CATIA V6R2009X - FACT SHEET
DESIGN EXCELLENCE

Unique real time rendering with new materials, new effects, and paint shaders for heightened virtual product realism throughout the entire V6 portfolio.

INTRODUCTION

CATIA V6 puts 3D collaborative innovation at the heart of the enterprise and helps accelerate companies' transformation toward a full PLM 2.0 approach, thanks to ground-breaking collaboration tools enabling 3D brainstorming.

AT A GLANCE

CATIA V6 enables the full spectrum of next generation collaborative virtual design, including shape design, mechanical and equipment engineering, as well as systems engineering integrating requirement, functional, logical and physical product definitions.

- Enables advanced systems simulation within the 3D digital mockup,
- Leverage “design anywhere, anytime” ability to capture and send intended design changes to offline collaborators,
- Composites Design brings In-context Composite Design for optimized design and mating with non-composite structural parts.

OVERVIEW:

CATIA V6 - Virtual Design offers, through a multidisciplinary approach, a full spectrum of virtual design capabilities and enables efficient design collaboration to encourage innovation across the extended enterprise.

Global Collaborative Innovation

- Extensions to Dassault Systèmes’ award-winning collaborative 3D environment, allowing an unlimited number of online users to participate with immersive video conference in virtual 3D brainstorming, leveraging the collective intelligence of geographically dispersed teams;
- Asynchronous collaboration to enable people offline to share new features asynchronously as well as in real time, supporting a “design anywhere, build anywhere” strategy.

Lifelike Experience

- Unique real time rendering capabilities with new materials, new effects, and paint shaders for heightened realism of virtual products through the entire V6 portfolio

Single Platform for IP Management

- The availability of knowledgeware beyond the design domain, to all disciplines, allowing capture/capitalize, reusing and enforce/automate companies’ best practices. CATIA V6R2009x delivers 2 new products to support creation and usage of companies processes template and accelerates design automation
- CATIA Virtual Product Management, allows for management of the product model at the right level of granularity, now provides automatic and intelligent data loading/unloading for unmatched performance during assembly updates
- A new kinematics simulation to enable collaborative multi-discipline engineering on the single IP platform

Online Creation and Collaboration

- Systems engineering improvements for virtual simulation of the system within the 3D mockup. In particular, systems engineering addresses the growing trend of electronic content in all industries

- Thanks to the “engineering connection” unified model, user can specify once any connection information such as part mating, FTA, etc. and reuse them across multiple disciplines spanning from assembly, to kinematics and structural analysis
- Two new composite design approaches, “grid” and “solid”, for optimized design and mating with structural parts.
- Support for collaborative V4/V5/V6 design scenarios enables gradual adoption of V6 for an OEM and its supply chain.

DETAILED DESCRIPTION

GLOBAL COLLABORATIVE INNOVATION

- Delivers intuitive access to the 3D product definition through the PLM toolbar and compass to make it possible for communities to gather, share, and experience around 3D. For instance users can access contextual PLM information, identify other contributors, and connect with them in real time. Once connected, online instant collaborative tools are available for 3D brainstorming, from chat, and snapshot, to co-review and co-design.
- CATIA 3DLive is becoming the 3D portal to get access to product definition for all the actors across the company.
- CATIA V6 provides tools to allow both online global collaboration and global instant collaboration which enable peer to peer connection through heads up display to "drag and drop" ideas using 3D between designers.
- Online global collaboration facilitates large scale collaborative innovation across all engineering disciplines and the extended enterprise by allowing engineers to be connected via the V6 Collaboration Platform. A highly scalable architecture based on SOA for managing the worldwide product development business processes of the extended enterprise. Designers, engineers and all casual users are able to collaborate on the product as if they are in the same room.

