Software Announcement
November 8, 2005

CATIA V5.16 accelerates collaborative product design

Overview

Process-centric: V5.16 accelerates product development processes such as relational design and optimization with 64-bit support of the CATIA portfolio on Microsoft™ Windows™. Taking full advantage of superior 64-bit memory capacity, V5.16 empowers customers to create and analyze even the largest and most complex products.

V5.16 extends CATIA Machining Solution′s leadership by integrating proven DELMIA simulation technologies through ISO code-based, realistic NC machine and material-removal simulation, reducing lead-time from programming to production.

V5.16 strengthens virtual product simulation, enabling customers to experience and validate their products. In particular, V5.16 improves the analysis assembly process, further promoting concurrent engineering across the extended enterprise.

Collaborative workspaces: V5.16 promotes collaboration between business partners working with ENOVIA or SMARTEAM. It fosters relational design across the supply chain with the new SMARTEAM Reconciliator, facilitating company interaction for concurrent engineering. V5.16 expands the usefulness of the open and extremely lightweight 3D XML format, designed specifically for fast and efficient 3D communication about the virtual product, with additional Product Lifecycle Management (PLM) product information.

V5.16 enables 3D-only paperless engineering design to manufacturing planning. Significant improvements in the ability to capture and share engineering requirements within the 3D model enable the full consumption of the design intent, further ensuring final product compliance to engineering specifications

Product, process, resource (PPR): V5.16 streamlines product development by strengthening the single desktop for designers to search and navigate, to communicate and collaborate, and to control and propagate. In V5.16, engineers can seamlessly manage and share more advanced design process information with the VPM Navigator, further facilitating formalized concurrent work processes.

Knowledge: Resulting from collaboration with industry leaders, V5.16 makes it easy to interactively capture and reuse corporate business process knowledge. This release further facilitates the use of engineering process knowledge using the Business Process Knowledge Template application, further increasing productivity in design generation.

Component Application Architecture (CAA) V5: V5.16 provides the only end-to-end, fully-integrated composites solution on the market. Third-party market leaders have chosen the powerful and flexible open V5 architecture to develop highly-specialized applications for composites parts manufacturing, complementing the CATIA V5 offering.

Key prerequisites

Refer to the **Hardware requirements** and **Software requirements** sections.

Planned availability dates

- December 2, 2005, CATIA V5.16 Solutions
- January 13, 2006, CATIA Web-based Learning Solutions V5.16

At a glance

CATIA V5.16:

- Accelerates product development with 64-bit support of the CATIA portfolio on Microsoft Windows.
- Promotes collaboration with suppliers with the new SMARTEAM Reconciliator.
- Expands the scope of the open and extremely lightweight 3D XML format by including additional PLM product information.
- Enables 3D-only paperless design-to-manufacturing, with significant improvements in the ability to capture and share engineering requirements.
- Extends CATIA Machining Solution′s technology leadership by integrating proven DELMIA simulation technology through ISO code-based, realistic NC machine simulation.
- Streamlines product development with the VPM Navigator by enabling seamless management and sharing of more advanced design process information.
- Provides the only end-to-end, fully integrated composites solution on the market with highly specialized composites parts manufacturing applications from market leaders to complement the CATIA V5 offering.

Description

Process-centric

V5.16 accelerates product development processes such as relational design and optimization, with 64-bit support of the CATIA portfolio on Microsoft Windows. Taking full advantage of superior 64-bit memory capacity, V5.16 frees customers to create and analyze even the largest and most complex products.

- 64-bit provides tremendous capacity for PLM users dealing with large amounts of data. It facilitates relational design methodology by offering a new way of doing daily design work.

  In complex automotive plastic parts design, for instance, a car dashboard is usually cut into several parts, which are then worked on by different users. This approach requires extra time to reconcile the parts. With CATIA V5 64-bit, one user can work on the whole dashboard, avoiding unnecessary duplication. This allows designers to fully benefit from V5, and particularly the *specification/result* mode, resulting in dramatic productivity gains.

- 64-bit support also enables unmatched performance in computationally demanding activities such as finite element analysis (FEA), which requires considerable memory capacity to correctly handle large amounts of computed data. With this extra memory space, it becomes possible to address the full analysis process in one operation.
- CATIA V5 32-bit users can also benefit from the 64-bit Windows underlying architecture. CATIA 32-bit processes can allocate 4 GB of memory space instead of 3 GB on Windows XP 32-bit.

In V5.16, proven DELMIA simulation technology leverages CATIA Machining Solution leadership by integrating ISO code-based realistic NC machine and material-removal simulation.

- The new CATIA — NC Machine Tool Simulation 2 product (MSG) enables NC machine ISO code-validation at an early stage avoiding problems and modification loops. This realistic virtual simulation enables NC programmers to validate that the part will be machined correctly the first time, thereby reducing lead-time from programming to production.

  Fully integrated in the V5 architecture, this seamless end-to-end solution enables NC programmers to easily switch between toolpath definition and toolpath validation without losing time due to data transfer or preparation. This eliminates interface issues and significantly increases cost effectiveness.

- The new CATIA — NC Machine Tool Builder 2 product (MBG) offers a unique environment for machine tool definition (with geometry, kinematics and technological information) as well as peripheral resources (tool and pallet changers). The definition of the NC resources can be used in the entire machining process, including NC planning and detailing, simulation and verification, post processing, and controller emulation, thus avoiding redundant machine tool data.

V5.16 strengthens virtual product simulation, enabling customers to experience and validate their products.

- It improves the analysis assembly process. To more accurately predict product behavior, V5.16 provides OEMs with a new load case management capability, enabling them to drive all preprocessing operations. This function enables OEMs to simulate all interactions in the product environment within the same analysis case. Suppliers and OEMs can transfer complete elementary preprocessing and post-processing analysis features to their partners, thereby increasing collaboration and further promoting more efficient concurrent engineering across the extended enterprise.
- The CATIA — Real Time Rendering product underpins the new nVIDIA shaders technology to deliver outstandingly realistic rendering that combines high-quality, real-time advanced effects on complex materials such as metallic car paint, varnished woods, and chrome materials. This enables early decision-making based on product appearance, reducing the need for expensive physical prototypes.

  Zebra mapping geometrical analysis also benefits from this new shaders technological breakthrough to provide real-time, high-end surface quality analysis, resulting in a shortened free-form analysis cycle.

- Further ensuring the safety and comfort of workers and users, V5.16 reinforces the prevention of troublesome postures using enhanced Rapid Upper Limb Assessment (RULA), as well as providing new back posture and load analysis. In addition, the new Human Ergonomic Analysis report enables users to consolidate a single view of all human analyses, ensuring an accurate evaluation of the project.
- Fluent and Mecalog reinforce the V5 simulation offering with two new products. The Fluent for CATIA product fully integrates Computational Fluid Dynamics (CFD) into the PLM process. In addition, MCrashV5 from Mecalog builds upon V5 analysis features such as meshes and material properties to seamlessly simulate crashes within a V5 environment in order to validate vehicle safety and reduce the need for costly physical crash testing.

V5.16 extends CATIA's unique position in 3D conceptual design, thereby facilitating product innovation, with comprehensive visualization management tools and more productive creation of skeleton geometry.

- V5.16 simplifies view background visualization with the 2D Layout for 3D Design product. It enables a 3D box definition that clips the representation of a view background using a clipping frame or a back-clipping plane. Moreover, additional filtering capabilities (such as display or mask filters, granular filtering of additional 3D elements, filtering objects from direct selection) and user-interface improvements (customize view creation process with filtering capabilities) facilitate conceptual design when working on complex parts.
- The new 2D mode facilitates the visualization and the manipulation of elements lying on a defined plane. It enables easy complex-part understanding in the conceptual design phase.
- In V5.16, the adaptive sweep function offers a powerful option to build one single complex surface based on a set of unconnected sketch profiles, accelerating the creation of automotive conceptual design surfaces.

V5.16 further consolidates the shipbuilding offering with a unique way to resolve physical design challenges. In V5.16, additional capabilities in the CATIA — Ship Structure Detail Design product (SDD) complement the CATIA — Structure Functional Design product (SFD) to deliver unprecedented levels of ship design productivity.

- SDD enables the automatic generation of blocks and design units based on planning break definitions from SFD.
- It completes the end-to-end ship design process by providing automation of detail work (slot or end-cut definition).
- At the end of the design phase, SDD allows piece-part generation for manufacturing preparation.

V5.16 consolidates the reverse engineering suite of products, enabling companies from the automotive, electronics, and consumer goods industries to take the best advantage of their assets and create robust and innovative products in a short cycle. The new automatic capture of physical specifications capability delivered by the CATIA — Quick Surfaces Reconstruction product enables an immediate reconstruction of complex surfaces from clouds and meshes in one operation. This increases shape creation productivity and makes reverse engineering easy.

V5.16 advances the adoption of the revolutionary functional modeling technology to serve productivity and innovation.

- This unique approach, which brings proven added-value to molded parts design, may also be extended to other design processes — in the aerospace industry, for instance.
- V5.16 introduces, in particular, a major change to a unified specification tree (Functional Body and Functional Solid become the Solid Functional Set).
- It significantly improves specification tree readability, especially in multi-body operations (divide, core and cavity extraction) and therefore fully organizes and manages the specification tree for better understanding of products.

V5.16 delivers to electrical, aerospace, and automotive OEM designers and their suppliers a consolidated 3D harness design toolset.

- Improved PLM management of shared supports refines the routing of 3D aerospace electrical systems.
- V5.16 continues to enhance design quality by enriching specialized harness design knowledge rules.
- New devices, such as multi-insert connectors, improve the definition of components and contribute to providing an accurate electrical solution to the automotive and aerospace industries.

Collaborative workspaces

V5.16 promotes unequalled collaboration between business partners working with ENOVIA or SMARTEAM. It fosters relational design across the supply chain with the new SMARTEAM Reconciliator, facilitating company interactions as well as concurrent engineering.

- The new SMARTEAM — CATIA Supply Chain Engineering Exchange (SEE) product promotes bidirectional exchanges of engineering packages between business partners. With the reconciliator, designers visualize, compare and select only relevant design modifications that have been received, and reconcile them within SMARTEAM.
- In addition, V5.16 fully supports global product simulation across the extended enterprise. It enables OEMs to reconcile analyses from suppliers through the ENOVIA Reconciliator, fostering collaboration for analysis processes.

V5.16 expands the usefulness of the open and extremely lightweight 3D XML format--designed specifically for fast and efficient 3D communication about the virtual product throughout the extended enterprise--with additional PLM product information such as annotations and animations.

- V5.16 delivers a richer 3D XML file format for all users to maximize use in downstream applications by adding PLM product data information to the 3D geometry. 3D annotations, along with functional tolerancing, provide significant benefits to the entire manufacturing process, while measurements and sections are key to technical publication applications.

  Animation from Fitting, Kinematics, and the DMU Navigator is also available as additional product information that can be put into 3D XML. Also in this release, 3D XML provides a full XML tessellated geometry description that further reinforces the openness of the 3D XML format.

- In addition to the full support of Microsoft Office applications, the 3D XML player is tightly integrated into IBM Lotus Notes®, facilitating easy 3D collaboration across the whole enterprise.

V5.16 enables 3D-only paperless engineering design to manufacturing planning and execution. Significant improvement in the ability to capture and share design intent and engineering specifications within the 3D model enables the full consumption of the design intent and eliminate the need for 2D drawings, further ensuring final product compliance to engineering specifications.

- V5.16 completes the definition of engineering requirements for manufacturing and enables all of them, including specifications, tolerances, flagnotes, standard parts, and so on, to be directly usable and reconcilable within DELMIA manufacturing process planning. Design engineers enrich 3D annotations with a “consumable” property, while DELMIA enables manufacturing engineers to partially or completely “consume” the intent of the design engineer with respect to annotated features and tolerances defined at a particular step in the process plan. This allows configured manufacturing planning evolution without forcing corresponding engineering changes.
- In parallel, V5.16 consolidates the unique 3D master approach with improved 3D annotation presentation. This facilitates information access and product definition understanding.
- The ability to get full technological content (technological results), not only geometry, through **Copy paste as result with link** facilitates manufacturing operations definition, such as drilling, on a final complex part.

V5.16 fosters concurrent engineering and communication with suppliers for tooling design. New link management and relationship analysis between sub-assemblies enable several designers to easily work on the same tooling, thus leading to significant lead-time reduction. Additionally, the new technological results capability enables a tighter integration between tooling design and downstream applications, such as drafting and machining. As a result, companies no longer need to send all of their part specifications in order to give sub-contractors access to components tooling information, further promoting intellectual property protection.

Product, process, resource (PPR)

V5.16 streamlines product development by improving the single desktop, which allows designers to search and navigate, communicate and collaborate, and control and propagate.

In V5.16, engineers can seamlessly manage and share more advanced design process information with the VPM Navigator, such as part maturity transition, and part transfer ownership, further facilitating formalized concurrent work processes and enterprise-wide communication.

The VPM Navigator strengthens its unique position to manage scalable relational designs within the most complex products. Thanks to the impact graph, engineers navigate directly through the rich content of the product and its relationships, accelerating the decision-making process for modifications. In V5.16, navigation of the relationships within the impact graph becomes easier and faster, with more natural and productive graph expansion and impact analyses. In addition, designers can identify at a glance the different types of links, thanks to the impact graph's enhanced display, and they benefit from new filtering capabilities for a comprehensive and efficient view of relationships within complex products.

Knowledge

As a result of working together with aerospace and automotive industry leaders, V5.16 promotes the reuse of company business process knowledge across the enterprise using the Business Knowledge Template application. It facilities the use of engineering process knowledge, further increasing productivity in design generation.

- V5.16 further consolidates the Business Process Knowledge Template (BKT) offering for the automotive and aerospace industries to best match their needs. V5.16 maximizes automated design and generative engineering tasks.
- V5.16 delivers a better synergy between language and interactive tasks for efficient management of low-level and high-level applications logic.
  - The BKT solution provides a "codeless" alternative to non-software engineers for developing knowledge applications. As a result, it reduces application development cycle time and maintenance costs.
  - In V5.16, BKT provides industrial management of knowledge applications by connecting the knowledge lifecycle application structure to development tools, such as the Component Application Architecture Rapid Application Development Environment (CAA RADE).

V5.16 expands generative design and morphing capabilities with the Product Knowledge Template product.

- V5.16 enables user-defined feature reference and instance modification.
- V5.16 eases deployment of user-defined features (UDFs). Users can now modify a UDF after creation, which provides more flexibility while ensuring protection when exchanging data, thanks to the new protected mode.
- In addition, this release improves morphing with the new knowledge pattern to manage the UDF lifecycle (creation, modification, deletion).
- The ability to define meta inputs eases template instantiation. Meta inputs improve template generation productivity and make them easier to control. Meta inputs also enhance the content of the template.

Component Application Architecture (CAA) V5

V5.16 delivers the only end-to-end, fully-integrated composites solution on the market. Our market-leading partners have chosen the powerful and flexible open V5 architecture to develop highly-specialized applications for composites parts manufacturing. These applications complement the CATIA V5 offering already in use at major aerospace manufacturers for developing largely composites-based new-generation aircraft.

- Two major manufacturers of machines for composites tape laying and fiber placement, Cincinnati Lamb and Ingersoll, join the CAA adopter community to extend the range of composites machine support.
- Airbus Cimpa, Majestic Systems, Inc., ESI Group, and MTorres deliver a complete offering based on V5 that addresses all composites manufacturing techniques, ranging from nesting, cutting, and tape laying programming to Resin Transfer Modeling (RTM).
- In V5.16, CATIA composites design products deliver to designers fully associative zone and ply design, as well as an enhanced ply exploder capability for unmatched digital mock-up (DMU) review, which enables early validation of the most complex designs.

In addition to composites design, this release strengthens the Aerospace Sheetmetal Design solution for high-end aerospace part modeling and manufacturing. Driven by aerospace manufacturers, the solution delivers significant enhancements that facilitate the sheetmetal designer's daily work, enhanced stamping capabilities, and new aerospace-specific drafting capabilities. Combined with the two new V5 specialized products for aerospace sheetmetal manufacturing preparation and nesting from Airbus Cimpa, V5.16 brings to the aerospace industry a key process-oriented, design-to-manufacturing solution with an unequalled ability to ensure conformance to company standards.

Based on the proven V5 CAA architecture, ICEM Shape Design enables the capture of the conceptual design to build up high-quality, visible Class-A surfaces. Together with the CATIA V5 Shape Design and Styling offering, they deliver a unique and fully-integrated automotive Class-A solution. By enabling a seamless workflow between conceptual and engineering design, this solution brings major productivity gains to the entire Class-A process.

Key PLM market leaders continue to adopt the V5 architecture. Since the introduction of CATIA V5.15, two new partnerships have been announced.

- Platform Computing, a leading grid-computing software vendor for industrial organizations that require computing-intensive resources to drive its virtual product development processes.
- Mtorres, a major machine tool manufacturer for composites parts. Mtorres is delivering a V5-based tape laying application for optimizing the manufacturing process.

More than 40 new, highly-specialized applications from V5 partners have been introduced since V5.15, extending V5 solution coverage.

Accessibility by people with disabilities

Owing to the graphics-intensive nature of its engineering design applications, this product has been granted a deviation for 2005.

Product positioning

CATIA V5 is the leading product development solution for all manufacturing organizations, from OEMs, through their supply chains, to small independent producers. The range of CATIA V5 capabilities allows application in a wide variety of industries, such as aerospace, automotive, industrial machinery, electrical, electronics, shipbuilding, plant design, and consumer goods, including design for such diverse products as jewelry and clothing.

CATIA V5 is the only solution capable of addressing the complete product development process, from product concept specification through product-in-service, in a fully integrated and associative manner. Based on an open, scalable architecture, it facilitates true collaborative engineering across the multidisciplinary extended enterprise, including style and form design, mechanical design and equipment and systems engineering, managing digital mock-ups, machining, analysis, and simulation. By enabling enterprises to reuse product design knowledge and accelerate development cycles, CATIA V5 helps companies speed-up their responses to market needs.

In conjunction with ENOVIA and SMARTEAM for lifecycle management and decision support and DELMIA for manufacturing engineering, CATIA V5 is a key component of collaborative PLM.

Much beyond pure CAD software packages, which provide geometry modeling features for design-centric companies, CATIA V5 delivers the keys to PLM for process-centric companies:

1. **Product to market.** CATIA is about product creation. From the earliest product concept to production tooling, its concurrent engineering and design-in-context capabilities create value by enabling companies to create products and bring them to the market.
2. **Time to market.** The unequalled process coverage of CATIA, combined with the native associativity among all of its applications, gives CATIA customers the means to shorten the time to market.
3. **Right to market.** CATIA′s integrated analysis, simulation, synthesis, and optimization applications provide product engineering validation at each design step to ensure product quality and market acceptance.
4. **Lead the market.** CATIA′s advanced capabilities for collaborative engineering, knowledge capture, and re-use boost innovation and help to lead the market.

The three CATIA V5 platforms (P1, P2, and P3) make it the most scalable solution for product creation. CATIA V5 P1 users benefit from PLM productivity in an affordable way with the security of potential growth. They can conduct associative product engineering based on CATIA V5 product design-in-context, product knowledge reuse, end-to-end associativity, product validation, and collaborative change management capabilities.

CATIA V5 P2 users can optimize their PLM processes through knowledge integration, process accelerators, and customized tools. They can drive generative product engineering based on the largest application portfolio. They can perform “design-to-target,” extensively capture and re-use knowledge, and stimulate collaborative innovation.

CATIA V5 P3 users access the highest productivity for specific advanced processes with focused solutions. They can lead expert engineering and advanced innovation, relying on unique and very specialized applications that integrate product and process expertise.

The CATIA V5 world is fully scalable because the P1, P2, and P3 platforms operate on the same data model, methodology, and management schemes. By enabling users to install P2 applications on top of P1 seats and by standardizing graphical user interfaces across platforms, P1 users can immediately do collaborative design of their extended enterprise with P2 users and later grow with P2 and P3 platforms as their business grows.

Product positioning — Globalization

CATIA V5 program integrated information (PII) and softcopy product information are available in:

- French
- German
- Italian
- Japanese

PII is stored in compressed unicode format (UTF-8) for world-wide exchange without the loss or misinterpretation of characters.

Hardware and software support services

SmoothStart™ /installation services

IBM SmoothStart/Installation Services are not available CATIA products.

CATIA V5 services plan

PLM Services, part of IBM Global Services, offers a robust portfolio of services to assist with the implementation of CATIA V5. Careful planning and implementation are essential to getting the most from CATIA V5. IBM PLM Services provides an Implementation for CATIA V5 Services Offering specifically designed to move an enterprise smoothly and successfully from CATIA V4 or other CAD systems to CATIA V5 via a flexible path that can be customized to the enterprise′s needs.

