CATIA V6R2012 - FACT SHEET
VIRTUAL PRODUCT

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VALUE AT A GLANCE

CATIA V6 LEVERAGES THE CAPACITY OF INNOVATION FOR COMPANIES OF ALL SIZES IN ALL INDUSTRIES BY DELIVERING BREAKTHROUGH PRODUCTIVITY DESIGN SOLUTIONS POWERED BY A HIGHLY COLLABORATIVE PLATFORM.

CATIA V6 redefines CAD from the purely physical product definition and expands it from digital mock-up to functional mock-up taking into account the multiple views that necessitates the product development (i.e. requirement, functional, logical and physical). Designed to operate on a PLM 2.0 platform, CATIA V6 is a 3D collaborative solution linking designers and non-CAD specialists.

CATIA V6 OVERVIEW

GLOBAL COLLABORATIVE INNOVATION

Broaden CATIA usage beyond designers to casual users within and outside the engineering department. Ground-breaking collaboration tools enable 3D brainstorming within the community of PLM users, to reach a new level of innovation.

LIFELIKE EXPERIENCE

CATIA introduces a paradigm shift to enable first- life experience and bring 3D product design to life with unmatched realism. In addition, CATIA V6 offers compelling simplicity and efficiency with in-context 3D manipulators and natural 3D operations.

SINGLE PLM PLATFORM FOR IP MANAGEMENT

Harnesses collective intelligence, making the always up-to-date product definition accessible to the various communities from anywhere, at any time. Facilitates multi-discipline collaboration among designers, engineering users, and manufacturing users from one unique IP repository, making the company knowledgeware assets available for all participants.

ONLINE CREATION AND COLLABORATION

Reaches new disciplines with CATIA Systems and widens the traditional scope of CATIA to the requirements, functional, and logical views of the physical product in a collaborative manner. This allows for direct traceability of the product, from the beginning to end phases of creation. CATIA V6 delivers PLM objects that match collaborative design innovation, eliminating heavy assembly files, enabling true concurrent design, and eliminating the needs for high references management between part, drawing, and products.

READY TO USE PLM BUSINESS PROCESSES

Opens new opportunities for new industries such as consumer packaged goods, consumer goods, and high tech. The CATIA product portfolio continues to cover more industry processes.

LOWER COST OF OWNERSHIP

Protects the V5 investment; as a natural extension to V5, CATIA V6 ensures a smooth upgrade and short ramp-up from V5 to V6. There is an easy transition to V6 with the use of the same modeler and ready–to-use migration path.

V6R2012 OVERVIEW

CATIA SYSTEMS

Systems functional and logical architecture definition reaches maturity in CATIA V6R2012
CATIA Systems Architecture Design,
CATIA Systems Logical Electrical & Fluidic Design,
CATIA Systems Conceptual Mechanical Design,
CATIA Systems Generative 3D Electrical,
CATIA Systems Generative Piping & Tubing

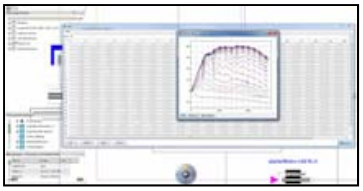
CATIA V6R2012 confirms the V6 platform readiness for easy design of functional and logical architectures during systems definition.

Traceability is at the heart of navigation as requirements, functional, logical and physical (RFLP) entities of any kind can be accessed across the network of their relationships.

Also, because requirements management capabilities are provided directly within CATIA, all systems-related objects are usable in the same authoring session. Automotive original equipment manufacturers (OEMs) have shown their trust in the Dassault Systèmes V6 systems engineering solution by selecting it to develop the future electrical, electronics, and embedded software (E/E) architecture of their new cars.

Unique integrated solution to develop complex multi-discipline systems
CATIA Dynamic Behavior Modeling

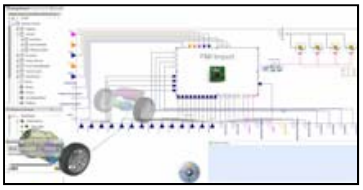
CATIA Dynamic Behavior Modeling solution (DBM) has now reached the same functional level as Dymola 2012.



CATIA Systems includes a comprehensive collection of libraries, ensuring that Dymola users find the functionalities to which they are accustomed.

Tighten systems engineers' collaboration for integrated product simulation
CATIA Dynamic Behavior Modeling,
CATIA Systems DBM Export for Simulink,
CATIA Systems DBM export for HILS,
CATIA Systems DBM binary export,
CATIA Systems DBM source code export,
CATIA Systems DBM source code export

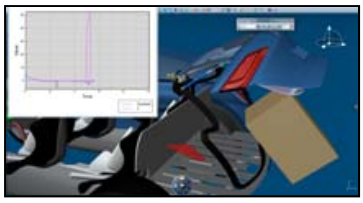
CATIA Systems V6 users and systems engineers using external applications such as Simulink benefit from four dedicated export options. These options ensure the simulation of both the controlled (plant) and controller models to fully validate the closed loop systems behavior.



At the same time, the Functional Mock-up Interface (FMI) standard is a key enabler for easy and secure exchanges of legacy data from other contexts, including non-Modelica tools, for an overall simulation within the CATIA DBM environment or in support of the supply chain schema, without exposing the intellectual property (IP) of the model.

