

# CLS YLC Companion

## CATIA V5 Mechanical Design (MDC-OC) Bundle

Online Companion Code	Companion Prerequisites	Course Code	Course Title	Training Duration (in hours)
<b>MDC-OC</b>	None	<i>List of courses included in this Online Companion product</i>		<b>220</b>
		ASM_A	CATIA Product Design Expert	16
		ASM_F	CATIA Product Design	8
		ASM_X	CATIA Product Design Added Exercises	8
		COM_F	Getting Started with CATIA V5	4
		DDR_F	CATIA Detail Drafting	16
		GDRA_F	CATIA Generative Drafting Fundamentals (ANSI)	8
		GDRI_F	CATIA Generative Drafting Fundamentals (ISO)	8
		GS1_F	CATIA Surface Design	8
		GS1_X	CATIA Surface Design Added Exercises	4
		PDG_A	CATIA Part Design Expert	12
		PDG_F	CATIA Part Design	8
		PDG_X	CATIA Part Design Added Exercises	16
		PRO_F	CATIA Tools For Proficient Users	8
		SKE_F	CATIA Sketcher	8
		V5E_A	CATIA V5 Mechanical Design Expert	40
		V5F_F	CATIA V5 Fundamentals	40
		KWF_F	CATIA Knowledge Fundamentals	8



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# COURSE CATALOG

V5 Online Companion Products  
Mechanical Design  
January 2014



**3DEXPERIENCE**

# 3DS Learning Solutions | Course Catalog

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## CATIA

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# CATIA

## CATIA Companion for Mechanical Design (MDC-OC)

CATIA Detail Drafting (DDR)	
Course Code	CAT-en-DDR-F-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
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"> <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-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
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 by adding basic dimensions.
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 CATIA Fundamentals
Available Online	Yes

CATIA Generative Drafting Fundamentals (ISO) (GDRI)	
Course Code	CAT-en-GDRI-F-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Mechanical 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 by adding basic dimensions.
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 CATIA Fundamentals
Available Online	Yes



CATIA Knowledge Fundamentals (KWF)	
Course Code	CAT-en-KWF-F-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
Duration	8 hours
Course Material	English
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>Use and manage the Knowledgeware working environment</p> <ul style="list-style-type: none"> <li>- Understand how collaborative work affects knowledge features</li> <li>- Use parameters, formulae, 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 CATIA V5 Assembly Design
Available Online	Yes

CATIA Part Design (PDG)	
Course Code	CAT-en-PDG-F-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	CATIA V5 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 solid feature creation methods</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 solid model in accordance with the manufacturing requirements</li> </ul>
Prerequisites	Students attending this course should be familiar with CATIA Fundamentals and CATIA Sketcher
Available Online	Yes

CATIA Part Design Added Exercises (PDG)	
Course Code	CAT-en-PDG-X-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
Duration	12 hours
Course Material	English
Level	Exercise
Audience	CATIA V5 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	<p>Upon completion of this course you will have:</p> <ul style="list-style-type: none"> <li>- Refreshed your Part Design skills</li> <li>- Put into practice recommended design methodology</li> </ul>
Prerequisites	Students attending this course should be familiar with CATIA Part Design and CATIA Knowledgeware
Available Online	Yes

CATIA Part Design Expert (PDG)	
Course Code	CAT-en-PDG-A-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
Duration	12 hours
Course Material	English
Level	Advanced
Audience	CATIA V5 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 be familiar with CATIA V5 Fundamentals, Getting started with CATIA V5, CATIA Sketcher, and Part Design Fundamentals
Available Online	Yes

CATIA Product Design (ASM)	
Course Code	CAT-en-ASM-F-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
Duration	8 hours
Course Material	English
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-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
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"> <li>- Refreshed your Product Design skills</li> <li>- Learned the recommended design methodologies to create complex designs</li> </ul>
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-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
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 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>- Set the required CATIA options that enable you to optimize its 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 exploded 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

## CATIA Product Design Expert (ASM)

Available Online

Yes



CATIA Sketcher (SKE)	
Course Code	CAT-en-SKE-F-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
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"> <li>- Work in the CATIA Sketcher environment</li> <li>- Create 2D sketch geometry</li> <li>- Analyze the sketch geometry</li> <li>- Edit existing 2D profiles</li> <li>- Dimension the sketch geometry and modify it using constraints</li> <li>- Manage the sketches within a 3D environment</li> </ul>
Prerequisites	Students attending this course should be familiar with CATIA Fundamentals
Available Online	Yes

CATIA Surface Design (GS1)	
Course Code	CAT-en-GS1-F-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
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 these exercises you will be able to:</p> <ul style="list-style-type: none"> <li>- Identify and use the tools that are specific to the Generative Shape Design workbench</li> <li>- Create simple reference geometry and wireframe geometry</li> <li>- Use the reference wireframe elements to create simple surfaces</li> <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-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
Duration	7 hours
Course Material	English
Level	Exercise
Audience	Experienced Mechanical Surface Designers
Description	<p>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.</p>
Objectives	<p>Upon completion of these exercises you will be able to:</p> <ul style="list-style-type: none"> <li>- Create basic wireframes and surfaces using the recommended techniques</li> <li>- Create a solid part after creating the surfacic shell</li> <li>- Create a wireframe before creating the surfaces</li> <li>- Work with multiple parts that are referencing a common part containing the basic specifications</li> </ul>
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-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
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 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 Fundamentals (V5F)	
Course Code	CAT-en-V5F-F-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
Duration	40 hours
Course Material	English
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-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
Duration	40 hours
Course Material	English
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-V5R23
Available Releases	V5R19 , V5R20 , V5R21 , V5-6R2013
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 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