

Course Catalog

V5

09 June 2017



3DEXPERIENCE®

3DS Learning Solutions | Course Catalog

CATIA Mechanical Design V5	58
2D Layout for 3D Design (LO1)	59
3D Functional Tolerancing and Annotation (FTA)	60
Advanced Drafting and Customization (DRA)	61
Aerospace Sheetmetal Design (ASL)	62
CATIA Composites Design V5R20 Update (UCPD20)	63
CATIA Detail Drafting (DDR)	64
CATIA Generative Drafting Fundamentals (ANSI) (GDRA)	65
CATIA Generative Drafting Fundamentals (ISO) (GDRI)	66
CATIA Generative Sheetmetal Design (SMD)	67
CATIA Generative Sheetmetal Design V5R19 Updates (SMD)	68
CATIA Generative Sheetmetal Design V5R20 Updates (USMD20)	69
CATIA Mechanical Design V5-6R2012 Update (UMD22)	70
CATIA Mechanical Design V5-6R2013 Update (UMD23)	71
CATIA Mechanical Design V5R19 to V5-6R2014 Update (UMD94)	72
CATIA Mechanical Design V5R19 Update (MD2)	73
CATIA Mechanical Design V5R20 Update (UMD20)	74
CATIA Part Design (PDG)	75

3DS Learning Solutions | Course Catalog

CATIA Part Design Added Exercises (PDG)	76
CATIA Part Design Expert (PDG)	77
CATIA Product Design (ASM)	78
CATIA Product Design Added Exercises (ASM)	79
CATIA Product Design Expert (ASM)	80
CATIA Sketcher (SKE)	81
CATIA Surface Design (GS1)	82
CATIA Surface Design Added Exercises (GS1)	83
CATIA Tools For Proficient Users (PRO)	84
CATIA V5-6R2014 to V5-6R2016 Update for Mechanical Designers (UMD46)	85
CATIA V5-6R2014 Update for Designers (UMSD24)	86
CATIA V5-6R2015 Update for Mechanical Designers (UMD25)	87
CATIA V5-6R2016 Update for Mechanical Designers (UMD26)	88
CATIA V5-6R2017 Update for Mechanical Designers (UMD27)	89
CATIA V5 Foundations for Aerospace Assembly Designers (V5AeA)	90
CATIA V5 Foundations for Aerospace Part Designers (V5AeD)	91
CATIA V5 Foundations for Aerospace Part Reviewers (V5AeR)	92
CATIA V5 Foundations for Body Designers (V5VB)	93
CATIA V5 Foundations for Chassis Designers (V5VC)	94
CATIA V5 Foundations for Powertrain Designers (V5VP)	95
CATIA V5 Fundamentals (V5F)	96
CATIA V5 Mechanical Design Expert (V5E)	97
CATIA V5-V6 Design Synchronization Essentials (DCE5)	98
Composites Grid Approach (CPG)	100
Composites Part Engineering (CPE)	101
Composites Part Manufacturing (CPM)	102
Core and Cavity Design (CCV)	104
Functional Molded Parts (FMP)	105
Getting Started with CATIA V5 (COM)	106
Healing Assistant (HA1)	107
Mold Tooling Design (MTD)	108
Part Design Features Recognition (FR1)	109
SEE Project : Assembly Design (SEEASM)	110
SEE Project : Drafting Design (SEEGDR)	111
SEE Project : Part Design (SEEPDG)	112
Tooling Design (TG1)	113

CATIA

CATIA Mechanical Design V5

2D Layout for 3D Design (LO1)

Course Code	CAT-en-LO1-F-V5R26
Available Releases	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R19 , V5R20 , V5R21
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	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R19 , V5R20 , V5R21
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

Advanced Drafting and Customization (DRA)

Course Code	CAT-en-DRA-A-V5R21
Available Releases	V5R19 , V5R20 , V5R21
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Draftsmen, Drafting Administrators
Description	This course will teach you how to set and manage all dimension and annotation standards contained in the standard files according to company or projects needs.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Use hints and tips on Generative and Interactive drafting - Perform administration tasks to set and manage all dimension and annotation standards - Generate coordinate tables - Create frames and title blocks with a macro
Prerequisites	Students attending this course should be familiar with CATIA V5 fundamentals and VB scripting
Available Online	Yes

