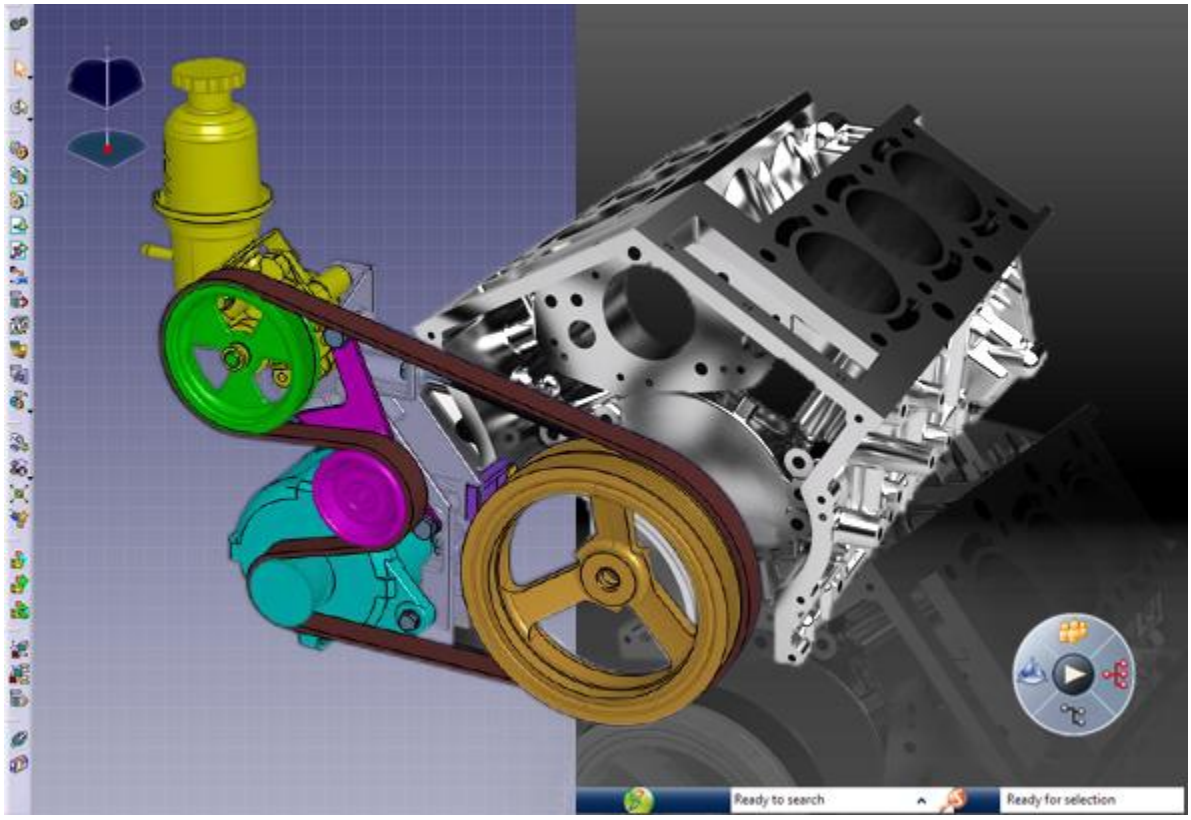
CATIA V5-6R2012 - FACT SHEET



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

- Delivers unique V5-V6 collaboration
- Allows V5 users to experience V6 core modeling technology.
- Enriches Industry Experience through enhancements.

OVERVIEW:

Unique feature level collaboration between V5 and V6 demonstrates unceasing enrichment of V5.

- **Unique new compatibility enables feature level collaboration between CATIA V5 and Version 6**

  3D Models created in CATIA V6R2012X can be sent to V5-6R2012, retaining their core features. These features can be modified directly in V5. A design can now evolve iteratively, with engineers having the freedom to create and modify the part at the feature level, whether they use CATIA V5 or V6. With this enhancement, the compatibility between 3D models in CATIA V5 and in CATIA V6 now exceeds that between two releases of V5. Mixed teams enjoy a *new collaborative potential*.