CATIA V6R2012 brings design automation using knowledge within the Systems domain
CATIA Systems Architecture Design,
CATIA Dynamic Behavior Modeling

By integrating V6 knowledge management capabilities, the CATIA V6 Systems Engineering solution promotes design automation that allows systems architects to capture and re-use design best practices.



CATIA SHAPE

Improved performance and productivity for mechanical surfaces design with multi-processing
All CATIA products

CATIA V6R2012 brings important productivity gains to multiple CATIA core applications by providing multi processing support for its core boolean operations.

Most of CATIA applications take advantage of this new performance, including the mechanical surfaces design with healing operation that can improve performance by up to 40%.

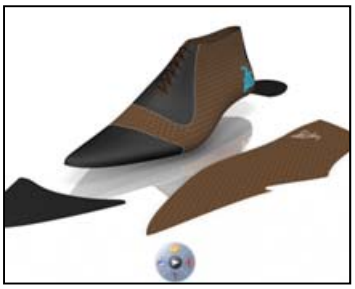
Reach functional completeness and V5-V6 feature equivalence for CATIA ICEM products
CATIA ICEM products

CATIA V6R2012 delivers perfection in virtual surface design quality. With this release, the CATIA Version 6 surfacing offer is fully mature and integrates industry leading ICEM technologies.

By combining freeform explicit modeling from ICEM with associative feature-based modeling, CATIA Version 6 delivers a powerful and intuitive set of tools to model, analyze, and visualize aesthetic and ergonomic shapes from the most basic to Class-A surfaces. This completes the feature set alignment of V5R19, V5R21, and V6R2012.

CATIA Industrial Design Refinement product includes techniques for design pattern and artwork on 3D
CATIA Industrial Refinement

CATIA Developed Shapes tools allow users to quickly and easily unfold any type of free form surfaces and to develop curves on a revolution surface.



Creative designers, both non-surface specialists and advanced-shape designers, can define an associative flattened pattern from their 3D models for manufacturing from flat sheets of raw materials.

CATIA Live Rendering delivers more content for interactive ray tracing
CATIA Rendering

CATIA Live Rendering for interactive ray tracing visualization is based on image based lighting with High Dynamic Range Images (HDRI). With this release, CATIA Live rendering brings new

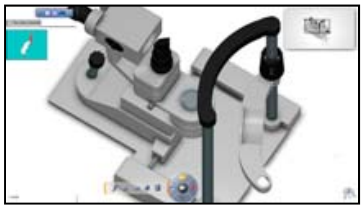
feature enhancements as well as a new library of HDRI content.

These enhancements add out-of-the-box, ready-to-use environments to create realistic, in-context rendering. Users can also import their own HDRI to recreate accurately any real scene.

CATIA MECHANICAL

CATIA Live Compose: the best tool you can rely on to conduct your architectural design
CATIA Live Compose

Your 3D concept is at the heart of your design. Select relevant items in your database and start an interactive "test and try" session, either alone or by collaborating with your project partners.



Positioning relationships are proposed on-the-fly using visual highlights as you approach parts or sub-assemblies. CATIA Live Compose makes it possible to build innovative proposals quickly in a 3D brainstorming spirit.

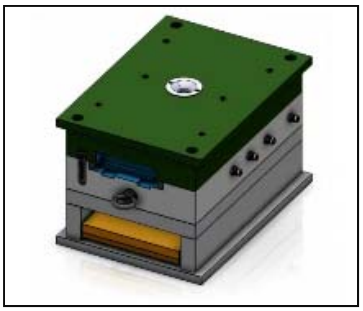
CATIA V6R2012 releases more creativity and collaboration with the new multi-touch experience
CATIA Live Shape

CATIA V6R2012 delivers the ultimate device-free way to boost the creativity of dedicated designers and casual users. The multi-touch experience is applied to 3D design through CATIA direct modeling in Live Shape and is the most natural way of modeling without constraints, delivering a true productivity gain.

It simplifies the collaborative process, improving the ease in which casual and management users can communicate ideas to the CAD specialist in 3D.

Design plastic products and molds with significant productivity gains
CATIA Plastic Part Design

The plastic part process from design to manufacturing benefits from enhancements in performance and capacity.

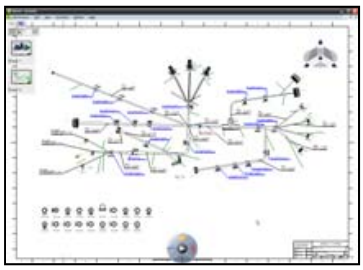


Embrace a wider scope of complex plastic parts and increase your design efficiency with modeling performances that are enhanced up to 40%, depending on the complexity of the part. In addition, in this dedicated and integrated process, you can choose among an extended library of mold base suppliers, natively incorporating positioning information for standard components.

Visualize CATIA V6 large drawings format in ENOVIA 3D Live for easy reviews
CATIA 3D Drafting & Annotation,
CATIA Industrial Design, CATIA 3D Design,

CATIA Mechanical Design,
CATIA Systems Conceptual Mechanical Design

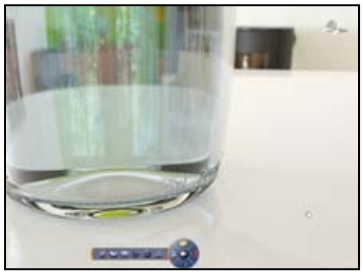
Perform your drawing reviews effectively with ENOVIA 3DLive and easy navigation among sheets.



