ISSN 2319 – 7757 EISSN 2319 –7765

Indian Journal of

Engineering

3D Modeling and virtual manufacturing of a machine component using CATIA V5 and MASTERCAM

Adarsh Adeppa[1] Shivkumar biradar[2]

1. Assistant professor, Dept of Mechanical Engineering BKIT, Bhalki Karnataka, India
2. Assistant professor, Dept of Mechanical Engineering BKIT, Bhalki Karnataka, India

Publication History
Received: 11 January 2014
Accepted: 19 February 2014
Published: 5 March 2014

Citation
Adarsh Adeppa, Shivkumar biradar. 3D Modeling and virtual manufacturing of a machine component using Catia V5 and Mastercam. *Indian Journal of Engineering*, 2014, 9(20), 6-13

ABSTRACT
It has been human nature to innovate, discover, invent new things and so has been his creation. Design may be pronounced as the synonym for creation. So there is no end to man's creation, design and hence CAD. By passage of time it will be even smarter, quicker and sophisticated. computer-aided design (cad) and computer-aided manufacturing (cam) hence there are several way of manufacturing similarly catia and mastercam. In the recent years product development is facing the challenges of maintaining mass productivity and reaching the customer recruitment. In this paper we are discussing about the method of combining the virtual prototype with design by analysis techniques. Final outcome of the report is to provide quality products, good accuracy by creating the prototype model. The main intension of virtual prototype for customized product development is to provide multidisciplinary design. . This design helps in utilization and capture of information generated during the design phase, and the simultaneous generation, at design time, of manufacturing, materials, costing, and scheduling data, together with visual evaluation of customer perception on target products, hence supporting the implementation of engineering.

Keywords: Virtual Prototyping, Mass Customization, Product Development, Design Management, Design by Manufacturing Simulation

1. INTRODUCTION

Narendra KomarlaM-Cme et al. (1999) CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systemes and marketed worldwide by IBM. Written in the C++ programming language, CATIA is the cornerstone of the Dassault Systemes product lifecycle management software suite. CATIA competes in the CAD/CAM/CAE market with Siemens NX, Pro/ENGINEER, Autodesk Inventor and SolidEdge. Several thousands of companies in multiple industries Worldwide have already chosen the Virtual Design capabilities of CATIA products to ensure their products Real Success. CATIA delivers solutions for the enterprise from large OEMs through their supply chains to Small and Medium Businesses. CATIA V5 is the leading solution for product excellence. It addresses all manufacturing

Figure 1

A Typical 3D model of an Aircraft Component generated in CATIA V5

organizations, from OEMs, through their supply chains, to small independent companies. The range of CATIA V5's capabilities allows for its application in a wide variety of industries, from aerospace, automotive, industrial machinery, electrical, electronics, shipbuilding, plant design, and consumer goods, to jewelry and clothing. CATIA V5 is the only solution that covers the complete product development process, from product concept specifications through to product-in-service, in a fully integrated manner. Based on an open, scalable architecture, it facilitates true collaborative engineering across the multidisciplinary extended enterprise, including style and form design, mechanical design, equipment and systems engineering, digital mock-up management, machining, analysis, and simulation. By enabling enterprises to reuse product design knowledge and accelerate development cycles, CATIA V5 helps companies speed-up 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.

1.1. Features

Commonly referred as a 3D Product Lifecycle Management software suite, CATIA supports multiple stages of Product development (CAx), from Conceptualization, Design (CAD), Manufacturing (CAM), and Engineering (CAE). CATIA can be customized via application programming interfaces (API). V4 can be adapted in the Fortran and C programming languages under an API called CAA. V5 can be adapted via the Visual Basic and C++ programming languages, an API called CAA2 or CAA V5 that is a component object model (COM)-like interface. Although later versions of CATIA V4 implemented NURBS, V4 principally used piecewise polynomial surfaces. CATIA V4 uses a non-manifold solid engine. CatiaV5 features a parametric solid/surface-based package which uses NURBS as the core surface representation and has several workbenches that provide KBE support. V5 can work with other applications, including Enovia, Smarteam, and various CAE Analysis applications. The version used in the current project is CATIA V5R17. It extends the unique 2D/3D associative approach for conceptual design and extends the 3D master approach by enabling the fast and convenient display of product information, such as tolerances and annotations, in a familiar drawing layout within the 3D environment. It boosts conceptual design within the 3D environment by enabling designers to easily create in-context 2D sketches from the automatic detection of existing 3D geometry displayed in the view background. Efficient, comprehensive, and standards-compliant CATIA V5 drafting capabilities always guarantee high quality when realizing drawing layouts and dress-up, whether immersed in 3D or in a separate drawing document (Figure 1). It is continually enhanced and updated to meet specialized needs, such as support for new customized symbols, strokes, and open-type fonts. CATIA V5R17 promotes 3D as the master reference for part and product definition. Designers can define and manage standards-compliant tolerance specifications and annotations linked to the 3D geometry, making them directly reusable for manufacturing planners and to be shared throughout the enterprise. Designers can easily present and share 3D tolerancing and annotation in a familiar drawing layout embedded in the 3D environment. V5R17 enables the rapid creation of associative views from Functional Tolerancing & Annotation views or captures. Users benefit from a more productive 3D annotation definition and layout process by realizing specific operations in a single step, such as directly managing the view ratio property for 3D annotation.

