



CATIA V5.10: From Entry PLM to Custom Applications for Best Practice Deployment and Business Process Optimization

Overview

In V5.10, the new Automotive Body-in-White Fastening Platform 3 (P3) product provides associative definition and management of welding and mechanical fastening. For powertrains, consumer goods, and plastic parts, blend vertex fillets improve user productivity, and access to tolerance values as parameters increases automation possibilities. The new Electrical Harness Flattening product validates the manufacturability of electrical wire harnesses during the design phase.

Large assembly management, drafting, welding, and electrical design are strengthened for the industrial machinery and components industries. New electrical harness design and generative shape design products are added to the entry Product Lifecycle Management (PLM) Platform 1 (P1) solutions portfolio along with the Part Design Feature Recognition product, allowing rapid modification of data from multiple sources (IGES/STEP/MultiCAD). For enhanced productivity for large assemblies, V5.10 introduces visualization mode updates, drafting generation on tessellated representations (CGR), and improved interactive drafting performance.

Definition and capture of best practices and process automation are now possible without specialist programming skills. Business process knowledge templates allow key knowledgeholders to capture and automate critical business process and design and engineering tasks, embedding core-knowledge rules within customer applications that can deliver dramatic improvements in productivity and competitiveness across a wide range of detail processes.

Design-in-context flexibility and usability are greatly improved, providing users with new capabilities to reuse components while preserving relational design integrity.

The CATIA-ENOVIA Work Package Exchange products maximize collaborative design efficiency along the entire supply chain. The P2 product allows OEMs to extract "workspaces" of subsystem data from their ENOVIAVPM product definition and to reconcile subsequent supplier modifications. The P1 product enables suppliers to complete design activities in the context of the work package and then send the updates to their OEM customer.

The effectiveness of collaboration in a multiCAD environment is significantly enhanced with the new Part Design Feature Recognition product and new healing analysis and repair tools for recovering imported geometry.

Key Prerequisites

CATIA V5 runs on selected system levels of:

- Microsoft™ Windows®
- AIX®
- Hewlett Packard HP-UX
- SGI IRIX
- Sun Solaris

Planned Availability Dates

- November 29, 2002, CATIA V5.10
- January 10, 2003, CATIA V5.10 Web-based Learning Solutions (WLS)

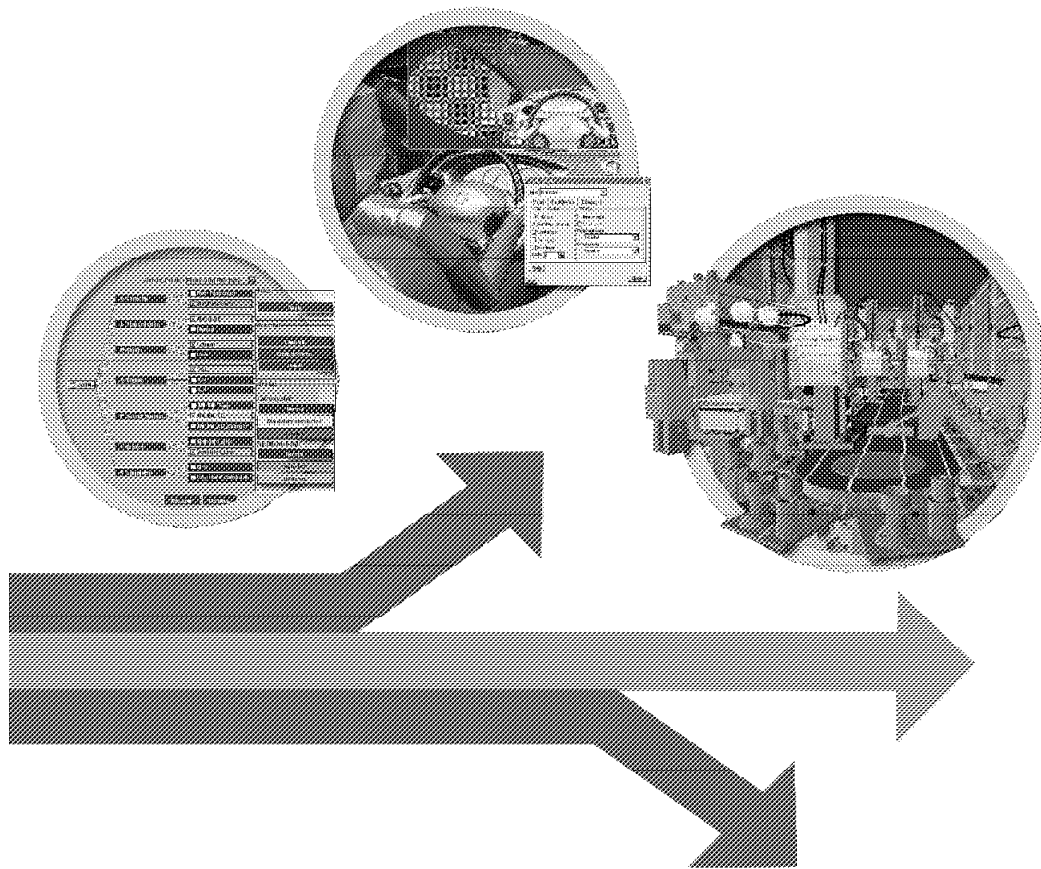
At a Glance

V5.10 increases the lead in end-to-end product development process coverage for automotive, aerospace, consumer goods, and the electrical and electronics industries with 13 new products and 81 enhanced applications. This release:

- Extends entry Product Lifecycle Management (PLM) process coverage, improves SMARTEAM integration, and optimizes solutions for small and medium enterprises
- Adds a new business process knowledge template product to allow companies to interactively capture their best practice processes within custom applications
- Extends the unique Product Process Resources (PPR) model to allow management of additional types of CATIA application data within an ENOVIA-configured product structure
- Enhances the collaborative workspace through extension in publication, assembly links management, and relational design.
- Delivers flexible supply chain collaboration with the CATIA — ENOVIA Work Package Exchange product for supplier sub-system design in the context of the OEM's ENOVIA virtual product definition

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Description



CATIA V5 is the cornerstone of the true integration of people, tools, methodologies, and resources within an enterprise. Its unique underlying PPR model and workplace approach provide a truly collaborative environment that fosters creativity, sharing, and communication. CATIA's built-in capacity to capture and reuse know-how drives the implementation of corporate design best practices and opens opportunities for innovation. In addition to the comprehensive CATIA V5.10 portfolio of over 140 applications, a wide range of tightly integrated third party applications built on the extensive V5 Component Application Architecture are now available.

Process-Centric

V5.10 extends entry PLM process coverage with six new P1 products, improved SMARTEAM integration, and optimized solutions for small and medium enterprises. Dedicated large assembly management, drafting, welding, and electrical solutions are strengthened for industrial machinery and industrial components. Shape design solutions are extended to better address the plastic parts process and to provide dedicated configurations for the consumer goods industries. V5.10 provides a comprehensive stand-alone line of P1 products to address end-to-end identified processes. New tailored electrical harness design and generative shape design products, inheriting from P2 expertise, strengthen our entry CAD/CAM/CAE solutions.

V5 portfolio scalability is improved by such new products as Part Design Feature Recognition to benefit P1 or P2 users working in multiple environments by enabling them to retrieve data in multiple standard formats. Large assembly update in visualization mode, drafting generation on tessellated representations (CGR) with the

support of occlusion culling technology, and improved interactive drafting performance on large data sets open new perspectives for large assembly management. DXF semantic export is delivered in V5.10 to answer a critical need among P1 users. Perfectly suited to the consumer goods domain, the new Styled Mechanical Design 1 (YM1) configuration is a 3D product creation package that includes comprehensive part and assembly design functions, as well as advanced surface creation tools and associative drawing extraction capabilities.

V5.10 increases the lead in end-to-end product development process coverage for the automotive, aerospace, consumer goods, and electrical and electronics industries with 13 new products and 81 enhanced applications that integrate intelligent productive features and dedicated expert solutions. The automotive body-in-white process is addressed through enhancements in tolerancing in the 3D Functional Tolerancing & Annotation 2 (FTA) product. Thickness and orientation surface management is enhanced in the Generative Shape Design 2 (GSD) product. Together, these enhancements improve the global design-to-manufacturing process.

At the same time, the new Automotive Body-in-White Fastening P3 product supports welding technologies and mechanical fastening. Detailed 3D design in powertrain, consumer goods, and plastic parts is strengthened by blend vertex fillets and the ability to generate functional tolerances in feature parameters, and also by sketcher improvements integration. The end-to-end automotive and aerospace harness design process is completed with the new Electrical Harness Flattening product, which enables extensive downstream drafting generation. The manufacturing cycle is accelerated by 3- and 5-axis operation productivity improvements and by the extended process coverage implemented by the new Prismatic

Machining Preparation Assistant (MPA) product, which offers advanced NC feature recognition techniques.

Collaborative Workspaces

V5.10 collaborative design evolutions allow the deployment of relational designs, not only in the conceptual design phase with a few users working in the same room, but up to detailed design with many users working together across the extended enterprise. More efficient collaborative engineering is provided through the extension of publication mechanisms, assembly links management, and relational design. Design-in-context flexibility and usability are greatly improved by providing users with new abilities to reuse components while preserving relational design integrity. A designer can now define contextual assembly links in a minimal context and use the closest common parent product to describe a self-contained sub-assembly that can be easily reused in another context.

The definition and modification of contextual links has been improved through the ability to perform link-by-link replacements in the **Define Contextual Links** panel through 3D selection or specification tree selection. This approach allows a designer to reuse contextual components in a new environment with complete flexibility to reroute contextual links and thus optimize relational design efficiency and productivity.

The CATIA — ENOVIA Work Package Exchange products provide flexible supply chain collaboration. They allow a supplier to work and iterate a CATIA sub-system design compliant with its OEM ENOVIA virtual product definition context. CATIA-ENOVIA Work Package Exchange is a P1/P2 product. The P2 product allows the OEM to define a work context in ENOVIAVPM V4 or V5 and load all associated data in CATIA V5. Then the OEM generates “mobile” files and provides them to the supplier. The supplier then uses a dedicated P1 product to make all data modifications or creations requested by the OEM and to return the updated “mobile” files to the OEM.

Finally, the OEM loads the data provided by the supplier into CATIA V5 and imports it into ENOVIAVPM V4 or V5. This new method greatly enhances collaboration between OEM and supplier by offering a comprehensive product to ensure consistency of PDM information through advanced and productive reconciliation tools.

Engineering methodologies for working in a heterogeneous environment are extended through feature recognition, healing, and multiCAD generative drafting capabilities. The new Part Design Feature Recognition product manages global and local feature recreation from B-representations to facilitate import and modification of foreign geometries for improved engineering collaboration. New analysis and repair tools in healing, such as local join, help users to perform precise and targeted joins for more efficiency in geometry recovery. The new capability to perform drafting operations in the CATIA CGR format (tessellated surface), such as views or sections on multiCAD parts, opens a considerable range of collaboration for designers. Indeed, they are able to carry out complex drafting operations on assemblies made of hybrid parts.

Sharing of best practices is enhanced by the ability to capture and reuse corporate standards. In V5.10, several applications, such as Electrical Harness Flattening 2, Automotive Body-In-White Fasteners 3, and Aerospace Sheetmetal Design 3, are enriched with abilities to integrate corporate standards into the core of the application. In Automotive Body-In-White Fastening 3 for instance, the company administrator can, for each process type, narrow the list of CATIA fasteners standard

attributes to relevant ones. For each, a set of company predefined values can be set up. This enables CATIA users to take advantage of technical expertise in defining their model specifications, and it ensures design quality and reliability through native integration of company standards at the design phase.

The PPR Model

CATIA-ENOVIA integration is extended with enhanced capabilities to capture and manage analysis results, kinematics definitions, and knowledge-driven products. Data inherent to kinematics, analysis, catalogs, digital mock-ups, tolerance analysis of deformable assemblies, and knowledge basis (rules, formulas and checks coming from the Object Manager and Knowledge Advisor), are now available and stored in ENOVIA. Through ENOVIA, design-in-context and concurrent engineering become a reality thanks to publishing (create and save modifications in the database), assembly management, and document-to-document link management.

Tolerance Analysis of Deformable Assembly increases its integration to manufacturing processes by sharing PPR information with DELMIA solutions. Processes and resources defined and managed in DELMIA can now be directly used and simulated by the CATIA — Tolerance Analysis of Deformable Assembly 3 (TAA) product to predict the tolerancing specified on assemblies considering the entire sheetmetal assembly process. Using ENOVIAVPM, it is possible to carry out results storage in a database for collaborative downstream use. CATIA V5.10 integration gives users the opportunity to inscribe PLM in their conception methodology with the flexibility of advanced applications and fluidity of transfer of information, skills and knowledge around the unique PPR model.

Knowledge

V5.10 delivers breakthrough capabilities for customers to build their own process-driven applications. Following the morphing capability introduced V5.9 (user-defined feature and product knowledge templates), the V5.10 Business Process Knowledge Template product enables companies to interactively capture their best practices within custom applications and to deploy them across the enterprise. Definition of best practices, know how, and capture of expertise can be achieved in a simple and intuitive way, with no required development skills. Doing so, an expert is able to express the company’s business processes, automate design and engineering tasks, and embed core-knowledge rules and checks within those tasks.

Those definitions are then presented as vertical applications that embody predefined methodologies derived from corporate best practices. For instance, complex automotive conceptual studies or aerospace engineering processes can be guided by skilled rules and checks. Freed from time-consuming and repetitive design tasks, and benefiting from knowledge-driven automation, users can then unleash their creativity and focus on innovation. The company increases dramatically its competitiveness by using specific knowledge-based environments and workbenches that exactly match its best practices, thus providing better products and considerably reducing costs.

A new constraint satisfaction extension brings innovative perspectives to product engineering optimization. The constraint problem solving function allows users to apply a mix of algebraic and numerical technologies to solve complex hybrid engineering problems. It disassociates the user from the underlying structure and design of the problem, allowing him to deal with it in terms of pure

constraint satisfaction. This brand new conceptual approach offers conceptual design engineers a new and powerful paradigm to abstract the end-user from the structure of the problem and the design, giving them the ability to find the solution that will perfectly suit their geometrical requests.

Component Application Architecture (CAA)

Openness is demonstrated through adoption of the V5 architecture by key players. V5.10 delivers Electrical Library and Team PDM APIs. With the addition of support for HP-UX 11.0 and Windows XP, CAA V5 now supports seven different operating systems to build and run CAA-based V5 applications.

CATIA V5 WLS

V5.10 updates courses to reflect new CATIA V5.10 capabilities, adds a course for Knowledge Expert, and provides additional instructor materials.

Accessibility by People with Disabilities

Owing to the graphics-intensive nature of its engineering design applications, this product has been granted a deviation for 2002.

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 its capabilities allows CATIA V5 to be applied in a wide variety of industries, such as aerospace, automotive, fabrication and assembly, 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. It facilitates true collaborative engineering across the multidisciplinary extended enterprise.

In conjunction with ENOVIA and SMARTTEAM product data management applications, ENOVIA Portal Web-based collaboration tools, and the DELMIA manufacturing process definition and simulation tools, CATIA V5 provides the core capability for IBM PLM Solutions.

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.

CATIA V5's three platforms (P1, P2, and P3) make it the most scalable solution for product creation. CATIA V5 P1 users benefit from 3D PLM productivity in an affordable way with the security of potential growth. They can conduct associative product engineering based on CATIA V5's 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.

Hardware and Software Support Services

SmoothStart™ /Installation Services

SmoothStart/Installation Services will not be provided for CATIA products.

CATIA V5 Services Plan

PLM Services, part of IBM Global Services, provides a robust portfolio of services to assist you 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 Version 5 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>

and click **Services** and then **Offerings**.

Enhanced Support Offering

Direct customer support is provided 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 ENOVIA Solutions V5.10, refer to Software Announcement 202-299, dated October 29, 2002.

For information about SMARTEAM V5.10, refer to Software Announcement 202-301, dated October 29, 2002.

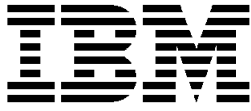
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IBM US Announcement Supplemental Information

October 29, 2002

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 products, visit the PLM Web site:

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

From the menu frame, select **CATIA Products** to get to the CATIA page. From the menu frame on the CATIA page, select **CATIA V5**.

CATIA V5 WLS Packaging and Products

Note: Beginning with this release, announcements about CATIA V5 WLS will be included in announcements about CATIA V5.

CATIA WLS products and configurations can be ordered in two ways, depending on the objectives of the customer:

1. As “shareable” products and configurations:
 - CATIA User Companion for Mechanical Design Configuration (5691-MDC)
 - CATIA User Companion for Hybrid Design Product (5691-HDS)
 - User Companion for DMU Product (5691-DNS)
 - CATIA User Companion for V4 Mechanical Design Configuration (5691-M4C)

Shareable products and configurations are suited to the customer who needs a limited number of licenses available.

These products and configurations are delivered with their own license key, allowing the user to obtain the license at the beginning of the session, or to leave it for another user.

2. As “enterprise” products and configurations:
 - CATIA User Companion for Mechanical Design Configuration —Enterprise Option (5795-MDC)
 - CATIA User Companion for Hybrid Design Product —Enterprise Option (5795-HDS)
 - CATIA User Companion for V4 Mechanical Design Configuration —Enterprise Option (5795-M4C)
 - User Companion for DMU Product — Enterprise Option (5795-DNS)

The terms regarding use of the Enterprise Option include configurations available in the P1 environment.

This licensing option offers a reduced Primary License Charge (PLC) and requires that the customer purchase licenses in a quantity equal to the quantity

of each licensed CATIA configuration or product as described by the following terms:

- The CATIA V5 WLS product 5795-MDC is licensed to customers on a one-to-one relationship between the 5795-MDC product and each licensed CATIA V5 Part Design 1 or Part Design 2 product. This includes each licensed standard configuration (and any future configuration) that contains Part Design 1 or 2, custom configurations built using 5693-PD1 or -PDG AOP and all shared 5691-PDG products.
- The CATIA V5 WLS product 5795-HDS is licensed to customers on a one-to-one relationship between the 5795-HDS product and each licensed Generative Shape Design 2 product. This includes each licensed standard configuration (and any future configuration) that contains Generative Shape Design 2, custom configurations built using 5693-GSD AOP, and all shared 5691-GSD products.
- The CATIA V5 product 5795-M4C is licensed to customers on a one-to-one relationship between the 5795-M4C product and each licensed configuration of 5626-SBD,-ADD, and -DWD.
- The User Companion for DMU Product (5795-DNS) is licensed to customers on a one-to-one relationship between the 5795-DNS product and each licensed CATIA — DMU Navigator 1 add-on product (5693-DN1) or CATIA — DMU Navigator 2 shareable product (5691-DMN) or add-on product (5693-DMN) and also each licensed configuration (and any future configuration) that includes the CATIA — DMU Navigator 1 or DMU Navigator 2 product.

Once the minimum requirements are met, a customer may also buy additional WLS licenses to support other CATIA V5 or V4 licensed configurations not listed in the enterprise option terms.

All subsequent orders for the applicable licensed configurations must be accompanied by an equivalent order for the supporting CATIA User Companion. The price of an enterprise seat is greatly reduced compared to the price of a shareable order.

There is no minimum seat requirement for CATIA WLS Enterprise Option products (5795-MDC, 5795-HDS, 5795-M4C, 5795-DNS).

Note: The following rules apply to the programs listed above:

- When the enterprise option is deployed, the shareable option is no longer necessary.
- A 5691 product (shareable) can only be added on top of a 5691 (shareable) configuration.
- A 5795 product (enterprise) can only be added on top of a 5795 (enterprise) configuration.

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- A customer may have 5691 type entry configuration and 5795 type entry configuration (such as 5691 — MDC + HDS and 5795 — DNC).

Controlled Availability of Selected CATIA V5.10 Products

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

- CATIA — DMU Space Engineering Assistant 2 Product (5691-SPE)
- CATIA — DMU Space Engineering Assistant 2 Add-on Product (5693-SPE)

The purpose of the controlled availability designation is to ensure your awareness of the introductory nature of these products and of the need to verify the readiness of support resources and product function to maximize successful introduction.

Additionally, the following new products are subject to controlled availability in this release:

- CATIA — Business Process Knowledge Template 2 Product (5691-BK2)
- CATIA — Business Process Knowledge Template 2 Add-on Product (5693-BK2)
- CATIA — Knowledge Definition 3 Configuration (5691-KD3)

It is strongly recommended that these products be purchased in conjunction with a dedicated services offering to facilitate proper education and to ensure optimal capture of customer best practices.

Customers interested in purchasing one of the above products should discuss their requirements with their IBM representative or Authorized IBM Business Partner.

Effective November 29, 2002, the following products are generally available and are no longer subject to controlled availability.

- CATIA — Weld Design 1 Product (5691-WD1)
- CATIA — Weld Design 1 AOP (5693-WD1)
- CATIA — Aerospace Sheetmetal Design 3 Configuration (5691-SL3)

CATIA P1

CATIA V5.10 delivers a new configuration, six new products, and enhancements to previously announced P1 configurations and products. Two products are withdrawn.

New Configuration

Mechanical Design Solutions

CATIA — Styled Mechanical Design (5691-YM1)

Perfectly suited to the consumer goods domain, the YM1 configuration is a 3D product creation package that includes comprehensive part and assembly design features, along with associative drawing extraction capabilities and advanced surface creation tools. Designers will also find all the 2D drafting features needed for efficient drawing production. This configuration is greatly empowered with a comprehensive package of the core shape and styling products needed to meet the industrial needs of the consumer goods

domain. In addition, import/export of IGES and STEP data make collaborative design within the extended enterprise easy. Users managing parts coming from external applications will appreciate the Healing Assistant 1 product. Healing Assistant 1 allows users to check the validity of imported geometry against V5 modeling criteria and to improve the topology and the geometry of the analyzed objects.

