

# Modern Machining Techniques for Mouldmaking

S. Docker, P.J. Dickin

Delcam plc, Small Heath Business Park, Birmingham B10 0HJ, UK

## Abstract

The increased globalisation of the mouldmaking industry has made it essential for all companies in the business to do everything they can to increase productivity. Newer machining techniques like high-speed machining and five-axis machining can give a crucial advantage in meeting these objectives. However, introducing these techniques involves much more than purchasing new machine tools. A modern CAM system is also essential, that can support the advanced strategies needed to maximise the productivity of these machines. These techniques, coupled with the ability to simulate toolpaths on the computer before they are sent to the machine, allow high-confidence machining, where top quality results can be guaranteed with minimal need for machine supervision.

## Keywords:

Machining, CAM, High-speed machining, five-axis machining

## 1 INTRODUCTION

The main current developments in machining technology are high-speed machining and five-axis machining. Mouldmakers that want to use these techniques will find a multitude of suppliers offering a wide range of new equipment in both these areas. After an initial period of scepticism, prompted mainly by some costly investments that failed to deliver the promised benefits, both mouldmakers and other toolmakers are becoming increasingly aware that these new technologies offer opportunities to shorten delivery times, boost productivity and increase profitability. It is also becoming increasingly apparent that all companies must adopt these high-productivity technologies rather than possibly relying on the lower labour costs that might exist in their countries.

Reducing costs have meant that there is less reason than ever to postpone any investment. Firstly, the price of both high-speed and five-axis machines has been falling steadily and this trend is forecast to continue. In addition, capabilities that were once only available on larger machines are now provided on the smaller ranges of equipment. It has, therefore, meant that technology that used to be economic only for the larger companies is now affordable by even small and medium size mouldmakers.

Similarly, the cost of the specialist cutters, particularly those required for high-speed machining, has also come down. When used correctly, they offer a longer cutting life and so can even offer more cost-effective metal removal. However, they are still more expensive than the standard cutters used for conventional machining, so the cost of a broken tool remains higher.

Finally, the cost of software and hardware to program both high-speed and five-axis machining operations has fallen. This has been most evident on the hardware side, with computing power that would have been unthinkable a few years ago now available at dramatically lower prices. Perhaps just as important as the reduction in costs on the software side have been the huge improvements in the ease of use of programming systems for these more sophisticated machining options. The time taken to learn

modern software has been reduced significantly, meaning that new users can become effective much more quickly and with much lower training costs.

## CAD for advanced machining options

The importance of starting with a high-quality CAD model when using high-speed machining and five-axis machining has become increasingly apparent, as there is more likelihood that problems with the CAD model will cause machining difficulties. Furthermore, the cost of repairing any damage to the cutter or the spindle will almost certainly be higher with either a high-speed machine or with five-axis equipment. After all, the key objective with both approaches is to cut faster, which inevitably leads to less time for operator intervention if things start to go wrong.

The most critical aspect is the tolerance used within the model, which must be set at a finer level than the planned machining tolerance. Adjusting the machining tolerance when manufacturing a poor-quality model will only mean that the defects will be machined more accurately.

This can be of particular importance if the STL format is used to pass the data between the CAD and CAM systems. STL is normally used for rapid prototyping, where the tolerances are not set as finely as those that are needed for machining. If the tolerances are not adjusted, the end result can be a faceted model that will need a lot of hand finishing to produce a smooth surface.

The negative side of machining to a finer tolerance is that calculation times are longer. Modern hardware does reduce the extra time needed. Even so, many CAM systems have a pre-defined limit below which the user cannot fix the tolerance. This does mean that calculation times will appear faster. However, the resulting surface will be of poor quality. Any time saved in the programming of the part will be lost many times over in the extra hand finishing that will be required.



Figure 1: Some CAM systems cannot work to fine enough tolerances. In this example, Delcam's PowerMILL produced the smooth finish shown in the top half of the surface, while other systems produced the faceted surfaces seen in two sections of the lower half.