Aerospace Sheetmetal Design (ASL)

Course Code	CAT-en-ASL-F-V5R21
Available Releases	V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Aerospace Designers
Description	This course will teach you how to use the CATIA Aerospace Sheetmetal Design workbench. You will learn how to create and modify the design of a Hydro-formed Sheetmetal Part by defining its internal features in this workbench. You will also learn how to create a drawing of a flattened part.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Define the parameters for an aerospace sheet metal part. - Create and modify the design of a Hydro-formed Sheetmetal Part. - Generate and draw a flattened part. - Create a Knowledge Expert Check using characteristic curves.
Prerequisites	Students attending this course should be familiar with Part Design, Assembly Design and Wireframe & Surface Design.
Available Online	Yes

CATIA Composites Design V5R20 Update (UCPD20)

Course Code	CAT-en-UCPD20-U-V5R20
Available Release	V5R20
Duration	4 hours
Course Material	English
Level	Update
Audience	Composite Designer, CATIA V5 Designer
Description	This course will teach you how to use the enhanced V5R20 functionalities of the Composites Design and the Composites Grid Design workbenches.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Use the new features available in Grid-based design. - Use the new general Composites design functionalities and enhancements. - Prepare a Composites part for manufacturing using the improved manufacturing preparation functions.
Prerequisites	Students attending this course should have knowledge of V5R19 CATIA Composites Design.
Available Online	Yes

CATIA Detail Drafting (DDR)

Course Code	CAT-en-DDR-F-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	16 hours
Course Material	English
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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
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-en-SMD-F-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Sheetmetal Designers
Description	This course will teach you how to create a sheet metal part using the standard wall, bend and stamping features. You will view how user features can be incorporated into a design and how you can make use of both standard and user-defined materials. Finally you will learn how to create a flat pattern and produce a detailed, annotated drawing.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Relate to 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 Generative Sheetmetal Design V5R19 Updates (SMD)

Course Code	CAT-fr-SMD-U-V5R19
Available Release	V5R19
Duration	4 hours
Course Materials	English , French , German , Japanese
Level	Update
Audience	Mechanical Designers, Sheetmetal designers
Description	This course will teach you learn the enhancements in the Extrusion, Recognize, and Paste Special functionalities. You will also learn about the new functionality of integrating the unfolded curve in the drawing.
Objectives	Upon completion of this course you will be able to use the enhanced functionalities in the Generative Sheetmetal Design Workbench for the V5R19 release.
Prerequisites	Students attending this course must have knowledge of CATIA Generative Sheetmetal Design V5R18.
Available Online	Yes

CATIA Generative Sheetmetal Design V5R20 Updates (USMD20)

Course Code	CAT-en-USMD20-U-V5R20
Available Release	V5R20
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 V5R20 functionalities of CATIA Generative Sheetmetal Design workbench.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create multiple Walls On Edge in a single step - Recognize solids having chamfers - Use the enhanced Cutout options for getting better results - Create a bend from flat using Bend Tangent Line (BTL) Support - Create a user stamp on both sides of a sheemetal part - Create a stamp that lie on a bend
Prerequisites	Students attending this course should have knowledge of V5R19 CATIA Generative Sheetmetal Design
Available Online	Yes

CATIA Mechanical Design V5-6R2012 Update (UMD22)

Course Code	CAT-en-UMD22-U-V5R22
Available Release	V5-6R2012
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 V5-6R2012 Mechanical Design workbenches. You will see for example, the way in which elements can now be projected and offset while in the Sketcher workbench and how rectangular patterns can now be created in the Part Design workbench. You will also see how to visualize part sections in 3D, use the improved wireframe functions in the Generative Shape Design workbench and simplify the creation of section profiles 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 V5-6R2012 for the following Mechanical Design workbenches:</p> <ul style="list-style-type: none"> - Sketcher - Part Design - Assembly Design - Generative Shape Design
Prerequisites	Students attending this course should be familiar with the V5R19 CATIA Mechanical Design workbenches.
Available Online	Yes

