

# Course Catalog

V5-6R2014 to V5-6R2017 Multiple Languages

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# CATIA

## CATIA Analysis V5

| 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"> <li>- Define and customize the material properties of the parts to be analyzed</li> <li>- Apply pressure, acceleration, and force density loads</li> <li>- Define virtual parts to simplify the analysis</li> <li>- Apply pivot, ball-joint, and user-defined restraints</li> <li>- Compute the frequency analysis for a single part</li> <li>- Create planar sections to visualize the internal result values</li> <li>- Compute and refine a mesh using adaptive meshing in order to achieve the pre-defined accuracy</li> </ul> |
| 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"> <li>- Understand why, when, and how to use Finite Element Analysis</li> <li>- Use different element types and shapes to mesh a part</li> <li>- Apply clamp, slider, and iso-static restraints</li> <li>- Apply force, moment, and displacement loads</li> <li>- Compute the static analysis for a single part</li> <li>- Visualize the images of the analysis results and produce the analysis reports</li> <li>- Refine existing meshes to produce more accurate results</li> </ul> |
| Prerequisites  | Students attending this course should be familiar with the fundamentals of CATIA V5  |
| Available Online                                       | Yes  |



# CATIA

## CATIA Infrastructure V5

| V5 Administration (ADM) |   |
|-------------------------|---|
| Course Code             | CAT-ja-ADM-F-V5R24  |
| Available Releases      | V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017   |
| Duration                | 24 hours  |
| Course Materials        | English , Japanese  |
| 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"> <li>- Install CATIA V5 and service packs</li> <li>- Manage CATIA licenses and environments</li> <li>- Manage CATIA settings and standards</li> <li>- Use CATIA V5 data management tools</li> <li>- Manage CATIA V4 data in V5, and CATIA V5 data in V4</li> </ul> |
| Prerequisites           | Students attending this course should be familiar with system administration.   |
| Available Online        | Yes   |