We can help with assessment, solution design, planning, installation, data migration, custom application development, best practices consulting, user and administrative training, support, and project management.

For additional information on service offerings and how IBM professionals can assist with the implementation of CATIA V5 in your environment, contact your IBM representative or IBM Services organization or visit

http://www.ibm.com/solutions/plm

Select **PLM Services** from the list of related links.

Enhanced Support Offering

Direct customer support is available under the Product Lifecycle Management Support Services — Enhanced Support Offering (ESO). Refer to this topic in the **Terms and conditions** section for additional information.

Reference information

For information about PLM V5.16, refer to Software Announcement 205-276, dated November 8, 2005.

For information about ENOVIA Solutions V5.16, refer to Software Announcement 205-277, dated November 8, 2005.

For information about SMARTEAM V5.16, refer to Software Announcement 205-278, dated November 8, 2005.

For information about CAA RADE V5.16, refer to Software Announcement 205-280, dated November 8, 2005.

Business Partner information

If you are a Direct Reseller - System Reseller acquiring products from IBM, you may link directly to Business Partner information for this announcement. A PartnerWorld ID and password are required (use IBM ID).

BP Attachment for Announcement Letter 205-279

https://www.ibm.com/partnerworld/mem/sla.jsp?num=205-279

IBM US Announcement Supplemental Information

November 8, 2005

Additional details

This section describes the new products and enhancements to previously announced products delivered in this release. For descriptions of previously announced CATIA V5 and CATIA WLS V5 products and other PLM solutions, visit the PLM Web site

http://www.ibm.com/solutions/plm

Controlled availability of selected CATIA V5 products

The following products, which were in a controlled availability status, are generally available with CATIA V5.16:

- CATIA — Ship Structure Detail Design 2 Product (5691-SDD)
- CATIA — Ship Structure Detail Design 2 Add-on Product (AOP) (5693-SDD)
- CATIA — Product Data Filtering 1 (5691-DF1)
- CATIA — Product Data Filtering 1 AOP (5693-DF1)

The following previously announced products remain in a controlled availability status in this release:

- CATIA — DMU Space Engineering Assistant 2 Product (5691-SPE)
- CATIA — DMU Space Engineering Assistant 2 AOP (5693-SPE)
- CATIA — Electrical System Functional Definition Configuration (5691-EF2)
- CATIA — Electrical System Functional Definition 2 Product (5691-EFD)
- CATIA — Electrical System Functional Definition 2 AOP (5693-EFD)
- CATIA — Business Process Knowledge Template 2 Product (5691-BK2)
- CATIA — Business Process Knowledge Template 2 AOP (5693-BK2)
- CATIA — Knowledge Definition 3 Configuration (5691-KD3)
- CATIA — Business Process Knowledge Template 3 AOP (5693-BKT)

CATIA Platform (P1)

CATIA V5.16 introduces one new product and enhances more than 20 previously announced products.

Products moved from P2 to P1: CATIA — Circuit Board Design 2 has been moved from P2 to P1. Product identifiers are unchanged. These products are now:

- CATIA — Circuit Board Design 1 Product (5691-CBD)
- CATIA — Circuit Board Design 1 AOP (5693-CBD)

Renamed products: CATIA — Engineering Work Package Exchange 1 AOP (5693-EW1) has been renamed to CATIA — ENOVIAVPM Supply Chain Engineering Exchange 1 AOP. The product identifier is unchanged.

New products

DMU Dimensioning & Tolerancing Review 1 (5693-DT1)

Previously announced for ENOVIA, DMU Dimensioning & Tolerancing Review 1 is now available for ordering with CATIA.

Enhanced products: Enhancements to a P1 product also apply to the corresponding P2 product.

Mechanical Design Solutions

CATIA — Assembly Design 1 (5693-AS1)

- Sections can now be imported or exported in the 3D XML format. (This function will become available via the service pack process.)
- A common 3D annotation menu and toolbar standardize the workbench user interface.
- A Visual Basic (VB) macro is available to check for constraints before saving objects to ENOVIA LCA.

CATIA — Weld Design 1 (5691-WD1, 5693-WD1)

A common 3D annotation menu and toolbar standardize the user interface for editing annotations from 3D Functional Tolerancing & Annotation.

CATIA — Part Design 1 (5693-PD1)

- Holes on non-planar surfaces — This enhancement keeps the hole direction associative to the normal surface. The positioning sketch is linked to the non-planar surface so that whenever the surface is modified or moved, the hole moves accordingly.
- Technological results — A new **Create and manage technological results** capability extracts technological information about features included in a body. This information can then be reused by downstream applications, such as Generative Drafting, or by downstream users. Benefits include:
  - Propagate technological information (such as user features or part design feature types and parameters and FTA dimensioning and tolerancing) to geometrical features (face, edge, and wireframe elements) resulting from commands or operations (such as paste as result with link, translate, rotate, symmetry, mirror, and patterning).
  - On demand, load and access valid (that is, consistently attached to the geometrical shape results of the part) technological information

without having to switch part documents in the design mode.

- Obtain technological results information about:
  - -- Hole definition
  - -- Thread definition
- This new capability improves productivity and delivers performance gains in downstream applications, such as large assembly drawing generation in Generative Drafting.

- Threads on conical shapes — This new function lets the user define threads on conical shapes, such as tapered holes and conical faces. The user can also generate drawings to represent threaded conical surfaces and threaded tapered holes.
- Sketcher
  - Sketch positioning direction normal to a surface — Using the new **Normal to surface** option, the user can now select a surface to define an orientation for positioning a sketch.
  - Update error if iso-constraints status is modified (solving status management) — This new option ensures that the Update Diagnosis dialog box appears when editing an under-constrained sketch. It reinforces design methodology application by forcing the user to fully iso-constrain all sketches, which is a recommended design methodology.
- Part infrastructure setting exposition — Users can now employ macros to access part settings.

CATIA — Wireframe & Surface 1 (5691-WS1, 5693-WS1)

- Ergonomics — Using the new 2D mode, the view is limited to elements that are on a given plane. The user can hide and make all geometries unselectable except the ones in a given plane. This capability is useful in both the concept phase, when manipulated objects are often planar (sections), and in the analysis phase, when final volumes and surfaces need to be analyzed throughout planar cuts.
- Assemble option for wireframes — A new extrapolation curve option gives additional flexibility by allowing the user to extract only the extrapolated curve from the entire assembled curve that was created using the extrapolation curve command.
- Freeze sub-menus — Sub-menus can be frozen to avoid automatic changes in point, line, plane, and circle sub-types when selecting geometry. For example, if the current sub-type is *coordinates*, and the user selects a *curve*, the sub-type will not change to *point on curve*.
- Join robustness — The join function is enhanced to enable rapid reconstruction of topological domains by rejecting the merge of overlapped edges. This improves the support of complex skin and surface reconstruction from CATIA V4 to CATIA V5. Based on the same topological components, the **Join** and the **Connect checker** commands optimize surface merge workflow.
- Bi-tangent lines — This release improves the stabilization of bi-tangent lines. A bi-tangent line is a line tangent to two curves. A new algorithm ensures the stability of a bi-tangent line after a design change. After the modification of the input curves of a bi-tangent line, the line is automatically recomputed to conform to the new curve specifications. This enhancement improves usability of the tangent line feature at the time of a **Replace** or **Power copy** instantiation.
- Polyline with corner — Previously, this command was available only in Generative Shape Design (GSD); now it is available in WS1.
- Selection propagation — Contextual menus now afford easy access to two new commands that allow the user to perform point or tangency propagation on sub-elements.

CATIA — Generative Drafting 1 (5693-GD1)

- Generation of threads in views pointing to multi-model methodology (MML) parts — Thanks to technological results, MML is now able to propagate technological information that is applied on the geometry for use by downstream applications. In Generative Drafting, technological results allow thread extraction on MML parts. This capability allows users to retrieve thread information from technological results to generate thread representation symbols in drafting and extends the current thread generation capability to MML parts created using *copy paste as result with link* user-defined features containing thread information.
- Generation of *WorkOnSupport3D* in drawing views — This enhancement incorporates the representation of Generative Shape Design *WorkOnSupport3D* in drafting views and the active *WorkOnSupport3D* when generating CATPart grids and labels. Use of this function requires a license for the CATIA — Automotive Body in White Template 2 (ABT) product.
- Multiselection of bodies or parts in the **Modify links** command — In the **Modify links** command, the selection of bodies/parts is improved and offers multiselection in the product specification tree using the same dialog as in the **View creation** command.
- Visualization mode compliant timestamp computation — Performance and capacity are improved when computing the up-to-date status of assembly views. The status computation previously required some product information to be loaded on top of visualization mode information.

  The new R16 algorithm requires only visualization mode information to compute the status so that no additional information is loaded if the product is in the visualization mode. The new algorithm improves performance when opening drawings or creating the first view (performance improvement varies depending on product content).

  Likewise, CPU usage is reduced by 25 to 95 percent, based on the time required to crate the initial view. The new computation algorithm is available for all view modes, but the greatest improvement is for views in the approximate, CGR, or raster modes, where the entire drafting process is performed with the product in visualization mode.

- Save V5 drawings in 3D XML format — V5 Drawings can be saved as 3D XML files using **File->Save As...**. (This function will become available via the service pack process.)

CATIA — Interactive Drafting 1 (5691-ID1, 5693-ID1)

- The product's DXF capability now supports the export of colored patterns.
- Support is added for importing drafts in 3D XML. (This function will become available via the service pack process.)

CATIA — 2D Layout for 3D Design 1 (5691-LO1, 5693-LO1)

- Clipping frame and back clipping plane for view backgrounds — These new functions allow the user to define a 3D box to clip the representation of a view background.
- View filter enhancement — The user can create display or mask filters, filter new 3D objects (such as FTA annotations and GSD geometry), filter objects by direct selection or by using any other V5 selection tool, and customize the view creation process to add filter capabilities.

Thanks to these clipping and filtering functions, view backgrounds can be simplified to handle the multiple view layout of large and complex parts.

CATIA — 3D Functional Tolerancing & Annotation 1 (5693-FT1)

- Create and review capture callouts (representation of captures in 3D) — This function allows users to create a 3D representation (a "capture callout") of captures.
  - It establishes a new way to directly identify in the 3D representation the captures the user wants to highlight.
  - It enhances the visualization of captures by delivering a customization of the 3D capture representation.
  - All the capture capabilities (contextual menu, selection, query, edit) are available from the capture callout.
    - It enables the display of a quick overview of the captures in the 3D representation. This display consists of text and a rectangular frame or pyramid to form the capture callout.
    - For tool tip display, a hidden text of a capture can be displayed in a tool tip when putting the mouse on the capture feature in the feature tree or on its callout in the 3D view. It enriches and enlarges the information.
  - It improves the usability of the 3D view and contributes to the global improvement of communication among designers.
- Manage show/hide status of bodies, open bodies, and part instances inside captures — Previously, captures controlled the show/no-show status of 3D annotations and constructed geometries. V5.16 enables captures to manage the show/hide status of part instances at the product level and of bodies and geometrical sets at the part level. This extension of captures facilitates the review process and allows the display to be customized.
- New consumable property for annotations for manufacturing purposes — An attribute called *Consumable* is attached to annotations. This highlight enables the user to decide if an annotation can be used for specifying design intent consumption during a manufacturing process. It makes the process easier for the user by reducing product definition redundancy between traditional assemblies and installations.
- Support for updating annotation set standards — This enhancement supplies users with the ability to change the standard of the tolerancing set and to update old 3D Functional Tolerancing & Annotation data with new company standards.
- Support of radius tolerances on constant sections of variable fillets — This improvement enables the user to create content and non-content annotations, such as radius, diameter, and geometrical tolerance, on constant sections of variable fillets.
- Visualization of the clipping plane — This capability enables the user to show clipping as hatching, as dotting, or as coloring. as in the drawing when activating the clipping plane command or displaying a capture with a clipping plane.
- 3D XML support — Functional tolerance annotations can be saved in 3D XML format. (This function will become available via the service pack process.)
- Support customer deployment — This release delivers 3D Functional Tolerancing & Annotation element macro-types for searching all types defined by the Functional Tolerancing & Annotation workbench.

CATIA — Tooling Design 1 (5691-TG1, 5693-TG1)

- Function to analyze block of concurrent engineering — When users insert components and create the associated holes, some links are created between sub-assemblies. This new tool enables the user to quickly analyze the impacted/impacting structure of such links. In conjunction with enhanced context management, this tool enables users to work independently on sub-assemblies and empowers concurrent engineering and communication with suppliers.
- Technological result management — The new technological results (TRs) make it possible to propagate attributes (threads, tolerances) of basic features (holes, in particular) through multi-model links. This capability, in turn, enables:
  - Tight integration between tooling and drafting/machining applications
  - Associativity between the plate to be machined and the tooling assembly design
  - The ability to use patterns, mirror, symmetry, and so on in the definition of the tooling components while preserving technological information.
  - The ability to protect intellectual property by eliminating the need to send part specifications to sub-contractors to convey component tooling information.
- Deactivate/activate tooling components — A new contextual command enables designers to easily deactivate/activate tooling components and all the associated impacted elements to try design alternatives. This is especially relevant for die casting, where designers are frequently using predefined templates of the mold and need to validate that standard components fit their needs.
- Component positioning in axis mode — This function enables users to accurately position tooling components by automatically constraining the components with fixed degrees of freedom. This capability is especially useful for prismatic components.
- Add/remove material enhancements — It is now possible to customize the name of remove/add material operations (Drill Hole, Tap Hole, Pad, Pocket) associated with components for a better understanding of the design structure and to meet company standards. In addition, a new Boolean operator (Assemble) can be used to add/remove features in a more efficient way than the add/remove function and is now the default for add/remove operations. This new operator makes it easy to reorder operations

without having to modify the type of the Boolean operation.

CATIA — Sheetmetal Design 1 (5693-SM1)

New set of parameters for flanged hole — All parameters of the flanged hole feature are now customizable and can be set through design tables to respect company standards and increase productivity.

CATIA — Functional Molded Parts 1 (5691-FM1, 5693-FM1)

- Specification tree — V5.16 introduces a major change in the specification tree presentation for functional features by consolidating functional bodies and functional solids. Functional specifications can now be organized into sets with wireframe and surface features. Previously created functional bodies and functional solids are amalgamated and become a solid functional set (unified solid functional set).

  The functional set enables the organization and management of groups of functional specifications. The capacities of a functional set are equivalent to a geometrical set (reorder children). The functional set affords the full capability to organize and manage the specification tree for the purpose of grouping features by function and to better capture design intent. It improves the readability of parts with multiple functional bodies produced by divided features or core and cavity extractions.

- Fillet intersection — V5.16 introduces the ability to select functional features in part design fillets with rule-based capabilities.
  - The user can select functional specifications instead of B-representation edges because the fillet is applied at the intersection of the selected features.
  - Edges to a fillet can be selected as the intersection between features.
  - Fewer edges require explicit selection, speeding up the filleting task and improving fillet robustness and stability to accommodate major design changes.
  - More information is captured to better reflect user intent.
- Remove, intersect, and rib fillet improvements
  - V5.16 adds to the remove and intersect local modifiers the capability to create fillets at the intersections with the targets. This part of the function (remove and intersect fillets) is conceptually similar to what is already available for the cut functional feature.
  - Fillets are done at the intersection with the target.
  - Improvements of intrinsic rib fillets and features based on open sketches is a logical improvement to add the coverage of fillets at the feature level. Fillet options (first, second, lateral) are also added to the rib feature.
  - The filleting task is made faster and more stable by eliminating the need of selecting individual edges.
  - Filleting capabilities are improved at a higher content level.
- Divide function improvement — Using a surface as a cutting tool, the divide function splits the volumes of a functional body into two functional parts. Each divided set of volumes is placed in an automatically generated functional body, thereby retaining the original functional characteristics.
  - The function includes an option to keep the protected volume and core undivided. The protected volume is duplicated in each of the divided bodies, and the core of either of the divided volumes is not limited at the division surface, allowing the internal volumes to be trimmed to the original extent of the core.
  - This enhancement makes it easier for the user to eliminate excess material within the divided bodies by applying the original protected volume.
- External shape improvement — This release adds:
  - Offset (+/-) capability
  - Different offsets on different faces
  - The ability to embed the offset in the reference, instead of doing a reference, offset, and second reference.
- Integration of functional holes in downstream applications (thread extraction in drafting) — Functional holes can be defined in FM1 at the same level of integration with the drafting application (considering thread representation) as holes defined in Part Design.
- Shellable feature improvements provide the ability to control whether the walls are created with an outside or inside thickness relative to the feature's selected boundary profile.
  - Offset (+/-) capabilities are added for external shape and core selection.
  - "Outward" shell thickness — The shellable shape defines the internal shape.
  - Profiles defined based on the desired core volume boundary make it easier to construct features.
- Transformation enhancement — Currently, the Functional Transform is a functional modifier that applies a translation, rotation, scaling, or reflection transform to a set of selected functional features. This enhancement adds transformation capabilities to this function, such as axis-to-axis and affinity. It also adds a new offset feature to the **Functional Modifier** toolbar.
- Faces can now be selected as profile inputs for functional and shape features.
- To Shell — This release adds:
  - An offset option
  - A To Shell option (with offset) to more features, including core, protected, cutout (floor), and pocket (floor)
- Ergonomics — V5.16 continues to enhance ergonomics by extending the availability of helping pictures to all functions.

Shape Design and Styling Solutions

CATIA — Generative Shape Design 1 (5693-GS1)

- Sweep
  - This release adds a graphical preview for angular sector selection to the sweep function. This ergonomic improvement allows the user to preview the angle that will be used during swept surface computation. The user can directly click on the arrow to indicate the orientation and get the desired output.

- V5.16 also adds automatic evaluation for intersections to the sweep function. In the case of an explicit swept surface with two guide lines, the system will automatically compute the intersection between the profile plan and the guide lines.

- Tolerant extract — Curves can now be extracted with specific distance and tangency criteria. In addition to the curvature threshold, the user can define a distance and angular threshold to use when extracting curves. In this way, the propagation of extracted curves can be controlled using the three criteria (continuity, tangency, and curvature) defined in the panel.

- User pattern — This new function allows the duplication of a feature or a list of features as many times as needed in a chosen location.

- Polyline with corner — This command is now available in GS1. Previously, it was available only in GSD.

CATIA — FreeStyle Shaper 1 (5693-FS1)

The following commands/functions, which were previously available only in FSS, are now available in FS1:

- **Control Points** command
  - Move inside compass plan
  - Move along compass direction normal
  - Move along mesh lines
  - Move along local normal
  - Smoothing button with weight

- **FreeStyle Extrude** command

- Associative and persistent curvature porcupine analysis

- 2D diagrams for porcupine curvature analysis and distance analysis

Product Synthesis Solutions

CATIA — Product Knowledge Template 1 (5691-KT1, 5693-KT1)

With this release, Product Knowledge Template 1 is available as a shareable product.

- Meta inputs for template definition and instantiation — Meta inputs improve productivity for template instantiation and allow better control of the templates. In this release, Product Knowledge Template implements a means for creating meta input and to associate real template input with this meta input. This new capability leverages the publication mechanism in the instantiation process. Meta input improves user productivity and, by allowing the designer to create the inputs, ensures that valid inputs will always be selected when the template is used.

- Quantifier feature for controlling the generation, modification, and deletion of user features — This release introduces a knowledge pattern feature based on the Knowledge language for managing the lifecycle of user-defined features (creation, deletion, modification). Instantiation templates are selected using the Knowledge language and may rely on previously generated templates. The function allows the user to define the relationships between the templates and enables access to templates through attributes, in the same way as any other V5 object.

CATIA — DMU Navigator 1 (5693-DN1)

- Exchange 3D information in 3D XML
  - Enriching 3D XML with DMU content — Measurements (**Measure Item** and **Measure Between**), annotations, scenes, and sections can be embedded in a 3D XML file for exchange purposes.
  - 3D mesh format — In addition to the CATIA Graphical Representation (CGR) format, this new tessellated format, which is fully described in the XML language, is introduced for saving in a 3D XML file. The 3D XML mesh format enables the user to describe any geometry in the XML language. Users now have the choice between the two tessellated formats: CGR and 3D XML mesh, plus the 3D exact geometry format. This capability is available as a user option in **Tools**- > **Options**- > **3D XML**.

    Because the 3D XML mesh format is fully described in XML, anyone can take advantage of its openness. All external formats supported by V5 (including those provided through the V5 MultiCAx solutions) can then be translated into the 3D XML file format.

  These functions will become available via the service pack process.

