CATIA V6R2011 - FACT SHEET
VIRTUAL DESIGN FOR PRODUCT EXCELLENCE

VALUE AT A GLANCE
CATIA V6 AND V6R2011 OVERVEIW
CATIA V6 DOMAINS AND V6R2011 ENHANCEMENTS

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 to requirement, functional and logical views where it also combines dynamic behaviour modelling of systems engineering. Designed to operate on a PLM 2.0 platform, CATIA V6 is a 3D collaborative solution linking designers and non-CAD specialists. CATIA V6R2011 reinforces systems engineering by delivering a supplemental suite of Dynamic Behavior Libraries to model complex multi-physical systems, helping users to design and validate better products faster. Today's release also delivers enhancements to CATIA Live Distiller, increasing usability and "share-ability" of 3D assets to new audiences. In addition, V6R2011 extends its unique functional modeling technology to surfacing for the Body-in-White community.

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.

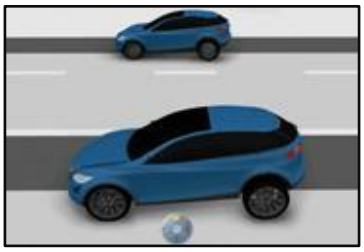
LOWER COST OF OWNERSHIP

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.

V6R2011 OVERVIEW

CATIA SYSTEMS

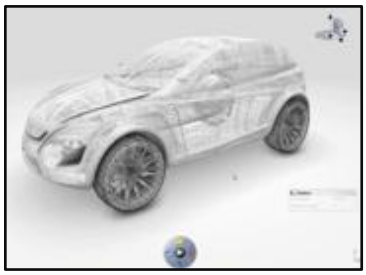
CATIA Systems is the unique systems engineering product suite which enables multidisciplinary design and integrates requirements, functional, logical, and physical (RFLP) product definitions. This release further advances its ability to model and simulate the dynamic behavior of systems under multiple simultaneous physical phenome-



na. Dassault Systemes' Dymola technology and component libraries are now augmented with 11 new libraries specifically designed for advanced modeling of complex systems in the automotive, aerospace, robotics, and energy sectors.

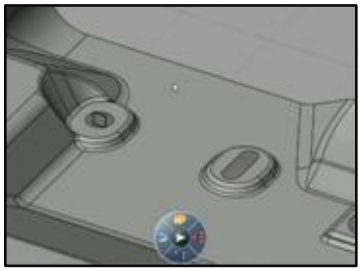
CATIA SHAPE

CATIA Distiller extends the already released CATIA Live Distiller functionality to further the use and "share-ability" of 3D assets to new audiences. It provides user-friendly content for consumers, enterprise functions such as marketing, and for specific purposes such as real-time visualization, virtual reality experiences, simulation or rendering. Data tessellation, reorganization and simplification create a lighter representation that is easily exported to other applications



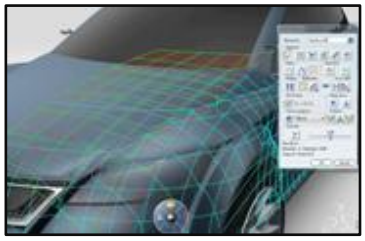
via the open, lightweight 3DXML format. 3D models can now be published easily to any online community such as 3DVIA.com.

CATIA pushes ahead and extends its unique functional modeling technology to surfacing for the Body-in-



White community. For years, CATIA's functional modeling capability has enabled users to focus on and capture their design intent, by freeing themselves from the sequence in which they model and by associating behaviors with design features, thereby decreasing design complexity. It also brings greater flexibility to evaluate multiple design variations in a short time frame.

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



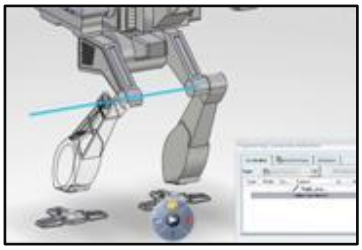
advanced surface modeling solution enables mechanical designers, shape designers, and stylists alike to create, validate, and modify surfaces, such as those in the automotive interior and exterior design, and aerospace lofting and interior cabin design domains.