# CATIA

## CATIA Mechanical Design V5

| 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"> <li>- Create an interactive view and draw a sketch on it</li> <li>- Add annotations to dress-up the view</li> <li>- Use advanced dimensioning tools</li> <li>- Perform 2D-3D links management</li> <li>- Customize the Drafting workbench in accordance with your requirements</li> </ul> |
| 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"> <li>- Create simple projection views and section views of 3D parts</li> <li>- Position the views on a drawing sheet</li> <li>- Add dimensions to the views</li> <li>- Manage the graphic properties of the drawing sheet</li> <li>- Finalize the drawing sheet by adding a title block</li> </ul> |
| 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-V5R27   |
| 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"> <li>- Create simple projection and section views of 3D parts</li> <li>- Position the views on a drawing sheet</li> <li>- Add dimensions to the views</li> <li>- Finalize the drawing sheet by adding a title block</li> </ul> |
| 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"> <li>- Understand the terminology and the design process for creating a sheetmetal part</li> <li>- Define and manage the sheetmetal part parameters</li> <li>- Design walls, bends, and flanges</li> <li>- Add features such as cutouts, holes, corners, and chamfers</li> <li>- Create standard and user-defined stamped features</li> <li>- Manage folded and unfolded views and export a finished flat pattern</li> </ul> |
| 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"> <li>- Sketcher</li> <li>- Part Design</li> <li>- Drafting</li> </ul> |
| 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-V5R27   |
| 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"> <li>- Design 3D mechanical parts using basic features</li> <li>- Create 3D solid features based on 2D sketches</li> <li>- Apply Dress-Up features to the 3D parts</li> <li>- Duplicate and move the 3D features</li> <li>- Modify a 3D part</li> </ul> |
| 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"> <li>- Create a part using 3D reference elements</li> <li>- Create advanced Sketch-Based Features</li> <li>- Apply advanced Dress-Up Features</li> <li>- Design 3D parts using Boolean operations</li> <li>- Work in a Multi-Model Environment and share your designs with others</li> <li>- Analyze parts and optimize them</li> <li>- Annotate the parts for review</li> </ul> |
| 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"> <li>- Create a new product and add components to it</li> <li>- Move the components within a product by positioning them using assembly constraints</li> <li>- Modify an existing product structure</li> <li>- Design new parts in the context of a product</li> <li>- Check the mechanical properties of a product and analyze its degrees of freedom</li> <li>- Analyze interferences between parts and perform measurements</li> </ul> |
| 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"> <li>- Optimize performance for large and complex designs</li> <li>- Manage contextual links between product documents using publications</li> <li>- Create and use parameters to drive a product design</li> <li>- Create sections to visualize the internal product structure</li> <li>- Create scenes and explode views of a product</li> <li>- Generate annotations and bills of material for assembly drawings</li> </ul> |
| 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"> <li>- Work in the CATIA Sketcher environment</li> <li>- Create 2D sketch geometry</li> <li>- Analyze the sketched geometry</li> <li>- Edit existing 2D profiles</li> <li>- Dimension the sketch and modify it using constraints</li> <li>- Manage sketches within a 3D environment</li> </ul> |
| 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-V5R27   |
| 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"> <li>- Create a clean topology from a set of surfaces and smooth sharp edges</li> <li>- Detect and correct the discontinuities on curves and surfaces</li> <li>- Create solids from surfaces</li> </ul>   |
| 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                                 |  |
| 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"> <li>- Create advanced replication features like Power Copies</li> <li>- Store components and Power Copies into a catalog and reuse them in a new context</li> <li>- Analyze and migrate CATIA V4 models to CATIA V5</li> </ul> |
| 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        | 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"> <li>- Sketcher</li> <li>- Part Design</li> <li>- Drafting</li> <li>- Generative Wireframe and Surface</li> </ul>  |
| 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"> <li>- Sketcher</li> <li>- Part Design</li> <li>- Generative Shape Design</li> <li>- Drafting</li> </ul>  |
| 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"> <li>- Sketcher</li> <li>- Part Design</li> <li>- Drafting</li> </ul>  |
| 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"> <li>- Sketcher</li> <li>- Part Design</li> <li>- Drafting</li> <li>- Generative Wireframe and Surface</li> </ul>  |
| 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 , Japanese  |
| 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"> <li>- Sketcher</li> <li>- Part Design</li> <li>- Drafting</li> <li>- Generative Shape Design</li> </ul> |
| Prerequisites   | Students attending this course should be familiar with the V5-6R2016 CATIA Mechanical Design workbenches.   |
| 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 , V5-6R2018  |
| 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"> <li>- Describe the CATIA V5 interface</li> <li>- Apply the correct solid and surface modeling methodology for body design</li> <li>- Create an automobile part in order to satisfy its design intent</li> <li>- Design and manage parts in the context of an assembly</li> <li>- Produce simple drawings and assembly layouts</li> </ul> |
| 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 , V5-6R2018   |
| 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"> <li>- Describe the CATIA V5 interface</li> <li>- Apply the correct solid and surface modeling methodology for Chassis design</li> <li>- Create an automobile part in order to satisfy its design intent</li> <li>- Apply advanced solid modeling technique necessary for Chassis design methodology</li> <li>- Design and manage parts in the context of an assembly</li> <li>- Produce simple drawings and assembly layouts</li> </ul> |
| 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 , V5-6R2018   |
| 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"> <li>- Describe the CATIA V5 interface</li> <li>- Apply the correct solid and surface modeling methodology for Powertrain Design</li> <li>- Create an automobile part in order to satisfy its design intent</li> <li>- Apply advanced solid modeling technique necessary for Powertrain design methodology</li> <li>- Design and manage parts in the context of an assembly</li> <li>- Produce simple drawings and assembly layouts</li> </ul> |
| 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-V5R24   |
| Available Releases          | V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018  |
| Duration                    | 40 hours   |
| Course Materials            | Chinese , English , French , German , Japanese , Russian , Spanish   |
| 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"> <li>- Understand and use the CATIA V5 interface</li> <li>- Plan the construction of a part in order to convey its visual and functional aspects</li> <li>- Create simple parts in CATIA V5</li> <li>- Construct an assembly using the parts</li> <li>- Produce simple drawings and assembly layouts</li> </ul> |
| 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-V5R26   |
| Available Releases                      | V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018  |
| 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"> <li>- Create a complex model in CATIA V5</li> <li>- Create and manage a structured model</li> <li>- Design parts in the context of an assembly</li> <li>- Re-use existing data to complete assemblies</li> <li>- Manage relationships between assembled parts</li> <li>- Analyze and annotate your design</li> </ul> |
| Prerequisites                           | Students attending this course should be familiar with the basics of CATIA V5 Mechanical Design  |
| 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"> <li>- Open CATIA V5 documents and use basic tools to modify them</li> <li>- Use the specification tree to browse and understand the structure of an object</li> <li>- Use the compass to manipulate the viewpoint</li> <li>- View and modify the graphic properties of an object</li> </ul> |
| Prerequisites                       | None  |
| 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"> <li>- Use and manage the Knowledgeware working environment</li> <li>- Understand how collaborative work affects knowledge features</li> <li>- Use parameters, formulas and design tables</li> <li>- Create parametric parts and assemblies</li> <li>- Share parameters and reuse relations</li> </ul> |
| Prerequisites                      | Students attending this course should be familiar with CATIA V5 Part Design and Assembly Design   |
| Available Online                   | Yes   |

