundations for Chassis Designers (V5VC)

Course Code	CAT-en-V5VC-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	56 hours
Course Materials	English , French , German
Level	Fundamental
Audience	Automotive Chassis Designers
Description	This course will introduce the fundamental concepts in CATIA V5 that are required to build simple automotive parts and assemblies in CATIA, and how to make simple drawings of those parts and assemblies. It will introduce you to the correct solid and surface modeling methodology necessary for chassis design.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Describe the CATIA V5 interface - Apply the correct solid and surface modeling methodology for Chassis design - Create an automobile part in order to satisfy its design intent - Apply advanced solid modeling technique necessary for Chassis design methodology - Design and manage parts in the context of an assembly - Produce simple drawings and assembly layouts
Prerequisites	Students attending this course should know the basics of Mechanical and Surface Design
Available Online	Yes

CATIA V5 Foundations for Powertrain Designers (V5VP)	
Course Code	CAT-en-V5VP-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	56 hours
Course Materials	English , French , German
Level	Fundamental
Audience	Automotive Powertrain Designers
Description	This course will teach you to use the fundamental concepts in CATIA V5 to build simple automotive parts and assemblies, and make simple drawings of those parts and assemblies. You will also learn how to use the advanced solid modeling techniques necessary for Powertrain design methodology.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Describe the CATIA V5 interface - Apply the correct solid and surface modeling methodology for Powertrain Design - Create an automobile part in order to satisfy its design intent - Apply advanced solid modeling technique necessary for Powertrain design methodology - Design and manage parts in the context of an assembly - Produce simple drawings and assembly layouts
Prerequisites	Students attending this course should know the fundamentals of Mechanical Design
Available Online	Yes

CATIA V5 Fundamentals (V5F)	
Course Code	CAT-en-V5F-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	40 hours
Course Materials	Chinese , English , French , German , Japanese
Level	Fundamental
Audience	Mechanical Designers with no CATIA V5 experience
Description	This course will teach you about CATIA V5. You will learn how to build simple parts and assemblies in CATIA, and how to make simple drawings of those parts and assemblies.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Understand and use the CATIA V5 interface - Plan the construction of a part in order to convey its visual and functional aspects - Create simple parts in CATIA V5 - Construct an assembly using the parts - Produce simple drawings and assembly layouts
Prerequisites	Students attending this course should be familiar with Mechanical Design and the Windows Operating System.
Available Online	Yes

CATIA V5 Mechanical Design Expert (V5E)	
Course Code	CAT-en-V5E-A-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	40 hours
Course Materials	Chinese , English , French , German , Japanese
Level	Advanced
Audience	Mechanical Designers
Description	This course will teach you how to start a complex design project from its specifications (top down approach) and complete it by reusing existing data. It will focus on advanced skills and concepts that enable you to create and analyze complex parts and assemblies.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create a complex model in CATIA V5 - Create and manage a structured model - Design parts in the context of an assembly - Re-use existing data to complete assemblies - Manage relationships between assembled parts - Analyze and annotate your design
Prerequisites	Students attending this course should be familiar with the basics of CATIA V5 Mechanical Design
Available Online	Yes

Composites Grid Approach (CPG)	
Course Code	CAT-en-CPG-F-V5R27
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	Composites Designers for Aerospace
Description	This course will introduce you to the Grid approach. You will learn how to generate plies, modify geometry, and create a solid or a top surface using the ply geometry. By the end of this course you will be able to create and modify a composite part using the Composites Grid Design approach.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Understand the concept of Grid approach in Composites Design - 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
Prerequisites	Students attending this course should be familiar with wireframe and surface design, drafting, and composites part design.
Available Online	Yes

Composites Part Engineering (CPE)	
Course Code	CAT-en-CPE-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	Composites Designers
Description	This course will first teach you how to design simple Composites Parts using a Manual approach. You will then learn how to use a Zone-based approach to complete the preliminary design and then the detailed design. You will also learn how to generate plies automatically, use the analysis tools and simulate fiber behavior. Finally, you will learn how generate exact solids and create composites drawings.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Define the composites parameters - Design a composite part using the manual approach - Design a composite part using the classical and solid zone approach - Perform and inspect the producibility analysis - Export and import the ply design data - Create a ply book
Prerequisites	Students attending this course should be familiar with Part Design, Assembly Design, Wireframe and Surface Design and Drafting.
Available Online	Yes