Using only the 3D Live navigation platform, you can visualize drawings with both a vector quality and a low memory consumption, as well as print them on the fly.

Make your words visible with an engraved or embossed packaging of your product
CATIA Systems Conceptual Mechanical Design,
CATIA 3D Drafting & Annotation,
CATIA 3D Tolerancing & Annotation,
CATIA Wire Harness Documentation & Formboard

During detailed design, 3D text can be used to define accurately any kind of text that will be embossed or engraved in your product.



Customize your text using creative fonts, original alignments and orientations, or circular shapes. This release eliminates the need for text outlines to be imported, resulting in time savings.

CATIA EQUIPMENT

Collaborative ECAD MCAD Integration for printed circuit board design
CATIA Printed Circuit Board Design

With CATIA V6R2012, ECAD and MCAD designers can now better focus and concentrate on their PCB design, thanks to the collaboration workflow being optimized to reduce manual tasks and potential errors in the process.

Standard exchange files (IDF format) can now be saved directly within the Version 6 platform, ensuring full synchronicity at all times, as well as efficient traceability of design changes.

Full support of complex connector assemblies in the flattening process of electrical wire harnesses
CATIA Wire Harness Documentation & Formboard

The entire wire harness flattening and formboard process including drawing generation now handles complex connector assemblies.

This enhancement facilitates and accelerates the process for harness manufacturers because they will now be able to see the representation of complex device assemblies in the drawing while preparing the harness for the installation step.

Optimized piping and tubing design preparation for manufacturing
CATIA Piping & Tubing Design

CATIA V6R2012 improves the detailed piping and tubing design and optimizes design preparation for manufacturing.

CATIA Piping & Tubing Design reaches a new level of performance for layout refinement and network resizing. It also offers new capabilities with assisted spool creation and insulation management to address growing design complexity.

V6 PLM EXPRESS
V6 PLM Express is a dedicated solution tailored to meet the needs of smaller organizations. It combines best-in-class Dassault Systèmes PLM solutions – such as CATIA, SIMULIA, DELMIA a – with key PLM 2.0 values including real-time seamless collaboration, online-enabled, design-anywhere functionality and a single, scalable platform adaptable to a variety of roles.

NEW IN THIS RELEASE:

Extended process integration between EBOM and MBOM processes with the new DELMIA process planning role.
The Manufacturing Process Planner role includes three new solutions:

- Starting with the DELMIA MBOM Planning Pack (MPK) users are able to define and manage the MBOM based on the EBOM within the V6 collaborative environment;
- MPK can then be scaled by adding process and resource planning with the DELMIA Process Planning Pack (PGK);
- Additionally, assembly processes can be planned, validated through simulations, and documented for shop floor implementation using the DELMIA Process Simulation Pack (PSK).

Enhanced deploy-ability with:

- Extended parameterisation of lifecycle and product identification practices to manage a greater number of product development scenarios
- User defined attributes policies and scalability to enable higher levels of PLM data to be managed between companies and V6 lifecycles.
- Secure automation tools for process automation whilst maintaining the low cost of ownership and simple system management philosophy.

Reduced cost of ownership and deployment with embedded MS SQLserver DB
Reduced cost of ownership and deployment with embedded MS SQLserver database. Thus, enforcing the "out of the box" approach of V6 PLM Express

Reach new collaborators with CATIA Live Shape and Compose tools for new or casual user.

CATIA V6 DOMAINS AND V6R2012 ENHANCEMENTS

CATIA SYSTEMS

Across industries, products are getting increasingly complex, involving many more engineering disciplines, with the value shifting from the products themselves to the actual services demanding customers expect from these products.

REQUIREMENTS ENGINEERING

The effective requirements engineering of complex products and systems ensures compliance to regulations and specifications, at the same time it improves time-to-market and helps reduce overall costs. It allows organizations to capture the "voice of the customer and translates that information into new products and systems in a timely and efficient manner.

SYSTEMS ARCHITECTURE

Delivering complex products and systems demands an open, extensible and versatile approach to system architecture definition, development, integration to modeling and simulation environments, as well as full lifecycle management of the underlying architecture entities and artifacts.

SYSTEMS MODELING & LIBRARIES

Today it is common practice to use many different models to simulate the behavior of complex systems and products. The challenge with this approach is that these models rarely interoperate with one another and do not exist in an aggregated environment for a 'whole-system multi-physics simulation' of the complete product. The Dassault Systèmes (DS) solution set provides a fully integrated systems modeling environment that leverages the open Modelica language as well as logic state machine based simulations.

EMBEDDED SYSTEMS SOFTWARE

Our embedded software management capabilities enables system architect to define, simulate and generate C-code for any control systems. The compiler is based on a technology, which is widely used in aerospace industry, to produce safety critical embedded software. Our solution comes with a set of editors including Grafcet, Statechart and Dataflow to enable components re-use as well as dynamic type and unit propagation.

