CATIA V5 CAM from IBM/Dassault Systèmes

A Product Review

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Foreword

CIMdata, Inc. prepared this product review as an independent and unbiased assessment of the functional capabilities of CATIA V5 R16 CAM, a CAD/CAM software product developed by IBM/Dassault Systèmes. CATIA is a registered trademark of Dassault Systèmes. This evaluation is one in a series of software product reviews produced by CIMdata, a worldwide consulting and marketing firm. CIMdata has authorized IBM and Dassault Systèmes to reproduce and distribute this document, without constraints from CIMdata.

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CIMdata is an industry-leading consultant on CAM software systems. It produces the NC Software Market Assessment Reports and the Compendium of NC Product Reviews. Market research has been conducted by CIMdata on a variety of CAM related topics. CIMdata provides consulting services to CAM software users and vendors and to the investment community.

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CATIA V5 CAM
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IBM/Dassault Systèmes is a worldwide leader in providing a process-centric PLM (Product Lifecycle Management) software solution. Their theme is “Innovation Drives Growth, Flexibility Makes it Happen.” The PLM solution is comprised of CATIA for collaborative product development, ENOVIA and SMARTEAM for product data and lifecycle management, collaboration and decision support, and DELMIA for digital development of factory and production processes. These offerings are all built on the Version 5 (V5) architecture that facilitates compatibility, data integration and data associativity. SolidWorks is owned by Dassault Systèmes, but operates as an independent entity and is not a component of the PLM offering.

The IBM/Dassault Systèmes PLM portfolio optimizes business processes by providing a wide range of process-centric applications, promoting the integration of engineering and manufacturing environments, utilizing a product-process-resource (PPR) model, and growing the intellectual capital of users by capturing and re-using business processes both at the enterprise level and at the engineering or manufacturing level. CATIA V5 is a general purpose CAD/CAM product that is in use by all types of companies large and small, in all manufacturing environments be it prototype manufacturing, tooling manufacture, or production part manufacturing, and in all industries.

IBM and Dassault Systèmes have substantial worldwide presence and credibility, there is a strong management commitment to manufacturing, the product strategy and direction for PLM and CAM is visionary and rational, the CAM product development resources being employed by Dassault Systèmes are the largest in the industry, and a viable CAM marketing and sales program is being implemented by IBM.

The overall IBM/Dassault Systèmes vision for CATIA V5 is to provide a solution that creates and simulates the entire product life cycle from conceptual design to maintenance of installed products. There are over 150 products in the CATIA application suite, including products from third-party CAA (CATIA Applications Architecture) partners. With Version 5 Release 16 (R16), 64-bit support is provided for the full CATIA portfolio under Microsoft Windows. IBM/Dassault Systèmes has established a strong position in both the CAD and CAM market segments. CATIA V5 is well recognized as a premiere product for conceptual and product design and most of the worldwide large aerospace and automotive companies utilize it for this purpose. A manager at a small German aerospace supplier commented, “We went with CATIA because it is the de facto standard CAD/CAM product for the aerospace industry.”

In CAM, the IBM/Dassault Systèmes strategy is to support both a process-centric solution for larger manufacturers implementing a PLM vision and a CAM-centric solution for small to mid-sized manufacturers that have a more narrow focus. CAM related products are available for part modeling, reverse engineering, process planning, core and cavity design, mold base design, die design, drilling, 2.5-axis milling to simultaneous 5-axis milling, feature recognition and feature-based machining, knowledge-based machining, turning, multi-axis turning, mill-turning, EDM, machine tool definition, product documentation, toolpath verification, and simulation of the full machining process. The range of manufacturing applications, depth of functionality, data integration, programming automation, and integrated simulation within the CATIA V5 manufacturing product set is impressive. CATIA V5 can be employed to machine all types of parts.

The Dassault Systèmes product architecture model defines the core products to be developed within Dassault Systèmes or supplied by CAA partners that develop and provide components, auxiliary products or solutions. The Dassault Systèmes core products include all the standard drilling, milling, turning, inspection, and simulation products. CAA partner products are employed for die design, wire EDM, laser cutting, sheet metal manufacturing, composite manufacturing, and post-processing.

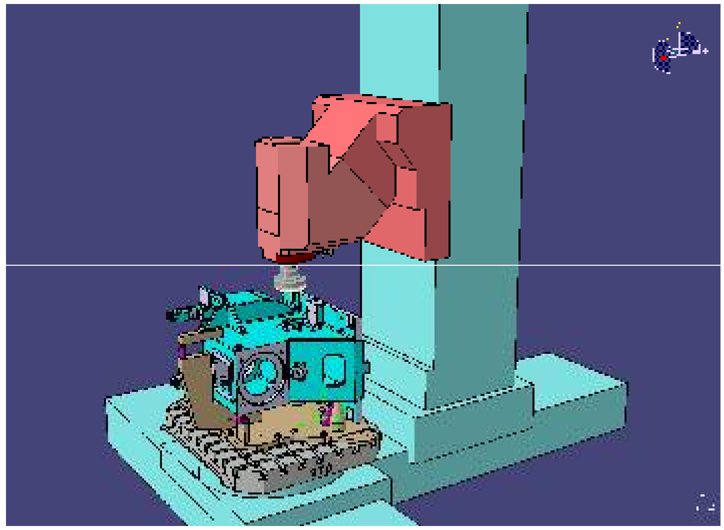

Figure 1—A Machining Simulation

In V5 R16, IBM/Dassault Systèmes is placing major emphasis on programming automation and integrated simulation. CIMdata is favorably impressed by the progress being made in both of these areas. Knowledgeware is a core product in CATIA V5. It facilitates automation by permitting all CATIA applications to capture best practices using defined settings and pre-defined machining processes, creating formulas, establishing parametric families of parts, and utilizing expert system rules to optimize programming. Three levels of programming automation have been defined to meet the needs of all levels of users.

The simulation capabilities in CATIA V5 R16 have been expanded to include full machine tool simulation as well as toolpath simulation. Simulation has been tightly integrated with toolpath generation to provide a user with quick viewing of the machining process as it occurs. A machined part analysis containing visualization of remaining material, undercuts, gouges and collisions is provided. Simulation can be performed on both the CATIA toolpath or APT file before post-processing or on the post-processed ISO file that is seen by a machine tool controller is produced. Controller operations are also simulated so that a user can visualize a machine tool operating as it will occur on the shop floor.