Composites Part Manufacturing (CPM)	
Course Code	CAT-en-CPM-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	20 hours
Course Material	English
Level	Fundamental
Audience	Composites Manufacturing Designers
Description	This course will teach you how to create a manufacturing document from a Composites Engineering design document. You will learn how to modify the Manufacturing Data structure and synchronize the link between the engineering and the manufacturing data. You will also learn how to apply the manufacturing and producibility constraints in the Composites Design process.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Design a Composite Part using the Manual Approach - Generate a Manufacturing Stacking from an Engineering Stacking - Synchronize the link between the Manufacturing and Engineering parts - Perform and inspect the Producibility Analysis - Compute and optimize a Flattening - Export the Ply Data - Create a Ply Book
Prerequisites	Students attending this course Must have knowledge of part design, assembly design, surface Design and drafting.

Composites Part Manufacturing (CPM)

Available Online

Yes

Functional Molded Parts (FMP)	
Course Code	CAT-en-FMP-F-V5R27
Available Release	V5-6R2017
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Molded Part Designers
Description	This course will teach you how to use the Functional Molded Part workbench to create molded parts using basic features and to finalize the part using additional dress-up features. You will also be taught the multi-body approach and will finally learn how to extract the cores and cavity from the final part.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create the main shapes of a molded part - Add functional features to the main part - Finalize the molded part using feature modifiers - Use the multi-body approach - Extract cores and cavities from the final part
Prerequisites	CATIA V5 Fundamentals
Available Online	Yes

Getting Started with CATIA V5 (COM)	
Course Code	CAT-en-COM-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	4 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	New CATIA V5 Users
Description	This course will teach you how to start working in CATIA V5. You will learn how to perform basic operations using the standard user interface elements and tools. You will also learn about graphic properties and how to use the basic visualization techniques to view objects in CATIA V5.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Open CATIA V5 documents and use basic tools to modify them - Use the specification tree to browse and understand the structure of an object - Use the compass to manipulate the viewpoint - View and modify the graphic properties of an object
Prerequisites	None
Available Online	Yes

CATIA

CATIA PLM Express V5

CATIA PLM Express Fundamentals (CTP)	
Course Code	CAT-en-CTP-F-V5R27
Available Release	V5-6R2017
Duration	40 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers with no CATIA V5 experience.
Description	This course will teach you how to use CATIA Team PLM configuration workbenches to build simple parts and assemblies. You will learn how to make simple drawings of those parts and assemblies. You will also learn about basic Wireframe and Surface creation.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Understand and use the CATIA V5 interface. - Plan the construction of a part in order to convey its visual and functional aspects. - Create simple parts in CATIA V5. - Construct an assembly managing the parts. - Produce simple drawings and assembly layouts.
Prerequisites	Students attending this course should be familiar with Mechanical Design and the Windows operating system
Available Online	Yes

CATIA PLM Express Fundamentals - Basic Surface (CTPB)	
Course Code	CAT-en-CTPB-F-V5R27
Available Release	V5-6R2017
Duration	40 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers with no CATIA V5 experience.
Description	This course will teach you how to use the CATIA Team PLM configuration workbench to build simple parts and assemblies in CATIA, and make simple drawings of those parts and assemblies. You will also learn how to create the basic wireframe and surface.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Understand and use 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 including basic surface geometry - Construct an assembly managing the parts - Produce simple drawings and assembly layouts
Prerequisites	Students attending this course should be familiar with Mechanical Design and the Windows operating system
Available Online	Yes

CATIA PLM Express Fundamentals - Surfaces (CTPS)	
Course Code	CAT-en-CTPS-F-V5R27
Available Release	V5-6R2017
Duration	40 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers with no CATIA V5 experience.
Description	This course will teach you how to use the CATIA Team PLM configuration workbench to build simple parts and assemblies in CATIA, and make simple drawings of those parts and assemblies. You will also learn how to create the basic wireframe and surface.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Understand and use 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 including basic surface geometry - Construct an assembly managing the parts - Produce simple drawings and assembly layouts
Prerequisites	Students attending this course should have knowledge of Mechanical Design and Windows operating system.
Available Online	Yes