The configuration includes the following products:

- CATIA — CATIA Generative Shape Design 1 (GS1)
- CATIA — Freestyle Shaper 1 (FS1)
- CATIA — Healing Assistant 1 (HA1)
- Real Time Rendering 1 (RT1)
- CATIA — Assembly Design 1 (AS1)
- CATIA — Part Design 1 (PD1)
- CATIA — Interactive Drafting 1 (ID1)
- CATIA — Generative Drafting 1 (GD1)
- CATIA — STEP Core Interface 1 (ST1)
- CATIA — IGES Interface 1 (IG1)
- CATIA — CADAM Interface 1 (CC1)
- CATIA — V4 Integration 1 (V41)
- CATIA — Object Manager 1 (CO1)

Updated Configurations (in this release)

- Wireframe & Surface 1 (WS1) is added to the Mechanical Design 1 (MD1) and Mechanical Engineering 1 (ME1) configurations.
- Weld Design 1 (WD1) is added to the Extended Mechanical Engineering 1 (XM1) configuration.
- Wireframe & Surface 1 (WS1) is removed from the Extended Mechanical Engineering 1 (XM1) configuration and is replaced with the new Generative Shape Design 1 (GS1) product.

Withdrawn Products: Effective October 29, 2002, the following P1 products are withdrawn:

- The CATIA — Team PDM 1 shareable product (5691-TD1) and add-on product (5693-TD1). The affected indicator feature numbers are withdrawn as listed in the **Ordering Information** section. Migration paths to the SMARTEAM CATIA Team PDM Configuration (5691-TDM) are provided in Software Announcement 202-301, dated October 29, 2002.

Note: Although CATIA P1 solutions are available on Windows® 98 through R10, the SMARTEAM CATIA Team PDM Configuration (5691-TDM) is not available on the Windows 98 platform.

- MultiCAX P Plug-in Product (5691-DPL). Migration paths to the MultiCAX PD Plug-in (5691-PDL) replacement product can be found in Software Announcement 202-297, dated October 29, 2002.

New Products

Mechanical Design Solutions

CATIA — Part Design Features Recognition 1 (5691-FR1, 5693-FR1)

Finds and recovers elementary part design features from an isolated B-representation. The product “recognizes” part design features in an imported B-representation and allows the user to retrieve a comprehensive CATIA data structure (product structure) from the representation.

FR1 facilitates downstream application integration to improve part simplification for tool definition and flexible local design. The product:

- Offers advanced tools for post-modification on isolated solids imported from external systems or as standard formats.
- Recreates part design features using a global recognition tool that analyzes the whole isolated solid.
- Recreates part design features based on face selection for precise and efficient local recognition.

CATIA — Generative Shape Design 1 (5691-GS1, 5693-GS1)

Helps design mechanical shapes that are based on a combination of wireframe and multiple surface features. GS1 includes all the functions and commands from the CATIA — Wireframe & Surface 2 (WSF) product, which is withdrawn in this release. It therefore provides an extensive set of tools for creating and modifying mechanical surfaces used in the design of complex shapes and hybrid parts. GS1 uses smart tools, such as power copy, to manage feature reuse. Its feature-based approach offers a productive and intuitive design environment in which design methodologies and specifications can be easily captured and reused.

GS1 features:

- Ease of use for all designers
- Coverage from preliminary to detailed design for parts that require surfacing in downstream processes
- Associative design
- Time-saving change management features
- End-to-end process coverage

Ease of Use: What makes GS1 unique in the market is that it can be used by either non-surface specialists and designers, thanks to the product's ease of use, or by advanced shape designers looking for a complete surfacing tool, owing to its a wide range of capabilities. Specification capture is completely transparent as if the user were designing explicit shapes.

From Preliminary to Detailed Design: CATIA — Generative Shape Design 1 provides a comprehensive set of features for shape design. These include many wireframe elements:

- Point
- Line
- Angle
- Plane
- Curve
- Circle (bi-tangent, through, and trimmed)
- Spline
- Parallel curves
- Corner-on-plane
- Connect 3D
- Spiral sphere
- Intersection
- Projection

Standard and advanced surface features include extrude, revolute, sweep, (including segment and circle), and fill. Standard combinations of elements use associative transformation, including symmetry, scaling, translation, affinity, extrapolation, and fillet.

Associative Design: Design in context allows concurrent work with user control of associativity. Wireframes and surfaces can be designed using the part or assembly context. When design changes are made, the user controls the propagation of modifications. The designer can reuse an existing surface and link-in additional parts to support concurrent engineering.

Managing Changes with Timesaving Features: Several GS1 features help to efficiently manage design modifications. For example, a datum curve or skin used in one feature can be replaced and a set of features can be isolated as a single feature (with no history) to facilitate design comprehension and accelerate design changes.

End-to-End Full Process Coverage: GS1 provides the process-centric solution for managing the complex skills that typify the design-to-manufacturing environment. The design and manufacture of plastic parts or shells is a good example. After importing surfaces, the designer can check and heal them with the CATIA — Healing Assistant 1 (HA1) product and then modify and add other surfaces using the powerful wireframe and surface creation tools of GS1. To finish, the design parts will be manufactured after surface machining programming with the CATIA 3-Axis Machining 2 (SMG) product.

Electrical and Systems Engineering Solutions

CATIA — Electrical Library 1 (5693-EL1)

Allows users to extend mechanical parts and assemblies with electrical behaviors to define electrical devices, such as connectors and equipment. It provides a consistent set of tools dedicated to heavy machinery electromechanical designers. The catalog used is common between the mechanical and electrical domains. Key product functions include:

- Creates and manages an extended range of electrical components, wires, and harness supports
- Manages compatibility rules between electrical devices
- Defines mechanical constraints according to the assembly of electrical devices
- Supports customization of electrical catalogs

Extended Range of Electrical Devices: Users can manage electrical equipment assemblies and electrical connectors. Definition of electrical devices includes features, in addition to electrical attributes.

Integration of Mechanical and Electrical Specifications: EL1 allows users to predefine how electrical devices will be assembled when they are connected, such as mating connector assemblies. The user can define a connection point to link a bundle segment, another connector, a contact, a filler plug, a bundle segment, or a cavity.

Definition of Harness Supports: To facilitate 3D harness installation, users can specify in the catalog how the support part will constrain the harness.

Editing Electrical Properties and Attributes: Properties of electrical objects can be edited using the CATIA V5 properties command.

Customization of Electrical Catalogs: Electrical catalogs are flexible to allow each user to define their own catalog structure. This flexibility is provided through a user-defined map that correlates electrical object types (wires) with catalog chapters.

Note: This product is available as an add-on product (AOP) only.

CATIA — Electrical Harness Installation 1 (5693-EH1)

Designs physical harnesses within the context of the 3D mock-up. Natively, the mechanical context is used as an input for electrical design and then to obtain complete integration. Easy modifications and associations between the 3D environment and the harness are available. Owing to the integration with mechanical assemblies, electrical harnesses can be connected either to mechanical parts or electrical devices.

EH1 delivers a consistent set of tools dedicated to heavy machinery electromechanical designers. Key functions include:

- Creates round bundle segments within the 3D mock-up
- Supports associative constraints
- Defines electrical connectivity between bundles, segments, and electrical devices
- Analyzes electrical links between objects

Bundle Segment Creation in the Digital Mock-up: Users can create bundle segments directly within the mechanical context. Bundle segments can be constrained by:

- Points
- Following the external body of a part
- Supports defined with CATIA — Electrical Librarian 2 (ELB)
- An existing curve (spline or helix)
- Electrical components (supports, connectors, and so forth)

Definition of Round Bundle Segment Properties: Properties can be defined for:

- Diameter
- Length
- Minimum bend radius

Integration: EH1 is completely integrated with Electrical Library 1 (EL1).

Infrastructure Solutions

ENOVIA Work Package Exchange (5693-EW1)

CATIA — ENOVIA Work Package Exchange 2 (EWE) supports the ability to export or import CATIA V5 product data and associated CATIA V5 links from an OEM site to a subcontractor site that is disconnected from the OEM ENOVIA database. CATIA — ENOVIA Work Package Exchange 1 (EW1) is the P1 product used by a subcontractor to recover a CATIA V5 session from OEM mobile files and then make required changes.

Using the ENOVIA Work Package Exchange 2 product, the OEM defines a work context in ENOVIAVPM V1.5 or ENOVIA LCA V5 and loads all associated data into CATIA V5. Then the OEM generates “mobile” files and provides them to the supplier. Afterwards, the supplier, using ENOVIA Work Package Exchange 1, makes all the data modifications or creations requested by the OEM and returns the updated mobile files. Finally, the OEM loads the data provided by the supplier into CATIA V5 and imports them into ENOVIAVPM V1.5 or ENOVIA LCA V5 with ENOVIA Work Exchange Package 2.

CATIA — ENOVIA Work Package Exchange 1 (EW1) is used at the subcontractor site to retrieve information from the

OEM. Users can read mobile files and then change or create CATIA V5 objects and save them in mobile files.

The product requires ENOVIAVPM V1.5 or ENOVIA LCA V5 and CATIA — ENOVIA Work Package Exchange 2 (EWE) on the OEM side. The subcontractor must have at least a P1 license.

PPR PDM Gateway (5691-PX1, 5693-PX1)

Provides a comprehensive gateway interface between CATIA V5 and any PDM application for fluid import/export of data in a heterogeneous environment. The export/import infrastructures or “engines” comprise several components, each of which performs a specialized function on the exchanged data: extraction, validation, mapping, and reconciliation. Each component can be specifically tailored to address different exchange formats and to implement exchange rules. Users have the ability to manage and streamline all product-related information throughout the product life cycle, regardless of what PDM systems are being used, while preserving the integrity and reliability of data through synchronization.

Key product functions include:

- Access a currently selected object in an active document in the session
- Access the root object in an active document in the session
- Navigate document-to-document links
- Access document properties (Read/Write) to support title block associativity
- Build document image in a standard format (JPEG)
- Define location strategy priorities
- Define a new location strategy
- Define a new document environment

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

Mechanical Design Solutions

CATIA — Assembly Design 1 (5693-AS1)

- Performance Enhancement — Constraints can now be displayed and updated in visualization mode, providing a major performance improvement. In addition to reducing memory consumption, this technique improves productivity by allowing the user to resolve constraints in the visualization mode instead of having to switch to the design mode.
- KWE Integration — URLs can be linked to constraints.
- Mechanical Assembly.
 - This release delivers a number of contextual link management enhancements to improve design-in-context operations.
 - Contextual links can be created in a minimal context by using the closest common parent product to describe a reusable and self-contained sub-assembly.
 - The contextual parts user interface now displays more extensive context information to inform the user when a selection of an external reference is not allowed. It provides current context information and proposes a **Change**

Context command with the context adapted to the selection the user wanted to carry out.

- The **Change Context** command now allows the user to manage contextual links in many ways. It has therefore been renamed **Define Contextual Links**. This command is also available in the part design workbench to facilitate design process fluidity. The user can now directly change the context when working in the part design workbench. Previously, the user had to return to the assembly design workbench to use the command.

- **Assembly Features** — This release improves the creation of assembly holes. The user can now define a different type of hole for each affected part of the assembly feature. For example, the user can define different diameters and depths that are integrated in the preview panel, such as creating a screw hole in three parts, the first one adapted to the screw head and the third one threaded and blind.

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

- This release expands coverage of weld types and shapes by enabling the user to specify the preparation length (full length or contact length). This capability allows the user to create a design for welding several parts on the same edge of a given part. Optionally, preparation can be done on the contact zone of the welded part instead of the entire edge.
- Downstream application integration is improved with a weld report. The report captures relevant data about welded parts:
 - ID
 - Type
 - Volume
 - Weight
 - Comment

The user can sort the data. The information can be used for downstream operations and for reporting purposes.

CATIA —Part Design 1 (5693-PD1)

- **Sketcher Enhancements**
 - Constraints schema can be created automatically when doing offsets.
 - Sketch analysis is improved with the addition of a **Diagnosis** tab to quickly re-frame on the relevant element, sort elements by column, and reduce the size of the dialog box to improve the usability of the command.
 - The user can now add points to a spline. Previously, only existing points could be selected.
- **Feature Enhancements**
 - **RIB/SLOT** — This release adds a thin solid option (similar to PAD/POCKET/STIFFENER).
 - **PAD/POCKET** — Thin solids are improved with **Up to plane** and **Up to surface** options.
 - **SHAFT/GROOVE Reverse Direction** — The user can determine the direction of the revolution along the axis of the profile when creating a shaft.
 - **HOLE** — This release adds a new way to specify a countersunk hole. The hole can be specified with two new parameters:

- Depth and diameter
- Diameter and angle

These parameters improve the specification-driven approach to provide better compliance to mechanical design processes.

- **Relational Design**

- **Permanent Display of 3D Parameters** — 3D parameters for all part design and sketcher features are now permanently displayed. This enhancement facilitates understanding and visualization of the part from design step to design step.
- **Acquisition Filter Toolbar** — This toolbar allows the user to decide whether or not to create a link on a sub-element. It improves user productivity because the user does not have to explicitly declare the sub-element.
- **Button Switch to Display only current solid** — This switch provides quicker access through a toolbar to this option.

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

Note: Beginning with this release, Wireframe & Surface 1 is available as a shareable product (5691-WS1).

- **General Mechanisms**

- **Parametrization Analysis** — This new command allows the user to analyze the CATPart structure.
- **Multiselection in One Feature** — This generic mechanism can be used by appropriate WS1 commands to handle a new multiselection mode. The multiselection mode allows WS1 commands to create multiple features in a single operation from reference-type arguments and list-type arguments. For example, in multiselection mode, the **Translate** command takes as input two reference arguments (a direction and a distance) and a list argument (all the features to be translated). The result represents every set of objects created by an appropriate WS1 multiselection command as single feature that includes all translations. This approach drastically reduces the size of the history tree. Editing a multiselection feature allows users to redefine it and consequently to modify the set of features it is intended to encapsulate.

- **Ergonomics and Productivity** — This release adds acquisition filters to the toolbar. Users can filter their selection by geometrical types and features/sub-elements.

- **Topology/Geometry**

- **Translate Point/Point or Vector** — The translate command is improved to allow users to create a translate feature either defining the translation vector using two points or explicitly by specifying the translation vector coordinates (parameters).
- **SPLIT/TRIM Keep/Remove Element:** The keep/remove element function for WS1 split and trim features allows the user to define the elements to be kept or removed while performing the split or trim operation. This enhancement avoids problems with an element's relative orientation and tangent contacts.
- **Guide Re-limitation by Planes for Sweep Operations** — This function consists of reducing longitudinally the domain of the sweep. It is performed by

selecting “re-limiters” (planes or points) for splitting the spine. It is available for all types of swept surfaces: explicit, implicit linear, implicit circular, and implicit conical profiles. This capability is useful when primary input elements lead to a longer swept surface than expected or necessary. Consequently, it saves a split creation, thus reducing the model size and complexity.

- Auto-search for Join Operations — This new function allows the user to automatically find and select curves connected to a reference curve.
- Non-manifold Join — Join non-manifold topology configurations.

CATIA — Generative Drafting 1 (5693-GD1)

- Generation and Update Performance — This release adds an occlusion culling capability. In the computation of views, only the impacted components are taken into account and hidden assemblies are ignored. This new facility significantly reduces memory consumption, allowing the user to manage much larger assemblies.
- View Generation in CGR Format — The user can now choose between generation in the design mode (typical) or visualization mode.
- Default Pattern Behavior for Assemblies (Orientation Behavior) — The pattern behavior of an assembly will respect the usual standard for the instantiated part automatic generation when creating 2D views.
- Interrupt Access — Drawing updates can now be interrupted.
- Dimension Generation for a Part Instance in a Product — This release adds the capacity to generate dimensions from part references in the context of an assembly instance.
- Section View Creation for Existing Break-out Definitions — Section views can now be created for current break-out definitions. This capability is useful for creating mixed views and produces more readable and shareable drawings.
- Progress Bars — Progress bars are now included for every view creation.

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

- Performance Enhancements for Large Data Volumes — This release adds support for:
 - Multiselection of a large set of objects
 - Large number of elements in no-show mode
 - Ability to create geometry on a large model
 - Ability to change color of a large number of elements
- Standards
 - Graphic definitions for line thicknesses are now stored in a standards file instead of being a tool option.
 - New annotation parameters have been added to ID1 standard management capabilities.
- GDT Enhancements
 - Support is added for all updates that have been made in the different international standards, such as JIS, ISO, and ASME (a new ANSI standard).
 - All normalized symbols are now supported.

- The product’s new user interface supports multi-line editing.
- Associative Dimensioning — Tubing and structure associative dimensioning is now enabled only for feature dimensioning (not between two features).
- Tolerancing — ISOCOMB tolerance description is now supported.
- Associative Re-routing — This release adds a re-route command to create or recreate the associativity for isolated dimensions. Though primarily a V4 migration tool, this capability can also be used for non-associative dimensioning.
- Visual Basic Script (VB Script) for Text Editing in an Instantiated Component — Through **Tools/Macro/Macros**, a user can run a VB Script macro to modify texts in 2D component instances. This new function provides more openness and flexibility. Users can automate operations for more productivity.
- Raster Editor — The user can now edit raster images in a CATDrawing, including scaling capabilities.
- Balloon Enhancement — The user can now fix the size of a balloon and fit text inside the balloon.
- Roughness Enhancements — This release provides a standard parameter for customizing roughness with the leader representation.
- Dimension Enhancements
 - Radius curvature dimension on spline
 - Min/max dimension on spline/curve linked to horizontal/vertical direction
 - Dimension value positioning management (on top or above the dimension line)
 - Foreshortening re-limitation management (ability to manage the end point of the foreshortening)
- Knowledge Integration
 - Dimensions, GDTs, and annotations can be included in a design table to be activated or deactivated. This gives the user the ability to manage a global design table configuration in 2D or 3D with the same behavior.
 - Table structure is associative to the 3D design table.
- View Isolation — All annotations and dimensions are kept as isolated semantic dimensions. When a generative is isolated, dress-ups are preserved.
- Split of Table Definitions in the Same Sheet — While editing a dimension table (a hole dimension table or a coordinate dimension table), the user can split the table on many parts of a defined number of rows. This gives more flexibility and usability to the command.
- Leader Management — This release adds the ability to:
 - Move or rotate leaders with annotations.
 - Easily move and position leader breakpoints.
 - Add a leader with an associative orientation to be perpendicular to the reference.
- Welding Enhancements — This release improves standards support by adding some primary symbols.
 - V flare for ISO
 - Spot for JIS
 - Finish symbols

- Bi-directional Modification of Generated Dimensions — This release adds the ability to edit 3D constraints through the 2D-generated dimensions.
- Symbol Suppression — No symbol visualization for leaders, datum features, and datum targets is supported. It is possible to suppress leader extremity symbols for all annotations.
- DXF — This release adds a number of export functions:
 - Export option for grouping all complex elements into graphic blocks
 - Export of layers (through semantic export option)
 - Export of all texts as semantic texts (through semantic export option)
- New Commands — This release delivers a number of new commands:
 - Text without leader
 - Text parallel to the screen
 - Flagnote (without leader)
- Non-semantic Geometric Tolerance Command Editor — The editor has a new look and feel, improved usability, and new capabilities.
 - Add upper and lower text
 - Define multiple modifiers per box
 - Fill the command panel from a selected geometric annotation (instead of rewriting)

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

- Automatic Grouping and Ordering of Annotations Commands — These capabilities help the user to group annotations with only one leader for the entire group. Two commands are provided, where ordering and grouping of a set of annotations apply to the same tolerancing feature:
 - Global automatic positioning. The order and position of annotations is proposed according to the standard definition and industry best practices.
 - One-by-one assisted positioning and ordering. The user has the ability to choose the position of a 3D annotation according to the position of a reference annotation through a new dedicated window.
- FTA/FT1 Generative Dimensions — The objective is to provide a new command for generating FTA/FT1 dimensions from part design feature constraints, such as constraint dimensions expressed at the sketcher level or part design internal parameters (pad length, hole diameter,...). The tolerance values of the FTA/FT1 dimensions and the part design constraints are associative, allowing the use of the mean dimensions command on FT1/FTA dimensions. The edit of the constraint is also proposed. This release addresses hole features only.
- Free/Perpendicular Leader Switch Command — A contextual command on a leader allows the user to switch among three modes:
 - Associative free
 - Associative perpendicular
 - Not associative

These modes give more flexibility and usability to the user.

- Feature Tree Sub-nodes — Sub-nodes enhance tree readability for:
 - Captures
 - Views
 - Datums and datum targets
 - Datum reference frames
 - Geometric tolerances
 - Dimensions
 - Roughness
 - Welds
 - Texts, NOAs, flagnotes
- Capture Management Command — New capabilities give the user the ability to query and manage how annotations are captured. For each annotation, the user can decide whether or not it is to appear in a particular capture.

CATIA — Mold Tooling Design 1 (5693-MT1)

- Mold Design Workbench
 - Enabling rules for positioning and dimensioning. A know-how rule can be attached to a component to modify or check its geometry. This allows the user to achieve perfect compliance with company standards or best practices.
 - Exact pre-visualization of a component when the catalog browser is selected (available also for assembly components). The user gets an interactive preview window in which the shape of the component can be directly visualized for dimensioning and positioning. The shape automatically adapts itself to the user's choices to provide efficient assistance.
 - Edit component to retrieve a list of original parameters. When editing a component, the user can retrieve the original parameters instead of a blank list. The user component can be instantiated faster by changing parameters from the original list.
- Ejection Component — This release adds a spring component.
- Standards Catalogs — This release updates the following catalogs:
 - HASCO
 - DME
 - DME-America
 - Rabourdin
 - Strack
 - Futaba
 - Misumi
 - EOC
 - National (USA)
 - Pedrotti (Italy)
 - PCS (USA)
 - Meusburger (Germany)

CATIA — Healing Assistant 1 (5693-HA1, 5693-HA1)

- Geometry Analysis Tools — This release introduces new tools for checking the validity of the imported geometry.
 - Check maximum G0 and G1 gap and keep the result of the analysis as a set of selected edges or faces to be used in local join or healing. The application's look-ahead capability improves user healing productivity.
 - Check curve connections on boundaries and keep them within the selected edges to make a local join.
 - Provide an icon to switch the **Display Surfaces' Boundary** option.