# CATIA

## CATIA Shape Design and Styling V5



| CATIA Generative Shape Design<br>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"> <li>- Wireframe creation</li> <li>- Curve creation</li> <li>- Surface Creation</li> <li>- Surface Topology</li> </ul> |
| Prerequisites  | Students attending this course should be familiar with the V5R19 CATIA Generative Shape Design workbench.   |
| Available Online   | Yes   |

| CATIA Surface Design Expert (GSD) |  |
|-----------------------------------|--|
| Course Code                       | CAT-de-GSD-A-V5R25   |
| 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"> <li>- Create wireframe features using existing curves and surfaces</li> <li>- Create advanced and parameterized swept surfaces</li> <li>- Perform advanced surface analysis and gap correction</li> <li>- Create advanced blend features</li> <li>- Improve the quality and stability of created geometries</li> </ul> |
| Prerequisites                     | Students attending this course should have attended the CATIA Surface Design course  |
| Available Online                  | Yes  |

| CATIA Surface Design Expert Added Exercises (GSD) |   |
|---|---|
| Course Code                                       | CAT-de-GSD-X-V5R25  |
| 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"> <li>- Create wireframe features using existing curves and surfaces</li> <li>- Create advanced and parameterized swept surfaces</li> <li>- Perform advanced surface analysis and gap correction</li> <li>- Create advanced blend features</li> <li>- Improve the quality and stability of create geometries</li> </ul> |
| Prerequisites                                     | Students attending this course should have attended the CATIA Surface Design Expert course  |
| Available Online                                  | Yes   |

| <h2 style="text-align: center;">CATIA V5-6R2014 to V5-6R2016<br/>Update for Shape Designers (UHD46)</h2> |   |
|--|---|
| 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"> <li>- Reference Geometry Creation</li> <li>- Curve Creation</li> <li>- Surface Creation</li> </ul> |
| 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:<br><ul style="list-style-type: none"> <li>- Generative Shape Design</li> </ul> |
| 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"> <li>- Reference Geometry Creation</li> <li>- Curve Creation</li> <li>- Surface Creation</li> </ul> |
| 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 , Japanese   |
| 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"> <li>- Reference Geometry Creation</li> <li>- Surface Creation</li> <li>- Generative Shape Design</li> </ul> |
| 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-en-V5S-F-V5R24  |
| Available Releases          | V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018   |
| 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"> <li>- Understand and use the tools of the Generative Shape Design workbench</li> <li>- Create good quality curves based on a sound and improved wireframe geometry</li> <li>- Assemble, relimit, and connect the surfaces to get a topology</li> <li>- Analyze the quality of surfaces and rectify the defects</li> <li>- Manage the surfaces in a multi-model environment</li> </ul> |
| Prerequisites               | Students attending this course should be familiar with the fundamentals of CATIA V5.  |
| Available Online            | Yes   |