CATIA MECHANICAL

CATIA V6R2011 strengthens manufacturing preparation processes with CATIA Bend Part Design and Mold Tooling

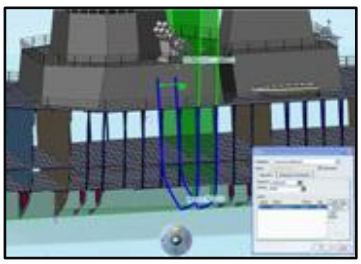
Design. Bend Part Design quickly creates bent part designs, from paper wrappers and cardboard boxes to aerospace sheet-metal components and provides access to both folded and unfolded representations.
Mold Tooling Design enriches its mold tooling offer with a new specialized product for mold tooling injection.

CATIA Mechanical is bridging the gap between direct modeling technology from Live Shape and the know-how of functional modeling solutions to offer a realistic, lifelike

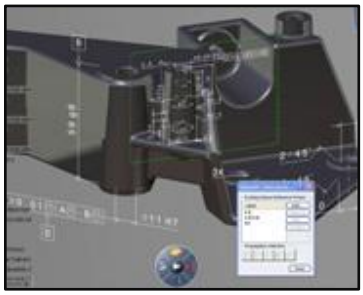


modeling experience that facilitates design and increases productivity with Live Smart Positioning.

CATIA Structure Functional Design delivers a preliminary structure design application to rapidly define, capture project intent, and create structural elements and materials thanks to parametric design, which allows multiple design variations and itera-



tions. Structure engineers are able to obtain early weight, material, labor and balance estimation of the structure as well as the classification drawings including conventions.



The automatic tolerancing and annotation of User Features provides significant productivity gains for defining tolerances and annotations of company standard features.

CATIA EQUIPMENT

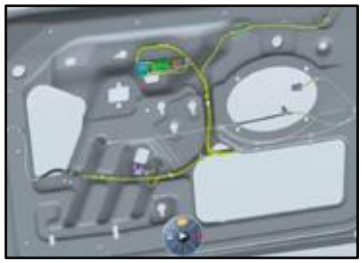
CATIA V6R2011 delivers state-of-the-art wire harness symmetry capabilities, incorporating best practices from

various industries. Users can automatically and interactively perform a symmetry of a complete wire harness while managing unnecessary duplication of reference and greatly reducing design time.

CATIA Systems 3D Logical Design introduces the CATIA Live Shape User Interface for designing 3D Logical Architectures. All tool bars have been removed and converted into contextual balloons that contain only commands relevant to the selected element. Pathways creation is improved with real time geome-

try creation that make modifications seem live and provide 3D architects with a new V6 lifelike user experience.

A 3D Master approach is introduced for the electrical discipline. Large Original Equipment Manufacturers (OEMs) can now extend their 3D Master approach to the electrical discipline by using the powerful ability to dress up the Functional Tolerancing Annotations (FTA) of 3D har-



nesses. They can now focus more on the design by exchanging only their 3D mock-up of FTA data with suppliers, saving the expensive and time consuming documentation process of drawing creation and maintenance.

CATIA V6 DOMAINS AND V6R2011 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.

SYSTEMS ENGINEERING

A collaborative system engineering methodology is critical to address these transformations. CATIA Systems uniquely captures, manages, and tracks product requirements with full traceability, ensuring that these early requirements are met accurately all along the product development cycle, from functional architecture and logical breakdown to physical design and testing.

BEHAVIOR MODELING & SIMULATION

With CATIA Systems, the components from multiple disciplines (such as mechanics, thermodynamics, and electricity including electronics) as well as the numerous interactions between them are modeled on a unique common platform to enable dynamic simulation of the complete system via a virtual prototype. The behavior of the product in operation is assessed while various design alternatives can be tested very early on.

Key V6R2011 ENHANCEMENT FOR CATIA SYSTEMS:

Delivers a Supplemental Suite of Dynamic Behavior Libraries

The CATIA Systems portfolio is enriched with new commercial products that provide libraries to expand creation methods of advanced and specialized models in Systems Dynamic Behavior (DBM). This enhancement allows users to create and simulate more complex models, and quickly find solutions to process- or industry-specific problems.

Displays 2D diagram animation in DBM

Users can now display the 2D diagram animations in the 2D viewer upon a Modelica simulation that contains descriptions of 2D diagram animations. This function provides an intuitive and easy-to-use tool for result analysis.