LIFELIKE EXPERIENCE

- CATIA V6 put the 3D at the heart of the company. It enables the users to literally experience the product as if it was in real life. This is achieve by providing to new shaders to enable more realism into the product definition view but also new working ambiance with floor, reflection, etc,.
- Designer can benefit from unprecedented ease of use and be immersed in their daily work. Casual users benefit from the CATIA 3DLive interface and can easily find information, experience the product, and collaborate with others in an immersive online 3D environment. CATIA V6 brings the same simplicity and effectiveness found in consumer's product.

SINGLE PLM PLATFORM FOR IP MANAGEMENT

- CATIA is built natively on a single PLM platform SOA based. This single platform enables support of the IP modeling spanning all engineering discipline as well as collaborative business processes (CBP) covering the entire product lifecycle. Design data, engineering data and manufacturing data are all stored in the same

base and accessed in a unified way. This makes breakthrough multi discipline collaboration. CATIA extends in depth the Shape, Mechanical and Systems.

ONLINE CREATION AND COLLABORATION

- CATIA V6 is optimized to enable concurrent work in real time across remote locations with only a Web connection. This capability is a major breakthrough for any company implementing a global engineering and manufacturing strategy. True collaboration is becoming a reality. CATIA V6 delivers key technologies with advanced capabilities able to address wide scale Intellectual Property (IP) modeling in a highly collaborative environment, designing a jointly scalable industrial project from the initial idea to a final product.

Highlights by CATIA domains

CATIA SYSTEMS

CATIA Systems provides a single simulation platform for hybrid, multi-disciplinary simulation. It delivers an embedded systems solution to model critical systems. It is based on a formal language which makes it possible to generate deterministic behavior. It enables intuitive and reliable simulation through modeling and execution of any system model. This 3D integrated simulation environment allows fast and easy systems development. CATIA Systems consolidates CATIA Systems Logical 3D Architecture with new functionalities, more agility and automation.

CATIA Logical Systems Routing, a new workbench, is dedicated to project logical connections into the logical 3D network, respecting installation and separation requirements, offering new functionalities:

- Routing of the logical connections on the 3D pathways
- Route logical connections on pathway network
- Visualize logical connection in 3D
- Query logical connections routed inside a pathway
- Manage compatibility rules with business rules
- It enables early pre-sizing (estimated signal length) of the connective network and offers automated rule checks for installation and separation requirements.
- User can select the pathways to define the route of a given logical connection. It is possible to optimize the route of the connections in the pathways early in the design process, prior to the detailed physical design with tubing and electrical applications.
- Thanks to a simplified data model, (no need to detail in 3D each tube or cable), the design time and cost to validate the 3D Architecture is reduced.

3D Architecture design agility, CATIA V6R2009x facilitates early definition of the 3D logical architecture.

- Round or Rectangular Pathway Section: Represent with a single pathway (round or rectangular) a set of routed systems. For example a user may define quickly a unique round or rectangular pathway to contain several pipes instead of representing each pipe in 3D.
- Equipment Center model (Bay Area, Panel, Rack…): The creation of 3D Equipments centers (Bay Area, Panel, Rack…) allows to quickly allocating logical instance to 3D. A user may build an equipment center and associate the list of the 2D logical entities contained in it, he does not need to associate a 3D representation to each component, when the logical definition is enough.
- Production Break on Pathways: A new production break entity allows user to specify that a pathway will be broken to respect a wall (for instance, an electrical wire will include a connector to go through a fire-break wall), avoiding to duplicate the pathways according to the space partition.

As a result, the 3D design is simplified while the logical systems model is kept. User does not need to get in the detailed specifications in 3D during the design of the systems architecture, which becomes quicker and easier and more productive. He gains agility from 2D to 3D.

Design Automation, CATIA V6R2009x brings more automation to the Logical Systems 3D Architecture:

- Auto 3D Pathway Routing from Logical Connection: It is possible to generate automatically a straight pathway by selecting a connection in the 2D logical view of the system, without using the dedicated creation commands for pathways, thus reducing 3D Design time.
- Knowledge exposition for 3D for Systems objects, attributes and relations: it is possible to store company specifications about the routing operation, for instance to avoid interference, heat or dripping problems, specifically to each product or industry.
- Business rules for Pre & Post Validation of 3D projection of logical connection: It is now possible to capture customer logical routing validation rules with knowledge ware business rules. These rules may check automatically that the pathway routing respects the equipments specifications, simultaneously when defining the routing by selecting pathway segments or after this definition. It allows a design right at first time.