1.2. CATIA V4 to V5/V6 Conversion

Narendra KomarlaM-Cme et al. (1999) CATIA V5 and V6 can directly use the CATIA V4 models, but changes in the CATIA data structure require data conversion from CATIA V4 to V5/V6. This is due to both a change in geometric kernel between CATIA V4 and CATIA V5, and changes in the CAD data structure between CATIA V5 and CATIA V6. Dassault Systemes provides utilities to convert CATIA V4 data to CATIA V5 with a one-to-one mapping. Still, cases show that there can be issues in the data conversion from CATIA V4 to V5, from either difference in the geometric kernel between CATIA V4 and CATIA V5, or by the modelling methods employed by end users. Experiment results show that there can be data loss during the conversion (from 0% to 90%). The percentage loss can be minimized by using the appropriate pre-conversion clean-up, choosing the appropriate conversion options, and clean-up activities after conversion. Engineering service providers have solutions, but mostly they are unique to a particular company and its processes / standard of modeling method. A common solution for 100% data conversion has yet to be devised. It is important to note that ANY change from one modeling kernel to another would cause similar problems; this issue is not unique to CATIA.

Adarsh Adeppa,
3D Modeling and virtual manufacturing of a machine component using CATIA V5 and MASTERCAM,
Indian journal of engineering, 2014, 9(20), 06-13,

1.3. Notable Industries using CATIA

CATIA is widely used throughout the engineering industry, especially in the automotive and aerospace sectors.

Aerospace

The Boeing Company used CATIA V3 to develop its 777 airliner, and is currently using CATIA V5 for the 787 series aircraft. They have employed the full range of Dassault Systemes' 3D PLM products — CATIA, DELMIA, and ENOVIA LCA — supplemented by Boeing developed applications. Chinese Xian JH-7A is the first aircraft developed by CATIA V5, when the design was completed on September 26, 2000. European aerospace giant Airbus has been using CATIA since 2001. Canadian aircraft maker Bombardier Aerospace has done all of its aircraft design on CATIA. The Brazilian aircraft company, EMBRAER, use Catia V4 and V5 to build all airplanes. The British Helicopter company, Westlands, use CATIA V4 and V5 to produce all their aircraft. Westlands is now part of an Italian company called Finmeccanica the joined company calls themselves AgustaWestland.

Automotive

Many automotive companies use CATIA to varying degrees, including BMW, Porsche, Daimler AG, Chrysler, Audi, Volkswagen, Bentley Motors Limited, Volvo, Fiat, Benteler AG, PSA Peugeot Citroën, Renault, Toyota, Ford, Scania, Hyundai, Škoda Auto, Tesla Motors, Proton, Tata motors and Mahindra & Mahindra Limited. Goodyear uses it in making tires for automotive and aerospace and also uses a customized CATIA for its design and development. Many automotive companies use CATIA for car structures — door beams, IP supports, bumper beams, roof rails, side rails, body components — because CATIA is very good in surface creation and Computer representation of surfaces.

Shipbuilding

Dassault Systems has begun serving shipbuilders with CATIA V5 release 8, which includes special features useful to shipbuilders. GD Electric Boat used CATIA to design the latest fast attack submarine class for the United States Navy, the *Virginia* class. Northrop Grumman Newport News also used CATIA to design the *Gerald R. Ford* class of supercarriers for the US Navy.