Manages experiment data for simulation of functional models

Users can create Simulation Objects within the ENOVIA database and attach them to a functional reference entity. They can then save the data related to System Experiment within the database, which allows them to iterate on the definition of those data. This is particularly beneficial during the functional system definition and validation process.

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

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 V6R2011 ENHANCEMENTS FOR CATIA SHAPE:

Introduces ICEM products in V6

CATIA V6 now integrates industry-leading ICEM surfacing technologies, incorporating a highly powerful and intuitive suite of tools to model, analyze, and visualize aesthetic and ergonomic shapes to the highest Class-A surface quality. This full integration supports the hybrid world of both explicit direct surface modeling techniques and associative feature-based methodology within the V6 collaborative environment and dramatically improves the productivity within the Class A-product development process.

This advanced surface modeling solution enables mechanical designers, shape designers and stylists alike to create, validate, and modify surfaces, particularly those which are visible and tangible to customers working within the automotive interior and exterior design and the aerospace lofting and interior aircraft cabin design domains.

Establishes new Functional Shape Design paradigm for the design of Body in White

The Body in White community benefits from a new functional approach that provides a new way of thinking the design, thanks to semantic features, which enable designers to focus on what they want to model and not how to model it. This Functional Modeling technology is unique in the marketplace and provides designers with great flexibility to quickly evaluate multiple design variations, while simultaneously enhancing productivity.

Process-oriented functional features like Hole, Diabolo, Dart, Contact Flange, Stamp, and Bead contain inherent specifications and behaviors that interact in specific ways. Users have the ability to dynamically modify, live, the values of existing parameters of these functional features by using 3D manipulators instead of having to enter edition command panels. in White, especially by optimizing parts for the tooling/die process. You can now rework the surfaces locally in order to enhance their quality and fulfill manufacturing requests. Sewing is an operation combining a surface upon an existing element. This capability adds or removes surface by modifying the existing element. A new feature will sew a given surface on an existing support. The output will be a new skin. This approach eliminates the need to create auxiliary geometries.

Offers 3D data optimization and preparation for multiple purposes

A 3D model can be used in different situations with different levels of definition and details. For multiple purposes, like industrial design, engineering, simulation, or marketing, the preparation and optimization of the data is critical. In some cases, there is no need for the invisible data; you just want to see the exterior aspect and visualize it in a 3D environment. Removing all of the invisible data significantly reduces the size of the model and the computation time of some applications.

The main goal of CATIA Distiller is to take an existing 3D model in the V6 database and to create new ones optimized for specific uses, such as, real-time visualization, virtual reality (VR) experiences, simulation or rendering. It allows you to optimize 3D models through data tessellation, re-organization and simplification. The “distilled” models are uploaded back into the V6 database, and can be exported to other applications, such as, 3DXML files or published on the 3DVIA.com online community.

Introduces Blend Corner command

The Blend Corner command, added to the Shape Design Workbench, provides part optimization of the Tooling/Die process for the Body in White (BiW) solution, by enabling local rework of surfaces to achieve better quality. This is adapted to some particular BiW tooling processes that forbid modifying the BiW part for the design of the tool and need greater round fillets with greater radius and fewer discontinuities for manufacturing purposes. You can now reshape some portions of a support locally without modifying the existing feature history thanks to this new

flexible command, which can be used on top of already existing fillets. Modifying the surface is very intuitive thanks to sliders in the 3D viewer.

Enhances Reverse Engineering User Interface (UI)

You can now quickly and easily create a cloud of points (or a mesh) from a file. Simply drag and drop a digit file describing a cloud of points (scanned or computed) or a mesh in the workbench to create the corresponding entity. This improves productivity by proposing a logical follow-up of the processes. Also, the user interface (UI) is improved, impacting the following commands: Activate, Remove, Split, Align by Best Fit, Power Fit, Basic Surface Recognition. The selection dialog box and the options available in the contextual menu are replaced by a balloon when selecting a cloud of points or a mesh, displaying only the available options.

