CATIA V5R21 - FACT SHEET

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INTRODUCTION

CATIA V5 is the leading solution for product success. It addresses all manufacturing organizations; from OEMs through their supply chains, to small independent producers. CATIA can be applied to a wide variety of industries, from aerospace, automotive, and

industrial machinery, to electronics, shipbuilding, plant design, and consumer goods. Today, CATIA is used to design anything from an airplane to jewelry and clothing. With the power and functional range to address the complete product development process, CATIA supports product engineering, from initial specification to product-in-service, in a fully-integrated manner. It facilitates reuse of product design knowledge and shortens development cycles, helping enterprises to accelerate their response to market needs. In conjunction with ENOVIA for collaborative product lifecycle management, SIMULIA for engineering quality and DELMIA for production performance, CATIA V5 is a key component of V5 PLM.

WHAT'S NEW AT A GLANCE

- **Support customers in production for production and quality improvement in all domains and industries**
- **Reach functional completeness for CATIA ICEM Design products**
- **Ensure easy transition to CATIA Version6**
- **Best in class machining strategies**

OVERVIEW:

CATIA V5R21 improves CATIA Sketcher for a better accuracy of design in context

CATIA Part Design brings time saving in Conceptual Design phase thanks to new associativity that exists now between reflect lines or internal profiles and the product shape. You can then refine accurately a design with 2D geometric details from the associative silhouette of its 3D faces/features. The area included in this outline can also be used to show a specific treatment area. Projection and intersection curves provides user with the same ability.

CATIA V5R21 enables exchanges of conceptual study data through external 2D formats.

Conceptual study may starts from 2D legacy data, with the direct import of DXF and DWG files, dealing the content of each 2D layer into a specific sheet. The concept can then be studied taking advantage of the overlaid sheets, as well as of any relevant 3D data. Generating all this data into a single drawing sheet ensures sharing it using 2D external formats whenever needed.

CATIA V5R21 delivers more flexibility for design pattern annotations

CATIA V5R21 brings more flexibility to the user in the definition of tolerances and annotations of pattern of holes. Each pattern instance can get its own semantic annotation. Distance and position annotations are automatically proposed to the user during the pattern creation step, thus providing a significant productivity gain.

CATIA V5R21 delivers top quality surfaces achievement for body work in the automotive industry, with superior user experience from CATIA Imagine & Shape

CATIA Imagine & Shape, the virtual clay modeler based on subdivision surface technology, delivers world-class surface tools to address body shape details such as cutting face plus superior usability tools for designers. Designers take also advantage of real-time rendering visual feedback during the creation phase. This makes CATIA Imagine & Shape fully suitable for body design in automotive industry.

CATIA V5R21 reach functional completeness and V5-V6 feature equivalence for CATIA Icem products

CATIA V5R21 extends its advanced and powerful freeform surface modeling, modification and analysis capabilities within the Class A modeling domain. V5R21 improves the Quality of the produced surfaces with more efficient tools to validate shape quality. User efficiency & Workflow are also improved thanks to a dedicated modeling environment embedding an adapted user interface. This completes the feature set alignment of V5R19, V5R21, and V6R2012.

CATIA V5R21 keeps improving mechanical surfacing design technologies with Loft command enhancements.

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.

CATIA V5R21 improves Harness Flattening ergonomy and robustness for Wire Harness manufacturing companies

This release delivers Ergonomic improvement through a new 3D Label selection process during the user manual mapping step for better user productivity. The Synchronies command has been enriched with a better branch management and can now also properly handle hybrid (support + mechanical part) parts components.

CATIA V5R21 delivers best in class machining strategy and continue to enhance machining simulation for better productivity gain

This release brings new machining strategy dedicated for Hard Material machining in order to increase the tool life duration. Numerous enhancements Multi-Pocket Flank Contouring such as feed rate reduction in corner or Control overlap for close tool path for unmatched programming and machining time reduction

CATIA V5R21 yet enhances its openness by supporting new format of CAD

New format support includes ProE WF5, UG NX6, SolidWorks 2009, SolidWorks 2010, ACIS V19 and Parasolid V21

CATIA V5 and CATIA Version 6 transition

To ensure that our customers can be as successful with deployment moving from V5 to Version 6 as they have in moving from a V5 release to another, DS delivers upward and downward compatibility between CATIA V5 and Version 6 data equivalent to the compatibility between two releases of V5.

To enable unmatched collaboration, within a company as well as between OEMs and their suppliers, Version 6 has been engineered to support relational design collaboration between all supported CATIA V5 releases and Version 6.

To protect customers' investment in their existing V5 projects, DS plans to continue the development and support of V5.

DETAILED DESCRIPTION

CATIA Shape

Materials on previsualization for CATIA Imagine & Shape product
This previsualization now does inherit from all graphic attributes and materials put on the feature and on its topological sub-elements. All view modes are also available, so that the previsualization is exactly the same as the feature's representation.

