

A Guide to investing in CAM software

► **Why invest in CAM?**

Without suitable CAM software it can be extremely difficult to optimize the productivity of CNC machinery as, depending on the parts being made, programming to produce reliable CNC code can be very time consuming and prone to error. Reasons for investing in suitable CAM software can be varied and include the following.

Faster and more reliable programming - Programming may be being done on the machine control, or with existing CAM software which may be inefficient. By purchasing new CAM software companies can greatly reduce the idle time on the machine tool. Time spent programming on the machine control with the machine stationary can be eliminated. Prove out time, program editing and program errors can also be a thing of the past with suitable new software. Furthermore, for large complex parts, calculation times can be significantly reduce and overall programming times can be shorter too.

Easier programming - Some CAM software can be complex to operate, making it necessary for the company to rely on the skills of one person. With easy to use CAM software, the skill required for programming is greatly reduced, enabling more people to become proficient in the software and, where appropriate, for programming to take place on the shop floor. The advantage here is that the machine operator is in a better position to understand the capabilities of his machine, and can program it while it is cutting, and whilst using the most efficient tooling, methods and cutting conditions.

Compatibility with CAD systems - Increasingly companies are receiving CAD models and designs from a wide range of CAD systems. To provide a service to these customers, manufacturers have to be able to import the model data from multiple systems, interrogate it and, where necessary, repair or modify it ready for machining, making this a crucial capability for retaining existing customers and gaining new ones.

More efficient machining processes - For manufacturers, the efficiency of toolpaths and their reliability can have a major impact on productivity. Having absolute confidence in the quality of the CNC code will enable companies to run machine tools unmanned for long periods of time, while maximizing tool life. The ability to carry out high speed hard metal machining and produce 5-axis toolpaths will enable more of a part to be cut in one setting, thereby reducing the use of EDM. Fewer electrodes, less setting, and highly efficient cutterpaths which take the shortest route, cut down on retract movements and keep feedrates as high as possible, will significantly reduce manufacturing costs.

► **Evaluating your needs**

As well as deciding on the objectives and the desired productivity improvements, factors that need to be examined include:

The type and range of work that the company produces. Is it 2D or 3D, does it require 5-axis machining, are there a lot of holes, are the materials it works with difficult to machine, are there particularly high quality standards required?

What CNC machinery needs to be programmed, multi-axis capability, high speed machining, positioning or continuous 5-axis, EDM?

What CAD data is normally received and in what format? Are there plans to find new customers that normally use different CAD systems, is a CAD system in use currently?

Skills available - is someone already skilled in CNC programming? Is shop floor programming anticipated?

► **Evaluating the supplier**

The level of support offered by the supplier is particularly important for CAM software, as delays in programming due to technical difficulties translate directly into production delays, missed deadlines and damage to the machine and the component being cut. Conversely, shorter programming times, faster CAD manipulation, improved quality, quicker machining times and longer tool life are all improvements that can be expected from the right CAM software and will significantly raise profitability and increase customer satisfaction levels.

It is a false economy to take shortcuts on training and support packages, as thorough knowledge of the system will not only make programming faster and more efficient but will greatly reduce the chance of error. Systems such as WorkNC include failsafe cutterpaths so that the system will not generate a colliding toolpath, and have defaults built in to make it difficult to make a mistake. Talking to other users of the software and visiting some with a similar application is definitely worthwhile, enabling managers to form an opinion on the quality of support, and the regularity and pace of software updates. Frequently, support is offered through remote access to the client's computer, enabling the vendor's technician to solve a problem directly over the Internet.

Visiting existing users also enables companies to see how reliable the software is, and how well tested new releases are. Postprocessors need to be 100% reliable so that no editing of the CNC code is required

before running the program. Where G code editing is necessary to make the program work, it will at some point result in an error on the machine, or worse still an expensive collision.

The CAM supplier itself needs to have sufficient resources to continue to develop its software, must have adequate local support, and have a history of producing significant technical enhancements for its customers. In this way, manufacturers can be sure of maintaining a competitive advantage and having software that will always be at the leading edge of machining technology.

► **Benchmarking**

Checking for CAD compatibility

It is a relatively simple task to get the CAM supplier to test some real CAD models for you. Direct translators for CATIA, Parasolid and IGES, STEP and DXF are available with most CAM software. However, once the model or design has been imported it may need repair or modification before it can be machined. It is also extremely useful to be able to interrogate the design before programming starts. WorkNC allows users to check dimensions and geometric items such as draft angles, and minimum radii. Knowledge of this information is important for determining items like tool diameters and moldability. The system also makes it possible to patch surfaces and holes, and repair surface models so that they are completely closed. The hybrid solid and surface modeling in WorkNC simplifies this task, and makes it easy to extract and re-orientate the part, add split lines, run-off surfaces or extract electrode shapes to prepare the model for machining. For some applications, it is not always necessary to completely close a surface model, so some tolerance of holes in the model during machining is a useful feature, which can save a considerable amount of time. WorkNC can happily cope with these inconsistencies, ignoring small gaps between surfaces during machining.

