Live Learning Series

CATIA V5: R19 & R20

Price: $100 per person, per session

What is the Live Learning Series?

The Live Learning Series is a sequence of live, Expert-Led classes delivered over the Internet on a regular schedule. They are designed to complement RAND Worldwide's classroom learning by providing an in-depth look at a specific subject within CATIA. The topics addressed in each class are based on suggestions submitted by registering students.

Our Live Learning Series enables students to interact with a live, expert instructor who facilitates, instructs, and demonstrates the lesson as well as answers questions and supplies feedback in real time. The instructor executes lessons from their desktop while students participate from their own computer.

Live Learning Series Benefits:

- Continuous improvement of general CATIA skills
- Answer specific questions about a workbench or procedure in CATIA
- Introduce a workbench for which CATIA users have not received classroom training
- Learn tips and tricks to improve modeling performance in CATIA
- Learn new functionality in the latest release of CATIA
- Interact with a live, expert instructor and follow the class from any location with Internet access and the minimum hardware and software requirements
- New classes to be added to the series, existing classes to be repeated throughout the year

Refer to the attached pages for the 2010 Live Learning Series schedule.
R20 Live Learning Series Webclasses begin as of April 29, 2010.

LLS Quick Facts:

- Each WebClass runs 45-60 minutes in length
- WebClasses are a mix of web-delivered lecture and step-by-step demonstration within CATIA V5
- Students receive a PDF copy of a Learning Aid created for each topic. The Learning Aid contains images, menu selections and instructions that enable the student to quickly apply any of the concepts and procedures presented in the class.
- Students are provided with an interactive learning experience, as they are encouraged to participate and ask questions throughout the presentation.
- Prerequisite for all sessions: CATIA: Introduction to Modeling or equivalent

System Requirements

- 56Kbps Internet connection or better
- Netscape 4.x or Internet Explorer 4.x+ (AOL5.0 and above only)
- Separate phone line for the audio
- A Windows PC
- WebEx Meeting Manager
- installed before the event: http://webexevents.webex.com under Set Up

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CATIA V4 to V5 Migration, Part I January 7	This class is useful for any user who works with any CATIA V4 legacy data to be used in CATIA V5. The basics of the class will focus on how to open a variety of different CATIA V4 models in V5. The first part of the class will focus on migrating 2D and 3D data from CATIA V4 to CATIA V5 through the CATIA V2, V3, V4 workbench.
CATIA V4 to V5 Migration, Part II January 14	A continuation from CATIA V4 to V5 migration Part I, Part II will build on the V4 to V5 topic by discussing the CATIA V5 Batch Utility and how it can be used to migrate CATIA V4 data to CATIA V5. The class will conclude by providing students with additional tips and tricks to assist with CATIA V4 to V5 migration.
Customizing the Catia Interface January 21	This class provides users with an introduction of how to customize their CATIA V5 Release 19 interface. The main areas of interest to be covered include Toolbar customization, Macros, customizing .CATSettings, and other tips and tricks within the CATIA V5 Release 19 interface.
File Management Techniques January 28	Learn the basics of CATIA V5 file management techniques. Students will be able to differentiate between the different techniques used to open, create, and store data. Students will also learn how to manage and edit the links of their CATIA V5 files.
Generative Sheetmetal Design, Part I February 4	If you have ever wondered how to create a sheet metal part using CATIA V5, this presentation is for you! The introduction and overview of the Generative Sheetmetal Design workbench will be followed by a comprehensive process based approach to creating your first sheet metal part. Various sheet metal specific features and best practices will also be covered.
Generative Sheetmetal Design, Part II February 11	This class picks up where Part I of Sheet Metal Design finishes. Now that you know how to begin the creation of a sheet metal part using CATIA V5, learn how to add sheet metal features resulting from forms, punches and dies to complete your sheet metal part.
Introduction to 3DVIA Composer February 18	3DVIA Composer offers a variety of tools for generating technical illustrations and work instructions linked to CATIA V5 designs. This class introduces users to the 3DVIA Composer interface, creation and updating of design views and basic animations.