The definition of the design and its checking process gets more and more automated enabling user to anticipate problems, including assembly or dismantling problems.

Systems Virtual Execution (SVE) functional enhancements

With a simple set of properties, the temporal characteristics (frequency, synchronicity, offset, delay information) of the functional and logical system models are easily defined. The Virtual Execution platform will schedule and synchronize the simulated systems according to those dynamic properties. Moreover, multi-behavior models are supported, allowing different facets for each system (nominal behavior, trivial behavior, dysfunctional behavior...). Also, SVE support hybrid simulation, i. e. mixing LOC & DBM behavioral model or multi-frequencies systems.

Functional Logical Editor (FLE) ergonomics enhancements

VPM System Functional Logical Definition: improve editors user interface & ergonomics.

Control & Logic Modeling (LOC) enhancements

Ergonomics has improved, and LOC capabilities have been extended with the user type & enum support.

CATIA SHAPE

CATIA Shape delivers ultra fast modeling technologies for designers to embrace innovative shape design creation. It provides an intuitive environment to easily generate and modify styled surfaces at the concept design phase, and develop high quality mechanical shapes for detailed design purposes.

For this release, it provides:

- Fast Surfaces creation from Curves. CATIA V6R2009x unleashes industrial designer creativity with Imagine and Shape product and allows fast surface creation from curves or network of curves. This enables designers to create their own primitives to start their design. Three new primitive commands have been added through intuitive and productive interface with no additional geometry creation.
    - Extruded profile : enables the designer to create an extruded profile as a primitive to start the design
    - Revolved profile : enables the designer to create a revolved profile as a primitive to start the design
    - Subdivision Net from a set of curves: enables the designer to select a set of curves as primitive to start the design
- Autofillet. Automatic filleting represents a significant process improvement, especially in the design for the plastic and molded part. Previously, the process of filleting sharp edges to conform to existing features was a repetitive and very time-consuming phase of the shape design process. This new capability can automatically fillet the sharp edges of a shape in a single operation. Automatic filleting is particularly helpful for fillet manufacturing preparation.
- Real Time Rendering. CATIA V6R2009x takes real time rendering one step further and lets designers experience the product as if it was real by providing real time material configuration test and try to leverage the perceived quality of the final product. Shadows have been made more realistic with the addition of multiple light sources. The user has now the ability to compute and to store light maps off-line and then load them so that a scene can be rendered in real time without waiting for calculations to be made.
- Fillet - User Interface improvement of Blend Corner for user productivity. Thanks to the enhanced user interface, the usage of blend corners for fillet definition is expanded to avoid selection of small edges or setback constraints and avoid redefinition of former blend corners when geometry is changed. The Blend Corner user interface has been enhanced (creation by selection of edges or vertices, edition of the value, remove) to improve productivity.
- Styling Fillet. The interface provides instant feedback of the Connect Checker Analysis when creating the fillet which is the deviation value for G0/G1/G2/G3 curvature for these operations:
    - Internal connections within ribbon (result fillet).
    - Connections between ribbon (result fillet) and supports.

CATIA MECHANICAL

CATIA Mechanical delivers highly productive technologies. It enables the concurrent creation of any type of 3D parts, from machined, sheetmetal, composites, molded and forged parts to the definition of mechanical assemblies. It provides all the tools to complete the product definition, including functional tolerances and annotations directly specified in 3D, as well as kinematics definition. Advanced process-driven functionalities, such as productive definition of specific drafts and fillets for forge or complex cast parts, avoid the need for manual and lengthy operations while improving manufacturability and productivity. Users are provided with a highly collaborative design environment with full concurrent engineering and unbreakable relational design across the complete extended enterprise.