- Check curvature.
- Topology Improvement Tools — Also in this release are a number of tools for improving the topology of analyzed objects.
 - Full parameters are available (such as global join and healing) for local joins.
 - An automatic correction option in the face checker command resolves self-intersection situations on a boundary.
 - Geometry Improvement Tools — This release provides enhanced diagnosis for failures that occur when smoothing the edges of a boundary face. A more precise and detailed analysis of the failure is displayed to help the user perform the healing operation.

CATIA — Sheetmetal Design 1 (5693-SM1)

- Split by Surface — This release adds the ability to split a sheetmetal part by a surface. The resulting cutout will be normal to the sheetmetal part.
- Standards Compliance — Cutouts normal to sheetmetal parts will be compliant with sheetmetal industry standards and manufacturing cutout processes. (Typically, cutouts are performed by a laser, so they are always normal to the sheetmetal part.) An option in the cutout panel allows the user to specify that the extrusion is to be normal to the part.

Product Synthesis Solutions

CATIA — DMU Navigator 1 (5693-DN1)

- CATDMUUtility — Enhancements for improved administration usability and productivity:
 - Multi-process tessellation that takes full advantage of multi-CPU machines and that allows for better management of errors during the tessellation process (SGI and IRIX only)
 - Ability to upgrade CGR versions to transform older CGR files into newer versions that take full advantage of semantic enhancements and size reduction relative to previous releases
 - User exit for password encryption
- File and Document Management — Enhancements for better support of customer processes involving loading large amounts of data:
 - Multi-process loading, taking full advantage of multi-CPU machines and allowing better management of errors during the loading process (SGI and IRIX only)
 - Loading progress bar, enabling users to monitor the completion status of the loading process
- DMU Capabilities — Create geometry (point, line, plane and axis system) in the context of DMU to be reused for measurement purposes.
- 2D Review — Add a move command to define the exact position of 2D documents inserted in a window.
- Navigation — Assign order to DMU application data based on customer processes.
- Measurements:
 - Exact measurement on skin elements
 - Measure the distance between arc centers defined by three points

- Sectioning — DMU Space Analysis 1 sectioning capabilities are now available in DMU Navigator 1.

Analysis Solutions

CATIA — Generative Part Structural Analysis 1 (5693-GP1)

New Preprocessing Function — This release adds a geometry sub-split (sewing) function for preprocessing specification definitions. It allows users to more precisely define the connection, restraint, and load specifications for increased accuracy of analysis results.

Enhanced Functions

- Preprocessing Process
 - Model Check Panel — Introduced in V5.9, the panel now shows the connection validity status and, if needed, can guide the user through the required connections.
 - Beam Sections — The predefined beam sections that became available in V5.9 are completed with new types of sections:
 - I
 - U
 - T
 - X
 - Box

These additional sections provide coverage for a wider field of applications.

- Post-processing Process — The creation of post-processing images used for deformation or vibration visualizations can now be done either interactively through journaling or by using VB Script macros. Either technique enables users to create complex images and to store and reuse them easily.

NC Manufacturing Solutions

CATIA — Lathe Machining 1 (5693-LG1)

Dwell for Roughing Cycles — This release adds the ability to define a decimal value for roughing cycle revolutions. Until now, the dwell time in a revolution could be defined only in terms of an integer value (one revolution, two revolutions ..). This release provides the ability to define dwell times as decimal values. For voluminous parts, where a considerable amount of material can be removed in a single revolution, this capability is useful for defining a dwell time of less than one revolution.

CATIA — Prismatic Machining 1 (5691-PG1)

- From Rough to Finish
 - This release extends High Speed Machining (HSM) capabilities for optimized and more productive machining with such functions as toolpath cornerization for contouring, feedrate reduction in corners for pocketing and contouring, and a slowdown rate option for pocketing.
 - The release also introduces additional flexibility for easier and quicker part programming with such enhancements as relative motion (godelta) in point-to-point operations, various bottom finish passes for contouring operations, the ability to handle multiple threading depths in one single threading operation, support of additional plunge options for circular milling and thread milling operations, and new capabilities in machining

pattern management to easily omit or retain holes within a pattern.

- Other enhancements include improvements to the Copy Order function for productivity gains, such as the ability to add/remove a toolpath, and extended support of transformations defined in the design environment.
- Productivity Improvements for a Streamlined Process — This release delivers additional productivity improvements to streamline customer processes.
 - Related to the manufacturing environment, such as the creation of tool catalogs from the resource list, the ability to store and retrieve macros from catalogs, definition of path names, and definition of post-processor integration
 - Related to business processes, such as batch queue management enhancements, with the ability to process single machining operations or to generate various outputs (APT, ISO, CLFILE)
 - Related to “day-to-day” tasks with such enhancements as the ability to manage Show/No-Show on activities or toolpaths and a new replay mode by NC command

Infrastructure Solutions

CATIA —IGES Interface (5691-IG1, 5693-IG1)

- Export — This release provides support for the export of faces in B-Spline mode. Faces from a part file are exported with 3D and 2D loops. The export process ensures that 2d curves are correctly oriented. This enhancement facilitates exchange with IGES processors that do not support interpretation of the 3D representation of surface boundaries.
- Import — This release adds phase 1 support for type 186 STEP entities (subordinated faces).

CATIA —STEP Core Interface 1 (5691-ST1, 5693-ST1)

- Export
 - Export of Selection Sets as Groups — During the export of a CATIA V5 file through the ST1 interface, the Selection Sets information is preserved through STEP's Group element. A particular entity can be included in more than one selection set (AP214).
 - Selective Export of NoShow Entities — This new function allows the user to specify in the STEP settings whether or not to export NoShow entities.
- Import — Insert a STEP file through the Assembly Design 2 (ASD) (among others) **Insert Existing Component** command. The STEP file is translated as a Document (CATPart or CATProduct), and this document is inserted.
- Transfer Reliability — This release completes the reliability improvements to the export and import processes with enhancements to geometric validation properties. To have a better assessment of transfer quality beyond the current OK/KO transfer information, specific properties for solids, products, and instances of products are included in the exported file. During import, the user knows if the data is reliable based on these properties. (AP214)

CATIA —Object Manager 1 (CO1)

- Interactive — This release adds support for:
 - Edit Link Replace capability on one document
 - Print

- Integration in batch monitor
- Log printings
- Light viewer
- Print
- Support for CGM, BMP, TIFF, SVG
- In the **Tools** options, add **Reset Background Color**
- Window multi-viewer with synchronization
- Graphic properties enhancements for points
- Nu, Nv settings for isopars
- Viewpoint update from local axis
- Snap on main directions in local transformations
- Ability to zoom plane and axis system
- Architecture
 - Dynamic licensing
 - 32-bit support

CATIA —V4 Integration 1 (5693-V41, 5693-V41)

- V4 Data Browsing — This release enhances the product's search and selection capabilities.
 - Details can be reordered in alphanumeric order.
 - A new keyword-based search dialog can be used to navigate through details.
- V4/V5 Data Interoperability Infrastructure
 - NT “forbidden” characters, such as the UNIX® asterisk (*), are converted by batch conversion when opening a model or session or on the fly for sequential files.
 - Support of DLNAMEs when opening a session is added.
 - The batch process used to migrate libraries is improved to reduce the risk of memory overflows.
- Interactive Drafting — This release adds:
 - ISOCOM tolerance
 - Radius curvature dimension
 - Layer filter on ditto — create a link to the number of configurations on the detail/ditto structure
 - Detail workspace navigation and selection in the CATIA V4 browser
 - Area fill blanking attribute migration
 - Batch report for as-result mode

CATIA P2

CATIA V5.10 introduces two new configurations, four new products, and enhances previously announced P2 configurations and products. Two products are withdrawn. Enhancements described for the P1 implementation of a P2 product also apply to the P2 product.

New P2 Configurations

Mechanical Design Solutions

CATIA —Styled Mechanical Design 2 (5691-YM2)

Perfectly suited to the consumer goods domain, the YM2 configuration is a 3D product creation package that includes comprehensive part and assembly design features, as well as associative drawing extraction capabilities and advanced surface creation tools. Designers will also find all the 2D drafting features needed for efficient drawing production. This configuration is greatly empowered with a comprehensive package of the core shape and styling products needed

to meet the industrial needs of the consumer goods domain. In addition, import/export of IGES and STEP data, as well as the ability to work in hybrid V4/V5 environment, make collaborative design within the extended enterprise easy.

Knowledge-based capabilities provided through Knowledge Expert 1 allows designers to import and use corporate knowledge stored in rule databases to ensure consistency and quality in their designs. Users managing parts imported from external applications, such as mold makers, will appreciate the Healing Assistant 1 product. Healing Assistant 1 allows users to check the validity of imported geometry against V5 modeling criteria and to improve the topology and the geometry of the analyzed objects.

The configuration includes the following products:

- CATIA — CATIA Generative Shape Design 1 (GS1)
- CATIA — Freestyle Shaper 1 (FS1)
- CATIA — Healing Assistant 1 (HA1)
- Real Time Rendering 1 (RT1)
- CATIA — Assembly Design 2 (ASD)
- CATIA — Part Design 2 (PDG)
- CATIA — Interactive Drafting 1 (ID1)
- CATIA — Generative Drafting 2 (GDR)
- CATIA — Knowledge Expert 1 (KE1)
- CATIA — STEP Core Interface 1 (ST1)
- CATIA — IGES Interface 1 (IG1)
- CATIA — CADAM Interface 1 (CC1)
- CATIA — V4 Integration 2 (V4I)
- CATIA — Object Manager 2 (COM)

Electrical and System Engineering Solutions

Electrical Wire Harness Installation 2 (5691-EI2)

This configuration provides in one seat all the necessary tools for creating and managing electrical wires and harnesses. This comprehensive package of 3D advanced products allows the user to design physical harnesses within the context of the 3D mock-up and, at the same time, manage the definition of electrical wires within the digital mock-up according to the functional definition of electrical signals. The Electrical Library provides the user with mechanical parts and assemblies with electrical behaviors to define electrical devices and to ensure a tight relationship between 3D physical implementation and its logical electrical representation.

This configuration also delivers a set of core CATIA tools that allow the user to perform advanced 3D design of parts and assemblies in the context of the full scale digital mock-up and to generate production drawings. In addition, it includes integrated real-time rendering capabilities. Knowledge basic capabilities provided through Knowledge Expert 1 allow designers to import and use corporate knowledge stored in rule databases to ensure consistency and quality in their designs.

As a CATIA P2 configuration, this solution offers integration tools for compatibility with CATIA V4. It includes data interfaces to most frequently used industry standards, including IGES, STEP, and CADAM drawings for smooth and fluid collaboration.

The configuration includes the following products:

- CATIA — Electrical Wire Routing 2 (EWR)
- CATIA — Electrical Library 2 (ELB)
- CATIA — Electrical Harness Installation 2 (EHI)
- CATIA — Generative Shape Design 1 (GS1)
- Real Time Rendering 1 (RT1)
- CATIA — Assembly Design 2 (ASD)
- CATIA — Part Design 2 (PDG)

- CATIA — Interactive Drafting 1 (ID1)
- CATIA — Generative Drafting 2 (GDR)
- CATIA — Knowledge Expert 1 (KE1)
- CATIA — IGES Interface 1 (IG1)
- CATIA — CADAM Interface 1 (CC1)
- CATIA — V4 Integration 2 (V4I)
- CATIA — Object Manager 2 (COM)

Updated Configurations: Wireframe & Surface 2 (WSF), which is withdrawn in this release, is replaced by the new Generative Shape Design 1 (GS1) product in the following configurations:

- Mechanical Design 2 (MD2)
- Mechanical Engineering 2 (ME2)
- Extended Mechanical Design 2 (XM2)
- Core & Cavity Design 2 (CV2)
- Reverse Engineering 2 (RE2)
- Mold & Die Machinist 2 (MO2)
- Preparation Machinist 2 (RM2)

In the Prismatic Machinist 2 (PM2) configuration, it is replaced by Wireframe & Surface 1 (WS1).

Weld Design 1 (WD1) is added to the Extended Mechanical Design 2 (XM2) configuration.

Withdrawn Products: Effective October 29, 2002, the following P2 products are withdrawn:

- The CATIA — Wireframe & Surface 2 shareable product (5691-WSF) and add-on product (5693-WSF). The affected indicator feature numbers are being withdrawn as indicated in the **Ordering Information** section. The withdrawn products are replaced by the new Generative Shape Design 1 (GS1) product in the following configurations:

- Mechanical Design 2 (MD2)
- Mechanical Engineering 2 (ME2)
- Extended Mechanical Design 2 (XM2)
- Core & Cavity Design 2 (CV2)
- Reverse Engineering 2 (RE2)
- Mold & Die Machinist 2 (MO2)
- Preparation Machinist 2 (RM2)
- Aerospace Sheetmetal Design 3 (SL3)

In the Prismatic Machinist 2 (PM2) configuration, they are replaced by Wireframe & Surface 1.

Paths for migrating to CATIA — Generative Shape Design 1 (GS1) are available with this release.

- The CATIA — Electrical Cable Definition 2 shareable product (5691-ECD) and add-on product (5693-ECD). The affected indicator feature numbers are being withdrawn as indicated in the **Ordering Information** section. Migration paths to CATIA — Electrical Connectivity Diagrams 2 (5691-ELD, 5693-ELD) are provided in this release.

New Products

Product Synthesis Solutions

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

Allows companies to capture their business processes and to share and deploy them through dedicated workbenches. In a simple and intuitive way, without coding, a user with expert process knowledge can model any company's best practices and know-how, capture and automate design and engineering tasks, and embed core knowledge rules and checks within those tasks. As a

result, the expert can deliver the captured best practices through a specific pseudo-application that includes all the required features and process knowledge with a dedicated user interface.

The CATIA — Business Process Knowledge Template solution takes advantage of the wide range of products in the CATIA V5 portfolio and leverages their capabilities in a powerful, process-driven way. End users have access to these high-level pseudo-applications in the same way as they have access to any CATIA V5 product. Users can create products while following a predefined methodology derived from the enterprise's best practices as captured by the experts. Guided by skilled rules and checks, freed from time-consuming, repetitive design tasks, and benefiting from knowledge-driven task automation, users can unleash their creativity and focus on innovation.

The Business Process Knowledge Template solution is divided into a buildtime product, CATIA — Business Process Knowledge Template 3 (BKT), and a runtime solution, CATIA — Business Process Knowledge Template 2 (BK2). The CATIA — Business Process Knowledge Definition 3 (KD3) configuration delivers the combined power of all of the CATIA V5 knowledge solutions in one seat.

Deploy your Know-how through Specialized, Easy-to-Use Applications: The pseudo-applications created using CATIA — Business Process Knowledge Template 3 (BKT) are deployed and accessible through the CATIA — Business Process Knowledge Template 2 (BK2) runtime solution. Thus, all the experts' knowledge and practices are available within the V5 framework. These pseudo-applications benefit from CATIA V5's world-class look and feel and architecture: they are easy to use, intuitive, and 100% integrated with the entire CATIA V5 product range. End users can have access to one or several pseudo-applications according to their skill or domain expertise. The embedded knowledge-driven processes guide and assist them to design with confidence while learning, understanding, and following the best practices being used.

Secure your Design Process: End users can take advantage of the best company know-how by using these high-added value pseudo-applications. Driven by the task sequencing characteristic of the pseudo-applications, compliance of their designs with all company methods and constraints is assured and the quality and performance of the products created are increased.

Accelerate your Design Development Cycle and Unleash Designers' Creativity: The pseudo-applications created with CATIA — Business Process Knowledge Template 3 (BKT) contain design methods that secure the design process and therefore save time and money. Benefiting from knowledge-driven task automation, end users can explore more design alternatives or focus on innovation, letting V5 do the painful repetitive work. This approach leads to previously unattainable gains in productivity.

Note: These products are subject to controlled availability in this release.

Equipment and Systems Engineering Solutions

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

Enables users to flatten 3D electrical harnesses created with Electrical Harness Installation 2 (EHI) to produce 2D associative drawings for inspection and documentation purposes. EHF extracts 3D electrical data and provides a flattened view of the electrical harness. The user can manipulate the position of bundle segments to arrive at an optimal solution. EHF also allows the user to generate 3D flattened views of a harnesses. This capability enables the user to determine whether or not a bundle segment that is very rigid must keep its original bent shape. 3D mechanical constraints defined with Electrical Harness Installation 2 are preserved so that the bend radius and length of the bundle segment are still constrained in the 2D view.

Electrical Harness Flattening 2 is fully integrated with Electrical Harness Installation 2. Additionally, its natural integration with Interactive Drafting 1 allows the user to define and manage the relevant set of annotations and dimensions to create full reports for documentation.

Key product functions include:

- Provides automatic flattening of a 3D electrical harness
- Allows the user to manipulated bundle segments
- Provides filtering management to deal with configurations of a 3D harness to generate several relevant harness drawings
- Provides integration with Electrical Harness Installation 2 (EHI) and Interactive Drafting 1 (ID1) and the Generative Drafting products for documentation purposes

NC Manufacturing Solutions

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

Prismatic Machining Preparation Assistant 2 delivers functions to prepare a design part for prismatic machining through geometrical machining feature recognition. The product helps the NC programmer to make the link between design and manufacturing. It provides a set of functions that generate all prismatic geometrical machining features for the design part to be machined. Therefore, it builds a real manufacturing view of the design part with all the drilling and milling features to be machined. Thanks to its imbedded machining features recognition technology, MPA enables the user to create geometrical machining features for all kinds of CATIA design parts, even those with no design feature specifications.

CATIA — Prismatic Machining Preparation Assistant is totally integrated with the CATIA — Prismatic Machining (PMG) product. It dramatically reduces programming times by automatically building up all geometric areas that must be machined with prismatic machining operations, decreasing time spent in geometry selection. All geometrical machining features can be fully machined with all CATIA — Prismatic Machining strategies, from elementary machining operation creation through automatic complex machining process application in both the drilling and milling domains.

Key product functions include:

- Automatically create all prismatic geometrical machining features of a design part for milling and drilling machining
- Locally create a prismatic geometrical machining feature of a design part for milling and drilling machining
- Integrate geometrical machining features into the Prismatic Machining product
- Manage the recognition of axial features in the manufacturing view (presentation, editing, highlighting)

Automatic Creation of all Prismatic Geometrical Machining Features for Milling and Drilling: The user can automatically generate all prismatic geometrical machining features that have to be machined from a given design part. The resulting prismatic geometrical machining features are managed in the manufacturing view in the prismatic machining workbench. All CATIA standard holes and milling features, such as pockets, steps and cutouts, are supported and can be recognized. If required, the user can easily choose feature types to be recognized.

Local Creation of a Prismatic Geometrical Machining Feature for Milling and Drilling: The user can locally create a particular geometrical machining feature on a specific area of the design part. Thanks to local feature recognition techniques, the user only has to select faces on the skin of a design part, and the system creates the feature automatically. All CATIA standard holes and milling features, such as pockets, steps and cutouts, can again be individually created.

Integration of Geometrical Machining Features in the Prismatic Machining Product: All generated geometrical machining features are fully integrated in the Prismatic Machining product. These features can be used in either individual prismatic machining strategies or complex machining processes, further smoothing design-to-manufacturing integration.

Management of Recognized Axial Features in the Manufacturing View (presentation, editing, highlighting): All recognized geometrical machining features are presented in the manufacturing view of the design part to be machined by the CATIA Prismatic Machining product. Therefore, the manufacturing view presents the tree of machining reference features to be machined: the "TO DO" list. Geometrical machining feature characteristics can be edited, and references to the design part are shown in the 3D view.

Note: In this release, MPA is available for the supported Microsoft™ Windows platforms only.

Infrastructure Solutions

CATIA — ENOVIA Work Package Exchange 2 (5691-EWE, 5693-EWE)

Supports the ability to export or import CATIA V5 product data and associated CATIA V5 links from an OEM site to a subcontractor site that is disconnected from the OEM ENOVIA database. Using CATIA — ENOVIA Work Package Exchange 2 (EWE), the OEM defines a work context in ENOVIAVPM V1.5 or ENOVIA LCA V5 and loads all associated data into CATIA V5. The OEM then generates mobile files and provides them to the supplier. Afterwards, the supplier, using CATIA — ENOVIA Work Package Exchange 1 (EW1), makes the data modifications or creations requested by the OEM and returns the

updated mobile files. Finally, the OEM loads the data provided by the supplier into CATIA V5 and imports them in ENOVIAVPM V1.5 or ENOVIA LCA V5 with CATIA — ENOVIA Work Exchange Package 2 (EWE).

CATIA — ENOVIA Work Package Exchange 2 (EWE) is used by the OEM to produce the mobile files to be sent to the subcontractor and then save them in its ENOVIAVPM V1.4 or ENOVIA LCA V5 database.

The product requires ENOVIAVPM V1.5 or ENOVIA LCA V5 on the OEM side and CATIA — ENOVIA Work Package Exchange 1 (EW1) on the subcontractor side.

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)

The publication function now allows the user to replace links in the Define Contextual Links window through 3D selection or through specification tree selection.

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

- Filleting
 - Blend Vertex Fillet — This new fillet mode for molded parts and press and die parts allows the user to perform styling and to create complex mechanical fillets for plastic parts.
 - Multi-Limiting — The user can now create fillets that are limited by several surfaces.
 - Fillet Reliability — Fillet reliability is enhanced with the addition of a local twist solving capability that improves the geometrical quality of fillet extremities.
- Drafting Neutral or Reflected Lines — This release adds the ability to draft with limiting surfaces for plastic parts. The draft can be limited by several surfaces, producing significant productivity gains for the user.