CATIA

CATIA Product Synthesis V5

CATIA Knowledge Fundamentals (KWF)	
Course Code	CAT-en-KWF-F-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	CATIA V5 Users
Description	This course will teach you how to embed knowledge within design and leverage it to automate modifications. You will learn how to create and use parametric parts and assemblies.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Use and manage the Knowledgeware working environment - Understand how collaborative work affects knowledge features - Use parameters, formulas and design tables - Create parametric parts and assemblies - Share parameters and reuse relations
Prerequisites	Students attending this course should be familiar with CATIA V5 Part Design and Assembly Design
Available Online	Yes

Knowledge Advisor (KWA)	
Course Code	CAT-en-KWA-F-V5R24
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	CAD Engineers
Description	This course will teach you how to embed knowledge in your designs using Knowledge Advisor tools. You will also learn how to leverage the knowledge to reduce errors and automate the design modifications.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create and use User Parameters and Formulae - Create Rules, Checks and Reactions to control the design - Create and use Design Tables to automate the design modifications - Use miscellaneous Knowledge Advisor tools
Prerequisites	Students attending this course should be familiar with the basics of CATIA V5.
Available Online	Yes

Knowledge Expert (KWE)	
Course Code	CAT-en-KWE-F-V5R24
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Mechanical and Electrical Design Engineers
Description	This course will show you how to build up and share corporate knowledge stored in rule bases, and leverage it across the company to ensure design compliance with established standards.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Embed the complex design knowledge in a parametric part using Knowledgeware Expert. - Automate design modifications using the specific Knowledge Expert tools
Prerequisites	Students attending this course should be familiar with the basics of CATIA V5 and knowledgeware.
Available Online	Yes

Product Engineering Optimizer (PEO)	
Course Code	CAT-en-PEO-F-V5R26
Available Release	V5-6R2016
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	CAD Engineers, Analysts
Description	This course will teach you how to use the Product Engineering Optimization workbench to optimize your designs by formulating and solving an optimization problem, considering the constraints and conditions involved in the problem.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Formulate an optimization problem - Select the appropriate algorithms to solve an optimization problem - Analyze the results of optimization
Prerequisites	Students attending this course should be familiar with the basics of CATIA V5.
Available Online	Yes

Product Knowledge Template (PKT)	
Course Code	CAT-en-PKT-F-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	CAD Engineers and Knowledge Engineers
Description	This course will teach you how to create and store interactive features and then reuse and adapt them to a new context.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create and reuse Power Copies and User Defined Features - Create and reuse advanced instantiation features like Knowledge Patterns - Create Part and Assembly Templates and reuse them in a new context
Prerequisites	Students attending this course should be familiar with the basics of CATIA V5 and knowledgeware.
Available Online	Yes

CATIA

CATIA Shape Design and Styling V5

CATIA Generative Shape Design V5R19 to V5-6R2014 Update (UHD94)	
Course Code	CAT-en-UHD94-U-V5R24
Available Release	V5-6R2014
Duration	8 hours
Course Materials	English , German
Level	Update
Audience	Surface Designers
Description	The Surface Update course will allow you to update your skills from CATIA V5R19 to CATIA V5-6R2014 and take advantage of the new tools and enhancements that are available in the Generative Shape Design workbench.
Objectives	<p>Upon completion of this course you will be able to take advantage of the new and enhanced tools in CATIA V5-6R2014 for the Generative Shape Design workbench for the following domains:</p> <ul style="list-style-type: none"> - Wireframe creation - Curve creation - Surface Creation - Surface Topology
Prerequisites	Students attending this course should be familiar with the V5R19 CATIA Generative Shape Design workbench.
Available Online	Yes

CATIA ICEM Shape Design Expert (IEX5)	
Course Code	CAT-en-IEX5-A-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	12 hours
Course Material	English
Level	Advanced
Audience	Surface Designers who are required to create high-quality surfaces
Description	This course will teach you how to use the advanced surface creation options, the advanced analysis tools, and the Expert tools of CATIA V5 Icem Shape Design. You will learn how to create high-quality surfaces, and analyze and improve the quality of the surfaces.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create high quality surfaces - Analyze surface quality - Correct surface defects
Prerequisites	<ul style="list-style-type: none"> - Students attending this course should be familiar with CATIA V5 Fundamentals and CATIA Icem Shape Design Fundamentals. - CATIA V5 for Surfaces is also recommended.
Available Online	Yes