Other

Architect Frank Gehry has used the software, through the C-Cubed Virtual Architecture Company, now Virtual Build Team, to design his award-winning curvilinear buildings. His technology arm, Gehry Technologies, has been developing software based on CATIA V5 named Digital Project. Digital Project has been used to design buildings and has successfully completed a handful of projects.

2. MASTERCAM

2.1. History

Founded in Massachusetts in 1983 CNC Software, Inc. is one of the oldest developers of PC-based computer-aided design / computer-aided manufacturing (CAD/CAM) software. They are one of the first to introduce CAD/CAM software designed for both machinists and engineers.

2.2. Introduction

Adhithan et al. (2013) has presented Mastercam, CNC Software's main product, started as a 2D CAM system with CAD tools that let machinists design virtual parts on a computer screen and also guided computer numerical controlled (CNC) machine tools in the manufacture of parts. Since then, Mastercam has grown into the most widely used CAD/CAM package in the world. CNC Software, Inc. is now located in Tolland, Connecticut. Mastercam's comprehensive set of predefined toolpaths—including contour, drill, pocketing, face, peel mill, engraving, surface high speed, advanced multiaxis, and many more—enable machinists to cut parts efficiently and accurately. Mastercam users can create and cut parts using one of many supplied machine and control definitions, or they can use Mastercam's advanced tools to create their own customized definitions. Mastercam also offers a level of flexibility that allows the integration of 3rd party applications, called C-hooks, to address unique machine or process specific scenarios. Mastercam's name is a double entendre: it implies mastery of CAM (computer-aided manufacturing), which involves today's latest machine tool control technology; and it simultaneously pays homage to yesterday's machine tool control technology by echoing the older term *master cam*, which referred to the main cam or model that a tracer followed in order to control the movements of a mechanically automated machine tool.

2.3. Mastercam Product Levels

Mikell P. Groover et al. (2007) with the release of Mastercam X (10), the application became a true Windows-based application, as opposed to one ported over from DOS. It also represented a fundamental shift in the way the application was configured. Mastercam X2 provided many enhancements over the previous version and adopted a true Windows application feel. Mastercam supports many types of machines, each with a choice of levels of functionality, as well as offers optional add-ins for solid modeling, 4-axis machining, and 5-axis machining. The following list describes the Mastercam product levels.

1. *Design*—3D wireframe geometry creation, dimensioning, importing and exporting of non-Mastercam CAD files (such as AutoCAD, SolidWorks, Solid Edge, Inventor, Parasolid, etc.).
2. *Mill Entry*—Includes Design, plus various toolpaths (top construction and tool planes only), posting, backplot, verify.
3. *Mill, Level 1*—Includes Mill Entry, plus surface creation, many additional toolpaths (for all construction and tool planes), highfeed machining, toolpath editor, toolpath transforms, stock definition.
4. *Mill, Level 2*—Includes Mill, Level 1, plus additional toolpaths, toolpath projection, surface rough and finish machining, surface pocketing, containment boundaries, check surfaces.
5. *Mill, Level 3*—Includes Mill, Level 2, plus 5-axis wireframe toolpaths, more powerful surface rough and finish machining, multiaxis toolpaths.
6. *5-Axis add-on*—5-Axis roughing, finishing, flowline multisurface, contour, depth cuts, drilling, advanced gouge checking.
7. *Lathe Entry*—3D wireframe geometry creation, dimensioning, importing and exporting of non-Mastercam CAD files (such as AutoCAD, SolidWorks, Solid Edge, Inventor, Parasolid, etc.), various toolpaths, backplot, posting.
8. *Lathe, Level 1*—Includes Lathe Entry, plus surface creation, C-axis toolpaths, stock definition, stock view utility.
9. *Router Entry*—3D wireframe geometry creation, dimensioning, importing and exporting of non-Mastercam CAD files (such as AutoCAD, SolidWorks, Solid Edge, Inventor, Parasolid, etc.), various toolpaths (top construction and tool planes only), toolpath transformation in top plane, backplot, verify, posting. CNC Software/Mastercam
10. *Router*—Includes Router Entry, plus surface creation, rectangular geometry nesting, additional toolpaths (for all construction and tool planes), highfeed machining, toolpath editor, full toolpath transformations, stock definition.
11. *Router Plus*—Includes Router, plus additional toolpaths, toolpath projection, surface rough and finish machining, surface pocketing, containment boundaries, check surfaces.
12. *Router Pro*—Includes Router Plus, plus True Shape geometry nesting, 5-axis toolpath functionality, multiple surface rough and finish machining, multiaxis toolpaths, toolpath nesting.
13. *Wire*—2D and 3D geometry creation, dimensioning, various 2-axis and 4-axis wirepaths, customizable power libraries, tabs.
14. *Art*—Quick 3D design, 2D outlines into 3D shapes, shape blending, conversion of 2D artwork into machinable geometry, plus exclusive fast toolpaths, rough and finish strategies, on-screen part cutting.