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Designing with Boolean Operations February 25	CATIA V5 is a feature-based solid modeler, yet also allows boolean operations to be performed. This powerful combination allows boolean operations to be performed on solid bodies developed from feature-based solid modeling. This design technique is ideal when designing molded parts and parts machined from castings.
Generative Sheetmetal Parameters March 4	Modeling parts in the Generative Sheetmetal Design workbench provides users with full formed and flattened parts, but how do users accurately know how much material is required to fabricate their models? This lesson covers creating sheetmetal parameters and customizing bend allowance and bend radius tables to account for the bend deformation.
Overview of NC Infrastructure March 11	CATIA V5 offers a whole suite of manufacturing (CAM) tools used to create tool paths. This class provides CATIA NC users, programmers, and managers an overview of the CATIA NC suite of workbenches. Users will learn how to navigate through the different NC workbenches, interrogate the NC models, and replay and analyze tool path data.
Introduction to Generative Drafting March 18	CATIA V5 is not only a 3D modeler, but also offers a 2D drafting solution. The Generative Drafting workbench allows designers to directly transfer 3D model geometry of parts and assemblies into parametric 2D views. This course will give an overview of the drawing creation process, including: creating views, detailing, adding tables, and selecting and applying formats.
Bill of Materials and Balloons March 25	The Generative Drafting and Assembly Design workbenches provide users with tools to create an assembly Bill of Materials (BOM) and assign balloons to each BOM item. This presentation will cover Bill of Materials functionality, customization through standards, and how to populate a drawing with the BOM table and balloons.
CATIA Administration Mode and Standards April 1	The CATIA Administration Mode and Standards presentation will provide students with tools to help administer a company-wide environment including defining standard CATSettings and Drafting Standards. The course will provide students steps to modify and define their own company drafting standards.

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Introduction to ENOVIA SmarTeam for Users April 8	ENOVIA SmarTeam allows user enhanced access to company data, even allowing for CATIA V5 integration over the web. This session will introduce users to the Smarteam interface, interacting with Smarteam from within Catia plus tips and tricks for day to day use.
Wireframe Elements April 15	Wireframe elements are important “building blocks” in creating support geometry for solid and surface features. This session will introduce wireframe elements available to a designer using the Wire Frame & Surface Design and Generative Shape Design workbenches. The class begins with an overview of the various types of wireframe elements, methods of creation and typical uses. Organization of Open Bodies, best practices, and analysis will also be covered.
Introduction to Surfaces April 22	Surface features are non-solid features, created using the Wire Frame & Surface Design and/or Generative Shape Design workbenches. Surface features are invaluable when modeling non-prismatic geometry, or manipulating 3D solid geometry. Various surface feature types, creation and design techniques will be covered.
What’s new in CATIA V5 R20 April 29	This class covers some of the major enhancements introduced to CATIA V5 Release 20 including the Mechanical Design Workbenches, Part, Product, Wireframe & Surface, and Drafting.
Customizing NC Tool Tables May 6	CATIA NC offers programmers a wide range of functions to creating NC tooling and tooling libraries in the CATIA NC environment. This session will provide users with an overview on how to create tools & holders, review tooling parameters, create and manage tooling catalogs, and custom tool design and usage.
Large Assembly Management May 13	As the design phase progresses, file size becomes an issue. System performance decreases and ‘normal’ operations become more time consuming. The Large Assembly Management class offers students tools and techniques to help improve the performance of CATIA V5 when working with and manipulating large assembly files. Topics include cache management, introduction to scenes, and visualization tools.

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Working with Scenes May 20	CATIA V5 has the ability to develop large assemblies with tens, hundreds, or even thousands of components. However, often the size of an assembly (number of components, file size) can greatly reduce the performance of the design project. This course offers users the ability to simplify the display of an assembly, reduce demands on the system, and allow designers to work with greater efficiency.
Generative Structural Analysis, Part I May 27	Throughout a typical design cycle, a part can be volleyed back and forth between the designer and the analyst many times, resulting in lengthy development times. CATIA V5 offers an engineering solution with its Generative Structural Analysis workbench that allows a designer to perform structural simulations throughout the design stage of a product, resulting in a structurally sound design ready to be sent to an analyst. This session will not only give an overview of the workbench, but will cover how to set-up models with constraints, forces, materials, and engineering analysis conditions.
Generative Structural Analysis, Part II June 3	CATIA V5 offers an engineering solution with its Generative Structural Analysis workbench that allows a designer to perform structural simulations throughout the design stage of a product, thereby resulting in a structurally sound design ready to be sent to an analyst. This session continues building upon the topics listed from Part I of Generative Structural Analysis.
Introduction to Tube Design June 10	The Tube Design workbench contains tools to assist in creation and mangement of tube systems according to industry standards. This class contains an overview of the Tube workbench as well as various tools associated with Routing and Object Catalogs
Skeleton Modeling Techniques June 17	Skeleton modeling is an advanced assembly modeling technique. This top-down design methodology is a very powerful way of capturing assembly design intent by locating critical design information at a high assembly level. Skeleton models are also used to define a space claim for parts or assemblies when working in the context of other components.

