DASSAULT SYSTEMES | The 3DEXPERIENCE Company

COURSE CATALOG

V6
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CATIA Equipments V6

CATIA 3D Electrical Design Essentials (EHD)

Course Code	CAT-en-EHD-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	Electrical Engineers who are new to Electrical Physical System Design using CATIA V6
Description	This course will teach you to create electrical physical system in CATIA V6 and thereby help you in designing the electrical physical systems. You will work with the catalogs to place the components from the electrical libraries. You will learn the routing of branches for creating electrical branch geometries, managing the electrical geometry content, and routing conductors through the electrical geometry. You will also learn the 3D Master Approach of annotating the electrical physical system.
Objectives	Upon completion of this course you will be able to: - Import CATIA V5 data into CATIA V6 - Create and use an Electrical Library using Project Resource Management (PRM) - Create an Electrical Geometry - Route Conductors through the Electrical Geometry - Annotate the Electrical Physical System using the 3D Master Approach
Prerequisites	Student attending this course should understand the Electrical Geometry Design process.
Available Online	Yes

CATIA ElectroMechanical Circuit Board Essentials (PCB)

Course Code	CAT-en-PCB-F-V6R130
Available Releases	V6R2012 , V6R2012x , V6R2013
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Engineers who need to prepare electronic circuit boards to exchange data with ECAD applications
Description	This is a process-based course that uses an industrial scenario to teach you how to use the CATIA Circuit Board Design workbench. First, you will learn how to work with a catalog of electronic components. Next, you will learn how to create a circuit board geometry in the context of a mechanical assembly, and how to create spatial and technological constraint areas. You will also learn how to exchange data with an ECAD application using IDF files (import / export). Finally, you will learn how to compare and update the MCAD data to sync it with the ECAD data.
Objectives	Upon completion of this course you will be able to: - Work with a catalog of electronic components - Design a circuit board in the context of an assembly - Exchange data between an ECAD application and CATIA V6 (MCAD) using the IDF format - Compare and update the circuit board design modifications
Prerequisites	- V6 users should have attended the CATIA V6 Mechanical Design Fundamentals (V6F) course

CATIA ElectroMechanical Circuit Board Essentials (PCB)	
	- V5 users should have attended the CATIA V5 to V6 Mechanical Design Transition (V6MT) course - All students should be well-versed with the basic electronics concepts
Available Online	Yes

CATIA Piping and Tubing Administration (PTS)

Course Code	CAT-en-PTS-F-V6R130
Available Releases	V6R2012x , V6R2013
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Senior CAD Designers and Fluid Systems Solution Administrators
Description	This course will teach you how to set up Fluid Systems Resources and create the piping components. You will learn how to manage component catalogs, design validation rules, and global naming conventions. You will also learn how to customize the GVS file for drawings. The course also features a master exercise that enables you to practice creating a piping and tubing setup.
Objectives	Upon completion of this course you will be able to: - Set up and administer the V6 Fluid Systems Solution - Create and manage Fluid Systems Resources - Build equipments, supports, and components - Reuse the piping standard data for design setup - Create and manage component catalogs - Define the global naming conventions - Create the checks and rules for design validation - Create templates for generating reports - Customize the GVS file for drafting
Prerequisites	Students attending this course should have attended the CATIA V6 Mechanical Design Fundamentals and the CATIA Piping and Tubing Design Essentials courses.

CATIA Piping and Tubing Administration (PTS)

Available Online	Yes

CATIA Piping and Tubing Design Essentials (PTD)

Course Code	CAT-en-PTD-F-V6R130
Available Releases	V6R2012 , V6R2013
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Piping or Tubing Designers, CATIA V5 or V6 Designers
Description	Upon completion of this course you will be able to route straight pipes or tubes, place parts, route flexible tubes, adjust the design of piping or tubing network, document and validate the design.
Objectives	Upon completion of this course you will be able to: - Route straight pipes or tubes - Position piping or tubing parts - Manage flexible tubes - Adjust the design of a piping or tubing network - Validate and document the piping and tubing design
Prerequisites	- Students attending this course should : - Understand the Piping or Tubing Design process. - Have attended CATIA V5 to V6 Mechanical Design Transition or CATIA V6 Mechanical Design Fundamentals.
Available Online	Yes

CATIA Systems Generative 3D Electrical Essentials (EGD)

Course Code	CAT-en-EGD-F-V6R130
Available Releases	V6R2012 , V6R2012x , V6R2013
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Electrical System Designers, System Architects, and Electrical Geometry Designers
Description	CATIA Systems Generative 3D Electrical unifies the logical definition of electrical geometry and its physical mock-up in a single workflow. 3D physical electrical geometry and placement can be automatically generated from its 2D schematic logical and 3D space reservation definition. Overall design change management cost is dramatically reduced as a result of the tight coupling between logical and physical aspect, and the quality is improved. Reusing logical information to build physical data will save time to the user, by avoiding him to do the work twice; one in the logical design, and the second time in the 3D design.
Objectives	Upon completion of this course you will be able to: - Explain the significance of the CATIA Systems Generative 3D Electrical product - List the benefits and capabilities of the product - Describe how to generate 3D electrical geometry from logical definitions.
Prerequisites	Students attending this course should be familiar with the CATIA Systems Logical Electrical and Fluidic

CATIA Systems Generative 3D Electrical Essentials (EGD)

	Design, CATIA Systems Logical 3D Architecture Design, and CATIA 3D Electrical Design courses.
Available Online	Yes

CATIA V5 to V6 Electrical Transition (V6VET)