CATIA — Wireframe and Surface 2 (5691-WSF, 5693-WSF)

This product is withdrawn in this release.

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

Enhancements for GDR are the same as for GD1.

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

- Fillet Dimension — The product now automatically recognizes any type of fillet in addition to previous types (cylindrical or torus). The user can now specify a radius tolerance for any kind of constant radius fillet surface.
- Automatic Support of COMMON ZONE (n SURFACES in ASME) and nX notes — The product automatically adds standardized notes for specification applied in a common zone or to a group of features (pattern).
- Tolerancing Feature Canonicity Respecification — All the semantic specifications that are controlled by FTA are based on the canonicity of the geometric features. When the geometric modeler is not able to provide the right canonicity for a given surface (very rarely), the user now has the ability to override this canonicity for tolerancing purpose only.

- Partial Surface Support — The standards define partial surface features that will be defined the following way:
 - Geometric feature definition are handled through GSD capabilities.
 - Partial surface creation command in FTA associates the partial surface geometric feature to the whole surface. The FTA semantic expert ensures partial surface integrity and warns the user in case of an inconsistency (partial surface is a sub-part included in the whole surface).
- V4 FD&T Data Migration — Copy/Paste of V4 tolerancing based on a product method is used to migrate V4 FD&T annotations into V5 FTA annotations. Only the annotations, the annotation planes, and the tolerancing features are migrated (not the geometry) and are applied to the same V4 geometry at the product level.

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

Enhancements to MTD are the same as for MT1.

CATIA — Core & Cavity Design 2 (5691-CCV, 5693-CCV)

- Full Design Capabilities for Core and Cavity Process — New standard design capabilities in the workbench make the product self-contained with respect to preparing a part for moldability and core and cavity design.
 - Add text with a leader
 - Add a flagnote
 - Add measurements
 - Add PowerCopy
- Draft Extract
 - Visualization of cast shadows.
 - Associativity in the definition of the main pulling direction.
 - Core and cavity and undercut areas are recomputed when the input of the main pulling direction is modified.
- Ergonomics — A new fly analysis option is available to enable a dynamic analysis of the draft value while flying on the part during the creation of the main pulling and slider/lifter directions. The option helps the core and cavity designer to obtain a quick definition of the right direction for the main pull and for sliders and lifters.

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

Enhancements for SMD are the same as for SM1.

Shape Design and Styling Solutions

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

- General Mechanisms — The user can now define the thickness and orientation for GSD features. Two thicknesses and the direction of the material on the surface can be defined. This enhancement improves integration for downstream processes, such as the new CATIA — Automotive Body in White Fastening 3 product.
- Topology/Geometry
 - Smoothing — The user can now freeze elements (vertices, edges) that are not to be deformed during a smoothing operation.

- Corner on Vertex — This new option allows the user to create a corner by specifying the internal vertex of a curve.
- Law Preview — This enhancement allows the user to pre-visualize the law evolution (for Sweep, Helix, and Parallel Curve) before applying it. It is available for all geometrical and knowledge laws.
- G1 Extrapolation of a Shell — Functional edges can be selected to take into account the internal edges of surfaces to extrapolate. The user controls the topology of the surface.
- Vertex Blend — Vertex fillets with several incident edges can now be created.
- Associative Global Untrim — The Untrim command is now associative. Associativity is very important in the IGES-to-shell shell process.

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

- Wrap Surface — This new function allows the user to deform surfaces with a law defined by a pair of definition surfaces.
- Wrap Curve on Multi-cell Curves — Surfaces can now be deformed based on the deformation of curve wrapping; that is, matching each reference curve onto a target curve. The deformation is then defined by the transformation of the reference curves into the target curves. It is now possible to have multi-cell curves for these reference curves and target curves.

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

- New Function: Continuity Constraints — The user can put a continuity constraint between two elements just by clicking an icon in the FreeStyle workbench and selecting the constraint later. These constraints, which are a key part of the user design intent capabilities of FSS, are now stored in the model. During any further design modification, the system will optimize the stored continuities. This concept is a free-form extension of part and sketcher constraints.
- Enhanced Functions
 - Feature Untrim — The Untrim command is now associative. Associativity is very important in the IGES-to-shell process.
 - Connect Checker — This release adds a color ramp (same as the existing one for distance analysis).
 - Match Curve — The original degree is kept.
 - Disassemble — Resulting elements can be created in a new open body.
 - Keep Continuity for Surface Converter Wizard — When performing a conversion on a surface, the user needs to keep continuities (tangency or curvature) on the surface. This capability is useful when converting several surfaces with common boundaries. Temporary Analysis (Name Replacement) — Analysis can now have the same life cycle as a command. The user can choose whether or not to keep a freestyle analysis when finishing a command. This is a time-saving enhancement.
 - Add Through-point Option for Surface Offset — Then user can select a point to specify the surface offset distance.
 - Curvature Porcupine Analysis — Create particular points as 3D GSD points (datum).

CATIA —FreeStyle Profiler 2 (5691-FSP, 5693-FSP)

Net Surface —The user can now create a net surface with no guide as input.

CATIA —FreeStyle Sketch Tracer 2 (5691-FSK, 5693-FSK)

The Sketch Tracer can now be used to create a painting using a CATIA screenshot. The user can refine a cylindrical or conical picture with any image editor using accurate information and use the updated painting to create the corresponding geometry. This function improves productivity by avoiding import definition and provides a full numerical workflow. Either an existing geometry and its engineering constraints (volume, dimensions, and so forth) can be taken into account, or a current, state-of-the-art styling study can be used to modify, improve, or add details.

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

- New Feature and Import Formats
 - Grouped or Distinct —When several files are given, this option is used to specify whether to group the files to create one global cloud (as in previous releases) or to create separate clouds for each file.
 - File IDs —The name of each distinct imported file appears in the specification tree.
 - New Formats
 - GOM-G3D
 - IGES —Type 116 entities
- Polygon Processing
 - 3D Tessellation with Control of Sag Value — This function allows the creation of less dense meshes while respecting the global shape within a given tolerance.
 - Increased Capacity — This release drastically increases the capacity to mesh huge clouds of points.
 - Polygon Smoothing
 - Deviation Control —The user can now limit the displacement of the vertices within a maximum distance.
 - New Option for filling Irregularities (Hollows) on a Surface —A follow-on to bump reduction, this option helps to control the volume of the model.
 - Curves and Scans — This release adds characteristic line protection against filtering and removal. It allows the user to protect cells or sub-cells, particularly for scans that correspond to the characteristic line in a cloud, against filtering and removal so that the points of a protected cell are kept.
 - Scan Creation on a Free Boundary — This new command can create scans or smoothed curves on a free boundary of a polygon.
 - Curve Projection —Curves can now be created from curve projections in two steps:
 - Create the scans with the **Curve Projection** command.
 - Optionally, smooth the resulting scans into curves to improve productivity.
 - The curves are projected along the compass direction or perpendicularly to the polygon.

- Planar Section — Option for output curve by smoothing.
- Curve from Scan — Point or tangency continuity constraint on an extremity. An existing entity can be used as a constraint (point, curve, or plane). The tangency direction is editable.
- Registration — This function is usable between CATIA points and clouds. It allows the user to specify a set of CATIA points as a pseudo-cloud for a target.

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

- Curve and Scan Creation
 - Scan Creation on a Free Boundary — This new command creates scans on the free boundary of a polygon.
 - Curve Projection —Curves can now be created from curve projections in two steps:
 - Create the scans with the **Curve Projection** command.
 - Optionally, smooth the resulting scans into curves to improve productivity.
 - The curves are projected along the compass direction or perpendicularly to the polygon.
- Curve from Scan —Continuity constraint on extremity. An existing entity can be used as a constraint (point, curve, or plane). Tangency direction is editable.
- Slice Curve — Curve re-limitation. The resultant polygon is the one relative to the curve section selected.
- Clean Contour — An option for global deformation instead of local deformation near curve extremities for better preservation of the initial shape.
- Adjust Node — This enhancement ensures point continuity or tangency continuity (to a plane) for a set of curves that arrive at a single corner. It improves corner continuity for power fit surfaces.
- Surface Creation —A new power fit option for NU, NV, degU, and degV allows the user to display and manage spans with defined directions.

Product Synthesis Solutions

CATIA —Knowledge Advisor 2 (5691-KWA, 5693-KWA)

- ENOVIAPM Integration — This release delivers the first step in knowledge integration with ENOVIAPM with the persistent storage of rules, checks, and formulas that reference part parameters at the product level for capitalization and re-use.
- Knowledge Relationships at the Product Level — In this release, the use of published parameters is extended to the knowledge relationships used at the product level. Such relationships include rules, checks, reactions, and sets of equations based on published parameters. Users can now parameterize a wider set of components based on published parameters, such as an assembly, and can ensure the relationships among parameters are preserved when a component is modified. For instance, at the conceptual stage of the design, an expert can define the global product architecture based on formulas, rules, and parameters. The publication of these parameters allows different users to enrich the product structure by plugging in their contribution (parts,

sub-assembly, and so forth). It is also an efficient and quick way to test different product configurations.

CATIA — Product Engineering Optimizer 2 (5691-PEO, 5693-PEO)

- New in this release is the extension of the Set of Equations capabilities (available in KWA) with the ability to solve constrained hybrid problems (mixing engineering and design). It allows design engineers to perform wider and more complete optimizations that take equations and geometries into account.
- In this release, the use of published parameters is extended to optimizations and design of experiments used at the product level. Users can now parameterize a wider set of components based on published parameters, such as an assembly, and can ensure the relationships among parameters are preserved when a component is modified. For instance, at the conceptual stage of the design, an expert can define the global product architecture based on formulas, rules, optimizations, design of experiments, and parameters. The publication of these parameters allows different users to enrich the product structure by plugging in their contribution (parts, sub-assembly, and so forth). It is also an efficient and quick way to test different product configurations.

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

- Data Flow — Data flow is enhanced for greater administrative usability and productivity.
 - A batch equivalent to the **Save as Frozen** function allows administrators to extract a product and associated data from a directory in batch mode.
 - A new builder utility allows the user to build products and associated cache from PSN.
- DMU — This release adds the ability for ENOVIAVPM users to create a symmetric part in ENOVIAVPM and to send it to DMU for positioning.

Equipment and Systems Engineering Solutions

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

- XML — Performance is improved for placing devices from XML.
- Smart Placement (Snap) — This release provides an easy, assisted way to define the orientation of a component.
- Device Part Number Change Control — To manage the integrity between external 2D electrical specifications and the 3D design, the system can track any part number inconsistencies and propose automatic reconciliation.

CATIA — Electrical Wire Routing 2 (5691-EWR, 5693-EWR)

- XML — This release improves performance for wire routing from XML.
- Filter Wires — Wires can be filtered based on an external configuration system. The system can retrieve a subset from a maximized harness based on criteria provided from an external legacy database or from business partners.

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

- V4 to V5 CATProduct Migration (Batch Method) — This release adds support for V4 to V5 bundle segment migration.
- Bundle Segment Split — The bundle segment split function is improved by allowing a branch on the main bundle segment without imposing a point location. A branch point is no longer imposed in space, but follows the segment curve. With this improvement, change management is facilitated because the whole harness will now follow the connector position.

CATIA — Electrical Cable Definition 2 (5691-ECD, 5693-ECD)

This product is withdrawn in this release.

CATIA — Tubing Design 2 (5691-TUB, 5693-TUB)

- Manufacturability Checking of Tubes — A dedicated command is provided to allow the user to do the following manufacturability checks on demand:
 - Minimum segment length
 - Invalid bend definition

The user can query the errors that are generated by the manufacturability check. The user will then be able to zoom in and find the exact area of the design where a proper adjustment is needed to ensure manufacturability.

- Enhanced Reporting
 - Bending data is now available (as dynamic attributes) through the standard reporting tool. This will provide the ability to associate reports that are inserted in a 2D drawing and linked to the 3D design. When the 3D design is modified, the user can simply update the 2D drawing to reflect the 3D changes.
 - The user can now insert a report on a 2D drawing to provide full associatively with the 3D design.

CATIA — Equipment Support Structures 2 (5691-ESS, 5693-ESS)

- Drafting Integration — Extract drawings from structural elements.
- Analysis Integration — Perform analysis on structural elements.
- Notching — This new assisted feature creates notches on structural elements.

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

This release enhances CATIA V5 to V5 model and catalog migration.

CATIA — Structure Preliminary Layout 2 (5691-SPL, 5693-SPL)

- Project Change from Root Product to any Top Product — The user can now incorporate multiple projects in the same product to deal with different purposes.
- Drafting Lines Drawings — The user can now extract waterlines, stations, and buttock lines in drafting.

- Multi-Molded Forms
 - This release delivers greatly improved speed.
 - Opening management in multicopy is improved so that multiple openings can be instantiated at the same time.

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

- PRM Integration — Remove settings defined in the PRM to avoid redundancy of information.
- Drawing Extraction for Functional Objects
 - Graphic Replacement — Simplified and standardized (customizable company standards) representation of elements (for example, a stiffener is symbolized by a single line with arrows)
 - Symbols
 - Annotations
 - Dimensions
- Knowledgeware
 - Sample Reports — Extract a bill of materials (BOM) from object attributes exposed in the knowledgeware package.
 - Sample Production Data — Create user attributes on stiffeners and plates.
 - Catalog of Design Rules — “Check” on opening location, for instance.
- Reporting — The reporter package (available in **Tools**) is integrated to do reporting through a dedicated application.

CATIA — Plant Layout 2 (5691-PLO, 5693-PLO)

An enhanced Advanced Local Axis shared command adds new define and set local axis capabilities. A local axis can be placed and set on a selected product and, at the same time, the compass can be lined up with the local axis. In this way, an object can be moved from one point to another without routing. The function includes a command option to either show or hide all attachments under a product. During the modification process, connectors can be used as manipulators.

Analysis Solutions

CATIA — Generative Part Structural Analysis 2 (5691-GPS, 5693-GPS)

In addition to the enhancements described for GP1, GPS now supports the persistent storage of analysis results and computation data in ENOVIAVPM for capitalization and re-use.

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

This release extends the product’s connections capabilities with a new connection type and enhancements to current connection types.

- This release introduces a fastened spring connection, a new type of face-to-face connection. This connection includes a spring in order to simplify complex modeling and to take into account know-how and experiments. The fastened spring connection introduces more flexibility in connection representation and is particularly relevant in the automotive domain for such tasks as cylinder head gasket modeling.

- Connection Enhancements
 - The Model Check window provides a better understanding of connections and a status on their validity and, where required, guides users through required corrections.
 - The creation of welding connections is greatly simplified as result of the ability to directly select the parts to connect instead of the geometry. In the case of arc welding only, users can directly select the entire wireframe geometry feature that represents the part junction so that they can work with the features quickly.

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

- Pre-processing
 - This release completes the data mapping function, which allows data to be imported from text or Excel files, with the ability to apply forces and moments to predefined points. This enhancement enables data to be imported from an external application for analysis in CATIA V5, fostering further collaboration between CATIA V5 and other customer applications.
 - The management of beam elements properties is enhanced in this release to allow precise modeling of a complex set of beams. For example, different kinds of beam sections, each with different associated values (length and diameter) can be applied to a geometry created in Generative Shape Design.
- Post-processing — This release adds the ability to create groups of geometry entities to make easier to visualize and understand the results.

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

- Assembly Mesh Compatibility Management — This release adds the ability to:
 - Implicitly capture the meshing of an external edge of a surface and make it compatible with the surface on which the edge rests, avoiding numerous iterations.
 - Automatically capture, during the meshing of a surface, the meshing of nearby different surfaces. This new function makes it easier to converge multiple surfaces.
 - Visualize mesh-free edges. This new functions allows the user to quickly check the status of mesh captures for an assembly.
 - Ensure the associativity between the captured mesh and a reference. This new function ensures the compatibility of the mesh with the reference when the reference is modified.
- Selections — Selections are simplified and more intuitive. The user can now:
 - Select a full edge by a path, avoiding repetitive selection of each edge segment.
 - Change edge constraint and face activity through a quick trap selection.
 - Make a multiselection by trap-on-mesh mapping.
 - Make a multiselection on mesh import files.
 - Analyze the mesh quality by MeshPart, completing the ability to perform a global analysis on an assembly to accurately pinpoint problems.

- Control interactively the pattern mesher (mapped, bead) for accurate specifications.
- Connections — The creation of welding connections is greatly simplified as result of the ability to directly select the parts to connect instead of the geometry and, in case of arc welding, to directly select the wireframe geometry feature that represents the part junction. Thus, users can employ these welding connections very quickly.

NC Manufacturing Solutions

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

This release adds collision detection between the tool assembly and the part during simulation.

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

This release delivers enhanced automation capabilities. The user can define machining features that can be shared by several machining operations and used directly within machining processes. Whether these machining features (such as pockets) are created manually or automatically by an automated feature recognition provided by the Prismatic Machining Preparation Assistant, company-specific machining processes that combine various machining operations can be applied to improve productivity. Additional enhancements, such as a new tool for hole selection and feedback on tool selection when applying machining processes, also deliver significant productivity gains.

CATIA — 3 Axis Surface Machining 2 (5691-SMG, 5693-SMG)

New Functions

- Semi-finishing, Finishing, and Finishing Rework — This release adds geometry parameters to pilot tool trajectory. This function reduces machining time.
 - Create NC points on the fly to determine approach and drilling points.
 - Use limit lines to determine machining limit zones.
- Roughing Operations — Stock can now be created by offsetting a part.
- Macros — Macros can be stored so that they can be read and reused.

Enhanced Functions — This release delivers a number of enhancements to the semi-finishing, finishing, and finishing rework processes.

- Rework Area Editor — This release adds the ability to create a split via a point. The cutting operation is done by filters (horizontal, vertical, and width). In some cases, the result of the cut operation is not valid, and it is useful to manually divide the area into parts. The user is now free to refuse the automatic option of the system and to choose a manual approach. This is a very important option for machining automotive parts, such as fenders.
- Machining Area and Rework Area — Additional parameters (tool, tool axis, tolerance) allow the user to initially fix parameters relative to slope and rework area operations. These new parameters speed up the tool path trajectory calculation, thereby reducing the machining definition time.

CATIA — Multi-axis Surface Machining 2 (5691-MMG, 5693-MMG)

- Curve Machining — This release delivers new multi-axis curve machining functions.
 - Automatic feedrate reduction in corners reduces machining time and makes high-speed machining possible.
 - An enhanced tangent axis strategy provides more flexibility for modeling and styling shop tasks in the automotive industry.
 - Along rules
 - Along drive surface iso-lines
 - Normal to drive curves
- Multi-Axis Fac-Iso Machining Operations — This release delivers a new tool axis strategy: 4 axis tilt. This strategy is important in machining turbo machinery parts.
- Macro Catalog — Also added in this release is the ability to store and retrieve macros from catalogs for multi-axis machining operations.

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

- From Rough to Finish
 - This release delivers better adaptation of roughing operations to aerospace machine parts. It adds the ability to define a different thickness on the floor and on the flank and pocketing improvements.
 - High Speed Machining (HSM) capabilities are extended for optimized and more productive machining with such functions as toolpath cornerization for contouring, feedrate reduction in corners for pocketing and contouring, and a slowdown rate option for pocketing.
 - The release also introduces additional flexibility for easier and quicker part programming with such enhancements as relative motion (godelta) in point-to-point operations, various bottom finish passes for contouring operations, the ability to handle multiple threading depths in one single threading operation, support of additional plunge options for circular milling and thread milling operations, and new capabilities in machining pattern management to easily omit or retain holes within a pattern.
 - Other related enhancements include improvements to the CopyOrder function for productivity gains, such as the ability to add/remove a toolpath, and extended support of transformations defined in the design environment.
- Enhanced Automation Capabilities — The user can now define machining features that can be shared by several machining operations and used directly within machining processes. Whether these features (such as pockets) are created manually or automatically by an automated feature recognition provided by the Prismatic Machining Preparation Assistant, company-specific machining processes that combine various machining operations can be applied to improve productivity. Additional enhancements, such as a new tool for hole selection and feedback on tool selection when applying machining processes, also deliver significant productivity gains.
- Productivity Improvements for a Streamlined Process — This release delivers additional productivity improvements to streamline customer processes.

- Related to the manufacturing environment, such as the creation of tool catalogs from the resource list, the ability to store and retrieve macros from catalogs, definition of path names, and definition of post-processor integration
- Related to business processes, such as batch queue management enhancements, with the ability to process single machining operations or to generate various outputs (APT, ISO, CLIFILE)
- Related to “day-to-day” tasks with such enhancements as the ability to manage Show/No-Show on activities or toolpaths and a new replay mode by NC command
- Multi-Axis Machining Operations
 - Multi-axis Fac-Iso Machining Operations — This is a new tool axis strategy (4 Axis Tilt) for turbo machinery parts.
 - Multi-axis Curve Machining — An enhanced tangent axis strategy provides more flexibility for handling complex multi-axis machining situations.
 - Along rules
 - Along drive surface iso-lines
 - Normal to drive curves

CATIA —STL Rapid Prototyping 2 (5691-STL, 5693-STL)

- Polygon Processing
 - 3D Tessellation with Control of Sag Value — Allows the creation of less dense meshes while respecting the global shape within a given tolerance.
 - Improved Mesh Capacity — This release drastically increases the capacity to mesh huge clouds of points.
 - Polygon Smoothing
 - Deviation Control — The user can now limit the displacement of the vertices within a maximum distance.
 - New Option for Filling Irregularities (Hollows) On a Surface — A follow-on to bump reduction, this option helps to control the volume of the model.
- New Import Format — When several files are given, a new group/distinct option can be used to specify whether to group the files to create one global cloud (as in previous releases) or to create separate clouds for each file.
- The name of each distinct imported file appears in the specification tree.