CATIA Mechanical Design V5-6R2013 Update (UMD23)

Course Code CAT-ja-UMD23-F-V5R23

Available Release V5-6R2013

Duration 4 hours

Course Materials English , French , German , Japanese

Level Fundamental

Audience Mechanical Designers

Description This course will teach you how to use the enhanced functionalities in the CATIA V5-6R2013 Mechanical Design workbenches. You will see for example, the way in which equivalent dimensions are now managed while in the Sketcher workbench and how circular patterns can now be created in the Part Design workbench. You will also see how to edit repetitive elements in the Generative Shape Design workbench to and modify tolerance display factors 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 V5-6R2013 for the following Mechanical Design workbenches:

- Sketcher
- Part Design
- Assembly Design
- Generative Shape Design

Prerequisites Student attending this course should be familiar with the V5-6R2012 CATIA Mechanical Design workbenches.

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 Mechanical Design V5R19 Update (MD2)

Course Code	CAT-ja-MD2-U-V5R19
Available Release	V5R19
Duration	8 hours
Course Materials	English , French , Japanese
Level	Update
Audience	Mechanical Designers
Description	This course will teach you how to use the enhanced functionalities in V5R19 CATIA Mechanical Design workbenches.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - - Upgrade and explode the sketch features such as edge fillet and tapered hole - - Modify the CATPart geometry in assembly context to create a new CATPart using new Associatively command - - Use the points and plane repetition command - - Customize a BOM using new command Advanced Bill of Material - -Demonstrate Broken and normal(non-broken) constraints
Prerequisites	<ul style="list-style-type: none"> - Students attending this course should have knowledge of - CATIA Mechanical Design V5R18
Available Online	Yes

CATIA Mechanical Design V5R20 Update (UMD20)

Course Code	CAT-ja-UMD20-U-V5R20
Available Release	V5R20
Duration	32 hours
Course Materials	English , French , German , Japanese
Level	Update
Audience	Mechanical Designers
Description	This course will teach you how to use the enhanced V5R20 functionalities of CATIA Mechanical Design workbenches.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Visualize a section of a part dynamically in the 3D Viewer - Add an edge fillet at the intersection of selected features - Create points on a curve along a direction - Repeat objects using datum mode and relative mode - Isolate a feature by breaking the links with its input - Create a mirrored extrude / cylinder - Create a V5 conic identical to a V4 conic - Position a section line at a specified distance from a selected edge - Customize a BOM to display the values for user-defined attributes - Upgrade your drafting data to the latest level
Prerequisites	Students attending this course should have knowledge of V5R19 CATIA Mechanical Design
Available Online	Yes

CATIA Part Design (PDG)

Course Code	CAT-en-PDG-F-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
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 <ul style="list-style-type: none"> - 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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	12 hours
Course Material	English
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	Apply your Mechanical skills in selected scenarios. <ul style="list-style-type: none"> - Employ the recommended methodology in various situations and efficiently use 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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	12 hours
Course Material	English
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 3D models. You will also learn to analyze 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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to create a simple product structure, 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 measure parts or products
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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
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	<p>Upon completion of this course, you will be able to:</p> <ul style="list-style-type: none"> - Examine your mechanical skills and exercise on selected scenarios - Apply the recommended methodology in various situations - Use 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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	16 hours
Course Material	English
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 optimally use CATIA 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 of 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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
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 a 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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
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	<p>These exercises will allow you to put your Shape skills into practice on selected scenarios.</p> <ul style="list-style-type: none"> - 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-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Advanced CATIA V5 Users
Description	This course will teach you how to use advanced CATIA functions such as Catalog Edition, Powercopy Feature 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	<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 V5-6R2015 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 Material	English
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 Foundations for Aerospace Assembly Designers (V5AeA)

Course Code	CAT-en-V5AeA-F-V5R26
Available Releases	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R20 , V5R21
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. You will learn how to create a structured assembly in order to best design parts in an assembly context. You will also learn 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	Students attending this course should have completed the V5 Foundations for Aerospace Part Designers course.
Available Online	Yes

