CATIA Version 6 Release 2013x FACT SHEET

The Digital Product Experience

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Value at a glance

CATIA Version 6 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

As the **3D**EXPERIENCE Company, Dassault Systèmes provides business and people with virtual universes to imagine sustainable innovations. Digitizing products and environments opens up new frontiers for all participants involved in a product's life, from creation to usage. Realistic experiences allow users to anticipate product behavior and impact on the environment, thus creating the conditions for sustainable innovation.

The **3D**EXPERIENCE Platform transforms innovation by connecting designers, engineers, marketing managers and even consumers, in a new 'social enterprise'. It powers Dassault Systèmes' Brand Applications, including CATIA, to deliver the full capabilities of 3D Modeling, Content & Simulation, Social & Collaborative innovation and Information Intelligence for industry leaders to build the right business value and experiences for their ultimate consumers.

More and more products are now a combination of physical, software and electronics, in other words, "smart products". This creates increasing pressure on all the specialists who must create these products, not only individually, in terms of their respective disciplines, but *collectively.* Designers, Engineers, Systems & Product Architects and Systems Engineers need multi-disciplinary collaboration like never before in the development of products.

The mission of CATIA is to serve this population. CATIA provides unparalleled user experiences which accelerate the innovation potential in creative design, engineering, systems & product architecture and systems engineering.

The CATIA Digital Product Experience:

The combination the Digital Mockup and the Functional Mock Up makes it possible to experience a product in its geometric as well as its functional view to realize the full product experience. This is the CATIA Digital Product Experience (DPE).

The DPE process is deployed to 4 communities of users:

- Designers
  - Create the shape and attractiveness of the product.
- Scientists
  - Create the proper functional behavior of smart products and components.
- Engineers
  - Responsible for the end-to-end process to ensure the product can be manufactured.
- Architects
  - Define the product and systems architecture and develop concepts.

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CATIA Version 6 R2013x Overview

CATIA Designer Community

CATIA Live Rendering, which makes the creation of high quality photorealistic images as easy as using a camera, now improves performances and offers the ability to render multiple photorealistic images in the background, while the user continues to work on CATIA .

CATIA Live Rendering is an interactive, physically correct, and photo-realistic rendering solution, based on ray tracing engine iray® from NVIDIA®, that is fast and simple to use. CATIA Live Rendering is as simple as a Camera. You select your lighting environment from a library of HDRI (High dynamic Range Image), then apply and tune your material color and settings and interactively choose your point of View. Producing your high end rendering is only a print screen or "save as" image workflow.
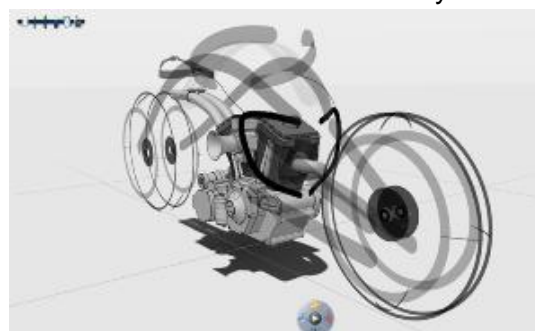

This new highlight offers the ability to render multiple images in background while you are still able to work inside CATIA. The user can continue to work in CATIA while the pictures are rendered, without being be impacted by

performance. All the rendering operations are performed in a background task, and depending on the hardware available render in background will run on CPU, GPU or both on the main computer. Monitoring and managing the queue remain possible even when CATIA is closed.

CATIA Natural Sketch adds new features, reaching a new level of maturity and delivering enhanced functionality which meets the real needs of creative designers.

This enhancement now allows you to use initial rough curves as support, as well as surfaces. It provides you with the ability to draw directly in 3D, directly using the work already done during the drawing of rough curves. The process is more efficient, quick and intuitive: you have not to draw the same model twice anymore.

Color picker - New brushes: This enhancement provides the ability to change the brush and the color of the stroke. You can define the color to create/modify a curve by using the color chooser panel.

Dimension and Position pictures: Natural Sketch allows the placement of images into the 3D world to be used as models or supports. This enhancement allows you to accurately size, measure, scale and position the images in the 3D environment. You have the ability to put some dimensions on pictures and to position them according to each other's position.