- **This new level of collaboration is made possible by bringing select Version 6 developments to V5**

  These enhancements bring immediate value to V5 users who now have access to many new features which were previously available only in CATIA V6. These include new surfacing and Part Design functionalities, developed in V6 and now available in V5-6R2012. This secures the investment our customers have made in V5, while making it easier for them to take advantage of the open V6 portfolio at their own pace. This capability will be available between synchronized releases of V6 and V5, beginning with V6R2012x and V5-6R2012. The change in name in this release, from V5 to V5-6 reflects the unparalleled compatibility between V5 and V6, the synchronization of V5 and V6 releases, and the enrichment of V5 with select V6 technology.

- **Extended Industry Experience, built on the significant enrichment of CATIA V5 over the last 2 releases, including:**
  - A fully integrated Automotive Class A solution with CATIA ICEM Shape Design
  - Top quality surfaces for body design in the automotive industry, with a superior user experience from CATIA Imagine & Shape.
  - Extended support for industry standards and long-term archiving, adding STEP exchange for large assemblies and composites.
  - Continued improvement to mechanical surfacing design technologies
  - New 3D Insight product
  - Enhanced Functional Modeling Part product
  - New capabilities in CATIA Analysis and the SIMULIA Extended Analysis products
  - CATIA Machining offers a new aerospace-validated machining strategy for hard material machining
  - The addition of Material Removal Simulation and Advanced Finishing

DETAILED DESCRIPTION

Unique feature level collaboration between V5 and V6 demonstrates unceasing enrichment of V5.

- **Unique new compatibility enables feature level collaboration between CATIA V5 and Version 6**

  - 3D Models created in CATIA V6R2012X can be sent to V5-6R2012, retaining their core features. These features can be modified directly in V5. A design can now evolve iteratively, with engineers having the freedom to create and modify the part at the feature level, whether they use CATIA V5 or Version 6.
  - With this enhancement, the compatibility between 3D models in CATIA V5 and in CATIA Version 6 now exceeds that between two releases of V5. Mixed teams enjoy a new collaborative potential.

- **New level of collaboration made possible by bringing select Version 6 developments to V5**

This gives CATIA V5 users access to many new features which were previously available only in Version 6. These enhancements bring immediate value to V5 users.

- New surfacing functionality developed in Version 6 and now available in V5-6R2012 includes:

  - Associative silhouette features
    Designer needs to compute the vision cones of objects that are inside a car, from the view point of the passengers. Until now there was no possibility to display the visible boundary of a three-dimensional curved surface onto a plane along a direction with entire parametrical functionality. The Silhouette function will do in one shot several current manual user actions, like the computation of reflect lines, the extraction of border edges, the removal of non visible areas of the contour is also done...Optionally, in 'Cylindrical' mode, the user can indicate a projection/shadow plane. The process is now reduced to a very few steps, providing significant productivity gains

  - Flexible fill surfaces created from multiple passing elements and inner boundaries

    Fill: Enable multiple passing elements (points, curves): This adds the capability of having multiple Passing Elements (Points and Curves) to constrain the Fill surface. This provides flexibility in the fill surface definition, and allows the user to have a better control over Fill surface created (until now only one Passing Point could be specified).

    Fill: Inner boundaries selection: This highlight adds the capability to define inner boundaries and corresponding support surfaces. It enables the user to input curves for

inner boundaries and supports as well. So the resultant Fill surface will be a Fill surface 'between' the outer and inner closed contours. Currently the user needs to perform many redundant steps, creating auxiliary geometry to by-pass the lack of this functionality. In many cases, these "by-pass" solutions are complex and time consuming. This improvement of the current Fill feature improves the user productivity.