The following capabilities are part of the solution:

- CATIA Live Shape enhancements. The 3D co-pilot (or snapping functionality) provides user with ease of use thanks to a straightforward alignment on existing geometry in a precise and intuitive way, during modifications through translation or rotation. The absolute dimensioning ensures precise dimensioning. When moving a face, a user may set the reference 0 of the measure wherever the user wants, even outside of the part (by selecting a face or an axis, in another part if desired), easily switching between relative and absolute dimensioning. CATIA Live Shape provides user with Meta-CAD modeling. The import of existing geometry enables him to reuse, modify and upgrade models from old versions or to benefit from 3D brainstorming between analysis and design fields. The user may imagine the shape during the concept phase and may also perform 3D red lining after the simulation phase or for manufacturing intents. The user may benefit from any source of data (V4, V5, V6 or multi-CAD) through import process or direct dedicated conversions.
- The flexible sub-assemblies allow the user to define an assembly as flexible, such as the assembly of a connecting rod and a piston. As a result, on each instance of this assembly positioned on the crankshaft, the user may overload the position of the connecting rod or of the piston, independently while preserving the reference of the assembly. This capacity avoids duplication of the same assembly for different positions of its instances. Moreover, it maximizes the reuse of components, as the synchronization of a new version can be managed automatically. In a nutshell, the mechanical specification is captured in a more precise way, consistently with the visible definition of the assembly, without creating fake parts only for handling purpose. The flexible assembly is integrated with kinematics mechanisms and can be simulated.
- The generative assembly symmetry workbench offers scalability and ease of use for symmetric assembly management. For basic use cases, user can generate an assembly symmetric to another assembly in only one click. For advanced scenarios, a dedicated workbench allows the user to manage the specifications of structure, shape and positioning independently with unsurpassed flexibility. For instance, each part of an assembly may be taken as it is or in a mirror shape, however the other parts are defined. On one hand, two symmetrical parts can exist in the database, even if they are not instanced in the same assembly. On the other hand, two different parts may be positioned symmetrically, thanks to

engineering connections. The associativity of the shape and/or the structure of the part or the assembly is guaranteed. Natively based on a PLM 2.0 foundation, generative assembly symmetry is stored in the database instance per instance, a level of granularity that enables concurrent engineering and partial open.

- The relational design ergonomic & capacity have been reinforced and the user experience of the relational design tools has been empowered (cross highlight in the selection panel) and the capability of the edition of the links & relations in collaborative environment has been enhanced (when creating import links between parts, the coupling between assembly parts has been reduced).
- Wall thickness analysis. CATIA V6R2009x improves and accelerates the part and tooling design review process for manufacturability, especially for casting and injection molding processes. The new wall thickness analysis feature ensures that the thickness meets the requirements of the manufacturing process in order to anticipate and fix potential manufacturing issues
- Composites solid and grid approaches. CATIA V6R2009x brings 2 new approaches for the end to end composite process: These two additional approaches are the grid and solid approaches. Industry in Aerospace but also Energy for the turbine wings design are provided with a better optimization in the composite design steps and a better mating with structural parts.
- Blend corner def/intersection edge. CATIA V6R2009x improves productivity by providing easier definition and better stability of the Blend Corner. This helps in extending its use for fillet definition (getting rid of small edges selection) and in preventing the redefinition of former blends corners when modifying an Edge Fillet. Its user interface has been enhanced (creation by selection of edges or vertices, edition of the value, remove) to improve productivity.
- The Intersection Edge Fillet is based on a concrete requirement from the field to increase the user's daily productivity. This Part Design functionality helps the user to create fillets with the appropriate definition, which is a definition by the intersection of several features, beyond the intersection of faces. As a consequence, it improves the update stability during design changes. Moreover, it facilitates the capture and the reuse of the part design features. For example, the user can define a power copy including a set of features and a fillet defined at the feature intersection level. When this power copy feature is re-instantiated in another context, the fillet is correctly updated as it does not depend on the intersection of the selected faces but on the intersection of the features.
- Assembly layout view creation and Assembly drawing generation from 2D layout located at assembly level. The assembly layout view can be created directly in 3D with reference to the assembly geometry. As in captures, drafting capabilities are provided, such as 3D seen in transparency, points of view, tolerance sets, ratios and filters, as well as a view box taking into account assembly model size. The user may then generate the assembly drawing from the 2D layout located at assembly level, so that the drawing representation is natively consistent with the 3D representation.
- Assembly FTA definition features within Engineering Connections. FTA data are now embedded in 3D Shape representations of assembly products. A single “Functional Tolerancing and Annotation” workbench allows both creating 3D annotations inside 3D Shape representation (Part FTA) and on the 3D Shape representations geometry of the product and its components (Assembly FTA). Assembly FTA definition features are stored in Engineering Connections to allow