The Router products are targeted to the woodworking industries but are virtually identical to the Mill line.

3. CNC PROGRAMMING

Adhithan et al. (2013) Programming is a means of defining Tool Movements, through the application of coded Letter Symbols. As shown in the figure 2, all the phases of production are included in the programming, beginning with the technical drawing and end with the final product. CNC machines use a special programming language called GN-code (technical name: RS274). MasterCAM is software that allows users to create GN-code programs that can be used to cut different geometric shapes on CNC machines. The main functions are:

(i) Describe the geometry of the part to be machined.
(ii) Create a tool database – this DB carries information about the available milling tools.
(iii) Create the GN-code program to cut the part.
(iv) Simulate the machining of the part (for visual verification of the program).
(v) Upload the program to the CNC machine controller.
(Vi) Virtual Machining Process
Before commencement of the actual machining, the followings steps need to be considered

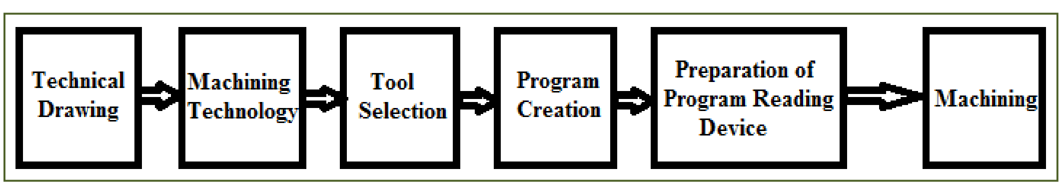

Figure 2
Phases of Production

Adarsh Adeppa,
3D Modeling and virtual manufacturing of a machine component using CATIA V5 and MASTERCAM,
Indian journal of engineering, 2014, 9(20), 06-13,

and performed carefully.

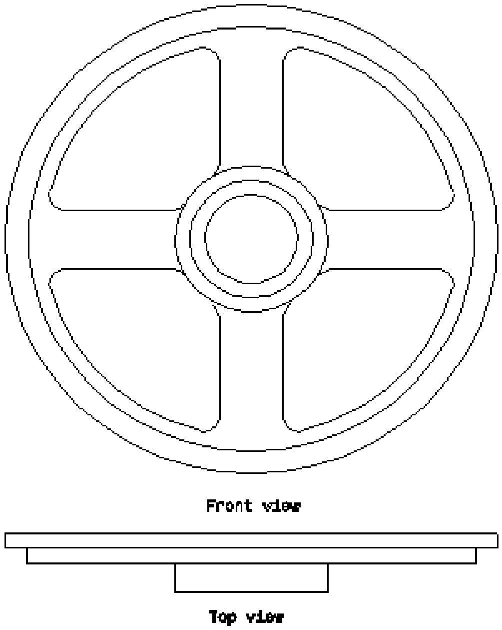

Figure 3

Front and Top views of the current Component

3.1. Drawing Study

Astheimer, P. and Gobel M et al. (1995) Drawing study is the very first basic and important step of manufacturing. It is the process of understanding the components geometry, identifying the machining processes and knowing various other parameters which are all together required to convert that component from raw material to final finished product. Also special processes like coatings, polishing finishes etc. are all identified. Drawings are normally represented in sheets of **'A'**-series. Based on the size & geometry of the part, no of details, other specifications, etc., the **'A'** sheet is used to represent it. Sometimes, based on the complexity of the part geometry, more than one sheet will also be used to provide all the details. Any mistake or error if arises while reading or understanding the drawing, it ruins the entire manufacturing process and finally the part gets rejected. So, one should have proper technical knowledge and sufficient experience for reading and understanding a Production Drawing. These drawings are generally read by Production In-charges or Shop floor Supervisors (Figure 3).