The strengths of the CATIA V5 CAM offering include the PPR architecture, the openness of the CATIA software, data integration throughout CATIA, the associativity that is provided, the integration with ENOVIA or SMARTEAM for collaborative product data management, CATIA design functionality, use of Knowledgeware for greater programming automation, the TDM system for tool storage and selection, feature recognition and feature-based machining, multi-slide machining, the tight integration between toolpath definition and simulation, and additional functionality provided by the DELMIA digital manufacturing product suite.

CATIA V5 CAM has often been perceived as a full function product, but one that is expensive and difficult to use. With V5 R16, these shortcomings have been addressed and considerable improvement has been made. In late 2005, IBM introduced the PLM Express for Machining offering that is intended for use by small and medium sized manufacturers. This offering greatly expands the scope of CATIA usage by providing a lower cost, easier to use product capability that can be employed by casual users.

Areas for potential improvement in CATIA V5 include support of 3 + 2 and 5-axis simultaneous machining for mold and die operations, a specific product and marketing program for shop floor machining, development of a specific product for electrode extraction, support of the trochoidal machining strategy, and support of plunge milling. In addition, several users indicated to CIMdata that they would like more direct contact with Dassault Systèmes CATIA development people and a more efficient process to request and receive product enhancements.

DELMIA, a Dassault Systèmes entity, is an industry leader in providing digital manufacturing software for manufacturing engineering and production operations. An integrated product suite is available for process planning, plant layout and design, simulation of factory floor assembly operations, worker ergonomic considerations, robotics programming, dimensional inspection programming, and machine tool simulation.

Dassault Systèmes is now integrating the DELMIA and CATIA product families both organizationally and technologically. The DELMIA product portfolio is built on the V5 architecture and a single person manages architectural control of all Dassault Systèmes PLM products. Development of all manufacturing products whether DELMIA or CATIA is also managed by a single person. DELMIA developed products have been embedded within CATIA V5 and CATIA developed products have been embedded within DELMIA. Currently, some products are available in CATIA V5 only, and some are available in DELMIA only, but a growing number of products are available under both brand names. The merging of CATIA V5 and DELMIA products on the V5 architecture has significantly broadened and strengthened both product families.

Dassault Systèmes encourages third party software vendors to implement on the V5 platform. Hitachi Zosen Systems (HZS) and Nihon Unisys are Japanese software vendors that are developing new CAM systems on the V5 architecture. HZS is developing a new version of Space-E and Nihon Unisys is developing a follow-on version of Togo that was originally developed for Toyota. HZS and Nihon Unisys are both major CAM suppliers in Japan, but currently have limited market presence in other geographies. Both companies are focused on the mold and die marketplace. Offering V5 as a product development platform permits Dassault to obtain maximum leverage from their own resources and the V5 architecture. It further provides for added market visibility, user acceptance, an increase in compatible software and additional revenues to Dassault Systèmes.

IBM provides the primary marketing, sales and customer support for the CATIA, ENOVIA, and SMARTEAM brands. The strength of IBM is in the sales and service provided to large aerospace and automotive enterprises. These two industries account for approximately two-thirds of CATIA-based revenues. Additionally, IBM has developed an extensive network of business partners to augment their sales and marketing. The business partners are mostly focused on working with small to medium sized companies. DELMIA has its own distribution channel, but IBM supports DELMIA products in large named accounts and sells DELMIA products that are available within CATIA. IBM interfaces closely with Dassault Systèmes, who provides product planning and development.

On the basis of end user payments, IBM/Dassault Systèmes was ranked by CIMdata as the worldwide largest CAM software vendor in 2004. With over 100 people in CAM related product development Dassault Systèmes has the largest number of CAM software product developers in the industry. IBM/Dassault Systèmes is expected to remain as one of the principal suppliers of CAM software for the foreseeable future.

CATIA Version 5 was introduced in 2000. CATIA V5, Release 16, which was announced in November 2005, was the product evaluated in this review. A release of CATIA is currently being made approximately every nine months. CATIA Version 4 is now in a support mode only. The majority of CATIA customers are now on V5.

1. Operating Environment

CATIA Version 5 supports a single design and manufacturing model and full data associativity among product design, tool design, and manufacturing operations. Data integration is provided among all CATIA V5 design and manufacturing applications. Using the same system in design and manufacturing provides synergy in training, reduces administration costs, and minimizes errors. There is no loss of time due to data transfer or preparation. A manager at a Danish pump manufacturer stated that "CATIA is a complete system and this is why we only employ a single system for design and manufacturing. We also like the tight integration among the modules." A change in the design model propagates to NC. This permits toolpaths to be quickly updated as design changes are made. As in other associative systems, a consistent topology is required for associative operations.

The CATIA V5 process-oriented architecture is based on a Process-Product-Resources (PPR) data model. Process includes a description of the part operation, manufacturing program and machining operations. It is composed of data, a user interface to access the data, and an operation sequence. The product is the full design definition of the product to be produced. It describes the NC setup including the geometry of the design part, stock, fixtures and the geometry. Manufacturing resources encompasses a description of what is physically required to produce a part. It includes the machine tools, pre-defined tool assemblies, cutters, holders and inserts.

The PPR model is the backbone for all applications. It captures process knowledge for best practices. It provides for a high level of associativity between product engineering, manufacturing processes and resources. The impact resulting from a change in design or manufacturing is managed by the PPR. All applications have a shared view of the PPR data model. The model is supportive of the concurrent engineering methodology being utilized in many companies.

The PPR tree allows users to see all operations linked to a process, as well as the geometry required for the operations definition and required resources. The PPR infrastructure also includes a tool library, catalogues, and other databases, the process modeler, software for material removal simulation, and an API for software customization. Effective change management and support of design variants, families of parts, and process templates is available by use of the PPR.

The CATIA V5 user interface is a native Windows implementation. It is common across all V5 applications, provides a clean look, makes full use of icons and is very visual. It incorporates intelligence and is intuitive and flexible. It features graphical feedback on user actions, mouse sensitive pictures, graphical help in defining parameters, drag and drop and cut/copy/paste capabilities, and customizable workbenches. An ample work area is available on the screen.