Fill: Allow continuity specification for individual support: This adds the capability in the fill surface definition to allow individual continuity on each support, providing the user more flexibility in creating a fill. The user has a greater control over the created fill by choosing the Continuity type on each Fill boundary (In previous releases the continuity was global to the Fill), with the help of widgets as well as through panel combo box.

- Sew surface based on surfacic targets
  This functionality is extended from Part Design workbench to support sew not only on solid context but also surfacic context. 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 is a gain to the process of the design of bodies in white. It avoids creating auxiliary geometry to by-pass the lack of this functionality. In many cases, these "by-pass" solutions are complex.

- Ability to create fill surfaces without complete boundaries.
  You can now create the Fill without specifying any outer boundaries. The outer boundary is no more a mandatory input, and when the outer boundaries are not specified, you are then required to input either the passing elements or inner boundaries, or both. This adds more flexibility while creating the fill surfaces.

- New Part Design enhancements include:
  - Enhanced location and editing of the values of a variable fillet
    Variable Fillet - Radius values edition: The definition of whole radius values of a variable or chordal edge fillet feature can be directly displayed and managed in one new dedicated panel. This enhancement offers a clear benefit when a fillet definition cannot be easily displayed on the geometry due to the amount of radius values around the same location or in case of a very long fillet running along the whole part like in Tooling Part design for instance. Designers are now able to have a clear overview of the whole definition of such fillets and easily navigate in the geometry from one radius value location to another. This increases user-friendliness and productivity.

  - Ability to shell a solid without propagating selected services
    Shell without face propagation: For some specific cases, designers need to avoid the extension of selected opening faces of a Shell feature with all faces coming from the

default propagation in tangency. Now, users can disable this propagation in tangency if needed, increasing design robustness and productivity.

- Staggered patterns with a shift between each row of the instance
  Create a wide range of patterns with the staggered pattern command: Specific staggered pattern configurations cannot be created using Rectangular Pattern command. For instance, a staggered pattern refers to a type of rectangular pattern in which, the alternate rows of the pattern have their instances offset by a certain distance.

- Enhanced Sketcher ease of use to indicate required partial extractions (near).
  Sketcher - Near & no canonicity use-edges: Near and No canonicity options can be used when creating use-edges.

  - This ***secures the investment our customers have made in V5***, while making it easier for them to take advantage of the open Version 6 portfolio at their own pace. Our customers can transition to Version 6, while continuing to collaborate seamlessly with departments, customers and suppliers who use V5.
  - This ***capability will be available between synchronized*** releases of V6 and V5, beginning with V6R2012x and V5-6R2012.
  - The change in name in this release, from V5 to V5-6 reflects the unparalleled compatibility between V5 and V6, the synchronization of V5 and V6 releases, and the enrichment of V5 with select V6 technology.

- **Builds on the significant enrichment of CATIA V5 over the last 2 releases, including:**
  - A fully integrated Automotive Class A solution with CATIA ICEM Shape Design delivers functional completeness and V5-V6 feature equivalence for CATIA ICEM products.
  - Top quality surfaces for body design in the automotive industry, with a superior user experience from CATIA Imagine & Shape. The virtual clay modeler, based on subdivision surface technology, delivers world-class surfacing tools to address body shape details. Designers also benefit from real-time rendering visual feedback during the creation phase. These enhancements make CATIA Imagine & Shape the market leader for body design in the automotive industry.
  - Extended support for industry standards and long-term archiving, adding STEP exchange for large assemblies and composites.
  - Continued improvement to mechanical surfacing design technologies with Loft command enhancements.

- The 3D Insight product has been developed to comply with FAA certification policies requiring only one model and one modifier, a primary engineer, during the entire development, deployment, manufacturing, and management lifecycle.
- 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.
- 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.
- CATIA Machining offers a new aerospace-validated machining strategy for hard material machining. Numerous enhancements in multi-pocket flank contouring, profile contouring, multi-point probing and drilling for unmatched programming and machining time reduction.
- The addition of Material Removal Simulation and Advanced Finishing reduce programming and machining time, respectively, saving companies time and money.