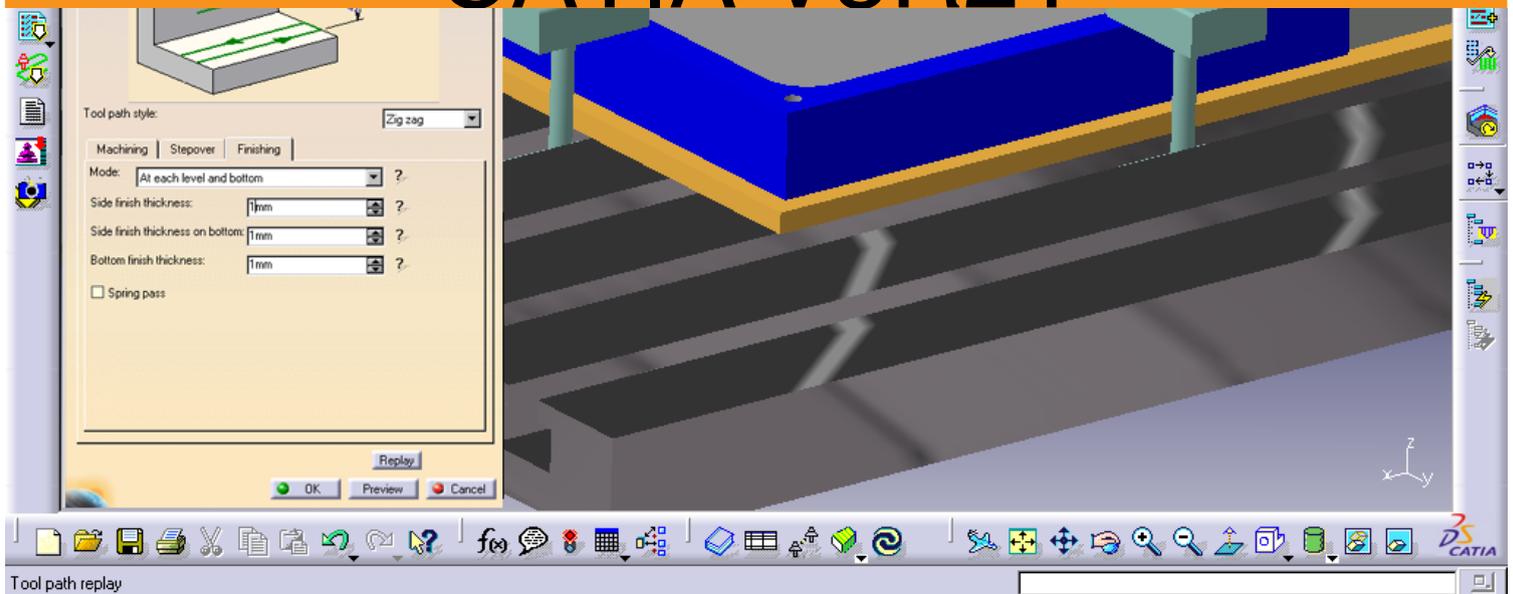


Machining

CATIA - Prismatic Machining 2 (PMG)

CATIA V5R21





Machining

CATIA - Prismatic Machining

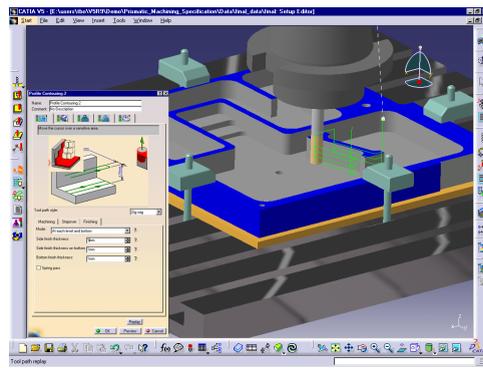
Easily defines NC programs dedicated to machining 3D parts using advanced 2.5-axis milling and drilling operations. It also provides high-level automation and knowledge reuse capabilities.

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.

Product Highlights

- Full set of 2.5 axis milling and drilling operations for accurate tool path definition, including support of High Speed Milling technology
- Flexible management of tools stored in file-based tool catalogs or in external databases
- High level of automation and standardization by capture and reuse of proven manufacturing know-how
- Tool path verification by material removal simulation and analysis of the in-part
- Associativity with CATIA design parts for efficient change management



Product Key Customer Benefits

Accurate tool path definition for drilling and 2.5-axis milling...

The user can elaborate a large range of machining operations for tool path definition like pocketing, facing and contouring operations. These operations can be defined as multi-level and multi-pass operations. Upon the user's request, the tool path can be optimized for High Speed Machining. Moreover, point-to-point machining operation is available, as well as 17 axial operations from standard drilling to more complex boring and chamfering operations. Finally, the user takes advantage of 3-axis to 5-axis NC machines, including 3-axis machines equipped with rotary table.

Quick tool path definition thanks to an intuitive user interface...

The user defines machining operations quickly using intuitive graphic dialog boxes. "Traffic lights" indicate if there are still parameters to be defined in order to complete the operation. Help icons assist the user for each parameter of the operation : when clicking on these help icons, an image describing the parameter pops up in

the panel. Moreover, the user takes advantage of copy/paste facilities for organizing program using specification tree. Finally, tool changes and machine rotations are automatically generated and can be visualized in the machining operation definition panel.

Flexible management of tools and tool

catalogs... Tools can be stored into file-based tool catalogs and retrieved using simple or complex queries. Tool assemblies (tools and tool holders) are supported so that the user can define the characteristics of the tool visualization. Tools can also be retrieved from external tool databases (like CATIA Version 4 TDB or WALTER TDM databases).

Automation and standardization... The user can define and store machining processes for 2.5 axis milling or axial machining operations in CATIA catalogs, and apply them to part geometry or to design features. By this, the company know-how can be capitalized and reused for efficient programming. NC objects and attributes can be handled as knowledgware objects in order to increase the level of automation and standardization.

Quick verification of tool path... Tool path replay allows generation and verification of individual operations or the complete programs. Alternative machining strategies can be tested and collision-free trajectories can be obtained.

In-process part visualization and material removal simulation... The user can visualize the in-process part and analyze remaining material, tool collision, etc. Furthermore, he can run a material removal simulation interactively or in batch-mode.

Seamless NC data generation... CATIA NC process is extended from tool trajectory (APT source) to NC Data generation (ISO format) thanks to an integrated postprocessor execution engine and a library of standard Post Processor (PP) samples. Moreover, the system allows the execution of Post Processors built with CATIA Version 4 Post Processor Builder product.

Efficient change management... Integrated the PPR data model CATIA NC offers a

high level of associativity product engineering, manufacturing processes and resources. Thus, CATIA allows an high efficiency in concurrent engineering and best support of design changes or design variants and a rapid creation of programs for families of parts.

ABOUT CATIA V5R21

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

www.3ds.com/products/catia