ENOVIA and SmarTeam both provide a cPDm (collaborative product data management) capability. ENOVIA is generally employed in enterprise environments and SmarTeam is better suited for dedicated installations. Both provide change management. The cPDm system tells a user that a change has occurred and which documents show the change. The design compare, in which one design version is compared to another and the differences between the two are highlighted, is done in CATIA.

Knowledgeware is a core element in CATIA V5. It provides the capability for all levels of automation. All CATIA applications can utilize Knowledgeware to capture process knowledge for a consistent set of best practices. It creates formulas, parametric families of parts, and machining rules based on customer know-how. All machining operations and parameters, machining features such as hole characteristics, tolerances, etc. and machining resources are captured. NC objects and attributes are treated as Knowledgeware objects. CATIA offers intelligent machining processes. CIMdata is favorably impressed by the Knowledgeware capabilities.

Figure 2—Querying the Knowledge Base

Knowledgeware can be utilized for both adaptive and generative knowledge-based machining. For adaptive knowledge-based machining it can be used to capture machining processes and then permit the user to reuse directly or modify these processes in a similar situation. For generative knowledge-based machining, a process can be defined by a set of rules to employ specific operations in a given order under a given situation. Machining processes can be interactively built and saved in a catalogue.

Rules in the form of checks, queries, and formulas to automate the programming process are employed in Knowledgeware. Checks are employed to constrain the machining operation or process. Queries are used to define criteria for cutting tool selection. Formulas are utilized to compute machining parameters.

IBM/Dassault Systèmes emphasizes the manufacturing knowledge management aspects of CATIA V5. It contains three levels of automation for machining processes. Level 1 is for simple machining processes without a link to feature parameters. In this case a user selects a process from a catalogue and manually applies it to points or a pattern. Level 2 applies to complex machining processes with a link to feature parameters. In Level 3, complex machining processes have a link to feature parameters and checks and formulas are employed.

An automation method employed in CATIA V5 is to define a complete set of potential operations for a process such as hole making. In this case, the full set of operations could include spot drilling, pre-drilling, drilling, tapping, and reaming. Then in a given situation and based on rules contained in Knowledgeware, some operations could be performed and some could be bypassed. This is an efficient methodology, as it reduces the number of processes to be defined.

The PPR data model is used to support CATIA V5 process templates. A process template includes the processes, product, and resources for all operations. Parts programs are captured and processes are re-used. The process document lists all operations including tool changes. A link is established from the process list to the product list and the resources list. To find prior processes employed one could use SmarTeam or create independent files. The process template from a prior part can be applied to the geometry of a new part. A graphic of both parts is shown to the user. A panel tells what needs to be done and what is left to do. Users can capture processes and place them in a catalogue. Experience has shown that the time to create a program from a process template is approximately an order of magnitude less than that required for the original program.

NC Setup and CATProcess are two key documents in CATIA V5. They are linked together and they support the data integration and links that are established. NC Setup is linked to the definitions for design parts, stock, fixtures, NC geometry, and the in-process model (IPM). CATProcess has links with the definitions for machining processes, tool representation, process knowledge, toolpaths, post-processor tables, simulation results, shop floor documentation and toolpath outputs.

Direct translators are provided between CATIA and other CAD products such as Multi-CAx Plug-ins for Unigraphics NX Design, PTC Pro/ENGINEER, and SolidWorks. CATIA supports IGES and STEP so that geometry translations can be done via these products. However, design intent can be lost by use of IGES and STEP as some information such as tolerances or features are not transferred by these methods.

2. Manufacturing Modeling

CATIA V5 is well recognized as one of the premier CAD products for conceptual and product design. Most of the worldwide largest automotive and aerospace manufacturers use CATIA. It permits a user to create product designs, manipulate imported data, prepare a design for manufacturing, edit and modify complex surfaces, create volumes and wire frame models without limitations, and design the tools required for manufacturing. At any time, a CATIA user has the tools necessary to create, analyze, and edit geometry. One can add fillets, split part geometry, smooth parting surfaces, create electrodes and inserts, find and implement engineering changes, and detail tooling components.

The CATIA V5 modeler is an integrated hybrid surface and solids-based feature-based design system. The same modeling tools are used independent of whether the model is defined in surfaces or solids or a combination of both. Complex geometries can be created in a single hybrid environment using the same user interface. CATIA is generally recognized as the industry leading CAD product for design of complex surfaces. Class A surfaces are routinely produced by users in the automobile industry. Surfaces can be manipulated and curvature continuity can be maintained. A range of parametric features are available for generating objects and defining machining tasks.

The Healing Assistant is available to perform a validity check on imported geometry with regard to V5 modeling criteria. It can be employed to prepare a part for V5 modeling by topological and geometrical healing of a model. Some of the capabilities include a draft angle function to show undercuts, a check of the validity of surfaces, edges, and faces, a function to join curves and surfaces, and a function to heal all surfaces or selected edges of a surface.

A model compare function is contained in the Digital Mockup or DMU module to detect and visualize product changes. Color is used to show differences between models, the area of change and what has changed. The software will identify machining processes that require changes. When a designer modifies a model, the PPR software generates a message for other users that a change has occurred.

For mold design, CATIA V5 automatically separates a model into the core, cavity, inserts, and sliders. An exploded view of the different mold areas is provided. Factors can be applied to a model to recognize either variable or non-variable shrinkage. A parting or split line can be established on either a surface or a solid model created with CATIA or any other design system. This is accomplished by knowing the pull direction on the model. The user has the option of modifying the parting line. The software automatically places surfaces on either side of the split line on either the core or cavity. If the software is unable to decide, the surface is identified and the user manually places the surface in the core or cavity. Since CATIA is associative, if a part dimension changes, the dimensions of the core and cavity will automatically change to compensate for the modification.

There is currently no specific program for creating and extracting electrode geometry. This function is performed by use of a macro within the CATIA V5 design module or through a third-party solution. The intent is to obtain this capability through a CAA partner.

