SYSTEM ANALYSIS AND EVALUATION OF AUTOMATED DESIGN & MANUFACTURING SOFTWARES

Mohsen Sadegh Amalnik
Department of Industrial Engineering, Faculty of Engineering
University of Tehran, Email: amalnick@ut.ac.ir

ABSTRACT

Automated Sofwares in design and manufacturing such as Computer Aided Design [CAD] / Computer Aided Manufacturing [CAM] are the important modules for advanced and complex manufacturing environment. This state of the art technology has revolutionized the manufacturing processes. The paper discusses the emerging advances in CAD/CAM Software that are presently available in the world market capable of NC code generation for multi-axes CNC machines , EDMs, Welding systems, and Laser systems. The switching over from DOS based systems towards windows based systems are also rapidly enhancing. Different control systems like Heidenhien, Siemens, Fanuc, etc. are also incorporating such CAD/CAM software as an optional choice for the customers. The paper discusses the software features and analyses the CAD/CAM software of ESPRIT, DELCAM, CAMTEK's CAD/CAM System for EDM [PEPS], LICOM's ALPHACAM, SURFCAM, MASTERCAM, etc. The paper provides an analysis and evaluation base for the superscript software listing that may be very helpful for the audience.

KEYWORDS

Automated System, Computer Aided Design [CAD], Computer Aided Manufacturing [CAM], Computer Numerical Control [CNC] Machine, Tool Path, NC Code.

1. INTRODUCTION

The Computer Aided Design (CAD) and Computer Aided Manufacturing (CAM) programs are meant to design a part and create the tool path code. Computer Numerical Control (CNC) machine uses the code to cut the part. The subscript describes the steps to generate the CNC code:
  - i. Create the part design
  - ii. Create the tool paths
  - iii. Manage the tool paths
    - a. Verify the tool paths
    - b. Edit the NC tool paths
  - iv. Create the CNC code
  - v. Communicate with the NC machine

2. MODERN AUTOMATED SOFTWARES IN DESIGN & MANUFACTURING

There is a lot of CAD/CAM software currently available in the world market. However, the following list highlights some of the top class modern CAD/CAM software like MASTERCAM, SURFCAM, DELCAM, ALPHACAM, PRO-E , ESPRIT, PEPS, etc.

3. ANALYSIS AND EVALUATION

The subscript topics describe the pros and cons of the enlisted Automated software. We discuss the prominent features of the software, the ease of use, the available options, the efficiency, and other characteristics in order to analyze and evaluate these software.

3.1 PART DESIGNING AND EDITING

Unigraphics supports the drawing editing in the CAM software. It supports the FEA program. Unigraphics collaborate enables multiple users to work on a design in a real time collaborative mode. It supports the batch processing. It supports Meshing.

Pro-E has the versatile capability of dynamic updating any editing in the part design. This feature brings this software a high ranking among other software. A very powerful tool called Pro-Detail is available in pro/e, which give you different views of a 3-d model in seconds. With one click you will get complete dimensions. You can edit the dimensions, you can edit the views.

Master CAM's Design and other general features exhibit improved handling of inch and metric parts. A new chaining option allowing users to select a surface edge when in Single selection mode is now available in Master CAM. Since the release of Mastercam's new Nesting product there has been a need to allow more control over the sorting of chains during selection. This is critical when creating tool paths for a Router because of how the material is held on the table. The machine needs to cut the inside nested chains first before the outer chains. Here users can select Inside to Out, Outside to inside, including optimization. There is also a new switch to reverse the direction of inner chains so compensation of Contour tool paths will happen on the correct side. Mastercam relies on endpoints, and in some cases the midpoints, of entities to create tool paths. In Version 9, users have the ability to dynamically change the start point of a chain to anywhere along the chains boundary. This can be done during the initial chain selection by selecting the Change Start menu option or by editing existing tool paths with the Start Point option found in the Chain Manager accessed through the geometry icon in the Operations Manager. This capability is available in both Mill and Lathe. The ability to create Spirals and Helixes has been added to the long list of Create options available in Mastercam. This function incorporates the abilities of the Spiral and Helix c-hook utilities into a single, easy to use, dialog box.
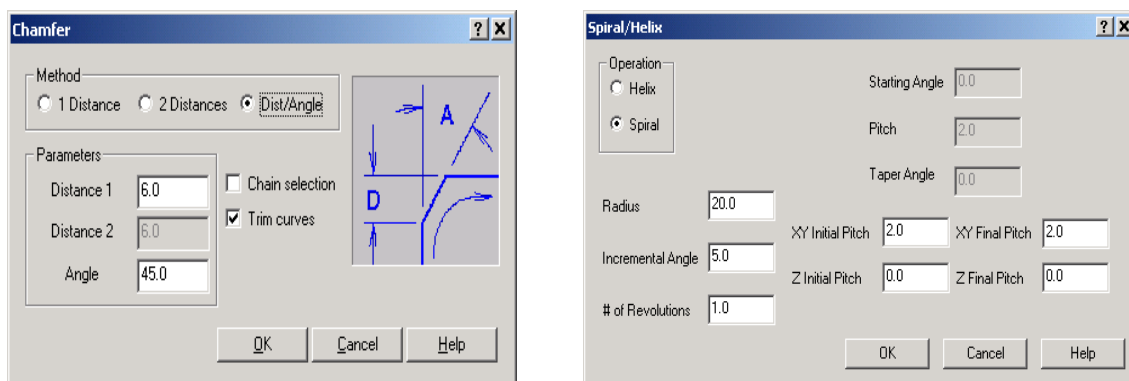