# ENOVIA

## Digital Mock-Up V5

| Digital Mock-Up Basics (DMB) |   |
|------------------------------|---|
| Course Code                  | ENOV-en-DMB-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                     | Mechanical Designers, Industrial Designers, Managers  |
| Description                  | This course will help you to understand the capabilities of each CATIA V5 Digital Mock-Up workbench and analyze which one suits your needs in a given situation. You will learn how to visualize and inspect a complex assembly in order to investigate the problem areas and highlight critical points.  |
| Objectives                   | <p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> <li>- Understand and use the capabilities of the Digital Mock-Up workbenches</li> <li>- Manage assembly components and explore their mock-up details</li> <li>- Manipulate view points</li> <li>- Perform measurements</li> <li>- Highlight critical areas using 2D and 3D annotations</li> <li>- Link information to external files</li> </ul> |
| Prerequisites                | Students attending this course should should be familiar with CATIA V5 basics   |
| Available Online             | Yes   |

| Digital Mock-Up Navigator (DMN) |  |
|---------------------------------|--|
| Course Code                     | ENOV-en-DMN-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, Managers   |
| Description                     | This course will teach you how to manipulate a Digital Mock-Up in the context of an engineering review. You will also learn how to create simulations for review presentations.  |
| Objectives                      | <p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> <li>- Use the basic and advanced functionalities of the DMU Navigator workbench</li> <li>- Modify the properties of components and position them</li> <li>- Create movies using the simulations</li> <li>- Manage the mock-up configurations using scenes</li> <li>- Save specific mock-up configurations for analysis</li> <li>- Create annotated views of a mock-up for sharing</li> </ul> |
| Prerequisites                   | Students attending this course should have attended the DMU Basics course  |
| Available Online                | Yes  |

| Digital Mock-Up Optimizer (DMO) |   |
|---------------------------------|---|
| Course Code                     | ENOV-en-DMO-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                        | Mechanical Designers, Engineering Managers  |
| Description                     | This course will teach you how to improve productivity by computing an optimized data geometric representation for rapid mock-up verification in the context of a collaborative design review environment.  |
| Objectives                      | <p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> <li>- Identify which DMU settings and capabilities are used to manage simplified representations</li> <li>- Select and use a simplified representation</li> <li>- Compute thickness and offset representations</li> <li>- Compute swept and vibration volumes</li> <li>- Compute Free Space and 3D Cut representations for performing measurements</li> </ul> |
| Prerequisites                   | Students attending this course should have attended the DMU Basics and DMU Space Analysis courses   |
| Available Online                | Yes   |

| Digital Mock-Up Space Analysis (SPA) |   |
|--------------------------------------|---|
| Course Code                          | ENOV-en-SPA-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                             | Mechanical Designers, Engineering Managers  |
| Description                          | This course will teach you how to review and validate designs throughout the product lifecycle, from design in context to maintenance review. You will also learn how to highlight interference problems and verify internal component clearances.  |
| Objectives                           | <p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> <li>- Perform measurements in the context of a digital mock-up</li> <li>- Create views to see the inner details of a digital mock-up</li> <li>- Analyze interferences to identify clashes, contacts, and component clearances</li> <li>- Compare different versions of a digital mock-up</li> </ul> |
| Prerequisites                        | Students attending this course should have attended the DMU Basics course   |
| Available Online                     | Yes   |

