



CATIA V5R20 - 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

- **Native Class A Automotive Surfacing Within V5:** ICEM Shape Design products are now delivered on standard CATIA V5 media supporting traditional V5 installation, licensing and administration procedures.
- **CATIA Composites Excellence:**
 - The new Light Manufacturing Part Generation product notably enhances productivity and collaboration by allowing users to work concurrently on the same composite part, later merging and synchronizing separately designed stacks into a single manufactured composite part. .
 - A new grid-based design methodology automatically generates plies by taking each cell's composite specifications defined during simulation and applying composite design best practices.
- **Enhanced STEP Interface:** CATIA V5R20 enhances openness and collaboration by introducing a unique new product, CATIA Extended STEP Interface. It advances large assembly archiving with nested assembly support and will include full composite data and FT&A in the standard STEP format. This enables long-term archiving of this data, as well as improves data exchange between different CAD formats. Mechanical
- **Part Design enhancements:** In particular, progress in functional modeling (from plastic to cast and forged parts) now addresses Powertrain processes, as well better collaboration on complex parts. Functional modeling allows users to design oil pans, gear boxes, or motor brackets up to 40% faster.

OVERVIEW:

Deliver CATIA integrated solution with CATIA ICEM Shape Design to fully address Automotive Class A

Delivered on standard CATIA V5 media supporting traditional V5 installation, licensing and administration procedures, ICEM Shape Design (ISD) V5R20 now becomes an integral part of your CATIA V5 deployment. ISD R20 extends its advanced and powerful freeform surface creation, modification and analysis capabilities within the Class A modeling domain.

Accelerate, unify, and streamline the conceptual design and surfacing workflow with new Imagine & Shape features

The powerful Subdivision Net Surfaces feature in Imagine & Shape allows users to combine the curve-based approach with subdivision surfaces clay modeling. This feature helps increase design quality and empowers the Creative Designer with greater capabilities. Imagine & Shape is particularly beneficial to the styling centers/design studios in the transportation, shipbuilding, product design, and life science industries.

Support composites data in the standard STEP format and increase productivity with Light Manufacturing Part Generation

CATIA V5R20 is the first solution that supports composites data in the standard STEP format. As a result, long-term archiving of composite data is becoming a reality, especially for the aerospace industry. This new support of composites in the STEP format also leverages the exchange of data between different CAD vendors. In addition, users can greatly increase productivity with Light Manufacturing Part Generation, which ensures that manufactured parts can be quickly created and synchronized with designed parts. This solution is especially beneficial to the aerospace, automotive, wind energy, and shipbuilding industries.

Improve design productivity and ensure high quality design with enhancements of Mechanical Part Design products and tools

The Functional Modeling Part product has been enhanced to address the design processes of customers in the powertrain domain as well as to support complex part design. The DS Functional Modeling breakthrough technology allows users to design oil pans, gear boxes, or motor brackets up to 40% faster. Enhancements have also been added to ensure fillet robustness, further increasing productivity and reducing design modification costs. In addition, the Wall Thickness Analysis tool enhancements allow users to assure design quality and manufacturability as well as increase productivity. All of these enhancements are particularly beneficial to the powertrain domain of the automotive industry, but are also useful across all industries.

Visualize complex multi-plane internal section views

This feature allows users to cut a part along multiple planes. Afterwards, they can instantly visualize several internal features, such as holes or slots, in a single view to better understand the geometry and all of its annotations. This instant display of complex views requires no computation and helps users to work faster. It is valuable across all industries.

Improve ergonomics and user productivity with Flex Simulation, Harness Installation and Harness Flattening enhancements

Harness analysis, and filtering and sorting capabilities in the device list have been improved to enhance ergonomics. In addition, knowledge parameters on bundle segments in harness flattening can be synchronized. These enhancements, among others, are particularly beneficial to the aeronautical and automotive industries.

Reduce programming and machining time

Two new features - material removal simulation and advanced finishing - reduce programming and machining time, respectively, so companies can save time and money. The material removal simulation feature reduces programming time by helping users to better understand the IPM (In Process Model) with the use of color coding. The advanced finishing feature reduces machining time by providing an optimized finishing route in only one operation while taking a dedicated strategy into account for the vertical and horizontal areas. These features enhance the machining process across all industries.

NEW STEP product for advanced functionalities

The CATIA - Extended STEP Interface enables long-term archiving with full validation properties and nested assemblies. It also makes it possible to manage very large assembly structures thru STEP thanks to nested assemblies support. This feature especially benefits the aeronautical and automotive industries.