Course Code	CAT-en-V6VET-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	Electrical Designers
Description	This course will teach you how to transition from CATIA V5 Electrical Design to CATIA V6. You will learn how to import CATIA V5 electrical data into V6, create Electrical Device Libraries, instantiate devices, and create Electrical Assemblies. In the V6 context, you will learn how to use electrical assemblies to create an electrical geometry network and route the conductors. Additionally, you will learn how to flatten the electrical geometry and create the corresponding electrical geometry document (electrical drawing). This course will also teach you how to use the Collaboration features of CATIA V6.
Objectives	Upon completion of this course you will be able to: - Manage, create and edit documents in V6 - Collaborate with the Community - Perform Impact Analysis and Propagation - Design parts in the assembly context - Migrate electrical data from V5 to V6 - Create and place electrical devices - Route Electrical Geometry and Conductors - Flatten the Electrical Geometry and create an electrical Drawing - Manage various product configurations

CATIA V5 to V6 Electrical Transition (V6VET)

Prerequisites	Students attending this course must be familiar with Electrical Design in CATIA V5.
Available Online	Yes

CATIA Wire Harness Documentation and Formboard Essentials (EFB)

Course Code	CAT-en-EFB-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Electrical System Designers
Description	This course will teach you how to create 2D drawings of electrical systems, which is required for manufacturing the wire harnesses. You will learn how to extract and flatten an electrical system, create drawings using the 2D catalogs and text templates, and add dimensions to the drawings.
Objectives	Upon completion of this course you will be able to: - Extract an electrical system into the Wire Harness Flattening workbench - Flatten the harness data on the desired 2D plane - Manipulate the flattened harness - Create an electrical drawing - Replace the 3D geometry of components with 2D details - Generate text templates - Create dimensions - Create Support Section views, Device Section views, and Segment Arrangement views
Prerequisites	Students attending this course should have attended the CATIA V6 fundamentals course. They should also be familiar with the Electrical System Design domain and engineering drawings.

CATIA Wire Harness Documentation and Formboard Essentials (EFB)

Available Online	Yes

CATIA

CATIA Infrastructure V6

CATIA V6 Automation Essentials (VBA)

Course Code	CAT-en-VBA-F-V6R131
Available Releases	V6R2012x , V6R2013x
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	CATIA V6 Administrators, VB Automation Engineers, Application Developers
Description	This course will introduce you to automation process in CATIA using VB.Net language. You will learn how to create automation scripts and macros in CATIA V6 using VBA and VB.Net.
Objectives	Upon completion of this course, you will be able to: - Create scripts using VB.Net - Create macros in CATIA V6 - Connect with the Automation server of Microsoft Excel and Word from CATIA - Work with various features of CATIA V6 like Sketches, Part Design Features and Drawings - Understand the V6 Product Model concepts - Learn the coding rules and the V6 adoption guidelines
Prerequisites	Students attending this course should be familiar with CATIA V6 Fundamentals, Scripting preferably with VB, and the Windows Operating System
Available Online	Yes

CATIA

CATIA Knowledge and Reuse V6

CATIA Knowledge Advisor Essentials (KWA)

Course Code	CAT-en-KWA-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers
Description	This course will teach you how to create Knowledge Advisor objects in order to embed parameters and design rules within your models. You will also learn how to check the models, reduce errors and automate the modifications.
Objectives	Upon completion of this course, you will be able to: - Customize the specification tree to display knowledgeware features - Create parametric models - Embed your design knowledge in the models - Automate the design and modification processes - Create design configurations using design tables
Prerequisites	Students attending this course should be familiar with the basics of CATIA V6 and Enterprise Knowledge Language (EKL).
Available Online	Yes

CATIA Product Knowledge Template Definition Essentials (PKT)

Course Code	CAT-en-PKT-F-V6R131
Available Release	V6R2013x
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	CAD Engineers and Knowledge Engineers
Description	This course will teach you how to create and store interactive features and then reuse and adapt them in a new context.
Objectives	Upon completion of this course you will be able to: - Create and reuse geometric features with power copies and user features - Create PLM templates to reuse products in a new context - Create advance instantiation and re-usability tools like the knowledge patterns and the product tables
Prerequisites	Students attending this course should have attended the CATIA V6 Mechanical Design Fundamentals course. They should also be familiar with the Knowledgeware domain.
Available Online	Yes

Introduction to Enterprise Knowledge Language (EKL)	
Course Code	CAT-en-EKL-F-V6R131
Available Releases	V6R2013x , V6R2013
Duration	4 Hours
Course Material	English
Level	Fundamental
Audience	CATIA V6 Designers
Description	This course will introduce you to Enterprise Knowledge Language, the driving force behind the Knowledgeware workbenches, which allow you to construct smart-models and automate design for maximum productivity.
Objectives	Upon completion of this course you will be able to: - Describe the EKL syntax and its usage - Directly manipulate CATIA objects through EKL scripts - Embed design logic in CATIA models using EKL
Prerequisites	Students attending this course should be familiar with CATIA V6 Mechanical Design fundamentals.
Available Online	Yes

CATIA

CATIA Mechanical V6

CATIA Aerospace Sheetmetal Design Essentials (ASH)

Course Code	CAT-en-ASH-F-V6R131
Available Release	V6R2013x
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 to create and modify the design of a Hydro-formed Sheetmetal Part. You will learn how to define its external and internal features. You will also learn how to create a drawing of the flattened part.
Objectives	Upon completion of this course you will be able to: - Define the parameters and standards for an Aerospace Sheet Metal part. - Create and modify the design of a Hydro-formed Sheetmetal Part. - Generate a Drawing with a Flattened Part.
Prerequisites	Students attending this course should be familiar with part design, assembly design and wireframe & surface design using CATIA.
Available Online	Yes

CATIA Composites Engineering Essentials (CEG)