Major V6R2012 ENHANCEMENTS FOR CATIA SYSTEMS:

Systems functional and logical architecture definition reaches maturity in CATIA V6R2012

CATIA Systems Architecture Design,
CATIA Systems Logical Electrical & Fluidic Design,
CATIA Systems Conceptual Mechanical Design,
CATIA Systems Generative 3D Electrical,
CATIA Systems Generative Piping & Tubing

Recent continued adoption of V6 Systems Engineering by automotive OEMs to develop their future electrical, electronics and embedded software (E/E) architectures demonstrates the appeal and readiness of V6's comprehensive systems engineering solutions. Indeed, CATIA V6R2012 offers numerous enhancements confirming the V6 platform readiness for easy design of functional and logical architecture during the systems definition phase. A spreadsheet-like editor enables fast and easy mass updates of functional and logical information. Requirements are now displayed within the same graphical navigator as functional and logical components in CATIA, making it natural to follow relationships between systems requirements and other RFLP entities. Additionally, requirements can also be authored from CATIA, and users can establish relationships between them and corresponding entities. It is even easier to access RFLP entities of any kind across the network of relationships that link them, because the visualization of these relationships can be filtered in the navigation graph.

3D modeling and dynamic systems behavior simulation are fully available within the same CATIA V6 design platform

CATIA Dynamic Behavior Modeling,
CATIA Systems DBM Export for Simulink,
CATIA Systems DBM export for HILS,
CATIA Systems DBM binary export,
CATIA Systems DBM source code export,
CATIA Systems DBM source code export

With CATIA Dynamic Behavior Modeling (DBM) now at the same functional level as Dymola

7.5, CATIA Systems provides a unique integrated solution. When modeling the dynamic systems behavior, systems engineers can benefit from ready-to-use model libraries and simulation tools. First, they can ensure state of the art design of complex physical systems through the latest level of the Modelica language Standard Library (MSL) 3.2 language support. Other commercial libraries covering a wide range of disciplines can be used in demanding industrial applications. This comprehensive collection has been enriched with Model Calibration and Experimentation and Validation modules, allowing engineers to gain an accurate knowledge through tuning and validation of the model behavior. For instance, Monte-Carlo Analysis explores the model response when varying multidimensional parameters at the same time. The Model Management library shortens the development time for complex systems for regression testing, library analysis, and encryption.

Four new CATIA Systems products have been introduced to tighten the collaboration between CATIA Systems V6 users and those using external applications such as Simulink. These products are used as interfaces to ensure the simulation of both the controlled (plant) and controller models to validate the closed loop system behavior:

- CATIA Systems DBM Export for Simulink
- CATIA Systems DBM Export for HILS
- CATIA Systems DBM Binary Export
- CATIA Systems DBM Source Code Export

In addition, the new CATIA Systems DBM Optimization product allows systems engineers to determine improved values of model parameters by multi-criteria optimization based on multiple simulation runs. Parameters are calculated to minimize criteria which express in mathematical terms what improvements are expected.

Requirements management and authoring for designers within CATIA V6

CATIA Systems Architecture Design,
CATIA Systems Logical Electrical & Fluidic Design,
CATIA Systems Conceptual Mechanical Design,
CATIA Systems Generative 3D Electrical,
CATIA Systems Generative Piping & Tubing

Requirements management capabilities are provided within the RFLP Navigator or in the Functional and Logical Editor workbench in CATIA, making all systems-related objects usable in the same design environment. Requirements can be authored from there and designers can establish relationships between them and corresponding entities, without the need to switch to an ENOVIA window.
This traceability and authoring capability allow systems engineers to perform frequent reviews of requirements, functional breakdown, and logical models at any stage of systems design, especially after validation steps. This increases quality through consistency as well as raising productivity, making it possible for any collaborator to benefit from the shared system modeling capabilities.

Functional Mockup Interface – A key enabler to integrated product simulation

CATIA Dynamic Behavior Modeling

Systems engineers or architects can import Functional Mock-up Unit (FMU) models for simulation purposes in CATIA Systems DBM thanks to the Functional Mock-up Interface (FMI) standards from the Modelisar project. This enables the import of legacy data or third-party models from other simulation environments, including non-Modelica tools, and the simulation of both the controller and plant models in the context of the whole physical system within CATIA DBM environment.
Secure and easy exports of your Modelica models can also be ensured to support supply chain schema. FMI Export allows systems engineers to export behavior models to dedicated downstream applications, without

exposing the IP of the model normally visible in a Modelica text file.

High design automation through knowledge management usage within the functional/logical model

CATIA Systems Architecture Design,
CATIA Dynamic Behavior Modeling

There are several different ways to leverage knowledge management in systems engineering. For instance, wizards can propose already existing functions for immediate reuse during the functional breakdown of a new function. Knowledgeware parameters, including exposed Modelica parameters, can be added to function and logical references by the user. Rules can control parameters to prevent them operating outside predefined limits (for instance). Administrators control the display, export, or update of functional and logical entities through knowledge scripts. For example, critical functions can be automatically colour coded (e.g. put in red) based on discipline or company rules. In addition, during a 3D simulation, meaningful graphic representations can display a part according to its current operating parameters such as its speed for a windmill wing or its varying state of charge for a battery. Similarly, physical components can be linked to logical ones so that physical parameters can drive the logical behaviors.