3.2. Development of Geometry CAD Model

You can create the geometry in one of two ways:

(a) By using the graphical design interface provided by MasterCAM

(b) By making the design in a CAD software, e.g. CATIA, Pro/Engineer, SolidWorks, and saving it in a format that MasterCAM can import (safest format to use: IGES; other possible formats include STL, STEP)

Method (a) is useful if you want to cut a simple shape; although MasterCAM provides some functions to generate 3D curved surfaces, defining the geometry in this environment is not convenient. So: if you want to make a part where the entire shape is defined using volumes made by sweeping 2D profiles normal to the plane of the geometry, use method (a). If you have access to any CAD system, use method (b). Method (b) generates the part by using a CAD system, and saved in any one of the Data Exchange Formats. Design the part (in, e.g. CATIA or SolidWorks). It is best if you already create the part such that the surfaces to machine are in the correct orientation and position for the machine. That is, the machining surfaces should be accessible from –Z direction, and the part should be located such that the covering, rectangular stock should be in the positive octant of the coordinate frame, with the corner at the origin. In the current project, the model (Figure 4) is developed using Catia V5R17 Narendra KomarlaM-Cme (1999).

3.3. Import the part into MasterCAM

MAIN MENU ? File ? Converters ? IGES ? Read File ? in the ***file browser***, select file ? OK (Figure 5).

3.4. Identifying and Planning Process Sequence

After developing the cad model, all the processes involved in manufacturing that component need to be identified and the sequence of those processes must be planned. For example, a particular component involves some processes, such as, Rough milling, Facing, Finish milling, Rough Turning, Grinding, Drilling and Tapping, etc. Planning the sequence of all these processes depends on many factors such as;

1. Complexity of the operation
2. Material being used
3. Geometry of the part
4. Degree of closeness of tolerances
5. Sometimes availability of the machinery
6. Any Heat Treatment Process specified, etc.

Based on its geometry and features, the current component involve, four milling operations (viz.., one contour milling and three pocket milling operations). All these operations are carried out in two steps (i) Roughing and (ii) Finishing.

3.5. Planning of Tools (Tooling)

For the completion of a machining process in-time and with desired surface finish conditions, proper planning of the required tools must be adopted (Adhithan et al. 2013). After identifying the processes involved, we need to select which tools are suitable for each individual process. This planning of tools not only helps to complete the operation in-time, but also brings down the manufacturing cost to a greater extent. If the jobs to produced are of large quantity (e.g. nuts, bolts, automobile components, etc), then it is advisable to use indexable milling inserts, because inserts can