CATIA V5 Foundations for Aerospace Part Designers (V5AeD)

Course Code	CAT-en-V5AeD-F-V5R26
Available Releases	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R20 , V5R21
Duration	40 hours
Course Material	English
Level	Fundamental
Audience	Aerospace Part Designers
Description	This course will introduce you to CATIA V5. It will first 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 also 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-V5R26
Available Releases	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R20 , V5R21
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Aerospace Part Reviewers
Description	This course will first explain how to use the CATIA V5 workbenches. It will then teach you how to review an existing part by verifying its properties, coordinates and measurements. You will also learn 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	Students 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	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R19 , V5R20 , V5R21
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	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R19 , V5R20 , V5R21
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	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R19 , V5R20 , V5R21
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	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R19 , V5R20 , V5R21
Duration	40 hours
Course Materials	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-V5R26
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R19 , V5R20 , V5R21
Duration	40 hours
Course Materials	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

CATIA V5-V6 Design Synchronization Essentials (DCE5)

Course Code

CAT-en-DCE5-F-V5R23

Available Releases

V5-6R2012 , V5-6R2013

Duration

4 hours

Course Material

English

Level

Fundamental

Audience

CATIA V6 designers who need to design in collaboration with CATIA V5 designers.

Description

This is a process-based course which will teach you how synchronised versions of CATIA V6 and CATIA V5 can be used to exchange data during product design. You will see how V6 models can be interactively converted to V5 solids and how V6 features can be preserved in V5, thus allowing a V5 user to modify them. You will see how a modified V5 model can then be imported into V6 and used to replace the original V6 model. Finally, you will see how the batch transfer mode can be used to perform mass data transfer and how it can improve performance.

Objectives

Upon completion of this course you will be able to:

- Convert a CATIA V6 product structure to CATIA V5 interactively.
- Convert a CATIA V6 part to CATIA V5 and modify it in V5.
- Import the modified V5 part into V6 and compare it with the original part.
- Replace a V6 part with a modified V5 part.
- Transfer products and parts between V6 and V5 using batch mode.

CATIA V5-V6 Design Synchronization Essentials (DCE5)

Prerequisites

Students should be familiar with CATIA V5 Fundamentals and CATIA V6 Mechanical Design Fundamentals.

Available Online

Yes

Composites Grid Approach (CPG)

Course Code	CAT-en-CPG-F-V5R26
Available Releases	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R20 , V5R21
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 Part Design, Assembly Design, 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	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R19 , V5R20 , V5R21
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 <ul style="list-style-type: none"> - 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	V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5R19 , V5R20 , V5R21
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 <ul style="list-style-type: none"> - 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

Core and Cavity Design (CCV)

Course Code	CAT-en-CCV-F-V5R21
Available Releases	V5R19 , V5R20 , V5R21
Duration	12 hours
Course Material	English
Level	Fundamental
Audience	Tooling designers with no experience on Core and Cavity Design
Description	This course will teach you how to create the Core and Cavity of a molded part model. You will learn the basic methods used to create the core and cavity areas of a part, including sliders and loose cores that are required to design a Plastic Injection Mold.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Split a shape into mold areas - Create the corresponding parting line and parting surface - Create the core surface, cavity surface and slider/lifter surfaces
Prerequisites	Students attending this course should be familiar with CATIA V5 fundamentals
Available Online	Yes

Functional Molded Parts (FMP)	
Course Code	CAT-en-FMP-F-V5R21
Available Releases	V5R19 , V5R20 , V5R21
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 by defining the material added or removed from the part mold. - Add functional features such as ribs or cutouts to the part. - Finalize the molded part using feature modifiers such as fillets or patterns. - Use the multi-body approach. - Extract cores, cavities and other EDM inserts from the final part.
Prerequisites	CATIA V5 Fundamentals
Available Online	Yes

Getting Started with CATIA V5 (COM)

Course Code	CAT-en-COM-F-V5R27
Available Releases	V5-6R2012 , V5-6R2013 , V5-6R2014 , V5-6R2015 , V5-6R2016 , V5-6R2017 , V5R19 , V5R20 , V5R21
Duration	4 hours
Course Material	English
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 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