CATIA V5 includes the Mold Tool Design, MTD product for design of mold bases. MTD is an intelligent, parametric, associative, easy to use, solids-based system that includes support for either single or multi-cavity molds. It supports design of the ejection, injection, and cooling systems within a mold. A process guide is available to lead a user through the mold design process. A variety of mold base catalogues, including DME and Hasco, are supported. Output from MTD includes an itemized BOM, assembly drawings, drawings of each plate, and an integrated application for automatic plate drilling.

An integrated die design and manufacturing application is provided in CATIA V5. It includes products for body in white modeling, stamping line simulation and optimization, and press die design and manufacture. Products, including VAMOS, a CAA product, are available for design and simulation of the parts to be produced, the dies to create the parts, and the processes to produce the components. To support process design, functions are available for process planning, workcell setup and assembly simulation. CATIA V5 also includes a capability for design of progressive dies.

3. Basic Machining

CATIA V5 machining contains two levels of products, P1 and P2, and multiple machining configurations. There are only four P1 products, which are Lathe Machining, Prismatic Machining, NC Review, and STL Rapid Prototyping. P2 level products include the P1 products plus Multi-Pocket Machining, Multi-Slide Machining, 3-axis and Multi-Axis Surface Machining, and multiple products for simulation. Specific configurations are available for prismatic machining, mold and die machining, and advanced machining. A user in a U.S. consumer products company commented, “CATIA is a very powerful product, it is versatile and there has been no job that I could not do.”

Depending upon the operation being performed, CATIA V5 machining is done on either a surface or a tessellated surface. The software controls which technique is employed, dependent upon the quality of surface desired. A tessellated surface is most commonly utilized by CAM software suppliers for machining, as it is typically faster and less likely to gouge. However, milling directly on a surface can result in greater accuracy and a smoother surface finish.

Dassault Systèmes has steadily enhanced the speed of computation in CATIA V5. As a benchmark, Dassault Systèmes selects several independent CAM systems and then strives to equal or exceed their speed of computation. Enhanced speed is key to obtaining highly interactive toolpath generation and simulation.

A focus of CATIA V5 R16 machining is on hard milling. In hard milling a primary objective is to avoid tool breakage. To avoid tool breakage, use of climb milling and avoiding full engagement of the tool are emphasized, even though this could result in a longer tool path. If soft material is being machined the shortest tool path is usually the objective.

Roughing or volume milling strategies include stock spiral, parallel cut, spiral cut inside-out or outside-in, roughing, re-roughing, and Z-level roughing. Roughing is typically done with Z-level roughing. Cutting can move along a part either level-by-level in which the inner and outer cuts are in the same operation or outer and inner cuts can be done sequentially. Roughing and re-roughing of material left behind can be accomplished in a single command. By use of the in-process model (IPM) the software knows where excess material remains.

Two new roughing strategies are available in V5 R16. They are concentric for roughing of internal surfaces, such as in a pocket, and offset on part for outer surfaces. Concentric roughing is a helical motion where movement is always in the same direction. Internal cuts are always done with

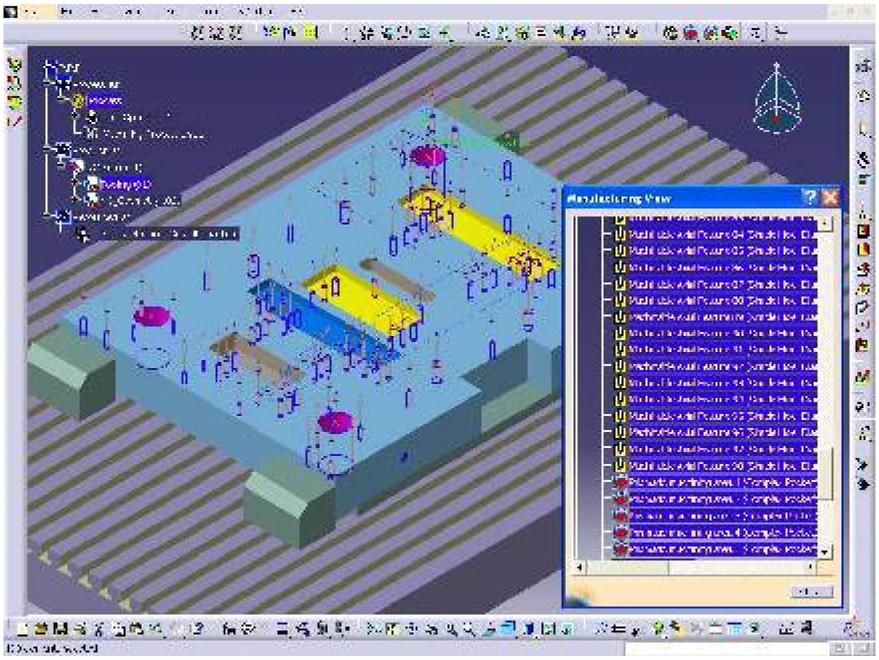

Figure 3—Feature Recognition

climb milling and a limited engagement of the cutter. The machining operations are managed through a process table.

Plunge milling is currently not included in CATIA V5, but IBM and Dassault Systèmes view this as a high priority for a subsequent release. They believe it to be an effective roughing strategy for hard material when a large amount of material is to be removed. It also has the benefit that machining can be done on older machines since accuracy is not required.

Feature Recognition and Feature-Based Machining

CATIA V5 includes a strong capability for feature recognition. This substantially accelerates the preparation of design parts for NC programming, particularly in those parts with a large number of features. The software recognizes design and manufacturing features in an isolated boundary representation part model. It converts part design features from native CATIA V5 into machining features taking into account geometrical and technological parameters such as tolerances and threads. Typically, design features are used to build up or create a model, whereas a manufacturing feature is obtain by material removal. The software also recognizes prismatic machining features in design parts without feature information such as those part models created in other CAD systems. It places the features and associated patterns into a process document for viewing.

A basic approach for users is to start with global feature recognition, find new features, and then convert design features into manufacturing features. A user can select only holes and then create patterns of holes of the same or similar size and tolerance and whether they are through holes or blind holes. A maximum diameter for a hole can be established and if the hole exceeds the diameter it is then presumed to be a pocket. The feature recognition library was developed in an R&D lab in India jointly owned by Dassault Systèmes and GSSL, an Indian company. Feature recognition in CATIA V5 R16 applies to prismatically shaped parts. It does not apply to free form parts.

