CATIA V5R17 - FACT SHEET

CATIA V5R17 boosts Innovation for Product Excellence

- **Introduction**
- **What's New at a Glance**
- **Overview**
- **Detailed Description**

INTRODUCTION

CATIA V5 is the leading solution for product excellence. It addresses all manufacturing organizations, from OEMs, through their supply chains, to small independent companies. The range of CATIA V5's capabilities allows for its application in a wide variety of industries, from aerospace, automotive, industrial machinery, electrical, electronics, shipbuilding, plant design, and consumer goods, to jewelry and clothing.
CATIA V5 is the only solution that covers the complete product development process, from product concept specifications through to product-in-service, in a fully integrated manner. Based on an open, scalable architecture, it facilitates true collaborative engineering across the multidisciplinary extended enterprise, including style and form design, mechanical design, equipment and systems engineering, digital mock-up management, machining, analysis, and simulation. By enabling enterprises to reuse product design knowledge and accelerate development cycles, CATIA V5 helps companies speed-up their response to market needs.
In conjunction with ENOVIA for collaborative product lifecycle management, SIMULIA for engineering quality and DELMIA for production performance, CATIA V5 is a key component of V5 PLM.

WHAT'S NEW AT A GLANCE

CATIA V5R17 allows you to:

- **Extend end-to-end industry process coverage** through major enhancements in 3D harness flattening, and the efficiency of CATIA Machining NC programming and simulation.
- **Rapidly explore design ideas** with new core styling enhancements.
- **Dramatically improve design productivity** with breakthrough auto-filleting.
- **Improve your ability to work concurrently** across the extended enterprise through key enhancements in the VPM Navigator and in the reconciliation mechanisms of ENOVIA SmarTeam.
- **Maximize collaboration** with 3D XML, to share information beyond the design office.

OVERVIEW:

CATIA V5R17's products and features listed below bring value to the customer reinforcing Dassault Systèmes' and IBM's CATIA fundamentals:

Integrated portfolio supports seamless industry processes

- CATIA V5R17 **strengthens the integration between 3D electrical harness design and documentation**, reducing the design-to-manufacturing cycle time for the automotive and aerospace industries.
- The release **increases CATIA Machining NC programming and simulation efficiency** to significantly reduce overall manufacturing process time.

High-performance design with knowledge capture and reuse encourages innovation

- CATIA V5R17 **facilitates design innovation** with core styling capabilities that enable designers to explore product ideas more rapidly, while permitting last-minute styling modifications.
- CATIA V5R17 **extends the 3D master approach** by enabling fast and convenient display of product information, such as tolerances and annotations, in a familiar drawing layout within the 3D environment.

Breakthrough technologies speed time-to-market

- CATIA V5R17 delivers a unique auto-filleting capability that enables automotive powertrain and chassis designers to **reduce the time required to fillet complex parts from hours or days to just minutes.**
- This release accelerates the adoption of **CATIA V5's breakthrough functional modeling** technology for the automotive powertrain process.

V5 PLM integration maximizes collaboration

- The V5R17 VPM Navigator **boosts the use of relational design** by making it easier for designers to work concurrently on different part versions, and to manage advanced product effectivity and configurations.
- Key enhancements in the reconciliation mechanisms of ENOVIA SmarTeam, such as advanced queries, **facilitate interaction between companies and concurrent engineering across the supply chain**.
- **A richer 3D XML format** now includes finite element analysis output to widen the scope for the use of this lightweight format and makes it easier to reuse and share 3D outside the design office.

Open and scalable architecture increases flexibility

- **Key PLM market players are adopting V5 architecture**. Additional partner V5 applications launched since V5R16 extend the process coverage of V5 solutions with highly specialized applications that cover many disciplines, such as sheetmetal design-to-manufacturing and digital mock-up (DMU) management.

DETAILED DESCRIPTION

Integrated portfolio supports seamless industry processes

CATIA V5R17 increases the value of the CATIA-Electrical Harness Flattening product for harness drawing production by enabling the reuse of existing models, enhancing productivity for faster layout editing, and improving harness drafting quality with automatically generated dress-up. The flattening process becomes more robust and completes the layout process more efficiently. The enhanced CATIA - Electrical Harness Design offering reduces design-to-manufacturing cycle time, especially for CATIA V5 automotive and aerospace electrical design customers.

V5R17 increases CATIA's machining programming and simulation efficiency, significantly reducing overall manufacturing process time.

- It offers three new operations which appreciably enhance NC programming and reduce machining time.
    - For hard material machining, V5R17 introduces a comprehensive operation for managing plunge milling. Developed with tool makers, it features a process-oriented tool path style and takes into account residual stock for a fully collision-free tool path. In addition, V5R17 adds the capability to automatically detect full-diameter engagement situations and manages tool overload (reducing the feed rate, adding extra machining planes, adding trochoidal paths) to greatly improve machining time and cutting tool life.
    - A new 4-axis sweeping operation machines parts with a sweeping tool path style, and a tool axis driven by a 2D curve.
    - Finally, V5R17 provides the NC programmer with a complete operation to support the machining of tubes and other parts that present an obvious central axis or curve, leading to major productivity gains in NC programming for these parts.
- V5R17 improves on the breakthrough technology of realistic integrated machine simulation capability introduced with V5R16. Simultaneous, synchronized

simulation of material removal and machine motion enables the realistic simulation of the entire machining environment, including collision detection between machine parts and in-process stock. The simulation can use NC code generated inside or outside of CATIA V5, enabling a final virtual buy-off of the NC program. Additionally, the user can now analyze machined stock during an integrated material simulation by accessing the 'Analyze', 'Video Measure' and 'Remove Chunks' commands, to validate the accuracy of the machined part.