Comply with FAA (Federal Aviation Administration) certification policies

The 3D Insight product has been developed to comply with FAA certification policies which require there be only one model and one modifier, a primary engineer, during the entire development, deployment, manufacturing and management lifecycle. This feature applies to the aeronautical industry.

New rule based meshing for creating high quality meshes

A new product, SIMULIA Rule Based Meshing, is available to automate the process of creating high quality surface meshing for all workflows that use CATIA meshing tools. The new product gives the user a means to specify globally the desired meshing treatment of entities such as holes, fillets, and beads. It also gives the user a means to specify acceptable element quality criteria, such as minimum edge length, aspect ratio, and skewness. Once the complete set of meshing rules has been specified, no additional user involvement is necessary, as the actual mesh generation is completely automated.

New analysis capabilities

Designers and engineers are able to use the new capabilities in CATIA Analysis and the SIMULIA Extended Analysis products to lower costs while accelerating the evaluation of how their products will perform under a variety of real-world loading conditions. The simulation results can be used to make design modifications to improve their products overall reliability and performance.

DETAILED DESCRIPTION

CATIA Shape

ICEM Shape introduction in CATIA V5 portfolio

Sustaining its principles of seamless data flow using one common data format throughout the CATIA V5 Portfolio between all phases of the design cycle, ISD R20 extends its advanced and powerful freeform surface creation, modification and analysis capabilities within the Class A modeling domain. Protecting your investment from previous releases and supporting seamless migration, ISD R20 further adopts and supports CATIA V5 trusted methodology, and proven update mechanisms

Combine curve-based approach with subdivision surfaces clay modeling

CATIA Imagine & Shape offers a new set of functions that accelerate and unify the concept design to surfacing creative workflow. A powerful Subdivision Net Surfaces feature allows users to combine the curve-based approach with subdivision surfaces clay modeling. These new Imagine & Shape features increase design quality and empower the Creative Designer with greater capabilities.

Alignment of points along any geometry

The Alignment command allows the alignment of vertices on plane or line defined by the compass. The new enhancements of the Alignment command extend the possible supports of projection; it provides a way to project some points of the base mesh onto any kind of geometry (curve, surface, etc.). This offers the user more freedom to use styled geometry as input for the design process.

Automatic Surface associativity

The Quick Surface Reconstruction workbench provides a fully automatic command in order to quickly build surfaces fitted on meshes. This Automatic Surface Reconstruction command is now associative. This enables interactive edition and automatic update of the surface in case of input mesh modification or replacement, increasing user productivity.

Mesh selection and propagation for Surface Creation

This function improves the user's selection on meshes. You can now quickly select plane areas on meshes. Two enhancements for mesh selection:

- Within Activate command, Flood could be achieved with an angle criterion
- Within PowerFit command, user can activate sub-part of mesh to avoid counter-draft treatment during the approximation

Shape Creation Enhancements in FreeStyle Shaper

It is now possible to more precisely control the styling fillet. The dialog box has been redesigned, the fictive display command as been enhanced (only ribbon is taken into account by fictive display), a new support for fictive edge has been introduced (no real intersection between support needed), as well as multi-result management (as in GSD).

Shape Modification Enhancements

Offers many new enhancements for control point functionality, extend command, and styling extrapolate command. The Break creation process is made easier, with more

possibilities for the creation of the datum feature, saving the time. New evolution of the matching constraint command will add more functionality to the command in order to improve productivity.

Isolate for all GSD features

This provides the user with a way to isolate any GSD feature. Isolating a feature means breaking any links with its inputs, the feature becomes a datum feature that cannot be updated anymore. Currently, there is no direct way to isolate any type of Generative Shape Design feature. The old process to perform the functionality was to copy as result the geometry of the feature and afterwards to replace this feature by its copy. This was a long and fastidious process, now spared thanks to the new functionality.

Fillet - Blend Corner robustness

It is now possible to define a blend corner with a setback exceeding the initial edge.

Dynamic sectioning in Part & GSD

This provides a dynamic sectioning command. The user has a tool to cut a part at a position he will choose dynamically by moving a 3D section directly in the 3D viewer. This functionality can be useful during the design itself, to visualize the being-designed-part by section, or at review time, to detect potential section issues.

