DASSAULT SYSTEMES | The 3DEXPERIENCE® Company

Course Catalog

Companion Mechanical Design

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Learning Experience | Course Catalog

CATIA

CATIA

CATIA Mechanical Design V5

CATIA Detail Drafting	
Course Code	CAT-en-DDR-F-V5R29
Available Release	V5-6R2019
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	Upon completion of this course, you will be able to: - Create an interactive view and draw a sketch on it - Add annotations to dress-up the view - Use advanced dimensioning tools - 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)	
Course Code	CAT-en-GDRA-F-V5R29
Available Release	V5-6R2019
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 dimensions to it.
Objectives	Upon completion of this course you will be able to: - 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)	
Course Code	CAT-en-GDRI-F-V5R29
Available Release	V5-6R2019
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 dimensions to it.
Objectives	Upon completion of this course you will be able to: - 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	
Course Code	CAT-en-SMD-F-V5R29
Available Release	V5-6R2019
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	Upon completion of this course you will be able to: - 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 Part Design	
Course Code	CAT-en-PDG-F-V5R29
Available Release	V5-6R2019
Duration	11 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	Upon completion of this course you will be able to: - Design 3D mechanical parts using basic features - Create 3D solid features based on 2D sketches - Apply Dress-Up features to the 3D parts - Duplicate and move the 3D features - Modify a 3D part
Prerequisites	Student attending this course must be familiar with CATIA V5 fundamentals and CATIA Sketcher.
Available Online	Yes

CATIA Part Design Added Exercises	
Course Code	CAT-en-PDG-X-V5R29
Available Release	V5-6R2019
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	Upon completion of this course you will be able to: - Apply your Mechanical skills in selected scenarios. - 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	
Course Code	CAT-en-PDG-A-V5R29
Available Release	V5-6R2019
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	Upon completion of this course you will be able to: - 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	Student attending this course must be familiar with CATIA V5 fundamentals and must have completed the CATIA Sketcher and CATIA Part Design Fundamentals courses.
Available Online	Yes

CATIA Product Design	
Course Code	CAT-en-ASM-F-V5R29
Available Release	V5-6R2019
Duration	12 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	Upon completion of this course you will be able to: - 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	
Course Code	CAT-en-ASM-X-V5R29
Available Release	V5-6R2019
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	Upon completion of this course, you will be able to: - 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 be familiar with Part and Assembly design in CATIA.
Available Online	Yes

CATIA Product Design Expert	
Course Code	CAT-en-ASM-A-V5R29
Available Release	V5-6R2019
Duration	17 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	Upon completion of this course you will be able to: - 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 Assembly Design and CATIA Part Design.
Available Online	Yes

CATIA Sketcher	
Course Code	CAT-en-SKE-F-V5R29
Available Release	V5-6R2019
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	Upon completion of this course you will be able to: - 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 be familiar with basics of CATIA V5 and mechanical design.
Available Online	Yes

CATIA Surface Design	
Course Code	CAT-en-GS1-F-V5R29
Available Release	V5-6R2019
Duration	16 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	Upon completion of this course you will be able to: - 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	
Course Code	CAT-en-GS1-X-V5R29
Available Release	V5-6R2019
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	These exercises will allow you to put your Shape skills into practice on selected scenarios. - You will apply the recommended methodology in various situations - You will enhance your understanding and usage of the Shape workbenches.
Prerequisites	Students attending this course should be familiar with CATIA V5 Surface Design.
Available Online	Yes

CATIA Tools For Proficient Users	
Course Code	CAT-en-PRO-F-V5R29
Available Release	V5-6R2019
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	Upon completion of this course you will be able to: - 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 V5-6R2019 Update for Designers	
Course Code	CAT-en-UMSD29-U-V5R29
Available Release	V5-6R2019
Duration	3.5 hours
Course Material	English
Level	Update
Audience	Mechanical Designers
Description	This course will teach you how to use the enhanced functionalities in CATIA V5-6R2019. You will learn how to create two tangent arc corners with different radius in the Sketcher workbench and create a blend corner in the Part Design workbench. You will also learn how to heal gaps between elements while joining them in the Generative Shape Design workbench and fill the text frames and balloons with the required color in the Drafting workbench.
Objectives	Upon completion of this course you will be able to effectively use the new and enhanced tools in CATIA V5-6R2019 for the following workbenches: - Sketcher - Part Design - Generative Shape Design - Drafting
Prerequisites	Students attending this course should be familiar with the V5-6R2018 CATIA Mechanical Design workbenches.
Available Online	Yes

Getting Started with CATIA V5	
Course Code	CAT-en-COM-F-V5R29
Available Release	V5-6R2019
Duration	4.5 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	Upon completion of this course you will be able to: - Open CATIA V5 documents and use basic tools to modify them - Use the specification tree to browse and understand the structure of an object - Use the Compass to manipulate the viewpoint - View and modify the graphic properties of an object
Prerequisites	None
Available Online	Yes

CATIA

CATIA Product Synthesis V5

CATIA Knowledge Fundamentals	
Course Code	CAT-en-KWF-F-V5R29
Available Release	V5-6R2019
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	Upon completion of this course you will be able to: - Use and manage the Knowledgeware working environment - Understand how collaborative work affects knowledge features - Use parameters, formulas and design tables - Create parametric parts and assemblies - Share parameters and reuse relations
Prerequisites	Students attending this course should be familiar with CATIA V5 Part Design and Assembly Design
Available Online	Yes


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