concurrent engineering at the granularity of individual FTA feature and to couple the life cycle of applicative data to the life cycle of the instances it connects.

- Integration of the drawings to PLM Update command. The drawing update becomes smarter: it determines the drawings requiring an update, loads the drawing and 3D required data, filters them before performing an overall assembly update. The reduction of the number of impacted drawings and the reduction of the interactions brings better performance (10- 15%) and lower memory consumption (which benefits to batch update too).
- Downward Compatibility to V5R18SP4. V6 representations can be exported from V6R2009 to V5R18 SP4 and higher V5 levels as V5 CATDrawing documents. The views are migrated as result where new data can be added and new sheets and views can be added too.
- Rigid position link for annotations. When the position of an annotation relative to its reference is not intended to change, the user may fix it by imposing a rigid behavior on the position link. Since the relative position may no longer be modified by direct manipulation of the annotation, the rigid position link offers security against involuntary moves.

CATIA EQUIPMENTS

CATIA Equipment delivers in the electrical domain a dedicated electro-mechanical end-to-end solution for designing and documenting electrical modules in all industries that design electric, electronic and electro-mechanic components. It reduces time and cost for creating electrical modules directly in the Digital Mock-Up, avoids interferences and create up-to-date manufacturing documentation.

- CATIA Equipment domain consolidates Electronics packaging
  - Import IDF with catalog of electronic component CATIA V6R2009x brings the possibility inside CATIA to access the ENOVIA catalog of electronic components in the context of the collaborative design between PCB boards 2D Layouts applications and CATIA V6 for PCB Boards 3D Design. This offers the possibility to develop a unified catalog of electronic component with ENOVIA library central
- CATIA Equipment domain consolidates Electrical Engineering
  - Improved Electrical Generative drafting: Line types, 2D symbols, Wire annotations, etc,.
  - V6R2009x Facilitates FBDI of V5 Wire Harness :
    - Improve FBDI electrical connection point with shared geometry
    - Improve FBDI of External Protections to Internal Protections with Post Treatment
- CATIA Equipment in the Piping and Tubing domain provides general layout tools for intelligent placement of parts. A full set of routing and parts placement methods allow users to choose the one that is right for a given context. Integration with the knowledge rules engine allows automation of the design process and ensures that company standards are followed throughout the design process. The product's comprehensive and flexible setup functions include an easy way to define project standards and catalogs that help users get into production rapidly V6R2009x brings the following highlights:

- Route Rigid Pipe/Tube through a Hole Port. It provides the ability to route a Pipe/Tube through a Hole Port, providing flexibility and ease of use. Users will be able to use existing objects, such as supports, to efficiently complete the required design
- Rigid Pipe/Tube Reference Selection using Port Data. When routing a NEW Pipe from a Part Port, use the port specification data to define the appropriate pipe reference. Example: Nominal Size. As added value, it enables design consistency and improves productivity. The user will not have to search for the required technological data to define the new route. The technological data is automatically derived from the selected port.
- Import 3D rigid pipe definition from a list of nodes and bend data. It allows the user to import one route path definition from an external file definition. The file will contain the list of nodes and bend data of the route. That provides flexibility during route path definition. This will improve productivity and efficiency.