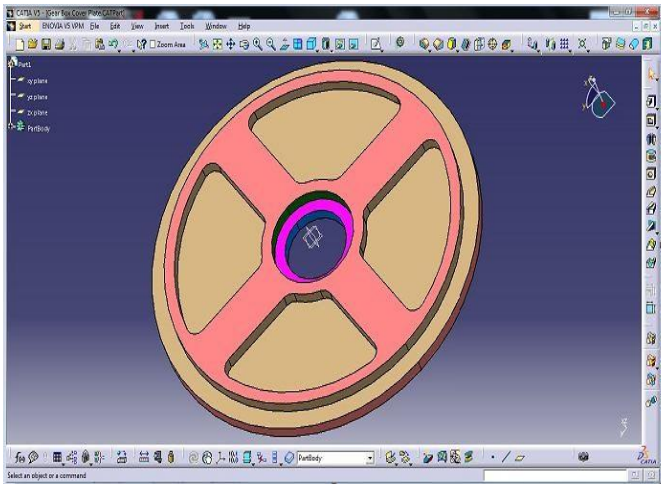

Figure 4

3D CAD Model of the component developed in CatiaV5R17

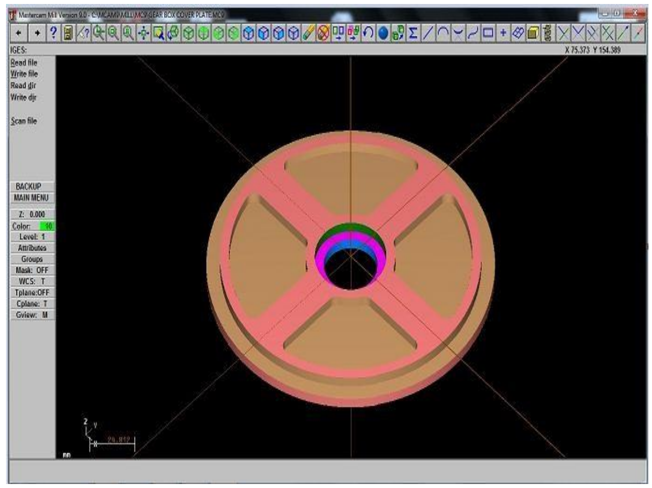

Figure 5

Imported IGES Model on which the tool paths are generated

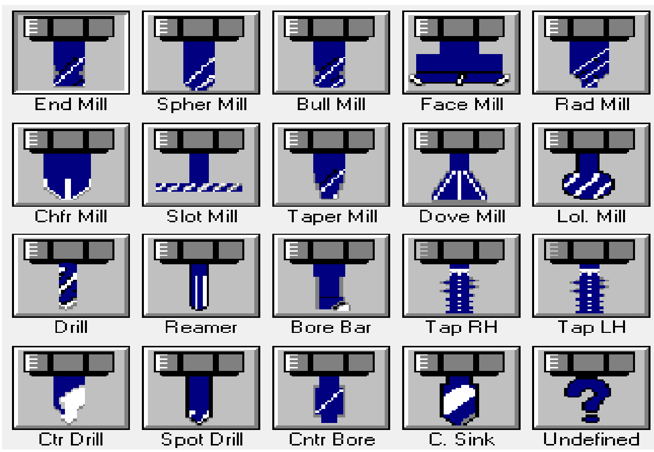

Figure 6

Schematics of Various Types of Cutting Tools

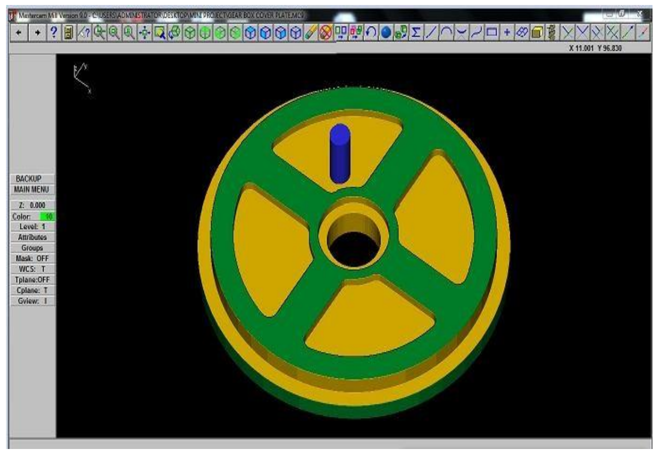

Figure 7

Graphical View of the Virtually Machined Component

machine more number of parts when compared to solid cutters. This automatically, reduces the tooling expenses. The tool database present in the Mastercam software describes the geometry of tools available for use in the workshop. The path that a tool travels in order to cut a shape depends on the size of the tool; some other information about the tool is also important – for example, the length of the cutting teeth on a drill or an end mill constraints the depth of the hole these can cut. The main functions for the tool database are:

(i) Look for an existing tool that you may want to use
MAIN MENU ? NC Utils ? Def. Tools ? Library
The Tools Manager window comes up, listing all available tools. Suppose we want to look for a flat end-mill of small diameter (e.g. less than 6mm). We can use the filter as follows:
Click Filter. In the Tools list Filter window, click Endmill1 Flat (top left icon) [if you leave the mouse cursor above the icon, its name appears] ? Tool diameter: [less than] value: 6 ? Unit masking: metric ? OK.

The tools Manager Window will show all tools that match these criteria. The figure below shows that my search gave five tools matching my criteria; If I want to use a 5mm end mill, double click on this tool, and its details will be shown in a ***Define Tool window***.

If the search results in **no existing tool**, and you would like to enter the data of a tool you have into the library: RIGHT-CLICK within the ***Tools Manager window*** ? Create new tool. This also pops up the ***Define Tool*** window; type the data

Adarsh Adeppa,
3D Modeling and virtual manufacturing of a machine component using CATIA V5 and MASTERCAM,
Indian journal of engineering, 2014, 9(20), 06-13,

for the new tool, and click Save to library (Figure 6). In the current case, for all the four operations, Flat-End Mill type cutters of different radii are used.