- Collaborate around 3D
  - New peer-to-peer 3D review capabilities
    - Mouse pointer position sharing
    - Co-highlight/tree expansion/show or hide specification
    - DMU presentation co-browsing

- Enterprise-wide 3D review capabilities (co-requires the VPM Navigator)
  - Mouse pointer position sharing .Co-highlight/tree expansion/show or hide specification
  - DMU presentation co-browsing

Equipment and Systems Engineering Solutions

CATIA — Electrical 3D Design & Documentation 1 (5691-EC1, 5693-EC1)

With this release, Electrical 3D Design & Documentation 1 is available as a shareable product.

Infrastructure Solutions CATIA — Object Manager 1 (CO1)

- Visualization
  - A new render mode that does not include vertices helps clarify edge visualization in complex models. A Custom View mode panel enables the user to choose whether or not to visualize vertices of edges.
  - A new *colored edges from faces* visualization mode enables border edges, internal smooth edges, and internal sharp edges to be colored according to the shading faces color. Each edge has the same color as its adjacent faces. This new mode improves edge selection when working in the NHR view mode by keeping the original color of each part for its edges.

- Printing and imaging
  - HP DesignJet 4000 Series support — V5.16 delivers a V5 driver for the HP DesignJet 4000 to take advantage of the benefits offered by this printer.

As a result, the printout generated with the driver will be compliant with the HP4000 printer and ensures tighter coupling of capabilities between CATIA V5 and the HP4000 printer.

- Output N-views for scheme printing — This release adds support for producing high-resolution N-views for scheme printing.
- Printed line thickness — The user can override the default line thickness to be used when printing models. A new option enables the user to map a color and thickness for each pen. This enhancement applies to HPGL drivers.

- Settings — A new **Tools**- > **Options** choice allows the user to permanently disable back face culling for all objects. As a result, all faces associated with objects are displayed, making 3D navigation and product analysis easier for the user.
- Enterprise architecture
  - Support of CATIA V5 32-bit on Microsoft™ Windows™ 64-bit (AMD Opteron and Intel™ EM64T) for all CATIA V5 and DMU V5 configurations, shareable products, and AOPs with the exception of:
    - CATIA — CADAM Interface 1 (5691-CC1, 5693-CC1)
    - CATIA — CADAM Drafting for V5 (CCD)
    - CATIA — DMU Space Engineering Assistant 2 (5691-SPE, 5693-SPE)
    - CATIA — Product Function Optimizer 2 (5691-PFO, 5693-PFO)

    This enhancement allows CATIA 32-bit to run on the Windows 64-bit platform. It increases memory allocation up to 4 GB of address space, which represents a 33 percent gain compared to the Windows 32-bit platform.

    Note: Integration of VisualBasic for Applications (VBA) on the Microsoft Windows XP 64-bit platform will become available via the service pack process.

  - Support of CATIA V5 64-bit on Windows 64-bit (AMD Opteron and Intel EM64T) for all CATIA V5 and DMU V5 configurations, shareable products, and AOPs with the exception at general availability of:
    - CATIA — DMU Space Engineering Assistant
    - CATIA — ENOVIAVPM Supply Chain Engineering Exchange

    These products will be certified for 64-bit via the service pack process.

    With this enhancement, addressable memory is limited only by the virtual memory of the hardware.

- PLM infrastructure
  - Ability to read SVG format — The user can now open the Scalable Vector Graphics (SVG) format based in XML and describe a 2D vector graphic (and text). This enhancement increases V5′s commitment to graphic format compliance. It can be initiated with **File**- > **Open** without any other specification. The 2D representation launches in the DMU workbench, making it available to all related DMU commands. It can also be imported as an image or as a drafting entity in a drafting sheet. Conversely, a drafting sheet can be saved in SVG format, and the user can set options to define the way the picture and text are saved.
  - **Edit**- > **Links** — This menu choice can now be used to reroute a reference link from an one publication to another. It helps to simplify publication management in a design iteration context because it allows the user to modify the original publication rather than modifying the geometry or the parameter represented by the modification.
  - **File**- > **Save** management with prefix and suffix support — This enhancement allows the user to prefix and/or suffix any CATIA documents selected in the Save Management panel where relationships exist for those documents. This avoids the cumbersome and time-consuming task of having to rename each document, such as when saving a subassembly of a product using a different name.
- Knowledge and collaboration
  - Find protocols — This release improves the capabilities of the Knowledge language to provide navigation function for accessing V5 objects across the Knowledge Advisor, Knowledge Expert, and Business Process Knowledge Template products. Easy access to parameter characteristics, such as ranges or lists of authorized values, makes it easy to write generic rules.
  - On demand Knowledge language dictionary — The dictionary increases the robustness of knowledge rules management (create, understand, execute) by loading on demand the correct package where these rules are stored. This also brings improvements in the PCS area, where packages that contain the rules are loaded only when needed, as opposed to loading all packages at once. The load process is fully transparent to the user, thus increasing usability.
  - Knowledge application management in RADE — All resources (.CATGScript, CATFct, CATNls) that are needed to build an application with the Knowledge suite of products can be gathered in file directories to enable management of those applications with the RADE suite of products like any CAA-written application.
  - CAA parameters — For the CAA developer, it is possible to integrate in their user interface an editor for a parameter that contains the ability to deactivate the parameter in a contextual menu.
- V5 framework
  - Dittos — V5.16 adds layer filter support for dittos.
  - Multi-view wizard configuration — CATIA V5.16 adds two more predefined views (vertical and horizontal tiling) to the manual multi-view configurations. Users can now choose a view configuration from a total of five predefined configurations.
  - Improved display of compass axis labels — Usability of the compass in the editing panel and in 3D scenes is improved by augmenting the font size of the axis label. Usability is also improved by highlighting the axis being operated on (by either rotation or translation).
  - Power input and selection — The user can now search for and select objects in a faster way. Different elements are added to the user interface of the **Search** command to give the user a quicker way to select scope, elements, and symbols. A default prefix, default scope, and predefined

favorite queries, and so on are added to the power input function to further increase user productivity.

- Search performance — Improved ergonomics allow the user to search and select in one click. A new button implements this capability. The search result is then selected and the Search dialog box closed.

- Workbench ordering in the **Start** menu — This usability improvement allows the user to sort a workbench in the **Start** menu. The ability to create sub-trees enables custom classifications and faster access to the workbench. New sub-trees are reflected in the graph tools option.

- Workbench list administration — The administrator can now hide a workbench from a user. A hidden workbench cannot be added to the favorite workbench list. However, hidden workbenches remain visible on the Tools Option tab page. This new option is invoked using the **Tools**- > **Customize Start** menu.

- Openess and 3D XML

  - PLM product information support — Many types of PLM product information can be saved in 3D XML documents. These include:

    -- Annotated views
    -- 3D annotations
    -- DMU presentations
    -- Scene management
    -- Results of **Measure Item** and **Measure Between**
    -- Sections
    -- Animations
    -- Materials applied to product references or product instances for graphical properties (saved or opened from CATIA — Real Time Rendering)

    Such information enriches the content of PLM metadata for 3D XML documents, enabling downstream applications to leverage the use of PLM data.

  - Drafting format support — From CATIA — Generative Design, users can save V5 drawings in the 3D XML format.

    The 3D XML data resulting from the draft can be imported back into CATIA — Interactive Drafting 1.

  - 3D XML mesh representation support — In addition to the CGR (CATIA Graphical Representation) format, a new tessellated format fully described in the XML language is provided for saving a 3D XML file. The 3D XML mesh format can be used to describe any geometry within the XML language. Starting in V5.16, users can choose between the two tessellated formats: CGR and 3D XML mesh, in addition to the 3D exact geometry non-tessellated format. The choice is made as an option in the **Tools** menu (**Tools**- > **Options**- > **3D XML**). Because the 3D XML mesh representation is fully described in XML, anyone can take advantage of its openness. All third-party formats supported by V5 (including those provided through the V5 Multi-CAx solutions) can then be translated into the 3D XML file format. In addition, the 3D XML mesh representation is supported in the 3D XML Player.

  These functions will become available via the service pack process.

CATIA — CATIA — ENOVIAVPM Supply Chain Engineering Exchange (5693-EW1)

Note: In prior releases, this product was called CATIA — ENOVIA Work Package Exchange 1. It is renamed in this release.

CATIA — Instant Collaborative Design 1 (5691-CD1, 5693-CD1)

- V5.16 delivers enhanced merge analysis and control of modifications so that designers can better understand the impact of the merge with a clear status for the features received relative to what they have in their own design. Moreover, they can explicitly control the merge behavior for each feature, such as to accept the merge or to keep their design as it is.

- The share assistant delivers more flexibility and understanding when building briefcases to be sent. It is especially productive when the features to be shared need to imbed bodies and sets in the briefcase as dependencies. The sender can set up rules for these dependencies to either overwrite or accept those on the receiver's side when they are merged.

- Instant Collaborative Design can be a light alternative to allow individual engineers to work informally on separate parts in parallel within a work team. V5.16 simplifies this approach by automating share and merge operations. When the auto share mode is in effect, all the new and modified features, at a predetermined time interval will be automatically added to a new briefcase and shared in the workspace.

  Similarly, the auto merge function simplifies the manual process of selecting and merging briefcases. It automatically merges the briefcases received during a given interval into the model. In addition, **Quick Share** is a new simple command for users who just want to share all new and modified features in the workspace.

- In addition, designers can easily share a briefcase already posted to other users of the workspace, such as new participants or forgotten ones, through the forward briefcase action.

- ENOVIA — VPM Navigator (VPN) is now the cornerstone for client-server enterprise-wide instant collaboration. It widens the scope CATIA — Instant Collaborative Design (CD1) to include asynchronous sharing and offers seamless integration to standard communication solutions for designers.

With this release, Instant Collaborative Design 1 requires ENOVIA — VPM Navigator (VPN) in a client-server mode. Additionally, the co-review function requires a license for CATIA — DMU Navigator 1.

Product Data Filtering 1 (5691-DF1, 5693-DF1)

With V5.16, users can create a new product structure using filtered parts from an existing product structure. This enhancement increases control over intellectual property by enabling the user to create a new product structure with filtered CATParts that are associated to the CATProduct. All applicative data are removed from the original product, but the product structure is kept. As a result, user knowledge is efficiently protected. This also increases productivity by enabling the processing of large product structures. With all parts being filtered and applicative data being removed, the customer will benefit from a substantial reduction in V5 file size.

Data kept in the product structure are left at the user option. These include:

- Product properties
- Colors and attributes for sub-elements
- Annotations
- Publications
- Selection sets
- Layer and filter selection

CATIA Platform 2 (P2)

Release 16 introduces 2 new products and enhances over 40 previously announced products.

Products moved from P2 to P1: CATIA — Circuit Board Design 2 has been moved from P2 to P1. Product identifiers are unchanged. These products are now:

- CATIA — Circuit Board Design 1 Product (5691-CBD)
- CATIA — Circuit Board Design 1 AOP (5693-CBD)

Renamed products: CATIA — Engineering Work Package Exchange 2 Product (5693-EWE) has been renamed to CATIA — ENOVIAVPM Supply Chain Engineering 2 Product. The product identifier is unchanged.

New products

Machining Solutions

CATIA — NC Machine Tool Simulation 2 (5691-MSG, 5693-MSG)

CATIA — NC Machine Tool Simulation 2 delivers an up-front, integrated environment for machine and material removal simulation based on tool path or ISO code during machining operation definition. NC Machine Tool Simulation 2 easily validates the machining setup for selected NC machines and tool paths during machining operation definition. It enables the NC programmer to assign a virtual machine to a part operation, simulate selected tool paths with the machine, determine interferences, modify tool paths or machining operations, generate ISO code, and simulate machine motions and material removal based on ISO code.

The product dynamically detects collisions, if any, during simulation. It enables the NC programmer to re-visit these collisions at the end of a simulation run and to modify machining operations to avoid collisions. It also detects axis limit errors, which can be interactively corrected by modifying the machining setup, thereby enabling the NC programmer to validate and finalize the part setup.

NC Machine Tool Simulation 2 allows NC programmers to:

- Simulate NC machine motion and material removal based on NC tool paths or ISO Code during NC programming
- Detect and correct interferences in tool paths and ISO code
- Interactively avoid collisions by teaching machines collision-free paths or by editing machining operations, thus reducing programming lead time

Key product functions include:

- Automatic workpiece mounting and machine setup validation
- Integrated simulation of NC machine motions and material removal based on tool paths or ISO code
- Validation of a manufacturing program, a single machining operation, or a zone of a machining operation
- Accurate cycle time calculation
- Checking of travel limits of the machine tool, collision detection, and distance analysis
- Synchronized display of ISO code during ISO-based simulation
- Interactive too path modification to avoid collisions

NC Machine Tool Simulation 2 offers the following user benefits:

- Shortened NC programming time, thanks to an end-to-end machining solution with seamlessly integrated NC machine tool simulation. Users can easily switch between definition and validation without losing time due to data transfer or preparation (no third-party software interface issues).
- Tool Path validation at an early stage avoids problem and modification loops. Thanks to realistic NC machine tool and material removal simulation based on ISO code, parts will be cut right the first time, thereby reducing lead time from programming to production.
- Increased process quality, thanks to easy problem identification and resolution. In case of collisions or exceeded travel limits, users can interactively edit machining operations or modify tool paths.
- Avoided redundant machine tool data, thanks to a single comprehensive machine tool description with geometry, kinematics, and technological information. The definition of the NC resource can be used in the entire machining process, including NC planning and detailing, simulation and verification, post processing, and controller emulation.

CATIA — NC Machine Tool Simulation 2 (MSG) requires CATIA — NC Manufacturing Review 2 (NCG). It is strongly recommended that CATIA Space Analysis — 2 (SPA) be used along with NC Machine Tool Simulation (MSG). A CATIA — Space Analysis 2 license is required for the distance and band analysis and clash detection and analysis functions in the context of machine simulation.

CATIA — NC Machine Tool Builder 2 (5691-MBG, 5693-MBG)

CATIA — NC Machine Tool Builder 2 delivers a unique environment for machine tool definition and peripheral resource (tool and pallet changers) in support of NC programming and simulation. Machine Tool Builder 2 easily models resources with kinematics, such as NC machines, tools, tool changers, pallet changers, and other peripheral devices for use and reuse in the entire machining process. Machine Tool Builder 2 supplies a comprehensive NC machine definition including geometry, kinematics, controller, and technological information.

The unique resources created with Machine Tool Builder can be saved to the DELMIA manufacturing hub and used by process planners, NC programmers, and operators to create machining process plans, validate machining setups, detail machining operations, validate and optimize tool paths, perform post processing, and emulate controllers. Machine Tool Builder can also import NC machines created in DELMIA D5 for use in CATIA V5 machining applications.

NC Machine Tool Builder 2 incorporates the following main functions:

- Provide a single user interface to completely define the NC machines including geometry, kinematics (forward and inverse), and attributes.
- Support technological information definition as required by NC detailing.
- Assign post processor and controller emulator information as required by post processing and simulation.
- Validate the defined kinematics, travel limits, tool change positions, and home positions using the jog function.
- Save a machine along with all its attributes in the Manufacturing Hub for use/reuse of this unique resource in machining process planning, NC detailing, and simulation.
- Reuse DELMIA D5 machine geometry and kinematics.

NC Machine Tool Builder 2 offers a single, comprehensive machine tool description with geometry, kinematics and technological information. One definition of the NC resource can be used for the entire machining process, including NC planning and detailing, simulation and verification, post processing, and controller emulation, avoiding redundant machine tool data. The product's jog capability also makes it easy to validate the defined NC machine tool, its kinematics, travel limits and home positions in the definition phase.

CATIA — NC Machine Tool Builder 2 (MBG) requires CATIA — NC Manufacturing Review 2 (NCG).

Enhanced products: For P2 products with P1 counterparts, only enhancements that are in addition to those described for the P1 product are included here.

Mechanical Design Solutions

CATIA — Assembly Design 2 (5691-ASD, 5693-ASD)

Enhancements are the same as for AS1.

CATIA — Part Design 2 (5691-PDG, 5693-PDG)

- Boolean operations can now be performed on bodies that have been inserted into an ordered geometrical set (OGS).
- After deleting a Boolean operation performed between a body and a body included in an OGS, the user can keep the operand body in its original position within the OGS. This is made possible using **Delete and Keep Operand in Context**, a new capability available from contextual menus.
- The new technological results capability described for Part Design 1 is integrated into the Part Design 2 thread analysis function.

CATIA — Generative Drafting 2 (5691-GDR, 5693-GDR)

Enhancements are the same as for GD1.

CATIA — 3D Functional Tolerancing & Annotation 2 (5691-FTA, 5693-FTA)

- Restriction area creation enhancement — Support is added for multi-geometric elements restricting and restricted features. This enhancement enables the user to create and manage a partial surface defined on a restricted feature made of several elements (if needed) and a restricting feature made of several elements (also if needed).
- Support of circle geometry for content-oriented linear dimensions —content and non-content annotations can now be created on constant sections of variable fillets. This enhancement aims at presenting new content-toleranced dimensions cases that can be taken into account and refers to ISO 8015 (Fundamental Tolerancing Principles) and ASME Y14.5.1M-1994 (Mathematical Definition of Dimensioning and Tolerance Principles).

  It implements the capability to create content-toleranced dimensions between a planar face, a cylindrical face, a spherical face, an axis, a point, or a plane and a planar face, a cylindrical face, a spherical face, an axis, a point, or a plane.

CATIA — Mold Tooling Design 2 (5691-MTD, 5693-MTD)

- Function to analyze block of concurrent engineering — When users insert components and create the associated holes, some links are created between sub-assemblies. This new tool enables the user to quickly analyze the impacted/impacting structure of such links. In conjunction with enhanced context management, this tool enables users to work independently on sub-assemblies and empowers concurrent engineering and communication with suppliers.
- Technological result management — The new technological results (TRs) make it possible to propagate attributes (threads, tolerances..) of basic features (holes, in particular) through multi-model links. This capability, in turn, enables:
  - Tight integration between tooling and drafting/machining applications
  - Associativity between the plate to be machined and the tooling assembly design
  - The ability to use patterns, mirror, symmetry, and so on in the definition of the tooling components while preserving technological information.
  - The ability to protect IP without the need to send the part specifications to sub-contractors to access component tooling information.
  - The ability to protect intellectual property by eliminating the need to send part specifications to sub-contractors to convey component tooling information.
- Deactivate/activate tooling components — A new contextual command enables designers to easily deactivate/activate tooling components and all the associated impacted elements to try design alternatives. This is especially relevant for die casting, where designers are frequently using predefined templates of the mold and need to validate that standard components fit their needs.
- Component positioning in axis mode — This function enables users to accurately position tooling components by automatically constraining the components with fixed degrees of freedom. This capability is especially useful for prismatic components.
- Add/remove material enhancements — It is now possible to customize the name of remove/add material operations (Drill Hole, Tap Hole, Pad, Pocket) associated with components for a better understanding of the design structure and to meet company standards. In addition, a new Boolean operator

(Assemble) can be used to add/remove features in a more efficient way than the add/remove function and is now the default for add/remove operations. This new operator makes it easy to reorder operations without having to modify the type of the Boolean operation.

CATIA — Sheetmetal Design 2 (5691-SMD, 5693-SMD)

Enhancements are the same as for SM1.

CATIA — Composites Engineering 2 (5691-CPE, 5693-CPE)

- Top surface from zones: Shape morphing for transition zones — When creating the top surface from zones, it is now possible to create the surface corresponding to the transition zones with a new shape morphing operator option that takes into account the inner mold surface curvature. This improvement creates a more realistic transition zone representation.
- "Full plies" associativity with zones contours — Plies design becomes fully associative with zones design as full plies (that is, the engineering edge of a part) are now associative with zones contour definition.
- Cut-piece management — The core sample, ply exploder, numerical analysis, and limit contour functions now take into account cut pieces resulting from splicing operations.
- Ply exploder with tessellated geometry — This function enables the user to get a more realistic view of the exploded composites stacking for unmatched analysis of the composite structure in DMU.

Note: To address strong customer requirements, the CATIA composites data model will be updated using the Release 16 service pack process. To ensure data compatibility, it is highly recommended that CATIA V5.16 customers use only Service Pack 1 or later for composites work.

CATIA — Composites Design for Manufacturing 2 (5691-CPM, 5693-CPM)

- Cut-piece management — The core sample, ply exploder, numerical analysis, and limit contour functions now take into account cut pieces resulting from splicing operations.
- Flattening command — Multiple point selection allows the user to create flattened features for several plies with one location point for each ply. This enhancement improves visualization of flattened plies and for nesting preparation.
- Ply exploder with tessellated geometry — This function enables the user to get a more realistic view of the exploded composites stacking for unmatched analysis of the composite structure in DMU.

Note: To address strong customer requirements, the CATIA composites data model will be updated using the Release 16 service pack process. To ensure data compatibility, it is highly recommended that CATIA V5.16 customers use only Service Pack 1 or later for composites work.

