CATIA V5R12 - FACT SHEET

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WHAT'S NEW AT A GLANCE

CATIA V5R12:
- **delivers a breakthrough in product design** through the integration of the revolutionary new functional modeling approach. This approach further demonstrates that "CAD is not a commodity". It is embedded in the new Functional Molded Parts 1 product, based on semantic features that capture design intent. It enables plastic part designers in the Electronics & Electrical and Consumer Goods industries to concentrate on what they want to model and not how to model it. This new paradigm provides significant productivity gains as well as all the benefits of being fully integrated in the CATIA V5 product design solution. Its availability, one year after the partnership announcement with pioneer ImpactXoft Inc., is a clear example of the power and the flexibility of the V5 architecture.
- **provides full-scale coverage of composite parts design** (functional, logical, detailed) enabling greater use of composite materials in customer products.
- **enhances the design and management of electrical equipment** with significant improvements such as full associativity between 3D design and harness documentation that provides users with a highly efficient 3D harness design solution.
- **delivers critical productivity gains in machining** including an important breakthrough for turbo-machinery parts through enhancements such as a new "5-Axis helix machining" function.
- **introduces two new products for the Shipbuilding Industry** dedicated to compartment and access objects definition, and heavy structures detailed design.
- **expands capabilities for the overall body process** including new tools to design advanced shapes.
- **strengthens the Entry PLM offer** with improved design for mechanically-welded products and sheetmetal parts, a new Develop Shapes 1 product for quick surface development, dramatic reduction in computation time for large assembly drawing extraction (up to 100 times in certain cases), and tighter CATIA-SMARTEAM integration.
- **enhances engineering change management and teamwork** along the product-process definition via the ability to use the publication

mechanism in Design-to-Manufacturing and Design-to-Analysis processes.

- **further develops the CATIA-ENOVIA engineering hub** to strengthen concurrent engineering best practices, and manage catalogs of features and equipment and system components.
- **reduces memory consumption up to 60%** through the extension of automatic selective loading of relevant product data from ENOVIA.
- **extends the reach of CATIA V5** with the addition of 55 new highly-specialized partner V5 applications since V5R11.

CATIA V5R12 DETAILED DESCRIPTION

Value to the Customer - The CATIA V5R12 products and features listed below are grouped according to Dassault Systèmes and IBM's PLM Fundamentals. The fundamentals are the core elements of the PLM business strategy that bring value to a customer.

In the **Process-Centric** fundamental category (basing customer solutions on industry-specific processes), CATIA V5R12:

- further strengthens and transforms business processes of targeted industries by turning experience into advantage:
  - o V5R12 provides full-scale coverage of functional, logical and detailed design for composite parts enabling customers to extend the use of composite materials within their products. In particular, the manufacturability of composite parts, designed through *CATIA - Composites Design 3 (CPD)* is enriched. More manufacturing constraints are taken into account earlier in the design stage, avoiding problems such as wrinkling, bridges of fiber, and shrinkage of material following the firing operation.
  - o V5R12 provides major enhancements for the design and management of electrical equipment, such as full associativity between 3D design and harness documentation that provides users with a highly efficient 3D harness design solution.
  - o V5R12 also delivers critical productivity gains across the machining process. The *CATIA - Advanced Machining 2 (AMG)* product delivers an important breakthrough for turbo-machinery parts: the new '5-Axis helix machining' operation allows creation a single helical path machining operation for turbine blade milling. *CATIA – Prismatic Machining 2 (PMG)* introduces an opposite hand machining process for symmetrical machining, supporting all axial machining operations and allowing the whole machining program to be quickly and easily defined. *CATIA - 3-Axis Surface Machining 2 (SMG)* introduces a new roughing operation adapted to hard material that is especially useful for mold and dies makers.

- V5R12 introduces two new products for the Shipbuilding Industry dedicated to compartment and access objects definition, and heavy structures design. These products consolidate ship definition process coverage. *CATIA - Compartment and Access 2 (CNA)* is dedicated to the definition of compartment and access objects within ship design. In particular, it provides users with the ability to define wall systems/non-structural bulkheads, extensive compartment boundaries, placement of closures, stairs, ladders, report generation, query and various analyses. *CATIA - Structure Detailed Design 2 (SDD*) completes design of heavy structures with realistic plating and stiffening elements and provides industry-specific features to create hull structure detailed parts.
  - V5R12 provides additional capabilities for the overall body process. This includes new tools to design advanced shapes within *CATIA - Generative Shape Design 2 (GSD)* such as "*3D curve offset"* to easily generate joint planes for mold and die makers, "*remove twisted area"* of sweep curve for manufacturing draft surfaces and "*multi-ribbon shape fillet"*. The production of best quality Class A shapes is promoted within *Automotive Class A 3 (ACA)* with both the automatic filleting of complex shapes and the creation of flanges at a given set of curves. In addition, *CATIA - Freestyle Shaper 2 (FSS)* delivers new high-end associative analysis tools: the "*Zebra Shading"* feature enables users to check a model's quality in detail, and the "*Highlight Analysis"* feature enables users to display curve highlights at the end of the analysis process. *CATIA - Freestyle Profiler 2 (FSP)* enables users to better respect guide curves segmentation and topology through "*Net quality"* enhancement. In the Body in White process, the modeling of fasteners within *CATIA - Automotive Body-In-White Fastening 3 (ABF) is* enriched with new types of fasteners including curve bead and spot projection welds.
  - V5R12 significantly reinforces product analysis and simulation. The accuracy of simulating car body and product structure behavior is increased through the support of new advanced meshing connections, including compatible spot connections with beam and hexahedron elements. In addition, V5R12 delivers critical time gains for large-model computation through both a new solving-renumbering method and the extension of external solving to all kernel steps within *CATIA - Elfini Structural Analysis 2 (EST)*. Finally, a 2D-to-3D extrusion tool allows users to mesh solid parts with hexahedron elements through *CATIA - FEM Solid 2 (FMD).*
- expands benefits to small and medium-sized businesses (SMB) through the unique value of the Entry PLM solution:
  - V5R12 further extends the intuitive and productive design of sheet metal parts within *CATIA - Sheetmetal Design 1 (SM1).* In