Another key issue is the trimming of the various surfaces that go together to make up the model. Overlaps or gaps between surfaces can cause a problem for any three-axis machine. However, the higher speeds that are used with HSM mean that any attempt to plunge into a minute gap between two surfaces will cause serious damage to the cutter at least and possibly also to the machine tool. These problems can be even more serious with five-axis machining since the cutter can be oriented to move between any gaps, even those that a three-axis toolpath might ignore. Again, this can be a common source of difficulties if data is translated between systems with different tolerances set in the sending and receiving systems.

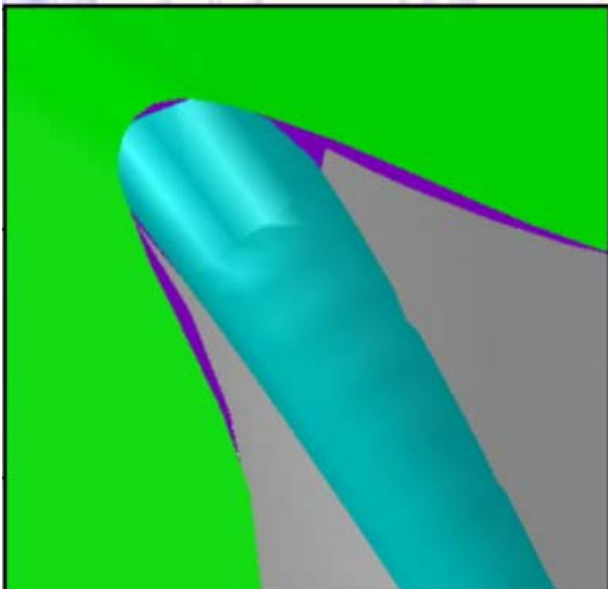


Figure 2: Poor trimming in the CAD model will lead to machining problems.

Similarly, it is important to cap over any holes in the model that will be drilled after machining the main surface. Failure to do this will otherwise again lead to the cutter attempting to plunge into the metal surface. Although, the potential for damage is less serious, it is also advisable to cap over any areas, such as small details, that will be removed later with spark erosion.

Another recommendation, which should again really be followed for all machining operations, is to ensure that the CAD model contains all the features that need to be machined. In the past, it was common for fillets to be left out of the CAD model and to be formed implicitly by the cutter during machining. With modern CAD systems, filleting is both much easier and quicker. Forming these features by machining inevitably involves the risk that the result will not be that which is required. Also, this approach puts excessive load on the cutter and on the machine tool, which will result in higher wear or even breakages.

## 2 HIGH-EFFICIENCY ROUGHING

The main requirements of high-efficiency rough machining strategies are to keep the load on the cutter as consistent as possible, and so maximise the life of the cutter, and to minimise any sudden changes in the cutting direction that will involve a slowing in the feed rate. One of the basic changes in strategy needed to achieve these conditions is the use of offset machining for roughing rather than the traditional raster approach. Whenever possible, machining should be completed from the centre of the job outwards to minimise any need for full-width cuts.

An even more efficient variant on conventional offset machining is called Race Line Machining, which was developed and patented by Delcam. With this option, the roughing passes are progressively smoothed out as the toolpaths move further from the main form. The resulting paths minimise any sudden changes in direction and so allow faster machining, with less wear on the cutter and lower stress on the machine tool.