Course Code	CAT-en-CEG-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	40 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, and how the Grid approach can be used for wing, fuselage or wind turbine blade design. You will also learn how to generate plies automatically, generate exact solids, how to use the analysis tools, simulate fiber behavior and create drawings.
Objectives	Upon completion of this course you will be able to: - Define Composites Parameters - Design a Composite Part using the Manual Approach - Design a Composite Part using the Classical Zone Approach - Design a Composite Part using the Solid Zone Approach - Design a Composite Part using the Grid Approach - Perform and inspect the Producibility Analysis - Export and import the Ply Data - Create a Ply Book
Prerequisites	Students attending this course should be familiar with CATIA V6 Fundamentals

CATIA Composites Engineering Essentials (CEG)

Available Online	Yes

CATIA Composites Manufacturing Essentials (CMP)

Course Code	CAT-en-CMP-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	16 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 beween the engineering and the manufacturing data after modification. You will also learn how to apply the manufacturing and producibility constraints in the Composites Design Process.
Objectives	Upon completion of this course you will be able to: - Design a Composite Part using the Manual Approach - Generate a Manufacturing Stacking from an Engineering Stacking - Synchronize the link between the Manufacturing and Engineering parts - Perform and inspect the Producibility Analysis - Compute and optimize a Flattening - Export the Ply Data - Create a Ply Book
Prerequisites	Students attending this course should be familiar with CATIA V6 Fundamentals
Available Online	Yes

CATIA Drafting Essentials (GDR)

Course Code	CAT-en-GDR-F-V6R131
Available Releases	V6R2013 , V6R2013x
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	Draftsmen
Description	This course will teach you how to create drawings using Drafting workbench. You will learn how to produce a drawing by creating projection views and section views of a 3D model and adding basic dimensions. You will also learn how to use advanced tools to dress-up, annotate the views and to customize the Drafting workbench to suit your needs.
Objectives	Upon completion of this course you will be able to: - Customize the Drafting workbench to meet your specific requirements - Create simple projection views and section views of 3D parts - Position the views on a drawing sheet - Add dimensions and annotations to the views - Finalize the drawing sheet by adding a title block - Work with large assemblies - Use Generative View Style to customize the drafting standards and settings
Prerequisites	Students attending this course should be familiar with CATIA V6 Fundamentals.
Available Online	Yes

CATIA Fabricated Part Design Essentials (FPD)

Course Code	CAT-en-FPD-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Mechanical and Structural Designers
Description	This course will teach you how to create a sheet metal part using standard wall, bend and stamping features. You will see how user features can be incorporated into the design and how to use both standard and user-defined materials. Finally you will learn how to create a flat pattern, create a welded part and produce a detailed, annotated drawing.
Objectives	Upon completion of this course you will be able to: - Create a sheet metal part using wall and bend features. - Create stamped features. - Use pre-defined sheet metal parameters. - Manage folded and unfolded views. - Export a finished flat pattern. - Create and manage a welded part. - Generate weld reports. - Create an annotated drawing.
Prerequisites	CATIA V6 Mechanical Design Fundamentals
Available Online	Yes

CATIA Fastener Design Essentials (FSR)

Course Code	CAT-en-FSR-F-V6R130
Available Releases	V6R2012 , V6R2012x , V6R2013
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Engineers
Description	This course will teach you how to create various types of fastener references using the Fastening workbench. You will learn how to instantiate these references in the assembly context. You will also learn how to review, modify and check the fastener instances. Finally, you will learn how to generate the drawing and the fastener report.
Objectives	Upon completion of this course you will be able to: - Prepare a fastening assembly - Create a fastener reference - Instantiate the fastener references - Modify the fastener instances - Review and check the fasteners - Generate drawings and reports
Prerequisites	Students attending this course should be familiar with CATIA Mechanical Design Fundamentals.
Available Online	Yes

CATIA Live Compose Essentials (LCE)

Course Code	CAT-en-LCE-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	- Mechanical Engineers - Mechanical Designers - Design Architects
Description	This course will teach you how to create and manage product structures. You will explore a product and modify its structure by adding new products and exploding existing products. You will then scan the structure to activate a working product level, search for and add existing parts and use constraints to position the parts. Finally, you will create a new sub-product from a components list and use it tocomplete the product.
Objectives	Upon completion of this course you will be able to: - Explore a product and modify its structure using CATIA Live Compose - Select the various working product levels, using the ladder in Live Compose - Search for a product and insert it in the existing assembly - Position the parts using constraints - Create a new sub-product from a component's list and use it to complete the product
Prerequisites	Students attending this course should be familiar with CATIA Live Shape.

CATIA Live Compose Essentials (LCE)

Available Online	Yes

CATIA Live FTA Review Essentials (LFT)

Course Code	CAT-en-LFT-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	- Product Reviewers and Presenters, Designers, Engineers - Documentation, Production, Program Management, Sourcing, Design, Quality, and other such departments where interrogating and annotating the 3D model is a frequent or occasional requirement
Description	This course teaches new users how to use CATIA Live Functional Tolerancing and Annotation Review to visualize, query, and filter mechanical dimensioning and tolerancing information contained within part and assembly files. Students will learn how to search and examine a part, view annotations and captures, filter and navigate FTA information, and how to use the dimensioning and tolerancing annotations to enhance understanding and improve decision making. The course also features a Master Exercise for live practice.
Objectives	Upon completion of this course you will be able to: - Access and visualize View, Capture and Annotation review features - Show / Hide individual 3D annotations and all annotations of a given type - Display FTA captures - Remove the FTA Clipping Plane of a capture - Filter the 3D annotations

CATIA Live FTA Review Essentials (LFT)

Prerequisites	Students attending this course should have taken the ENOVIA 3DLive Essentials course and be familiar with the Windows Operating System
Available Online	Yes