the mechanical design process, *CATIA - Weld Design 1 (WD1)* extends welding capabilities and standards consistency with: seven double-weld types added to cover specific welding processes; the ability to weld parts of different heights; and the creation of normalized welds (shape, name, parameters) according to ISO standards.

- o V5R12 delivers a new product *CATIA - Develop Shapes 1 (DL1)* that enables users to both easily develop curves on a revolution surface and to unfold a ruled surface. Working with flat surfaces, user can more easily manage bulk calculations, raw material costs calculation, manufacturability calculation, etc.
- o V5R12 strengthens the CATIA V5 Drafting solution across all industries with increased productivity and usability of drafting tools. This includes, in particular, the support of new dimensions (dimension along any reference direction, curvilinear total length dimension, and technological dimension), the automatic recognition of intersection points for rapid dimensioning, and multi-tolerance with associative numerical value automatically computed by CATIA V5. A new algorithm (Approximate mode) for drawing generation on CGR mode is also added within *CATIA - Generative Drafting 1 (GD1)* enabling a dramatic reduction in computation time for large assembly drawing extraction (up to 100 times shorter).
- o Finally, CATIA-SMARTEAM collaboration is strengthened through the extended capability of applicative resources management within SMARTEAM. This contributes to tighter integration with Plant Layout, and Equipment and System applications.

In the **Collaborative Workspaces** fundamental category (delivering a shared, real-time, 3D working environment), CATIA V5R12:

- enhances engineering change management and teamwork via the ability to use the publication mechanism in Design-to-Manufacturing and Design-to-Analysis processes.
- improves the generation of high quality photo-realistic images with the new "*Fresnel Shader*" application for realistic glass-rendering enabling greater collaboration between design and marketing teams.
- simplifies systems design and process development through improvements in harness data management that enable users to store electrical connectivity XML files in ENOVIA LCA.
- further increases the ability for post-modification on isolated solids coming from external systems or standard formats through new capabilities in feature recognition, such as drafts and volumes recognition. In addition, V5R12 delivers extended capabilities in mechanical parts tolerant modeling, such as automatically removing twisted areas during sweep operations and an edge fillet ability to roll on sharp edges. CATIA V5R12 also delivers a completed migration report that reinforces V4 transition to V5.

- extends CATIA-SMARTEAM integration by enabling the CATIA user to be automatically connected to SMARTEAM when launching CATIA, and to the SMARTEAM object status display through dedicated icons directly in the drawing feature tree.

In the **Product, Process, Resource Model (PPR)** fundamental category (integrating product development, manufacturing processes, and resources) CATIA V5R12:

- further develops the CATIA-ENOVIA engineering hub to strengthen concurrent engineering best practices, and manage catalogs of features and equipment and system components.
- reduces memory consumption up to 60% through the extension of automatic selective loading of relevant product data from ENOVIA.
- dramatically improves productivity for prismatic machining tasks. When a milling machine is described in DELMIA, a new capability in the CATIA Prismatic Machining application automatically generates the transition path between two machining operations.

In the **Knowledge** fundamental category (capturing, sharing, and reusing information) CATIA V5R12:

- includes the revolutionary new functional modeling approach. This approach further demonstrates that "CAD is not a commodity". It is embedded in the new Functional Molded Parts 1 (FM1) product. FM1 is based on semantic features that capture design intent. It enables plastic part designers in the Electronics & Electrical and Consumer Goods industries to concentrate on what they want to model and not how to model it. This new paradigm provides significant productivity gains as well as all the benefits of being fully integrated in the CATIA V5 product design solution. The availability of FM1 one year after the partnership announcement with functional modeling pioneer ImpactXoft Inc., is a clear example of the power and the flexibility of the V5 architecture.
- permits finer tuning of large multi-constraint optimization for more accurate and easy-to-analyze solutions. Users can prioritize the constraints applied to a system, allowing greater control of constraint resolution and optimization of over-constrained systems. This enables better visualization of results and more accurate analysis.
- introduces new automation enhancements that perfect morphing capabilities. These include:
  o the ability to plug in a behavior at the end of an assembly template instantiation,
  o the quick access to formulas in 3D,
  o the ability to instantiate a user-defined feature from a VisualBasic script,
  o the ability to select inputs in a panel (instead of write VB code) and integrate a loop in PowerCopies.

In the **CAA V5 - Component Application Architecture** fundamental category (encouraging development of new PLM solutions by independent software vendors), CATIA V5R12:

- extends the reach of CATIA V5 with the addition of 55 new highly-specialized partner V5 applications since V5R11.

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