Feature-based machining is accomplished by sequentially applying a process to each feature. The available processes are in a catalogue. A user at a major German automobile manufacturer remarked, “By defining processes for each feature, storing them in a catalogue, and then reusing the process in a similar situation we are able to reduce our programming time in half.” Knowledgeware is utilized to determine which operations in a process are to be employed. Rules and constraints can be applied. For example, spot drilling could only be performed if the feature or hole size is greater than a given amount. Formulas can also be applied. The proper tool to be employed is defined in a tool query, which is linked to a feature. The tool characteristics

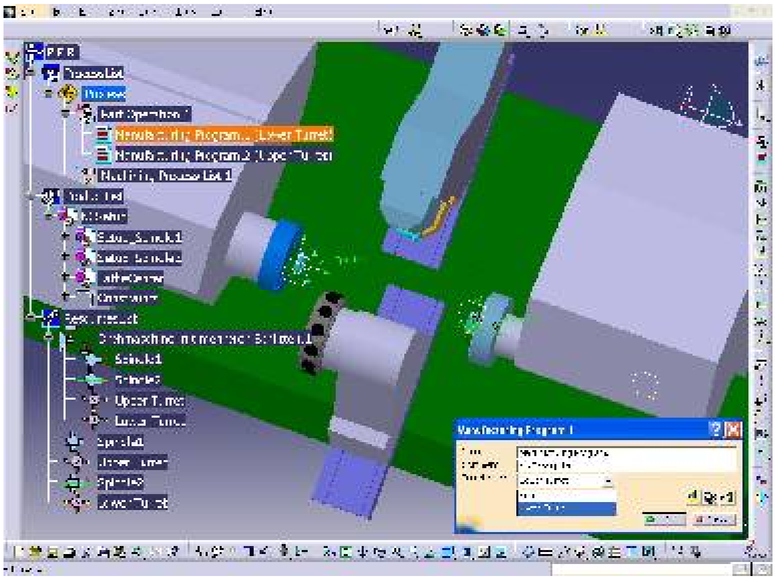

Figure 4—Lathe Center Programming

required could specify tool diameter and length or they could be specified in relation to the hole characteristics such as diameter, depth, tolerance, through hole or blind hole, etc. Tool changes are established and listed. Processes can be optimized and re-ordered to minimize tool changes. Users can establish a link between a process and features or it can be done automatically.

Complex Turning and Mill-Turn

CATIA V5 has a robust capability to support complex turning machines and mill-turn machines. These types of machines represent a growing trend in the machine tool market. They reduce the number of machine tools required for turning, milling, and drilling by performing multiple operations on a single machine, reduce the shuttling time between stations, and limit the number of setups.

This type of complex machining is facilitated by use of the Multi-Slide Lathe Machining module (MLG) of CATIA V5 R16. MLG is used to program parts on turning machines with multiple turrets and spindles. It does not create a toolpath. Instead, it manages and synchronizes all movements of a machine tool. An objective is to minimize setups. Turrets and spindles can be defined when programming milling and turning operations in a multi-task and multi-axis context. All turning operations are performed in MLG, the basic lathe product. Swiss turn machines are not supported in V5.

CATIA V5 supports balanced or pinch turning on a 4-axis lathe, multi-axis lathes, sub-spindles, c- and b-axes, and mill-turn machines. There are no limits on the number of turrets supported. For 4-axis lathe operation, one can start with a standard turning program utilizing one turret and one spindle and then alter it to function with two turrets and one spindle, or the user can program directly for 4-axis operation. Two turrets, upper and lower or master and slave turrets are selected. The two turrets concurrently move along a spindle cutting a part in a balanced or pinch turning operation. A program is created for each turret. There are as many programs as there are turrets. Any spindle can be addressed within a program. A spindle is selected and turrets are assigned to a spindle.

For each turret, a Gantt chart is prepared that depicts the amount of time used on each operation, tool change time, and the amount of dead time. Synchronizations between operations can be set. The synchronization is done manually. The user establishes the synchronization marks to control when each operation is performed. The Gantt charts also list the total duration time of the operation. An objective is to minimize the total time by maximizing the cutting times for each turret and minimizing dead time. If more idle time is shown on one turret than another, operations are moved from one turret to another.

Mill-turning is becoming an increasingly important machining function as most lathes now include a milling attachment. The capability to perform milling and turning on the same tool is particularly important in production machining operations, and it is also required in most general purpose machine shops. CATIA V5 is one of the limited number of CAM software products to effectively support an integrated mill-turn capability.

Mill-turn machines are programmed similarly to that of 4-axis lathes. The primary difference is that one of the turrets supports a milling head instead of a turning tool. The milling turret is programmed using a standard CATIA V5 milling module, which could be a 5-axis operation. All CATIA V5 milling operations are supported in mill-turn.

A visual time-based replay is shown in CATIA V5 to simulate all operations by time, including tool changes as well as cutting operations. There is a different color for each operation. For example, roughing operations could be in red and tool changes in black. The tool and full turret are shown. If a part is moved from the main spindle to a sub-spindle this is shown in the simulation. The output can be a single file or a program. Synchronization is done in the post-processor. Checks are made for any collisions among the moving turrets and the part. The replay can be from the start time of any operation selected. Simulations are only done on the CATIA toolpath file prior to post-processing. At a later time simulation will be done on the ISO code after post-processing. All programs can be output to a single file.

Tool Data Management

For tool data management, IBM/Dassault Systèmes offers either pre-defined tool catalogues or the TDM system. Catalogues provide a basic capability. Data is generally stored on an Excel worksheet or on a Lotus Workbook. Tools and tool holders can be visualized in both toolpath replays and material removal simulations. A user defined dynamic creation of a cutter and holder based on geometric parameters can be developed. However, some users have indicated that the time required to define cutting tools can be excessive.