CATIA Live Shape Essentials (LSE)

Course Code	CAT-en-LSE-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Conceptual Designers, Stylists, Simulation and Manufacturing Engineers
Description	This course will introduce you to CATIA Live Shape and its radically different working environment. You will learn how to use CATIA Live Shape to quickly conceptualize, create, and modify mechanical parts and shapes. The course is process-based and it uses an industrial scenario to teach you how to use the tools in the context of creating a design from conceptual data. It features short-duration demos followed by exercises to allow you to practice using the tools. You will learn the related theory, tips and recommendations while performing the exercises.
Objectives	Upon completion of this course you will be able to: - Create a conceptual design directly in 3D - Use the hybrid design environment to quickly conceptualiz your designs - Work on the structures to create the 3D parts - Navigate the structures and position the parts - Reuse the existing designs in your 3D models
Prerequisites	- Students attending this course should know the fundamentals of CATIA V6 Mechanical and Shape. - They should also be familiar with the Microsoft Windows operating system.

CATIA Live Shape Essentials (LSE)

Available Online	Yes

CATIA Mechanism Simulation Essentials (MSI)

Course Code	CAT-en-MSI-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Engineers
Description	This course will teach you to simulate a mechanism. You will learn how to complete and animate a mechanism, then learn how to define behavior by manually recording an animation and by using laws. You will learn how to include analysis of measurements, interferences, speeds and accelerations. Finally, you will learn how to generate traces, swept volumes and snapshots which can be used when reviewing the simulation results.
Objectives	Upon completion of this course you will be able to: - Complete and animate a mechanism architecture - Create a new mechanism from existing sub-mechanisms - Include dress-up components to complete the mechanism - Create a scenario manually or by using laws - Include measurement and interference analyses - Generate results - Create snapshots for review - Export the final simulation
Prerequisites	Students attending this course should have attended the CATIA V6 Mechanical Design Fundamentals course.

CATIA Mechanism Simulation Essentials (MSI)

Available Online	Yes

CATIA Mold Tooling Essentials (MOT)

Course Code	CAT-en-MOT-F-V6R130
Available Releases	V6R2012x , V6R2013
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Mold Tooling Designers
Description	This course will first teach you how to import data and prepare a Mold Base catalog then how to create the Molded Part from the design part. You will learn how to create the Mold Tool and add additional components from the catalog created previously. Finally, you will create a new version of the original part, compare the two part versions and update the Mold Tool with the new version.
Objectives	Upon completion of this course you will be able to: - Prepare a Mold Tooling catalog - Create a Molded Part from a Design Part - Create and update the Project Unit - Create a completely equipped Mold Tool - Create an Electrode and generate its documentation - Modify the Molded Part and update the Mold Tool
Prerequisites	Students attending this course should have attended the CATIA V6 Mechanical Design Fundamentals and CATIA V6 Advanced courses.
Available Online	Yes

CATIA Plastic Part Design Essentials (PPD)

Course Code	CAT-en-PPD-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Engineers and Plastic Part Designers
Description	This course will teach you how to create a molded plastic part from a set of styled surfaces. You will use functional modeling to integrate basic features on a rough shell before completing the detailed design. Then you will learn how to analyze, prepare, and integrate the styling surfaces. Finally, you will extract the core and cavity for mold tooling design.
Objectives	Upon completion of this course, you will be able to: - Import, analyze, and repair a set of styled surfaces using the Healing Assistant workbench - Reserve space for the components that will be present inside the part - Integrate the Styling surfaces in Functional Design - Use Functional features to create / modify shapes - Manage shell and draft properties and cores - Use external shapes to design in context - Extract the core and cavity models
Prerequisites	Students attending this course should be familiar with the CATIA V6 Mechanical Design Fundamentals and CATIA V6 Mechanical Surface Design courses.
Available Online	Yes

CATIA V5 to V6 Mechanical Design Transition (V6MT)

Course Code	CAT-en-V6MT-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	12 hours
Course Materials	Chinese , English , French , German , Japanese
Level	Fundamental
Audience	Mechanical Designers, CATIA V5 Designers
Description	This course will teach you how to import data and search for models in the CATIA V6 database. You will learn how to perform modifications, check impacts and propagate modifications using a role-based scenario. You will also learn how to design in context, replace components with new versions and analyze a product.
Objectives	Upon completion of this course you will be able to: - Connect to a CATIA V6 database and start a V6 session - Import data into the V6 database - Search for data in the database - Open V6 parts for modification - Propagate the modifications to the database - Manage large assemblies and use assembly-level features - Create and use catalogs - Analyze and review a product
Prerequisites	Students attending this course should be familiar with CATIA V5 Fundamentals and ENOVIA 3D Live Essentials.
Available Online	Yes

CATIA V5-V6 Design Synchronization Essentials (DCE6)

Course Code	CAT-en-DCE6-F-V6R131
Available Releases	V6R2012x , V6R2013x
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 (DCE6)

Prerequisites	Students should be familiar with CATIA V5 Fundamentals and CATIA V6 Mechanical Design Fundamentals.
Available Online	Yes

CATIA V6 Automotive Body Transition (V6VBT)

Course Code	CAT-en-V6VBT-F-V6R130
Available Releases	V6R2012 , V6R2012x , V6R2013
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Automotive Designer, CATIA V5 Designer
Description	This course aims at teaching Automotive Mechanical Designers specializing in Body how to shift from CATIA V5 to CATIA V6. Students will learn how to search for models in the V6 database and how to import existing V5 data. Using a role-based scenario in the context of an assembly they will learn how to design parts in collaboration with other users, perform modifications, check impacts and propagate modifications to the database. They will also see how to manage assembly architecture and contextual links, reuse catalog data, work with large assemblies and analyze the resulting design. Finally, they will practice on a specific master project. The objectives of this project are first to create a part, add it on an assembly and position it with constraints, then to create a new part in the context of an assembly. Finally, students will replace the input curves, manage update errors and create an assembly drawing.
Objectives	Upon completion of this course you will be able to: - Import existing CATIA V5 data and store in V6 - Search for data in the V6 database - Open V6 parts for modification - Share information with other users - Analyze the impacts of modifications - Propagate modifications - Load a product configuration

