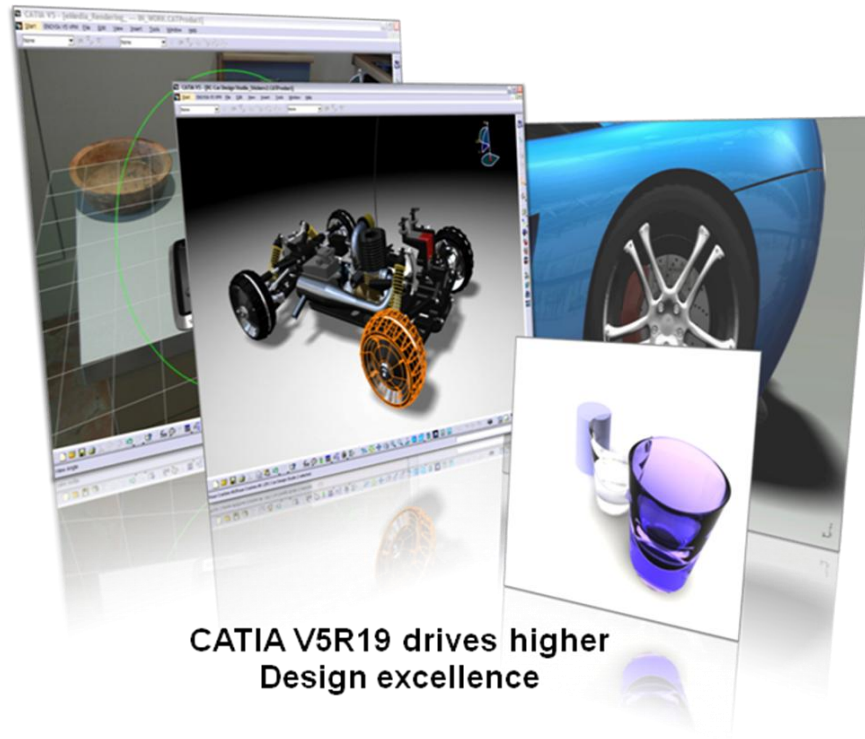


## CATIA V5R19 - FACT SHEET



- [Introduction](#)
- [What's New at a Glance](#)
- [Overview](#)
- [Detailed Description](#)

### [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

**CATIA V5R19 allows you to:**

- **Consolidates end-to-end industry process** coverage for plastic part design and enables new terrain modeling in context process
- **Enables In-context Composite Design** for optimizing the definition of large composites parts
- **Accelerates long lasting productivity tools** for the design of plastic part and tooling design review process
- **Add a global collaboration environment** tailored to CATIA V5 designers
- Becomes fully compatible with **Windows Vista**

## OVERVIEW:

**CATIA V5R19's products and features listed below bring value to the customer reinforcing Dassault Systèmes' and IBM's CATIA fundamentals:**

**Your CATIA PLM, Designed for you, designed with you**

- CATIA V5R19 enables Terrain modeling process by delivering large scale modeling of meshed surfaces and volume capabilities. It enables the modeling and planning of earthwork activity, modeling construction in-situ, allows earthwork engineering and planning and accurate foundation engineering.
- CATIA V5R19 reinforces plastic part process and leverage industrial communication on product through easy label painting creation & placement.

**Design performance for Innovation, Just be the imagineer**

- In-context Composite Design in CATIA V5R19 for optimizing the definition of large composites parts in context of mating sub-structures such as, spars, ribs, and stringers; and for optimizing the definition of plies in small parts or re-engineered metal parts
- CATIA V5R19 unleashes industrial designer creativity with the Imagine and Shape product which allows fast surface creation from curves or network of curves and allows the user to create their own primitives to start their design.

**Breakthrough technologies, Your innovation starts with ours**

- CATIA V5R19 brings superior productivity in the surface modeling used in the design of plastic and molded parts using the breakthrough auto-fillet capability.