Exhibit 1: Chamfer and Spiral / Helix Options in CAD / CAM Software

3.1.1   TRANSLATORS

The ability to translate part data from other Automated Systems into Mastercam is possible. DWG, DXF, and Inventor supports are available in the software. The engine used for translation of

AutoCAD files has been changed. In making this change, the reliability of translations has been increased, including support for version earlier than R-13. At the same time it supports Inventor version 5.0 files. The ability to modify and save STL files has been added with a new Xform capability. Users can Mirror, Rotate, Scale, Change Normals and other things to manipulate STL models for use in Tool path creation or for stock in Tool path verification. Users are automatically prompted when an SLDPRT [Solid works part] file contains more than one configuration. This allows users to load specific sections of an SLDPRT file. CNC Software has entered into an agreement with Pro-E to use their newest translator engine called Granite. This gives us the ability to directly read in Solids from Pro-E and turn them in to Solids that Mastercam can use. Solid healing during translation is a configurable option that allows Mastercam to automatically attempt to heal imperfect Solids during the translation process. This switch can be set in Screen, Configure, Files, in the Converters dialog box.

3.2 TOOL PATH GENERATION

Multiaxis toolpaths in Master CAM's Version 9 have undergone extensive changes giving users access to new powerful functions. Geometry Interface offers a new method for defining Cut Patterns, Tool Axis Control, Drive Surfaces, and Check Surfaces has been implemented for most Multiaxis toolpaths. This dialog box will appear first when creating Multiaxis toolpaths or when the geometry icon in the Operations Manager is selected. See the Multiaxis sample parts for specific examples. The cut pattern works as a guide to determine how the toolpath will be generated. Users can pick an existing Surface or define parametric Cylinders, Spheres, and Boxes that guide initial tool motion. Tool Axis Control option allows users complete control on how the tool axis motion will be controlled during toolpath generation. Using a surface for both the cut pattern and the tool axis control will result in a Flowline like toolpath with the surface normals controlling the too axis. Additional options give the ability to force the tool axis to behave specific to the users needs. Including options to focus the toolpath to or from a specific position, and the use of a closed boundary to contain the tool axis. Cut Surfaces option enables users to choose drive surfaces that are to be cut. Surfaces other than the Pattern surface can be included. Check Surfaces option allows users to define surfaces as check surfaces similar to 3 axis milling. Multisurface 5 axis toolpath gives users the ability to Rough and Finish with the same interface. The interface gives users control over Lead and Lag angles. Toolpath stepover along and across the part. Options for zigzag, one-way, or spiral toolpath motion, roughing parameters including number of depth cuts and values for depth of cut. Point Generators, which are used to add additional tool moves in areas of the part that change rapidly. Other options include the ability for users to set Axis limits so toolpaths can not be created with angles greater than the Milling machine can support. Multiaxis toolpaths in Version 9 can also compensate for Conical and Tapered cutters, giving greater flexibility in creating parts with undercuts.
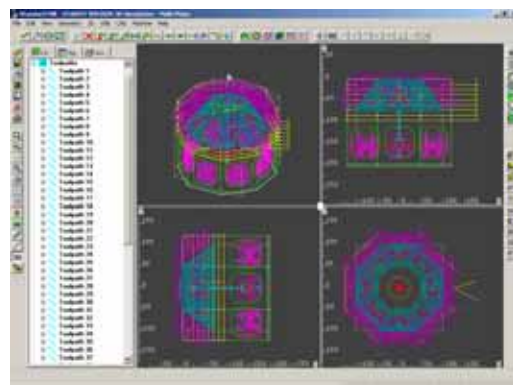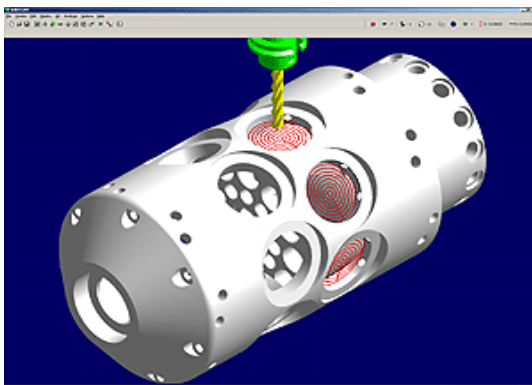