CATIA — Functional Molded Parts 2 (5691-FMP, 5693-FMP)

- Clip function — A clip is a specialized feature that incorporates all the best practice parameters. It may be designed with a constant beam section or a combination of tapered thickness and tapered width. Its use of process-oriented features, such as those for plastics, increases productivity.
- Flange — This function provides the capability to add a flange as a support for the mating of two components. It is a multi-body feature that requires at least one divided half body (such as a lip). Its process-oriented features increase productivity. Flanges are very common in several internal components, such as fluid tanks for the automotive industry and valve enclosures for household appliances.
- Additional functions improve the **Join** command.
  - Inside/Outside shape improvement — The join can be positioned at an external position.
  - Integration with the screw database — Integration allows definition of screw parameters by selecting the screw from an existing screw catalog.

Shape Design and Styling Solutions

CATIA — Generative Shape Design 2 (5691-GSD, 5693-GSD)

Note: These enhancements are in addition to those for GS1 and WS1.

- Adaptive sweep
  - Generation from sketches — A sweep can now be generated from an output or a profile of a sketch. This enhancement improves usability by allowing the use of only a sub-part of a complex sketch while keeping constraints set effective.
  - Positioning sketch integration — In the context of the **Sweep**, command, the user can define the position of the sketch according to the surface normal and aggregate it under the sweep. This improvement increases sweeping stability and ensures perfect correlation between the sketch and the sweep.
  - Stack join for reference surfaces — In the context of the **Adaptive Sweep** command, the user can now launch the **Join** command to define a reference surface. This enhancement increases productivity by enabling a rapid creation of joined surfaces as reference surfaces in the adaptive sweep context.
- Edge robustness — A new option for defining intersecting edges allows modification of both intersecting edges and the features based on them. For example, if the user defines an intersecting edge between two surfaces with the new edge definition option and then defines an extruded surface based on this edge, the user will be able to translate the extruded surface seamlessly. This improvement ensures good edge stability to better manage part modifications.
- Mask box to limit part display — The mask box is a 3D box in which geometry is displayed. The view is limited to elements inside or intersecting the 3D box. The box improves user productivity by offering the ability to limit the visualization only to selected geometries inside the 3D box.
- User deviation for robust offset — A new option makes the offset's smoothing capability accessible to user manipulations. To perform an offset on non-offsetable surfaces, the user can manually smooth the geometries by defining the maximum deviation allowed between the theoretical and the smoothed surface. This enhancement increases the robustness of the offset while controlling the validity of the offset result.

- Sweep — The management of removed twisted areas is improved in this release. Twisted areas can be automatically removed during the preview.
- Sweep two guides and a radius for a full circle solution — When using two guides and a radius to create a sweep, the system initially suggested only four choices related to arc sweep solutions. This enhancement offers two additional solutions that correspond to complete circular solutions.

CATIA — Generative Shape Optimizer 2 (5691-GSO, 5693-GSO)

V5.16 introduces a new wrap surface type with deformation direction. The user is asked to specify a deformation direction and two surfaces: a reference surface and a target surface. If a direction is provided, these two surfaces could be multi-cell surfaces. The deformation of a shape consists of projecting it onto the target surface according to the reference surface direction. This approach gives the user better control of global deformation and increases its robustness.

CATIA — FreeStyle Shaper 2 (5691-FSS, 5693-FSS)

- Generative Shape Design (GSD) mask availability — The GSD mask is a 3D box in which geometry is displayed. The view is limited to elements inside or intersecting the 3D box. The mask improves productivity by enabling the user to limit the visualization to only the selected geometries inside the 3D box. This capability is imported from GSD, which is not a prerequisite.
- Shaders for zebra mapping — New shader technology improves zebra mapping. A new, free-form technological breakthrough can be used to evaluate quickly and with high quality the shape flow and the continuity between geometries. Rather than assigning a color to each of the vertices of the visualization mesh, the color is assigned to each pixel that is related to the analyzed geometry. This enhancement significantly improves the quality of the results of the analysis process. It is available only on the Microsoft Windows or IRIX platforms with nVIDIA or ATI graphic cards.
- Panel improvements for associative isoparametric curves — Introduced in CATIA V5.15, this release delivers two enhancements to improve workflow and foster ease of use when creating associative isoparametric curves.
  - User interface — A new dialog box interface makes it easier to use the function.
  - GSD element creation stack command — To create an isoparametric curve, the user must define a support, and this support must be a surfacic element. These support elements are provided with GSD stack commands, such as **Join**, **Extract**, and **Extrude**.
- Break command improvement and new ergonomy
  - There are three types of breaks: Curves by Points, Curves by Curves, and Surfaces by Curves. For Curves by Points and Surfaces by Curves, it is now possible to define a point as a cutting element within the **Break** command.
  - The user interface is improved to optimize workflow.
- Color ramp enhancements
  - Min/Max pushbutton — A new option in the **Distance Analysis/Curvature Analysis** command allows the minimum and the maximum computed analysis values associated with the current analysis to be used as the extreme values of the color scale.
  - Special color associated to undefined results range — The user can now chose a color to be associated with erroneous areas for measurements.

  These enhancements produce more accurate analysis results by allowing the user to customize the user interface.

CATIA — Digitized Shape Editor 2 (5691-DSE, 5693-DSE)

- Alignment by constraints — Because constraint priority is much more intuitive to the user than tolerance values, V5.16 enables users to define a level of priority for each constraint The higher the priority assigned, the more it has to be respected without deviation. The result is independent of the constraint input order. It depends only on the priority. This enhancement increases the flexibility of the alignment process and ensures its robustness and is consistent with the user's expectation.
- Project a mesh onto a plane — This new option allows the user to create planar scans by projecting meshes or clouds onto a plane. It can be used in association with the Quick Surface Reconstruction product to retrieve sketches from planar scans. It could also be used in aerodynamic simulations to calculate vehicle resistance.

CATIA — Quick Surface Reconstruction 2 (5691-QSR, 5693-QSR)

- New automatic surface reconstitution without curve segmentation — Usually in the network approach, the reconstitution of surfaces from meshes requires the user to create a network of curves to fill the surfaces. The new command significantly improves productivity and ease of use by automatically building surfaces of meshes/clouds, eliminating the need to create a network of curves.
- Recognition and creation of lines and ellipses on 2D scans — The aim of this enhancement is to recognize primitive shapes (lines, circles, ellipses) in the planar scan. Once the shape is recognized, a sketch can be built using the primitives. This approach allows users to convert the digitized contours of production tools into curves for electro-erosion programs. It improves productivity during the alignment process by making it easier to quickly find the characteristic points for the alignment (centers of circles, for example).

Product Synthesis Solutions

CATIA — Product Knowledge Template Definition 2 (5691-PKT, 5693-PKT)

Note: In prior releases, this product was called CATIA — Product Knowledge Template 2. It is renamed in this release.

- User feature reference modification — User features no longer have to be recreated from the beginning when modifying a components list. This enhancement greatly improves ergonomics, productivity, and user perception and enables a more efficient continuous development process for user-defined features.
- User feature debugging capability — Debugging in this case should be taken as opening and closing the user defined feature. The debug capability enables several schema for instantiating user features:

- White box — Intended for debugging only.
- Black box — Instantiated as before. A specific debug command can still be used to swap from black box to white box, but only with a Product Knowledge Template Definition license. This mode enables the deployment of secured best practices within the company, while maintaining security for a small number of expert users so they can debug the user features if a problem is encountered.
- Black box protected — Same as black box, but does not allow swapping. This mode enables secured supply chain exchanges.

These new capabilities bring flexibility for user defined feature instance exploitation and increase the value of the functional design process. By enabling industrial companies to deploy user defined features, these capabilities are a major step in removing inhibitors for deploying the overall solution. They also enhance the ability to manage intellectual property protection between OEMs and their supply chains and ensure secure industrial use within the company by enabling the opening and closing of user defined features once they have been deployed.

- Meta inputs for template definition and instantiation — When creating user features or document templates, the user can now assemble "high-level" objects together and connect them to their environments through their inputs. This capability improves productivity for template instantiation and allows better control of the templates.

  Product Knowledge Template Definition supplies a means for creating meta input and to associate real template input with this meta input. The meta input mechanism is a way to ease knowledge template instantiation (user defined features and document templates). It uses a component-based approach to assemble the inputs. Meta input specifications are stored in knowledge templates.

  Meta input also increases the usability of templates within the Business Process Knowledge Template 3 (5693-BKT) product, which is often used to instantiate numerous templates, each from the output of the previous template instance.

  Instantiation through the knowledge language is also improved.

- Quantifier feature for controlling the generation, modification, or deletion of user features — This release introduces a knowledge pattern function based on the knowledge language for managing the lifecycle of user-defined features (creation, deletion, modification). Instantiation templates are selected using the knowledge language and previously generated templates. It allows the user to define the relationships between the templates and enables access to templates through attributes, in the same way as any other V5 object.

CATIA — Business Process Knowledge Template 2 (5691-BK2, 5693-BK2)

- Behavior executing a Knowledge language script — A new behavior is introduced in the Business Process Knowledge Template (BKT) product to allow the definition of the logic of the application with the Knowledge language. Once developed, the behavior runs in the user workbench and can easily access V5 features, thanks to dedicated functions in the language (such as query, find, and access). This new behavior supplies BKT architects the full power of the Knowledgeware language to access and manipulate V5 features and offers an interesting alternative to pure interactive creation of the application logic, which is particularly suitable for "low-level" logic.
- Interactive definition of sub-functions — This enhancement enables direct access to the sub-function interactive creation process with the Knowledge language but without being dependent on C++ development directly in BKT. The new sub-functions appear in the tree view along with behaviors and technological types. The sub-functions can be called from anywhere the Knowledge language is used, such as Knowledge Advisor reactions, BKT behaviors, or Knowledge Expert rule bases. This new capability allows architects to better structure their application logic and to make this logic reusable in BKT contexts.
- Reusability of a sub-process within a main behavior — This improvement allows the reuse of existing behavior patterns without duplicating data. The knowledge architect can define the sub-process and then use it from its knowledge application, thanks to a new behavior for calling it. This new capability allows architects to better structure their application logic and to make this logic reusable in BKT contexts.
- Isolation of the knowledge application from its knowledge resources — This enhancement makes deployment of knowledge solutions much easier by separating the resources used by the application, such as a template or a rule base, from the kernel of the application using application resources management (ARM) files. Any further change of the resources will not require editing and modification of the application (the CATFct); only the ARM catalog needs to be updated to point to the right resources.

  This isolation has three advantages:
  - It clearly separates the development of an application from its deployment.
  - It separates the lifecycle of the application from the lifecycle of its resources (which are usually not the same).
  - It simplifies the usage of the resources within BKT.

  Finally, the resources can be managed in ENOVIA while the application is managed with RADE on a classic file directory structure.

- Separate Technological type data model from product data model — When V5 products and features are extended with technological objects, they no longer require the presence of the application (the CATFct) in the runtime environment. This clearly separates the generated data from the application that has generated the data, which enables easy downstream usage of the data. It also affords more flexibility in the design of the application because technological types can be modified without having to regenerate all the data previously generated.
- Behavior enhancement — Behavior scripts written in Visual Basic and behavior commands developed in CAA now rely on official and stable exposure: these behaviors are critical for establishing the connection between a BKT application and legacy data or algorithms.
  - The system behavior is enriched to provide a simple call to a C++ function.

- The CAA command behavior is used for integrating the CAA user interface.
- VB behavior is enhanced to call Visual Basic for Applications (VBA) interpreted code, which makes it easy to integrate specific user interfaces.
- VB behavior is improved to provide more direct access to the behavior variables (input and output).

These improvements bring new capabilities to BKT, such as a dedicated user interface to the knowledge applications (thanks to VBA) or connection to legacy databases, and extend BKT to the full range of CAA V5 APIs.

- Workbench deployability — Users can now create workbenches and group them by categories. This capability fosters deployment of workbenches generated in BKT. The definition of the workbench no longer relies on CATIA settings, but becomes part of the data that can be managed within RADE.
  - Workbench access can be managed according to user categories.
  - Workbench management provides a hierarchical level of administration based on user profiles.

CATIA — DMU Navigator 2 (5691-DMN, 5693-DMN)

- When importing applicative data, creates the DMU review at the highest level.
- Enriches 3D XML with DMU content. Presentations can now be embedded in a 3D XML file for exchange purposes. (This function will become available via the service pack process.)

Equipment and Systems Engineering Solutions

CATIA — Electrical Library 2 (5691-ELB, 5693-ELB)

- Multi-insert connector — This new feature enables the user to create a multi-insert connector with as many terminations as needed. This new connector type allows one cavity connection point and one or more connector connection points and terminations. It improves harness detailed design for the aerospace and automotive industries.
- Connection point for shells — V5.16 introduces a command to define a connection point for connecting two shells together. The command can be used to:
  - Define a shell connection point on a shell.
  - Connect two shells together using drag and drop or **Connect Devices**.
  - Define a backshell connection point on a shell.
  - Connect a backshell on a shell using its backshell connection point.
  - Define a modular connector (assembly of connectors) in a harness.
- Defining cavities on devices — This release enables the user to add cavities to terminal strips, terminal blocks, and ground studs and to connect these components to contacts.
  - Connect contacts on device cavities (drag and drop, connect, or place from XML).
  - Definition of BOMs of aerospace electrical systems is improved.

CATIA — Electrical Harness Installation 2 (5691-EHI, 5693-EHI)

- Placement of bundle segments inside supports — This release adds dedicated commands to help define the layout of bundle segments in supports. The commands enable the user to arrange bundle segments in a support and to propagate an arrangement of bundle segments from one support to another. This enhancement improves productivity when routing 3D aerospace electrical systems when supports are shared by several harnesses.
- Bundle segment links — V5.16 improves the process of creating links to supports that are outside of the geometrical bundle in the work package.
  - A new dedicated command updates the geometry of bundle segments that are to be routed to supports located outside the geometrical bundle saved in ENOVIA as work packages.
  - Links between bundle segments and the segments supports that are not yet assembled in the harness are maintained, even if the harness is saved as an ENOVIA work package.
  - The capability is available with ENOVIA VPM V4 and ENOVIA VPM V5.
  - The function allows more flexibility in the definition of electrical product structures in ENOVIA, especially for supports shared by several harnesses.
- Knowledge rules — New knowledge rules improve the quality of a design by checking the harness using automated knowledge checks.
  - Electrical connections — Check the electrical connections on bundle segments by counting the number of objects connected to a given bundle segment extremity (connectors or other bundle segments).
  - Check compatibility between a bundle segment and supports used. This new knowledge function retrieves the list of supports used for a given bundle segment.
  - Retrieve the contact part number associated with a wire extremity (from and to) and expose new attributes of the wire object in knowledgeware:

    -- Elec_FromContactPartNumber
    -- Elec_ToContactPartNumber

    Additionally, generate a manufacturing wire list report that includes the associated contact part numbers.

CATIA — Electrical Harness Flattening 2 (5691-EHF, 5693-EHF)

This release improves the process for orienting 2D detail devices in drawings. It provides the user with the ability to orient the 2D detail (symbol) to follow the tangency of the bundle segment during the drawing process. This enhancement improves productivity when creating 2D documentation for harnesses.

CATIA — Electrical Connectivity Diagrams 2 (5691-ELD, 5691-ELD)

- Rename pins — A new command, **Rename Pin**, enables the user to rename a pin connector after adding pin connectors to a component. Previously, the system assigned default names to pin connectors as pin1,

pin2, pin3, and so on. The user can now give each pin connector a meaningful name.

- Replace cable references — The user can now replace the cable reference associated with a cable instance. In the shipbuilding domain, there is a very significant requirement to design and manage electrical cables and cable routes. As part of the design process, the user may determine that an incorrect cable has been used in a certain circumstance. In this case, the user wants to be able to replace the cable reference without having to redefine the from/to information or any previously defined routing information. The new replace function allows the user to make replacements in this way. This capability improves the logical electrical design phase.
- Provide computed attributes for wire objects (FromEquipment, ToEquipment, FromPinID, ToPinID, CablePartNumber) — Computed attributes are used in various tools for evaluating attribute values in a specific schematic diagram. In V5.16, the following computed attributes are available for electrical wires:
  - FromEquipment — The "from" equipment to which the wire's cable is connected
  - ToEquipment — The "to" equipment to which the wire's cable is connected
  - FromPinID — The pin ID on the "from" equipment to which the wire is
  - ToPinID — The pin ID of the "to" equipment to which the wire is connected
  - CablePartNumber — The part number of the cable that contains the wire

  Thanks to knowledge integration, users can now create rules and checks using these computed attributes. This improvement facilitates the capture and sharing of corporate intellectual capital.

CATIA — Electrical Cableway Routing 2 (5691-ECR, 5693-ECR)

- Access and management of cable routings — The new ENOVIA — VPM Electrical Cable Route Management (ECV) product facilitates access and management of cable routing when used with CATIA — Electrical Cableway Routing 2 (ECR). It enables concurrent work in a complex cableway network on large projects. Multiple users can perform cable routing through the same cableway network.

  The cableway network is never modified. It is used only to define the optimum cable path using the cable routing rules, such as maximum cable length, percent fill, and minimum bend radius. It allows the user to use a very large network for the purpose of cable routing. An optimized light cableway network is visualized in CATIA by directly getting the relevant data from the database. Only data that is needed for the cable routing is extracted and loaded in session.

  This approach reduces memory consumption and allows the user to work with very large networks to perform cable routing. Using Cableway Routing 2 on with the electrical database enables the following route operations:
  - Specify the list of cables to be routed using various filter options: all cables in a schematic, all cables connected to a piece of equipment, all cables in the project.
  - Route cables (automatic and partial)
  - Recommend or forbid a route path
  - Visualize cable routes (using a light cableway network)
  - Validate cable routes
  - Get cable route status
  - Delete cable routes

  This function requires a license for the ENOVIA — VPM Electrical Cable Route Management product.

- Network definition for an electrical database — The user can define a cableway network using hangers, MCTs, or lofts. This function also requires a license for the ENOVIA — VPM Electrical Cable Route Management product.

Systems Diagrams 2 (5691-SDI, 5693-SDI)

- Define a default color for object types — Users can define their own object types (based on default types) using the feature dictionary editor. This enhancement enables the user to associate default colors to the types defined in the feature dictionary editor. It improves the graphical content and communication of the diagram.
- Use multi-discipline equipment to create multiple occurrences of the same schematic object — This enhancement enables for the creation of multiple occurrences of the same schematic object for multi-discipline equipment. For example, the user designs a piping system that contains piping-related equipment, such as a pump. The pump will be included in a piping diagram for the piping system design, but the user will also need to supply electrical power to the pump. So, the user will also need to include the same pump object in an electrical diagram. This enhancement implements this capability by enabling an object to be shared across diagrams of different disciplines.

  A new command enables the user to place an occurrence of a piece of multi-discipline equipment by simply selecting it from an opened schematic diagram and placing it into the active schematic document. The original equipment is considered the master occurrence and can be edited by the user. All other occurrences of the object will be considered images of the master occurrence and cannot be edited by the user.

  A new command also enables the user to update the attributes of an image from the master occurrence if the master has been modified.

  Together, these enhancements improve communication between applications.

- Off sheets can be defined as multiple occurrences of same object — In this scenario, the user requires that a single line route (for example, a pipe function or HVAC function) needs to be shown across multiple sheets of a diagram. Therefore, the same logical object must be represented across multiple diagram documents. With V5.16, whenever the user creates an off-sheet connection between two line routes in different diagrams, the system will define one of the line routes to be a master occurrence and the other to be an image. This enhancement ensures the two line routes will be treated as one logical object.
- Support for impact analysis between schematic diagram and related 3D design document — A new

command allows the user to explicitly create a link between a 2D schematic diagram and associated 3D design documents. Once the link is established, the VPM Navigator **Impact On/By** commands will recognize this relationship and display it to the user.

Piping Design 2 (5691-PIP, 5693-PIP)

- Generative View Style (GVS) support for text template definition by object class — This enhancement adds the ability to associate standard text templates to a specific object of a discipline, such as piping and HVAC, in the GVS file definition. Examples include the IDs of all piping objects and the length of pipes. This customization capability enables the user to standardize the text templates for specific drawing types. For example, the user may have different text template call-outs for different drawings: installation drawings, fabrication drawings, and so on.
- Customize text template positioning and layout on a drawing — This enhancement lets the user customize the layout of text templates (call-outs) on a drawing. A leader is created for each text that string that is associated with an object. It enables the user to standardize drawing layout definition.
- Loose parts definition — A new dedicated command allows the user to model loose parts that are associated with piping parts. Loose parts are such items as bolts, nuts, and washers. They are associated with base parts, such as flanges.