A new level of maturity for CATIA Icem. The surface modeler expert now reaches his ultimate goal: achieving the perfect class A shape at the first creation step.
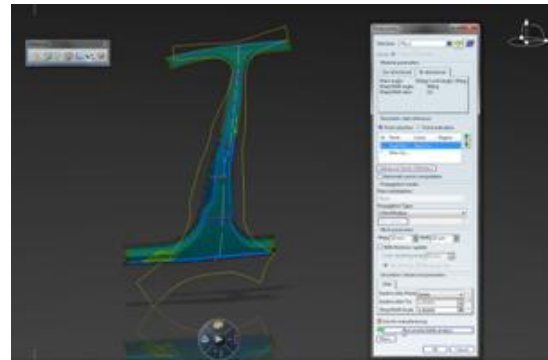

In V6R2013x, CATIA Icem introduces a significant step in terms of functional content but also enables users' daily workflows. Many improvements are provided for daily tools (light manager, ruler, transparency for zebra highlights, iso manipulation in blend surface…), reducing repetitive tasks. The CATIA Icem Expert module now provides a set of major enablers, which make the surface expert get closer to his ultimate goal: achieving the perfect class A shape at the first creation step. Major enhancements are introduced in patch from curve (coupling options), corners (ball and blend corner networks) feature modeling (now automatically retrieving continuity constraints), and flanges (corner manual fine tuning).
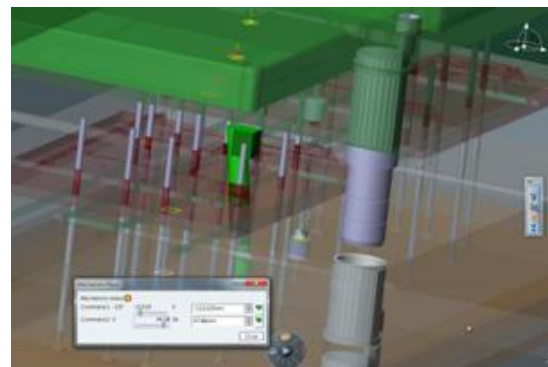
CATIA Engineer Community

Integration of Simulayt Fiber Simulation Technology into CATIA V6

From aerospace and automotive to consumer goods or energy, all industries may need to predict and optimize fiber reinforced materials' behavior, minimizing their weight while increasing their performance.

The Simulayt simulation technology is routinely used to analyse complex surfaces. Its rapid generation of accurate data streamlines the entire composites development process. This adds the proven manufacturability assessment to CATIA Composite Solutions.

Process-oriented mold design workflow – Easy creation of complex mold mechanisms.

Design of the tool assembly is made simpler by guiding user actions through the stages of the mold design process, by enabling easy creation of complex mechanical systems and by allowing fast definition of the plastic injection system.

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CATIA Systems Community

More flexibility for systems designers and more traceability of the impact between components.

Branching and Merging abilities offer an increased flexibility for Systems Designers. They can design on their own and synchronize with others when ready. Navigation among components brings a quick and clear view on impacts.

CATIA V5R2013X delivers multiple advances in systems engineering, the multi disciplinary design methodology which integrates RFLP product definitions. 5 new model libraries, based on the Modelica standard, are added to CATIA Systems.

This improves product design and accelerates time to market by providing a comprehensive set of libraries for the study of Engine, Electrical Power, Hydro Power, Liquid Cooling and Thermal Power systems.

Engine Dynamics Library -Combustion engine system modeling, simulation and analysis, including the complete gas

exchange.

Electric Power Library - Modeling, simulation and analysis of electric power systems

Hydro Power Library - Modeling and simulation of hydropower plant operation

Liquid Cooling Library - Liquid cooling system design

Thermal Power Library - Modeling of thermal power plant operation including transient operation
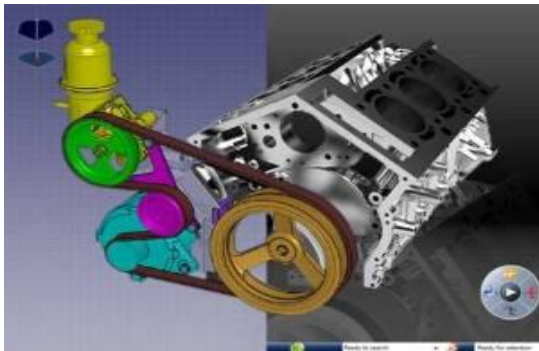
CATIA Product Experience

CATIA V6R2013x demonstrates increased openness with the support for STEP AP242 , Managed Model Based 3D, an automotive and aerospace standard for data exchange and long term archiving. It enables import and export of ISO standard BREP and Tessellated data.

