

MACHINERY

THE MAGAZINE FOR PRODUCTION ENGINEERS

FEBRUARY 2011

Presetting pointers

Making sure that tools measure up

Gripping tales

Workholding under the spotlight



Rules of engagement

Renewed push behind toolpath optimisation

Rules of engagement

CNC milling toolpath optimisation has been around for a while, but there has been a recent spurt of activity from CAD/CAM vendors. **Andrew Allcock** reviews recent happenings, reveals why and gets into the detail (see much extended online version – <http://bit.ly/gSIXrv>)

CNC toolpath optimisation can mean a number of things, including the elimination of fresh air cutting or taking the shortest cutter path. But recent activity, and *Machinery's* main focus, is the optimisation of cutter/material engagement within a CAM system to deliver maximum/consistent cut volume or consistent cutter load or chip thickness – and 'feedrate optimisation' or 'adaptive feedrate' as a post-CAM operation, where output NC code is modified.

Within that broad classification, there are further differentiators, such as whether the technology supports 3D contoured toolpaths or 2½-axis cutting (X-Y moves at different Z depths), and whether roughing or both roughing and finishing are supported.

Key claimed benefits for toolpath and feedrate optimisation are reduced cycle times, improved tool life, improved surface quality and reduced machine tool wear.

At the International Manufacturing Technology Show (IMTS) in Chicago,

USA, last September, the technology was prominently highlighted by three machine tool makers – Okuma (NCMT, 020 8398 4277), Hardinge (0116 286 9900) and MAG (0121 306 5600). All these machine tool makers were employing American company Surfware's TrueMill technology, applied within its Surfcam CAD/CAM system to generate optimised roughing cuts for 2½-axis machining.

CUTTING TITANIUM

For Okuma, the focus was on demonstrating that smaller machines employing TrueMill-generated toolpaths can deliver benefits, even when tackling titanium. The company's new compact, high speed Millac-44H – a 320 mm diameter table horizontal machining centre with 8.2 kW spindle and Okuma OSP/THINC CNC – was shown machining 6AL4V titanium at a rate of over 4.5 m/min, using a 12 mm diameter end mill, working at a 25 mm depth of cut: it achieved peak material removal rates of almost 100 cm³/min. In parallel, improved

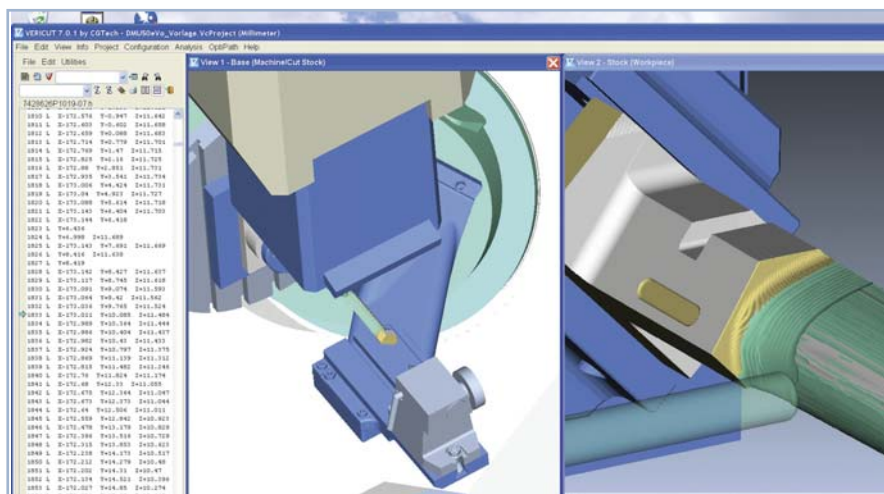
tool life was also highlighted.

Surfware's TrueMill, meanwhile, (www.truemill.com) began life in 2002, so has been around a while. It is the subject of US patents and is described by the company as "a quantum leap in CAM toolpath technology." The technology's key factor is that it maintains a constant engagement angle between tool and the material being cut to maximise the material removal rate (MRR). This eliminates, for example, dramatic increases in tool engagement when tools enter corners in pockets by producing novel toolpaths (see pictures, right), rather than modifying 'traditional' toolpaths. Indeed, Surfware CEO Alan Diehl offers this: "Toolpath optimisers [that work on NC code] are not capable of taking an existing conventional toolpath and modifying it, so as to achieve constant engagement."