Hiroki Takama from the Japanese Riken Forge's Production Engineering Department said, *"WorkNC was the only system capable of opening our JAMA-IS format CAD files. It was also obvious that WorkNC was better for direct machining of dies, producing high quality NC data with customized templates – it is an improvement in every way."*

When evaluating CAM it is worthwhile looking at other aspects of the production process and how these will fit with your choice of CAM software. SESCOI has a view, mark-up and analysis CAD viewer, WorkXPlore 3D. This extends the analysis and

collaboration capabilities of WorkNC, enabling many more people from an organization and its supply chain to simultaneously work on the design before manufacture starts. CAD data from many different design systems can be imported and combined. Users can then add comments, and review the design to shorten delivery times, reduce costs and optimize the finished product, making this a useful addition to the manufacturing process. SESCOI's WorkPLAN Enterprise manufacturing ERP software and MyWorkPLAN job shop software control other aspects of production, helping companies to produce accurate quotations, keep track of engineering changes and production progress, control costs, delivery times and quality, and ensure key performance indicators are on target.

Darrel Gallier from CA Tooling, Lynchburg, Virginia says *"WorkNC and MyWorkPLAN systems are both vital parts of our operation and they make a significant contribution to cost reduction, productivity and business growth. By expanding our SESCOI software to meet the demands of our new facility we will ensure our success for the long term."*

Ease of use, speed of operation and capacity

The skills required to operate CAM software need to be minimized. There is a growing and serious shortage of skilled engineers in many countries. Software that is complex to operate is likely to fall into disuse, as only one or two people will be able to program it effectively, greatly reducing flexibility and adding to the cost of running the system. WorkNC has been designed with ease of use in mind. Its ergonomic interface guides the user through the process and its automatic machining routines have been designed to automatically apply safe and efficient toolpaths to the model.

Haruki Matsui of Japanese company TOMCO Co Ltd, says, *"WorkNC certainly met our expectations. It lives up to its claims that it is easy to use and implement, unlike our old software, which required a lot of adjustments to its cutting strategies to achieve a satisfactory result. We also found that WorkNC's toolpath editing facilities were first class, enabling us to get exactly the cutterpath we wanted very simply indeed."*

CAM software is frequently used by machine operators on the shop floor. Advantages reported for this way of working include flexibility as many more people are able to operate the system; making better

use of the machining knowledge and skills of the machine operators; multi tasking with operators programming while the machine is cutting; improved job satisfaction; and higher quality levels through pride in a job well done. Whether the system is used in a programming office or on the shop floor, ease of use is one of the most important attributes of a system. By making it difficult to make a mistake and automating the process, CNC programming becomes a routine exercise rather than an aspect of production which is time consuming and error prone. Within WorkNC, users can set up sequences of operations for similar components. By applying these to new geometry, all the tooling, methods and feeds are all automatically set. Programming can then be completed with one mouse click ensuring consistency of methods on the shop floor, assured quality, and consolidation of the range of tooling used.

Roy Thomas of Patterson Mold & Tool (USA) comments on WorkNC shop floor programming, *"The operators understand their tooling, machines and the metal cutting process significantly better than our offline programmers did. Productivity has gone up by 30% with better toolpaths and much less time wasted making changes. In particular, roughing operations take half the time to program and run 70% faster."*

For complex parts, machining times can be many hours, requiring copious amounts of CNC code. Generating this can be time consuming, so calculation speeds, batch processing and multi tasking will significantly cut waiting time. WorkNC's 64 bit algorithms support the latest parallel processors, cutting calculation times by up to 10 times. Additionally, the number of instances where recalculation needs to be carried out has been reduced. Toolpath parameters such as lead-in and retract moves can be altered without the need for complete toolpath recalculation.

Gerhard Ammon of Playmobil (Germany) says, *"The software is worth its weight in gold. Many processes are easier thanks to WorkNC. It paid for itself in less than a year - the calculation is easy - today two WorkNC workstations ensure that five machining centers are in use simultaneously. That's what I call productivity."*

How the software is licensed can make a big difference. The ability to start programming another job while an existing one is calculating greatly increases the capacity of the system and cuts out waiting time. Floating licenses increase flexibility enabling the software to be accessed from many

locations - particularly useful for shop floor programming, while batch processing allows long programs to be calculated overnight, making the maximum use of the software's processing power.

Quality of cutterpaths

The most elegant geometry handling and high quality interface will be of little value if the CNC code and toolpaths produced by CAM software are inefficient or need editing before they can be run on the machine. The most common problem is excessive air cutting and unnecessary retract and rapid movements. These will extend machining times, greatly increasing the cost of manufacture. SESCOI has spent many years optimizing its toolpaths to overcome these problems. Stock and rest models enable the system to know exactly where material is located and smooth machining algorithms ensure that unnecessary retract movements are eliminated, and that feedrates can be kept as high as possible by avoiding sudden changes in direction.