Support of STEP AP242, an Automotive and Aerospace initiative for creating a new standard for data exchange and long term

archiving. AP242 is a merge and enhancement of AP203 and AP214. Support included for import and export of STEP AP242 data. Includes support for tessellated STEP AP242. This provides a vendor independent ISO standards-based approach to exchanging compressed and tessellated data.

With V6R2013 we announced the industry's best multi-version compatibility, allowing V5 and V6 users to share and edit parts at the feature level. CATIA V6R2013x extends the scope of this capability to include features created in the Freestyle workbench.

V6 parts created in V6R2013x containing features created in the FreeStyle workbench can be transferred with all their specifications to V5-6R2013. FreeStyle features are now included in V6 to V5 "As Specifications" export

Kinematic Mechanism specifications can be exchanged from V6 to V5 including Joints, Commands and Dress-up. Once migrated, mechanisms are ready to be simulated in V5.

When exporting FTA data to V5, FTA data, the links between FTA features and geometries are maintained.

For additional flexibility and mobility, travelling users who don't have an internet connection can now continue to design with CATIA V6R2013x even when they are not connected to the server. Once back online, un-checking the "work offline" switch reconnects to the server and work can be re-synchronized and shared.

With CATIA Offline, "On –the-go", users can access their current data even when it is not possible to connect to the enterprise ENOVIA V6 Server. Simply selecting "work offline" downloads the required data. CATIA function will then be available "on the go", when working from home or when no network is available. Once back on site, un-checking the "work offline" switch reconnects to the ENOVIA V6 Server, and work can be re-synchronized and shared.

CATIA Version 6 R2013x Enhancements

CATIA Designer Community

Industrial Design – Reverse Engineering – Mesh to Brep.

From a tessellation mesh, you can built exact topology, thanks to this new command, even if you don't have construction history. You can then modify them with live shape, are do some drafting with it. The meshes can come from 3dvia.com or from other software or from former CATIA versions. Before you had to do it manually, with a command called basic recognition, so it was tedious. So now this saves some time. Exemple of usage : When you have only the CGR of old data model and you need to do some maintenance, or remodel some assemblies, you need to re build the B rep to have exact characteristics.

Industrial Design – Distiller - User interface Ergonomic & workflow enhancement

This enhances and refreshes the global ergonomy of the Distiller workbench. Several panels have been updated (Decimation, HSR, statistics) and global access to the application have changed. Provides to the user a simple way to browse and edit structure of a distilled product. The object of this enhancement is to describe the view in the distiller workbench which will allow the user to easily edit the distiller product structure. This view is based on the similar Live Compose-Structure view. It shows the product tree-hierarchy on turntables and the user may rearrange the structure through drag-and-drop interactions on these turntables.

Industrial Design – Imagine & Shape – Circular alignment

This enhancement provides a new option of alignment allows aligning circularly the selected summits. You now have the ability to align vertices along a dynamic circle. This new option is added to the modification tools palette, when alignment mode is active.

Industrial Design – Natural Sketch – Trace on Sketch

Natural Sketch allows users to draw two different kinds of curves: rough curves and trace curves. Firsts are like "drafts" for the second ones. However, the user was not able to draw on them: for creating trace curves above rough, he had to redo all the work he already did for the first step ie select plane, draw, select another plane… This enhancement now allows you to use rough curves as support, as well as surfaces. It provides you with the capability to draw directly in 3D, not using the plane selection step (the work being already done during the rough curves drawing). The process is more efficient, quick and intuitive: you have not to draw twice the same model anymore.

Industrial Design – Natural Sketch – Dimension & position pictures

Natural Sketch allows you placing images into the 3D world to be used as models/supports. This enhancement allows you to size, measure, scale and position accurately the images in the 3D environment. You have the ability to put some dimensions on pictures & to position them according to each other's position.