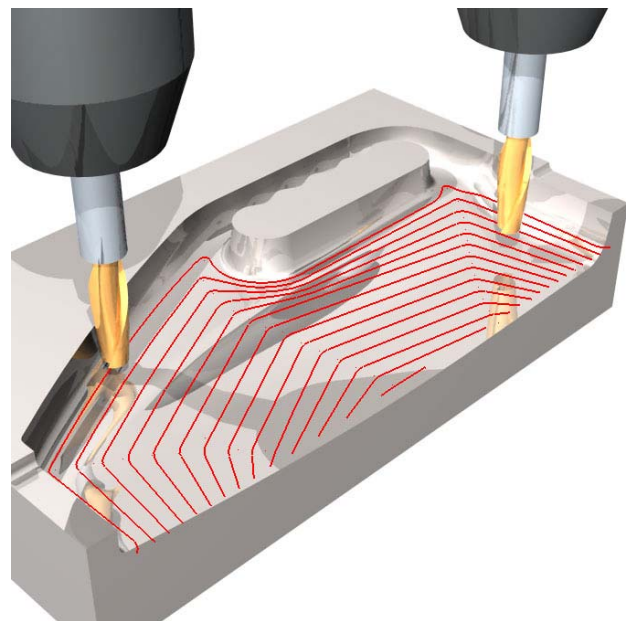


Figure 3: Race Line Machining smooths out the toolpath as it moves away from the main form

An alternative roughing strategy, which is used in a number of CAM systems, combines offset roughing with trochoidal machining toolpaths that progressively shave material from the block in a circular motion. This automatically avoids high-load cuts that could occur with conventional offset strategies, by switching to a trochoidal motion when the cutter encounters large amounts of material. Using a trochoidal action to remove material in these areas reduces the load on the tool to a more

consistent level, allowing higher machining speeds to be maintained.

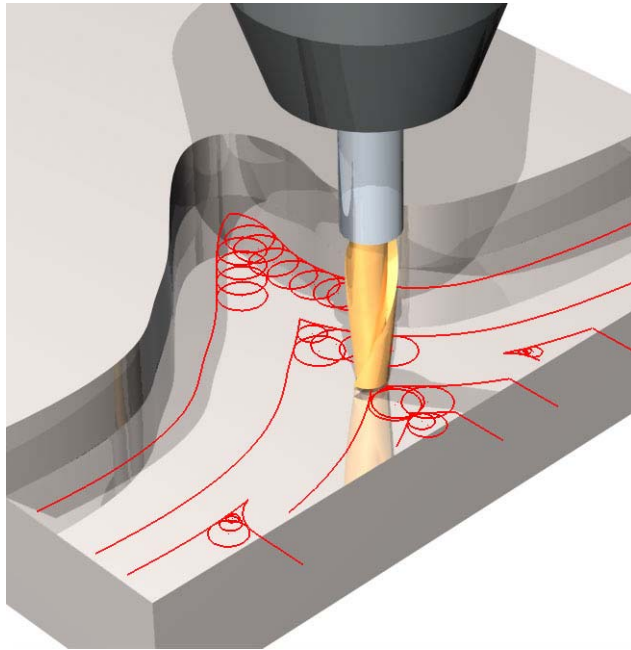


Figure 4: The machining strategy is automatically converted into a trochoidal motion in areas of high engagement to reduce tool loading.

The initial advice for high-speed machining was to keep both the step-over and step-down small compared to conventional machining. Recent developments in cutting tool technology mean that the latter restriction no longer applies. Roughing with deeper cuts is now possible by using a ball-nose cutter with four or six flutes that can cut with the side of the tool. The step-over, however, must still be comparatively small. While this high-efficiency approach uses lower feed rates than high-speed machining, the increased depth of cut gives increased metal removal, as can be seen in the table below from cutting tool supplier Mitsubishi Carbide<sup>(1)</sup>.

	Conventional method	High-speed 10,000~	High Efficiency
RPM	30~2,000 rpm	50.000 rpm	1,000~12,000 rpm
FEED	30~300 mm/min	1,000~20,000 mm/min	500~10,000 mm/min
Cutting depth (mm)	AD=10~15mm	AD=0.3~4mm	AD=10~15mm
usina			
	1~20 cm <sup>3</sup> /min	1~60 cm <sup>3</sup> /min	1~150 cm <sup>3</sup> /min

Table 1: The increased depth of cut with high-efficiency machining gives higher metal removal even though the feed rate is lower.