Composite creation of Light Manufacturing Part offers dramatic improvement of Manufacturing Part generation

Creation is possible 'as result' of ply contours to ensure the manufacturing part is exactly the same as the design part. In addition, only mandatory objects are transferred to manufacturing parts, which reduces the number of transiting object up to 90%, increases performances of generation/synchronization, and increases robustness. Plus, a manufacturing part can still be modified and synchronized easily.

CATIA Mechanical

Section & Thread management

Even though threads definition does not generate geometry, CATIA Wall Thickness Analysis is able to optionally take existing Thread and Tap features into account in order to measure real thickness material. Moreover, since users are able to section its geometry, they can dynamically store sections in the Wall Thickness Analysis command, in order to save critical zones and to get a quick access to them. They can be used for review during design, thanks to status, comment and thickness values.

2D Layout for 3D Design - Multi-plane section views

Offset and aligned section views or section cuts can be created in layouts, rather than in drawings only.

Such multi-plane views can be created either from an FTA view or capture, or from a polyline located in an existing layout view. In offset section views, parts are cut by several parallel staggered planes, in order to display the hole silhouettes in a cylinder head for instance. In aligned section views, the part is cut by secant planes, and drawn as if this cutting surface was unfolded.

3D model is displayed according to the cutting profile, using 2D Layout view filters to insure proper display of both 3D geometry and annotations.

Boundaries' objects can be created and edited to get relevant display of cutting profile corners in the multi-plane view.

These multi-plane views can of course be generated to drawings.

Automatic tolerancing of User Features

Enables administrator to:

- Upgrade of reference User Features
- Creation of tolerancing schema features
- Creation of catalogue(s) of tolerancing schemas
- User default tolerancing schema catalogue definition

and User to benefits from automatic creation of Tolerances for User Features as well as:

- User Features selection propagation
- Group of User Features Tolerances creation
- User Features tolerancing schema query and filter

Global copy of FTA data from one CATPart to another one

Allow duplicating an existing annotation set feature from a CATPart to another CATPart document. This includes automatic reroute to geometric elements that have the exact same geometric characteristics.

Clipping plane enhancement: support of offset/aligned section view/cut

If the active view is a component of an Aligned/Offset Section Cut/View feature, the corresponding 3D Aligned/Offset clipping is displayed

Wall On Edge Propagation

To define a Wall on edge feature, the user can select several edges which will generate several folds. Folds and their tips are automatically relimited according to a minimum gap value between them.

User stamp on both sides

In addition to a better user interface, this option allows users to create stamps in two opposite directions within a single operation.

Chamfer Recognition

The feature recognition in Generative Sheetmetal Design now enables users to recognize the chamfers.

Intersection Fillets on Functional modeling

This function improves Intersection Fillet stability by computing them at the minimal intersection level. More modifications on previous features may occur with no failure on intersection fillet definition and computation.

Local Work on Functional modeling

This command, similar to the 'define in work' command, allows the user to work locally in a simplified context, such as a sub-part of the part, by selecting a functional set or a list of features. It reduces the impact of the part complexity.

SDNF (Steel Detailing Neutral File) Import/Export Command

This provides the capability to Export Structures information into a neutral file format called SDNF. SDNF is designed to be expandable to meet future needs and capabilities. All information in the file is in ASCII format and is structured into “packets” of information. It also provides the capability to Import a valid SDNF file with structure information and use it to create these structures under a specified product.

Advanced Bill Of Material: Support of user-defined product attributes

This enables the user to extract not only the predefined CATIA attributes of products, but also the attributes added by users through "Other Properties" from the Bill Of Material (BOM).

Section and auxiliary view callouts

Within a 3D Master process, this function allows to natively get a normalized representation of callouts in layouts, rather than in drawings only.

Callout representations corresponding to Auxiliary views, Section views, Section cuts and Multi-Planes Section Views can be added in existing design views.

This can be done while creating such views or afterwards, using the same styles and properties as Drafting's callouts.

These callouts will of course be generated in Drawings while generating a corresponding layout view.

Dynamic sectioning in Part Design

This provides a dynamic sectioning command. The user has a tool to cut a part at a position he will choose dynamically by moving a 3D section directly in the 3D viewer. This functionality can be useful during the design itself, to visualize the being-designed-part by section, or at review time, to detect potential section issues.

Intersection Edge Fillet - Additional "Intersection against selected features" definition mode in Part Design

When the user wants to create a fillet between features, he may define two lists of features by selection. The fillet will run on the intersection edges existing between these two lists.

Joggle enhancement request

This option allows the user to locate the extremity of the joggle regarding its joggle plane. Either the End radius extremity or the Run out extremity could be located exactly on the joggle plane.