Industrial Design – Natural Sketch – Photo sketching

Natural Sketch allows users to draw 3D models. However, most of them are more used to draw in 2D. It is necessary to give them the possibility to import their 2D models in Natural Sketch. This enhancement provides the ability to automatically extract some rough sketches from a picture. You can now import an image, launch the sketch from picture command, select an image support, and a set of rough curves representing the image content is created.

Technical Surfacing – Generative Shape Design – Pattern a multi selection

This provides the capability to pattern in Geneative Shape Design workbench multiple features in one go, through a single pattern command like it was already possible on the solid pattern in Part Design. The user will be able to transform a selection of multiple GSD features (either all surfacic or all volumic, but not mixed), through the pattern command. This functionality will greatly add to the productivity of the user.

Technical Surfacing - Shape Morphing: Multiple limiting elements

This provides the capability to define multiple limiting elements to restrict the deformation. In addition to enabling to create shape morphing that had not been able to be made so far (some configurations with more than one limiting element), the highlight enables to create more complex shape morphing (i.e. with more than one limiting element) in one single step, while beforehand the user had to perform several steps to get the same result.

Generative Shape Optimizer - Surface Simplification: Deviation diagnosis improvements

This enhancement improves the precision to the diagnosis of the deviation for Surface Simplification command. This highlight provides a way to visualize for each type of deviation, the points that exceed the maximal deviation. Each point indicates a percentage of the maximum allowed deviation value. This new functionality helps the user to visualize the locations where deviation is high. Then he can do some operations around those locations in order to reduce the deviation.

Live Shape enhancements for more productive product design

Designers benefit from a better integration between Live Shape design and traditional Features. Design intent is retained: in addition to fillets, there is a new declarative chamfer and declarative pattern evolutions make it complete. Constraints can now be set on edges and vertices, in addition to faces. The user can structure his design with rigid blocks which group bodies to set constraints between them.

Moreover, as constraints are converted when splitting a Live Shape design into new parts, the designer can start design in CATIA Live Shape, and switch to CATIA Live Compose or Assembly workbench without losing specifications. A 'first draft' design can smoothly evolve into an organized assembly structure.

CATIA Systems Engineering Community

Five new model libraries complement the CATIA Systems Engineering behavior modeling and simulation tools.

The Engine Dynamics Library brings Combustion engine system modeling, simulation and analysis, including the complete gas exchange,

the Electric Power Library for modeling, simulation and analysis of electric power systems,

the Hydro Power Library for modeling and simulation of hydropower plant operation,

the Liquid Cooling Library for Liquid cooling system design,

and the Thermal Power Library for modeling of thermal power plant operation including transient operation.

More flexibility for systems designers with Branching and Merging abilities

With these new abilities, systems engineering becomes more flexible and ensures collaboration more easily. Systems designers can define and simulate systems components on their own, in their own workspace, and synchronize with others when ready.

Navigation among components offers a quick and clear view on impacts between components

The related objects panel brings more traceability on impacts as it gives a global relational view of a system object. Systems engineers can navigate on Parents/Children relationships, Implemented / Implementing objects, Connected objects, or they can follow Exchanged data. The highlight of objects related to the selected entity provides them with a clear view of the Functional and Logical connection network. The different levels of connection are visible as colors display them according to their nearness.

Accurate exploration & fast open with Predefined Configuration Filter

Configuration is taken into account more widely. Thus, any collaborator benefits from an easy access to the RFLP (Requirements – Functional – Logical and Physical) definition for a given configuration by the application of a predefined configuration on all the explored or opened trees.

CATIA Engineer Community

Electrical Design -Torsion Management between Design and flattening

One of the most critical steps for the harness manufacturing process is to ensure the final harness will be installable in its environment without any problems.

By orientation of the support and devices in the flattened harness extra torsion is introduced in the flattened harness on top of the torsion present in the design harness. The user needs to minimize this relative torsion between the design harness and flattened harness to ensure installation is possible.

This highlight gives relative torsion information between the 3D Harness and 3D flatten data. It also provides an interactive and visual decision-making tool to analyze and fix these twisting issues before the actual installation. By defining maximum acceptable torsion values you are now able to visualize in one shot the most critical portions of the harness. This development also provides an interactive tool to fix these twisting issues before the actual installation by integrating "Component Orientation" command with the "Analyse Torsion" command.