  The new **Loose Parts Management** command implements light, non-graphical modeling of loose parts for generating reports and for creating call-outs in a drawing. The user can manage the lifecycle of loose parts and also extract loose part information in downstream processes, such as reporting and material take-off.

  The new command is added to all of the 3D Detail Design workbenches (Piping Design, HVAC Design, Equipment Arrangement, and Compartment & Access) to promote ease-of-use across these related products.

- Extract a bend diameter factor for pipe nodes — This enhancement enables the user to extract/read the bend radius definition for pipe or tube turn nodes. The bend radius definition can be defined as a diameter factor multiplier. The extraction of the bend radius at a pipe or tube turn is necessary to feed it to a bending machine during the manufacturing process.

CATIA — Equipment Arrangement 2 (5691-EQT, 5693-EQT)

This release introduces a dedicated command that allows the user to model loose parts that are associated with piping parts. Loose parts are such items as bolts, nuts, and washers. They are associated with base parts, such as flanges.

The new **Loose Parts Management** command implements light, non-graphical modeling of loose parts for generating reports and for creating call-outs in a drawing. The user can manage the lifecycle of loose parts and also extract loose part information in downstream processes, such as reporting and material take-off.

The new command is added to all of the 3D Detail Design workbenches (Piping Design, HVAC Design, Equipment Arrangement, and Compartment & Access) to promote ease-of-use across these related products.

CATIA — Compartment & Access 2 (5691-CNA, 5693-CNA)

- Drawing production — Users can now define different line styles for each graphic representation. For access parts, such as doors, hatches and ladders, the user normally has multiple graphic representations defined, such as layout and envelop representations. In many cases, the user wants to include multiple representations in a drawing, but they want the different representations so each has its own line style for drafting lines, arcs, and curves. Individual line styles, such as solid, dash, and dotted, can be defined for each graphic representation of an access part.
- Support for loose part definitions — A new **Loose Parts Management** command is added to the Compartment and Access workbench. It enables the user to create and manage the lifecycle of loose parts, such as bolts, nuts, and washers. These loose parts are associated to base parts, such as doors and hatches. The user can manage the lifecycle of loose parts and can also extract loose parts information in downstream processes, such as reporting and bills of materials.

CATIA — HVAC Design 2 (5691-HVA, 5693-HVA)

CATIA V5.16 introduces a dedicated command that allows the user to model loose parts that are associated with piping parts. Loose parts are such items as bolts, nuts, and washers. They are associated with base parts, such as flanges.

The new **Loose Parts Management** command implements light, non-graphical modeling of loose parts for generating reports and for creating call-outs in a drawing. The user can manage the lifecycle of loose parts and also extract loose part information in downstream processes, such as reporting and material take-off.

The command is added to the all of the 3D Detail Design workbenches (Piping Design, HVAC Design, Equipment Arrangement, and Compartment & Access) to promote ease-of-use across these related products.

CATIA — Structure Functional Design 2 (5691-SFD, 5693-SFD)

- Power copy and user-defined feature (UDF) integration allow users to duplicate plates and beams.
- A new project plane definition capability can be used to create measurement reference planes.
- The shipbuilding grid is enhanced by the ability to display permanent 3D text.
- A new option in the **Split Plate** command allows the user to break plates.
- A new option in the **Split Stiffener** command lets the user to break stiffeners.

CATIA — Ship Structure Detail Design 2 (5691-SDD, 5693-SDD)

SDD now uses the same data model and the same enhanced shipbuilding geometrical modeler that was introduced in V5.15. SDD offers the same user interface and functions, but with a different focus. This release adds:

- The ability to split a structure functional design system by functional volumes defined previously in Structure Functional Design 2.
- Improved ability to extract Structure Design 1-compliant data through the piece part engine.

- Support for extracting classification drawings from detail design
- Power copy and user-defined feature (UDF) integration for duplicating plates and beams.
- A new project plane definition capability for measurement reference planes.
- The ability to display permanent 3D text in the shipbuilding grid.
- A new option in the **Split plate** command for breaking plates
- A new option in the **Split stiffener** command for breaking stiffeners

Analysis Solutions

CATIA — Generative Assembly Structural Analysis 2 (5691-GAS, 5693-GAS)

- Connections and connected parts information in assembly part summaries — When selecting an analysis part, all parts connected to the selected analysis part and their node-to-node connections are listed in a report file. The connection information can be dumped to an HTML file for external usage. This enhancement saves data access time and enables the user to select only the pertinent information.
- Assembly of displacements — Across an assembly of displacements, the OEM can post-process all the vehicle components that come from suppliers based only on computations made at the component level. In this way, the OEM can rapidly compare the component's displacement in the context of the complete assembly without having to build the assembled analysis for the entire vehicle. This capability reinforces collaboration between OEMs and their partners and increases the robustness of the engineering package exchange process.
- Assembly of loads and additional mass sets — Users can now create assembled loads and assembled masses to complete a vehicle-level analysis by referencing load/mass sets from sub-analyses. By taking advantage of associativity, this capability avoids data duplication. The purpose is to stimulate collaboration by encouraging the reuse of preprocessing analysis data between partners and to improve performance.
- Bolt-on user-defined connections created between two points authorized without revolution geometry — Previously, the tightening direction was defined only by the revolution axis of one of the geometrical supports of the analysis connection. Now, if the connection supports are defined between two points, the bolt tightening connection property can be defined using those two points. The definition of the geometry with its revolution axis is no longer mandatory. This enhancement gives the user more flexibility in defining bolt connections.
- Face-to-face contact between non-coincident surfaces — Prior to V5.16, the user needed to have 3D body elements in order to define a face-to-face contact. With V5.16, if a gap exists between the 2D bodies, the user can create a face-to-face contact using 2D bodies as supports of the contact. This improvement gives the user more flexibility in defining face-to-face contacts according to the type of finite element models involved.

CATIA — FEM Surface 2 (5691-FMS, 5693-FMS)

- Duplication element checker — This new function automatically identifies elements that share the same nodes. It is very useful in checking for duplicate elements after a 2D/1D extraction or an import operation from an external system.
- Connection summary for an assembly —This capability permits pertinent and easy analysis of connections between analysis documents or between mesh parts. By selecting an analysis document, the user can visualize and get information from all analysis documents that are connected to the selected document within the assembly. Likewise, by selecting a mesh part, the user can visualize mesh parts that are connected to the selected mesh part.
- Group definition by boundaries —This release adds to FMS the same capability that is available in the ELFINI Structural Analysis (EST) product.
- Group usage in the **Move node** operator —Groups can be used in the **Move node** operator to associate orphan finite element models with geometrical parts by connecting each mesh node to a geometrical point. The purpose of this enhancement is to resolve ambiguity when several mesh nodes are in the same geometrical location. The user can select a group of nodes under a mesh part instead of the entire mesh part and then employ a user-defined tolerance to associate each mesh node to a geometrical point near to it. This enhancement improves productivity during the CATIA V4-to-V5 migration process.
- Null size element quality checker — A new option, "Maximal Gap," is introduced to check BAR connection elements for excessive lengths. This criterion applies only to NSBAR (Null Size BAR) elements that are generated by mesh parts. With this new option, the user can detect the maximum gap between the two extremities of NSBARs present in the finite element model. This capability permits validation of the quality of the connections between analysis parts.
- Persistent **Save** and **Undo/Redo** on **Shrink visualization** options — This improvement increases customer productivity by taking advantage of a persistent save of all shrink adjustments parameters and of undo/redo capabilities when shrinking visualizations.

CATIA — ELFINI Structural Analysis 2 (5691-EST, 5693-EST)

- Linear combination of load and mass —Users can now create combined load-mass entities which are a linear combination of existing loads and masses. This allows users to define a load-mass once and then reuse it to define many derived load-masses. This capability is especially useful for vehicle-level processes. It promotes collaboration and boosts performance.
- Distribution of mass and inertia —Previously, analysis of the distribution of inertia was only possible on a single virtual part. Now, inertia and mass distribution analysis is possible using multiselected geometries or groups that belong to different mesh parts. This enhancement extends the capability of mass and inertia distribution analysis.
- Group definition using geometrical boundaries — It is now possible to create surface groups of 1D or 2D elements linked to a selected support.
  - Group all 2D elements that are located within an area bounded by previously defined boundary curves. The 2D elements will be linked to the selected support.

- Include all 1D elements that are located within an area bounded by previously defined boundary curves or previously defined boundary points. The 1D elements will be linked to the selected support.

This enhancement offers powerful capabilities for improving productivity by extending the associativity update capability between design changes and analysis.

- Multiple load case management — This new type of load set case references several load sets defined in other cases. All of these load sets are computed in the same analysis case. The solution of a static case containing a multiple load set is a multi-occurrence in which each occurrence corresponds to a load set.

  In the context of vehicle-level processes, users deal with large volumes of preprocessing data. Multiple load case management offers the ability to assemble a large number of loads in one multiple load set. This capability improves performance by consolidating the load cases to be managed.

- Sensors on load and mass sets — Sensors are currently available to measure solution output values, such as displacement and stress. The purpose of this improvement is to supply sensors to measure input values, such as those resulting from applied loads and inertia. It incorporates a tool to quickly check if applied loads and inertia are consistent. The availability of this function in CATIA V5 facilitates migration from CATIA V4, where it already exists, to CATIA V5.
- Visualization of loads and mass sets evaluated at the node degree of freedom — Users can now visualize how the solver interprets the preprocessing specifications (loads and masses) before launching a computation.
- Transfer of loads/displacements from assemblies to components — Computed loads/displacements can now be transferred from an assembly to a sub-analysis or from one finite element analysis (FEA) component to another. This new capability reinforces collaboration between OEMs and suppliers, making it easier to exchange engineering packages.
- Visualization of material names in images — The visualization function now displays a discrete color map of the material in which each material is represented by a specific color and label.
- Option to exclude structural mass when additional mass has been defined — The structural mass can be excluded from the total mass during solver computation.
- Axis for 3D orthotropic properties defined by a surface and a direction — This new capability extends the definition of an accurate axis for 3D properties to complex 3D shapes.
- Advanced post-processing visualization options for text and symbol images — This enhancement ensures better visibility of text and allows the user to choose the text size and the text position. It gives users control over improving the visualization of a symbol image's text.
- Option to add mesh part identifiers in the export data image — The export data image allows the export of image content into an external file (.txt, or .xls file). With this new option, the user can also export the name of the mesh part, thereby increasing the accuracy of information transmitted to partners in the external file.

Machining Solutions

CATIA — Lathe Machining 2 (5691-LMG, 5693-LMG)

V5.16 extends tool support for turning operations. The product now supports the increasingly used Trigon (WNMM ISO code) inserts and groove insert-holders to optimize cutting data and boost productivity.

CATIA — Prismatic Machining 2 (5691-PMG, 5693-PMG)

The product now offers the capability to define, model, program, and simulate machine-tools with external axes CATIA machining. It extends the scope of supported machine kinematics with:

- Management of axes names defined for CATIA machines for automatic generation of transition paths.
- The ability to define manual machine rotation using CATIA machines.
- In conjunction with the new CATIA NC Machine Simulation product (MSG), the ability to manage machines with peripheral and parallel axes. Machine instructions offer the capability to move/lock a machine axis. Automatic transition path generation implements these new machine instructions.

CATIA — Advanced Machining 2 (5691-AMG, 5693-AMG)

For enhancements in 2.5-axis milling and drilling, see **CATIA — Prismatic Machining 2 (PMG)**.

CATIA — NC Manufacturing Review 2 (5691-NCG, 5693-NCG)

- Revision management through the ENOVIA Manufacturing Hub — V5 Machining applications now automatically manage design changes made in the ENOVIA Engineering Hub through the Manufacturing Hub. This enhancement makes it easy to re-version scenarios (to switch from product version 1 to product version 2) in the Manufacturing Hub context.
- ISO code based video simulation — This enhancement validates machining programs with unprecedented realism and accuracy, thanks to material removal video simulation based on ISO code. This function requires a license for CATIA — NC Machine Simulation 2 (MSG).

CATIA — NC Manufacturing Verification 2 (5691-NVG, 5693-NVG)

ISO code based machined part analysis — This enhancement allows users to analyze a machined part with better accuracy. This function requires a license for CATIA — NC Machine Tool Simulation 2 (MSG).

CATIA — Prismatic Machining Preparation Assistant 2 (5691-MPA, 5693-MPA)

- Additional feature management through technological results (created in the Part Design workbench) — Feature recognition capabilities are enhanced in order to retrieve technological information (thread, tolerances) that have been propagated and to associate them to created machinable axial features. This enhancement allows the user to obtain technological information for transformed (translated, rotated, mirrored, patterned) Part Design hole and thread features and for multi-model geometry without loading all the original documents, which is key for mold and die machining.

- Merge coaxial holes — Coaxial holes with the same diameter can now be merged automatically or manually to fit a particular machining methodology.
- Patterns — New color criteria are available for use in automatic pattern creation.

Infrastructure Solutions

CATIA — ENOVIAVPM Supply Chain Engineering Exchange 2 (5691-EWE, 5693-EWE)

Note: In earlier releases, this product was called CATIA —ENOVIA Work Package Exchange 2. It is renamed in this release.

- Reconciliation for analysis information —Suppliers are increasingly asked to deliver a comprehensive definition of their design, one that contains drawings or analysis information. Reconciliation of analysis specification and results information for both parts and assemblies enables integration of the supplier's analysis of the product reference.

  It extends and promotes the assembly of analyses, which is an important contributor to concurrent engineering practices in PLM: It significantly reduces the time required to analyze large assemblies represented by single part analyses by introducing a new, innovative process for combining individual analyses into an overall analysis assembly.

- Reusable XML file generation for reconciliation automation — Using the simulation mode for reconciliation, which previously built the reconciliation without saving the result, now generates an XML file that contains all the information needed for re-use in the next batch of simulations. Because the completed XML file can be used directly for reconciliation purposes, it promotes timely back-and-forth exchanges between an OEM and its suppliers.
- Sort tree capability for the reconciliator — An assembly structure and the existing structure in ENOVIA with which it is to be reconciled can be visualized in parallel windows (Instance View). This side-by-side visualization of instance tree structures enables the user to easily map the CATIA part onto its ENOVIA counterpart by selecting an instance of the part in the CATIA tree viewer followed by an instance of the part in the VPDM tree viewer. The same sorting capability by instance name can then be applied on both tree, enhancing usability and productivity.
- Microsoft Windows support for the reconciliator —This enhancement allows companies to perform the reconciliation on more supported operating system environments for ENOVIA VPM.

CATIA — Object Manager 2 (COM)

Enhancements are the same as for CATIA — Object Manager 1.

CATIA Platform 3 (P3)

Withdrawn configurations: Effective immediately, the CATIA — Automotive Class A 3 Configuration (5691-AC3) and its included product, CATIA — Automotive Class A 3 (ACA), are withdrawn. For migration paths, contact your IBM representative. This configuration is not delivered on the CATIA V5.16 media.

Withdrawn products: Effective November 8, 2005, the CATIA — Automotive Class A Optimizer 3 Product (5691-ACO) and the CATIA — Automotive Class A Optimizer 3 AOP (5693-ACO) are withdrawn. For migration paths, contact your IBM representative. These products are not delivered on the CATIA V5.16 media.

Enhanced products

Mechanical Design Solutions

CATIA — Aerospace Sheetmetal 3 (5693-ASL)

Beginning with this release, this product is available as a P3 AOP.

- New set of parameters for flanged hole — All parameters of the flanged hole feature are now customizable and can be set through design tables to respect company standards and increase productivity.
- Standard management of ASL feature characteristic curves for drawing and manufacturing purposes — Characteristic curves of sheetmetal parts (Bend Tangent Line, Inner Mold Line, Outer Mold Line) are now customizable (line type, thickness, and color) through an XML standard file (only for the CATIA administrator) to respect company standards. These characteristic curves are displayable in 3D and in the drawing, enabling a better understanding by the manufacturing department.
- Support of two flanges on corners —This enhancement enables users to create two flanges on a corner without modifying the web for better productivity. The user can then create a corner relief or a cutout to remove the sharp vertex in the corner.
- Second outer mold line characteristic curve: ASL now automatically generates second outer mold lines, avoiding tedious manual characteristic curve creation.

CATIA — Composites Design 3 (CPD)

- Top surface from zones (shape morphing for transition zones) —When creating the top surface from zones, it is now possible to create the surface corresponding to the transition zones with a new shape morphing operator option that takes into account the inner mold surface curvature. This improvement ensures a more realistic transition zone representation.
- "Full plies" associativity with zones contours — Plies design becomes fully associative with zones design. Full plies (the engineering edge of a part) are now associative with zone contour definition.
- Cut-piece management — The core sample, ply exploder, numerical analysis, and limit contour functions now take into account cut pieces resulting from splicing operations.
- Flattening command — Multiple point selection allows the user to create flattened features for several plies with one location point for each ply. This enhancement improves the visualization of flattened plies and for nesting preparation.

Note: To address strong customer requirements, the CATIA composites data model will be updated using the Release 16 service pack process. To ensure data compatibility, it is highly recommended that CATIA V5.16 customers use only Service Pack 1 or later for composites work.

Shape Design Solutions

CATIA — Automotive Body in White Fastening 3 (ABF)

- New body-in-white command — Generate and attach to any body-in-white fastener a 3D fastener and 3D fastener series annotations to enable the display of:
  - Weld ID and weld parameters and the weld series and fastener series count number
  - Fastener-joined parts or user-defined text information in a comment
  - User-defined complementary drafting symbols — Body-in-white fastener annotations can be extracted to a CATIA drafting document. Extracted drafting annotations are associative to body-in-white fastener design changes.
- This release completes "3D Master" process coverage, making the 3D the master representation, especially when driving production documentation.

Product Synthesis Solutions

CATIA — Business Process Knowledge Template 3 (BKT)

Beginning with this release, this product is available as a P3 AOP. Enhancements are the same as for Business Process Knowledge Template 2.

Analysis Solutions

CATIA — Tolerance Analysis of Deformable Assembly 3 (5693-TAA)

Beginning with this release, this product is available as a P3 AOP.

- Input defect referencing a digitized part — Instead of manually creating input deviation data to simulate defects from a digitized manufactured part, the user can now use Realistic Shape Optimizer features to model deformations. This approach improves productivity when creating a deformation and delivers a powerful method for easily simulating defects. Likewise, in conjunction with the Generative Shape Optimizer shape morphing function, it can be used to model defects.
- Gravity modeling — This new function takes gravity into account during a simulation. Gravity is a new object created in TAA by the end user. Two types are available:
  - *Gravity annotation*, which is stored in the component document under the annotation set. It describes the operating condition of the measurement operation.
  - *Gravity load*, which is stored in the CATProcess document under a new separator. It describes the operating condition of the assembly operation.

  Gravity modeling enables users to perform more accurate simulation during an assembly process to obtain a better prediction of assembly quality.

- Contact management enhancements
  - Enhancement of contact representation — The contact specification orientates its constraint from the first component to the second in order to distinguish the interference case (negative clearance) from the non-interference case (positive clearance) and traction forces (positive forces) from compression forces (negative forces).
  - Direct assignment of a contact to an activity — Previously, a contact could be assigned to a process only through a fastener. With V5.16, the assignment is done directly between the contact and the activity.
  - Contact restitution — The user can obtain the local component-to-component distance and reaction force relative to any contact point at any activity that manipulates the two components of the contact.
  - Contact check option — After computation, a message appears indicating the name of the first activity that has invalid contacts.

  These enhancements extend the capabilities of contact management for tolerance analysis of deformable assemblies.

Infrastructure Solutions

CATIA — Object Manager 3 (CO3)

Enhancements are the same as for CATIA — Object Manager 1.

Common products

These products are announced by other PLM brands but can be ordered with CATIA V5.

Enhanced common products

Shape Design and Styling Solutions

Real Time Rendering 1 (5693-RT1)

3D XML support for applied materials — This capability enables materials applied to product references or product instances to be saved and subsequently opened in the 3D XML file format.

Note: Only graphical properties can be saved.

(This function will become available via the service pack process.)

Real Time Rendering 2 (5691-RTR, 5693-RTR)

V5.16 adds support for new shaders through nVIDIA CgFX1.4. It embeds the advanced shading language provided by nVIDIA to obtain the maximum benefit from best-in-class graphic cards (nVIDIA, ATI) and then offer impressive realistic renderings of V5 designs. This enhancement:

- Improves the optical behavior of the existing materials provided in V5 with:
  - Native support for bumps, reflection maps, and texture
  - Per-pixel lighting to avoid jagged edges on spotlights
  - Level of detail (LOD) management to optimize performance on textures and bumps
- Extends the number of materials available to include new metallic car paint, leathers, varnished woods, and plastic and chrome materials.
- Enables users to develop their own materials and include them in the V5 material library.