# ENOVIA

## ENOVIA SmarTeam V5

## ENOVIA SmarTeam Administration for Foundation, Editor & Web Editor (STA)

|                    |  |
|--------------------|--|
| Course Code        | ENOV-en-STA-F-V5R26  |
| Available Releases | V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017  |
| Duration           | 24 hours   |
| Course Materials   | English , French , German , Japanese   |
| Level              | Fundamental  |
| Audience           | New ENOVIA SmarTeam Administrators   |
| Description        | This course will teach you how to perform administrative tasks in ENOVIA SmarTeam. You will learn how to create and manage user profiles, data model structures, lifecycle rules, and workflows. You will also learn how to add and modify user-defined commands and menus.  |
| Objectives         | <p>Perform basic and advanced configuration tasks for SmarTeam Foundation, Editor, and Web Editor</p> <ul style="list-style-type: none"> <li>- Create and modify data model structures</li> <li>- Create user profiles and assign authorizations</li> <li>- Add and modify user-defined commands and menus</li> <li>- Create and modify workflows</li> <li>- Manage the lifecycle rules</li> </ul> |
| Prerequisites      | Students attending this course should have attended the ENOVIA SmarTeam Fundamentals course and the ENOVIA SmarTeam Editor course  |
| Available Online   | Yes  |

| ENOVIA SmarTeam - CATIA Integration (TPU) |   |
|---|---|
| Course Code                               | ENOV-de-TPU-F-V5R24   |
| Available Releases                        | V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018   |
| Duration                                  | 8 hours   |
| Course Materials                          | English , French , German , Japanese  |
| Level                                     | Fundamental   |
| Audience                                  | Engineers, CAD Designers, Suppliers, and Team Leaders involved in product development   |
| Description                               | This course will teach you how to manage CATIA Parts and Assemblies, and maintain the dependencies and data integrity while performing lifecycle operations using ENOVIA SmarTeam. You will also learn the concepts of Collaborative Design and Relational Design, and how to apply these concepts using ENOVIA SmarTeam.   |
| Objectives                                | <p>Manage CATIA products using ENOVIA SmarTeam</p> <ul style="list-style-type: none"> <li>- Manage the various CATIA links and lifecycles associated with CATIA products</li> <li>- Understand how the concepts of Relational Design and Collaborative Design are implemented</li> <li>- Use Properties Mapping</li> <li>- Use Standard CATIA Parts and Catalogs</li> </ul> |
| Prerequisites                             | Students attending this course should have attended the ENOVIA SmarTeam Fundamentals course and the ENOVIA SmarTeam Editor course   |
| Available Online                          | Yes   |



| <h2 style="text-align: center;">ENOVIA SmarTeam - CATIA Supply Chain Engineering Exchange (SEE)</h2> |  |
|--|--|
| Course Code  | ENOV-de-SEE-F-V5R24  |
| Available Releases   | V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018  |
| Duration   | 8 hours  |
| Course Materials   | English , French , German , Japanese   |
| Level  | Fundamental  |
| Audience   | Design Managers and Design Engineers who are working in a collaborative environment  |
| Description  | This course will teach you how to use the ENOVIA SmarTeam - CATIA Supply Chain Engineering Exchange product to exchange data. You will learn to perform two types of data exchanges: - ENOVIA SmarTeam file-based exchange - Exchange between two independent ENOVIA SmarTeam installations  |
| Objectives   | <p>Build upon your knowledge of ENOVIA SmarTeam - CATIA Integration and use the ENOVIA SmarTeam - CATIA Supply Chain Engineering Exchange product proficiently</p> <ul style="list-style-type: none"> <li>- Perform file-based data exchanges in ENOVIA SmarTeam</li> <li>- Exchange data between two independent ENOVIA SmarTeam installations</li> </ul> |
| Prerequisites  | Students attending this course should have attended the ENOVIA SmarTeam - CATIA Integration course   |
| Available Online   | Yes  |