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Design In Context June 24	Learn how CATIA V5 promotes concurrent engineering and collaborative design through the use of designing in context methodologies. Designing in context is a powerful top down design approach that enables design engineers to create interfacing geometry quickly and accurately. Creation of external references and links management will be covered and followed by best practices.
Introduction to ENOVIA DMU Navigator July 1	CATIA V5 offers incredible assembly design functionality. The CATIA V5 DMU workbenches take assembly design further by offering users the ability to simulate how products are assembled together and perform digital measurement analysis, as well as analyze assembly motion to simulate the assembly function. An excellent tool useful for project design managers through to shop floor personnel. This session will introduce students to the DMU Navigator workbench and the standard tools of the workbench.
Formulas, Parameters, and Measures July 8	CATIA V5 includes tools to provide users with the ability to drive design intent and company standards through the use of model parameters, formulas, and measures. This Live Learning Series class offers students an overview of how to create measures and parameters, and then automate their design through the creation and manipulation of formulas.
Design Tables and Catalogs July 15	Use Design Tables in CATIA V5 to create model catalogs and instances. Students will learn how to work with Design Tables to vary dimensions, parameters, features and components. Catalogs are organizations' tools within the CATIA V5 environment. Students will also receive an introduction to the CATIA V5 Catalog Workbench.
ENOVIA DMU Kinematics, Part I July 22	DMU (Digital Mock-Up) Kinematics Simulator is a CAD-independent CATIA/ ENOVIA V5 workbench that offers a series of tools to perform detailed assembly motion simulations. This class covers an introduction to Kinematics Simulator and introduces constraint based joints. There will also be an introduction to the simulations functionality of Kinematics Simulator.

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ENOVIA DMU Kinematics, Part II July 29	DMU (Digital Mock-Up) Kinematics Simulator is a CAD-independent CATIA/ ENOVIA V5 workbench that offers a series of tools to perform detailed assembly motion simulations. This class builds on the DMU Kinematics Part I class by introducing more advanced functionality like curve and surface based joints, ration based joints, along with converting constraints to joints. There will also be an introduction to using multiple mechanisms to create a single product.
Introduction to Digitized Shape Editor August 5	Point cloud data isn't ready to start modeling. The Digitized Shape Editor workbench and this lesson will introduce the tools used to filter, manipulate, and merge point cloud data in order to prepare the points for wireframe and surface generation.
Introduction to Quick Surface Reconstruction August 12	The Quick Surface Reconstruction workbench provides users the ability to quickly define curve and surface geometry from imported point cloud data. This class will introduce students to the basic tools of the Quick Surface Reconstruction workbench.
NC Toolpath Automation August 19	The CATIA Manufacturing workbenches provide programmers many options to generate tool paths. There are often times when programmers come across similar shapes or models to be machined. This lesson covers the basics of re-using and grouping machining operations and theory through catalogs to then be re-used with only a few clicks of the mouse.
CATIA V5 Batch Utility August 26	The CATIA V5 Batch Utility provides users a variety of tools to automate the export of multiple CATIA V5 files through a single action. This presentation will provide information to create and run batch programs for data exchange, CATDUAV5, plotting through batch, and Downward Compatibility.
Assembly Design Tools September 2	The Assembly Design workbench provides tools to assemble components to develop the desired product. Additional tools frequently used within the Assembly Design workbench included in this presentation are Assembly Features, Flexible Sub-Assemblies, and duplication tools.