Wire Harness Documentation & Formboard - Associative Generated Device Section

Device section views are created because the Front view is not able to provide information about relative angles between devices in a device assembly. After rotation of a branch in 3D, front view and section view get updated. Now with this new development the viewing direction gets updated according to the manipulation in 3D Flatten data, so that the section view still remains relevant. It enables the correct update of section view according to the changes done in 3D Flatten data. This will increase the productivity of customer.

Wire Harness Documentation & Formboard - 3D Display of Electrical Attributes

This highlight provides a way to share necessary information to the supplier without exposing the data model through CATIA or 3DLive under Review mode. The goal is to give access to the attributes of the branches, segments and protective coverings and to display them for wide sharing through 3DLive (connected or not) for review. It enables quicker harness data review between OEMs, manufacturers, suppliers. During the review, through single selection it is possible to access to a cross-highlight and a frame display. The "bubble" messages will contain the attributes and the knowledge attributes set on the feature and their irrespective values. This mechanism will be active only under the Live FTA Review workbench. Automatically and with transparency, the users will be able to export their data through .3dxml files (For Authoring, For Review) with the necessary information for the suppliers to make the harnesses. Suppliers will import the .3dxml files in their 3DLive environment and will be able to review the customer data and access to the necessary information for the manufacturing.

Wire Harness Documentation & Formboard - Smart Synchronize Simulation

A new Immersive Panel U.I is created for reporting the modifications during Synchronization process and as this report will be a part of CATIA UI, interaction between report and 3D elements of the model will become possible. It provides facility of cross highlight in both 3D harness and Flatten data when click on a particular element in report. It also provides facilities of sorting, filtering, search, exporting the report in excel format, etc. The new UI for reporting is much more user friendly and provides live feedback on every action performed during synchronization. It enhances user productivity as user does not have to re-run command again for running multiple simulations.

3D Electrical Design – Arrange Junction Command

It is normal scenario in industrial practice that a large bundle of wires that start from a Energy input source get divided into more than one sub-bundles which end in different devices. This development introduces a lean and intuitive way to arrange branches at junctions by providing a new 'Arrange Segments at Junction' Command. This development is

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positioned in the design simulation of Electrical Network, especially where Bundles further subdivide into small Sub bundles of wires each continuing to different devices in the network. Thanks to the capability to arrange at Junction, a realistic view can be created with less manual efforts and the harness can be analyzed close to reality situations.

3D Electrical Design – Symmetry : avoid over constrained assemblies

The combination of the "fix symmetrically" constraint and the electrical engineering connections sometimes generated over constraint system and an error when updating the assembly. A new development has been integrated in the assembly symmetry process to avoid an over constrained configuration. Even is the connections can be manually deleted by the user to recover a correct situation, an enhancement of the assembly symmetry has been developed to provide a correct result to end user: the symmetrical electrical parts must be correctly connected and no over constrained system will be generated.

3D Electrical Design – Auto insert mode in the Branch routing panel

This enhancement provides a new mode to automatically find a position in the routing elements lists to insert a new routing element. Thanks to the new Automatic routing mode capability, you don't have to choose the position of insertion in the route of routed elements to insert one element in the route of the branch. With mode "add auto", the element is inserted automatically at its best position in the branch route. This enhancement will give a better productivity to the branch routing capability.

Piping & Tubing Design - Export .pcf to ISOGEN or other software

You can now generate the .pcf file from the piping designs from CATIA V6, which contains the information of piping design and is to be used by ISOGEN to create piping isometric drawings (.dxf), using their stand alone application I-RUN. These in turn are used to create (import) ISO metric drawings. This will help the user to speed up the design of piping systems, .i.e. drawing generations.

Piping & Tubing Design - Multiple catalogs define in Project Resource Management(PRM)

This enhancement allows you to declare multiple catalogs in project resource management (PRM). In Industrial cases with some partners, the Piping/tubing catalog is not unique, so it was critical to provide this capability to use in the same time multiple catalogs. A new type of immersive dialog box is also introduced. It is possible to see through this dialog box the back ground and give the feeling that the dialog box takes less space in windows. A new capability of Cache memory also makes it possible to put a software component always accessible to avoid a long elapse time to get the resources.