This brings to users a breakthrough technology for an order of magnitude in productivity

- CATIA V5R19 accelerates the part and tooling design review process for manufacturability, especially for casting and injection molding processes. The new wall thickness analysis feature ensures that the thickness meets the requirements of the manufacturing process in order to anticipate and fix potential manufacturing issues.
- CATIA V5R19 takes real time rendering one step further to allow users to experience the product as if it was real, to provide real time material configuration testing, and to leverage the perceived quality of the final product.

### **Collaborative PLM, Networking brains to speed up innovation**

- Designer Central V6 for CATIA V5R19 provides a global collaboration environment tailored to CATIA V5 designers to manage their product IP and change process. It prepares V6 transition from existing V5 seat by providing migration and co-existence support

### **Easy, Open and Smart, Adaptable to you**

- CATIA V5R19 improves long term archiving for the aerospace industry by aligning with the latest PROStep/PDES.inc recommended practices for the control of STEP transfers.
- V5R19 provides the user with large scale management capabilities to enable large scale modeling of industrial projects such as dams in the construction industry.
- V5R19 is fully compatible with Microsoft Windows Vista.

## **DETAILED DESCRIPTION**

### **Your CATIA PLM, Designed for you, designed with you**

CATIA V5R19 enables Terrain modeling process by delivering large scale modeling of meshed surfaces and volume capabilities. It enables the modeling and planning of earthwork activity, modeling construction in-situ, allows earthwork engineering and planning and accurate foundation engineering.

CATIA V5R19 reinforces plastic part process and leverage industrial communication on product through easy label painting creation & placement.

CATIA V5R19 improves the end to end electrical process with more knowledge capture and reuse for electrical design. The user may define knowledge user parameters on internal Harness covering and copy from catalog definition to harness design. A knowledgware check allows him to validate the 3D harness segment diameters versus the harness segment diameters computed from wires. He may also synchronize Equipment reference designators attributes between electrical an tubing definition using knowledge.

CATIA V5R19 provides with more productivity in Wire and Harness Design & Flattening. A more efficient harness flattening is provided with the ability to keep bundle tangency from 3D harness design. It allows the usage of XML electrical files in local repository

when using ENOVIA VPM V5. The arrange segment in support has been improved. This ensures wire routing but also complex connector assemblies (shell & connector).

CATIA V5R19 improves V4 to V5 electrical Harness data migration. V4 to V5 migration brings to the user numerous benefits in this release, such as the migration of V4 Protection as V5 Light Protection. This enhancement enables user to use the better V5 Internal Protection if he/she migrates Electrical Data from V4 to V5 containing Protections. The migration offers a better consistency of V4 and V5 Bundle Segment Shape & Length.

CATIA V5R19 brings a new EndCut command for Structures design. The creation, edition and deletion of end cuts is provided and the end cut for Beam is integrated in the Piece Part Engine. The user is able to translate an SFD/SDD feature during the explode mode through Piece Part process. In addition, it is integrated in Drafting: the new end cut command enables a user to define end cut Graphic replacement. The user is able to manage different graphic replacement for Beam pillar.

CATIA V5R19 enhances technological results usages in the tooling design. Mold to CAM enhances to permit the creation of technological results report. Reporting of Die/Mold Technological Results is accessible via the new command Display Technological Results (with MPA license). Any user of Tooling Design 1 who has an MPA license can benefit from enhanced capabilities for Mold to CAM. A reporting command allows the user to create a report (\*.csv) on the Technological Results existing in a given Die. The company can then define some in house processes between the Design and the Manufacturing stage. It also provides with a powerful control tool of the integrity of the Die design, preventing costly iteration between design and manufacturing

CATIA V5R19 increases design productivity in sheet metal design. Design productivity increased with powerful Extrusion explodes improvement. It offers an explode mode allowing user to generate associative wall on edge. Creating an extrusion becomes an accelerator of the design. Exploding extrusion allows user to locally modify design keeping associativity.

CATIA V5R19 for Analysis allows the mode shapes of two models to be animated simultaneously. This allows for an easier comparison of the vibration characteristics of two design alternatives. The animation makes it much easier to visualize and understand the dynamic vibration of a product.