Climb machining is recommended for all roughing operations, as this will reduce tool wear. Although this approach does involve extra air moves, maximising the

speed of the cutting moves can make up the time. A combination of conventional and climb machining can be used safely in finishing operations where less than 0.3 mm of material is being removed.

With both rough and finish machining arcing moves should be used when approaching and leaving the job. Plunging onto the surface of the part should be avoided as this slows down the cutter and leaves a dwell mark on the surface. Similarly, arcing moves should be used to link the various cutting moves. The traditional approach of using right-angle moves will again require the cutting speed to be reduced and also leave dwell marks.

An additional feature to protect the cutter is the automatic fitting of arcs to toolpaths as the tool approaches any steep face. Without the arc, there will be a sudden increase in the load on the tool as it engages the steep surface. To prevent damage, the feed rate must be reduced so overall machining times will be increased. With the arcs added, a smoother toolpath results that can be run at higher speed with less wear.

When cutting corners, the radius of the cutter needs to be considerably less than the radius of the corner, so that the maximum contact distance can be kept to less than 30% of the circumference of the cutter. This allows sufficient cooling to take place and also avoids a sharp increase in the load on the tool as it enters the corner.

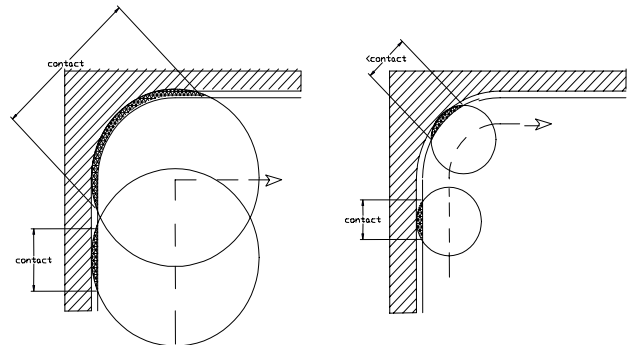


Figure 6: A smaller cutter size should be used than the corner radius to give a more even contact area as the tool enters the corner.

### 3 HIGH-SPEED FINISHING

The same basic requirements for roughing – consistent tool loading and the fewest possible sudden changes in direction – are also needed for high-speed finishing. To meet these priorities, a combination of strategies is recommended, with 3D offset finishing used on flatter areas and Z-level finishing used on steeper areas. The 3D offset toolpaths should be calculated with the surface stepover varying so that a constant cusp height is applied over the part, as this will give more even loading on the tool as well as reducing the amount of hand finishing needed.

The ability of modern tooling to cut with the side of the tool can also be used with finishing. This allows a spiral path to be used as an alternative to more commonly used Z-level technique. Instead of using the conventional approach of machining down the job level by level, this new strategy uses a single, smooth toolpath that machines upwards in a spiral from the bottom of the surface. This keeps the cutter in constant contact with the job and so eliminates air moves. The smoother

toolpaths also minimise any dwell marks as well as protecting the cutter and machine tool.