Exhibit 2: Tool Path Generation in SurfCAM and Licom's AlphaCAM

Pro E offers tool path generation with the options of Pocket Milling, Raster Milling, Uni and Bi direction, Spiral Milling, Shallow region milling, Radial Milling, ZIG-ZAG, etc. Pro/e is very powerful in machining and you can find, pocketing, profile cutting, volume cutting, surface cutting, engraving, drilling, deep hole drilling, threading, reaming and so many more options.

Unigraphics has the options of Mixed cut pattern, Manual and Auto Operation, Sequences outside in alternate, Inside out Alternate, ZIG, ZIG-ZAG, ZIG-ZAG with lifts, etc.

Surfcam offers a variety of methods for tool path generation as others do. It gives users access to multi axes tool path generation.

3.3 OPTIMIZATION

Feed / Speed

Feed rate dialog available in Unigraphics sets the spindle mode and speed settings. A new dialog can be used to correct the feed speed. Pro-E is value dependent. Parameters can be set, feed cutting speed could be changed or improved. MasterCAM, SurfCAM, and DelCAM offers the dialog boxes for feed and speed calculations according to the respective process in order to optimize the process.

Calculation / Optimization

Unigraphics calculates and optimizes the design size. If the design size changes, the other dimensions will be changed automatically. Pro-E calculates the design value and optimizes the size. The optimization calculations are dependent upon the feed, speed, material, depth of cut, and other parameters in MasterCAM, SurfCAM, and DelCAM.
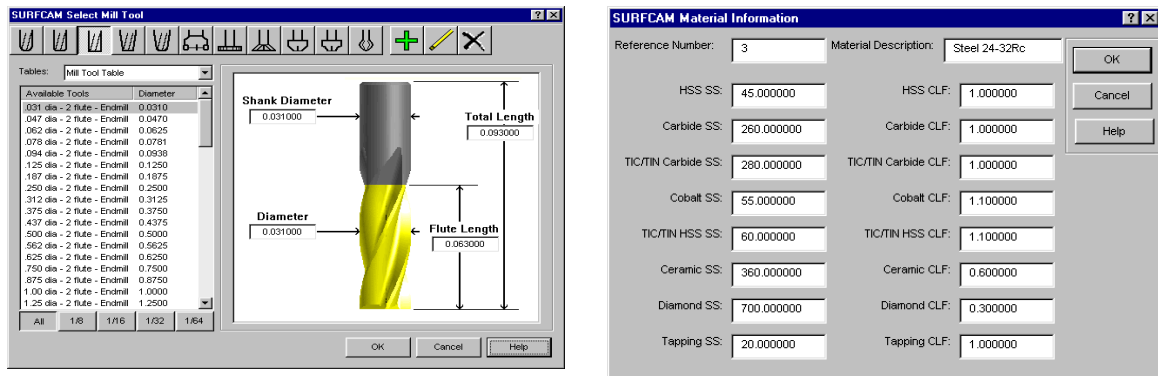