V5R17 widens the scope of the end-to-end collaborative composites solution to reduce overall design-to-manufacturing time.

- An automatic mirrored-part generation function enables the designer to create a fully-associative mirror-image of a part, such as a wing, with a single click, saving design time.
- The robust composites analysis and simulation tools (core sampling, nonstructural ply management, fiber direction indication) are improved, easily managing more complex geometries, and enabling the designer to validate the design early in the process.
- In addition, V5R17 offers an interactive way to easily migrate CATIA V4 models (materials, geometry, data structure) to CATIA V5 and enables the user to define zones where a splice or an overlap is not allowed, for example, for structural reasons. The cut pieces will be automatically generated according to these specifications.
- The release improves composites manufacturing export capabilities for nesting and cutting processes and offers a new automatic ply book generation capability. This associative document enables the manufacturing operator to easily and accurately locate the ply on the mold, to avoid incorrect draping. The book includes all the information required to perform the process, such as 2D geometries, ply manufacturing annotations, ply group, sequence and material names, and ply thickness.

In conjunction with DELMIA, CATIA V5R17 reinforces the entry-level end-to-end shipbuilding process, performing the manufacturing preparation of heavy structures directly on 3D design data in a semi-automatic way.

V5R17 greatly improves CATIA V5's core and cavity design, especially for the stamping die process.

- The CATIA - Core & Cavity Design product uses rough offset technology to offer a new 'light surface'. The die designer can substitute this light surface for the original during the die design phase, providing major capacity and performance gains on ultra large designs. In addition, designers can rapidly and easily identify and modify the fillet radii of the stamping dies prior to milling, leading to significant cycle-time reduction. The enhanced associativity between the parting line and the design contribute to improved productivity.
- Finally, V5R17 enhances mold tooling design efficiency and intuitiveness and facilitates the classification of tooling elements in the specification tree by the designer. At the same time, designers can more easily manage the component replacement process and its possible impact.

V5R17 delivers major productivity gains to sheetmetal designers on complex designs. Users can define a wall-on-edge in the assembly context and intuitively define most of the attributes of the wall-on-edge and bend properties directly in 3D. Combined with the sketch-based wall definition, these enhancements significantly reduce design time. The ability to create several corner reliefs in a single operation contributes to the unmatched efficiency of CATIA V5 sheetmetal design. V5R17 simplifies the complex sheetmetal part prototyping and strip layout processes with the ability to locally fold or unfold bends with stamping features, making the design fully compliant with manufacturing processes.

V5R17 extends the reach of CATIA aerospace sheetmetal design by enabling it to handle more complex designs. Sheetmetal designers can select a ruled surface for the web and can design more complex joggles. In addition, V5R17 automates the design validation and compliance with company standards, thanks to knowledge rules or checks based on characteristics curves, or joggle parameters based on design tables.

CATIA V5R17 delivers high-end surface quality with the CATIA V5 reverse engineering portfolio. Dedicated to surface reconstruction, the new Deviation Analysis command enables the designer to control the accuracy of generated surfaces. With advanced options, it performs distance analysis of a shape according to reference elements.

CATIA V5R17 ensures the manufacturability of non-ruled surface shapes. This release embeds a new breakthrough technology to support complex non-ruled surface flattening operations, enabling accurate manufacturing preparation of any designed shape, from rough fabrics in the textile industry to raw materials in aerospace.

High performance design with knowledge capture and reuse encourages innovation

CATIA V5R17 facilitates design innovation with core styling capabilities that enable designers to explore product ideas more rapidly, while permitting last-minute styling modifications.

- Unmatched offset technology offers unlimited complex styled surface creation, even for sharp-edged shapes.
- Powerful symmetry technology helps designers achieve major styling productivity gains, while conserving G2 continuity.
- Linking subdivision elements enables the user to perform accurate global deformation on the components of several models in a single operation.
- Integrating the advanced operators of Generative Shape Design optimizes the styling-to-design workflow.

CATIA V5R17 extends the unique 2D/3D associative approach for conceptual design and extends the 3D master approach by enabling the fast and convenient display of product information, such as tolerances and annotations, in a familiar drawing layout within the 3D environment. It boosts conceptual design within the 3D environment by enabling designers to easily create in-context 2D sketches from the automatic detection of existing 3D geometry displayed in the view background. Efficient, comprehensive, and standards-compliant CATIA V5 drafting capabilities always guarantee high quality when realizing drawing layouts and dress-up, whether immersed in 3D or in a separate drawing document. It is continually enhanced and updated to meet specialized needs, such as support for new customized symbols, strokes, and open-type fonts.