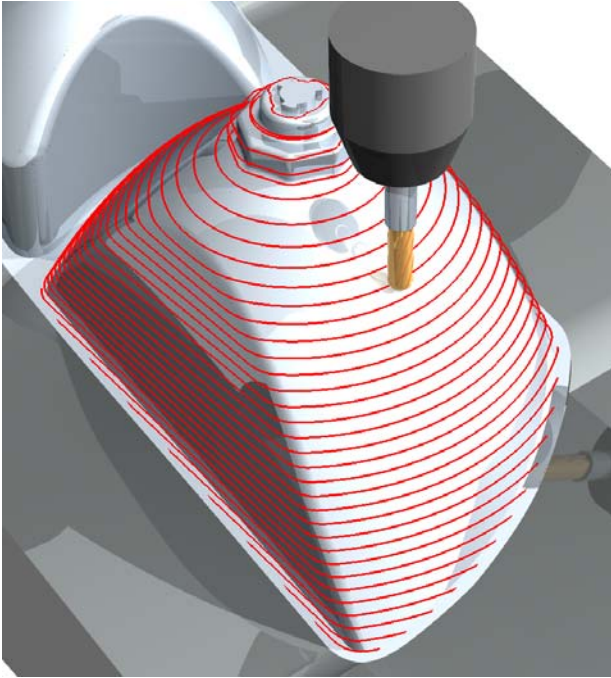


Figure 7: A spiral finishing toolpath keeps the cutter in contact with the surface and minimises air moves.

#### 4 POSITIONAL FIVE-AXIS MACHINING

In the five-axis machining area, two distinct types of machine are now being offered. Firstly, there are those machines that are, effectively, smaller versions of the equipment used for many years by the large companies in the aerospace industry, with all five axes of movement being generated through changing the orientation of the head.

The more popular option with smaller components, and for most mouldmaking operations, is to have a fixed orientation for the head, with the extra two axes being produced by tilting and/or rotating the bed holding the workpiece.

In the aerospace industry, there has been considerable use of continuous five-axis machining, in which the relationship between the cutting tool and the surface being cut is constantly changing in all five axes. In contrast, the most commonly used type of five-axis machining in the toolmaking industry is positional five-axis machining. With this technique, which is also known as 3-plus-2 machining, the head is oriented into a series of positions and machining carried out as a set of discrete operations.

With both positional and continuous five-axis machining, shorter cutters can be used, since the head can be lowered towards the job and the cutter oriented towards the surface. As a result of the shorter length, there is less vibration in the cutter so higher cutting speeds can be achieved with no loss in accuracy.

A second major benefit is the ability to machine complex shapes in a single set-up. This obviously saves time compared to performing the job in a series of set ups, and also reduces the time and cost of making fixtures.

Furthermore, with multiple set-ups, there is always a possibility of incorrect alignment each time the part is moved.

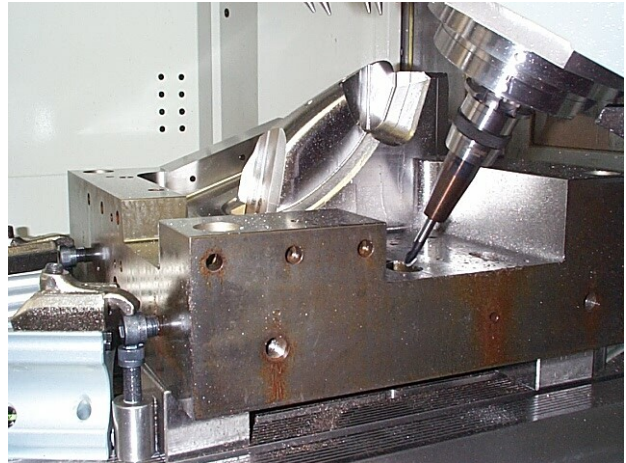


Figure 8: Five-axis machining allows shorter cutters to be used, so allowing faster, more accurate machining.

Another area where five-axis machining can give huge time savings is the drilling of holes. While this may seem trivial in comparison to the complexity of machining a complex core or cavity, drilling a series of holes with different compound angles is extremely time consuming, since, if a three-axis machine is used, a different set-up must be used for each hole. Most modern CAM software now incorporates feature recognition, which allows the positions of holes to be recognised and the different sizes and types to be grouped automatically. Templates can be developed within the software for different styles of hole, e.g. tapped holes, drilled holes, countersunk holes etc. These templates make programming much faster and more consistent, while the ability to machine holes in a variety of orientations in a single set-up makes machining times much shorter. For example, patternmaker Rojac found that drilling times for a typical component were reduced from two days to one hour.

In addition to the extra time required, it is very easy to make a mistake when changing between a long series of complicated set-ups. With a five-axis machine, the head simply has to be oriented along the required series of axes, allowing the drilling to be completed much more quickly.