Photo Studio 2 (5691-PHS, 5693-PHS)

This release certifies Photo Studio 2 for 32-bit and 64-bit application on the Microsoft Windows XP 64-bit platform. Photo Studio 2 requires extra memory when an assembly is loaded for image computation. 64-bit certification:

- Avoids disk swapping and increases performance dramatically.
- Enables the generation of photo-realistic images for very large assemblies that could not be computed on 32-bit operating systems.

Product Synthesis Solutions

DMU Space Analysis 1 (5693-SP1)

V5.16 enriches 3D XML with DMU content. Sections can now be embedded in a 3D XML file for exchange purposes. (This function will become available via the service pack process.)

DMU Fitting Simulator 2 (5691-FIT, 5693-FIT)

- 3D XML
  - Enriching 3D XML with DMU content — In V5.16 animations can be embedded in a 3D XML file for exchange purposes (save, open, insert).
  - 3D XML specifications — Users can now exploit 3D XML animation schema and extend/customize them or add applicative or user data to the format.

  These functions will become available via the service pack process.

- Highlight currently simulated actions in the sequence editor — Users can now easily identify the current set of actions that are being simulated in a sequence. This capability provides users with quick feedback on actions to address or modify, based on the analysis results obtained during the simulation of a sequence.
- Simulate attached sections during move operations — This enhancement allows the user to analyze the section curves for minimum distance and clash when editing a **Move** track or **Move** activity or when running a simulation.

DMU Optimizer 2 (5691-DMO, 5693-DMO)

Performance and capacity are improved in V5.16 with the ability to generate an optimal wrapping around the interpolated trajectory of vibrating parts in a large assembly. A wrapping of up to 1.2 million triangles can be computed to surround a part that is moving along up to 500 positions on a track.

Fastening Review 2 (5691-FAR, 5693-FAR)

Body-in-white fastener functional annotations — In V5.16, users can visualize body-in-white fastener functional annotations as created in CATIA — Automotive Body-In-White Fastening 3 (ABF), including 3D fastener annotations (such as weld ID and weld parameters) and 3D fastener series annotations (such as weld series ID and fastener series count number). This capability is especially useful for sharing information and enriching documentation.

Human Builder 2 (5691-HBR, 5693-HBR)

- Manikin reality — V5.16 brings human manikins closer to reality by adding the ability for the manikin to look at its moving hand in the reach and inverse kinematics modes.
  - The manikin can “see” what it is about to reach for or manipulate.
  - The head or line-of-sight is oriented interactively in the direction of the working hand.
  - A new vision inverse kinematics behavior for the line-of-sight simulates the movement of the eyes only.
- Manikin appearance reuse — Users can customize the eyes, mouth, and hair of a manikin and store a manikin’s segment colors and head appearance in catalogs for future use.
- Multiple offsets per segment — Users can now create multiple offsets per segment and make them available to knowledgeware tools for reuse.
  - A new **Offsets** command creates offsets on any segment.
  - New offsets can be stored in catalogs and retrieved when needed. This capability enables the user to quickly change the manikin segment’s position, which is useful, for instance, when changing grabbing tools.
  - New reporting capabilities for all analyses, including the new vision analysis
    - The report is updated when the analysis evolves.
    - The output format can be TXT of HTML.
    - An option allows the user to specify a custom ID for each report update.
    - The vision analysis report contains the length of the line of sight (focal distance) and a snapshot (image) of the manikin’s vision window.
- New catalogs for offsets, angular limitations, appearance, and reports — This capability extends and facilitates the reuse of offsets, angular limitations, appearance, and reports through catalogs. This enhancement improves performance when reusing data for multiple manikins, as well as when applying modification on manikins in different scenarios.
- Interference analysis for manikins in interactive manipulation — Interference objects (that is, collision queues) can be used when manipulating a manikin in direct mode (outside simulation). This enhancement better integrates clash capabilities with the usual manipulation tools (posture editor, inverse kinematics, reach mode).
  - The user can activate/deactivate all clash detection queues through the standard user interface.
  - Full visual highlighting of the manikin clash analysis (instead of red curves) is performed when an object attached to the manikin collides with another part within its trajectory.
- Fix-on and fix constraint usability — This release improves the usability of the fix-on and fix constraints by adding the ability to re-establish a new target offset (position and orientation) between the segment and the object (fix-on) or the world (fix) by simply creating another constraint at the new location.
- Properties panel ergonomics — The Properties panel has been modified to display only one level of tabs for selected nodes.

Human Measurements Editor 2 (5691-HME, 5693-HME)

This release enables the user to change a population with respect to values or percentiles. In prior releases, this function was available as **Copy/Paste behavior** in the **Options** choice in the **Tool** menu. Only the name has been changed for better understanding.

Human Posture Analysis 2 (5691-HPA, 5693-HPA)

- Customized joints — This release adds the ability to create customized joints for preferred angles and postural scores.
  - Virtual joints can be created with a set of upper body segments of the spine and the head for postural scores analysis.
  - Back posture and load can be analyzed to prevent work injuries.
- User interface enhancements to improve usability performance
  - Preferred angles and angular limitations are merged in the user interface.
  - The lock function is integrated with the angular limitation function. Angular limitations are now easier to consult and more coherent.
  - More than one degree of freedom (DOF) can be accessed at a time in the Human Posture Analysis workbench.

Human Activity Analysis 2 (5691-HAA, 5693-HAA)

- Posture analysis — V5.16 implements better identification of problematic postures through direct manikin segment coloring for Rapid Upper Limb Assessment (RULA) analysis. Two types of coloring are available for the RULA analysis:
  - General — For two body segment groups
  - Detailed — For individual segments
- Productivity — The **Load** command is now available in the Human Activity Analysis workbench to eliminate repetitive back-and-forth operations between it and the Human Builder workbench.

MultiCAx Plug-ins

MultiCAx PD Plug-in (5691-PDL)

This release:

- Adds support for ProE WildFire2 data.
- Certifies the plug-in for 32-bit and 64-bit operation on the Microsoft Windows XP 64-bit operating system.

MultiCAx ID Plug-in (5691-IDL) — V5.16 certifies the plug-in for 32-bit and 64-bit operation on the Microsoft Windows XP 64-bit operating system.

MultiCAx UD Plug-in (5691-UDL) — This release adds support for UG NX2 data.

MultiCAx STEP Plug-in (5691-DTL) — V5.16 certifies the plug-in for 32-bit and 64-bit operation on the Microsoft Windows XP 64-bit operating system.

MultiCAx IGES Plug-in (5691-DGL) — V5.16 certifies the plug-in for 32-bit and 64-bit operation on the Microsoft Windows XP 64-bit operating system.

APIs

This release adds APIs for CATIA — Aerospace Sheetmetal Design 3 (ASL) and enhances the APIs for the following products:

- CATIA — 3D Functional Tolerancing & Annotation 2 (FTA)
- DMU Kinematics Simulator 2 (KIN)
- CATIA — Business Process Knowledge Template (BK2, BKT)
- CATIA — Piping Design 2 (PIP)
- CATIA — Electrical Wire Routing 2 (EWR)
- CATIA — PPR PDM Gateway 1 (PX1)
- CATIA — Instant Collaborative Design 1 (CD1)
- CATIA — Object Manager (CO1, COM, CO3)

CATIA WLS V5

Enhanced products

CATIA — User Companion for Mechanical Design (5691-MDS, 5795-MDS)

Note: This product is available only via the CATIA — User Companion for Mechanical Design Configuration (5691-MDC, 5795 MDC).

- A V5.15 to V5.16 update course allows users to self-train on all the new capabilities delivered with the Sketcher, Part Design, Assembly Design, Generative Shape Design (GS1) and Drafting workbenches, as well as on the Mechanical Design Infrastructure. Its extensive coverage of the new release enhancements and its synchronization with product availability make this update course unique in the marketplace.
- The Product Design course structure has been optimized and enriched by a new exercise. Recap exercises for the Part Design course have been reworked and enhanced.

CATIA — User Companion for Hybrid Design Product (5691-HDS, 5795-HDS)

- A V5.15 to V5.16 update course allows users to self-train on all the new capabilities delivered with the Generative Shape Design (GSD) workbench. Its extensive coverage of the new release enhancements and its synchronization with product availability make this update course unique in the marketplace.
- The Surface Design Expert course structure has been reworked and enriched with new exercises.

CATIA — User Companion for Structural Analysis (5691-SAS, 5795-SAS)

The CATIA User Companion for Structural Analysis has been enriched to reflect the V5.16 enhancements of CATIA — Generative Part Structural Analysis.

CATIA — User Companion for Sheetmetal (5691-SMS, 5795-SMS)

A V5.15 to V5.16 update course allows users to self-train on all the new capabilities delivered with the Generative Sheetmetal Design workbench. Its extensive coverage of the new release enhancements and its synchronization with product availability make this update course unique in the marketplace.

User Companion for DMU (5691-DNS, 5795-DNS)

- A V5.15 to V5.16 update course allows users to self-train on all the new capabilities delivered with the DMU Navigator, DMU Space Analysis, and DMU Optimizer workbenches. Its extensive coverage of the new release enhancements and its synchronization with product availability make this update course unique in the marketplace.
- A new master exercise has been created for the DMU Optimizer course.

User Companion for Extended Structural Analysis (5691-EAS, 5795-EAS)

- A V5.15 to V5.16 update course allows users to self-train on all the new capabilities delivered with the Generative Assembly Structural Analysis workbench. Its extensive coverage of the new release enhancements and its synchronization with product availability make this update course unique in the marketplace.
- The master and recap exercises for the Generative Assembly Structural Analysis course have been reworked.
- The recap exercises for the Generative Dynamic Response Analysis course have been reworked.

Companion Development Studio (5691-CDS)

V5.16 delivers:

- New management and lifecycle capabilities for tracking any modification made, down to the slide level.
- A new tool for reviewing Companion pages by screen filtering and modification management.
- A new Microsoft Word-based English spell checker for simulations (assessments, Show Me, and Discover It). Microsoft Word is a prerequisite.
- Installation procedure for deployment projects.

Education support

Schedules and enrollment

Call the IBM PLM North America (PLM/NA) response line at 800-395-3339 for schedules and enrollments.

CATIA Higher Education and Training (HEAT) program

Under the HEAT program, two configurations are available for qualifying institutions to order CATIA V5. CATIA — Educational HEAT Configuration 1 (5691-ED1) and CATIA — Educational HEAT Configuration 2 (5691-ED2) comprise the CATIA V5 products in the P1 and P2 platforms, respectively. When new CATIA P1 or P2 products are announced, they are automatically included in these configurations. When P1 or P2 products are withdrawn in a release, they are removed from these configurations.

An order for the CATIA —Educational HEAT Configuration 1 (5691-ED1) or 2 (5691-ED2) will automatically deliver the P1 or P2 box kit and a set of the MultiCAx CDs (LCD4-7456).

When the CATIA — Educational HEAT Configuration 1 (5691-ED1) or 2 (5691-ED2) is ordered, the currently available version of the CATIA WLS V5 products can also be requested. If the WLS products are requested, the following user companions will be shipped to the HEAT institution:

- CATIA — User Companion for Mechanical Design Configuration (MDC)
- CATIA — User Companion for V4 Mechanical Design Configuration (M4C)
- CATIA — User Companion for Hybrid Design Product (HDS)
- CATIA — User Companion for Structural Analysis Product (SAS)
- CATIA — User Companion for Extended Structural Analysis Product (EAS)
- CATIA — User Companion for Sheetmetal Product (SMS)
- User Companion for DMU Product (DNS)

Technical information

Hardware requirements

Summary of changes for V5.16: This release adds support for:

- The Sun Blade 1500+ system unit.
- The SGI Terzo workstation and R16000 processor.
- The HP Fire GL T2-128 and Fire Gl X3 graphics adapters.

Common hardware requirements: The following requirements are common to all operating systems supported by this release. System unit and graphics adapter requirements are platform-specific and are specified in subsequent topics below.

- Required components and features
  - Disk drive: An internal or external disk drive (minimum recommended size is 4 GB) is required to store program executables, program data, the user environment, and to supply paging space. Installation of all CATIA V5 products requires 2.0 GB on Windows, 2.4 GB on AIX®, 2.7 GB on HP-UX, 2.5 GB on IRIX, and 2.3 GB on Solaris.
  - Memory: 256 MB of RAM is the minimum recommended for all applications. 512 MB of RAM is recommended for DMU applications on large assemblies and for the Digitized Shape Editor 2 (DSE) product. Requirements may be greater when large amounts of data are used.
  - Internal/external drives: A CD-ROM drive is required for program installation and for access to the online documentation, which can optionally be downloaded to disk.
  - Display: A graphics color display compatible with the selected platform-specific graphics adapter. The minimum recommended size for usability reasons is 17 inches. Minimum resolution for Windows workstations is 1024 x 768 and 1280 x 1024 for UNIX® workstations. When selecting a graphics adapter, hardware texturing capability is strongly recommended when using CATIA V5 products that use texture mapping, in which case the amount of texture RAM has to be adequate for the number and complexity of textures to be used.

- Keyboard: A specific keyboard compatible with the selected installation locale may be required for national language support.
- Pointing device: Three-button mouse. On Windows workstations, a two-button mouse may alternatively be used (the third button is emulated with a keyboard sequence). The three-button mouse is recommended for usability reasons. IntelliMouse (two buttons plus a wheel instead of the third button) is an alternative to the three-button mouse on Windows workstations. The wheel acts as the middle button to allow additional manipulations, such as panning and scrolling.

- Optional components and features: For CATIA P2 and P3, SpaceBall or SpaceMouse, in addition to the standard mouse, can be used to perform graphic manipulations (zoom, pan, rotate). The required drivers are delivered with these devices. Support of these devices is also available with the DMU Navigator 1 and DMU Space Analysis 1 P1 products.

The robustness of the overall solution is dependent on the robustness of the operating system and the hardware environment used. Windows and UNIX hardware configurations certified by Dassault Systemes for running CATIA products are published, respectively, at

http://www.3ds.com/implementation/technology/windows/certified-workstations-list/

http://www.3ds.com/implementation/technology/unix/unix-based-workstations/

Although CATIA V5 products might run on configurations or with graphic adapters other than those specified for each of the platforms below, incidents specific to such configurations or adapters will not be accepted for support.

Hardware requirements for Windows 2000 and Windows XP

- System unit: An Intel Pentium™ III- or Pentium 4-based workstation running Microsoft Windows 2000 Professional Edition or Windows XP Professional Edition.
- Graphics adapter: A graphics adapter with a 3D OpenGL accelerator is required.

  Note: Graphics performance on local transformations (panning, zooming, rotating model) will depend on the selected graphics adapter. The graphics adapter should have the following capabilities:

  - 24-bits, true color, double buffered visual
  - 24-bits, Z-buffer
  - Stencil buffer
  - Minimum supported resolution: 1024 x 768; a resolution of 1280 x 1024 is recommended for usability reasons

- Network adapter: An active LAN adapter (Ethernet or Token Ring, installed and configured) is required for license key purposes.

Hardware requirements for IBM AIX

- System unit: Any RS/6000®, based on PowerPC 604™ (166 MHz minimum clock speed), POWER2™, POWER3™, or POWER4™ processor families, supported on AIX V5.2 or AIX V5.3.

  Note: Standard support for AIX V5.1 will end on April 1, 2006. To ensure a consistent level of maintenance and service for CATIA and ENOVIA, V5.15 is the last release to support AIX V5.1. Additionally, because end of support (EOS) for AIX V5.2 is September 1, 2007, customers installing CATIA V5.16 are encouraged to migrate to AIX V5.3.

- Graphics adapter: One of the following graphics adapters is required.
  - GXT500P
  - GXT550P
  - GXT800P
  - GXT800M
  - GXT2000P
  - GXT3000P
  - GXT4000P
  - GXT4500P
  - GXT6000P
  - GXT6500P

Hardware requirements for HP-UX

- System unit: Any B-Class, C-Class, or J-Class workstation based on the PA8000 processor family and supported on HP-UX Version HP-UX 11.11 (HP-UX 11i).
- Graphics adapter: One of the following graphics adapters is required:
  - Visualize-FXE
  - Visualize-FX2
  - Visualize-FX4
  - Visualize-FX5
  - Visualize-FX6
  - Visualize-FX10
  - Fire GL-UX
  - Fire GL T2-128
  - Fire GL X1
  - Fire GL T2-128
  - Fire GL X3

Hardware requirements for SGI-IRIX

- System unit: Any O2, Indigo2, Octane, Octane2, Onyx2, Fuel, Onyx3000, or Terzo workstation based on an R5000, R10000, R12000, R14000, or R16000 processor supported on IRIX 6.5.
- Graphics adapter: One of the following graphics adapters is required.
  - Integrated graphics adapters on O2 workstations
  - Solid Impact, or SI/SE
  - Super Solid Impact, or SSI/SSE
  - High Impact
  - Maximum Impact, or MXI/MXE
  - VPro V6
  - VPro V8
  - VPro V10
  - VPro V12
  - InfiniteReality
  - InfiniteReality 3
  - InfinitePerformance

Hardware requirements for Sun Solaris

- System unit: Any Ultra1, Ultra2, Ultra10, Ultra30, Ultra60, Sun Blade 100, Sun Blade 150, Sun Blade 1000, Sun Blade 1500, Sun Blade 1500+ (1.5 GHz), Sun Blade 2000, or SunBlade 2500 workstation based on the UltraSPARC processor supported on Solaris 8.
- Graphics adapter: One of the following graphics adapters is required.
  - Creator3D

- Creator3D Series III
- Elite 3D (U10-440MHz only for U10 workstations)
- Elite 3D Lite
- Expert 3D
- XVR-500
- XVR-600 (Requires Solaris 8 H/W 05/03 and OpenGL 1.3)
- XVR-1000
- XVR-1200

Hardware requirements for WLS: There are no specific requirements beyond those for CATIA V5.

Software requirements

Summary of changes for V5.16

- Support for the x86-64-bit architecture on Microsoft Windows XP 64-bit is added.
- The minimum level for HP 11 is HP 11.11 (HP 11i).
- The minimum level for Windows 2000 Professional Edition is Service Pack 4.
- The minimum level for Windows XP is Service Pack 1 or Service Pack 2.
- The minimum level for WMQ is 5.3.
- The level of IBM License Use Management (LUM) shipped with the product is 4.6.8.

Prerequisite for instant collaboration users on Microsoft Windows 32-bit client

Note: Two Microsoft .NET software packages are a prerequisite for instant collaboration users. CATIA users working in a Microsoft Windows 32-bit client environment are affected by this prerequisite.

The Microsoft software prerequisites are as follows:

- Microsoft .NET Framework Version 1.1 Redistribution Package
- MicrosoftVisual J# .NET Version 1.1 Redistribution Package

To download free of charge from the Microsoft site, visit

http://www.microsoft.com/downloads/details.aspx?FamilyID=e3cf70a9-84ca-4fea-9e7d-7d674d2c7ca1&displaylang=en

Common software requirements: CATIA V5 runs on selected levels of:

- Microsoft Windows 2000
- Microsoft Windows XP
- IBM AIX
- Hewlett Packard HP-UX
- SGI IRIX
- Sun Solaris

Refer to the *Program Directory* or contact your IBM Support Center for appropriate corrective service to apply to the software described below.

Software requirements for Microsoft Windows 2000 and Windows XP: One of the following:

- Windows 2000 Professional Edition with Service Pack 4 or higher
- Windows XP Professional Edition
- For the x86-64 64-bit architecture, Windows XP Professional x64 Edition

With the above, the following components at the minimum levels indicated are required:

- A Microsoft implementation of OpenGL libraries as delivered with Windows 2000 and Windows XP. These libraries may have to be modified to accommodate the selected graphics adapter when installing the graphics adapter and its associated drivers. For recommendations related to driver levels based on certified configurations, visit

  http://www.ibm.com/solutions/plm

- A localized version of the operating system may be required when the selected installation locale differs from Latin 1.

Software requirements for IBM AIX 32-bit and 64-bit

- AIX 5.2 ML02, with the following components:
  - IBM C Set++ for AIX Application Runtime (5765-F56) at a minimum level of 6.0.0. The C Set++ Application Run-time is shipped with the AIX operating system.
  - IBM XL Fortran Runtime Environment for AIX at a minimum level of 8.1.1.
  - OpenGL and GL3.2 Run Time Environment (delivered with the operating system)
  - Common Desktop Environment (CDE; delivered with the operating system).
  - Java™ Run Time Environment (JRE) 1.3.1.

  Or

- AIX 5.3 ML02, with following components:
  - XL C/C++ Enterprise Edition Run-time Environment at a minimum level of V7.0.0 with the IBM C++ Run-time Environment Component for AIX
  - XL Fortran Enterprise Edition for AIX Run-time at a minimum level of V9.1.0
  - Java 1.3.1 32 bit SR7P

Note: Standard support for AIX V5.1 will end on April 1, 2006. To ensure a consistent level of maintenance and service for CATIA and ENOVIA, V5.15 is the last release to support AIX 5.1. Additionally, because end of support (EOS) for AIX V5.2 is September 1, 2007, customers installing CATIA V5.16 are encouraged to migrate to AIX V5.3.