CATIA V6 Automotive Body Transition (V6VBT)	
	- Use assembly-level features - Analyze a product
Prerequisites	Students attending this course should be familiar with the basics of CATIA V5 (Part Design, Assembly Design, Drafting.)
Available Online	Yes

CATIA V6 Automotive Chassis Transition (V6VCT)

Course Code	CAT-en-V6VCT-F-V6R130
Available Releases	V6R2012 , V6R2012x , V6R2013
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Automotive Designer, CATIA V5 Designer
Description	This course aims at teaching Automotive Mechanical Designers specializing in Chassis how to shift from CATIA V5 to CATIA V6. Students will learn how to search for models in the V6 database and how to import existing V5 data. Using a role-based scenario in the context of an assembly they will learn how to design parts in collaboration with other users, perform modifications, check impacts and propagate modifications to the database. They will also see how to manage assembly architecture and contextual links, reuse catalog data, work with large assemblies and analyze the resulting design. Finally, they will practice on a specific master project. The objectives of this project are to create parts, add the parts to an assembly and position them with constraints, create a contextual part in an assembly, finalize the design intent and create an assembly drawing.
Objectives	Upon completion of this course you will be able to: - Import existing CATIA V5 data and store in V6 - Search for data in the V6 database - Open V6 parts for modification - Share information with other users - Analyze the impacts of modifications - Propagate modifications - Load a product configuration - Use assembly-level features

CATIA V6 Automotive Chassis Transition (V6VCT)	
	- Analyze a product
Prerequisites	Students attending this course should be familiar with the basics of CATIA V5 (Part Design, Assembly Design, Drafting.)
Available Online	Yes

CATIA V6 Automotive Powertrain Transition (V6VPT)	
Course Code	CAT-en-V6VPT-F-V6R130
Available Releases	V6R2012 , V6R2012x , V6R2013
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Automotive Designer, CATIA V5 Designer
Description	This course aims at teaching Automotive Mechanical Designers specializing in Powertrain how to shift from CATIA V5 to CATIA V6. Students will learn how to search for models in the V6 database and how to import existing V5 data. Using a role-based scenario in the context of an assembly they will learn how to design parts in collaboration with other users, perform modifications, check impacts and propagate modifications to the database. They will also see how to manage assembly architecture and contextual links, reuse catalog data, work with large assemblies and analyze the resulting design. Finally, they will practice on a master project focusing on a powertrain assembly. The objectives of this project are to modify a part, add and position it in an existing assembly, finalize its design intent and create the assembly drawing.
Objectives	Upon completion of this course you will be able to: - Import existing CATIA V5 data and store in V6 - Search for data in the V6 database - Open V6 parts for modification - Share information with other users - Analyze the impacts of modifications - Propagate modifications - Load a product configuration - Use assembly-level features

CATIA V6 Automotive Powertrain Transition (V6VPT)	
	- Analyze a product
Prerequisites	Students attending this course should be familiar with the basics of CATIA V5 (Part Design, Assembly Design, Drafting.)
Available Online	Yes

CATIA V6 Collaborative Design Essentials (CDE)

Course Code	CAT-en-CDE-F-V6R131
Available Release	V6R2013x
Duration	4 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Engineers
Description	This course will teach you how collaborate with other users during design and review, using the PLM workplace to manage your contacts. You will learn how to exchange messages and annotated views and how to start a collaborative review session. You will then learn how to exchange data directly using the synchronous mode and how the asynchronous mode can be used to exchange data using a collaborative design workspace. Finally, you will see how to create a collaborative design project and define project tasks.
Objectives	Upon completion of this course you will be able to: - Start a collaborative design session - Create and manage user groups - Exchange messages - Send and receive snapshots - Create and share annotations - Collaborate to review a design - Share and merge 3D geometry - Manage collaborative design projects
Prerequisites	Students attending this course should be familiar with CATIA V6 Fundamentals.
Available Online	Yes

CATIA V6 Mechanical Design Advanced (V6E)

Course Code	CAT-en-V6E-A-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	32 hours
Course Materials	Chinese , English , French , German , Japanese
Level	Advanced
Audience	Mechanial Designers
Description	This course will introduce you to complex modelling techniques. You will learn how to create structured models and complex parts, how to define a product architecture and use it to design in an assembly environment. You will also learn how to manage complex product structures and product configurations, and create part families using parameterized models. Finally, you will learn how to analyze the impacts of design modifications and review a product.
Objectives	Upon completion of this course you will be able to: - Design complex parts - Manage a complex product structure - Design in an assembly environment - Use assembly-level features - Use product configurations - Analyze impacts of modifications - Analyze a product - Review a product
Prerequisites	CATIA V6 Mechanical Design Fundamentals
Available Online	Yes

CATIA V6 Mechanical Design Fundamentals (V6F)	
Course Code	CAT-en-V6F-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	36 hours
Course Materials	Chinese , English , French , German , Japanese
Level	Fundamental
Audience	Mechanical Engineers
Description	This course will teach you how to build parts using feature-based and functional modeling techniques and how to apply design rules in CATIA V6. This course also teaches you how to create a simple assembly, simulate a mechanism, create a rendered image and generate a simple detail drawing.
Objectives	Upon completion of this course you will be able to: - Find documents in the V6 database - Open, explore and save documents - Create mechanical parts - Check parts using existing rules - Create and animate an assembly - Create rendered images - Produce a simple detail drawing
Prerequisites	Students attending this course should be familiar with the basics of ENOVIA 3D Live.
Available Online	Yes