There are several factors that have made positional five-axis machining the preferred approach for most toolmakers rather than continuous. A study undertaken last year by US analysts CIMdata<sup>(2)</sup> found that around three quarters of toolmakers with five-axis capabilities used this technique. One of the main reasons is that machining can usually be undertaken more quickly. Once the correct orientation has been achieved, the fact that the head is then fixed means that it is possible to operate at high spindle speeds. With continuous movement of the head, it is difficult to operate at higher speeds because of the inertia that has to be overcome. Early machines often offered much slower rotational acceleration along the b and c axes than the linear acceleration in the x, y and z axes. More recent machines offer faster rotational acceleration and so can be classed as both high-speed and five-axis equipment.

In addition, with positional five-axis operation, once each new workplane has been set, it is easy to check for possible collisions or gouging. These calculations are much more difficult with continuous five-axis machining. Thus, users can program positional five-axis equipment with a higher degree of confidence in the results.

## **5 CONTINUOUS FIVE-AXIS MACHINING**

With continuous five-axis machining, there can be simultaneous movement in all five axes. The main challenge for companies using this approach is to ensure that the head does not collide with the job as the orientation is changing. For toolmakers, this is less difficult when machining a core but can be much more problematic when machining inside a small but complex cavity. The benefit compared to the positional five-axis approach is that the angle of engagement can be continuously changed. This can allow increased metal removal as the angle can always be optimised. However, for most mouldmakers, the ability to give the best possible surface finish by adjusting the contact angle is usually more important.

One of the most common techniques promoted for continuous five-axis machining is based on keeping the cutter approximately normal to the surface being machined. While this approach has benefits when machining structures in the aerospace industry, it is less appropriate for other components, where access to the surface being cut will be more difficult. It can also cause problems in toolmaking, since the parts being machined, and the individual details within them, are typically much smaller.

Firstly, machining normal to the surface will require the machine head to travel a greater distance than necessary when machining a convex surface and so will increase machining times. It may be found that the normal position is inaccessible by the machine head, while, in other cases, it will be impossible to maintain the 90° angle without a collision with another section of the model.

Secondly, maintaining the cutter at a 90° to the surface of the part means that only a small part of the cutting surface of the tool is ever used. As a result, tool life is much shorter. Also, the tool tip travels a shorter distance at a given rotational speed than the full width of the cutter so the material removal rate will be lower when using ball-nose cutters.

An alternative approach is to generate five-axis toolpaths based on machining through a point or to a point, or to or from a line, or from a simplified surface. Software now available incorporates automatic adjustment of tilt angles for both lead and lag (Lead is measured in the cutting direction; lean at right angles to the cutting direction.). Control of both lead and lag can give better cutting conditions, while the lean angle is mainly used to avoid collisions. The optimum angle between the cutter and the workpiece will vary according to the materials and machinery being used but will normally be between 20 and 30 degrees away from normal.

While the software will maintain the specified angle for the majority of the toolpath, most CAM systems incorporate automatic collision avoidance to change the tool axis when collisions might occur. With these systems, the cutter is tilted away from the obstacle by the specified tolerance automatically and then returned to the value set for the overall toolpath once the obstacle has been

cleared. Various choices are normally available to control the direction in which the tool is tilted. The software will also calculate of gouge-free leads and links as the machine tool moves between the various surfaces within a part. These ensure that gouges are prevented and that collisions do not occur between the cutter and the part being machined.

As mentioned above, as well as increasing the range of strategies available, much work has been undertaken to make programming easier for five-axis machining. For example, secondary axis limiting has been added to ensure that the program generated will not attempt to move outside any limits in the machine tool's range of movement. Contact point analysis allows the user to ensure that only the cutting surface touches the component and that rubbing with the shank is avoided, and contact point feed rate control enables the feed rate to be set based on the cutting speed rather than the rotation rate of the tool tip. This last enhancement is of great importance when cutting some of the more difficult materials found in the aerospace industry but can also be useful when machining hardened steels used in toolmaking.