CATIA V5R19 brings new command to users in structures design. A new EndCut command to support the creation, edition and deletion of end cuts. With the integration of end cut for Beam in the Piece Part Engine: the user is then able to translate an SFD/SDD feature during the explode mode through Piece Part process. With the integration of end cut for Beam in Drafting: the new end cut command enables a user to define end cut Graphic replacement. The user is able to manage different graphic replacement for Beam pillar.

CATIA V5R19 ensures new improvement in the machining simulation domain.

- Capability to support milling machine with interchangeable heads. This allow the user to simulate Milling machines with interchangeable heads, thus enabling the user to verify NC tool paths for travel limits, collisions and modify if necessary.

- Machine configuration management within an operation. This provide with new options in the existing tool path modify dialog and the tool path trace so that he can force the selection of a particular machine configuration during machine simulation. This will allow the user to associate and persist a machine configuration along with a particular tool path point from within the tool path modify dialog. This machine configuration will be applied during machine simulation when the particular tool path point is to be reached.

CATIA V5R19 ensures the design of Machine Complex part with easy to use Multi axis Machining: operations. The designer is provided with a new operation which is the Multi Pocket Flank contouring. The multiple tool axis is supported for the 'point to point' operation. And the tool axis may be driven by an auxiliary surface (sweeping, contour driven, iso-parametric, curve following) through the 5x operation.

CATIA V5R19 reduces the Programming time and reduces the Machining time. To get an accurate information, the user can display the in-process stock on each milling or turning operations. The tool path replay includes the capability to check collision with part and fixture defined in the PO. During a contour driven operation, an extra guide may be defined: by 2 points, point and direction, and an automatic guide contour may be computed to enable contact point on guide curve. Multiple radial passes may be operated for thread and circular milling operations.

CATIA V5R19 enlarges and optimizes Machining process and technology. New operations are brought for probing in context of NC machines. Regarding the plunge milling, a new mode « by offset on contour » is offered. The clearance management has been federated and the tool path cornerization is proposed. As for the Mill-Turn, the capability of C axis management on a turret is available, it is possible to process machining on main spindle and counter spindle part with the same tool and the user may invert the tool orientation (0°-180°). Concerning DPM Machining, conical tool and user profile tool are supported in IPM generation for profile contouring.

### Design performance for Innovation, Just be the imagineer

CATIA V5R19 brings 2 new approaches for the end to end composite process. These two additional approaches are the grid and solid approaches. Industry in Aerospace but also Energy for the turbine wings design are provided with a better optimization in the composite design steps and a better mating with structural parts.

CATIA V5R19 expands the power of 3D to terrain modeling discipline that requires large scale modelization capabilities.

- The user gets the ability to work with meshed surfaces and volumes (Tessellation, Multi-slice commands, etc...), and the capability to easily edit the meshes. He may create and manage Z-Level curves. The building implantation analysis benefits from dedicated improvements. The manipulation performances have been greatly enhanced.

CATIA V5R19 provides with the Intersection Edge Fillet to improve the update stability during design changes:

- Part Design increases the user daily productivity: the Intersection Edge Fillet. This functionality helps the user to create fillets with the appropriate definition,

which is a definition by the intersection of several features, beyond the intersection of faces. As a consequence, it improves the update stability during design changes. Moreover, it facilitates the capture and the reuse of the part design features. For example, user can define a power copy including a set of features and a fillet defined at a feature intersection level. When this power copy feature is re-instantiated in another context, the fillet is correctly updated as it does not depend on the intersection of the selected faces but on the intersection of the features.

CATIA V5R19 improves productivity by an easier definition and a better stability of the Blend Corner:

- The Blend Corner capability and stability improve. This helps extend its usage for fillet definition (getting rid of small edges selection) and prevent the redefinition of former blend corners when modifying an Edge Fillet. Its user interface has been enhanced (creation by selection of edges or vertices, edition of the value, remove) to improve productivity.

CATIA V5R19 allows user to define the Gage Taper Hole main diameter using the selected face as the reference

- The user interface of the Gage Taper Hole has been improved: a new computation mode enables user to compute the tapered hole by using the selected face as the reference for the main diameter.