TDM provides a much more extensive capability for tool data management. It is offered through TDM Systems, a CAA partner. TDM is integrated within CATIA V5, provides a scalable approach and avoids redundant tool information. It includes information on cutting tools and tool assemblies. It provides a realistic representation of a tool as well as many characteristics of the tool geometry, the tool technology, and appropriate feeds and speeds for each tool. A 3D graphical tool representation can be created using geometrical parameters or DXF files stored in the TDM database. A realistic representation in toolpath replay and material removal simulation is provided. TDM supports milling, drilling and turning tools, cutting conditions and NC simulation.

Users have direct access to the TDM database. Tool queries can be done inside CATIA V5 or TDM using the CATIA Search Tool panel or TDM search capabilities. Tool lists can be loaded from the TDM database into CATIA and exported back into TDM. The technology index specifies the appropriate speeds and feeds for each tool and these can be automatically updated. TDM is most appropriate for large aerospace or machinery manufacturers. See Figure 2.

4. 3-Axis Milling

3-axis milling can occur on a model containing solids, surfaces or both. A wide variety of machining strategies for surface milling are supported. Finishing machining strategies available to a user in CATIA V5 R16 include:

- Single surface machining
- Parallel plane machining
- Spiral inside out and outside in
- Radial cutting with an angle stepover around a point
- Layer-by-layer milling both horizontal and vertical
- Combination cuts in which different strategies are employed for flat and steep areas separated by an angle within a single toolpath

- Flowline machining either parallel or perpendicular to flowlines
- Free path machining along any curve
- Engraving by tracing geometry and projecting a 2D curve onto a surface
- Re-machining including rest machining and pencil tracing
- Use of a 3D stepover

Trochoidal machining, a strategy that utilizes a circular tool motion that is partly on and partly off the workpiece, is not available in V5. This strategy is often employed in high-speed machining to obtain an improved surface finish and in roughing when deep material is present.

Rest material machining is accomplished on either block stock or a 3D cast model. The amount and location of material remaining is computed by having knowledge of the original stock, final part model, toolpaths previously generated, and cutting tools that have been employed. An in-process model is generated and after each successive machining step, the IPM is automatically updated. In rest machining, a smaller cutter is selected, the toolpath is re-computed, and machining only occurs in those areas in which material remains. All rework is done in a single operation. The depth of stock remaining is not displayed to the user in machining, but is presented in the toolpath replay simulation. Pencil tracing is also provided. In this case a final single pass is made with a small tool along the intersection of surfaces to provide a fillet and smooth the surface transitions.

For control of toolpath stepover, CATIA V5 contains perhaps the widest variety of options available in the industry. It includes both 2D and 3D stepover, control by maximum scallop height left on a surface, by the number of passes, by a relationship such as percent of tool size, by an angle stepover for radial machining, and by use of an aiming surface strategy. Some techniques can be used in combination, such as control by maximum scallop height and 2D stepover. In a 3D stepover, the stepover distance is measured along the surface of the part, as compared to being measured on a plane above the surface, which is the case in a 2D stepover. In an aiming surface strategy, which is relatively unique, a virtual surface is created. Flow lines are defined above the surface and then projected down on the actual surface. This stepover option is particularly appropriate if there are patches on the surface.

High-speed machining continues to gain in importance in cutting of molds and dies. CATIA V5 includes a strong capability to support this type of machining. IBM considers the three primary criteria for high-speed milling to be no sharp corners, constant engagement of the cutter with the material, and to perform climb milling whenever possible. A large number of features are included in CATIA to specifically support high-speed milling including NURBS output as an option in all 3-axis and 5-axis machining strategies, a helical approach to the material, a constant chip load in roughing, loops added to the toolpath, rounded rapid moves on an arc, and control of tool penetration.

CATIA V5 includes a process focused solution to machine multi-cavity parts such as mechanical prismatic parts and aerospace structural parts. It provides users with a solution to machine these parts with a mix of roughing and finishing toolpaths. The available toolpaths include back and forth with cornerization, helical, and concentric. Z-level passes are synchronized with the center passes. Feedrate optimization is provided in sharp corners and inside arcs for the helical strategy.

A process oriented cycle for multi-pocket milling is offered in CATIA V5 R16. It can be performed either pocket-by-pocket or level-by-level over multiple pockets if thin walls are present. One operation can be used for the entire pocket to cut both the sides and the bottom. A pocket can be roughed and finished with one tool and one toolpath. Or a user could machine all areas of the pocket with a large tool and finish with a smaller tool. As in other operations an IPM is employed so the user knows the amount of material left after each cut. Multi-pocket machining is only available in 3-axis milling.

5. 5-Axis Milling

CATIA V5 is one of the leading products for programming continuous 5-axis machines. Simultaneous 5-axis machining is becoming more common as the price of these machine tools continues to decrease, they become easier to program, and the benefits become more accepted. Continuous 5-axis machining offers a number of advantages over 3-axis, and other machining processes. These include a reduction in the number of individual operations and setups required to machine complex parts, use of the most rigid cutters since smaller and shorter tools can be employed, greater accessibility, and the ability to machine undercuts. It often results in reduction or elimination of electrodes required for EDM operations.

Simultaneous 5-axis milling permits keeping the tool axis normal to the work surface, which results in an improved surface finish. Machine time is reduced and product quality is usually better since workpieces do not have to be moved from machine to machine for multiple operations. For complex shapes, users can machine 5-sides of a part without special fixturing, jigs, or manually re-positioning. The ability to re-orient a tool in mid-cycle to allow milling, drilling, and tapping with only one setup also can be a significant benefit.

An APT-based approach is employed in which the drive, check and part surfaces are determined. Machining is typically done along a drive surface, normal to an auxiliary surface placed over the surface to be machined, while taking check surfaces into account. The user can specify the reference lead angle and maximum and minimum lead angle and the fixed tilt angle of the tool. A change in the lead angle would be performed if collision checking is active and the algorithm can compute a better tool position within the allowed change in lead angle. The variable lead mode is intended to avoid a collision between the rear side of a tool and the part.

Five-axis only machining strategies include flank or swarf cutting, and multi-axis helix machining. In flank machining cutting is done on the side of a tool. It is not dependent upon geometry. It is mainly used for semi-finishing and finishing 5-axis walls in structural parts. The flank contouring code is provided to Dassault Systèmes by NCCS, a California-based NC software firm.