CATIA ICEM Shape Design Fundamentals (ICM)	
Course Code	CAT-en-ICM-F-V5R24
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	40 hours
Course Material	English
Level	Fundamental
Audience	High quality surface designers
Description	This course will teach you how to use the ISD workbench to create good quality curves and Class A surfaces. You will learn how to analyze the wireframe and surface quality and interpret the results in order to correct visual defects.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create robust class A surface models - Create good quality curves - Assemble, re-limit and connect the surfaces - Analyze surface quality - Correct surface defects - Manage surface models
Prerequisites	<ul style="list-style-type: none"> - CATIA V5 Fundamentals - Some knowledge of Mechanical Surface Design is advisable
Available Online	Yes

CATIA Imagine and Shape (IMA)	
Course Code	CAT-en-IMA-F-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Product Stylists, Industrial Designers
Description	This course will teach you how to use the Imagine and Shape workbench in CATIA to create new product shapes. You will also learn how to improve product styles.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create and modify curves - Create subdivision surfaces using tools specific to the Imagine and Shape workbench - Modify product style surfaces
Prerequisites	Students attending this course should know the CATIA Generative Shape Design workbench.
Available Online	Yes

CATIA Surface Design Expert (GSD)	
Course Code	CAT-en-GSD-A-V5R27
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	16 hours
Course Materials	English , French , German , Japanese
Level	Advanced
Audience	Mechanical Designers and Surface Designers
Description	This course will first recall and summarize the tools taught in the Surface Design course. It will then capitalize on this knowledge and teach you advanced surface creation tools, quality checking and correction techniques, and surface creation in a multi-model environment. This course covers only those Generative Shape Design tools that are specific to the HD2 license.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create wireframe features using existing curves and surfaces - Create advanced and parameterized swept surfaces - Analyze advanced surfaces and fill gaps - Create advanced blend features - Improve the quality and stability of existing geometries
Prerequisites	Students attending this course should have completed the CATIA Surface Design course
Available Online	Yes

CATIA Surface Design Expert Added Exercises (GSD)	
Course Code	CAT-en-GSD-X-V5R26
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	16 hours
Course Materials	English , French , German , Japanese
Level	Exercise
Audience	Mechanical Designers and Surface Designers
Description	This course provides you with an extensive database of exercises for additional practice on advanced topics of CATIA Surface Design. The exercises have been created based on Industry practices.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create wireframe features using existing curves and surfaces - Create advanced and parameterized swept surfaces - Perform advanced surface analysis and gap correction - Create advanced blend features - Improve the quality and stability of create geometries
Prerequisites	Students attending this course should have attended the CATIA Surface Design Expert course
Available Online	Yes

CATIA V5-6R2014 to V5-6R2016 Update for Shape Designers (UHD46)	
Course Code	CAT-en-UHD46-U-V5R26
Available Release	V5-6R2016
Duration	4 hours
Course Materials	English , German
Level	Update
Audience	Surface Designers
Description	This course will teach you how to use the enhanced functionalities in CATIA V5-6R2016. You will learn different ways to create axis systems. You will also learn enhanced methods to extrapolate curves and extract surfaces.
Objectives	<p>Upon completion of this course you will be able to take advantage of the new and enhanced tools in V5-6R2016 in the Generative Shape Design workbench for the following domains:</p> <ul style="list-style-type: none"> - Reference Geometry Creation - Curve Creation - Surface Creation
Prerequisites	Students attending this course should be familiar with the CATIA Generative Shape Design V5-6R2014 workbench.
Available Online	Yes

CATIA V5-6R2015 Update for Shape Designers (UHD25)	
Course Code	CAT-en-UHD25-U-V5R25
Available Release	V5-6R2015
Duration	1 hours
Course Materials	English , French , German , Japanese
Level	Update
Audience	Shape Designers
Description	This course will teach you how to use the enhanced functionalities in CATIA V5-6R2015. You will see different ways to extract surfaces maintaining associativity in the Generative Shape Design workbench.
Objectives	Upon completion of this course you will be able to take advantage of the new and enhanced tools in CATIA V5-6R2015 for the following workbench: <ul style="list-style-type: none"> - Generative Shape Design
Prerequisites	Student attending this course should be familiar with the V5-6R2014 CATIA Generative Shape Design workbench.
Available Online	Yes