The CATIA Systems Engineering offer continues to be enhanced for the academic market

Academic communities benefit from three more products for 3D virtual modeling and simulation of complex multi-physical systems:

- CATIA Systems Generative 3D Electrical Design (EGD)
- CATIA Systems Logical 3D Architecture (TDS),
- CATIA Systems Generative Piping & Tubing (PTA)

CATIA SHAPE

CATIA Shape provides industrial designers, Class A modelers, and mechanical engineers with a full suite of surfacing, reverse engineering, and visualization solutions to create, modify, and validate any type of complex innovative shapes and help streamline the transition and collaboration between Design, Class A and Engineering departments. From subdivision, styling, and Class A surfaces to mechanical functional surfaces, CATIA Shape covers all the surface creation and modification needs. It also supports the complete reverse engineering process from the import of digitized data to the recovering and finalization/completion of high quality surfaces.

INDUSTRIAL DESIGN

Combining a unique and intuitive virtual clay modeling approach with free-form surfaces, CATIA Shape is at the service of industrial designers and their creativity. Starting 3D ideation from scratch or from 2D sketches, industrial designers can manipulate shapes with unrivaled freedom, but also take full advantage of a true creativity accelerator to quickly explore and test many more ideas in the early conceptual phase. CATIA Shape also provides real-time visualization for instant evaluation and photorealistic Mental Ray rendering, allowing designers to make better and faster decisions

SURFACE REFINEMENT (CLASS A & LOFTING)

CATIA fully addresses the Automotive Class-A process with a solution for surface refinement that integrates industry leading ICEM surfacing technologies. By combining the V6 technology strengths, which include knowledge capture-and-reuse paradigm, it delivers a powerful and intuitive suite of tools for modeling, analyzing and visualizing aesthetic and ergonomic shapes for the highest Class-A surface quality.

MECHANICAL SURFACES

CATIA Shape provides advanced technologies for mechanical surfacing, based on a powerful specification-driven modeling approach. This solution brings high-end quality surface modeling for detailed designers and promotes efficient concurrent engineering between styling and engineering worlds in order to optimize the product design workflow.

COMPOSITES

Spanning from preliminary to engineering detailed design and manufacturing preparation, CATIA Shape provides unique capabilities for designers of composites structures to work in dedicated design in-context environments to integrate structural, assembly, and manufacturing requirements early in the composites design process and thereby anticipate and avoid problems.

Key V6R2012 ENHANCEMENTS FOR CATIA SHAPE:

Reach functional completeness for ICEM products

CATIA ICEM products

CATIA V6R2012 delivers perfection in Virtual Surface Design. With this release, CATIA V6's surfacing offer is fully mature and integrates industry-leading ICEM technologies. By combining freeform explicit modeling from ICEM with associative feature-based modeling, CATIA V6 delivers a powerful and intuitive set of tools to model, analyze, and visualize aesthetic and ergonomic shapes from the most basic to Class-A surfaces. This advanced surface modeling solution enables mechanical designers, shape designers, and stylists alike to create, validate, and modify surfaces, such as those in the automotive interior and exterior design and in aerospace lofting and interior cabin design domains.

CATIA Industrial Design Refinement product now includes techniques for design pattern and artwork on 3D

CATIA Industrial Design Refinement

CATIA Developed Shapes tools allows users to quickly and easily unfold any type of free form surfaces and to develop curves on a revolution surface. Creative designers, both non-surface specialists and advanced-shape designers, can define an associative flattened pattern from their 3D models for manufacturing from flat sheets of raw materials.

Improved performances and productivity for mechanical surfaces design with multi-processing
All CATIA products

CATIA V6R2012 takes advantage of multi-core hardware architecture to improve the performance of the CGM Boolean operation. The new CGM Boolean operator mode is called the *multi-processed* Boolean. With this mode, some internal operations within the Boolean are done in parallel, improving performance on some scenarios. If the user setting is activated, the multi-processed Boolean is used for all CGM Boolean operations within all functionalities. CATIA V6R2012 supports multi threading and brings important productivity gains to multiple CATIA core applications by providing multi threading support for its core Boolean operations. Most of the CATIA applications take advantage of this new performance, including the mechanical surfaces design with healing operation that can improve performance by up to 40%.

CATIA Live Rendering delivers more content for interactive ray tracing
CATIA Rendering

CATIA Live Rendering for interactive ray tracing visualization is based on image based lighting with High Dynamic Range Images (HDRI). With this release, CATIA Live rendering brings new feature enhancements as well as a new library of HDRI content. These enhancements add out-of-the-box, ready-to-use environments to create realistic, in-context rendering. Users can also import their own HDRI to recreate accurately any real scene.

Materials management and application enhancements
CATIA Industrial Design
CATIA ICEM Class A Surface Design

CATIA V6R2012 makes it possible to assign covering materials on features inside 3D shapes, further increasing detail and giving you more freedom when designing. This release also provides the PLM Compass mode so you can easily visualize which parts of the structure have a core material and which parts do not. The compass provides enough information so you can easily correct errors from a material point of view. A new command is included to display a balloon each time you click on geometry; the balloon enables you to select the core or the top-most covering material assigned onto a geometry and to choose an action such as remove or re-apply.