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Introduction to Photo Studio September 9	The CATIA V5 Photo Studio workbench a tool for users who want to generate more realistic images for presentations without resorting to a full-scale Rendering tool. This class will cover tasks such as Cameras, Lights and Material selections as well as Stickers, Animations and basic Rendering
ENOVIA DMU Fitting Simulator September 16	DMU (Digital Mock-Up) Fitting Simulator is a CAD-independent CATIA/ENOVIA workbench that offers a series of tools to simulate the motion of parts during the assembly or disassembly operations. The DMU Fitting Simulator workbench offers tools to record video and to review the path and space occupied by the part motions.
Introduction to Imagine and Shape September 23	The Imagine and Shape workbench provides users the ability to quickly convert a conceptual design to 3D through the use of 3D primitive shapes and sculpting tools, all the while remaining in the CATIA V5 environment. Users can quickly swap between the Sketch Tracer, Part Design, and other part level workbenches to continue with the design development. This class will provide users with an introduction to the Imagine and Shape workbench and show users how to perform basic tasks within the workbench.
Overview of Hybrid Design September 30	Hybrid design provides users of CATIA V5 R14 (and above), a new modeling methodology for creating solid and non-solid geometry models. This class will provide an overview of the new Hybrid design functionality, including pros and cons of the functionality, and suggested techniques of this new design theory.
Introduction to Functional Molded Part October 7	The Functional Molded Part workbench offers users a new way of developing 3D part models in the CATIA V5 environment. The workbench methodology focuses more on the ‘functional goals’ and ‘design constraints’ of the final product versus individual feature creation used in normal Part Design methods. This new methodology results in less geometry being created to arrive at the final design. This presentation will provide students with an overview of the Functional Molded Part workbench, review the standard tools of the workbench, and go over the basic design process within the workbench.

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Opposite Hand Modeling October 14	Often times when modeling a part in CATIA, designers are required to create a mirrored copy of the part to be assembled to the opposite side of an assembly. This class will provide students with different time saving methodologies and best practices for designers to follow in order to quickly duplicate an opposite hand model of the original design.
Introduction to Functional Tolerancing and Annotation October 21	The 3D Functional Tolerancing & Annotations workbench of CATIA V5 offers engineers the ability to document tolerances and notes directly on 3D parts and assemblies. Having critical tolerancing information on the 3D models saves engineers' time from having to reference or rely on 2D drawings. This class introduces students to the basics of creating geometrical tolerances and annotations directly on 3D models using the 3D Functional Tolerancing & Annotations workbench.
Knowledge Templates October 28	This class covers the creation and usage of PowerCopies which are very powerful when you need to replicate data many times, in the same or different models. The use of PowerCopies increases designer efficiency and promotes company standards.
Introduction to Multi-Axis Flank Contour & Multi-Axis Flank Pocket Machining for CATIA V5 NC November 4	Thin and/or drafted walls present many challenges when designing manufacturing programs. This class will introduce CATIA V5 NC users and programmers to machining operations such as Multi-Axis Flank Contouring and Multi-Axis Flank Pocketing, in addition to other techniques, useful in coping with these scenarios
Designing with Fillets November 11	Fillets or Rounds are an essential part of any design. This class will introduce students to the basics of fillet creation in CATIA V5. Topics such as advanced fillet creation and fillet techniques and tips are also covered in this lesson.
Introduction to Healing Assistant, Part I November 18	The Healing Assistant workbench offers users tools to analyze, and work on the geometry and topology of surface geometry. This session provides students with an overview of the Healing Assistant workbench and provides users with the basic process to follow in order to repair surface geometry.
Thanksgiving Day	

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Introduction to Healing Assistant, Part II December 2	The Healing Assistant workbench offers users tools to analyze and work on the geometry and topology of surface geometry. This session builds upon the information gained from Part I. Users will learn advanced techniques to 'heal' native CATIA V5 surface, and imported surface geometry, and how to create a solid from the resulting 'healed' model.
Feature Duplication Tools December 9	Duplication of features enables designers to reuse existing features quickly and accurately, increasing design efficiency and maintaining design intent. Functionality covered will include various pattern types and options, transformation tools, and copy and paste techniques.
Part Design Feature Recognition December 16	Have you ever imported a 3D solid and wished you had the ability to modify a feature and have the model update, rather than having to recreate the feature? The Part Design Feature Recognition workbench may be your answer! This workbench provides users with the ability to automatically create/convert unintelligent data to native CATIA V5 features by selecting on the 3D solid. This class will introduce students to this useful tool for those who work with imported STEP, CATIA V4 solids, and other non-parametric solids.