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

- Maximum Feed Rate — The user can now define a maximum feed rate for a machine. Errors will be reported during material removal simulation if the current machining feed rate exceeds the maximum value.
- Error Reporting — Error information generated during material removal simulation is now accessible in the NC programming context so it can be used for error correction.
- Machined Part Geometry — Accurate point information is now available. This information can be used to measure the center and radius of circular entities.
- Measurements — This release adds the ability to measure the center-to-center distance between two circular entities on the machined part.

- Collision List — The collision list can be filtered based on the type of collision.

Infrastructure Solutions

CATIA — Object Manager 2 (COM)

- Knowledgeware
 - Product level relations based on publication
 - VPM integration of relations at product level
- Interactive
 - Multiselection for the **New From** command. If a selected document has links to other documents, the user can extend the document list to apply the command to an entire product (CATProduct or CATPart) in a single operation without manually rerouting the links.
 - New drill selector ergonomics.

CATIA —V4 Integration 2 (V4I)

- CATIA V5 to CATIA V4 Data Interoperability — Increase start model definition and content. The initial V4 model is enriched with a broader coverage of V4 semantics (text and dimensions, 3D standards).
- Transfer of Infrastructure Information
 - Transfer of SPACE Entities as Result
 - Quality enhancements to V5 to V4 topological transfer.
 - Isolated solid mock-up migrated as exact solids through a setting. Previously, solid mock-ups were translated only in CGR.
- Transfer of SPACE Entities as Spec
 - Associative migration of Solid MML structures.
 - Sweep migration optimization.
 - Sewing migration on new sewing feature. Depending on the setting, support of the two new Part Design sewing feature intersection modes (IntersectSewing and NoIntersectSewing).

CATIA P3

V5.10 delivers two new configurations and two new products. The new products are available only in the new configurations.

New Configurations

Shape Design and Styling Solutions

CATIA — Automotive Body in White Design 3 (5691-AB3)

This configuration provides in one seat advanced 3D tools for designing body-in-white fasteners specifically for the automotive industry. In addition to a full portfolio of productive and innovative applications for design and drafting, the configuration contains applications for shape design and image rendering, plus a number of useful interfaces. It gives the user a productive end-to-end solution for mastering the complexities of body-in-white part design while benefiting from CATIA V5’s advanced knowledge and V4 integration products to produce a ready-to-manufacture mock-up.

The configuration's core product, CATIA — Automotive Body-in-White Fastening 3 (ABF), provides the intuitive tools for creating and managing spot-type fasteners. The user can switch, as needed, from a 3D point shape definition to a 3D hemispherical shape specification. In addition to placing fasteners, the application can issue reports to list such attributes as fastener location coordinates and the properties of the joined parts at each fastener location.

The framework of this solution is the CATIA V5 P3 platform. By integrating comprehensive product and process expertise, this P3 solution allows the user to maximize the productivity of specific automotive body-in-white fastening processes.

The configuration includes the following products:

- CATIA — Automotive Body-In-White Fastening 3 (ABF)
- CATIA — Generative Shape Design 2 (GSD)
- Real Time Rendering 1 (RT1)
- CATIA — Assembly Design 2 (ASD)
- CATIA — Part Design 2 (PDG)
- CATIA — Interactive Drafting 1 (ID1)
- CATIA — Generative Drafting 2 (GDR)
- CATIA — Knowledge Expert 2 (KE1)
- CATIA — IGES Interface 1 (IG1)
- CATIA — CADAM Interface 1 (CC1)
- CATIA — V4 Integration 2 (V4I)
- CATIA — Object Manager 3 (CO3)

Product Synthesis Solutions

CATIA — Business Process Knowledge Definition 3 (5691-KD3)

The CATIA — Business Knowledge Process Definition 3 (KD3) configuration accelerates a company's business processes while ensuring compliance with its best practices and taking advantage of its collective know-how. The KD3 configuration offers users access to advanced design parameterization and knowledge capture tools and allows them to define standard rules and checks for design quality assessment. It helps them to optimize the design process by making synergistic use of CATIA V5's suite of powerful knowledgware tools and by leveraging corporate knowledge with V5-empowered best practices workbenches.

"Capture, re-use, automate, explore, optimize, validate" is the unique philosophy that guides the KD3 configuration: Get it right the first time!

The configuration includes the following products:

- CATIA — Business Process Knowledge Template 3 (BKT)
- CATIA — Knowledge Advisor 2 (KWA)
- CATIA — Knowledge Expert 2 (KWE)
- CATIA — Product Engineering Optimizer 2 (PEO)
- CATIA — Product Knowledge Template 2 (PKT)
- CATIA — V4 Integration 2 (V4I)
- CATIA — Object Manager 3 (CO3)

Note: This configuration is subject to controlled availability in this release.

Updated Configurations

Shape Design and Styling Solutions

CATIA — Automotive Class A 3 (5691-AC3)

With this release, CATIA — Real Time Rendering 1 (RT1) is replaced by CATIA — Real Time Rendering 2 (RTR) in this configuration.

Shape Design and Styling Solutions

CATIA — Automotive Body in White Fastening 3 (ABF)

CATIA — Automotive Body in White Fastening 3 is a new generation CATIA product dedicated to the design of automotive body-in-white fasteners. It supports welding technologies and mechanical clinching, along with such other fastening techniques as adhesives, sealers, and mastics.

The user is provided with intuitive tools to create and manage spot-like fasteners. The user can switch, if needed, from a 3D point shape definition to a 3D hemispherical shape specification. In addition to placing the fasteners, reports can be issued from the application to list such attributes as:

- Fastener location coordinates and the joined parts properties at each fastener location
- Crossed thicknesses and flange materials
- Normal flanges
- Joined parts stacking order

The CATIA V5 generative feature infrastructure enables the associative update of fastener feature placement from a part design change (modification of the flange shape, flange thickness, or material property) or assembly structure change (Move joined parts, Replace joined part). Automotive Body in White Fastening 3 can be cooperatively used with other current CATIA V5 applications, such as Part Design and Assembly Design.

Key product functions are:

- Supports fastener parameter set-up file customization according to company standards
- Enables spot fastener placement and shape definition
- Supports welding, adhesive, sealant, and clinching process types
- Uses a setup file to make it easy to assign fastener attributes
- Associatively updates fastener locations from joined part design changes
- Enables collaborative engineering/manufacturing work
- Ensures access to customer proprietary tools via a neutral ASCII text exchange format
- Includes an intuitive, task-optimized user interface
- Provides native integration with other CATIA V5 applications

Tools Dedicated to Automotive Joining Technologies:

CATIA Automotive Body in White Fastening 3 is dedicated to the design of body-in-white fasteners for joining thin automotive sheetmetal parts. The application workbench commands are packaged into toolbars that reflect typical automotive joining technologies:

- Welding
- Adhesives
- Sealers and mastics
- Body-in-white mechanical fasteners

This process-based approach offers a highly productive and intuitive fastener design workflow. During the design phase, engineering and manufacturing fastener

parameters are taken into account, and several positioning methods are provided for spot-like fasteners. These include **On surface** and **From curve on support**. The created fasteners can be grouped to connect a set of functional contact zones specified on the parts to be joined. Whenever a fastener is inserted, the product can be updated.

Feature-Based Fastener Modeling: CATIA ABF provides body-in-white specific spot-like fasteners most commonly used in the automotive industry:

- Welds
- Points
- Adhesives drops
- Sealant thumbable
- Clinch point

The fasteners are created in a CATIA CATProduct document at the assembly level of the common node of the joined parts instances product path elements.

Associative Fastener Update from Part Design Changes: The specification of the fastener location can be generated on the fly when the user selects an external geometry reference issued from the joined parts (a support surface, a surface edge, a reference point). Whenever the location, shape, or properties (surface material or thickness) of the reference is changed, the fastener location is updated automatically along with the information (connected parts thicknesses and materials) provided. Hence, such an associative and integrated fastener design process delivers benefits not only for product development, but also for the industrial process.

Industry Standards Compliance: For each general joining technology, specific process types can be specified in the application parameters standard setup file. For each process type, the company administrator can narrow the list of CATIA fastener standard attributes to those that are relevant to the process type. For each relevant attribute, a set of company predefined values can be set up.

Intuitive User Interface: CATIA ABF offers the standard Windows interface. Fastener parameters can be set by using combo editors that provide the list of company standard predefined values. Fastener shape representation is a symbolic point or a symbolic wireframe depending on the shape definition of the fastener, which can be a point or a hemisphere. A specific graphic symbol can be assigned to each process type. The 3D visualization of this symbol makes it easy to differentiate among the different fastener process technologies.

Integration with other CATIA V5 Applications: CATIA Automotive Body in White Fastening 3 is natively integrated with other CATIA V5 applications.

Product Synthesis Solutions

CATIA — Business Process Knowledge Template 3 (BKT)

The CATIA — Business Process Knowledge Template (BKT) solution allows companies to capture their business processes and to share and deploy them through dedicated workbenches. In a simple and intuitive way, without coding, an expert in the subject process can model any company's best practices and know-how. From complex conceptual studies in the automotive domain to airframe structure generation, the expert captures and automates design and engineering tasks and then embeds core knowledge rules and checks within those tasks. The result is a specific pseudo-application that delivers the captured best practices, including all the required features and process knowledge.

To reach this objective, the CATIA — Business Process Knowledge Template solution takes advantage of the wide range of products in the CATIA V5 portfolio and leverages their capabilities in a powerful, process-driven way. End users have access to these high level pseudo-applications in the same way as they have access to any CATIA V5 product. Users can create products while following the predefined methodology derived from the enterprise's best practices as captured by the experts. Guided by skilled rules and checks, free from time-consuming, repetitive design tasks, and benefiting from knowledge-driven task automation, users can unleash their creativity and focus on innovation.

The Business Process Knowledge Template solution is divided into a buildtime product, CATIA — Business Process Knowledge Template 3 (BKT), and a runtime solution, CATIA — Business Process Knowledge Template 2 (BK2). CATIA — Business Process Knowledge Template 3 (BKT) is the foundation of the CATIA — Business Process Knowledge Definition 3 (KD3) configuration, which delivers in one seat all the power of the CATIA V5 knowledgeware solutions.

Captures Business Processes and Best Practices: Creation of a design assistant supposes that the company's processes and best practices have been formally identified beforehand. Using CATIA — Business Process Knowledge Template 3 (BKT), an expert can capture and capitalize on this knowledge, packing and combining the enterprise's processes and best practices within a pseudo-application. To ensure a global process coverage, these applications can be enriched by several experts, each representing a different discipline. This knowledge capture is made in the CATIA V5 environment through its intuitive interface and without coding. Thus, formerly complex implicit design processes and practices are made explicit, making them easily understandable, reusable, and shareable.

Automates Design and Engineering Tasks: From the processes and practices identified, the expert organizes the different design tasks and creates task sequences. All the CATIA VT product functions can be inserted and reused in a design sequence as a result of CATIA V5's powerful knowledge-based architecture. The created tasks and the entire sequence can be automated. Time-consuming, complex, or repetitive sequences, for example, can then be run automatically. Of course, if needed, users can also perform these sequences manually, task by task. For example, in the automotive field, crankshaft design successively uses repetitive geometry generation based on parameters, balance, and section computations.

The computations form an iterative loop on which the designer chooses the intervals to explore. The entire loop can be automated and optimized afterwards. The resulting added value is the dramatic design time reduction that knowledge-driven automation enables. Products are developed more quickly and in a surer way. The designers can test more design configurations in less time, giving the company a considerable boost in its competitiveness.

Generates Design Environments in a Simple and Intuitive Way: The creation of a pseudo-applications with CATIA — Business Process Knowledge Template 3 benefits from all the CATIA V5 functions. An expert can use, without coding, the different building blocks provided by the CATIA V5 portfolio of products and combine them together. The result of this building process is a dedicated pseudo-application workbench that is totally and seamlessly integrated into the CATIA V5 environment. Finally, the end user benefits from the

CATIA V5 products and generated pseudo-application synergy as a result of global knowledge integration and consistent look and feel.

Deploys Best Practices and Know-how in an Easy-to-Reuse Way: The pseudo-applications created with CATIA — Business Process Knowledge Template 3 benefit from the easy to use and intuitive CATIA V5 interface. They embody the deployment of the enterprise knowledge and know-how within the CATIA V5 framework and consistency with all other CATIA V5 portfolio applications. Complementing the other CATIA V5 products, these high added value “design assistants” can be defined and used according to the skill of each user. Thus, even inexperienced designers can use them to learn the company’s know-how and best practices.

Accelerates and Secures Design Process Time: Finally, the end user has access, in a CATIA V5 environment, to one or several specific pseudo-environments, each embodying specific best practices. Designers are driven and guided throughout the conceptual work, while being confident in design consistency and time gain thanks to knowledge-driven automation. This leads to better design quality, increased product innovation, and real strengthening of the company’s competitiveness.

Enhanced Products

Shape Design and Styling Solutions

CATIA — Automotive Class A 3 (ACA)

- Control Points Toolbar — This new toolbar improves user productivity by providing direct access to the most frequently used control point options, such as coordinate system, row and cross diffusion, and attenuation.
- Direct Break at Iso-lines — With this option, the user can trim a surface directly at a selected iso-line without creating a curve. The iso-line is produced by a 3D tool that displays both the line’s position and its direction. This enhancement is a significant productivity gain.

Analysis Solutions

CATIA — Tolerance Analysis of Deformable Assembly 3 (TAA)

- Assembly-level Image Editing — The user can now modify the image parameters of all products associated with an assembly process step at the same time, significantly improving designer productivity.
- 3D Tolerancing Support Enhancements — Productivity and reliability are improved by directly using tolerances defined in the 3D Functional Tolerancing & Annotation products.
- Datum Reference Frames — Several datum reference frames are now supported for the same tolerancing scheme.
- Process Activity Items — Datum reference frames, datums, and datum targets can be used as process activity items (to specify the positioning of the parts).
- Worst Case Analysis — This new type of analysis is now available to provide broader process coverage.
- Integration with PPRHub — This integration gives the TAA user the ability to read information from DELMIA (products, processes, and resources) and, reciprocally, to write TAA information to DELMIA databases.

Infrastructure Solutions

CATIA — Object Manager 3 (CO3)

In addition to the enhancements for CO2, this release provides interactive support for the MPK workbench and VR device button mapping.

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)

This release adds the ability to manage materials and material catalogs using VB Script to deliver such powerful capabilities as importing external catalogs and configuring the material of the mock-up.

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

New Functions

- Reality Center Configuration Workbench — A workbench specifically designed to allow users to easily configure any reality center display device is added. The workbench makes it easier to set up multi-screen displays using 3D visualization files instead of typing installation files.
- ClearCoat 360 Texture Generator (SGI IRIX Only) — The generator adds the ability to produce realistic reflective painted surfaces.
- Catalog Management — Materials and materials catalogs can be managed using VB Script, which offers powerful capabilities, such as external catalog import and mock-up material configuration.

Enhanced Functions

This release adds:

- Refined object lighting using ambient, diffuse, and specular coefficients on rendering lights
- Improved contextual menu with direct access to materials by clicking on the geometry

Note: In this release, Real Time Rendering 2 does not support dynamic licensing.

Photo Studio Optimizer 2 (5691-PSO, 5693-PSO)

- Depth of Field — This new visual effect makes objects that are very close or very far appear blurry, giving an additional sense of depth to the image.
- Lens Flare — This visual effect appears where the light intensity is very high. It provides a dazzling effect by generating a halo, flare, and radian lines.

DMU Space Analysis 2 (5691-SPA, 5693-SPA)

In addition to the enhancements to SP1, this release delivers improvements to DMU Space Analysis 2’s clash management capabilities:

- A new computation mode enables the user to group the computation of all interference detection information in one step.

- XML clash files can be imported directly using VB Script macros.

Product Synthesis Solutions

DMU Space Analysis 1 (5693-SP1)

- Sectioning — Perform automatic comparison of two sections in the context of the 3D compare command for thorough inspection and better understanding of differences between two versions of a part or assembly.
- Measurements
 - Perform exact measurement on skin elements.
 - Measure an arc center defined by three points.

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

- Ergonomy — This release adds the ability to do dynamic snaps based on smart targets. Positioning assistance with automatic snap while the user manipulates an object provides greater accuracy and ease of use.
- Ergonomics — In conjunction with Human Builder 2, the user can link a manikin to a part and automatically update the manikin's position when the part moves. This capability provides enhanced support of early serviceability and maintenance validation processes.
- Track Definition and Use — The user can now select multiple track positions (shots) in order to delete them all at once.
- Simulation
 - Color and Opacity Transition — This release adds the ability to manage smooth transitions from one color to another or from transparency to opacity. Smooth gradual color variation or gradual transparency/opacity variation is provided for special visual effects in marketing videos.
 - Multi-magnitude Player — The user can open the player to customize the simulation units to suit the application. A track can be simulated in seconds or in shots, depending on whether it is an animation or an analysis of passage points, respectively.
 - Sequence Delay — The user can delay the starting time of an action in a sequence to allow for track overlapping (that is, to allow for a track to start while another is still playing).
 - Keyboard Shortcuts — This release adds keyboard shortcuts for controlling the player in full-screen mode.
- Explode — Generate a sequence based on tracks from an exploded view for a better understanding of initial dismounting specifications and generation of marketing materials with special animations.

DMU Kinematics Simulator 2 (5691-KIN, 5693-KIN)

- Mechanism Modeling

ENOVIAVPM integration is exposed, allowing ENOVIAVPM users to store, retrieve and manage kinematics information from their VPDM system.

A CATProduct contained mechanism definition should be saved as *Expose publication*.

 - A CATProduct at an upper level, embedding another product containing mechanism, will expose a dress-up application data block in ENOVIAVPM:

1. The conceptual mechanism architect first saves the wireframe mechanism as *Expose publication* in ENOVIAVPM.
2. 3D designers create the detailed design for each part of the mechanism.
3. A digital mock-up integrator loads the wireframe plus the detailed design from ENOVIAVPM to create relevant dress-up relationships.
4. Upon saving, the product structure created is exposed in ENOVIAVPM, and the dress-up information is stored as an attribute of the integration level product.
 - Dress-up objects are visible, can be simulated, and are synchronized (no need to import).

- Mechanism Editing — This usability enhancement allows the user to check and eventually change the positive directions of commands.
- Mechanism Simulation — Collisions or distance results can be selected as simulation sensors. This allows the user to retain the history of the results, improving the simulation report (spreadsheet, X/Y plot, simulation dashboard).

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

This release delivers:

- A new vibration volume command. The command generates a maximum wrapping envelope for a vibrating part or assembly according to its positions.
- Wrapping command quality and performance:
 - Hole-free, smoother wrapping display for high-quality visualization
 - Enhanced memory management that allows for finer accuracy than ever before. The user can generate wrapped representations as close to the original geometry as possible with limited memory consumption.
- Improved mesh simplification commands performance for smoother rendering of simplified representations.

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

- Manikin-environment Interaction: Auto-update Mode for Constraints — When this mode is switched on, the manikin's position will automatically be re-adjusted to reflect the modification in one of the constraints. This enhancement addresses interactive manipulation and also serviceability scenarios (when used in conjunction with the DMU Fitting Simulator) by linking a manikin to a part and automatically updating the manikin's position when the part moves.
- Manikin Manipulation and Positioning
 - This release adds the ability to build a plane constraint with a specific orientation.
 - New Inverse Kinematics (IK) Mode — In addition to the existing IK mode that relies on the orientation of the pelvis, there is a new IK mode based on the orientation of the hand. Whereas the pelvis-based mode is used to find a general posture for the leg or arm, the new IK mode can be used to fine-tune the general posture and adjusting the extremities, such as the feet or hands.

- Improved balance management in IK behaviors ensures the manikin will be kept in balance when being manipulated using IK behaviors.
- Manikin Appearance — Smoother skin rendering and light reflectivity produce greater realism in the manikin's appearance.
- Vision Cone Enhancement — The color of the cones can be customized.

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

- This release adds the ability to mirror/copy/reset/swap preferred angles for upper and lower limbs.
- Visual feedback is enhanced with the coloring of the preferred angle with the worst score among all the Degrees of Freedom (DOF) of a corresponding segment in the postural score.

Infrastructure Solutions

MultiCAx Plug-ins

Withdrawn Product

5691-DPL — Effective October 29, 2002, the MultiCAx DPL Plug-in shareable product (5691-DPL) is withdrawn. It is replaced by the MultiCAx PDL Plug-in product. The PDL plug-in provides the same functions as the DPL Plug-in except that it supports the direct interface only. Migration paths to the MultiCAx PDL Plug-in product are provided in Software Announcement 202-297, dated October 29, 2002.

Enhanced Products

This release delivers the following improvements to the MultiCAx Plug-ins:

- Greater visualization quality for the U plug-in with suppression of some gaps between surfaces
- IDEAS MS9 support for indirect mode (NT only)
- UG V18 support for indirect mode (NT and AIX® only)

Complementary Solutions

New Complementary Solution

CATIA Team PDM Configuration (5691-TDM)

This SMARTEAM configuration provides a rapidly implemented, scalable, customizable, Windows-based, and seamlessly integrated PDM solution to CATIA V5. The configuration comprises the SMARTEAM Foundation, SMARTEAM Editor, and the SMARTEAM CATIA Integration products.

For details about these products and ordering information, refer to Software Announcement 202-301, dated October 29, 2002.

CATIA V5 Migrations and Upgrades

CATIA V5 provides migration paths from CATIA V4 workstation products and from withdrawn CATIA V5 products to CATIA V5 products on UNIX and Windows.

Upgrade paths have been developed to protect your V4 investment by providing, in most cases, a path to upgrade existing V4 configurations or combinations of V4 complementary (a la carte) products to their V5 functionally equivalent packages with no additional primary license (PLC) charge. Also, upgrades are offered

to take advantage of the evolving capabilities of CATIA V5. Upgrade paths will be announced over time as additional CATIA V5 applications become generally available.