Enhances Sketch Tracer image edition and update

This provides a way to quickly edit a Freestyle Sketch Tracer image with the dedicated software, and to update the painting when modifications are performed. Using a new icon, the user starts the software associated by Windows to the file type of the image, and CATIA will automatically detect the software closing and update the Painting with the new version of the image. This significalnly improves productivity and ergonomy for the modification of Freestyle Sketch Tracer paintings in V6.

Integrates the new Painter command throughout all workbenches

Through a new Painter command, available in the Part and Surface Design Workbench, the user is able to flexibly and easily apply an effect from a predefined effect library onto every geometrical element of a CAT Shape (body, solid, surface, up to the face level), enhancing the model with superb realism. The visual effect can be removed through the same command. Effects will have the same behavior as existing graphical properties (colors, opacity).

Offers UV Integration in RTR and PH1

Visualizing a 3D mock-up with very large data is always a challenge. V6R2011 introduces a new UVR format that is lighter. As opposed to CGR, UV is not a fixed accuracy file format; rather it contains a low accuracy polygonal representation (called base mesh) which can be refined to reach a given accuracy. UVR does not bring additional behavior specific to rendering, but by reducing the amount of triangles used to represent a shape, it improves performance (file size, memory consumption and download time) and scalability for large data on all the V6 applications. The user is now be able to load a large amount of data on screen, saving a considerable amount of time for preparing data for an interactive review.

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

Breakthrough direct 3D modeling technology opens the doors of creation and 3D experience to new contributors who can quickly and easily sketch any idea in 3D. These new design ideas can then be reused as preliminary shapes by designers further down the process.

DETAILED DESIGN

Highly integrated tools automate the detailed design process and provide a smart management of complex assemblies, from part positioning through mechanical assembly constraints definition to drawing generation and assembly consistency checks thanks to mechanisms simulation.

MANUFACTURING PREPARATION

Advanced process-driven functionalities, such as the automatic definition of complex drafts and fillets to optimize foundry and forge tooling design of forged and

complex cast parts, ensure that manufacturing intent is captured in the early stages of design to avoid manual, lengthy operations, as well as improve manufacturability and productivity.

Key V6R2011 ENHANCEMENTS FOR CATIA MECHANICAL:

Expands into manufacturing preparation

CATIA V6R2011 introduces Bend Part Design to quickly create folded parts across multiple disciplines spanning from sheetmetal to paper boxes. Users benefit from a highly productive universal hybrid modeling tool, which empowers users to put their ideas directly into an industrial context early in the concept phase without going into too much detail. CATIA V6R2011 also introduces Mold Tooling Design, offering users complete tooling process coverage for tooling mold injection designers. It allows users to create standard and company-specific mold bases with ease. Users can also position components and calculate their environmental impacts with ease.

Advances toward lifelike modeling

CATIA Mechanical incorporates functional modeling know-how with the freedom of CATIA Live applications so that mechanical part designers can benefit from Live Smart Positioning of Assembly constraints. This permits live visualization and modification of assembly constraints with a new contextual command. It is also possible to calculate intelligent positioning solutions based on publications and preview and validate any resulting constraints upon smart positioning.

Responds specifically to mechanical in production

Global performance robustness is enhanced with a 30% increase of design creation and modification productivity in Functional Modeling. Latency delays between global systems have been reduced and users can embrace a wider scope of manufactured parts with Functional 3D Modeling. It is also possible to design more complex plastic parts.

In addition, a new dynamic UVR visualization format (replaces CGR) is available and provides better visualization accuracy and reduced memory consumption**.**

Automates the tolerancing and annotations of User Features

Users can benefit from significant productivity gains for defining tolerances and annotations by capitalizing and reusing the tolerancing schema of standard features in catalogues. This also helps to ensure the accuracy of tolerances and annotations.

Finds the most natural position automatically

The user can now launch the Live Smart Positioning algorithm as soon as the mouse is moved over an element. The algorithm combines relationships between publications and maintains the most natural position solutions. It helps the user to preview and to validate any resulting constraints, increasing productivity substantially for positioning product instances.

Introduces Associative Bill of Material (EBOM)

Users can benefit from a new Bill of Material command for both Generative Drafting and 2D Layout products. It offers an automated, associative and customizable functionality that ensures the validity of the consistency between drawings and product definition. It displays the product as it is defined in the CBP/EBOM.