Software requirements for HP-UX: HP-UX Version 11.11 (HP-UX 11i) with the following components at the minimum levels indicated:

- ANSI C++ Runtime Environment (aC++, at a minimum level of 3.30, delivered with the operating system)
- HP Fortran 90 Runtime Environment (delivered with the operating system)
- HP-UX 700 OpenGL 3D API Runtime Environment
- CDE (delivered with the operating system)

A localized version of the operating system may be required when the selected installation locale differs from ISO code pages.

Software requirements for SGI IRIX: IRIX 6.5.15m with the following components at the minimum levels indicated:

- C, C++ and Fortran77 standard execution environment (delivered with the operating system)
- OpenGL (delivered with the IRIX execution environment)
- IRIX Interactive Desktop (delivered with the operating system)
- WorldView when the selected installation locale differs from ISO-1.

Software requirements for Sun Solaris: Sun Solaris 8 HW 05/03, with the following components at the minimum levels indicated:

- C and C++ runtime environment (delivered with the operating system)
- OpenGL runtime environment (delivered with the operating system)
- Fortran runtime environment (delivered with CATIA V5)
- CDE (delivered with the operating system)

Or

Sun Solaris 8 H/W 05/03, with the following components at the minimum levels indicated:

- C and C++ runtime environment (delivered with the operating system)
- OpenGL runtime environment at level 1.3
- Fortran runtime environment (delivered with CATIA V5)
- CDE (delivered with the operating system)

A localized version of the operating system may be required when the selected installation locale differs from ISO-1.

Requirements for interoperability with ENOVIA products

- CATIA V5 and ENOVIA V5 must be at the same release, service pack, and hot fix level for proper interoperability.
- CATIA V5 and ENOVIAVPM interoperability is supported between various CATIA V5 release and ENOVIAVPM PTF levels. Refer to the most recent ENOVIAVPM PTF program directory for the current prerequisite and recommended interoperability PTFs.

Product-specific software requirements for CATIA P2

- CATIA — V4 Integration 2 (V4I) requires, on the CATIA V5 client, for interoperability with CATIA V4 CDM and ENOVIAVPM:
  - If the database server is DB2®,
    - On AIX, HP-UX 11.11 or Solaris: DB2 UDB V8.1 FP 3
    - On IRIX: DB2 UDB V7.2 FP 10a
  - If the database server is Oracle,
    - On AIX, HP-UX or Solaris — Oracle Version 9.2.0.3
    - On IRIX — Oracle Version 8.1.7

  Note: Contact your IBM or Oracle representative for support and planning information on DB2 UDB or Oracle.

  CATIA V4 CDM and ENOVIAVPM interoperability is available through CATIA — V4 Integration 2 (V4I) for the following products:

  - CATIA — Assembly Design 2 (ASD)
  - CATIA — DMU Kinematics Simulator 2 (KIN)

- Interoperability of CATIA V5 on the supported Microsoft Windows platforms with ENOVIAVPM through ENOVIA 3d com requires, on the client side, either (depending whether the database server is a DB2 or an Oracle server):
  - DB2 Universal Database® V8.1 FP3 Client for Windows
  - Oracle Client Version 9.2.0.3 for Windows
- Math Kernel Libraries — When on an Intel architecture in a Windows environment, the performance of the following products can be improved through the use of Intel Math Kernel Libraries at level 5.2 or 6.0:
  - CATIA — Generative Part Structural Analysis 1 (GP1)
  - CATIA — Generative Part Structural Analysis 2 (GPS)
  - CATIA — Generative Assembly Structural Analysis 2 (GAS)
  - CATIA — ELFINI Structural Analysis 2 (EST)
  - CATIA — Tolerance Analysis of Deformable Assembly 3 (TAA)
  - CATIA — Product Engineering Optimizer 2 (PEO)
  - CATIA — Generative Dynamic Response Analysis 2 (GDY)

  MKL Libraries at level 6.0 can be obtained from

  http://developer.intel.com/software/ products/mkl/

  This requirement is optional. Refer to “Before you Begin” in the *Generative Structural Analysis User’s Guide* for additional information.

Specific software requirements for CATIA WLS V5: CATIA WLS V5 has the same browser requirements as CATIA V5. Additionally:

- The Microsoft Virtual Machine must be at a minimum level of 3158.
- Browser Plug-ins — Some WLS products may provide learning resources using the Portable Document Format (PDF). To view these documents, Adobe Acrobat Reader, at minimum level 5.0, must be installed as a plug-in on the browser. The latest level of the Reader can be downloaded from

  http://www.adobe.com/

  Although access to WLS might work on other HTML browsers, incidents specific to browsers other than the products mentioned above are not eligible for support.

- An installation of CATIA V5.16, though not a strict requirement, is recommended to practice the provided exercises automatically loaded by the User Companion Desktop in the active CATIA session.

- For the CATIA User Companion for V4 Mechanical Design product, an installation of CATIA V4 at level R2.1, R2.2, or R2.3 is recommended to practice the provided exercises automatically loaded by the User Companion Desktop in the active CATIA session. For software requirements, visit

  http://www.ibm.com/solutions/plm

The Companion Development Studio Product (5691-CDS) requires:

- Microsoft Windows 2000 Professional Edition or Microsoft Windows XP Professional Edition
- JRE at a minimum level of 1.3.1. The supported version of JRE 1.3 can be downloaded from

  http://java.sun.com/products/archive/index.html

The license management environment is the same as for CATIA V5.

The Companion Development Studio is designed to communicate with other applications. The following are not requirements, but are recommended to take full advantage of the product.

- Access to one of the following browsers is mandatory for the preview of screens:
  - Microsoft Internet Explorer 4.01, beginning with Service 4.01 SP1 (Microsoft VM version number 3158, at a minimum)
  - Microsoft Internet Explorer 5.0 (Microsoft VM version number 3158, at a minimum)
  - Microsoft Internet Explorer 6.0 (Microsoft VM version number 3158, at a minimum)
  - Netscape Communicator 4.7 series, beginning with 4.70

  Note: Access to the screens developed with the Companion Development Studio might be accessible directly through the Companion Desktop. The Companion Desktop has been a license key managed product available with any WLS entry configuration since V5.9. Installation of any WLS standard configuration will provide the Companion Desktop license key and will allow access to customer-developed screens and skillets.

- Access to Microsoft PowerPoint and Microsoft Word (version Office 2000) through OLE automation.

Note: Although different versions of Microsoft PowerPoint and Microsoft Word may work, incidents specific to other than the versions specified above are not eligible for support.

Access to product information: Product information is delivered with the product CDs in HTML format. An HTML browser is required to access this documentation. Online documentation may be installed and used only in the same supported operating environments as CATIA V5.

- In a UNIX environment (AIX, HP-UX, IRIX, and Solaris), Mozilla 1.4 with the Java plug-in installed
- In a Microsoft Windows environment, one of the following browsers is required:
  - Microsoft Internet Explorer at a minimum level of 5.0
  - Mozilla 1.4 with the Java plug-in installed

In addition to a Java-enabled Web browser, the Java Plug-in at level 1.4 is required to search online documentation.

- For AIX, it is delivered with the operating system or can be downloaded from

  http://www.ibm.com/developerworks/java/jdk/aix/service.html

- For HP-UX, it can be downloaded from

  http://www.hp.com/products1/unix/java/java2/jpi/downloads/index.html

- For IRIX, it can be downloaded from

  http://www.sgi.com/products/evaluation/

- For Sun Solaris, it can be downloaded from

  http://java.sun.com/j2se/

- For Microsoft Windows, only the 1.4 Plug-in distributed by Sun is supported. It can be downloaded from

  http://java.sun.com/j2se/

Although access to the online documentation might work on other HTML browsers, incidents specific to browsers other than those specified are not eligible for support.

Product information is also supplied with the product CDs in Portable Document Format (PDF) form. Viewing and printing of the PDF files requires the Adobe Acrobat Reader at a minimum level of 5.0. The latest level of the reader can be downloaded, at no charge, from

http://www.adobe.com

Prerequisites for the license management environment: Microsoft Windows workstations must have an active LAN card (Ethernet or token ring) and TCP/IP installed and properly configured, even in the case of nodelock keys, though for nodelock there is no need to have the workstations connected to the network. No additional license management software is required when accessing nodelock license keys.

IBM License Use Management (LUM) is required to serve concurrent license keys across a network. A LUM configuration file (i4ls.ini) is required on CATIA V5 clients to access concurrent license keys from these servers. Server and Nodelock license management mechanisms are available for CATIA V5 P1, P2, and P3 on all supported operating environments.

IBM LUM level:

- V4.6.8 with Patch 4.6.8.3 is the certified, recommended level.
- V4.6.5 is the minimum level for UNIX, Windows 2000 Professional, Server, Advanced Server, or XP license key servers.
- V4.6.7 is the minimum level for Windows Server 2003 license key servers.
- V4.6.5 is the minimum level when the High Availability Licensing (HAL) mechanism offered by LUM is used.
- V4.6.7, or later, is required when the Concurrent Offline Licensing mechanism offered by LUM is used.
- V4.6.8 supports LUM service across a firewall and other new features.

LUM 4.6.8 with Patch 4.6.8.3 is shipped with CATIA V5. Various release levels of LUM can be downloaded at no charge from

http://www.ibm.com/software/lum

For details about LUM license management for CATIA V5, refer to the **The license management model** section.

Macro replay capabilities: CATIA V5 has built-in macro record and replay capabilities. For UNIX, the interpreter is VB Script 3.0 from Mainsoft. Its components are included in CATIA V5 as shared libraries.

For Windows, the interpreter is either:

- VB Script at a minimum level of V5.0. It is delivered with Microsoft Internet Explorer. VB Script libraries at level 5.0.0.3715 are delivered with Microsoft Internet Explorer 5.0 or at later levels with later versions of Internet Explorer. Use of VB Script is recommended for developing Windows- and UNIX-compatible macros.
- Microsoft VisualBasic for Applications (VBA) at a minimum level of 6.0. VBA is delivered and installed by default with CATIA V5.

  Note: Integration of VBA with CATIA V5 64-bit applications on the Microsoft Windows XP 64-bit platform will become available via the service pack process.

Printer and plotter support

- UNIX — CATIA V5 supports the following plotter/printer languages:
  - CGM-ISO, ATA, CALS
  - Hewlett Packard HP-GL/2-RTL and HP-GL or IBM-GL subsets
  - OCE Graphics GPR50: VDF plotting routines
  - PostScript
- Windows — Printers and plotters are supported through the vendor′s drivers for the targeted printer or plotter relative to the targeted version of the operating system. Contact the printer or plotter vendor for requirements and support.

Batch monitoring using WebSphere® MQ (WMQ) (formerly MQSeries®): Using WMQ communication tools, some batch operations can now be launched remotely. When implemented at the batch level, this optional feature requires WMQ at minimum level 5.3. For availability of client and server components on supported platforms, visit

http://www-306.ibm.com/software/integration/websphere/mqplatforms/supported.html

WMQ Client is required on systems where the transaction is initiated. WMQ Server is required on systems where remote batches are executed.

CATIA V5 general packaging principles

- A CATIA P1 product or a CATIA P1 configuration requires or must include (in the case of configurations) CATIA — Object Manager 1 (CO1). P1 products can be used on P2 or P3, and in such cases, they operate with CATIA — Object Manager 2 (COM) or CATIA — Object Manager 3 (CO3), respectively.
- A CATIA P2 product or a CATIA P2 configuration requires, or must include (in the case of configurations), CATIA — Object Manager 2 (COM). P2 products can be used on P3, and in such cases, they operate with CATIA — Object Manager 3 (CO3).
- CATIA — COM 1 to 2 Extension (5693-C12) allows all P2 products to be added on to P1 configurations. The C12 product effectively upgrades the P1 infrastructure (CO1) to the P2 infrastructure (COM). The C12 product does not change the underlying P1 configuration; included products remain P1. The C12 product can be used as a shareable (5691-C12) or AOP (5693-C12).
- A CATIA P3 product or a CATIA P3 configuration requires or must include (in the case of configurations) CATIA — Object Manager 3 (CO3). P1, P2, and P3 shareable and AOPs are available to P3 configurations as shareable or AOPs.
- License keys for CATIA configurations are acquired and released for the total configuration.
- The functions within a configuration cannot be shared.
- A configuration is required for each CATIA V5 seat.
- CATIA V5 add-on and shareable products may require prerequisite products that are not included in a standard purchased configuration. When a prerequisite product is not included in the selected standard configuration, both the AOP and its prerequisite products must be purchased and included as AOPs within a custom configuration. Prerequisites for shareable products can be satisfied by a standard configuration, by an AOP within a custom configuration, or by a shareable product.
- CATIA — V4 Integration 2 (V4I) is included in all P2 and P3 configurations, and CATIA — V4 Integration 1 (V41) is included in all P1 configurations.

Product-specific prerequisites for products announced in this release

- CATIA — NC Machine Tool Simulation 2 (MSG) requires CATIA — NC Manufacturing Review 2 (NCG). It is strongly recommended that CATIA Space Analysis — 2 (SPA) be used along with NC Machine Tool Simulation (MSG). A CATIA — Space Analysis 2 license is required for the distance and band analysis and clash detection and analysis functions in the context of machine simulation.
- CATIA — Machine Tool Builder 2 (MBG) requires CATIA — NC Manufacturing Review 2 (NCG).

Changed prerequisites: CATIA — Instant Collaborative Design 1 (CD1) requires ENOVIA — VPM Navigator (VPN) in client-server mode.

Planning information

Direct customer support: Direct customer support is offered under a Product Lifecycle Management (PLM) enhanced support contract. This service for a fee enhances the customer′s productivity by providing voice and electronic access to the PLM Support Center. For eligible products, the PLM help desk can answer questions pertaining to the installation, administration, use, and handling of suspected software defects.

For additional information about the enhanced support contract and other available PLM services offerings, visit the PLM Technical Support Web site

http://www.ibm.com/software/applications/plm/support/

Click on the **Enhanced support** link.

Packaging

CATIA V5 packaging: The deliverables associated with each of the three platforms (P1, P2, and P3) are shipped

in separate box kits. The Program Directory is shipped as a CD.

Platform 1 box kit (LK3T-5100)

Ships with all orders for P1 configurations.

- 15 CDs (LCD4-4077)
  - AIX
  - HP
  - IRIX
  - Solaris
  - Windows

  These CDs include P1 products and configurations and the national language product integrated information (PII).

- Program Directory CD (LCD4-5571)
- LUM CD (LCD4-7882)
- Publications
  - English softcopy publications collection kit (SK3T-4124)
  - IBM informal documents
    - Memorandum to Current Users (GI11-4403)
    - Registration Memorandum (GI11-4404)

P2 box kit (LK3T-4132)

Ships with all orders for P2 configurations, 5691-C12, or 5693-C12.

- 20 CDs (LCD4-3756)
  - AIX
  - HP
  - IRIX
  - Solaris
  - Windows

  These CDs include P1 and P2 products and configurations and national language PII.

- Program Directory CD (LCD4-5571)
- LUM CD (LCD4-7882)
- Publications
  - English softcopy publications collection kit (SK3T-4124)
  - IBM informal documents
    - Memorandum to Current Users (GI11-4403)
    - Registration Memorandum (GI11-4404)

P3 box kit (LK3T-7778)

Ships with all orders for P3 configurations.

- 20 CDs (LCD4-5339)
  - AIX
  - HP
  - IRIX
  - Solaris
  - Windows

  These CDs include P1, P2, and P3 products and configurations and national language PII.

- Program Directory CD (LCD4-5571)
- LUM CD (LCD4-7882)
- Publications
  - English softcopy publications collection kit (SK3T-4124)
  - IBM informal documents
    - Memorandum to Current Users (GI11-4403)
    - Registration Memorandum (GI11-4404)

The following table describes the contents of the platform box kits in terms of the installable configurations and products each contains.

Box kit	Installable configurations	Installable products
P3	P2 and P3	P1, P2, and P3
P2	P2	P1 and P2
P1	P1	P1

Shipping package

In addition to the box kits, the shipping package contains:

- Licensed Program Specification (GH52-1328)
- With an order for the CADAM Drafting for V5 (CCD) AOP, the three CCD product CDs (LCD4-4076)
- With an order for one or more MultiCAx products, the eight MultiCAx product CDs (LCD4-7456)

Note: The CATIA API CDs are shipped with CAA RADE as LCD4-7422. Refer to Software Announcement 205-280, dated November 8, 2005, for information on ordering CAA RADE.

CATIA V5 national language packaging: Product integrated information (PII) is included on the world-wide distribution media in the box kits for each operating system at general availability.

National language versions (NLVs) of the softcopy collection kits in French, German, Italian, and Japanese at the current release level will be available after general availability. Each NLV contains six CDs.

- CATIA V5 Softcopy Collection Kit — French (SK3T-4125)
- CATIA V5 Softcopy Collection Kit — German (SK3T-4126)
- CATIA V5 Softcopy Collection Kit — Japanese (SK3T-4127)
- CATIA V5 Softcopy Collection Kit — Italian (SK3T-0575)

CATIA WLS V5 packaging: CATIA user companions for UNIX can be ordered together under a single form number and all user companions for Windows under another. The lists below show which companions are included in the platform deliverables.

- CATIA WLS V5 User Companions for UNIX (LCD4-7846, eight CDs)
  - CATIA V5 User Companion for Mechanical Design
  - CATIA V5 User Companion for V4 Mechanical Design
  - CATIA V5 User Companion for Hybrid Design
  - CATIA V5 User Companion for Structural Analysis
  - CATIA V5 User Companion for Extended Structural Analysis
  - CATIA V5 User Companion for Sheetmetal
- CATIA WLS V5 User Companions for Windows (LCD4-7847, six CDs)

- CATIA V5 User Companion for Mechanical Design
- CATIA V5 User Companion for Hybrid Design
- CATIA V5 User Companion for Structural Analysis
- CATIA V5 User Companion for Extended Structural Analysis
- CATIA V5 User Companion for Sheetmetal

Common WLS product packaging: The DMU WLS products common to ENOVIA and CATIA are packaged as the User Companion for DMU — UNIX and Windows (LCD4-7848, four CDs).

The Companion Development Studio Product is LCD4-5559 (one CD).

Informal documents for WLS

- CATIA WLS Licensed Program Specifications (GI11-1328)
- CATIA WLS Registration Memorandum (GI11-4404)
- CATIA WLS Memorandum to Current Licensees (GI11-4403)

National language versions (NLVs) of WLS products: National language versions of WLS products in French, German, and Japanese at the current release level will be available after general availability. The standard ordering process applies.

- CATIA WLS V5 User Companions — UNIX (six CDs) contains:
  - CATIA V5 User Companion for Mechanical Design
  - CATIA V5 User Companion for Hybrid Design
  - CATIA V5 User Companion for Structural Analysis
  - CATIA V5 User Companion for Sheetmetal

  The form numbers are:
  - LCD4-7857 (French)
  - LCD4-7851 (German)
  - LCD4-7854 (Japanese)

- CATIA WLS V5 User Companions — Windows (five CDs) contains:
  - CATIA V5 User Companion for Mechanical Design
  - CATIA V5 User Companion for Hybrid Design
  - CATIA V5 User Companion for Structural Analysis
  - CATIA V5 User Companion for Sheetmetal

  The form numbers are:
  - LCD4-7858 (French)
  - LCD4-7852 (German)
  - LCD4-7855 (Japanese)

- User Companion for DMU — UNIX and Windows (four CDs) contains:
  - User Companion for DMU — UNIX
  - User Companion for DMU — Windows

  The form numbers are:
  - LCD4-7860 (French)
  - LCD4-7853 (German)
  - LCD4-7856 (Japanese)

Security, auditability, and control

The announced programs use the security and auditability features of the operating systems software. The customer is responsible for evaluation, selection, and implementation of security features, administrative procedures, and appropriate controls in application systems and communication facilities.

Ordering information

Current licensees

Current licensees will receive this update from IBM Software Delivery and Fulfillment automatically.

Should you require further assistance, contact your country IBM representative.

Shipment of this release of CATIA V5 is scheduled to be completed by December 16, 2005; shipment of this release of CATIA WLS V5 is scheduled to be completed by January 27, 2006.

New licensees

Orders for new licenses will be accepted now. For new licensees, shipment will begin on the planned availability date.