Flank machining includes a fanning capability that can be employed anywhere on the toolpath or throughout the toolpath. Fanning is required when going around a corner or in other non-linear moves. In a basic fanning strategy, the tool axis continually and uniformly moves on an interpolation path between the start and end position of a curve. It ensures good continuity through different tool motions. The tool is tangent to the drive surface at a given contact height. In other fanning strategies the tool axis stays normal to the part in the forward direction with fanning at the beginning and at the end of the tool motion or the tool follows the isoparametrics of the surface with fanning at the beginning and at the end of the tool motion.

Multi-axis helix machining is analogous to potato peeling as the tool encircles a blade following a defined contour. It is mainly employed for semi-finishing and finishing blades and disks in turbo machinery parts. An interpolation tool axis strategy allows the user to manually define axes in order to have better control on the tool. A user can select existing axes and adjust parameters in a dialog box or select axes that have previously been created and stored. In each case one can use the display tool option to avoid a collision.

Other multi-axis surface milling strategies in CATIA V5 include:

- Multi-axis drilling on a surface where drilling is normal to the curved surface
- 5-axis roughing
- Profiling
- A multi-axis sweeping operation in which the toolpath is executed in parallel planes
- A multi-axis contour driven operation in which the tool is driven along a contour. Three machining modes are: parallel contour in which the cutter moves parallel

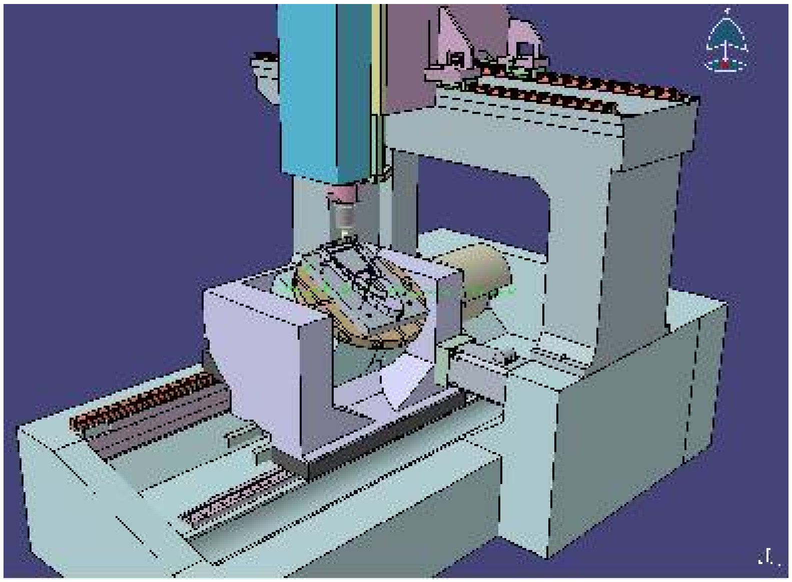

Figure 5—Machine Tool Simulation for Aerospace

to a guide curve, between two contours along the iso lines of a closed contour, and spine contour in which the cutter moves in a straight line normal to a guide curve

- A multi-axis curve operation in which the tool's side, tip or contact point is driven along a curve
- A multi-axis isoparametric operation in which the user selects strips of faces and machines along their isoparametrics

In CATIA V5 R16 there are nine ways for a user to define the tool axis. They are by lead and tilt, fixed lead and tilt, through a point, normal to a line, 4-axis lead and lag, optimized lead, interpolation, tangent axis and 4-axis tilt.

CATIA V5 outputs NURBS in 5-axis as well as in 3-axis milling. This is a relatively unique capability for 5-axis machining. However, Cenit appears to be the only firm to provide a post-processor that supports NURBS in 5-axis. IBM believes that the use of NURBS provides a better surface finish and results in less finishing time.

In CATIA V5, gouge checking is performed on all moves and excellent gouge avoidance is provided. If a gouge does occur it will be shown by color. If the toolpath is truncated to avoid gouging, the software will record the location and mark the toolpath file as having been truncated.

The Advanced Machining module combines Prismatic Machining, Surface Machining, and Multi-Axis Machining into a single package that includes all capabilities required for aerospace or other complex machining users. Dassault Systèmes has developed their 5-axis capability by targeting the needs of aerospace companies.

Whereas some vendors have developed specific cycles for 5-axis machining to perform tasks such as machining of turbine blades, impellers, tubes, pipes, and tire molds, Dassault Systèmes does not plan to have multiple granular cycles, but instead to offer more generic cycles.

CATIA V5 R16 does not include a capability for 3 + 2 milling. This type of machining can be applied to the cutting of high, steep walls that are often found in dies, cores, and deep cavities. In this operation the tool can be tilted relative to a machining plane. Or a 2-axis tilting rotary table can be added to a 3-axis machining center so that parts can be positioned at different angles by tilting the table and rotating the parts. This type of machining permits the use of shorter tools for deep cavities and helps avoid collisions. Instead, CATIA employs several 3-axis operations, each with a different sized tool to machine along a wall level-by-level.

6. Toolpath Simulation and Machine Tool Control

CATIA V5 R16 includes several capabilities for visualization and simulation. The basic CATIA V5 visualization capabilities are in Photo and Video. These have been components of CATIA for some time. NC Machine Tool Builder and NC Machine Tool Simulation have been DELMIA