Loft command enhancements
CATIA Mechanical Surface Design

The Loft command has been enhanced to define continuity in curvature (G2) with section supports or guide supports where previously only tangent continuity was supported. This new capability increases the quality of the Loft because you can define precisely which continuity you want to apply on sections and guides. Another improvement lets you create a surface with a punctual extremity in one shot. In the Loft command, one section and two guide curves intersecting each other at one extremity are sufficient to create such shapes, resulting in increased productivity.

Sweep untwist at C1 discontinuities for better productivity
CATIA Mechanical Surface Design

This new option of the Sweep feature allows better management of C1 discontinuities on the guide of a sweep operation. The purpose of the option is to consider the C1 discontinuities as twisted areas and to benefit natively from the sweep removal or fill possibilities for the twisted areas. Because C0 vertices areas can automatically be filled or removed without any manual handling, productivity is increased.

Apply global deformations on wireframe
CATIA Generative Shape Optimizer
CATIA Body in White Modeling

This enhancement extends the possibilities of several GSO commands, including Bump, Shape Morphing, Wrap Curve, and Wrap Surface. Historically, such functionalities were only

applicable to surfacic inputs. However, it is now possible to deform a wireframe skeleton in the same manner, enabling additional usage scenarios. Previously, surface deformation could lead to an unsuitable surface quality; however, now you can deform the wireframe skeleton instead and rebuild surfaces on the deformed result.

Enriched scope of functionalities for Functional Shape Design

CATIA Body in White Modeling

Ergonomy of the properties function panel has been enhanced with an interactive picture to help the user to understand the manipulated concepts, homogenizing the creation command panels available in the workbench. CATIA V6R2012 allows you to create draft properties that contain draft angle and draft direction specifications. You are now able to share draft specifications data between functional features. This release also provides the ability to set up variable draft angles for the stamp functionality directly in the 3D viewer and to define G1-constant angle law by defining draft values for each tangent piece of input profile.

CATIA MECHANICAL

Finding ways to reduce design-to-manufacturing cycles and improving productivity are key priorities. CATIA Mechanical delivers a highly collaborative and flexible design environment with full concurrent engineering and high performance change management through relational design to enable the efficient definition and engineering of any type of 3D parts and assemblies, from the simplest to the most advanced. In V6, CATIA expands 3D design to user communities outside of the design office, addressing each profile with the right modeler capabilities: direct 3D modeling, geometrical surfaces handling, feature-based design and history-free functional modeling.

CONCEPTUAL DESIGN

Whether you are a casual user or a CAD-specialist, CATIA enables you to quickly create new designs or modify existing ones, even from other CAD solutions. Creating a new assembly structure becomes as easy as assembling toy building blocks. Manufacturing constraints can be embedded early on the preliminary shape, enabling downstream users to access the design and insert their specifications directly in the 3D models.

MECHANICAL SYSTEMS ENGINEERING

Based on the RFLP (Requirements, Functional, Logical and Physical) approach, CATIA fosters the collaborative definition of a product across its different views from conception to production and operation. This enables designers, engineers and system architects to define the product functional breakdown, the logical entities representing the technological solutions and the corresponding physical parts and assemblies.

PRODUCT DESIGN

CATIA enables the creation of any type of 3D part, from rough sketches up to the definition of mechanical assemblies. It provides all the tools needed to complete product definition, including functional tolerances and annotations, as well as kinematics definition. Cast and forged product design is addressed, as well as plastic or molded parts through a functional modeling approach. CATIA also covers also welding, fastening and sheetmetal processes.

TOOLING DESIGN

CATIA provides a wide range of application for tooling design, for generic tooling as well as mold and die. A rich catalog of industry-standard components is provided to automate tooling definition. Specific tools are also provided to address the needs of mold tooling injection designers.

Key V6R2012 ENHANCEMENTS FOR CATIA MECHANICAL:

CATIA Live Compose is your solution for managing mechanical structures

CATIA Live Compose

CATIA Live Compose helps to achieve the objective of putting the 3D product in the center of the design by getting rid of administrative tools and enabling you to reach non-CAD designers or non-specialists. In a CATIA Live Compose session, you can search for parts in the database, position them easily and rapidly compose a new assembly. You get a clear idea of the assembly structure since turntables display how it is broken down. Navigation becomes very natural, with easy movements up and down the different levels. Editing of the structure is easy, too: select a sub-assembly and simply drag and drop it on the right level and position. Engineering connections capture mechanical constraints between the different parts, automatically or on demand. You can build consensus through a N to N collaboration, during a validation scenario for instance. Collaborative sessions offer real 3D brainstorming because anyone can take the lead and modify a product structure to make propositions. Synchronous product composition and sharing and merging of components ensure a fast design project check and generate new ideas.

Design using only your fingers on a multi-touch screen or tablet on the Windows 7 operating system

CATIA Live Shape

With CATIA V6R2012, using the mouse or keyboard is no longer necessary. Manipulate the viewpoint with one or two fingers and

access all basic functionalities such as pan and zoom by sliding, spreading, or moving two fingers closer or with a single touch. The ability to draw with one finger also dramatically changes the user's experience.