CATIA

CATIA PLM Express V6

V5 to V6 PLM Express Design Transition (V6MTX)

Course Code	CAT-en-V6MTX-F-V6R130
Available Releases	V6R2012 , V6R2012x , V6R2013
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Designers, CATIA V5 Designers
Description	This course will introduce you to PLM Express V6. You will learn how to search for models and import existing V5 data. Using a role-based scenario you will learn how to design in collaboration with other users, perform modifications, check impacts and propagate modifications. You will then learn design in context, replace components with new versions and analyze a product.
Objectives	Upon completion of this course you will be able to: - Import existing CATIA V5 data and store in V6 - Search for data in the V6 database - Open V6 parts for modification - Share information with other users - Propagate modifications - Use assembly-level features - Analyze a product
Prerequisites	Students attending this course should be familiar with CATIA PLM Express Fundamentals.
Available Online	Yes

V6 PLM Express Essentials (V6FX)

Course Code	CAT-en-V6FX-F-V6R130
Available Releases	V6R2012 , V6R2012x , V6R2013
Duration	40 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Engineers
Description	This course will introduce you to PLM Express V6. You will learn how to build parts using feature-based and functional modeling techniques and how to apply design rules. You will be able to collaborate with other users to review designs. Finally, you will be able to create a simple assembly, simulate a mechanism and produce a rendered image and simple detail drawing.
Objectives	Upon completion of this course you will be able to: - Find documents in the V6 database - Open, explore and save documents - Collaborate with other users - Create mechanical parts - Check parts using existing rules - Create and animate an assembly - Create rendered images - Produce a simple detail drawing - Generate and compare Bills of Materials
Prerequisites	Students attending this course should be familiar with the Microsoft Windows Operating System.
Available Online	Yes

CATIA

CATIA Shape V6

CATIA Imagine and Shape Essentials (IMS)

Course Code	CAT-en-IMS-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Shape Designers, Product Stylists, and Industrial Designers
Description	This course will teach you how to use the CATIA V6 Imagine and Shape workbench to create, modify, and improve product shapes and styles. You will learn how to use the Freestyle Sketch Tracer workbench to import stylist's images in V6. You will also learn how to use the Real Time Rendering workbench to create an environment around a model and render it.
Objectives	Upon completion of this course you will be able to: - Import and position sketches in CATIA V6 - Create subdivision surfaces using tools specific to the Imagine and Shape workbench - Modify the style surfaces using Shape Design tools - Create the required environment around a model - Apply materials, textures, and 3D textures to your models
Prerequisites	Students attending this course should be familiar with the fundamentals of CATIA V6 Mechanical and Shape.
Available Online	Yes

CATIA Natural Sketch Essentials (NTS)

Course Code	CAT-en-NTS-F-V6R131
Available Release	V6R2013x
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Industrial Designers
Description	The main objective of CATIA Natural Sketch (NTS) is to Sketch in 3D to express and communicate your creative ideas.
Objectives	Upon completion of this course you will be able to: - Create a 2D sketch - Create a 3D sketch - Create a sketch on a surface
Prerequisites	This course has no prerequisites needed.
Available Online	Yes

CATIA Rendering Essentials (REN)

Course Code	CAT-en-REN-F-V6R120
Available Release	V6R2012
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Industrial Designers and Engineers, Visualization Experts and Team Reviewers.
Description	This course will teach you the concepts of rendering in CATIA workbenches. You will learn how to use the RTR and LRE workbenches to create realistic images. You will learn how to create an ambience, apply materials on an object, tune the viewpoint and render the image
Objectives	Upon completion of this course you will be able to: - Apply materials on an object - Create cameras - Create a scene - Tune a viewpoint - Render and save an image
Prerequisites	Students attending this course should be familiar with the Windows Operating System.
Available Online	Yes

CATIA Reverse Engineering Essentials (REV)

Course Code	CAT-en-REV-F-V6R130
Available Releases	V6R2012 , V6R2012x , V6R2013
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Mechanical Engineers and Shape Designers
Description	This course will teach you how to use the Digitized Shape Editor (DSE) workbench to import and process the digitized data (scans or clouds of points), and how to use Quick Surface Reconstruction (QSR) workbench to create the surface from the digitized data. You will learn how to create a mesh and extract characteristic curves to create surfaces. You will also learn how to use CATIA features in the Reverse Engineering phase to quickly create surfaces using a given point cloud data. The course also provides you with real time industrial examples for your practice.
Objectives	Upon completion of this course you will be able to: - Import and process digitized point cloud data - Create tessellated mesh on the point cloud data - Extract characteristic curves from the data - Create scans from point cloud data - Create curves on mesh data - Create surfaces from curves
Prerequisites	Students attending this course should be familiar with the CATIA V5 to V6 Surface Transition, CATIA V6 Mechanical Design Fundamentals, and CATIA V6 Mechanical Surface Design courses.