3.6. Generation of Machining Tool path

There are many different options and settings in generation of the tool path. Respective option must be chosen based on the requirement. Using the Tool parameters tab, you can set the Spindle speed, Feed rate and Plunge rate; if you know the values, enter them; otherwise just use the default – you can modify these when you actually machine the part, on the CNC machine. Hit OK, and if all goes well, the software will generate the tool path for you, and display it. Notice that the tool path where it is cutting is shown in different color than when it is moving up or down, and moving between pockets. Also notice how the tool path for removing the region inside the pocket is zigzag, but the tool path also goes once around the contour of each pocket.

3.7. Verification of the Generated Tool Paths

Once after proper tool paths with desired operations are developed, these are verified by simulating them one after the other, so that the final product after total machining can be viewed graphically (Figure 7).
In Mastercam, we can see a 3D animation of the cutting by the following method:
MAIN MENU ? Tool paths ? Operations ? The operations Manager window pops up; ? Verify, and then click the Machine button on ***Verify: simulation window***. You can speed up the simulation by moving the slider to the right:

3.8. Job Stock Set-up

This step defines the block from which you will cut the final part. It is essential to create the cutting plan. This enables proper viewing and understanding of the Virtual Machining process. This step has to be performed prior to the tool path verification process. MAIN MENU ? Tool paths ? Job Setup ? in the ***Job Setup window***, Click Bounding Box [Window will vanish], MasterCAM will prompt you to select the entities: All ? Entities ? Done ***Job setup window*** re-appears, and the size of the bounding box is displayed in the correct boxes ? OK. You will see the bounding box around the part (different colored lines) 4.

3.9. Completion of the Given Task

Following all the above operations properly with proper attention and care leads to completion of the task as per the specifications. After completion of all these steps, the NC code part programme must be developed for real time machining. The tool path generated by MasterCAM has all information required to control a CNC machine tool to cut the part. However, different CNC machines use slightly different versions of GN-code. The conversion of the machining data to the GN-code specific for a particular CNC machine is called ***Post-Processing***. The exact format of the GN-code is stored in different post-processing files, and the system will use whichever post-processing format you select. Later, the generated NC-Part file is subjected to proper Header- Footer changes before being transferred into the CNC machine controller. You can set the communications settings for your CNC machine controller, and if your PC is connected to the CNC machine controller, the program will be uploaded to the CNC machine. Else, the program can also be transferred using various external Data Transfer Devices.

4. CONCLUSIONS

As the concept of virtual machining is not just viewing of the machining process graphically, it has to be carried out with at most care. So, while performing virtual machining, the following precautions must be taken into consideration:

1. No mistake or error should takes place while creating the model. All the dimensions and sizes must be maintained as given in the drawing.
2. The model has to be converted into proper Data Exchange formats in which very less or no geometry losses occur.
3. After importing the model into the CAM software, it should be transformed into proper orientation as per the machining convenience.
4. While generating the tool paths for various operations, precautions should be followed keeping in mind, the real time machining hurdles, like spindle speed , feed, depth of cut, etc.
5. It is always advisable that tool path for each operation is verified until you get visually satisfied.

After generating the NC Code part file, it is mandatory that necessary Header- Footer changes are made before transforming it into the CNC machine controller.

REFERENCES

1. Astheimer P, Gobel M. Virtual Design II: an advanced VR development environment", in Proceedings of Eurographics Workshops, Monte Carlo, Monaco, 1995
2. Choi B, Barash MM. STOPP: an approach to CAD/CAM integration, *Computer-Aided Design,* 1985, 17(4), 162-168
3. *Automation Production Systems and Computer Integrated Manufacturing,* by Mikell P. Groover.24 july 2007
4. Narendra KomarlaM-Cme. Grid generation and analysis using CATIA V5, 1999
5. Mitchell M Tseng, Jianxin Jiao, Chuan-Jun Su. Virtual Prototyping for Customized Product Development. Integrated Manufacturing System, 1998, 9(6)
6. Operational Manual of "*Vertical Milling Machine. Model: BMV 50, Controller: Fanuc Series-OiMB*"
7. "*CNC MACHINES*" by B.S. Pabla, M.A. Adhithan. 14 july 2013
8. "*MECHATRONICS*" by HMT