CATIA V5R17 promotes 3D as the master reference for part and product definition. Designers can define and manage standards-compliant tolerance specifications and annotations linked to the 3D geometry, making them directly reusable for manufacturing planners and to be shared throughout the enterprise. Designers can easily present and share 3D tolerancing and annotation in a familiar drawing layout embedded in the 3D environment. V5R17 enables the rapid creation of associative views from Functional Tolerancing & Annotation views or captures. Users benefit from a more productive 3D annotation definition and layout process by realizing specific operations in a single step, such as directly managing the view ratio property for 3D annotation.

Breakthrough technologies speed time-to-market

CATIA V5R17 includes new cutting edge solutions to speed time to market. Unique in the marketplace, it allows the creation of cast and forged fillets in a single operation rather than repetitive, step-by-step manual operations. Only CATIA V5 auto-filleting enables automotive powertrain and chassis designers to reduce the time for filleting complex parts for manufacturing from hours or days to just minutes.

This release enhances CATIA automotive powertrain solutions with a suite of unique software. Mechanical design applications, especially Functional Modeling with process-specific behaviors encapsulated in its features, allow designers to develop parts with a higher level of productivity and flexibility. CATIA V5 considerably reduces the costly and time-consuming creation and modification of the design.

In the context of hybrid shape creation with components coming from heterogeneous environments, for example in the body design process, CATIA V5R17 offers a powerful geometrical visualization option that enables engineers to perform accurate analysis of surface connections.

V5 PLM integration maximizes collaboration

CATIA V5R17, with the ENOVIA VPM Navigator, boosts the use of relational design by making it easier for designers to work concurrently on different part versions and to manage advanced product effectivity and configurations. Designers can easily develop their own studies, relying on the approved version of parts made by their co-workers, when ‘in-work’ versions exist concurrently. V5R17 enables designers to synchronize the links of their designs to other parts, and to any versions of these parts. Enabling designers to explore more allows them to better anticipate changes, resulting in a shorter lead-time for manufacturing, and more innovative products.

CATIA V5R17 broadens the scope of VPM Navigator for configuration management, with enhanced capabilities for the designer to filter and analyze configured products. Filters using a combination of date, range, milestones, or specifications can be defined, saved and modified. In addition, modification filters can be added to these configuration filters.

CATIA V5R17 strengthens navigation and enhances user interface customization to search and evaluate product and change impacts, improving ease of use and the ability to interpret relationships and changes. It provides a new software openness, making it easier to define the display names of ENOVIA VPLM documents for all generic CATIA

V5 menus and windows, as well as customized icons for ENOVIA entities displayed in the VPM Navigator tree. In the impact graph, naming of nodes can now be managed through boxes with multiple lines for better layout and enhanced visibility.

Key enhancements in reconciliation mechanisms of ENOVIA SmarTeam facilitate interaction between companies and concurrent engineering across the supply chain. These include enhanced reporting, advanced queries and visualization, batch support, improved ergonomics, and tighter synchronization with ENOVIA VPLM.

This release supports efficient collaboration with a richer 3D XML format throughout the product development cycle.

- Display of finite element analysis (FEA) output facilitates the reuse and sharing of 3D beyond the design office. In CATIA V5R17, FEA analysis output is embedded in 3D XML format. This enables the analysis project manager or engineering specialist to make the right product development decision without having an FEA seat.
- More realistic viewing of 3D XML products enables efficient communication and marketing for the final product, thanks to rendering material support.
- In V5R17, new automation support enables easy integration of 3D XML player in customized applications, such as web.
- The 3D XML player in V5R17 enables cross highlight management between the BOM and the 3D representation, or integrated multi-viewer representation within the same application.

Open and scalable architecture increases flexibility

Key PLM players are adopting V5 architecture. V5 applications launched by partners since V5R16 extend process coverage of V5 solutions with highly specialized applications including:

- IGE+XAO Group: contribution to full electrical PLM scenarios with SEE Topology V5
- ESI Group: fiber simulation for composites with CAA V5-based PAM-QUIKFORM
- Forming Technologies: CostOptimizer CAA V5 for sheet metal parts processes
- Kineo CAM: a new partner - DMU Path Planner, for improved automatic motion and path planning for DMU Fitting Simulator

New partners, Theorem Solutions and Elysium Co ltd. will deliver native V5 integration of external data (from ProE, UGS, Ideas) within a V5 session. These integrations will leverage dedicated CAA V5 APIs and infrastructure to improve data quality and consistency.

CATIA V5R17 delivers a secure and robust way to integrate data management with CATIA V5, with CATIA – PPR PDM Gateway. The gateway guarantees the accuracy of CATIA V5 behavior and data in a product data management context, thanks to thorough checking mechanisms. Its open development interfaces ensure high-quality integration of product data management applications. In V5R17, the gateway is at the interactive integration level. Check-in and check-out status can be applied and visualized in the CATIA tree structure. Design rules checking secures the data before saving it. The XPDM gateway provides a way to display the links between CATIA documents.