New command for Mesh Creation / Edition in CATIA Imagine & Shape
You can create a new base mesh or modify an existing one, enriching the possibilities of mesh modification with the management of n sides or faces. The command offers various tools for modifying this mesh, including Add Face, Move Vertex, Cut Face, Cut by Plane, Merge Vertices, Regularization, and Deletion of Face, Edge or Vertices.

Obj files import command available in CATIA Imagine & Shape
This provides a way to import "Subdivision Surfaces" coming from other software through OBJ files. It adds a simple command that permits you to select the obj files you want to import, see a preview of the imported surfaces and meshes, and choose the normal orientations and global model

Loft with G2 continuities for mechanical surfaces creation
This enhancement provides in the Loft command the capability 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. The curvature continuity is particularly expected by people designing plastic parts.

Loft with punctual extremity for mechanical surfaces creation
This new capability enables you to create a surface with a punctual extremity in one shot, rather than building this shape with several other features. In the Loft command, one section and two guide curves intersecting each other at one extremity will be enough to create such shapes, increasing your productivity.

New option for the Sweep command with untwist at C1 discontinuities
This new option of the Sweep feature allows a 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 benefit natively from the sweep removal or fill possibilities for the twisted areas. This increases your productivity because C0 vertices areas can automatically be filled or removed without any manual handling.

Offset with Local Regularization option
In some cases, the Offset command cannot compute the offset surface due to a local curvature defect or a pointed surface. A new option is added to improve the robustness of the Offset command, providing the ability to regularize the offset result locally instead of globally. The local regularization option improves the number of successful computations by adding some local adaptations to the offset surface. With the new option the offset result is closer to the offset theoretical result, but is also more topologically complex with additional sub-elements.

Global Deformation is now available on Wires
This enhancement extends the possibilities of several GSO commands (Bump, Shape Morphing, Wrap Curve, Wrap Surface). Historically such functionalities were only applying to surfacic inputs. Now it is possible to deform the same way a wireframe skeleton. This enables additional usage scenario. Typically it could occur that the surface deformation was leading to a non suitable surface quality. Now you can deform the wireframe skeleton instead and rebuild surfaces on the deformed result.

Simplification Surface function enhancements
The new Surface Simplification functionality allows you to simplify the topology of a surface within given deviations tolerances. The simplified surface will better undergo downstream treatments or downstream operations used in the industrial processes. This will lead to a better robustness and better performances.

Automatic Surface Reconstruction Enhancement
A new option is added in the automatic surface reconstruction command panel to obtain more fluent stream lines on the resulting surface. The objective is to improve output surface quality and aspect, computation time, and to reduce the weight of the output surface. This new option is not available in case of closed mesh.

CATIA Mechanical

CATIA V5R21 improves productivity by updating only impacted views after 3D graphic attribute changes:

After the modification of the graphic attributes of a 3D body, only generative drafting views in which the modified body appears require an update. This improved update management reduces significantly the number of view or drawing updates.

CATIA V5R21 strengthens the standardization of your drawing with additional representations from ISO 14405-1, ISO 10135, AWS A2.4, ISO 2553 and JIS Z3021. Increase the standardization of your views with additional representations from these standards. This includes the upgrade of Fonts DSISO1 and SYM2, additional engineering symbols for linear sizes characteristics, application zones and welding symbols.

CATIA V5R21 brings more efficiency with associative offset of 3D elements and 2D background's geometries. A new command "Offset 3D Elements" allows user to create a 2D associative offset of those elements, saving time after design changes, when later elements depend on the offset curve.

CATIA V5R21 delivers Multi-domain Use-Edges to simplify their management: Projections and intersections of the 3D model can be associative regardless of the number of domains generated, improving conceptual design associativeness to actual 3D model.

Increase productivity and flexibility when importing IGES 2D files directly in a layout. This new import of data in the IGES 2D format enables user to enrich 2D layouts of the 3D representation from external resources. This leverages the exchange of data between different CAD vendors.

CATIA Equipment

Auto Update Flatten Links Ergonomics Improvements
During the Auto Update Flatten Link process, when algorithm stops and asks the user to manually select the appropriate object to map, a new behavior is added to the labels displayed in the flatten data editor to improve the usability of the Auto Update Flatten Link command.

Synchronize Branch with Inverted Route Orientation
This enhancement handles in Synchronize process the branches whose route orientation has been inverted in harness data. This helps the user to synchronize their flatten data with their modified 3D data with more confidence.

Handle Hybrid Parts in Synchronize algorithm
The goal of this function is to manage properly the supports with publications used for branch routing in the EHF process (Extract/Synchronize). As of today, the support are handled, the mechanical parts are handled but not this "hybrid" support-mechanical part.

This enhancement helps the user to use the product with more confidence since the Hybrid Part will be now handled properly.