Mechanical Design

CATIA - Extended Mechanical Design 2
Configuration (XM2)

CATIA V5R21
Configuration overview

Perfectly suited to such various domains as Industrial Machinery or Consumer Goods, the XM1 configuration is a 3D product creation package including comprehensive part and assembly design features, as well as associative drawing extraction capabilities. Furthermore, designers will find all necessary 2D drafting features for efficient drawing production. As a CATIA P2 configuration, this solution offers advanced 3D oriented productivity features such as Analysis of the degrees of freedom of a component, in order to quickly understand (graphically) how the component is constrained. Knowledge basic capabilities allow designers to import and use corporate knowledge stored in rule databases to ensure consistency and quality of their designs. To match industrial needs, the configuration has been reinforced with sheetmetal design productive capabilities and advanced wireframe and surfacing features. Customer benefit from a high productivity solution which is natively interoperable with other CATIA Version 5 sets of solutions and which offers integration tools for compatibility with CATIA Version 4 and data interfaces to most frequently used industry standards, which makes it a configuration of choice for existing Version 4 customers.
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Product overview

CATIA - Knowledge Expert 1 (KE1) allows designers to import and use corporate knowledge stored in rule bases created using CATIA - Knowledge Expert 2 (KWE). Thus design compliance is ensured with established standards. These rule bases automate knowledge processes such as:

- best practices,
- application processes, and,
- design validation and corrections.

The reports generated provide better identification of standards violations and help to implement necessary corrections. As an integrated product, CATIA - Knowledge Expert 1 (KE1) can be used in conjunction with all other Version 5 products to share process applications throughout the enterprise. The Knowledge Expert solution consists of a buildtime product, CATIA - Knowledge Expert 2 (KWE), and a runtime solution, CATIA - Knowledge Expert 1 (KE1).

Real Time Rendering

Enable designers to leverage technological material specifications to produce realistic renderings of their model.

Product overview

Real Time Rendering (RT1) Enables designers to leverage technological material specifications to produce realistic renderings of their model. Texture can be created from scratch, modified from imported digital images, or selected from the included library. Associativity is maintained between the material library and the material applied to the parts. Materials can be applied through a specification-driven approach or through simple selection. Real-time display computations quickly convert models to realistic renderings.
CATIA - Assembly Design
Easily define mechanical assembly constraints, automatically position parts and check assembly consistency.

Product overview
CATIA - Assembly Design 2 (ASD) offers a new generation CATIA P2 product for managing assemblies. Assembly Design products integrate with other CATIA Version 5 applications like part design and drawing generation. CATIA - Assembly Design 2 (ASD) establishes mechanical assembly constraints using mouse movements or graphical commands to easily snap parts into position. Assembly Design 2 helps designers managing large, hierarchical assemblies of CATIA V4, V5, VRML or STEP parts using a top-down or bottom-up approach. Parts and sub-assemblies are easily reused in the assembly without data duplication. Productivity tools like automatic exploded view generation, collision and clearance checking. Automated BOM generation greatly reduce time and increase quality. Flexible Sub-Assembly gives user the ability to dynamically unlink product structure and mechanical behavior. This unique command allows to move individual component in the parent assembly or to manage different internal positions of instantiated sub-component. Introducing an intuitive user interface, user interface across NT and UNIX, offering productivity, ease of use, and low training costs.

CATIA - Weld Design 1
Design advanced welded products based on process definition and rules.

Product overview
CATIA - Weld Design 1 (WD1) is an application addressing welded assemblies. 15 weld types (ISO standard) are provided to the user to create welds, part preparation and relevant annotations.

This product addresses Machinery & tooling industries through advanced welding processes. The creation of the weld in the 3D digital Mock-Up will allow designer to manage digital pre-assembly, mass inertia, space reservation and drafting annotation.
CATIA - Part Design
Design complex parts with highly productive and robust modeling features in an intuitive environment

Product overview
This CATIA-P2 offers a new generation CATIA product for part design with one interface on NT and UNIX. Interoperates with CATIA Version 4 data and methodologies. The "smart-solid" design kernel combines high productivity feature-based toolsets and boolean methodologies, delivering a flexible solution enabling multiple design approaches. CATIA - Part Design 2 (PDG) can be used with CATIA V5 applications like Assembly Design, Drafting, Wireframe & Surface.

CATIA - Generative Drafting
Generate drawings from 3D part and assembly designs with assistance to manage complex schema

Product overview
Generative Drafting for CATIA-P2 is new-generation CATIA products which allows users to automatically generate associative drafting from 3D mechanical designs and assemblies produced with CATIA Version 5. When combined with Interactive Drafting 1, the Generative Drafting product 2 benefits both from integrated 2D interactive functionality and from a higher productive environment for drawings dress-up and annotation. Generative Drafting 2 product offers a flexible and scalable solution to create associative drawings from 3D mechanical designs, surfaces, hybrid parts and assemblies created with CATIA Version 4 or Version 5. 3D dimensions can be automatically generated with control over their placement. Designers are able to add post-generation annotations with standards-based dress-up features. Associativity of the drawings to the 3D master representation enables users to concurrently work on their design and drawings.
CATIA - Interactive Drafting
Address 2D design and drawing production requirements.