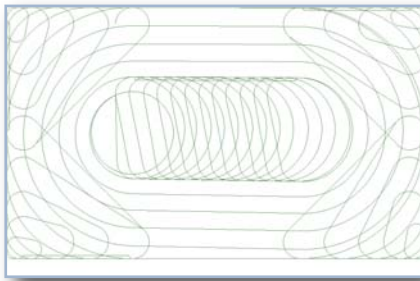
The only variation to the application of constant engagement angle is, highlights Mr Diehl, in 'corner picking' or areas where it is not possible, with a conventional toolpath, to machine the corner without exceeding engagement. Here, several tool passes of decreasing radii are arranged, such that engagement is not exceeded while machining each pass. This process allows machining corner radii only 10% larger than the tool radius without exceeding engagement.

The clue to why the technology, after almost eight years, is now attracting attention is because of the increasing use of difficult-to-cut materials, such as titanium, the company says.

According to Surfware, TrueMill toolpaths have been proven to increase material removal rates by up to 10x and increase tool life anywhere from 30% to 100%, while at the same time decreasing cycle times and reducing the



Post-NC code generation optimisation via CGTech's Vericut software



TrueMill generates 'novel' toolpaths, not offsets of the geometry, as is the norm

stress on CNC machines. The company will soon be licensing TrueMill for use by third parties.

Vericut CNC toolpath simulation software (CGTech, 01273 773538) is another software package that has had toolpath optimisation as an optional element for many years – OptiPath. Vericut is not a CAM system, but is instead used to review and check actual CNC G-code generated by CAM systems, including 3D contouring paths, which Optipath also supports.

UK agent CGTech's managing director, John Reed, suggests Vericut has an advantage over CAD/CAM systems. "The key issue is that Vericut produces a constantly updated in-process cut model and, therefore, has the information about the tool and the part used to calculate constant volume and chip thickness cutting. This in-process cut model is not available in CAM systems – it is a bi-product of the simulation process."

And he questions how the calculations are being performed within CAM systems concurrent with toolpath generation. Responding for Surfware, Mr Diehl highlights that Mr Reed is correct in respect of 3D contouring toolpaths, but that "it is not true for 2D or 2½D machining, where the cuts are made at various Z levels".

Vericut's OptiPath modifies existing 'traditional' toolpaths for roughing and finishing – it does not alter the trajectory. During roughing, the goal is to remove as much material, as quickly as possible. Here, OptiPath keeps the cutter at its maximum safe rate-of-advance into material for the varying cutting conditions.

When it comes to finishing, chip loads typically vary widely, as the tool profiles through the material left behind during



Oliver Burston

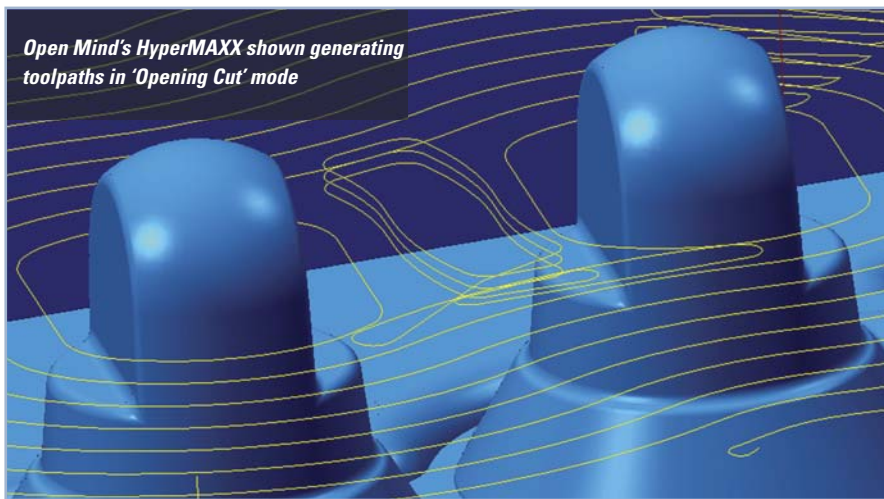
roughing cuts and over the contours of the workpiece to near net shape. OptiPath adjusts the feedrates to maintain a constant chip load – consistent chip loads are recommended by cutting tool makers to reduce 'chip thinning' – with the result that tool life and surface finish are improved.

Like Surfcam, Mr Reed agrees that the benefits of such optimisation are magnified when machining challenging materials, but machining complex shapes using 5-axis machines, regardless of material, is also cited as an area of beneficial application.

"What people tend to do, to avoid cutters diving in to material as geometry changes, is to program for the worst-case scenario, which obviously gives

problems. Conversely, people using OptiPath don't worry about the lateral feedrate; they put a really high one, because they know OptiPath will reduce it." (See extended article for details of Vero and SESCOI's NC code optimisation software.)