CATIA V5-6R2016 Update for Shape Designers (UHD26)	
Course Code	CAT-en-UHD26-U-V5R26
Available Release	V5-6R2016
Duration	4 hours
Course Materials	English , French , German , Japanese
Level	Update
Audience	Surface Designers
Description	This course will teach you how to use the enhanced functionalities in CATIA V5-6R2016. You will learn different ways to create axis systems. You will also learn enhanced methods to extrapolate curves and surfaces.
Objectives	<p>Upon completion of this course you will be able to take advantage of the new and enhanced tools in V5-6R2016 in the Generative Shape Design workbench for the following domains:</p> <ul style="list-style-type: none"> - Reference Geometry Creation - Curve Creation - Surface Creation
Prerequisites	Students attending this course should be familiar with the CATIA Generative Shape Design V5-6R2015 workbench.
Available Online	Yes

CATIA V5-6R2017 Update for Shape Designers (UHD27)	
Course Code	CAT-en-UHD27-U-V5R27
Available Release	V5-6R2017
Duration	2 hours
Course Materials	English , French , German
Level	Update
Audience	Surface Designers
Description	Upon completion of this course, you will be able to effectively use the new and enhanced tools in CATIA V5-6R2017 for the Generative Shape Design workbench.
Objectives	<p>Upon completion of this course, you will be able to effectively use the new and enhanced tools in CATIA V5-6R2017 for the following workbenches.</p> <ul style="list-style-type: none"> - Reference Geometry Creation - Surface Creation - Generative Shape Design
Prerequisites	Students attending this course should be familiar with the CATIA Generative Shape Design V5-6R2016 workbench.
Available Online	Yes

CATIA V5 for Surfaces (V5S)	
Course Code	CAT-de-V5S-F-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	24 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Surface Designers
Description	This course will teach you how to create curves and surfaces using the Generative Shape Design workbench. You will learn how to analyze the wireframe and surface quality and rectify the defects. You will also learn how to work in a multi-model environment with published surfaces.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Use the tools of the Generative Shape Design workbench - Create good quality curves based on a sound and improved wireframe geometry - Assemble, relimit and connect the surfaces to get a topology - Analyze the quality of surfaces and rectify the defects - Manage the surfaces in a multi-model environment
Prerequisites	Students attending this course should be familiar with the fundamentals of CATIA V5.
Available Online	Yes

Developed Shapes (DL1)	
Course Code	CAT-en-DL1-F-V5R27
Available Release	V5-6R2017
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Surface Designers
Description	This course will teach you how to use CATIA Developed Shape functionalities to create unfolded surfaces from a ruled surface. You will learn how to develop wires and points onto a revolution surface.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create unfolded surfaces from a ruled surface using the CATIA Developed Shapes functionalities - Develop wires and points onto a revolution surface
Prerequisites	Students attending this course should be familiar with the fundamentals of CATIA and Generative Surface Design
Available Online	Yes

FreeStyle Shaper, Optimizer and Profiler (FSS)	
Course Code	CAT-en-FSS-F-V5R25
Available Releases	V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017
Duration	12 hours
Course Material	English
Level	Fundamental
Audience	Surface Designers
Description	This course will teach you how to create flawless, styled shapes from scratch using three-dimensional free-form curves and surfaces or using digitized data. You will also learn how to analyze and improve the quality of existing curves and surfaces.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create styled shapes using digitized data - Create surfaces using a curve-based approach - Create surfaces using a surface-based approach - Analyze and correct the curve quality - Analyze and correct the surface quality
Prerequisites	Students attending this course should know Surface Design in CATIA V5.
Available Online	Yes

Freestyle Sketch Tracer (FSK)	
Course Code	CAT-en-FSK-F-V5R27
Available Release	V5-6R2017
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Shape Designers
Description	This course will teach you how to import images in the CATIA V5 environment and use them as a background or a basis for your design.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Import an image into CATIA V5 - Position the image in the CATIA V5 environment - Use the image as a background or as a basis for the design
Prerequisites	Students attending this course should know the basics of CATIA V5.
Available Online	Yes

Realistic Shape Optimizer (RSO)	
Course Code	CAT-en-RSO-F-V5R27
Available Release	V5-6R2017
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Surface designers, Tooling designers
Description	This course addresses the needs of Reverse Shape Optimizers. It will teach you how to create a deformation law with the CATIA R27 platform. You will learn how to use the deformation law in Digitized Morphing for Curve and Solids. It will teach you about optimization of a vector field and also how to filter a vector field to check quality of vectors.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Morph surfaces with a computed deformation field - Optimize vectors field from deviation analysis - Filter vectors field to check the quality of the vectors
Prerequisites	Students attending this course should be familiar with the basics of wireframe and surfaces creation.
Available Online	Yes