Assembly enhancements: engineering connection visualization, assembly pattern and live smart positioning
CATIA Mechanical Design,
CATIA Systems Conceptual Mechanical Design

Engineering connections are visualized in 3D with a totally new look allowing for immediate readability. Through this 3D visualization, the user easily finds engineering connections related to a selected or preselected product and can launch editing of the engineering connections without needing to look in the specification tree. Patterns can be reused from former designs, pattern libraries, or from defined skeleton patterns, since you can instantiate a product on a list of axes. Enhanced ergonomics make it very supple because it is possible to define an assembly pattern and then to replace some of its instances or to define local modifications such as different representation and different positioning on them. Easily find the positioning solution for your sub-assembly thanks to the Live Smart Positioning algorithm. The list of constraints that correspond to a browsed solution helps users to choose the most adequate solution among solutions with same position. Available Smart Positioning when moving a product under constraint makes assembling very realistic and clear.

CATIA Live Shape lifelike experience enhancements
CATIA Live Shape

Modify parts as intuitively as possible by push-pulling edges or vertices or adding required edges on the fly. Feel as if you are drawing in real life, and choose on demand the geometric elements to snap on, rather than being faced with a multitude of them in the same time. As a result of these enhancements, copilot performances improve and you can get a clear view of the part specification by filtering meaningful constraints, depending on the context. Save valuable time by creating several constraints of the same type on- the- fly using multi-selection.

More visibility of weight and inertia attributes helps users effectively manage these aspects along the product lifecycle
CATIA Industrial Design,
CATIA 3D Design,
CATIA Mechanical Design

To follow up product attributes such as mass, volume, declared or computed weights, and inertia information, these attributes can be added in the Bill Of Material (BOM) columns. This associative BOM is displayed and editable in generated drawings or in 3D views.

Fastener modeling enhancements make your industrial process very productive
CATIA Fastener Design

Standards of fasteners can be administrated safely by organization or project, as they are integrated in the Project Resources Management (PRM) infrastructure. Your fastening process can be optimized by using the same spot fastener for the different stacking of different configurations. Design quality is ensured by easily checking fasteners' consistency relative to defined and customized rules to take advantage of company know how. Also, time can be saved during a fasteners review by storing the Fasteners Analysis results into a report text file with delimited sections including comments that can be opened in a spreadsheet application for reading comfort.

Create 3D text for an engraved or embossed packaging of your product in a very flexible way
CATIA Systems Conceptual Mechanical Design ,
CATIA 3D Drafting & Annotation,
CATIA 3D Tolerancing & Annotation,

CATIA Wire Harness Documentation & Formboard

During detailed design, 3D text can be used to define accurately any kind of text that will be embossed or engraved in your product. Direct creation of 3D profiles saves time as text outlines are created like any 2D geometry defined in a sketch and do not need to be imported.
Customize your text with creative fonts, original positions, alignments and orientations, or circular shapes for instance. Positioning is a flexible process, as the positions of the geometrical outlines are globally taking into account all the relative positioning relationships. The text command makes them become thick or hollow for an embossing or engraving operation. This is available in the Sketcher, Part Design, Generative Shape Design and 2D Layout for 3D design workbenches.

Visualize CATIA V6 large drawings format in ENOVIA 3D Live for easy reviews
CATIA 3D Drafting & Annotation,
CATIA Industrial Design,
CATIA 3D Design,
CATIA Mechanical Design,
CATIA Systems Conceptual Mechanical Design

Perform your drawing reviews effectively with ENOVIA 3DLive and easy navigation among sheets. Using only the 3D Live navigation platform, you can visualize drawings with both a vector quality and a low memory consumption, as well as print them on the fly.

Since sheets are displayed in a maximized window, you are able see the widest possible area of a large format drawing (such as an A0 roll). As a first step, create non-persistent measures on the drawing sheets to feed a validation process.

CATIA EQUIPMENT

CATIA Equipment provides an integrated environment that enables the collaborative detailed design of electronic, electrical, and fluidic systems in context of a virtual product. While design is driven by the system logical definition to ensure conformity with product specifications, full traceability, and configuration management, knowledge rules are integrated to enable the automatic compliance to standards throughout the design process, all the way to the production of associative documentation for manufacturing. Such an integrated environment improves design quality, drastically reduces time needed for modifications, and minimizes errors.

ELECTRICAL WIRE HARNESS DESIGN

CATIA delivers a dedicated electro-mechanical, end-to-end solution for designing and documenting electrical wire harnesses in all products that include electric, electronic and electro-mechanic components. Creating electrical modules directly in the product digital mock-up reduces time, costs and part interferences. It also enables automatic creation of manufacturing documentation.

PIPING & TUBING DESIGN

CATIA provides general layout tools for intelligent placement of piping and tubing parts. A full set of routing and part placement methods allow users to choose the one that is right for a given context. The CATIA knowledge and rules management capabilities enable automation of the design process and compliance of company standards. Rules setup is easy with project standards and catalogs.

ELECTRONICS

Today's consumers require compact electronic devices with greater functionalities. To create high-quality products faster, companies need the integration of realistic Printed Circuit Boards (PCBs) inside a virtual product and simplified collaboration between mechanical and electronics specialists.