The software also has the ability to ignore surfaces that are not required for a particular calculation. This option can be helpful when machining multi-cavity moulds, where the same shape needs to be duplicated a number of times within the component. The user can select the surfaces making up a single cavity and instruct the software to ignore the remainder of the model. The toolpaths generated can then be copied around the die to produce the complete machining program needed for all the cavities.

## **6 MACHINING SIMULATION**

One of the most important elements in making both high-speed and five-axis machining more reliable and predictable has been the introduction of full machine tool simulations inside the computer. These simulations allow the user to see exactly what will happen on the machine tool before the program is passed to the shop floor. Warnings are displayed whenever there is any possibility of a collision, either between the machine and the job or between any parts of the machine itself. The main benefit of machine tool simulation is for five-axis machining since the added complexity makes a programming error more likely. As well as helping to avoid problems, the simulations can also be used to optimise different aspects of the machining process, such as the best position of the job in the machine bed or the optimum orientation for the fixture in which the job will be held.

- 2) NC Software and Related Services Market Assessment, Version 15, May 2006, Alan M. Christman, CIMdata Inc.

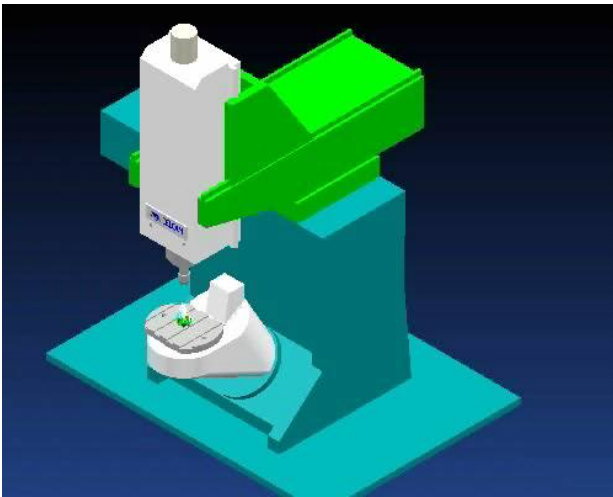


Figure 9: Machine tool simulations give confidence that the programme will run successfully on the shop floor.

## 7 CAM ISN'T EVERYTHING

Further development of strategies to support the latest machining developments continue at all of the leading CAM companies. However, no matter how good the CAM system may be, it is essential that all parts of a company's machining operations are reviewed when making the transition to these new types of machining. Many of the unsuccessful installations referred to at the start of this article encounter problems by ignoring one or more of the other factors contributing to efficient operation.

Buying the latest machine tool will give little benefit unless it is fitted with a modern control system that can cope with the larger volumes of data associated with high-speed and five-axis machining. Similarly, using slower CAM software or a less powerful computer will lead to frustrating delays, with the new machine tool lying idle while NC programs are being generated. Finally, suitable cutters must be used, even though they may be more expensive, or any gain in machining efficiency will be lost in the increased downtime needed to replace broken or damaged tooling. To achieve the correct return on investment and to give the expected improvement in delivery times, a balanced investment in each aspect of the process is required.

## 8 CONCLUSIONS

This paper has outlined some of the benefits of new technology, in particular, high-speed machining and five-axis machining. The new techniques do offer the potential to increase productivity, improve quality and consistency, and reduce lead times. They are, therefore, able to make mouldmaking companies more competitive, not only with their local rivals but also with the increasing number of toolmakers in international markets. However, just buying new machines tools is not enough to ensure success. Other parts of the overall programming and machining process must also be considered to gain real benefits from any investment

## 9 REFERENCES

- 1) Paper by Adrian Barnacle of Mitsubishi Carbide at a 2004 Delcam seminar on High-Speed Machining