An order for CATIA V5 basic licensed programs consists of:

- A licensed standard configuration program order (5691-XXX)
- A licensed custom configuration (5691-XXX) with AOPs (5693-XXX)
- Optional licensed shareable product program order (5691-XXX)
- One system program order (SPO), 5628-ACN

An order for CATIA WLS V5 basic licensed programs consists of:

- A licensed shareable product program order 5691-XXX
- A licensed standard configuration — enterprise option (5795-XXX)
- One system program order (SPO), 5628-WLS

The 5691-XXX and 5693-XXX orders are required for billing and asset registration. The 5628-ACN and 5628-WLS orders are required to ship machine-readable materials and publications.

CATIA V5 and CATIA WLS V5 are available only on CD-ROM.

Basic license: To order a basic license, specify the program number and feature number 9001 for asset registration. Specify the feature number of the desired distribution medium shown below.

New products in this release

Description	Program number
CATIA — Product Knowledge Template 1 Product	5691-KT1
DMU Dimensioning & Tolerancing Review 1 AOP	5693-DT1

(Previously announced in ENOVIA V5.8)

CATIA — Electrical 3D Design & Documentation 1 Product	5691-EC1

New CATIA V5 P2 products

CATIA — NC Machine Tool Simulation 2 Product	5691-MSG
CATIA — NC Machine Tool Simulation 2 AOP	5693-MSG
CATIA — NC Machine Tool Builder 2 Product	5691-MBG
CATIA — NC Machine Tool Builder 2 AOP	5693-MBG

New CATIA V5 P3 Products

CATIA — Aerospace Sheetmetal Design 3 AOP	5693-ASL
CATIA — Business Process Knowledge Template 3 AOP Product	5693-BKT
CATIA — Tolerance Analysis of Deformable Assembly 3 AOP	5693-TAA

Withdrawn configurations and products: The following are withdrawn:

Description	Program number
CATIA — Automotive Class A 3 Configuration	5691-AC3
CATIA — Automotive Class A Optimizer 3 Product	5691-ACO
CATIA — Automotive Class A Optimizer 3 AOP	5693-ACO

There are no replacement products.

Products moved from P2 to P1: CATIA — Circuit Board Design 2 Product (5691-CBD) and CATIA — Circuit Board Design 2 AOP (5693-CBD) are moved from CATIA P2 to CATIA P1 and are renamed as CATIA — Circuit Board Design 1 Product and CATIA — Circuit Board Design 1 AOP. Product identifiers are unchanged.

Renamed products

The following products are renamed as indicated. Product identifiers are unchanged.

Product identifier	Old name	New name
5691 PKT	CATIA Product Knowledge Template 2 Product	CATIA Product Knowledge Template Definition 2 Product
5693 PKT	CATIA Product Knowledge Template 2 AOP	CATIA Product Knowledge Template Definition 2 AOP
5693 EW1	CATIA ENOVIA Work Package Exchange 1 AOP	CATIA ENOVIAVPM Supply Chain Engineering 1 AOP
5691 EWE	CATIA ENOVIA Work Package Exchange 2 Product	CATIA ENOVIAVPM Supply Chain Engineering 2 Product
5693 EWE	CATIA ENOVIA Work Package Exchange 2 AOP	CATIA ENOVIAVPM Supply Chain Engineering 2 AOP

Customization option: Feature code 3444 is used for serial number purposes only; no media or documentation will ship with an initial or MES order.

Product ordering: To order a basic license for a standard configuration or shareable product, specify:

- The configuration/shareable product program number
- The appropriate charge option (PLC/ALC or YLC)
- The platform and quantity

Note: The configurator will automatically add to your order:

- The appropriate billing feature numbers
- Feature number 9001 for asset registration
- The appropriate SPO information

To order a basic license for a custom configuration, specify:

- The standard configuration program feature number
- The appropriate charge option (PLC/ALC or YLC)
- The platform and quantity
- The AOPs (up to a maximum of 80). This support requires LUM at a minimum level of 4.6.5. Previous levels of LUM are limited to a maximum of 17 AOPs.

Note: The configurator will automatically add to your order:

- The AOP indicator codes and 5693 AOP PIDs
- The appropriate billing feature numbers
- Feature number 9001 for asset registration
- The appropriate SPO information

Billing feature numbers for all new configurations, shareable products, and AOPs are contained in this section.

Billing feature numbers for new configurations and products

Note: For order quantities exceeding 250, contact your IBM representative.

New P1 shareable products

CATIA — Electric 3D Design & Documentation 1 Product (5691-EC1)

0011 PLC EC1 Product
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

CATIA — Product Knowledge Template 1 Product (5691-KT1)

0011 PLC KT1 Product
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

New P2 shareable products

CATIA — NC Machine Tool Builder 2 Product (5691-MBG)

0011 PLC MBG AOP
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+

0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

CATIA — NC Machine Tool Simulator 2 Product
(5691-MSG)

0011 PLC MSG AOP
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

New P1 AOPs

DMU Dimensioning & Tolerancing Review 1 AOP
(5693-DT1)

0011 PLC DT1 AOP
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

New P2 AOPs

CATIA — NC Machine Tool Builder 2 AOP
(5693-MBG)

0011 PLC MBG AOP
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

CATIA — NC Machine Tool Simulator 2 AOP
(5693-MSG)

0011 PLC MSG AOP
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

New P3 AOPs

CATIA — Aerospace Sheetmetal Design 3 AOP
(5693-ASL)

0011 PLC ASL AOP
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

CATIA — Business Process Knowledge Template 3 AOP
(5693-BKT)

0011 PLC BKT AOP
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

CATIA — Tolerancing Analysis of Deformable Assembly 3 AOP (5693-TAA)

0011 PLC TAA AOP
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

Withdrawn configurations and products: The following configuration and products are withdrawn effective November 8, 2005. There are no replacement products.

Withdrawn P3 configuration

CATIA-Automotive Class A 3 C
(5691-AC3)

0011 PLC AC3 Configuration
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

Withdrawn P3 shareable product

CATIA — Automotive Class A Opt 3 P
(5691-ACO)

0011 PLC ACO Product
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

Withdrawn P3 AOP

CATIA-Automotive Class A Opt 3 AOP
(5693-ACO)

0011 PLC ACO AOP
0006 ALC User 1-9
0007 ALC User 10-25
0008 ALC User 26+
0001 YLC User 1-9
0002 YLC User 10-25
0003 YLC User 26+

Billing feature numbers for upgrades and migrations: The product upgrade charge applies to orders for a replacement product when it replaces a predecessor product that was obtained for a PLC/recurring charge. The predecessor product is discontinued upon installation of the new version. For purposes of data model migration and validation, a customer may continue to use the predecessor product for up to three months without IBM approval, but may not use both the new version product and the predecessor product simultaneously for productive purposes at any time.

If both licenses are used simultaneously the customer will be subject to the full PLC and recurring charges that apply to the replacement program, as well as the recurring charges applicable to the predecessor program. The three-month parallel usage of the replacement and predecessor product may be extended to more than three months on an individual basis with prior IBM approval.

New upgrade paths to CATIA V5 products from CATIA V4 products

To CATIA V5 primary license charge upgrade feature number	From CATIA V4
0930 5693 GS1 + 0930 5693 FSS	5626 SUR
0930 5693 GS1 + 0930 5691 FSS	5626 SUR
0931 5693 GS1 + 0931 5693 FSS	5626 SUD + 5626 ASU + 5626 FRF
0931 5693 GS1 + 0931 5691 FSS	5626 SUD + 5626 ASU + 5626 FRF

Non-billable indicator feature numbers: No-charge indicator codes are set up under each of the 5691 configurations for which the AOP is available.

New AOP indicator codes

New AOP indicator codes for products announced in this release for each configuration are listed below.

5691-MD1

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-ME1

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-DR1

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-XM1

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-YM1

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-MS2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-MD2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-ME2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-DP2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-SD2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-XM2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-CV2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-HD2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-YM2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-FS2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-SO2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-RE2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-EF2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-ER2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-EI2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-PL2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-SA2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-PM2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-RP2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-MO2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-RM2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-AM2

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-SL3

4361 5693-BKT AOP Indicator Code
4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code
4364 5693-TAA AOP Indicator Code

5691-CD3

4360 5693-ASL AOP Indicator Code
4361 5693-BKT AOP Indicator Code
4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code
4364 5693-TAA AOP Indicator Code

5691-AB3

4360 5693-ASL AOP Indicator Code
4361 5693-BKT AOP Indicator Code
4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code
4364 5693-TAA AOP Indicator Code

5691-KD3

4360 5693-ASL AOP Indicator Code
4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code
4364 5693-TAA AOP Indicator Code

5691-TA3

4360 5693-ASL AOP Indicator Code
4361 5693-BKT AOP Indicator Code
4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

5691-ADD

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

4306 5693-DT1 AOP Indicator Code
4362 5693-MBG AOP Indicator Code
4363 5693-MSG AOP Indicator Code

Configuring orders

Note: This section describes how to configure orders for CATIA V5. For CATIA WLS V5, refer to Software Announcement 202-139, dated June 11, 2002.

To configure orders, specify the platform feature number(s) specified below with a quantity equal to the number of users.

Note: The following platform feature numbers apply only to 5691 program numbers; they do not apply to 5693 program numbers.

Software platform	Feature number
Microsoft Windows	5353
AIX	5350
Hewlett Packard HP-UX	5351
SGI IRIX	5352
Sun Solaris	5354

Ordering examples

Refer to Software Announcement 202-142, dated June 11, 2002, for ordering examples.

CATIA V5 system program order (SPO)

CATIA V5 programs for all platforms (UNIX and Microsoft Windows) are shipped under the 5628-ACN System Program Order. The SPO is required for all program shipments and future updates.

Initial orders placed for a program type 5691 without a corresponding order or MES for the 5628-ACN SPO will either fail order validation, or will not generate a media shipment. Each customer number must have its own SPO.

5628-ACN program feature numbers for new products: Because the products announced in this release do not ship deliverables, no new supply feature numbers are added to the 5628-ACN SPO.

5628-ACN program feature numbers for withdrawn products: Effective November 8, 2005, the program feature numbers for the following products are withdrawn:

Product	Program feature number
CATIA — Automotive Class A3 Configuration (5691-AC3)	1842

Initial orders: When ordering the first 5691 configuration for CATIA V5, an order must also be placed for the 5628-ACN SPO. Both the 5691 program order and the SPO must be for the same CPU system type/system number and must have the same scheduled shipment date.

Subsequent orders: The SPO **must** have a feature number for every CATIA V5 configuration that is installed or on-order. Therefore, when a configuration that was not previously installed is ordered, the SPO must be updated.

To update an on-order system: When a license for a new 5691 program is ordered for an on-order system, the 5628-ACN SPO must be updated to reflect the feature number of the licensed program desired. Also, for asset registration and billing purposes an order for the individual licensed program type 5691 is required.

To update an installed system: When a license for a new 5691 program is ordered for an installed system, the 5628-ACN SPO must be updated to reflect the feature number of the licensed program desired.

5628-ACN media feature number

Environment	Feature number	Distribution medium
All platforms	3410	CD-ROM

CATIA WLS V5 system program order

The following no-charge program feature numbers are used with the 5628-WLS System Program Order to show which configurations and products are installed. This information is used for shipment of media. The maximum quantity of each of the features that may be specified is one for each of the 5691 or 5795 programs.

Environment	Feature number	Distribution medium
UNIX platforms	3410	CD-ROM
Microsoft Windows platforms	3432	CD-ROM

Customization options: Select the appropriate feature numbers to customize your order to specify the delivery options desired. These features can be specified on the initial or MES orders.

For example, if publications are not desired for the initial order, specify feature number 3470 to ship media only. For future updates, specify feature number 3480 to ship media updates only. If, in the future, publication updates are required, order an MES to remove feature number 3480; then the publications will ship with the next release of the program.

Note: Features pertaining to documentation do not apply to WLS.

Description	Feature number
Initial Shipments	
Ship media only (suppresses initial shipment of documentation)	3470
Ship documentation only (suppresses initial shipment of media)	3471
Update Shipments	
Ship Media updates only (suppresses update shipment of documentation)	3480
Ship documentation only (suppresses update shipment of media)	3481
Suppress updates (suppresses update shipment of media and documentation)	3482

Expedite Shipments

Local IBM office expedite (for IBM use only)	3445
Customer expedite process charge ($30 charge for each product)	3446

Expedite shipments will be processed to receive 72-hour delivery from the time IBM Software Delivery and Fulfillment (SDF) receives the order. SDF will then ship the order via overnight air transportation.

For CATIA V5, one English softcopy publication CD-ROM is automatically included in all shipments. National Language Versions (NLVs) of the softcopy publications in French, German, Italian, and Japanese will be available after general availability at no charge. To order, specify the language using the feature numbers below. Only one additional copy of a kit can be ordered per license.

Publication	Feature number
CATIA V5 Softcopy Collection Kit (English)	7100
CATIA V5 Sftcpy Collection Kit (French)	7101
CATIA V5 Softcopy Collection Kit (German)	7102
CATIA V5 Softcopy Collection Kit (Japanese)	7103
CATIA V5 Sftcpy Collection Kit (Italian)	7108

By using the following numbers, additional softcopy collection kits can be ordered through the IBM Publication Ordering System

http://www.elink.ibmlink.ibm.com/public/applications/publications/cgibin/pbi.cgi

- CATIA V5 Softcopy Collection Kit — English (SK3T-4124)
- CATIA V5 Softcopy Collection Kit — French (SK3T-4125)
- CATIA V5 Softcopy Collection Kit — German (SK3T-4126)
- CATIA V5 Softcopy Collection Kit — Japanese (SK3T-4127)
- CATIA V5 Softcopy Collection Kit — Italian (SK3T-0575)

There are no softcopy publications for CATIA WLS V5.

Terms and conditions

Agreement: IBM Customer Agreement

Designated machine: Not required

Variable charges apply: No

Location license applies: No

Use limitation applies: Yes, as implemented by the license management model described.

The license management model

CATIA V5 license management on workstations: In workstation environments, CATIA controls the number of concurrent users of a CATIA configuration or product, according to the number of license keys acquired for the configuration or product.

CATIA V5 delivers identical license management mechanisms on UNIX and Windows environments, based on IBM License Use Management (LUM). The following license management principles apply:

- A CATIA V5 configuration (standard or custom) will require a license key. License keys for CATIA V5 configurations are acquired and released for the total configuration. The products within a configuration cannot be shared.
- Each CATIA V5 shareable product will require a license key, in addition to one for the prerequisite configuration and any prerequisite product, if applicable.
- In all cases, configuration license keys are acquired at the beginning of the process and are released at its termination.
- CATIA V5 add-on and shareable products may require license keys for prerequisite products that are not already included in a standard configuration. Prerequisites for shareable products can be satisfied by a standard configuration, by an AOP within a custom configuration, or by a shareable product. However, because all AOPs are defined within one custom license key, any AOP prerequisites must be satisfied by either a standard configuration or by other AOPs purchased and defined within the same custom configuration.

CATIA V5 can be used in three license management modes: nodelock, with concurrent usage of license keys on a network, or concurrent offline license management.

Nodelock usage: The use of a local display of the hardware configuration is mandatory for CATIA V5 usage in nodelock mode. There is no limit to the number of CATIA V5 processes launched for a given license key (configuration or product). For instance, a user can launch the following simultaneous processes:

- A V5 interactive session
- A V5 process executed through an OLE container application
- Replay of macros recorded from captured sequences of V5 user interactions

In the nodelock mode of operation, only one CATIA license key per configuration and shareable product can be registered by machine, and only one CATIA user can run at a time on that machine. If multiple license keys per configuration or shareable product, or multiple users on a single machine, are required, refer to the **Concurrent usage** section.

Concurrent usage: A user on one machine on one display uses one license key per configuration or shareable product used, regardless of the number of processes. If the display changes, then an additional license key is taken for the corresponding process.

License keys for CATIA configurations are acquired and released for the total configuration. The functions within a configuration cannot be shared.

Dynamic license management: Shareable product license keys may be acquired at the beginning of the session and released at its termination or, upon user request, acquired and released during the session. (The ability to acquire and release licenses is not available for configurations.) Shareable license keys acquired at the beginning of the session cannot be released before the end of the session; only license keys dynamically granted upon user request during the session can be released during the session.

Concurrent offline license management: A concurrent license key control technique is available via the LUM server. It gives CATIA, ENOVIA DMU, DELMIA, and RADE applications running on a Windows laptop the ability to disconnect from the license key server for a specific period of time so that users can take advantage of the full license key capability while mobile. During the checkout period, the license key is unavailable for use by another concurrent user.

This feature is designed to add additional flexibility to a user's work environment. It is offered to accommodate short-term travel needs and collaboration while away from a fixed office environment or server connection. All ICA terms and conditions, including cross-border licensing terms, are unchanged, and users will check-out and check-in license keys at their home server, where rules and procedures are controlled by LUM.

Note to users of the following SMARTEAM-branded CATIA integration products:

- CATIA Team PDM Configuration (TDM)
- CATIA Integration Product (CAI)
- CATIA Xtended Team PDM Configuration (TDX)
- CATIA Xtended Integration Product (CIX)

These and other SMARTEAM products do not support dynamic or concurrent offline license management. Therefore, dynamically releasing a CATIA key will not release any SMARTEAM integration product key that may have been checked out at the same time, and these integrations cannot be used with offline-type LUM certificates.

Educational allowance available: No. CATIA V5 and CATIA WLS V5 products are available under the CATIA Higher Education Academic Training (HEAT) program.

Contact Vijay Srinivasan (vasan@us.ibm.com) or 914-642-6587 for additional information and approvals.

Volume orders: Contact your IBM representative.

Warranty applies: Yes

Licensed program materials availability

- Restricted materials of IBM: None
- Non-restricted source materials: None
- Object Code Only (OCO): All

Program services: For a list of all currently supported releases of CATIA, ENOVIA and SMARTEAM products, visit

http://www.ibm.com/software/applications/plm/support/

Sign on to the site. On the right side, select **End of Support** in the **Announcements** column.

Note: If you have not yet obtained an IBM common registration user ID, visit

http://www.ibm.com/registration/selfreg

Program Services offer a method of reporting code-related problems for CATIA V5 and WLS licensed software products. Program Services are available electronically using the Problem Reporting process at

http://www.ibm.com/software/applications/plm/support/

Sign in, then select **New problem** under Problem Reporting. On the next page, select "Free Support" for the appropriate product. When using the "Free Support" option, all communications will be either through the PMR or by e-mail, and must be in English. (If the customer has ECSR (Electronic Customer Support Representative) enhanced support (reference PRPQ 5799-C88) with the CATIA support feature, select "Contract Support" for the appropriate product.) Customers that do not have enhanced support will only be able to enter data into the PMR; all subsequent communications will be through e-mail. Response time for "Free Support" PMRs will be within two business days.

Not all options of the Product Lifecycle Management (PLM) technical support e-services are available in all countries.

If the problem reported is not known to be a code-related problem, the customer will be informed that work will continue on it provided the customer has ECSR (Electronic Customer Support Representative) enhanced support (reference PRPQ 5799-C88) with the CATIA support feature. Enhanced support for WLS is included in the base product's feature code; no separate enhanced support is required.

For additional information about the enhanced support contract and other available PLM services offerings, visit

http://www.ibm.com/software/applications/plm/support/

Click on the "Enhanced support" link or contact your PLM marketing representative or authorized IBM Business Partner for more information.

Preventive Service is delivered through the next release of CATIA V5 products. The new release also includes corrections to problems, depending on the time of their submission and their severity.

During the Program Services period, Corrective Service for CATIA V5 releases is delivered through service packs on a regular basis. A service pack includes corrections for Blocking Problems in production systems reported on this release and all corrections available for all components at the time it is built. Service packs are provided at the same time for all platforms currently supported. Each service pack supersedes the previous one and may be installed on top of the released level or on top of a previous service pack. (The exception is WLS products where Corrective Service is provided in the next release of the product. No maintenance for WLS products is provided in the service packs.)

Customers may request a correction via a service pack for Blocking Problems. A Blocking Problem is defined as:

- A problem that stops production: The customer is currently using the level for which a fix is requested in a production environment.
- A problem that prevents migration: The customer must provide the migration plan.
- A problem that halts testing of a given level: A fix will allow customer to continue the testing.
- Installation problem: A problem that prevents the customer from installing or using the product.
- Regression: Problems reported as regressions may be due to an operation that was being performed erroneously or created incorrect data with a previous release and the current release no longer permits these operations. Therefore, each problem reported as a regression must be evaluated. True regressions will be handled as Severity 1 problems.

CATIA and WLS V5 Products

Program Services will be provided for this release of CATIA and WLS V5 products until June 2, 2007.

The end of service (EoS) date for CATIA and WLS V5.14 will be May 5, 2006.

IBM Operational Support Services — SupportLine: No

Other support: PLM Support Centers

Prices

Contact your IBM representative for pricing information for this announcement.

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