Other factors include fluid entry into the material, trochoidal movements and combined roughing and finishing operations, which will eliminate duplicated cutting operations and minimize the tool loading, greatly extending tool life.

Hiroki Takama of Riken Forge (Japan) says, *"WorkNC has made it much easier to create CNC data. The machining sequence, which is one of the features of WorkNC, saves us a substantial number of man-hours. What's more it creates the toolpath quickly, with a choice of re-machining paths, enabling us to improve the quality of our products. We are particularly satisfied with WorkNC's Re-machining toolpath."* Saori Sekine from the Production Engineering Department, adds, *"A Z-Level Finishing toolpath is required to finish some dies. However we find that WorkNC's 3D Drive Curve Finishing toolpath works much better for these parts. It eliminates unnecessary retracts and provides a smoother finish by machining on the surface with a constant stepover. A toolpath can be created without subsequent modification so we can leave WorkNC to calculate in batch mode overnight. When the toolpath is ready for machining the next morning, we do not even need to check the data. This is a huge advantage for us."*

Where 4 and 5-axis programming is required, the complexity of achieving a reliable result can increase dramatically, with potential tool and holder collisions becoming a far greater risk. WorkNC solves this with its Auto 5 and recently launched Auto 3+2 modules. These automate the process by allowing engineers to

program the part in 3 axis and then allow the software to automatically transform the program into 4 or 5-axis movements. For specialist applications the software also includes blade and impeller machining routines as well as conventional 5-axis machining operations, allowing for maximum flexibility.

Mr Ono of RYOBI (Japan) says, *“Adopting Auto 5 enabled us to reduce five surface finishing processes to three, while electrode manufacture and EDM die sinking has been much reduced as we can machine the bulk of the cavity using 3/5-Axis toolpaths. The result is a reduction from 12 processes to 8.”*

Applications which include many holes, such as stamping die manufacture, rely on the accurate interpretation of the model. Feature recognition automates the process by gathering hole information directly from the design and automatically applying the appropriate tools and toolpaths. Not only does this speed up a repetitive programming task, but also eliminates the possibility of an error.

Ed Busman of Walker Tool and Die (USA) says *“Thanks to WorkNC’s Feature Recognition module the time saved has been huge, with a 90% reduction in the hours required for CNC hole programming.”*

Benchmarking to test the quality of toolpaths and the cycle times will quickly show the advantages of one CAM software over another. Advantages to look for include smooth tool trajectories, few retract moves, no air cutting, constant tool loading and the quality of the finish on the part. Poor entry and exit strategies, rapid direction changes, incorrect feedrates, excessive tool loading, and not using climb milling will all contribute to a rough surface finish.

Simulation and collision avoidance

Simulation of the toolpath is an important final check, making sure that the cutterpath is as planned and that there are no collisions between the tool, its holder, the workpiece and the machine, and for comparing the finished part against the original model to ensure it has been completely machined. The checking in WorkNC goes a stage further, giving options for creating just the cutting operations, which can be reached with an existing tool and then recommending a tool length capable of finish machining the remainder of the part. This type of functionality helps the programmer to make a decision about the next operation. He can then decide on the feasibility of a longer cutter and, if

necessary, examine alternatives such as spark eroding, reorientating the part for further 3-axis machining, or moving to 5-axis machining.

If the decision is spark eroding, WorkNC is able to extract the electrode geometry from the model, locate it on a blank electrode, set a datum, and finish machine it. For 5-axis machining, programming can be continued in Auto 5, letting the software tip the machine head so that a shorter tool can be used to reach the rest of the part. Here, the collision avoidance goes further, introducing flip and unwind movements when the machine tool’s axis limits are reached.

Pierre Gauthier, Managing Director of Zedes (France) can cite some impressive results, *“We have halved the time required to create an electrode, and with WorkNC’s HSM strategies we save the same time again when machining”.*

Graphical toolpath editing can also be very useful in optimizing cutterpaths. Here, the user should be able to graphically select parts of the toolpath on the screen and delete or modify them. This makes it possible to have a much greater degree of control when special circumstances arise, allowing the programmer to customize the toolpath to his exact requirements.

► Conclusion

There are significant differences between CAM software systems, but the key factors are the ability to handle CAD data; the ease with which the system can be learnt and used; the quality and power of the toolpaths it generates; the simulation and collision avoidance capabilities; the reliability of the postprocessed CNC code; the pace of product development by the software vendor and the support it can provide. Carrying out a benchmark test will help to answer most of these questions, as will talking to existing users. However, companies need to be clear about why they want to purchase a CAM system and the benefits they are expecting to see. With products like Sescoi’s WorkNC, the advantages of installing the software are frequently revolutionary, completely changing the way of working and achieving productivity gains which were previously thought to be impossible, paying for the investment in the software many times over.

► Sources

Author - Sescoi International SAS - Blvd General de Gaulle, BP 75, 71009 Macon Cedex, France Tel.: +33 385 216621 Fax: +33 385 21662 info@sescoi.fr

Sescoi International SAS www.sescoi.com/us