Increases performance for Mechanical in Production with solutions built with synergy

Global performance robustness is also improved with enhancements that increase design creation and modification productivity by 30% in Functional Modeling. Users also experience reduced latency delays between global systems, are able to handle a wider scope of manufactured parts with Functional 3D modeling, and they can design more complex plastic parts. The new UVR dynamic visualization format replaces CGR and provides better visualization accuracy while reducing memory consumption.

Generates 2D drawings with Generative View Styles (GVS)

Users increase productivity by storing the generation specifications at the layout view level so that drawing generation is a one-step action.

Enforces automatic creation of View Texts

Users can enforce automatic creation of View Texts that comply with international standards such as ISO, JIS, or ASME, or with company standard definitions through drafting customization. In addition, users can quickly position the View Text according to a view's overall dimensions, assuring readability.

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

CATIA Equipment delivers 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.

PIPING & TUBING

CATIA Equipment also provides general layout tools for intelligent placement of parts as well as a full set of routing and parts placement methods.

Key V6R2011 ENHANCEMENTS FOR CATIA EQUIPMENT:

Offers live pathway routing

V6R2011 brings a significant improvement in the Designing/Routing pathways user experience. Pathways are now updated in real time to give the impression of modifying them live; the preview is no longer represented by a green dotted line, bringing a new lifelike experience to pathway routing.

Introduces new ergonomic workbench

This provides a totally new, intuitive, and lifelike user interface for the 3D Logical Architecture workbench. All tool bars have been removed and converted into contextual balloons that contain only commands relevant to the selected element. Some commands used for creating 3D shapes are now available in the West Compass tool bar, providing a new, cleaner, and more lifelike experience for the workbench.

Introduces new route component

This enables you to use any PLM Object other than Pathway Segments to route the routable. You are no longer constrained to using Pathway Segments as the only route component. This addresses, for example, specific High Tech requirements that enable you to consider some conducting surfaces are routable by the system.

Enhances spreadsheet user interface

A new spreadsheet-like user interface for the Requirement, Functional, Logical, and Physical (RFLP) data is available. Instead of the 2D or 3D representations, it displays a table of all of the objects in the model, allowing you to browse or edit the existing objects, and create new ones. You can create sub function/logical components, create ports and connections, edit properties, delete, etc. This is very useful if you are dealing with heavy models with a large number of components and links, as the default representations are filled quickly. The spreadsheet allows you to manage a massive amount of data without taking care of the representation and productively perform all of the operations possible.

Generate wire harness symmetry

In many customer processes, the reuse of complete electrical harnesses is a daily need. When many electrical systems are identical for both sides of a vehicle, it is very useful to be able to generate the symmetrical harness with a minimum number of interactions rather than totally redesigning the harness. In V6R2011, the Assembly Symmetry command of the Assembly Design workbench is added to Wire Harness Assembly workbench, and has been specialized to implement specific behavior for electrical parts or products. You now have the ability to automatically and quickly get the symmetry of a complete

harness. This greatly reduces design time and global engineering costs, while ensuring the design quality level of the original harness.

Synchronizes wire harness with logical harness

A new command is introduced that allows users to automatically use the design content/changes from the 2D electrical schematic and the design content/changes from the 3D space reservation in order to create or update the 3D physical design. It is now possible to generate physical harness bundles and branches based on logical pathway segments. Reusing logical information to build physical data will save time and avoid doing twice the work, firstly in the logical design, and secondly in the 3D design. This also facilitates the adoption of the CATIA System application for electrical design since users will take advantage of the easier transition to the physical design compared to working with an external ECAD application, and having to import the data. With this new design change management tool, the logical to physical process is very efficient.

Simplifies FTA creation

With this new command, users can directly select devices, supports, and segments to create the dimensioning, instead of having to select planes and lines. Users can very quickly create multiple dimensions between their selected points in just a few clicks. Advanced electrical dimensioning enables you to dimension a specific path of segments between electrical components. Ultimately, you now have a more productive tool for processing FTA work.

Defines insulation for piping parts

This enhancement allows you to insulate a piping part. It helps you to model thermal insulation and to avoid human contact with the pipe. You are able to define rectangular insulation geometry. This helps you to check that there is enough space for the insulation and that there is no clash with the environment. The insulation process is candidate for specification-driven part placement.

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.