CATIA Reverse Engineering Essentials (REV)

Available Online	Yes

CATIA V5 to V6 Mechanical Surface Design Transition (V6ST)

Course Code	CAT-en-V6ST-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	8 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Mechanical Surface Designer
Description	This course will introduce you to CATIA V6. You will learn how to search for models and import existing V5 data. Using a role-based scenario you will learn how to design in collaboration with other users, perform modifications, check impacts and propagate modifications.
Objectives	Upon completion of this course you will be able to: - Import existing CATIA V5 data and store in V6 - Search for data in the V6 database - Open V6 parts for modification - Share information with other users - Analyze the impacts of modifications - Propagate modifications to the database
Prerequisites	Students attending this course should be familiar with CATIA V5 (V5 Fundamentals or Part / Assembly Design, Surface Design and Drafting)
Available Online	Yes

CATIA V6 Icem Shape Design Advanced (IEX)

Course Code	CAT-en-IEX-A-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	12 hours
Course Material	English
Level	Advanced
Audience	Surface Designers who are required to create high-quality surfaces
Description	This course will teach you how to use the advanced surface creation options, the advanced analysis tools, and the Expert tools of CATIA V6 Icem Shape Design. You will learn how to create high-quality surfaces, and analyze and improve the quality of the surfaces.
Objectives	Upon completion of this course you will be able to: - Create high quality surfaces - Analyze surface quality - Correct surface defects
Prerequisites	- Students attending this course should be familiar with CATIA V6 Mechanical Design Fundamentals and CATIA Icem Shape Design Fundamentals. - CATIA V6 Mechanical Surface Design Essentials is also recommended.
Available Online	Yes

CATIA V6 Icem Shape Design Fundamentals (ISH)	
Course Code	CAT-en-ISH-F-V6R131
Available Releases	V6R2013x , V6R2013
Duration	40 hours
Course Material	English
Level	Fundamental
Audience	High quality surface designers
Description	This course will teach you how to use the ISD workbench to create good quality curves and Class A surfaces. You will learn how to analyze the wireframe and surface quality and interpret the results in order to correct visual defects.
Objectives	Upon completion of this course you will be able to: - Create robust class A surface models - Create good quality curves - Assemble, relimit and connect the surfaces smoothly to meet connectivity constraints - Analyze surface quality - Correct surface defects - Manage surface models
Prerequisites	- CATIA V6 Mechanical Design Fundamentals - Some knowledge of Mechanical Surface Design is advisable
Available Online	Yes

CATIA V6 Mechanical Surface Design Essentials (SUR)

Course Code	CAT-en-SUR-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	40 hours
Course Materials	English , French , German , Japanese
Level	Fundamental
Audience	Surface Designers and CATIA V5 Designers
Description	This course will teach you how to use the FreeStyle and Generative Shape Design workbenches to create quality curves and surfaces. You will learn how to analyze the wireframe and surface quality, and rectify the detected defects. You will also learn how to work in a multi-model environment with published surfaces.
Objectives	Upon completion of this course you will be able to: - Understand and use the FreeStyle and the Generative Shape Design workbenches - Create good quality curves and improve the imported wireframe - Create good quality surfaces based on sound wireframe geometry - Assemble, re-limit and connect the surfaces smoothly to get the topology - Analyze the surface quality and heal the defects - Manage surfaces in a multi-model environment
Prerequisites	Students attending this course should be familiar with the CATIA V6 Fundamentals or Part/Assembly Design and Drafting.
Available Online	Yes

CATIA

CATIA Systems / Geensoft V6

CATIA REQTIFY Administration: Analysis Types (RQT)

Course Code	CAT-en-RQT-A-V6R130
Available Releases	V6R2012 , V6R2013
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Future administrators: Quality Engineers, Software Engineers, Project Managers, etc.
Description	REQTIFY provides all the functionalities to exactly fit the methodologies and standards that have to be implemented in an automated requirement management process. Hence, customization of input formalism is key for a complete integration of requirement traceability through REQTIFY. Participants will become confident on all the import topics and will be able to support their internal team.
Objectives	Upon completion of this course you will be able to: - Describe the structure of each element in Reqtify - Write regular expressions - Customize types for ASCII and XML formats - Create Modification types - Customize the tagger
Prerequisites	Students attending this course should have attended the Introduction to CATIA REQTIFY course.
Available Online	Yes

CATIA REQTIFY Administration: Report Generator (RQR)

Course Code	CAT-en-RQR-A-V6R130
Available Releases	V6R2012 , V6R2013
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Quality Engineers, Software Engineers, and Project Managers
Description	REQTIFY provides all the functionalities to exactly fit the methodologies and standards that have to be implemented in an automated requirement management process. Dedicated report generation and specific analysis rules are some mandatory aspects to fulfill commitments in requirements management. Hence, participants will become confident on report generation and will be able to support their internal team.
Objectives	Upon completion of this course, you will be able to: - Understand the global principles of templates customization - Customize the rtf templates - Use the model editor - Understand the OTScript and the language of REQTIFY - Customize simple rules for your project - Manage the customization files to adapt REQTIFY projects to existing processes

CATIA REQTIFY Administration: Report Generator (RQR)

Prerequisites	Students attending this course should have attended the Introduction to CATIA REQTIFY and the CATIA REQTIFY Administration: Analysis Types courses.
Available Online	Yes

CATIA Systems Architecture Design Essentials (SAR)	
Course Code	CAT-en-SAR-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	12 hours
Course Material	English
Level	Fundamental
Audience	Systems Architects, Systems Engineers, Mechanical Designers
Description	This course will teach you the basic concept of the RFLP system design approach. You will learn the creation of a Requirement, Functional, Logical Design, and Physical model. You will also learn about the Implement Relations.
Objectives	Upon completion of this course you will be able to: - Capture the requirements from an MS Word document - Define and formalize data using the RFLP Editor workbench - Create Implement Relations among Requirement, Functional, Logical and Physical entities - Use Search and Navigation tools for RFLP - Generate traceability reports
Prerequisites	- Students attending this course should have attended: - Introduction to Systems Engineering - ENOVIA Requirements Central Essentials
Available Online	Yes

CATIA Systems Dynamic Behavior Modeling Essentials (DBM)