Exhibit 3: Optimization in Automated Softwares based upon Tooling Data

3.4 CNC CODE GENERATION

SurfCAM offers post processors for the controllers like Agiecut 150F AG-150F, Allen Bradley 7360, Amadan 04P-C AM-04PC, Ampex AP, Anilam 1100 AN-1100, Bandit 1 BA-1, Bosch CC220 BO-CC220, Bridgeport Boss 2 BP-BOSS2, Brother Tapping Center BR-TC, Burney 2.5RS BU-25RS, Charmilles Robofil 300 CH-R300, Cincinnati Acramatic 850 CN-AC850, CNC Inovation RTE-1000 CI-RTE1000, Siemens GN-8M SI-8M, Sinumeric System 3 SS-3, Sodick Mark 20 SO-MARK20, Southwestern Prototrak MX2 SP-MX2, Strippit Hecc 80/3 ST-HECC803, Traub TX-8 TR-TX8, Tree TREE, Umac 6B20 Cascade UM-6B20, Yasnac LX-1 YN-LX1, etc.
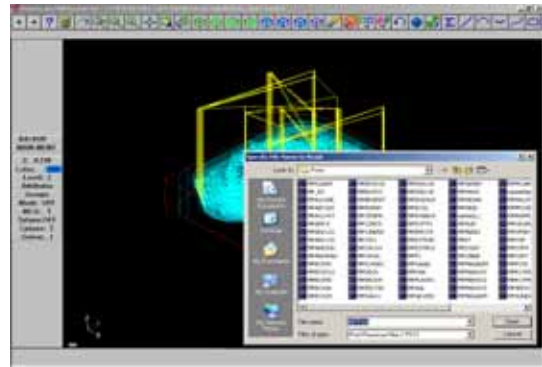
Exhibit 4: CNC code generation in Automated Softwares using desired post processors

Pro-E, Mastercam, and Delcam creates any program for machining. If the machine has the similarity, then it uses the default or matches the settings. By default there are postprocessors for controllers like, Fanuc, HAAS, FADAL, DYNA, CINCINNATI, FANUC, HEIDENHAIN, etc. but one can prepare customized postprocessor according to controller of the machine. UniGraphics supports post processors including EDM Processors as well. But no core wire is used.

Simulations

Pro-E automatically gives the machining time. There is an addin in Pro-E called VERICUT, which is a very powerful tool for 5-axes simulation and optimization of tool path. Simulation is free and when you create a tool path you can simulate it using VERICUT. For tool path optimization, you have to purchase license from CG-Tech, the manufacturers of VERICUT.
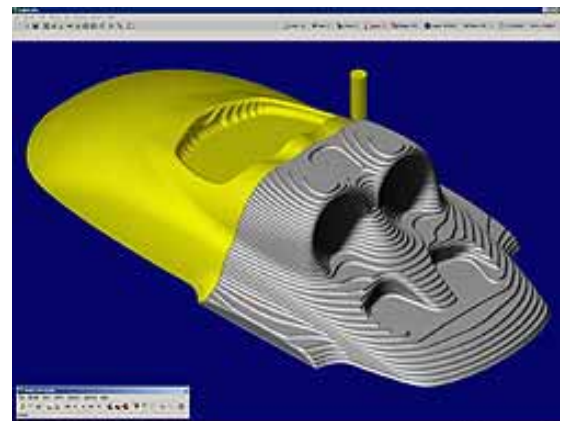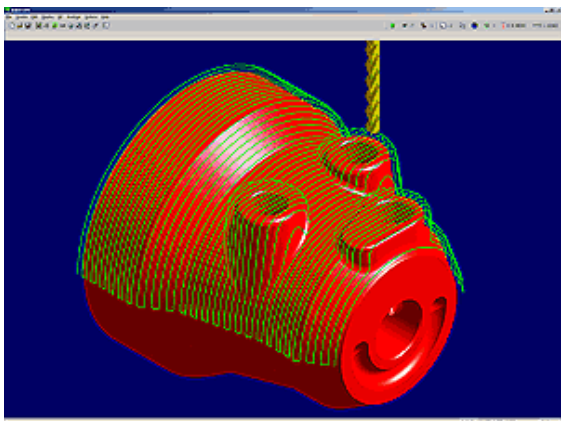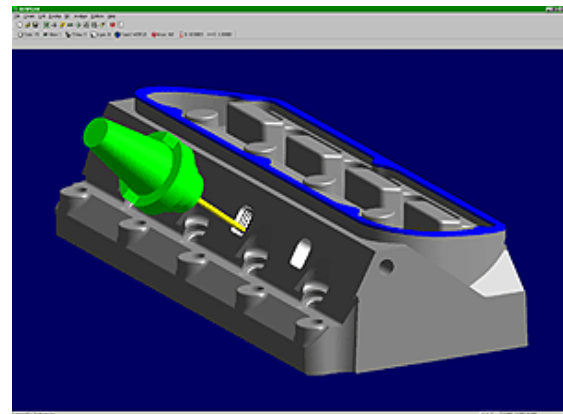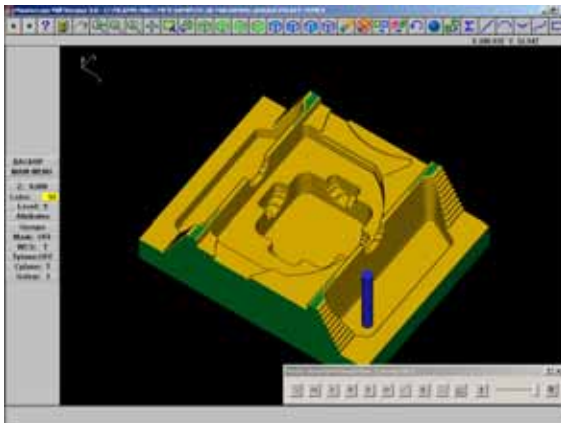