Each upgrade or migration can be ordered up to the current quantity of the existing "from" program install base. The new upgrade/migration paths for this release are:

From Earlier Releases of CATIA V5 to CATIA V5.10

5693-GS1 Upgrade from 5693-WSF
5693-GS1 Upgrade from 5691-WSF

5693-ELD Upgrade from 5693-ECD
5691-ELD Upgrade from 5691-ECD

From CATIA V4 Products to CATIA V5 Products

5693-GS1 Upgrade from V4 WF3+SUD

5693-GS1 Upgrade from V4 SUD

5693-GS1+PDG Upgrade from V4
WF3+DYS+SOE+PA3+SUD (GS1)
5693-GS1+PDG Upgrade from V4
WF3+DYS+SOE+PA3+SUD (PDG)

5693-EHF Upgrade from V4 ELG
5691-EHF Upgrade from V4 ELG

From CATIA V4 Configurations to CATIA V5 Configs/Products/Custom Configurations (CC)

5691-EI2 Upgrade from V4 EEM

5691-EI2+EHF CC Upgrade from V4 EEF (EI2)
5691-EI2+EHF CC Upgrade from V4 EEF (5693-EHF)
5691-EI2+5691-EHF from V4 EEF (5691-EHF)

Withdrawn Migration Paths

Effective October 29, 2002, the following migration paths are withdrawn:

5693-WSF from V4 WF3+SUD
5691-WSF from V4 WF3+SUD

5693-WSF from V4 SUD
5691-WSF from V4 SUD

5693-PDG+WSF Upgrade from V4
WF3+DYS+SOE+PA3+SUD
5693-PDG+WSF Upgrade from V4
WF3+DYS+SOE+PA3+SUD

5691-PDG+WSF Upgrade from V4
WF3+DYS+SOE+PA3+SUD
5691-PDG+WSF Upgrade from V4
WF3+DYS+SOE+PA3+SUD

5693-PMG Upgrade from V4 PMP+TSA
5691-PMG Upgrade from V4 PMP+TSA

5693-SMG Upgrade from V4 SMP+TSA
5691-SMG Upgrade from V4 SMP+TSA

APIs

This release delivers a number of new interfaces for each of the CATIA V5 solutions in CATIA — CAA API Products (5691-CDC).

CATIA — Mechanical Design Solutions

CATIA — Interactive Drafting API

- DraftingInterfaces
 - CATAnnInfoDimension — Creation mode on circles
 - CATIDftBalloon — Interface for Balloon objects
 - CATIDftDatumFeature — Interface for Balloon objects
 - CATIDftDatumTarget — Interface for DatumTarget objects
- InteractiveInterfaces
 - CATIActivate — Activate/deactivate an element

CATIA — 3D Functional Tolerancing & Annotation

- CATTPSInterfaces
 - CATITPS3DGeoVisu — Drive specific visualization behavior of features within a tolerancing context
 - CATTPSDimensionType
 - CATTPSLinearDimensionSubType

CATIA — Shape Design & Styling Solutions

CATIA — Generative Shape Design API

- GSMInterfaces
 - CATGSMtolerantModelingSmoothingType — Smoothing types for tolerant modeling
 - CATGSMtranslateVectorType — Types for translation vector definition type
 - CATIGSMthickness — Get or/and set the thickness of a surface
- GSOInterfaces
 - CATGSOWrapSurfaceRefDirType — Used for WrapSurface creation and modifications
 - CATIGSOWrapSurface — Access data from the WrapSurface

CATIA — Product Synthesis Solutions

DMU Space Analysis API

- SpaceAnalysisInterfaces
 - CATISpaEnvironment — Initialize and/or clean up the space analysis environment.

CATIA — Knowledge Expert API

- KnowledgeInterfaces
 - CATDelegateInstanciationAdapter — Used by knowledgeware to instanciate objects.
 - CATDelegateInstanciation — Allows several types to use the same instanciation function.
 - CATInstanciationContext — When instanciating an object through CATIInstanciation, this class is used to pass some arguments.
- LiteralFeatures
 - CATICkeAddParametersDlg — Gets a dialog to append parameters on the feature that implements it

- CATICkeExpression — Interface dedicated to expressions management
- CATICkeOleSheet — An encapsulation of an array of arrays (a sheet)
- CATIOptConstraint — Encapsulates an optimization constraint
- CATKWEErorSeverity — Sets error severity (info, warning, error)
- CATKweInstanceAdapter — For adapter management

CATIA — Equipment & Systems Engineering Solutions

Solution Interfaces (CATSchPlatformInterfaces)

- CATEASchAppCntrName — Super class to derive implementation for interface CATISchAppCntrName Role: Adaptor
- CATEASchAppGroup — Super class to derive implementation for interface CATISchAppGroup Role: Adaptor
- CATISchAppCntrName — Manage application connector names
- CATISchAppGroup — List application group objects found in the document application

CATIA — Electrical Library API

- ElecDeviceltf
 - CATIElbBackShell
 - CATIElbBundleCnctPt
 - CATIElbCavity
 - CATIElbConnectorShell
 - CATIElbEquipment
 - CATIElbExternalSplice
 - CATIElbInternalSplice
 - CATIElbSingleConnector
 - CATIElbTermination

CATIA — Analysis Solutions

CATIA — Generative Part Structural Analysis API

- AnalysisMeshingModel
 - CATIMSHAdvWorkbenchAddin — Create an add-in within the advanced meshing tools workbench
- CATAnalysisVisulInterfaces
 - CATISamImage
 - CATISamImageColorMap
 - CATISamImageDisplayManager
 - CATISamImageFactory
 - CATSamVisuTypeName

CATIA — Manufacturing Solutions

CATIA — NC Manufacturing Review API

- ManufacturingInterfaces
 - CATIMfgUserDataAccess — Add a user data feature to a manufacturing setup or a manufacturing program
 - CATIMfgUserDataFeature — Manage a user data feature

- CATIMfgUserEditorActivity — Offers services to retrieve user tabpage frame of the activity object editor

CATIA — Infrastructure Solutions

CATIA — PPR PDM Gateway API

- CATPDMInterfaces
 - CATIPDMItem — Any CATIA document can be stored in the database tie to this interface
 - CATIPDMSession — Manage a V5 session by retrieving the current V5 object

CATIA — Object Modeler API

- AdvancedTopologicalOpe
 - CATCreateTopBlend_0 — Create a topological operator that computes a connecting surface between two wires
 - CATCreateTopBlend_1 — Create a topological operator that computes a connecting surface between two wires
 - CATCreateTopGeodesicDistanceTool — Construct the object for the geodesic computation
 - CATCreateTopSweepWireSkinCircleVariable — Create a CATTopSweepWireSkinCircle operator
 - CATTopBlend — Class defining the topological operator that connects two wires with a skin
- BasicTopologicalOpe
 - CATCreateTopInfiniteLineFromDirection — Create an infinite line passing through a point in a given direction
 - CATCreateTopLineInfiniteOperatorFromDirection — Create an infinite line passing through a point in a given direction
 - CATIAApplicationFrame
 - CATMMUpdateSettings — Class to return the settings of the update section of Part Design
- MecModInterfaces
 - CATITTRSAdvanced — Interface dedicated to manage additional information for Technologically and Topologically Related Surface (TTRS)
 - CATMmrTTRSAdmissibleType — Enumeration
 - CATMmrTTRSAssociationCase — Enumeration
 - CATMmrTTRSFeatureOfSize — Enumeration
 - CATMmrTTRSSurfaceCanonicity — Enumeration
- System
 - CATBaseCollection — This class provides a partial default implementation for the CATIACollection interface.
 - CATBaseObject — This class provides a default implementation for the CATIABase interface.
 - CATCreateEnumVARIANT — Creates a new object which implements IEnumVARIANT.
 - CATIACollection — Represents the base object for collections.
 - IEnumVARIANT — Interface for traversing CATVariant collections.

- TopologicalOperators
 - CATCreateTopEuclidianDistanceTool — Constructs the object for euclidean computations
 - CATCreateTopTrim_0 — Constructs an operator that mutually trims two bodies
 - CATFilletRadiusType — Defines the way the fillet radius is defined
 - CATTopSweepWireSkin — Class defining the topological operator that creates a sweep between a skin body and a wire body
 - CATTopSweepWireSkinCircle — Class defining the topological operator that creates a sweep between a skin body and a wire body
- VisualizationBase
 - CAT3DArrowGP — Creates the graphic primitive of a 3D arrow
 - CATReadCgr_4 — Reads a list of CGR files and returns their graphic data as an array of pointers to CAT3DRep objects
 - CATWriteCgr_3 — Saves, in the CGR format, the representation of a graphic scene in a buffer
 - CATWriteCgr_4 — Saves the representation of a graphic scene in a CGR file
 - CATWriteCgr_5 — Saves the representation of a graphic scene in a CGR file
 - MARKER_MULTIPLE
 - MARKER_SINGLE
 - MAX_VIEWPOINT_BUFFER
- VPMInterfaces
 - CATICfgGroupModif — Manage modification
 - CATICfgMileStone — Manage a milestone
 - CATICfgProgram — Create a milestone
 - CATListOfCATICfgBasicEffectivity_var — Manage collections
 - CATListOfCATICfgMileStone_var — Manage collections
 - CATListOfCATICfgProgram_var
 - CATListOfCATIVpmAFLProduct — Collection class for CATIVpmAFLProduct_var
 - ENOVIDisplayableValue — Return the object as a displayable string
 - ENOVIPrdConnection — Interface to connection pointing objects, connection objects, and connectors through branches
 - GetVPMIdentifierGeneratorMgr
 - VPMIdentifierGeneratorMgr — Manage the identifier generator

Though delivered with CATIA V5, these APIs are for use with the CAA Rapid Application Development Environment (RADE) tools. For more information about the RADE products, visit the PLM Web site:

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

From the menu frame, select **ENOVIA Products** to get to the ENOVIA page. From the menu frame on the ENOVIA page, select **RADE**.

Also, for the latest announcement about the CAA RADE tools, refer to Software Announcement 202-298, dated October 29, 2002.

CATIA V5 WLS

V5.10 delivers a number of enhancements to the WLS portfolio.

User Companion for Mechanical Design Configuration (5691-MDC)

- Courses have been upgraded to reflect new CATIA V5.10 capabilities.
- A new course for Knowledge Expert has been added.
- Exercises have been reviewed and new ones have been developed to better capitalize on experience acquired and to reflect best practices, especially in the automotive industry.
- CATIA P2-only capabilities have been marked in the Companion using a P2-specific logo so that P1 users know that they do not have access to the capability.
- Feedback and requests from customers have been integrated into the Companion when these are of general interest to any industry and any company.
- Knowledge expert capabilities have been included to reflect the utilization of company rules and knowledge-based design.
- The skilet dedicated to CATIA V4 users has been upgraded to show the benefits of CATIA V5 utilization.
- Additional instructor material is provided (foils plus a recommended agenda of the courses).
- The quality of the material has been reviewed and improved to reflect recommendations from customers and business partners.

User Companion for Hybrid Design Product (5691-HDS)

- Courses have been upgraded to reflect new CATIA V5.10 capabilities.
- Exercises have been reviewed and new ones have been developed to better capitalize on experience acquired and to reflect best practices, especially in the automotive industry.
- Feedback and requests from customers have been integrated into the Companion when these are of general interest to any industry and any company.
- Additional instructor material is provided (foils plus a recommended agenda of the courses).
- The quality of the material is reviewed and improved to reflect recommendations from customers and business partners.

User Companion for DMU Product (5691-DMU)

- Courses have been upgraded to reflect new CATIA V5.10 capabilities.
- A new "Discover It" type simulation is included in all courses.
- More automotive-oriented exercises have been added.

Education Support

Schedules and Enrollment

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

CATIA V5 WLS

Training, whether classroom or computer-based, always plays a critical role in how well users perform their job.

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) consist of the CATIA V5 products in the P1 and P2 platforms, respectively. CATIA — Educational HEAT Configuration 2 (5691-ED2) will include the latest version of CATIA V5 WLS products, including the User Companion for DMU, upon request.

The new P1 and P2 products announced in this release are included in these configurations.

P1 and P2 products withdrawn in this release are removed from these configurations. For current HEAT customers, the SMARTEAM CATIA Team PDM Configuration (5691-TDM) will be available via special bid to replace CATIA — Team P DM 1 Product that was formerly available in the CATIA Educational HEAT configurations (5691-ED1 and 5691-ED2).

Note: Although CATIA P1 solutions are available on Windows 98 through V5.10, the SMARTEAM CATIA Team PDM Configuration (5691-TDM) is not available on the Windows 98 platform.

Technical Information

Hardware Requirements

Summary of Changes for V5.10

- This release is the last to be supported on Windows 98 and Windows NT®. The next available Windows release will only be supported on Windows 2000 and Windows XP.
- This release is the last to be supported on Sun Solaris 2.6 and Solaris 7. The next available Solaris release will be supported only on Solaris 8.
- Support for HP-UX workstations based on the HP-UX PA7000 processor is withdrawn. Only workstations based on the PA8000 processor running HP-UX 11.0 ACE are supported.
- Support is added for:
 - Sun Blade 2000 workstations
 - Sun XVR-500 graphics adapter

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 provide paging space. Installation of all CATIA V5 products requires 1.7 GB on Windows, 2.1 GB on AIX, 2.0 GB on HP-UX and IRIX, and 1.8 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 graphic 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 800 x 600 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 at:

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

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 in a Windows 98 Environment

Important: The Windows 98 environment applies only to CATIA V5 P1. CATIA V5 P2 and P3 products do not run on Windows 98. Additionally, the capability supplied by the CATIA — COM 1 to 2 Extension (C12) product for

running P2 products on P1 configurations does not apply to Windows 98.

Note: This release is the last to be supported on Windows 98. The next available Windows release will only be supported on Windows 2000 and Windows XP.

Windows 98 clients can use nodelock licensing or can run with Windows NT, Windows 2000, Windows XP, or UNIX license servers.

- System Unit: An Intel® Pentium® II-, Pentium III-, or Pentium 4-based workstation running Windows 98.
- Graphics Adapter: An OpenGL-capable graphics adapter is required.

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

- 16-bit, high-color, double buffered visual
- 16-bit, Z-buffer (when hardware accelerated)
- Stencil buffer (1-bit)
- Minimum supported resolution is 800 x 600; a resolution of 1280 x 1024 is recommended for usability reasons.

Whereas Dassault Systemes has certified complete configurations (workstation model and graphics adapter) for CATIA V5 running on Windows NT, Windows 2000, or Windows XP, only generic graphics adapters that have been tested on Windows 98 will be documented. An updated list of these adapters is published at:

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

- Network Adapter: An active LAN adapter (installed and configured) is required for licensing purposes.

Hardware Requirements for a Windows NT, Windows 2000, or Windows XP Environment

Note: This release is the last to be supported on Windows NT. The next available Windows release will only be supported on Windows 2000 and Windows XP.

- System Unit: An Intel Pentium II-, Pentium III-, or Pentium 4-based workstation running Windows NT Workstation Version 4.0, Windows 2000 Professional Edition, or Windows XP Professional Edition.

- Graphics Adapter: A graphics adapter with a 3D OpenGL accelerator is required. Note that graphic 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: 800 x 600; 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 licensing purposes.

An updated list of supported Windows NT, Windows 2000, and Windows XP configurations certified by Dassault Systemes for running CATIA V5 products can be found at:

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

Hardware Requirements in an AIX Environment

- System Unit: Any RS/6000®, based on PowerPC 604™ (166 MHz minimum clock speed), POWER2™, or POWER3™ processor families, supported on AIX V4.3.3 or AIX V5.1.
- Graphics Adapter: One of the following graphics adapters is required.
 - GXT500P
 - GXT550P
 - GXT800P
 - GXT800M
 - GXT2000P
 - GXT3000P
 - GXT4000P
 - GXT4500P
 - GXT6000P
 - GXT6500P

Hardware Requirements in an HP-UX Environment

- System Unit: Any B-Class, C-Class, or J-Class workstation based on a PA8000 processor and supported on HP-UX Version 11.0 ACE (Additional Core Enhancements November, 1999).

Note: Beginning with this release, workstations based on the PA7000 processor family are no longer supported.

- Graphics Adapter: One of the following graphics adapters is required:
 - Visualize-FXE
 - Visualize-FX2
 - Visualize-FX4
 - Visualize-FX5
 - Visualize-FX6
 - Visualize-FX10

Hardware Requirements in a SGI-IRIX Environment

- System Unit: Any O2, Indigo2, Octane, Octane2, Onyx2, Fuel, or Onyx3000 workstation based on an R5000, R10000, R12000, or R14000 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

Hardware Requirements in a Sun Solaris Environment

- System Unit: Any Ultra1, Ultra2, Ultra10, Ultra30, Ultra60, Sun Blade 100, Sun Blade 1000, or Sun Blade 2000 workstation based on the UltraSPARC processor supported on Solaris 2.6.0, Solaris 7, or Solaris 8.

Note: This release is the last to be supported on Sun Solaris 2.6 and Solaris 7. The next available Solaris release will be supported only 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

Software Requirements

Common Software Requirements

CATIA V5 WLS has the same software requirements as CATIA V5 unless otherwise noted.

Summary of Changes for V5.10

- This release is the last to be supported on Windows 98 and Windows NT. The next available Windows release will only be supported on Windows 2000 and Windows XP.
- This release is the last to be supported on Sun Solaris 2.6 and Solaris 7. The next available Solaris release will be supported only on Solaris 8.
- Support for HP-UX workstations based on the HP-UX PA7000 processor is withdrawn. Only workstations based on the PA8000 processor running HP-UX 11.0 ACE are supported.
- Support is added for AIX V5.1.
- Support for Versatec and Calcomp formats is withdrawn.

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

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

Software Requirements in a Windows 98 Environment

Important: The Windows 98 environment applies only to CATIA V5 P1. CATIA V5 P2 and P3 products do not run on Windows 98. The capability supplied by the CATIA — COM 1 to 2 Extension (C12) product for running P2 products on P1 configurations does not apply to Windows 98. CATIA V5 WLS is not supported on Windows 98.

Note: This release is the last to be supported on Windows 98. The next available Windows release will only be supported on Windows 2000 and Windows XP.

Windows 98, with the following components at the minimum levels indicated, is required:

- A Microsoft implementation of OpenGL libraries, as delivered with Windows 98. These libraries may have to be modified to accommodate the selected graphic adapter when installing the graphic adapter and its associated drivers. For recommendations related to driver levels based on tested graphics adapters, 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 in a Windows NT, Windows 2000, or Windows XP Environment

Note: This release is the last to be supported on Windows NT. The next available Windows release will only be supported on Windows 2000 and Windows XP.

One of the following:

- Windows NT Workstation Version 4.0 with Service Pack 4, Service Pack 5, or Service Pack 6a,
- Windows 2000 Professional Edition, or
- Windows XP Professional Edition

with the following components at the levels indicated, is required:

- A Microsoft implementation of OpenGL libraries as delivered with Windows NT 4.0, Windows 2000, and Windows XP. These libraries may have to be updated 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.

Note: For remote access from networked clients, Terminal Server is supported by the Windows NT 4.0 Terminal Server Edition at the Service Pack 4 level on the Windows NT server. Access through standard browsers requires Citrix MetaFrame 1.0 to also be installed on the server. Terminal Server is also available with Windows 2000 Server, Windows 2000 Advanced Server, and Windows XP Professional Edition.

Software Requirements in an AIX Environment

AIX V4.3.3 or AIX V5.1 with the components listed below at the minimum levels is required.

- IBM C Set++ for AIX Application Runtime
 - At a minimum level of V4.0.2 (5765-D52) for AIX V4.3.3
 - At a minimum level of V5.0.2 (5765-E26) for AIX V5.1

The C Set++ Application Runtime is shipped with the AIX operating system.

- IBM XL Fortran Runtime Environment for AIX:
 - 5765-C11 or 5801-AAR-7070 (part number 04L2123) at a minimum level of 5.1.0 or
 - 5765-E03 at a minimum level of 7.1.0 (AIX V5.1 requires XL Fortran Runtime Environment 7.1.0)
- OpenGL and GL3.2 Runtime Environment (delivered with the operating system)
- Common Desktop Environment (CDE; delivered with the operating system)

Software Requirements in an HP-UX Environment

Note: Support for HP-UX workstations based on the HP-UX PA7000 processor is withdrawn. Only workstations

based on the PA8000 processor running HP-UX 11.0 ACE are supported.

HP-UX Version 11.0 ACE (Workstation Additional Core Enhancements for HP-UX 11.0 — November, 1999) with the following components at the minimum levels indicated, is required:

- 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 in an SGI IRIX Environment

IRIX 6.5.3m, with the following components at the minimum levels indicated, is required:

- 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 in a Sun Solaris Environment

Note: This release is the last to be supported on Sun Solaris 2.6 and Solaris 7. The next available Solaris release will be supported only on Solaris 8.

Sun Solaris 2.6.0 H/W 5/98 or higher, Solaris 7, or Solaris 8 with the following components at the minimum levels indicated, is required:

- 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)

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

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 ENOVIA VPM 1:
 - For IBM DB2 database servers —DB2 Universal Database® (UDB) V7.1 FP3 or 7.2
 - For Oracle database servers —Oracle V8.1.7

CATIA V4 CDM and ENOVIA VPM 1 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 5 on the supported Windows platforms with ENOVIA VPM 1 through ENOVIA 3d com requires, on the client side, either (depending whether the database server is a DB2® or an Oracle server):
 - DB2 UDB V7.1 FP3 or V7.2 Client for Windows
 - Oracle Client Version 8.1.7 for Windows
- Math Kernel Libraries —When on an Intel architecture in a Windows NT, Windows 2000, or Windows XP environment, the performance of the following products can be improved through the use of Intel Math Kernel Libraries (MKL) 5.1.e or 5.2:
 - 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)

MKL Libraries 5.2 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.