Piping & Tubing Design - Add Flexible pipe/tube in Spool

The spool can contain some piping part and rigid pipe/tube. With this development you can now also manage the flexible pipe/tube inside the spool. This new function provides the capabilities to create new spool from Flexible pipe and to Add/remove Flexible pipe in Spool.

Piping & Tubing Design - Transfer of component between spool

This enhancement provides the capability to move directly some component between two spools. The targeted spool can be an existing or a new spool. The link (contextual link, engineering connection and implement link) is updated with the same level of capability of drag and drop function. This capability was already possible with the standard drag and drop function but without the intelligent selection. This is important to allow an easy way to modify the spool.

Generative Piping & Tubing Design - New Merge Line ID command

This enhancement delivers new Merge Line ID Command, which helps user to merge a Line ID into another Line ID (both Line IDs exist in the current document). The command will work same in PTD and PLE workbench.

Integration of Simulayt Fiber Simulation Technology into CATIA V6

Real-world components incorporate awkward features like sharp corners, splits and rapid changes in curvature. These requirements routinely trap fiber simulation tools that have not been developed with real geometry of complex surfaces in mind. For example, a typical Formula 1 monocoque contains over two thousand separate plies. These structures are designed in the space of two weeks, so product performance is critical.

Proven manufacturability simulation technologies, such as sophisticated fiber simulation algorithms, advanced propagation modes, instantaneous flat pattern capabilities and real-time simulation yield excellent results for commonly-used woven fabrics used for composites structures. The accuracy and resolution of the surface analysis increases with the surface curvature to avoid errors due to fiber bridging. A precise control of the draping is given by a Flexible Seed Point and Seed Curve constraints. Engineers can optimize manufacturing processes and ensure producibility assessment quickly and accurately.

Process-oriented mold design workflow – Easy creation of complex mold mechanisms

A more productive and intuitive user interface reflecting stages of the mold design process enables to display only useful information and provide only relevant commands. Complex mechanical systems such as Sliders, Lifters, Interlocks are easily created in the tool assembly thanks to user assistance in defining proper connections and kinematic interactions.

A new 3D-driven dedicated function makes design of the plastic injection system much less time-consuming. The Mold Assembly workbench and the integrated Tooling systems design enable compatibility with off the shelf or customer specific systems.

Table templates and BOM evolutions bring 3D Master enhancements

A more accurate view of the Bill Of Material (BOM) is now possible within CATIA through the inclusion of parts which do not require CAD design, such as grease or glue. Some other enhancements make the BOM more productive with the ability to create quickly an inverted BOM by using default mode or to customize the display of deleted parts.

In addition, templates can be used to automatically fill any table of a Drawing. These predefined and reusable tables may contain simple information, such as on site fabrication instructions, or product related information, such as title blocks or revision blocks.

Manage efficiently annotations presentation using views and captures

Dedicated captures can be created as visual filters related to each view. Engineers can make a relevant capture more visible by setting it as current when activating a view. And they can present several annotation plans at the same time thanks to axonometric views.

Layout View Filtering for a more productive 3D Master approach

When working on large and complex 3D Assemblies, users can use a combination of predefined filters to create specific visualization views. This simplifies the work of the designer and increases productivity. For individual layout view creation each component had to be manually added to a visual filter. It is now possible to create a mask or display filter by choosing one or several PLM Filters ,in addition to explicitly filterable elements, and by applying a visualization mode to each selection. It enables the use of PLM filters that might have been created by the person responsible for the components or which may have already been used to define the Favorite Context of the layout.

Performances and memory consumption gains in Mechanical Systems design (Assembly)

Kinematic simulation and the realistic movement of constrained assembly parts can be performed with greater speed. This is achieved by enabling the simulation to take place without loading the full 3d Shapes content, thus reducing memory and processor requirements.

Completion of Bend part design constraints

Sheetmetal designers can make their design intent persistent with constraints between Walls. The position of all faces from the sheetmetal solid can now be controlled by constraints. Even top and bottom faces for which unfold transformations are impacted. Wall position can be specified with five different types of constraints and variants to move or not mobile parts together with the wall.

Accelerated design of Surface Flanges for Aerospace Sheetmetal products

With the new support of complex surfaces flanges, designer can create them directly, without the need of additional Cut Outs, even in case of complex Sides shapes. Side and joggle compensation can be integrated. In addition, any sheetmetal features can now be upgraded easily.

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

http://www.catia.com/

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