Exhibit 5: Advance simulations in Automated Softwares

Master CAM provides Simulation options up to 5-axes. It provides the user facilities to simulate in a required direction. It can also show the tool along with the tool head. A back plotting facility is also provided. A simulating video is available to user with the functions to make the movements fast or slow as the user demands. Unigraphics simulates the CAM activities up to 3 axes.

3.5 COMMUNICATION WITH CNC MACHINES

Most of the Automated Softwares in manufacturing provide the user to convert the prepared CAM program to the CNC machine through RS 232 or RS 422 according to the data transfer parameters standardized with the machine.
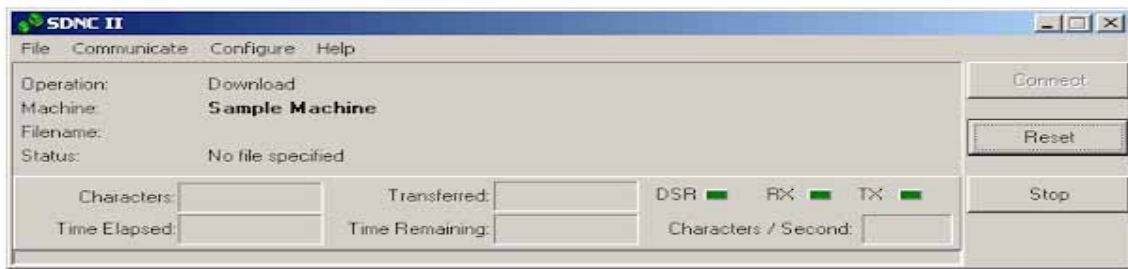

Exhibit 6: Surf CAM DNC Software for data transfer from CAM station to CNC machine

4. CONCLUSIONS

The modern Automated Softwares in design and manufacturing like MasterCAM, SurfCAM, Pro-E, AlphaCAM, UniGraphics, and DelCAM are being analyzed and evaluated in this research. All of these software are among the top class of the Automated Softwares presently available in the market. All have the unique features like design, editing, standard design transformers / converters like IGES, DXF, etc., tool path generations, optimization of parameters like feed, speed, etc., simulations, back plotting, NC code generation according to the required post processor, customized post processors, data transfer from CAM station to the CNC Machine, etc. All these features are available in these software with some differences like Pro-E is very strong in design and editing facilities while MasterCAM and SurfCAM are very strong in their available post processors and the facility to customize post processors. DelCAM is stronger in its front end that is claimed to be better than Pro-E and is more likely to be used for e-solution by DelCAM experts.

ACKNOWLEDGEMENTS

We are grateful to the developers of the Automated Softwares discussed in this research for providing us evaluation versions of the software.

REFERENCES

[01]   Manuals and Help, "Master CAM Version 9", CNC Software Inc. USA, 2002.
[02]   User's Guide, "Unigraphics V 18.0", Unigraphics Solution Inc. USA, Pp. 153-192, 2001.
[03]   Manuals and Help, "Pro Engineer Wilde Fire", Parametric Technology Corp. USA, 2003.
[04]   Manuals and Help, "Surf CAM", Surfware Inc. USA, 2003.
[05]   Manuals and Help, "Alpha CAM", Licom Systems Inc. USA, 2002.
[06]   Manuals and Help, "Del CAM", Del CAM Inc. UK, 2003.