Key V6R2012 ENHANCEMENTS FOR CATIA EQUIPMENT:

CATIA 3D Electrical Design now covers cable routing and cableways
CATIA 3D Electrical Design

The current harness design product capabilities are now extended to be usable for applications in the electrical domain such as cableway designing and routing. The product is now more generic for electrical applications and is not merely restricted to wire harnessing. The look and feel of the Wire Harness workbench has evolved accordingly, along with the User Interface, Knowledge Exposition, and CAA exposition.

Large scale automatic systems routing in batch mode for the aerospace, shipbuilding and energy industries
CATIA Systems Logical 3D Architecture

CATIA V6R2012 provides automatic signal routing in batch mode directly on the Version 6 platform, supporting industries with massive data such as aerospace, shipbuilding and energy. This out-of-the-box solution respects installation and separation requirements and enables early pre-sizing of the connective network, enhancing cost efficiency during the design of large scale logical architectures.

Enhanced collaboration between systems architects and 3D fluidic designers
CATIA Generative Piping & Tubing

CATIA V6R2012 provides automated 3D physical generation from 2D logical specifications of fluidic systems. CATIA Systems Generative Piping & Tubing unifies the logical definition of a pipe or tube and its physical mockup in a single workflow. Overall design change management cost is dramatically reduced and quality is improved thanks to the tight coupling of both the logical and physical aspects of designs.

Loop branch in support for 3D Electrical Design product
CATIA 3D Electrical Design

This enhancement allows looping of a single branch on a single support section thus enabling you to simulate real-world practices that involve looping branches on a support for temporary phases in harness design or final

phases of the design wherein looping is necessary to eliminate sagging branches in the manufacturing phase.

Harness flattening of complex connectors
CATIA Wire Harness Documentation & Formboard

The entire wire harness flattening and formboard process (extraction + flattening + drawing generation) can now handle complex connector assemblies, making the installation easier and quicker for harness manufacturers. They will now be able to see the representation of complex device assemblies in the drawing while preparing the harness for installation step. A complex device assembly is a set of devices linked to the end of a branch route through the segment connection point. This set can have several layers or levels starting from the branch end all the way to the equipment connector.

New command for assisted spool creation in Piping & Tubing Design product
CATIA Piping & Tubing Design

A Spool is defined as a group of physical objects that are manufactured into an assembly. It is used to group the piping parts and rigid pipes that are placed in the model. This command provides the ability to create spools. The resulting spool objects will be saved in the database, and this command allows you to define and delete spools and to add or remove objects from a spool.

New Resize Network command for pipes and tubes makes it possible to propagate a nominal size change along the network
CATIA Generative Piping & Tubing

During the tubing and piping process, some resizing of the piping components is necessary. The goal of the Resize Network enhancement is to replace a sub-part of the existing network with some other references that have same attribute with the exception of the Nominal Size, allowing you to propagate a nominal size change along the network.

Piping line extremities definition in spreadsheet editor and checking in PLM east compass
CATIA Systems Logical 3D Architecture

CATIA V6R2012 introduces the ability to edit connections to the spreadsheet editor,, especially for extremities of piping lines. It also introduces the ability to review piping line extremities using thee PLM East Compass. The compass will give the user visual feedback on his piping line extremities design and will allow a quick review of the user's work. This enhancement provides a very quick way to review a piping design with a user-friendly interface. In one look the user gets enough information to know if they have correctly connected all piping components and pipes belonging to a piping line to the correct equipment.

Cross highlighting to implement link between Logical 2D and Physical 3D
CATIA Systems Logical 3D Architecture

This enhancement provides the capability of cross-highlighting between logical equipment (and ports) with their physical 3D implementation. You now have the ability to see the relationship between an item defined in a logical system and the physical item implementing it at-a-glance. This enhancement leverages the level of integration between the system definition and the physical design.

Collaborative ECAD-MCAD circuit board Design with ENOVIA V6
CATIA Printed Circuit Board Design

Collaboration between MCAD and ECAD designers is based on the exchange of IDF files. To improve this exchange, it is necessary to be able to save an IDF file in the PLM database. In order to facilitate the "Import from IDF" functionality of CBD, it is now possible to select directly a PLM document containing an IDF file in the database for generating a PCB assembly. Previously, users were able to save an IDF file in the database but had to rename

the *.idf as *.txt before saving it in the database and then to revert to an *.idf file for loading in CBD.

Easy 3D component placement on flex circuit board

CATIA Flexible Printed Circuit Board

The Place Component on Flexible Board command allows 3D electronic components to be placed and creates the necessary data for moving the component during the fold and unfold process. The manual creation of the axis system for placing the component was not very easy and productive for the user. This enhancement allows easy placement and movement of 3D components on the flexible board and axis systems are created automatically in the right place.

DESIGN KNOWLEDGE & REUSE

Design Knowledge & Re-use enables companies to model, capitalize and re-use the full complexity of their engineering knowledge in order to accelerate and secure drastically their product development processes. Design Knowledge and Re-use accelerates a company's business processes while ensuring compliance with its best practices and taking advantage of its collective know-how. It provides an access to advanced design parameterization, knowledge capture as well as optimization tools, and enables the definition of standard rules and checks for design quality assessment.

For more information on the product content, come visit us at:

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