Speaking for Planit's Edgecam (0118 975 6084), strategic product director Raf Lobato suggests that the post-CAM optimisation of toolpaths is really an admission of poor strategies within the CAM system. "It is a failing when a toolpath that comes out needs optimising: it is the wrong place to do it, because you have lost all associativity with the CAD model, while it is just addressing weaknesses in a CAM system. "I used such a product on a



Open Mind's HyperMAXX shown generating toolpaths in 'Opening Cut' mode

toolpath not generated by Edgcam and got a 30% cycle time improvement. However, we remodelled the toolpath within Edgcam and, low and behold, discovered they were using the wrong diameter tool. They were machining a large pocket with small corners, using the cutter that suited the corner radius. By using a roughing and rest roughing strategy in Edgcam – so a large diameter cutter, followed by a smaller diameter cutter – the time saving wasn't 30%, it was nearer 75%.

"Optimising afterwards does have benefits, but it's a bit like 'souping up' your car and driving faster down a country road while ignoring the motorway alongside on which you could be travelling along even faster." And Mr Lobato adds that, when applied to Edgcam toolpaths, the improvements are "minimal".

"Additionally, CAM output must also take account of the machine control. Some modern ones will want output in the form of many small G01 moves (straight line moves), older controls cannot cope, so we can offer output options such as arcs or even NURBS. Without the data appearing at the machine tool in the correct format, any optimisation might be wasted, if the machine can't keep up with the data flow," he adds.

But while the company claims it has efficient roughing strategies, it is about to unveil a new strategy – Wave Form – which supports constant chip load (constant volume) 2½ axis milling. Wave Form will result in novel-shaped toolpaths

that are not a simple offset of the shape of the final geometry, like TrueMill. Edgcam's technology has been developed with Sandvik Coromant to obtain best entry and exit paths, while eliminating repeated entry and exit that blunts the tool – a weakness of trochoidal milling, for example. "In this area, we believe we have leapfrogged the competition," says Mr Lobato.

The major benefit will be on hard materials and he suggests a 35% improvement over existing roughing cycle times "is not unrealistic"; assuming machine and cutter technology that is able to exploit Wave Form.

NEW CLUB MEMBERS

Two recent members of the toolpath optimisation club are CAD/CAM companies Delcam (0121 766 5544) and Open Mind Technologies (01865 338026). Open Mind's offering is hyperMAXX, a roughing strategy powered by the established Volumill toolpath engine from America's Celeritive Technologies – set up by ex-Surfware personnel, in fact. (GibbsCAM (Tech CAD/CAM, 01284 754781), Mastercam (4D Engineering, 01285 650111, Esprit (Cam Supplies & Services, 0871 218 3001) and Sigmanest (SigmaTEK Europe, 024 7632 3065) can also employ Volumill technology, although it may not necessarily be integrated within the package.)

Volumill is, like Truemill, a technology for 2½D rough machining, producing 'novel' toolpaths. And while it does not support 3D contouring, it does support machining of 3D geometry at multiple Z

levels, via use of two, distinct depths of cut within the same toolpath.

This approach is said to minimise roughing and overall cycle times, machining the part to a near-net shape and requiring the use of fewer tools. Surfware offers a similar approach, called Step Reduction Milling, which is combined with TrueMill, while Edgcam's 'Intermediate Slices Technology' is also similar.

Volumill has two other cutting strategies: 'opening cut', suited for full cut machining of areas that are difficult to reach; and effective in softer materials, such as aluminium; and 'side cutting only', which sees lateral in-feed movements, without full cutting, designed for machining harder materials. Open Mind claims that roughing time savings of 50% or more are typical with hyperMAXX.

Delcam's technology supports both rough and finish cutting, plus it accommodates 3D contoured surfaces. The optimisation capability is a feature of the 2011 version of its PowerMILL CAM system for 5-axis and high speed machining. Key is the new stock-model-engagement option.

A number of CAM programs incorporate strategies based on the extent of cutter engagement to give more consistent loading on the tool and so allow higher feedrates, explains Delcam, but these options are "usually limited to the initial roughing operations, or to only roughing and rest-roughing". In comparison, PowerMILL's new stock engagement technology can additionally be employed with all of the CAM system's finishing and rest-finishing strategies.

So, optimised toolpath generation concurrent with CAM programming or after the event feedrate optimisation, with all this recent development it might be prudent for machinists to revisit this area to glean what benefit there is to be had. □

Note: Extended feature covers Vero, SESCOI and FeatureCAM feedrate optimisation technology; NC code optimisation for slower CNCs; CNC adaptive feedrate control; further reading.