- SCSL Scientific Library —When running on an R5000, R10000, R12000, or R14000 MIPS architecture in an IRIX environment, the performance of the ELFINI Structural Analysis 2 (5691-EST) product can be improved through the use of the SGI SCSL Scientific Library. SCSL Scientific Library 1.3 can be downloaded free of charge from:

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

This requirement is optional.

Specific Software Requirements for CATIA V5 WLS

- An HTML browser is required.
 - In a UNIX environment (AIX, HP-UX, IRIX, Solaris) —Netscape Navigator at minimum level 4.75.
 - In a Windows environment — Either Microsoft Internet Explorer at minimum level 5.0 Service Pack 1. The level of the Microsoft Virtual Machine must be at minimum level 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 3.0, 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 V5.10, 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>

Access to Product Information

Product information is delivered on the product CDs in HTML format. An HTML browser is required to access this documentation.

- In a UNIX environment (AIX, HP-UX, IRIX, Solaris), Netscape Navigator at a minimum level of 4.75.
- In a Windows environment, one of the following:
 - Microsoft Internet Explorer at a minimum level of 5.0
 - Netscape Navigator at a minimum level of 4.75

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 on the product CDs in PDF form. Viewing and printing of the PDF files requires the Adobe Acrobat Reader at a minimum level of 3.0. The latest level of the reader can be downloaded, at no charge, from:

<http://www.adobe.com>

Prerequisites for the Licensing Environment

- Windows workstations must have a LAN card (Ethernet or token ring) and TCP/IP installed and properly configured, but there is no need to have the workstations connected to the network.
- No additional licensing software is required when accessing nodelock licenses. Nodelock licensing for Windows 98 became available beginning with CATIA V5.5.
- License Use Management (LUM) is required to serve concurrent licenses across a network. A LUM configuration file (i4ls.ini) is required on CATIA clients to access concurrent licenses from these servers.

LUM Summary

Operating Environment	License Type for P1	License Type for P2	License Type for P3
Windows 98	Nodelock or UNIX server or Windows NT server or Windows 2000 server or Windows XP server	Not supported	Not supported
Windows NT Windows 2000 Windows XP	Server and nodelock	Server and nodelock	Server and nodelock
UNIX (AIX, HP UX, IRIX, Solaris)	Server and nodelock	Server and nodelock	Server and nodelock

IBM LUM at a minimum level of:

- V4.5.5 is required on UNIX or Windows NT license servers.
- V4.5.8 is required on Windows 2000 Professional license servers.
- V4.6.2 is required for Windows 2000 Server and Advanced Server license servers.
- V4.6.5 is required when the High Availability Licensing (HAL) mechanism offered by LUM is used.
- V4.6.4 is required on Windows XP license servers.

Windows 98 is not supported as a LUM license server. Windows 98 clients require either:

- Nodelock licensing
- or
- A UNIX, Windows NT, Windows 2000, or Windows XP LUM license server

LUM is shipped with CATIA V5. Various release levels of LUM may be obtained, at no charge, from:

<http://www.software.ibm.com/is/lum/download.html>

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.

Printer and Plotter Support

- UNIX — CATIA V5 provides support for 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.

Note: Support for the Versatec (VCGL and VGS) and Calcomp 907 formats is withdrawn in this release. Customers have access to several work-arounds:

- A CAA V5 print partner solution, such as ZEH Print. Such solutions have Versatec or C907 support and are fully integrated into the V5 print environment.
- Development of driver support using CAA.
- Use of other file formats, such as PostScript, HPGL2, or TIFF supported by the printer.
- Conversion of other file formats into the Versatec or C907 file format using external tools:
 - Fuji Xerox (Japan only).

- gralxConvert from GTS-GRAL (info@gtsgral.de).
- ZEH Print from ZEH (www.zeh.com).

- 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 MQSeries®

Using MQSeries communication tools, some batch operations can now be launched remotely. When implemented at the batch level, this optional feature requires MQSeries at minimum level 5.2. For availability of client and server components on supported platforms, visit:

<http://www.ibm.com/software/ts/mqseries/messaging/>

MQSeries Client is required on systems where the transaction is initiated. MQSeries 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).
- The add-on product CATIA — COM 1 to 2 Extension (5693-C12) allows all P2 5693-XXX add-on products (AOPs) 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 add-on (5693-C12) product.
- 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 add-on products are available to P3 configurations as shareable or add-on products.
- CATIA V5 add-on and shareable products require a license for a configuration that includes at least the prerequisite products. Licenses 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 — V4 Integration 2 (V4I) is included in all P2 and P3 configurations, and CATIA — V4 Integration 1 (V4I) is included in all P1 configurations.

Product-Specific Prerequisites

- CADAM Drafting for V5 (CCD) requires a configuration that includes CATIA — Interactive Drafting 1 (ID1).
- CATIA — CADAM Interface 1 (CC1) requires CATIA — Interactive Drafting 1 (ID1).

- CATIA — Circuit Board Design 2 (CBD) requires CATIA — Part Design 2 (PDG).
- CATIA — Generative Shape Optimizer 2 (GSO) requires CATIA — Generative Shape Design 2 (GSD).
- CATIA — FreeStyle Optimizer 2 (FSO) requires CATIA — FreeStyle Shaper 2 (FSS).
- CATIA — Generative Assembly Structural Analysis 2 (GAS), CATIA — FEM Surface 2 (FMS), and CATIA — ELFINI Structural Analysis 2 (EST) require CATIA — Generative Part Structural Analysis 2 (GPS).
- Human Measurements Editor 2 (HME), Human Posture Analysis 2 (HPA) and Human Activity Analysis 2 (HAA) require Human Builder 2 (HBR).
- CATIA — Piping & Instrumentation Diagrams 2 (PID), CATIA — HVAC Diagrams 2 (HVD), CATIA- Electrical Connectivity Diagrams 2 (ELD), CATIA — Waveguide Diagrams 2 (WGD), and CATIA — Tubing Diagrams 2 (TUD) require CATIA — Systems Diagrams 2 (SDI).
- CATIA — Systems Diagrams 2 (SDI) requires CATIA — Interactive Drafting 1 (ID1).
- CATIA — Automotive Class A Optimizer 3 (ACO) requires CATIA — Automotive Class A 3 (ACA).
- CATIA — Lathe Machining 1 (LG1) and CATIA — Prismatic Machining 1 (PG1) require CATIA — NC Manufacturing Review 1 (NG1).
- CATIA — Lathe Machining 2 (LMG), CATIA — Prismatic Machining 2 (PMG), CATIA — 3 Axis Surface Machining 2 (SMG), CATIA — Advanced Machining 2 (AMG), CATIA — NC Manufacturing Verification 2 (NVG), and CATIA — Prismatic Machining Preparation Assistant 2 (MPA) require CATIA — NC Manufacturing Review 2 (NCG).

Note: This prerequisite does not apply when adding a product that requires CATIA — NC Manufacturing Review 2 (NCG) to a configuration in which it is already included (5691-PM2, 5691-MO2, and 5691-RM2).

- CATIA — Multi-Axis Surface Machining 2 (MMG) requires CATIA — 3 Axis Surface Machining 2 (SMG).
- CATIA — Electrical Cableway Routing 2 (ECR) requires CATIA — Systems Space Reservation 2 (SSR).
- CATIA — DMU Space Engineering Assistant 2 (SPE) requires DMU Space Analysis 2 (SPA). ENOVIA LCA V5 corequisite products are the Enterprise PPR Infrastructure and CATIA Hub. These are included in the following ENOVIA LCA V5 user roles:
 - System & Data Administration (RVR)
 - Professional User (MGR)
 - Engineering User (EGR)

HTTP Server V1.3.19 is a corequisite. The HTTP Server can be downloaded from the Apache Web site:

<http://www.apache.org>

- CATIA — Weld Design 1 (WD1) requires CATIA — Assembly Design 1 (AS1)
- Photo Studio Optimizer 2 (PSO) requires Photo Studio 2 (PHS)
- CATIA — Part Design Feature Recognition 1 (FR1) requires CATIA — Part Design 1 (PD1)

Customers planning to use CATIA V5.10 products with ENOVIAVPM V1.5 must install an ENOVIAVPM corequisite

Program Temporary Fix (PTF) to obtain the required interoperability level. The required PTF level could vary depending on whether the base level or a subsequent service pack level of CATIA V5.10 is being used. Refer to the CATIA V5.10 Program Directory for interoperability information or contact your local PLM Support Center for specific details on the PTF and service pack levels and how to obtain them. Or visit the PLM Technical Support Web site:

<http://techsupport.services.ibm.com/engineering/support>

On the left side, select **Fixes, tools**.

Planning Information

Direct Customer Support: Direct customer support is provided under the Product Lifecycle Management (PLM) enhanced support offering. This fee service enhances the customer's productivity by providing voice and electronic access to the PLM Support Centers. The PLM help desk will answer questions pertaining to the installation, administration, use, and handling of suspected software defects for eligible products.

Additional information about the enhanced support offering and other available PLM service offerings is available by visiting the PLM Web site:

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

On the left side, click on **Support**. Then select **The PLM Technical Support Web site**, and then click on **Service and Support Offerings**.

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 (SK3T-5100)

Ships with all orders for P1 configurations.

- Seven CDs (LCD4-4077)
 - AIX
 - HP (2)
 - IRIX
 - Solaris
 - Windows NT/2000/XP
 - Windows 98

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

- Program Directory CD (LCD4-5571)
- LUM CD (LCD4-4195)
- Publications
 - English softcopy publications collection kit (SK3T-4124)
 - IBM informal documents
 - Licensed Program Specification (GH52-1328)
 - Registration Memorandum (GI10-8107)
 - Memorandum to Current Users (GI10-9873)

Platform 2 Box Kit (SK3T-4132)

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

- Nine CDs (LCD4-3756)
 - AIX (2)
 - HP (2)
 - IRIX (2)
 - Solaris (2)
 - Windows NT/2000/XP

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

- Program Directory CD (LCD4-5571)
- LUM CD (LCD4-4195)
- Publications
 - English softcopy publications collection kit (SK3T-4124)
 - IBM informal documents
 - Licensed Program Specification (GH52-1328)
 - Registration Memorandum (GI10-8107)
 - Memorandum to Current Users (GI10-9873)

Platform 3 Box Kit (SK3T-7778)

Ships with all orders for P3 configurations.

- Nine CDs (LCD4-5339)
 - AIX (2)
 - HP (2)
 - IRIX (2)
 - Solaris (2)
 - Windows NT/2000/XP

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

- Program Directory CD (LCD4-5571)
- LUM CD (LCD4-4195)
- Publications
 - English softcopy publications collection kit (SK3T-4124)
 - IBM informal documents
 - Licensed Program Specification (GH52-1328)
 - Registration Memorandum (GI10-8107)
 - Memorandum to Current Users (GI10-9873)

Shipping Package

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

- Licensing IBM's Product Lifecycle Management (PLM) Solutions Software (GI11-3619)
- COE Brochure (GH52-1319)
- With an order of the CADAM Drafting for V5 (CCD) add-on product, the three CCD product CDs (LCD4-4076)
- With all orders for P1, P2, or P3 configurations, three CATIA V5 API CDs (LCD4-5342)

CATIA V5 WLS Packaging

This product includes the following deliverables in English:

- Code CDs (in separate packages as follows):
 - Two CDs (LCD4-4667) CATIA V5 User Companion for Mechanical Design — Windows

- Two CDs (LCD4-4668) CATIA V5 User Companion for Mechanical Design — UNIX
- One CD (LCD4-5233) CATIA User Companion for V4 Mechanical Design — UNIX
- One CD (LCD4-5234) CATIA V5 User Companion for Hybrid Design — Windows
- One CD (LCD4-5235) CATIA V5 User Companion for Hybrid Design — UNIX
- One CD (LCD4-5452) User Companion for DMU — Windows
- One CD (LCD4-5453) User Companion for DMU — UNIX

Informal documents:

- CATIA WLS Licensed Program Specifications (GI11-1328)
- CATIA WLS Registration Memorandum (GI11-1329)
- CATIA WLS Memorandum to Current Licensees (GI11-2463)
- WLS Licensed Program Specifications (GI11-2632)
- WLS Registration Memorandum (GI11-2631)
- WLS Memorandum to Current Licensees (GI11-2633)
- Licensing IBM's Product Lifecycle Management (PLM) Solutions Software (GI11-3619)

The CATIA WLS products are currently available in English. National language versions (French, German, and Japanese) will be available at a later date.

Security, Auditability, and Control

The announced programs use the security and auditability features of the following operating systems:

- AIX for the RS/6000 family
- HP-UX for the HP platform
- IRIX for the SGI platform
- Solaris for the Sun platform
- Windows NT, Windows 2000, Windows 98, and Windows XP for Intel Pentium platforms

If sensitive data is sent over external communications facilities, users may wish to pursue the use of cryptography.

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.

Shipment of this release of CATIA V5 is scheduled to be completed by December 20, 2002; shipment of this release of CATIA V5 WLS is scheduled to be completed by January 31, 2003.

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 add-on products (5693-XXX)
- Optional licensed shareable product program order (5691-XXX)
- One system program order (SPO), 5628-ACN

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

CATIA V5 is available only on CD-ROM.

Basic License

Program Numbers for Products Announced in V5.10

Note: Products without the CATIA brand prefix are available for ordering with CATIA V5 and ENOVIA Portal V5.

New Configurations

CATIA — Styled Mechanical Design 1 Configuration	5691-YM1
CATIA — Styled Mechanical Design 2 Configuration	5691-YM2
CATIA — Electrical Wire Harness Installation 2 Configuration	5691-EI2
CATIA — Automotive Body in White Design 3 Configuration	5691-AB3
CATIA — Business Process Knowledge Definition 3 Configuration	5691-KD3

New Shareable Products

CATIA — Part Design Feature Recognition 1 Product	5691-FR1
CATIA — Wireframe & Surface 1 Product	5691-WS1
CATIA — PPR PDM Gateway 1 Product	5691-PX1
MultiCAX PD Plug-in	5691-PDL
CATIA — Business Process Knowledge Template 2 Product	5691-BK2
CATIA — Electrical Harness Flattening 2 Product	5691-EHF
CATIA — Hanger Design 2 Product	5691-HGR
CATIA — Prismatic Machining Preparation Assistant 2 Product	5691-MPA
CATIA — ENOVIA Work Package Exchange 2 Product	5691-EWE

New Add-on Products (AOP)

CATIA — Part Design Feature Recognition 1 AOP	5693-FR1
CATIA — Generative Shape Design 1 AOP	5693-GS1
CATIA — Electrical Library 1 AOP	5693-EL1
CATIA — Electrical Harness Installation 1 AOP	5693-EH1
CATIA — ENOVIA Work Package Exchange 1 AOP	5693-EW1
CATIA — PPR PDM Gateway 1 AOP	5693-PX1
CATIA — Business Process Knowledge Template 2 AOP	5693-BK2
CATIA — Electrical Harness Flattening 2 AOP	5693-EHF
CATIA — Hanger Design 2 AOP	5693-HGR
CATIA — Prismatic Machining Preparation Assistant 2 AOP	5693-MPA
CATIA — ENOVIA Work Package Exchange 2 AOP	5693-EWE

Withdrawn Products: Effective October 29, 2002, the following products are withdrawn:

- CATIA — Team PDM 1 Product (5691-TD1) and CATIA — Team PDM1 AOP (5693-TD1).

AOP indicator code 4279 is withdrawn from the following configurations:

5691-MD1	5691-ME1
5691-SD1	5691-SL1
5691-DR1	5691-XM1
5691-MR1	

5691-MS2	5691-HD2
5691-MD2	5691-ME2
5691-DP2	5691-SD2
5691-XM2	5691-EF2
5691-ER2	5691-SA2
5691-FS2	5691-SQ2
5691-RE2	5691-PM2
5691-MO2	5691-PL2
5691-RM2	5691-CV2
5691-DI2	

5691-TA3	5691-AC3
5691-SL3	

5691-ADD	5691-SBD
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- MultiCAX P Plug-in Product (5691-DPL).
- CATIA — Wireframe & Surface 2 Product (5691-WSF) and CATIA — Wireframe & Surface 2 Add-on Product (5693-WSF).

AOP indicator code 4282 is withdrawn from the following configurations:

5691-MD1	5691-ME1
5691-SD1	5691-SL1
5691-DR1	5691-XM1
5691-MR1	

5691-MS2	5691-DP2
5691-SD2	5691-ER2
5691-EF2	5691-FS2
5691-SA2	5691-PL2
5691-SQ2	

5691-TA3	5691-AC3
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5691-SBD	
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- CATIA — Electrical Cable Definition 2 Product (5691-ECD) and CATIA — Electrical Cable Definition 2 AOP (5693-ECD). AOP indicator code 4218 is withdrawn from the following configurations:

5691-MD1	5691-ME1
5691-SD1	5691-SL1
5691-DR1	5691-XM1
5691-MR1	
5691-MS2	5691-HD2
5691-MD2	5691-ME2
5691-DP2	5691-SD2
5691-XM2	5691-EF2
5691-ER2	5691-SA2
5691-FS2	5691-SO2
5691-RE2	5691-PM2
5691-MO2	5691-PL2
5691-RM2	5691-CV2
5691-TA3	5691-AC3
5691-SL3	

Updated Configurations: In this release:

- Wireframe & Surface 1 (WS1) is added to the Mechanical Design 1 (MD1) and Mechanical Engineering 1 (ME1) configurations. CATIA — Wireframe & Surface 1 AOP (5693-WS1), indicator code 4281, is therefore withdrawn from these configurations.
- In the CATIA — Automotive Class A 3 (5691-AC3) configuration, CATIA — Real Time Rendering 1 (RT1) is replaced by CATIA — Real Time Rendering 2 (RTR). CATIA — Real Time Rendering 2 AOP (5693-RTR), indicator feature number 4288, is therefore withdrawn from this configuration.

Customization Option: Feature number 3444 is being assigned to each new program number for use in customizing orders. It can be used to suppress shipment of media and documentation. This feature can be specified on the 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)

Note: This support requires LUM at a minimum level of 4.6.4. 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 V5.9 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 Configuration

5691-YM1 CATIA — Styled Mechanical Design 1 Configuration

0011 PLC YM1 Config
 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 Shareable Products

5691-FR1 CATIA — Part Design Feature Recognition 1 Product

0011 PLC FR1 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+

5691-PX1 CATIA — PPR PDM Gate 1 Product

0011 PLC PX1 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+

5691-WS1 CATIA — Wireframe & Surface 1 Product

0011 PLC WS1 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 P1 Add-On Products (AOP)

5693-FR1 CATIA — Part Description Feature Recognition 1 AOP

0011 PLC FR1 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+

**5693-GS1 CATIA — Generation Shape
Design 1 AOP**

0011 PLC GS1 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+

**5693-EL1 CATIA — Electrical
Library 1 AOP**

0011 PLC EL1 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+

**5693-EH1 CATIA — Electrical Harness
Installation 1 AOP**

0011 PLC EH1 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+

**5693-EW1 CATIA — ENOVIA Work Package
Exchange 1 AOP**

0011 PLC EW1 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+

5693-PX1 CATIA — PPR PDM Gate 1 AOP

0011 PLC PX1 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 Configurations

**5691-YM2 CATIA — Styled Mechanical
Design 2 Configuration**

0011 PLC YM2 Config
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+

**5691-EI2 CATIA — Electrical Wire
Harness 2 Configuration**

0011 PLC EI2 Config
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

**5691-BK2 CATIA — Business Process Knowledge
Template 2 Product**

0011 PLC BK2 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+

**5691-EHF CATIA — Electrical Harness
Flattening 2 Product**

0011 PLC EHF 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+

5691-HGR CATIA — Hanger Design 2 Product

0011 PLC HGR 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+

**5691-MPA CATIA — Prismatic Machine
Preparation 2 Product**

0011 PLC MPA 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+

**5691-EWE CATIA — ENOVIA Work Package
Exchange 2 Product**

0011 PLC EWE 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 Add-On Products (AOP)

**5693-BK2 CATIA — Business Process Knowledge
Template 2 AOP**

0011 PLC BK2 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+

**5693-EHF CATIA — Electrical Harness
Flattening 2 AOP**

0011 PLC EHF 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+

5693-HGR CATIA — Hangar Design 2 AOP

0011 PLC HGR 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+

**5693-MPA CATIA — Prismatic Machine
Preparation 2 AOP**

0011 PLC MPA 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+

**5693-EWE CATIA — ENOVIA Work Package
Exchange 2 AOP**

0011 PLC EWE 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 Configurations

**5691-AB3 CATIA — Automotive Body-in-White
Design 3 Configuration**

0011 PLC AB3 Config
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+

**5691-KD3 CATIA — Business Process
Knowledge Definition 3 Configuration**

0011 PLC KD3 Config
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/Migration Paths

**PLC
Upgrade
Feature
Number**

From CATIA V5 to CATIA V5.10

0913 5693-GS1 Upgrade from 5693-WSF
0914 5693-GS1 Upgrade from 5691-WSF

0915 5693-ELD Upgrade from 5693-ECD
0915 5691-ELD Upgrade from 5691-ECD

From CATIA V4 Products to CATIA V5 Products

0918 5693-GS1 Upgrade from V4 WF3+SUD

0919 5693-GS1 Upgrade from V4 SUD

0920 5693-GS1+PDG Upgrade from V4
WF3+DYS+SOE+PA3+SUD (GS1)

0920 5693-GS1+PDG Upgrade from V4
WF3+DYS+SOE+PA3+SUD (PDG)

0921 5693-EHF Upgrade from V4 ELG
0921 5691-EHF Upgrade from V4 ELG

**From CATIA V4 Configurations to CATIA V5
Configs/Products/Custom Configs (CC)**

0922 5691-EI2 Upgrade from V4 EEM

0923 5691-EI2+EHF CC Upgrade from V4 EEF (EI2)
0923 5691-EI2+EHF CC Upgrade from V4 EEF (5693-EHF)
0923 5691-EI2+5691-EHF from V4 EEF (5691-EHF)

Withdrawn Migration Paths

Effective October 29, 2002, the following migration paths are withdrawn:

0055 5693-WSF from V4 WF3+SUD
0021 5691-WSF from V4 WF3+SUD

0056 5693-WSF from V4 SUD
0022 5691-WSF from V4 SUD

0057 5693-PDG+WSF Upgrade from V4
WF3+DYS+SOE+PA3+SUD
0058 5693-PDG+WSF Upgrade from V4
WF3+DYS+SOE+PA3+SUD

0024 5691-PDG+WSF Upgrade from V4
WF3+DYS+SOE+PA3+SUD
0024 5691-PDG+WSF Upgrade from V4
WF3+DYS+SOE+PA3+SUD

0084 5693-PMG Upgrade from V4 PMP+TSA
0084 5691-PMG Upgrade from V4 PMP+TSA

0801 5693-SMG Upgrade from V4 SMP+TSA
0801 5691-SMG Upgrade from V4 SMP+TSA

Non-Billable Indicator Feature Numbers: P2 add-on products (AOPs) are available on P1 configurations and P1, P2 and P3 AOPs are available on P3 configurations. No-charge indicator codes are set up under each of the 5691 configurations for which the AOP is available. New AOP indicator codes for each configuration are listed below. These codes are for use worldwide and must be key-enabled.