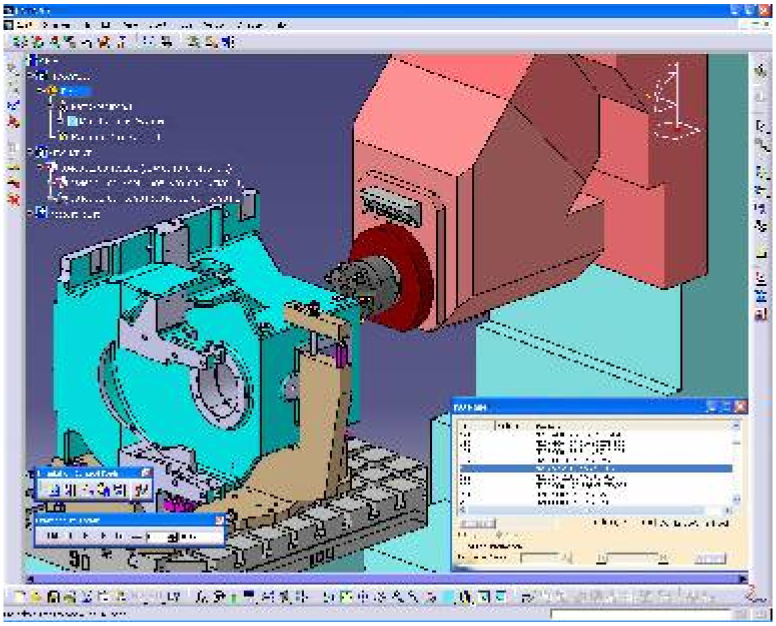

Figure 6—Simulation of ISO Codes

products in the past, but since V5 R16 they are available in both the CATIA V5 and DELMIA portfolios. The graphics and visualization provided in the simulation suite of products is excellent.

Photo and Video are integrated with CATIA toolpath generation. Photo provides a snapshot or photo of the updated stock after a cutting operation. The remaining material can be seen and any collisions or gouges are shown. One can zoom on a local area for more precise analysis. By picking geometry a user can obtain relevant information such as the tool used and depth of cut.

Video provides a replay of the toolpath depicting the material being removed, the remaining material, gouge and collision detection considering the tool, holder and fixtures, and analysis and validation of the toolpath. The integrated simulation of the cutting parameters helps deliver an optimized toolpath. Tests are made such that if the feed rate is greater than the machine capability, it is shown. MachineWorks code is utilized as the core capability in the material removal simulation provided in Video. Video simulation is done on either the CATIA toolpath or the ISO code after post-processing.

When simulating on the ISO code the simulator sees the same code as is provided to the machine tool controller. The controller is also emulated in the simulation so an accurate representation of machine tool movement is obtained.

Video shows the thickness of material remaining after each operation and the operation and tool that was used in the process. The material remaining is computed by comparing the IPM with the final part model. The video result can be saved for the next operation and one can start a simulation from a previously saved result. The simulation shows both a graphic of the operation and a line by line listing of the code being simulated. A user can pick any line and a graphic shows the tool on the part.

In collision detection the complete tool, tool holder and fixtures are considered in both 3-axis and 5-axis milling. The system depicts which operation and tool generated the collision. The collision detection computations are performed against the in-process model. The finishing IPM from one operation becomes the stock for the next operation. If a collision occurs it is up to the user to manually change the operation in some way to avoid the collision. The user can stop the process, consider optional tool holders and shanks that have been previously defined, change the toolpath or switch from 3-axis to 5-axis.

Being highly interactive, testing can be done in the CATIA V5 CAM system during program creation. It is not necessary to create a separate simulation file and/or wait until programming is complete to run Photo or Video.

NC Machine Tool Builder is a resource modeling tool employed to create a model of a machine tool. There is a single user interface and a single machine tool description with geometry, kinematics and technological information. It is useful in planning, simulation and verification, post-processing and controller emulation. The same information is required to build a model as is needed to create a post-processor. A user is provided with a machine tool definition from a catalogue, post-processor table, and emulation of the controller to be employed.

The Builder incorporates machine attributes such as travel limits for the X, Y, Z, A, and B axes, home positions, tool change positions, machine table and spindle definition, and machine kinematics. Speed and acceleration control of the axes is provided. The controller emulator translates ISO code into machine moves and allows the NC machine simulation to be driven by the ISO code file. One can save machine information with all its attributes for re-use in process planning, NC detailing and simulation. The Builder supports milling machines. Mill-turn and turning machines will be supported in a subsequent release.

Machine tool models could be developed by the end user, IBM, a CATIA reseller, or one of the three post-processor business partners supporting CATIA. A number of example models are shipped with the CATIA V5 R16 software. Use of the Builder can also be delivered to the end user through a service offering.

Once a model of a complete machine tool, including all peripheral devices is completed, a full simulation of the machining operation can be run using the NC Machine Tool Simulation product. Fixtures and other holding devices can be modeled. Any collisions with the part, machine tool or holding devices can be detected. This interactively avoids collisions by teaching machines collision-free paths or by editing machining operations and thereby reducing programming lead time. The user can visualize the collision, analyze collision curves, determine the amount of interference and print collision reports. Material removal is not shown in the simulation. Use of the Machine Tool Builder and Simulator is expected to be most useful when expensive parts are being machined and with complex machine tools. As with Video, simulation can be performed on either the CATIA toolpath or the ISO machine tool code generated in a post-processor. There is a synchronized display of the machining operations and the ISO code during simulation.

Typically, simulation is done on the CATIA toolpath during programming development and simulation on ISO code at the end of the process. Simulation of ISO code provides greater confidence for the user. Photo is mainly used for two- through three-axis machining, Video supports two- through five-axis milling and turning.

Toolpath generation and simulation can be computed quickly and with all products being under CATIA V5, a highly interactive capability is available. Simulation is a push button operation. No time is required to prepare the file for simulation. One can use the Photo/Video capability for material removal simulation and then do a full machine tool simulation, if appropriate.

For post-processing, Dassault Systèmes utilizes post-process builders and libraries from ICAM, IMS, and Cenit. All three companies are CAA partners of Dassault Systèmes. The intent is to allow customers to select and utilize the post-processor of their choice. Each of these products has been integrated into the V5 architecture. The toolpath file is directly passed to each of these post-processors. Other post-processors could also be utilized with CATIA V5, but the level of integration would be less. In this situation an APT file is generated in CATIA as the input to a third-party post-processor.

7. Product Plans

The IBM/Dassault Systèmes near term priorities for further product development of CATIA V5 include:

- Continued improvements in simulation to provide for stronger integration with toolpath generation, greater realism, and realistic simulation of machine tool controllers

- Offering process planning for machining and specifically for engine components
- Greater automation for prismatic and complex surface machined parts
- Maintaining a leadership position in 3- to 5-axis machining of molds and dies
- Provide a 3 + 2 milling capability
- Increased investment in 5-axis machining
- Further support of complex machines by moving from being driven by geometry to management of machining systems
- Expanding the network of CAA partners for product development