| ENOVIA SmarTeam - Editor (SED) |   |
|--------------------------------|---|
| Course Code                    | ENOV-de-SED-F-V5R24   |
| Available Releases             | V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018   |
| Duration                       | 16 hours  |
| Course Materials               | English , French , German , Japanese  |
| Level                          | Fundamental   |
| Audience                       | Reviewers, Engineers, Designers, Sales & Support Staff, and Managers  |
| Description                    | This course will teach you how to use the ENOVIA SmarTeam - Editor and Workflow products. You will learn how to create data and manage its lifecycle and workflow using ENOVIA SmarTeam. To complement the theory a detailed PLM-based Master Exercise, split into short steps, allows you to practice working with ENOVIA SmarTeam in an industrial context.   |
| Objectives                     | <p>Build upon your knowledge of ENOVIA SmarTeam Fundamentals and use the ENOVIA SmarTeam - Editor and Workflow products proficiently</p> <ul style="list-style-type: none"> <li>- Create, search, view, and manage your Product Data</li> <li>- Use the various Workflow modules to create, work with, manage, and customize your Business Processes</li> </ul> |
| Prerequisites                  | Students attending this course should have attended the ENOVIA SmarTeam Fundamentals course   |
| Available Online               | Yes   |

| ENOVIA SmarTeam Fundamentals (SFF) |   |
|------------------------------------|---|
| Course Code                        | ENOV-de-SFF-F-V5R25   |
| Available Releases                 | V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018   |
| Duration                           | 4 hours   |
| Course Materials                   | English , French , German , Japanese  |
| Level                              | Fundamental   |
| Audience                           | <ul style="list-style-type: none"> <li>- Users who are new to PLM, and more specifically to ENOVIA SmarTeam.</li> <li>- Users who want to test their knowledge on ENOVIA SmarTeam fundamentals.</li> </ul>  |
| Description                        | This course will introduce you to the concept of PLM and show how it is implemented by ENOVIA SmarTeam. You will become conversant with the terminology used in ENOVIA SmarTeam and learn the basic concepts of ENOVIA SmarTeam Data Management, Lifecycle Mechanism, and Workflow. |
| Objectives                         | Outline the basics of PLM and ENOVIA SmarTeam <ul style="list-style-type: none"> <li>- Describe how ENOVIA SmarTeam stores and manages different types of Product information</li> <li>- Explain the basic concepts of Lifecycle Management and Workflow</li> </ul>                 |
| Prerequisites                      | None  |
| Available Online                   | Yes   |

| ENOVIA SmarTeam - Web Editor (WED) |   |
|------------------------------------|---|
| Course Code                        | ENOV-de-WED-F-V5R24   |
| Available Releases                 | V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5-6R2018   |
| Duration                           | 8 hours   |
| Course Materials                   | English , French , German , Japanese  |
| Level                              | Fundamental   |
| Audience                           | Engineers, Designers, Managers, Sales & Support Staff, and Suppliers  |
| Description                        | This course will teach you how to work with ENOVIA SmarTeam - Web Editor. You will learn how to view projects and documents, manage their lifecycle, and use the various search functions to retrieve data. You will also learn about Workflow functions in brief. The course also contains a scenario-based Master Exercise to allow you to practice what you have learnt.                           |
| Objectives                         | <p>Create and manage Projects and their related data using ENOVIA SmarTeam - Web Editor</p> <ul style="list-style-type: none"> <li>- Search for different types of product data</li> <li>- Use the Viewer to view CAD data</li> <li>- Manage the lifecycle of your product data</li> <li>- Use the integrated Workflow functionality to work with processes in a collaborative environment</li> </ul> |
| Prerequisites                      | Students attending this course should have attended the ENOVIA SmarTeam Fundamentals course   |
| Available Online                   | Yes   |