CATIA V5R19 for Analysis now allows frictional effects to be included in contact analyses, allowing these types of products to be simulated, as well as making it easier to analyze models that are under constrained if friction is not included. The technology and methods used are the same as those used in SIMULIA's Abaqus product, so they have been proven over many years of production use.

### **Breakthrough technologies, Your innovation starts with ours**

CATIA V5R19 improves and accelerates the part and tooling design review process for manufacturability with the Wall Thickness Analysis for Cast & Forge

- After having delivered the Automatic Fillet and the Automatic Draft for the Cast & Forged Optimizer (CFO) product, DS keeps on delivering CFO add-ons with CFO Wall Thickness Analysis. This new capability improves and accelerates the part and tooling design review process for manufacturability, not only in casting et Forging processes but also in Injection Molding processes for Plastic Part Design. It allows analyzing the part thickness either by projecting a ray from the selected point inside the part and normally to its surface, or by rolling a sphere of a maximal diameter inside the part. Moreover, these tools are coupled with a dynamic sectioning view and a transparent mode display, allowing a detailed and accurate analysis. As a result, it provides with a very quick identification of thin or thick critical areas.

CATIA V5R19 enables automatic filleting in Shape Design. Automatic filleting represents a significant process improvement, especially in the design for the plastic part, mold part.

Previously, the process of filleting sharp edges to conform to existing features was a repetitive and very time-consuming phase of the shape design process. This new capability can automatically fillet the sharp edges of a shape in a single operation. Automatic filleting is particularly helpful for fillet manufacturing preparation.

CATIA V5R19 delivers Fast Surfaces creation from Curves in Imagine and Shape. Two new primitive commands have been added through intuitive and productive interface with no additional geometry creation. The extruded profile command enables the designer to draw directly the profile from the extrude command. The modification of the profile can be made dynamically within the extrude command. Another command is provided to revolve the profile around an axis definition.

CATIA V5R19 brings advanced effects in rendering like soft shadows from multiple light sources. Shadows have been made more realistic with the addition of multiple light sources. The user has now the ability to compute and to store light maps off-line. He may then load them so that a scene can be rendered in real time without waiting on calculations to be made.

When used to simulate design performance, CATIA V5R19 for Analysis features improved meshing in thin sections. Many parts have thin sections, and it is desirable that the finite element mesh has at least two elements through the thickness to ensure accurate simulation results, which often required manual intervention. In V5R19, a single mouse click ensures that an accurate mesh is obtained in these situations, resulting in saved time and improved accuracy of the analysis.

CATIA V5R19 for Analysis has enhanced quadrilateral shell meshing for more precise design simulation results. The mesh is better aligned with the primary geometric directions, resulting in an improved mesh and more accurate results.

### **Collaborative PLM, Networking brains to speed up innovation**

Designer Central for CATIA V5: provides a multi-site CATIA V5 design data management solution on the V6 platform

CATIA V5R19 improves 3DXML export with shader support. A full compatibility with CgFX is provided for shadows. More materials have been added to the materials library, such as CarPaint and external shaders can be imported for even more effects.

CATIA V5R19 improves 3D annotations review using 3D XML. The FT&A features defined in CATProduct and in CATProcess documents can be saved in .3dxml documents as well as .CATPart annotations. It allows user to review them either using the DMU Dimensioning & Tolerancing Review product or the 3D Live FT&A Review product.

CATIA V5R19 is now supporting new CAD format. Regarding the multiCAD capability, new formats are supported, such as SW2007, SW2008, SolidEdge V20, Parasolid V18 and UG NX4, NX5.

### **Easy, Open and Smart, Adaptable to you**

CATIA V5R19 provides CATIA application infrastructures with a large scale management. After having addressed Micro-Electro-Mechanical Systems Industry in V5R18 with the Small scale management, DS provides with a large scale management (from 0.1mm up to 10 km). Mechanical Infrastructure enables to design 3D large scale objects, as dams for example. This large scale management benefits from a whole integration in Generative Drafting and 2D Layout.

CATIA V5R19 ensures alignment to latest PROStep/PDES.inc recommended practices for the control of STEP transfers. New STEP validation properties are taken into account for assemblies. This allows checking assemblies' conversion even when the geometry is not exchanged (e.g. nested assemblies).