Machining

CATIA - Preparation Mold & Die Machinist 2
Configuration (RM2)

CATIA V5R21
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Configuration (RM2)

Configuration overview

This CATIA P2 Machining configuration integrates the powerful standard 3D and 2D mechanical design tools with generative 2.5-axis and 3-axis surface machining tools. It is targeted at Machining users who need to make design modifications or create additional geometries, fixtures or even drawings to explain the machine set-up. The easy implementation and propagation of modifications are only possible through the associativity of CATIA V5.
# CATIA - Preparation Mold & Die Machinist 2 Configuration (RM2) Product List

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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

Define mechanical assembly constraints, automatically position parts and checks assembly consistency.

Product overview

CATIA - Assembly Design 1 (AS1) is a P1 product for managing assemblies in a highly productive way. It gives the user the ability to design in assembly context with a user-control associativity, to perform concurrent engineering between the design of the assembly and the design of individual components through a hierarchical assemblies using a top-down or bottom-up approach. Assembly Design 1 establishes mechanical assembly constraints using mouse movements or graphical commands to easily and intuitively snap parts into position. Through a manual/automatic update choice, the designer can reorder the product structure, drags dynamically parts into position and makes collision and clearance checks. A productive work is provided through a catalog access for standard parts or assemblies, the access of advanced and powerful assemblies features, automatic exploded view generation and automated BOM generation. Assembly Designers get great time reduction and increased quality. Introducing an intuitive user interface, common user interface across NT and UNIX, offering productivity, ease of use, and low training costs.
CATIA - Part Design
Design parts in a highly productive and intuitive environment

Product overview

CATIA - Part Design 1 (PD1) product for CATIA-P1 offers a new generation CATIA product for part design with one interface on NT and UNIX. The "smart-solid" design kernel combines feature based design with the flexibility of Boolean approach, offering a highly productive, intuitive solution enabling multiple design methodologies.

CATIA - Part Design 1 (PD1) product can be used with other CATIA V5 applications such as: Assembly Design, Generative Drafting, Wireframe & Surface.

CATIA - Generative Drafting
Generate drawings from 3D part and assembly designs

Product overview

Generative Drafting for CATIA-P1 is a 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, the Generative Drafting product benefits both from integrated 2D interactive functionality and from a higher productive environment for drawings dress-up and annotation. Generative Drafting 1 product offers a flexible and scalable solution accommodating drawing generation requirements for sheetmetal, surface and hybrid parts or assemblies. A drawing creation assistant, or "wizard" simplifies the generation of multiple-view drawings. 3D dimensions can be automatically generated. Designers are able to associate patterns with part material specifications and 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 designs 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.

CATIA - NC Manufacturing Review
Access, visualize, and browse 3D NC Manufacturing data from CATIA V5 and previous versions.

Product overview
CATIA NC Manufacturing Review 2 (NCG) offers basic NC capabilities such as tool path verification, material removal simulation, remaining material analysis, tool path edition and creation of shopfloor documentation. It provides the infrastructure for all V5 NC programming products and allows NC programmers or machine operators to review V5 part operations. Furthermore, different manufacturing data such as APT files, CL files and ISO code can be imported and reviewed. NCG also allows the reuse of CATIA V4 NCMILL and NCLATHE programs.
CATIA - Prismatic Machining
Easily defines NC programs dedicated to machining 3D parts using advanced 2.5-axis milling and drilling operations. It also provides high-level automation and knowledge reuse capabilities.

Product overview
CATIA - Prismatic Machining 2 (PMG) easily defines drilling and 2.5-axis milling operations including support of High Speed Cutting technology. Quick tool path definition is ensured thanks to an intuitive user interface based on graphic dialog boxes. Tools can be easily created and stored in tool catalogs. Entire manufacturing process is covered from tool path definition and computation, tool path verification including material removal simulation to NC code and shop floor documentation generation. Associativity with CATIA design parts allows efficient change management.
CATIA - 3-Axis Surface Machining
Easily define NC programs dedicated to machining 3D parts using 3-axis surface machining operations.

Product overview
CATIA - 3 Axis Surface Machining 2 (SMG) easily defines 3-axis milling and drilling operations. High Speed Milling Technology is supported for all milling operations. Quick tool path definition is ensured thanks to an intuitive user interface based on graphic dialog boxes. Tools can be easily created and stored in tool catalogs. Entire manufacturing process is covered from tool path definition and computation, tool path verification including material removal simulation to NC code and shop floor documentation generation. Instant Cycle Update Technology (TM) allows fast tool path update after modification. Associativity with CATIA design parts allows efficient change management.
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.

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 - 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.
ABOUT CATIA V5R21

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

www.3ds.com/products/catia