Product overview
Interactive Drafting is a new generation CATIA product to address 2D design and drawing production requirements. It offers a highly productive, and intuitive, interactive drafting system. It also enriches Generative Drafting with both integrated 2D interactive functionality and with a productive environment for drawings dress-up and annotation.
Product overview

The Next Generation CATIA Solution Version 5 is built on a totally new scalable architecture, which combines the best of the current CATIA technologies with new generation standards. It offers total single system image across Native Windows and UNIX environments, and an extensible environment capable of supporting all aspects of the Digital Enterprise, from Digital Mockup, Digital Manufacturing, to Digital Operation and Digital Plant definition. The Version 5 system architecture offers a unique scalability environment which gives to customers the choice of selecting the optimum set of solutions, given the CAD specialization of the intended users, the complexity of the project and corresponding functional requirements. The 3 possible choices are CATIA-P1, CATIA-P2 and CATIA P3 installations.
**CATIA - CADAM Interface**
Share drafting information between the CATIA-CADAM Drafting and V5 drafting products.

**Product overview**
The product CATIA - CADAM Interface 1 (CC1) provides the user with an integrated capability to share 2D drafting information between CATIA CADAM Drafting (CCD) and V5 Drafting products.

This integrated capability provides existing CCD users with the flexibility to easily integrate the V5 CATIA product portfolio into their environment, while maintaining their current practices and procedures based on their CCD product use.

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**CATIA - IGES INTERFACE**
CATIA - IGES Interface 1 (IG1) helps users working in a heterogeneous CAD/CAM environment to exchange data through the neutral IGES format.

**Product overview**
CATIA - IGES Interface (IG1) helps users working in a heterogeneous CAD/CAM environment to exchange data in a neutral format. This utility supports IGES Version 5.3 and provides naming management to map IGES and CATIA identifiers. Users can perform reliable bi-directional 2D and 3D data exchange between dissimilar systems with direct and automated access to IGES files. CATIA - IGES Interface 1 (IG1) can process wire frame geometries, surfaces and trimmed surfaces, conic curves and color.
CATIA - STEP Core Interface
CATIA - STEP Core Interface 1 (ST1) allows users to read and write data in STEP AP214 and STEP AP203 data formats.

Product overview
CATIA - STEP Core Interface 1 (ST1) helps users working in a heterogeneous CAD/CAM environment to exchange data through a neutral format. This utility allows users to interactively read and write data in STEP AP214 and STEP AP203 data formats allowing reliable bi-directional data exchange between dissimilar systems. To facilitate access to data, CATIA Version 5 offers a homogeneous user interface for all supported formats, using Windows-compliant user interface controls (such as File > Open, File > Save as) and automatic recognition of the STEP file type.

CATIA - V4 Integration
CATIA - V4 Integration 1 (V41) helps existing CATIA Version 4 customers take advantage of the advanced technologies of Version 5 while preserving their investment in V4 data.

Product overview
Dassault Systemes has an unprecedented record of accomplishment in delivering upward compatible CATIA Solutions within a version life cycle and across major new versions. Thanks to a unique approach of gradual introduction of Next Generation software components within Version 4, started since V4R1.3, this new product line offers a smooth progression for current users of CATIA Solutions Version 4. Beyond the best of CATIA Version 4 Technologies brought forward in the new generation product line, CATIA - V4 Integration 2 (V4i), which is only part of CATIA-P2 set of solutions, offers numerous integration features which make seamless hybrid installations possible, involving both applications from CATIA Version 4 and applications from CATIA Version 5, while benefiting from data compatibility between the two product lines.
CATIA - Generative Shape Design
Create in-context, specification-driven shapes based on a combination of wireframes and extensive multiple surface features.

Product overview
CATIA - Generative Shape Design 1 (GS1) helps design mechanical shapes based on a combination of wireframe and multiple surface features. CATIA - Generative Shape Design 1 (GS1) provides an extensive set of tools for creating and modifying mechanical surfaces used in the design of complex shapes or hybrid parts. GS1 brings smart tools, like powercopy to manage the feature reuse. Its feature-based approach offers a productive and intuitive design environment where design methodologies and specifications can be captured and reused.

CATIA - Sheetmetal Design
Designs sheet metal parts in a very intuitive and productive environment

Product overview
CATIA - Sheetmetal Design 2 is a new generation CATIA product dedicated to the design of SheetMetal parts. Its feature-based approach offers a highly productive and intuitive design environment. It allows concurrent engineering between the folded or unfolded representation of the part.

CATIA - Sheetmetal Design 2 can be cooperatively used with other current or future applications of CATIA Version 5 like part design, assembly design and drawing generation. As sheetmetal design may start from scratch or from an existing solid, communication between suppliers and contractors is empowered. It includes many standard design features, for stamped, bended or rolled parts and compliant with E&E and F&A industries requirements. CATIA - Sheetmetal Design 2 offers the same ease of use and user interface consistency as all CATIA V5 applications to drastically reduce training cycle time and unleash designer creativity. It also provides multiple interoperability functions with CATIA V4 sheetmetal products. ...
ABOUT CATIA V5R21

CATIA is Dassault Systemes’ PLM solution for digital product definition and simulation.

www.3ds.com/products/catia