Course Code	CAT-en-DBM-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	20 hours
Course Material	English
Level	Fundamental
Audience	Systems Architects, Systems Engineers and Mechanical Designers
Description	This course will teach you how to model and simulate the dynamic behavior of a multi-engineering system. You will learn how to search, open, and manage the Dynamic Behavior Modeling (DBM) libraries. You will also learn how to manage the link between a logical component and a DBM model, how to add a 3D Representation to the DBM model, and how to simulate the Logic Control Modeling (LCM) and DBM models together.
Objectives	Upon completion of this course you will be able to: - Search and open the Dynamic Behavior Library - Use the DBM workbench to edit and replay an existing model - Create a new DBM model - Create drawings and layers for representation - Attach the DBM model to a logical component - Simulate the DBM model
Prerequisites	Students attending this course should be familiar with the CATIA Systems Architecture Design (SAR) product and the Modelica language.

CATIA Systems Dynamic Behavior Modeling Essentials (DBM)

Available Online	Yes

CATIA Systems Logical 3D Architecture Essentials (TDS)

Course Code	CAT-en-TDS-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	12 hours
Course Material	English
Level	Fundamental
Audience	Systems Architects, Systems Engineers, and Mechanical Designers
Description	This course will teach you how to create the 3D Geometry for the system, to manage the 3D Representation by creating and editing the pathway, associating a logical connection to the pathway, managing the zone and the equipment center. This course will also teach you how to use the knowledge check rules.
Objectives	Upon completion of this course you will be able to: - Add a 3D representation to a logical component - Add a 3D representation to a logical connection - Add a Zone representation - Manage the 2D / 3D representations - Use the knowledge rules to check the clashes, if any, between two 3D components
Prerequisites	- Students attending this course should be familiar with: - CATIA Systems Architecture Design Essentials - CATIA Live Shape Essentials - CATIA V6 Mechanical Design Fundamentals - CATIA V6 Mechanical Surface Design Essentials

CATIA Systems Logical 3D Architecture Essentials (TDS)

Available Online	Yes

CATIA Systems Logical Electrical and Fluidic Design (ELS)

Course Code	CAT-en-ELS-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Electrical Systems Designers and Piping Systems Designers
Description	This course will teach you how to build Electrical Logical Systems and Fluidic Logical Systems. You will learn how to create the different logical components of a system and connect them. You will also learn how to import and export the logical systems.
Objectives	Upon completion of this course you will be able to: - Define electrical logical components - Use the Spreadsheet Editor to create logical components - Use the Live Symbol Editor to create a schematic layout - Manage Connector Ports and Pins - Define Nets and create Net Groups - Manage Electrical Nets and Net Groups - Create and manage Wires and Cables - Associate Nets and Net Groups with Wires and Cables - Define a Harness and manage its Content Links - Import and export Electrical Logical Systems - Create a Fluidic Logical System and its components
Prerequisites	Students attending this course should have attended the CATIA Mechanical Design Fundamentals and the

CATIA Systems Logical Electrical and Fluidic Design (ELS)

	Introduction to Systems Engineering courses. They should also be familiar with electrical systems and piping systems design domains.
Available Online	Yes

Introduction to CATIA REQTIFY (RQS)

Course Code	CAT-en-RQS-F-V6R130
Available Releases	V6R2012 , V6R2013
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Software Engineers, Hardware Engineers, Electronics Engineers and System Engineers.
Description	Although REQTIFY is intuitive and simple to use, it has many capabilities that might not be immediately identified. This course will describe all the basic notions of requirement management and how to implement such a process in a tool-oriented solution like REQTIFY. After this training you will have the opportunity to explore all the REQTIFY features and you will be able to become autonomous on REQTIFY usage. This course belongs to the V2010-1b version of REQTIFY.
Objectives	Upon completion of this course, you will be able to: - Understand the REQTIFY solution - Create a new project - Use the predefined types and templates to generate reports - Use REQTIFY to manage traceability and perform impact analysis
Prerequisites	There are no prerequisites for this course.
Available Online	Yes

Introduction to Systems Engineering (RFLP)

Course Code	CAT-en-RFLP-F-V6R131
Available Releases	V6R2012 , V6R2012x , V6R2013 , V6R2013x
Duration	2 hours
Course Material	English
Level	Fundamental
Audience	Systems Architects, Systems Engineers, Mechanical Designers, Sales Engineers, Managers
Description	This course will introduce you to the System Engineering and RFLP methodology. It will explain the significance of Requirement, Function, Logical Design, and Physical model in the RFLP methodology.
Objectives	Upon completion of this course you will learn to: - Use the Systems Engineering approach to manage the concurrent multi-disciplinary engineering processes - Apply the Requirement, Function and Logical approach to optimize the design process
Prerequisites	Students taking this course should be familiar with Systems Engineering.
Available Online	Yes

Companion

V6 Companion Development Studio

V6 Companion Development Studio Essentials (V6CDS)

Course Code	WLS-en-V6CDS-F-V6R131
Available Releases	V6R2012x , V6R2013x
Duration	12 hours
Course Material	English
Level	Fundamental
Audience	Course Developers
Description	This course will introduce you to the V6 Companion Studio and the various new features that it offers. You will learn how to develop and publish courses using V6 Companion Development Studio and the basics of Companion Learning Space. This is a process based course which highlights the features of Studio while following the ideal process for developing a course. This course also explains the underlying Companion concepts at each stage and relevant tips, notes, and recommendations are included throughout.
Objectives	Upon completion of this course you will be able to: - Create projects and components - Create learning objects (Lessons, Skillets, and Job Aids) - Create a course structure - Use the Rapid Skillet Wizard to create a skillet - Create an assessment - Publish and view the output
Prerequisites	None
Available Online	Yes