FAA (Federal Aviation Administration) Compliance

The 3D Insight product has been developed to comply with FAA certification policies which require there be only one model and one modifier, a primary engineer, during the entire development, deployment, manufacturing and management lifecycle. It is especially dedicated to collaborators who need to access and use the 3D product definition, while disallowing them to save or export created/modified data. This prevents any changes of the 3D product definition.

CATIA Equipment

Enhancement For Bend Radius Ratio (BRR) Usage

The Branchable will now have two new attributes added to the list of existing attributes: the Creation Mode attribute, and Bend Radius Ratio attribute. These two attributes will complement each other and provide a more improved information on the Branch especially when it is created using Bend Radius Ratio button. The addition of these two new attributes for the Branch:

- Will enable every Branch to hold the information about the BRR value set for its Curve
- While performing Wire Routing, this new BRR attribute can now be used and respected to recompute Branch/(BNS) Diameter and BendRadius appropriately.

While editing multiple branchables via Branch Definition Command, each branchable's BRR value can be invoked from this new attribute

Filter Device List

Enable several ways to filter and sort the device list. Show the harness defined for each device

CATIA Machining

Advanced Finishing

Machine a whole part with a dedicated tool path style for both vertical & horizontal area Useful for finishing or for finishing rework (material left by a bigger cutter)

Machine simulation

Display of stock based on tool color during integrated material removal and machine simulation

CATIA Infrastructure

NEW STEP product for advanced functionalities

The CATIA - Extended STEP Interface enables long-term archiving with full validation properties and nested assemblies. It also makes it possible to manage very large assembly structures thru STEP thanks to nested assemblies support. This feature especially benefits the aeronautical and automotive industries.

Healing assistant: Compare for review

Enable the review & approval process for any design geometric change
Speed the approval process for a new revision of a part geometry. This allows a better understanding of part modifications impact to downstream processes such as tooling, machining, analysis or inspection

CATIA Analysis and SIMULIA Extended Analysis

New Rule Based Meshing for Creating High Quality Meshes

A new product, SIMULIA Rule Based Meshing (RBM), is available for test in R20 & R20SP1 and for industrial usage in R20SP2 to automate the process of creating high quality surface meshing for all workflows that use CATIA surface meshing tools. The new product gives the user a means to specify globally the desired meshing treatment of entities such as holes, fillets, and beads. It also gives the user a means to specify acceptable element quality criteria, such as minimum edge length, aspect ratio, and skewness. Once the complete set of meshing rules has been specified, no additional user involvement is necessary, as the actual mesh generation is completely automated. This new product used in combination with FMS can only be installed on top of it and provides on top of automation a very significant improvement of mesh quality as compared to FMS V5R19.

Perform Design Analysis Computations in Batch Mode in ENOVIA

CATIA Generative Part Structural Analysis enables analysis jobs that are ready to solve on an ENOVIA server to be updated in batch mode.

Improvement in the Post-processing of Composite Design Computation

The user will have access to the ply names as well as the Ply numbers, as defined in CATIA Composite Design (CPD), when analyzing simulation results which significantly eases the process of design optimization for composites.

Performance Improvements in Automatic Contact Detection

Automatic detection of likely contact surfaces through the 'Find Interaction' Wizard is a valuable tool in both SIMULIA Nonlinear and Thermal Analysis products. The new release provides substantial performance improvements in searching the model for surfaces likely to come into contact during an analysis.

Import of Temperature Results from ATH into CATIA Analysis

ATH uses thermal loading conditions as input and provides a temperature distribution throughout the model as its key output. The new release enables an ATH temperature field to drive a thermal-stress analysis in CATIA Analysis in addition to ANL, the SIMULIA Nonlinear Structural Analysis product.

PEO Optimization Support

You can now use the Product Engineering Optimizer (PEO) workbench with ATH and ANL to optimize a desired model parameter. In the PEO workbench, you create and run the optimization problem, for example, minimizing the thickness at a key point in the model while keeping the stress within an acceptable range. The optimizer will run multiple consecutive analyses, varying the input parameters to achieve the desired result.

Improve Interpretation and Visualization of Design Analyses

CATIA Generative Part Analysis provides the ability to animate simulation models under various conditions of loads for improved comparison studies. CATIA - Generative Dynamic Response Analysis provides the ability to display and animate complex values after filtering operations are complete. It also enables results to be visualized at a given modulus with phase.

Virtual Design for product Excellence