5691-MD1

4316 5693-FR1 AOP Indicator Code
4317 5693-GS1 AOP Indicator Code
4319 5693-BK2 AOP Indicator Code
4314 5693-EL1 AOP Indicator Code
4313 5693-EH1 AOP Indicator Code
4320 5693-EHF AOP Indicator Code
4322 5693-HGR AOP Indicator Code
4323 5693-MPA AOP Indicator Code
4315 5693-EW1 AOP Indicator Code
4318 5693-PX1 AOP Indicator Code
4321 5693-EWE AOP Indicator Code

5691-ME1

4316 5693-FR1 AOP Indicator Code
4317 5693-GS1 AOP Indicator Code
4319 5693-BK2 AOP Indicator Code
4314 5693-EL1 AOP Indicator Code
4313 5693-EH1 AOP Indicator Code
4320 5693-EHF AOP Indicator Code
4322 5693-HGR AOP Indicator Code
4323 5693-MPA AOP Indicator Code
4315 5693-EW1 AOP Indicator Code
4318 5693-PX1 AOP Indicator Code
4321 5693-EWE AOP Indicator Code

5691-SD1

4316 5693-FR1 AOP Indicator Code
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4289 5693-QSR AOP Indicator Code
4265 5693-KT1 AOP Indicator Code
4216 5693-SP1 AOP Indicator Code
4248 5693-KWA AOP Indicator Code
4249 5693-KWE AOP Indicator Code
4263 5693-PEO AOP Indicator Code
4267 5693-PKT AOP Indicator Code
4266 5693-PFD AOP Indicator Code
4264 5693-PFO AOP Indicator Code
4319 5693-BK2 AOP Indicator Code
4214 5693-DMN AOP Indicator Code
4217 5693-SPA AOP Indicator Code
4211 5693-FIT AOP Indicator Code
4212 5693-KIN AOP Indicator Code
4215 5693-DMO AOP Indicator Code
4210 5693-ANR AOP Indicator Code
4301 5693-SPE AOP Indicator Code
4242 5693-HBR AOP Indicator Code
4243 5693-HME AOP Indicator Code
4244 5693-HPA AOP Indicator Code
4241 5693-HAA AOP Indicator Code
4314 5693-EL1 AOP Indicator Code
4313 5693-EH1 AOP Indicator Code
4208 5693-CBD AOP Indicator Code
4278 5693-SRT AOP Indicator Code
4276 5693-SSR AOP Indicator Code
4221 5693-EFD AOP Indicator Code
4223 5693-ELB AOP Indicator Code
4222 5693-EWR AOP Indicator Code
4220 5693-EHI AOP Indicator Code
4320 5693-EHF AOP Indicator Code
4219 5693-ELD AOP Indicator Code
4297 5693-ECR AOP Indicator Code
4259 5693-PID AOP Indicator Code
4245 5693-HVD AOP Indicator Code
4277 5693-SDI AOP Indicator Code
4258 5693-PIP AOP Indicator Code
4280 5693-TUB AOP Indicator Code
4226 5693-EQT AOP Indicator Code
4225 5693-ESS AOP Indicator Code

4291 5693-HVA AOP Indicator Code
4292 5693-SPL AOP Indicator Code
4293 5693-SFD AOP Indicator Code
4302 5693-WGD AOP Indicator Code
4303 5693-TUD AOP Indicator Code
4304 5693-WAV AOP Indicator Code
4312 5693-RCD AOP Indicator Code
4322 5693-HGR AOP Indicator Code
4260 5693-PLO AOP Indicator Code
4237 5693-GP1 AOP Indicator Code
4238 5693-GPS AOP Indicator Code
4236 5693-GAS AOP Indicator Code
4224 5693-EST AOP Indicator Code
4227 5693-FMS AOP Indicator Code
4250 5693-LG1 AOP Indicator Code
4261 5693-PG1 AOP Indicator Code
4283 5693-TL1 AOP Indicator Code
4285 5693-NG1 AOP Indicator Code
4251 5693-LMG AOP Indicator Code
4262 5693-PMG AOP Indicator Code
4201 5693-SMG AOP Indicator Code
4254 5693-MMG AOP Indicator Code
4294 5693-AMG AOP Indicator Code
4295 5693-STL AOP Indicator Code
4296 5693-NCG AOP Indicator Code
4273 5693-NVG AOP Indicator Code
4323 5693-MPA AOP Indicator Code
4272 5693-ST1 AOP Indicator Code
4315 5693-EW1 AOP Indicator Code
4318 5693-PX1 AOP Indicator Code
4274 5693-STC AOP Indicator Code
4321 5693-EWE AOP Indicator Code
4206 5693-CCD AOP Indicator Code

5691-KD3

4204 5693-AS1 AOP Indicator Code
4298 5693-WD1 AOP Indicator Code
4255 5693-PD1 AOP Indicator Code
4316 5693-FR1 AOP Indicator Code
4281 5693-WS1 AOP Indicator Code
4233 5693-GD1 AOP Indicator Code
4247 5693-ID1 AOP Indicator Code
4202 5693-FT1 AOP Indicator Code
4252 5693-MT1 AOP Indicator Code
4299 5693-HA1 AOP Indicator Code
4269 5693-SM1 AOP Indicator Code
4271 5693-SH1 AOP Indicator Code
4205 5693-ASD AOP Indicator Code
4256 5693-PDG AOP Indicator Code
4234 5693-GDR AOP Indicator Code
4203 5693-FTA AOP Indicator Code
4275 5693-STD AOP Indicator Code
4253 5693-MTD AOP Indicator Code
4287 5693-CCV AOP Indicator Code
4270 5693-SMD AOP Indicator Code
4268 5693-RT1 AOP Indicator Code
4317 5693-GS1 AOP Indicator Code
4230 5693-FS1 AOP Indicator Code
4288 5693-RTR AOP Indicator Code
4257 5693-PHS AOP Indicator Code
4300 5693-PSO AOP Indicator Code
4239 5693-GSD AOP Indicator Code
4240 5693-GSO AOP Indicator Code
4231 5693-FSS AOP Indicator Code
4228 5693-FSO AOP Indicator Code
4229 5693-FSP AOP Indicator Code
4232 5693-FSK AOP Indicator Code
4209 5693-DSE AOP Indicator Code
4289 5693-QSR AOP Indicator Code
4216 5693-SP1 AOP Indicator Code
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4212 5693-KIN AOP Indicator Code
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4242 5693-HBR AOP Indicator Code
4243 5693-HME AOP Indicator Code
4244 5693-HPA AOP Indicator Code
4241 5693-HAA AOP Indicator Code
4314 5693-EL1 AOP Indicator Code
4313 5693-EH1 AOP Indicator Code
4208 5693-CBD AOP Indicator Code
4278 5693-SRT AOP Indicator Code
4276 5693-SSR AOP Indicator Code
4221 5693-EFD AOP Indicator Code
4223 5693-ELB AOP Indicator Code
4222 5693-EWR AOP Indicator Code
4220 5693-EHI AOP Indicator Code
4320 5693-EHF AOP Indicator Code
4219 5693-ELD AOP Indicator Code
4297 5693-ECR AOP Indicator Code
4259 5693-PID AOP Indicator Code
4245 5693-HVD AOP Indicator Code
4277 5693-SDI AOP Indicator Code
4258 5693-PIP AOP Indicator Code
4280 5693-TUB AOP Indicator Code
4226 5693-EQT AOP Indicator Code
4225 5693-ESS AOP Indicator Code
4291 5693-HVA AOP Indicator Code
4292 5693-SPL AOP Indicator Code
4293 5693-SFD AOP Indicator Code
4302 5693-WGD AOP Indicator Code
4303 5693-TUD AOP Indicator Code
4304 5693-WAV AOP Indicator Code
4312 5693-RCD AOP Indicator Code
4322 5693-HGR AOP Indicator Code
4260 5693-PLO AOP Indicator Code
4237 5693-GP1 AOP Indicator Code
4238 5693-GPS AOP Indicator Code
4236 5693-GAS AOP Indicator Code
4224 5693-EST AOP Indicator Code
4227 5693-FMS AOP Indicator Code
4250 5693-LG1 AOP Indicator Code
4261 5693-PG1 AOP Indicator Code
4283 5693-TL1 AOP Indicator Code
4285 5693-NG1 AOP Indicator Code
4251 5693-LMG AOP Indicator Code
4262 5693-PMG AOP Indicator Code
4201 5693-SMG AOP Indicator Code
4254 5693-MMG AOP Indicator Code
4294 5693-AMG AOP Indicator Code
4295 5693-STL AOP Indicator Code
4296 5693-NCG AOP Indicator Code
4273 5693-NVG AOP Indicator Code
4323 5693-MPA AOP Indicator Code
4207 5693-CC1 AOP Indicator Code
4246 5693-IG1 AOP Indicator Code
4272 5693-ST1 AOP Indicator Code
4315 5693-EW1 AOP Indicator Code
4318 5693-PX1 AOP Indicator Code
4274 5693-STC AOP Indicator Code
4321 5693-EWE AOP Indicator Code
4206 5693-CCD AOP Indicator Code

5691-TA3

4316 5693-FR1 AOP Indicator Code
4317 5693-GS1 AOP Indicator Code
4319 5693-BK2 AOP Indicator Code
4314 5693-EL1 AOP Indicator Code
4313 5693-EH1 AOP Indicator Code
4320 5693-EHF AOP Indicator Code
4322 5693-HGR AOP Indicator Code

4323 5693-MPA AOP Indicator Code
 4315 5693-EW1 AOP Indicator Code
 4318 5693-PX1 AOP Indicator Code
 4321 5693-EWE AOP Indicator Code

System Program Order

5691-ADD

4316 5693-FR1 AOP Indicator Code
 4319 5693-BK2 AOP Indicator Code
 4314 5693-EL1 AOP Indicator Code
 4313 5693-EH1 AOP Indicator Code
 4320 5693-EHF AOP Indicator Code
 4322 5693-HGR AOP Indicator Code
 4323 5693-MPA AOP Indicator Code
 4315 5693-EW1 AOP Indicator Code
 4318 5693-PX1 AOP Indicator Code
 4321 5693-EWE AOP Indicator Code

5691-SBD

4316 5693-FR1 AOP Indicator Code
 4317 5693-GS1 AOP Indicator Code
 4319 5693-BK2 AOP Indicator Code
 4314 5693-EL1 AOP Indicator Code
 4313 5693-EH1 AOP Indicator Code
 4320 5693-EHF AOP Indicator Code
 4322 5693-HGR AOP Indicator Code
 4323 5693-MPA AOP Indicator Code
 4315 5693-EW1 AOP Indicator Code
 4318 5693-PX1 AOP Indicator Code
 4321 5693-EWE AOP Indicator Code

Configuring Orders

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

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

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.

Media Feature Number

Environment	Feature Number	Distribution Medium
All Platforms	3410	CD-ROM

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

5628-ACN System Program Order

CATIA V5 programs for all platforms (UNIX and Microsoft Windows) will be shipped under the 5628-ACN System Program Order (SPO). 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. The SPO is designated to the processor on which the program type 5691 is licensed. Each customer number must have its own SPO.

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 Licenses: 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 Program Feature Numbers for New Products: The following no-charge program feature numbers are used with the 5628-ACN 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 programs.

Program Feature Number	Configuration or Product
------------------------	--------------------------

New Configurations

2345	5691-YM1 SPO
2348	5691-YM2 SPO
2349	5691-EI2 SPO
2350	5691-AB3 SPO
2351	5691-KD3 SPO

New Shareable Products

2352	5691-BK2 SPO
2354	5691-EHF SPO
2355	5691-HGR SPO
2357	5691-MPA SPO
2359	5691-EWE SPO
2361	5691-FR1 SPO
2297	5691-PDL SPO
2366	5691-PX1 SPO
2368	5691-WS1 SPO

Program Feature Number	Configuration or Product
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New Add-on Products (AOP)

2353	5693-BK2 SPO
2356	5693-HGR SPO
2358	5693-MPA SPO
2360	5693-EWE SPO
2362	5693-FR1 SPO
2363	5693-EL1 SPO
2364	5693-EH1 SPO
2365	5693-EW1 SPO
2367	5693-PX1 SPO
2369	5693-GS1 SPO
2370	5693-EHF SPO

5628-ACN Program Feature Numbers for Withdrawn Products: Effective October 29, 2002, the program feature numbers for the following products are withdrawn:

Program Feature Number	Configuration or Product
1931	TD1 CATIA —Team PDM 1 Product
1992	5691-TD1 CATIA —Team PDM 1 Product for U.S. English
2296	5693-TD1 CATIA —Team PDM 1 AOP for U.S. English
2343	5691-TD1 CATIA —Team PDM 1 for Japanese
2344	5693-TD1 CATIA —Team PDM 1 AOP for Japanese
2298	5691-DPL MultiCax P Plug-in Product
1924	5691-WSF CATIA —Wireframe & Surface 2 Product
2279	5693-WSF CATIA —Wireframe & Surface 2 AOP
1834	5691-ECD CATIA —Electrical Cable Definition 2 Product
2217	5693-ECD CATIA —Electrical Cable Definition 2 AOP

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 under the 5628-SPO.

Description	Feature Number
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Update Shipments

Suppress updates (suppresses update shipment of media and documentation)	3482
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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.

Unlicensed Documentation: 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 Softcopy Collection Kit — French	7101
CATIA V5 Softcopy Collection Kit — German	7102
CATIA V5 Softcopy Collection Kit — Japanese	7103
CATIA V5 Softcopy Collecting Kit — Italian	7108

Additional Softcopy Collection Kits can be ordered through the IBM Publication Ordering System:

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

using the following numbers:

- 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)

Displayable Softcopy Publications

CATIA V5 product publications are offered in displayable (HTML) and printable (PDF) softcopy form. All unlicensed manuals are included. These softcopy manuals are part of the basic machine-readable material that automatically ships with an order for a standard or custom configuration. The manuals are shipped on CD-ROM.

Terms and conditions for use of the machine-readable files are shipped with the product.

The displayable manuals are in HTML format and can be viewed in any of the supported environments using one of the HTML browsers specified under **Software Requirements** in the **Technical Information** section. The printable manuals are in PDF format and can be displayed and printed with Adobe Acrobat Reader at minimum level 3.0. The Adobe Acrobat Reader may be downloaded, at no charge, from:

<http://www.adobe.com/prodindex/acrobat/readstep.html>

Additionally, customers with the Enhanced Support Offering can view the latest CATIA V5 publications in HTML form by visiting:

<http://techsupport.services.ibm.com/engineering/support>

and clicking on the **Technical knowledge, data bases, publications** choice.

Licensed Documentation: There are no licensed publications for CATIA V5.

Subsequent updates (technical newsletters or revisions between releases) to the publications shipped with the product will be distributed to the user of record for as long as a license for this software remains in effect. A separate publication order or subscription is not needed. National language editions of CATIA V5's softcopy collection kits are available as follows:

- SK3T-4125, for French
- SK3T-4126, for German
- SK3T-4127, for Japanese
- SK3T-0575, for Italian

Terms and Conditions

Licensing: IBM Customer Agreement (ICA)

Designated Machine: Not required

Variable Charges Apply: No

Installation License or Location License Applies: No

Usage Restriction Applies

Yes, as implemented by the licensing mechanism described.

The Licensing Mechanism

CATIA V5 Licensing on Workstations: In workstation environments, CATIA controls the number of concurrent users of a CATIA configuration or product, according to the number of licenses acquired for the configuration or product.

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

- A CATIA V5 configuration (standard or custom) will require a license. Licenses for CATIA V5 configurations are acquired and released for the total configuration. The products within a configuration cannot be shared.
- CATIA V5 shareable products will require a license, in addition to one for the prerequisite configuration and any prerequisite product, if applicable.

CATIA V5 can be used in two licensing modes, either nodelock or with concurrent usage of licenses on a network.

Nodelock Usage: The use of 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 (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 nodelock mode of operation, only one CATIA license per configuration and product can be registered by machine, and only one CATIA user can run at a time. If multiple licenses per configuration or product, or multiple

users, are required, refer to the **Concurrent Usage** section.

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

CATIA add-on and shareable products require a license for a configuration, which includes at least the prerequisite products.

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

Dynamic Licensing: Beginning with this release, shareable product licenses 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 configuration licenses.) Shareable licenses acquired at the beginning of the session cannot be released before the end of the session; only licenses dynamically granted upon user request during the session can be released during the session.

Note: In this release, the Real Time Rendering 2 Product (5691-RTR) and Real Time Rendering 2 Add-on Product (5693-RTR) do not support dynamic licensing.

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

Contact Buz Nowicki (qnowicki@us.ibm.com) or 248-552-5981 for additional information and approvals.

Volume Discount: Contact your IBM representative.

Warranted: Yes

Licensed Program Materials Availability

- Restricted materials of IBM: None
- Non-restricted source materials: None
- Object code only (OCO): All

Testing Period: None

Program Services: Program services for CATIA V5 and CATIA V5 WLS will be available until discontinued by IBM upon six months' written notice.

Customers may report problems against a given **release** of CATIA V5 for a limited time only. This service period ends four months after the general availability of the second subsequent release of CATIA V5. Therefore, the **end of service** date for CATIA and CATIA WLS V5.8 will be four months after the general availability date of this release. For a list of all currently supported releases of CATIA V5, go to:

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

On the left side, select **Support**. Then select **The PLM Technical Support Web site** and then click on **Release nomenclature and end of service dates** in the **What's new?** column.

If the user has not yet obtained an IBM common registration user ID, one can be obtained at:

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

Program Services offer a method of reporting code-related problems for CATIA and CATIA WLS V5 licensed software products.

The preferred method for technical assistance is electronically through the PLM Web page:

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

On the left side, select **Support**. Then select **The PLM Technical Support Web site**, and then click on the **Service requests, problem reporting** link, and then **Fastpath to Submit a Problem Management Report (PMR)**. When using the PMR option, communications will be either through the PMR or by e-mail, depending on country-specific guidelines.

Note: Not all options of the PLM technical support e-services are available in all countries.

Problems may also be reported by facsimile, e-mail or conventional mail and will be responded to via the same medium used to submit the problem. IBM will respond to customer inquiries within two business days of e-mail or facsimile receipt and seven days of conventional mail receipt advising of the problem disposition.

Country-specific e-mail, facsimile or conventional mail information can be found at the following Web site:

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

On the left side, select **Support**. Then select **The PLM Technical Support Web site** and then click on **Service requests, problem reporting**. On the subsequent frame expand **Contracts, Other resources** and select **IBM PLM Warranty Support contacts as an alternative to this site**.

If the problem reported is not known to be a code-related problem, the customer will be informed that IBM will continue to work on it provided the customer has an enhanced support contract.

Additional information about the enhanced support contract and other available PLM services offerings is available by going to the PLM Web page:

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

On the left side, select **Support**. Then select **The PLM Technical Support Web site**. Then click on **Service and Support offerings**. On the next frame expand **Support Offerings**. Alternatively, contact your IBM PLM marketing representative or authorized IBM Business Partner for more information.

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

Corrective service for CATIA V5 Releases is delivered through service packs on a regular basis. A service pack includes corrections for Severity 1 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 can be installed on top of the released level or on top of a previous service pack. No individual corrections will be delivered between two service packs. No updates to online documentation will be provided through service packs.

Customers may request a correction via a service pack for Severity 1 problems. A Severity 1 problem is defined as:

- A problem that is stopping production — This means the customer is already using the level for which he requests a fix in a production environment.
- A problem that is preventing migration — In this case, the customer must provide the migration plan.
- A problem that is halting testing of a given level — In this case, a fix will allow the customer to continue the testing.
- Installation problem — A problem that prevents the customer from installing or using the product.
- Regression — Problems reported as regressions are due to an operation which 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, and true regressions will be handled as Severity 1 problems.

Requests for corrections to a given release of CATIA V5 may be made up to four months after the next release of CATIA V5 becomes generally available. After that date, no new service packs will be built for that release. This final service pack will still be available for ordering on CD-ROM via the IBM Support Centers for the duration of the service period of each release. (This service period ends four months after the general availability of the second subsequent release of CATIA V5.)

With the availability of this release, the **end of corrective service date** for CATIA and CATIA WLS V5.9 will be four months after the general availability date of this release; the **end of service (EOS) date** for CATIA and CATIA WLS V5.8 will be four months after the general availability date of this release.

For help guides and information on education, go to the CATIA/ENOVIA TechSupport Web page at:

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

On the left side, click on **Education**.

IBM Operational Support Services — Support Line: No

Other Support: PLM Support Center

Prices

Contact your IBM representative for pricing information for this announcement.

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