Healing Assistant (HA1)

Course Code	CAT-en-HA1-F-V5R21
Available Releases	V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Tooling Designers, Mechanical Designers, Surface Designers.
Description	This course introduces you to CATIA Healing Assistant's user interface and its basic tools. You will learn how to analyze and repair the imported data (IGES 3D or CATIA V4 files). You will also learn how to compare two versions of a Part, and how to customize the workbench to suit your needs.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Analyze the imported data - Repair the imported data - Compare two versions of a Part - Customize the workbench
Prerequisites	Students attending this course should be familiar with the Wireframe and Surfaces.
Available Online	Yes

Mold Tooling Design (MTD)

Course Code	CAT-en-MTD-F-V5R21
Available Releases	V5R19 , V5R20 , V5R21
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Mold Tooling Designers
Description	This course will teach you how to design an injection mold and its components using standard and user-defined catalogs. You will learn the design process with the help of industrial examples.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create a mold base using guided and fixed components - Build a Plastic Injection Mold assembly from scratch
Prerequisites	Students attending this course should be familiar with CATIA V5 fundamentals and Tooling Design fundamentals
Available Online	Yes

Part Design Features Recognition (FR1)

Course Code	CAT-en-FR1-F-V5R21
Available Releases	V5R19 , V5R20 , V5R21
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you to use the Feature Recognition tools of the Part Design workbench. You will learn how to build a comprehensive V5 data structure for solids whose specifications are lost or are unreachable. You will also learn how to perform flexible local design modifications on all kinds of models.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Build a feature-based model from a CATIA V5 isolated BRep solid model - Build a feature-based model from a solid imported from another CAD system
Prerequisites	Students attending this course should be familiar with CATIA V5 Fundamentals and CATIA Part Design.
Available Online	Yes

SEE Project : Assembly Design (SEEASM)

Course Code	CAT-en-SEEASM-F-V5R25
Available Release	V5-6R2015
Duration	12 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	A Self-paced, eLearning solution for CATIA Assembly Design. An Interactive and motivating multi-media experience under the guidance of a virtual teacher. There are 3 modules.
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 - Position components using assembly constraints - Design new parts in the context of a product - Analyze interferences between parts - Measurement distances between component
Prerequisites	CATIA Part Design
Available Online	Yes

SEE Project : Drafting Design (SEEGDR)

Course Code	CAT-en-SEEGDR-F-V5R25
Available Release	V5-6R2015
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	New CATIA V5 Users, Mechanical Designers
Description	A Self-paced, eLearning solution for CATIA Drafting. An Interactive and motivating multi-media experience under the guidance of a virtual teacher. There are 2 modules.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create a single sheet part drawing and add a title block - Create simple projection and section views - Position the views on a drawing sheet - Add dimensions and annotations - Create an assembly layout - Add bills of materials and balloon references
Prerequisites	CATIA Part Design, CATIA Assembly Design
Available Online	Yes

SEE Project : Part Design (SEEPDG)

Course Code	CAT-en-SEEPDG-F-V5R25
Available Release	V5-6R2015
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	New CATIA V5 Users, Mechanical Designers
Description	A Self-paced, eLearning solution for CATIA Part Design. An Interactive and motivating multi-media experience under the guidance of a virtual teacher. There are 6 modules.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Design and dimension 3D mechanical parts - Create solid features based on 2D sketches - Add rounded corners and bevels - Shell and draft a part - Create pocket and hole patterns - Modify a part
Prerequisites	None
Available Online	Yes

Tooling Design (TG1)	
Course Code	CAT-en-TG1-F-V5R21
Available Releases	V5R19 , V5R20 , V5R21
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Tooling Designers
Description	This course teaches you the basics of the tool design. You will also learn how to create and instantiate different components of the mold.
Objectives	<p>Upon completion of this course you will be able to:</p> <ul style="list-style-type: none"> - Create the die and mold components using the Mold Tool Design workbench - Instantiate the components in a die or mold structure
Prerequisites	Students attending this course should be familiar with CATIA V5 fundamentals
Available Online	Yes