



The Mill and Drill

Vol 1, Issue 36 (Apr. 1, 2005)

DEAR CUSTOMER

Welcome to your weekly installment of "*The Mill & Drill*", TriMech Mfg.'s Technical Newsletter for FeatureCAM. This newsletter is intended to keep you, the hip TriMech FeatureCAM User, on the cutting edge of your FeatureCAM seat. Don't forget to visit www.trimech.com for additional information.

NOTICES

Delcam Acquires EGS

Delcam has consolidated its position as the leading world-wide supplier of specialist machining software and associated services with the acquisition of the US-based CAM software specialists Engineering Geometry Systems, the developer of the FeatureCAM CAD/CAM software line.

FeatureCAM will continue as an independent product line, developed in the USA and marketed worldwide. "The only immediate difference existing FeatureCAM users will see is a change in the company name to Delcam USA," said Delcam's Managing Director, Hugh Humphreys. "I am pleased that all EGS' management and staff have agreed to stay with the company. Both companies produced record sales in 2004, so it only makes sense to continue to strengthen both product lines and aggressively market them worldwide."

Introduced in 1995, FeatureCAM was the world's first Windows- and feature-based milling system. The product range has grown significantly to include turning, wire-EDM, mill-turn packages and feature recognition for imported CAD files.

"The acquisition will help provide the additional exposure and resources necessary to move FeatureCAM to an even higher level of automation to better serve our customers," says Glenn McMinn, President of Engineering Geometry Systems. "Delcam is an experienced player in the CAM business and, as a result, the company has a much broader reseller network with more than 150 offices in 60 countries. Being part of Delcam will help us grow our sales significantly and increase our international market share."

With this acquisition, Delcam will provide an even broader range of quality products to address the worldwide need for CAM at various price points. "We recognize the importance of the FeatureCAM brand name in the industry and its unique position. The addition of the FeatureCAM family will accelerate our sales growth by giving us strength in automated and feature-based production machining. This will complement our industry-leading products for toolmaking and other complex manufacturing operations," commented Mr. Humphreys. "In addition, FeatureCAM's high reputation with its users will further raise Delcam's profile and market share in the CAM industry, especially in the important North American market."

We think this is a very positive move and will prove to be great news for you, the TriMech customer. In addition to having a better, stronger product in FeatureCAM, you will still enjoy the

services and support that TriMech Manufacturing offers. If you have any questions or concerns at all, please don't hesitate to give us a call at (540) 949-7703.

FeatureCAM TRAINING SCHEDULE

Note: Our main training facility is in Fishersville, VA, but we can make special arrangements for classes at any of our other TriMech offices (Columbia, MD; Pittsburgh, PA; Richmond, VA; Durham, NC; and Charlotte, NC). We also provide on-site training.
Call (540) 949-7703 for rates and details.

Fishersville, VA	
FeatureCAM Basic (2 days)	Apr. 6,7 May 4,5
FeatureCAM Advanced Milling	Apr. 14 May 12
FeatureCAM Turning	TBD

TIPS AND TRICKS

Tip of the Week

The great thing about having such smart customers is that we often have the opportunity to learn from you. Here's a little nugget that the guys from EVI showed us.

You can change tools, speeds, and/or feeds for more than one operation at a time by using the Operations List. Here's how: Open the Results window and ctrl-select all the operations that you want to share the same tool (or feed or speed). Order is important here; the attribute from the first selected operation (in machining order, not selection order) will be set as an override for all the selected operations. Once they are selected, right-click on the selection. In addition to the regular navigation options, you will have the choice to select an override.

In the following example, the #2 center drill will be used for all of the hole operations, so it will be selected, and all of the operations that are to be changed will be selected. Then, right click on the operations to get the override options.

F	Operation	Feature	Tool	Feed	Speed
↑	spotdrill	hole3	center_2	0.7 IPM	868 RPM
↑	drill	hole3	TD_00995_39:J	0.7 IPM	960 RPM
↑	spotdrill	pattern1.hole1	center_3	0.7 IPM	707 RPM
↑	spotdrill		center_3	0.7 IPM	707 RPM
↑	spotdrill		center_3	0.7 IPM	707 RPM
↑	drill		TD_01250_1 8:J	0.7 IPM	764 RPM
↑	drill		TD_01250_1 8:J	0.7 IPM	764 RPM
↑	drill		TD_01250_1 8:J	0.7 IPM	764 RPM
↑	spotdrill		center_5	0.7 IPM	248 RPM
↑	spotdrill		center_5	0.7 IPM	248 RPM
↑	spotdrill		center_5	0.7 IPM	248 RPM
↑	drill		TD_03750_3 8:J	0.7 IPM	255 RPM
↑	drill		TD_03750_3 8:J	0.7 IPM	255 RPM
↑	drill		TD_03750_3 8:J	0.7 IPM	255 RPM
↑	Results				

Dimensions

Location

Strategy

Misc

Tools

Feeds and Speeds

Drilling Attributes

Drilling Cycle

Override with center_2

Override to 0.7 IPM

Override to 868 RPM

Here are the results. This method sure beats changing the tool one feature at a time.

F	Operation	Feature	Tool	Feed	Speed
↑	spotdrill	pattern1.hole1	* center_2	0.7 IPM	707 RPM
↑	spotdrill	pattern1.hole1	* center_2	0.7 IPM	707 RPM
↑	spotdrill	pattern1.hole1	* center_2	0.7 IPM	707 RPM
↑	spotdrill	pattern2.hole2	* center_2	0.7 IPM	509 RPM
↑	spotdrill	pattern2.hole2	* center_2	0.7 IPM	509 RPM
↑	spotdrill	pattern2.hole2	* center_2	0.7 IPM	509 RPM
↑	spotdrill	hole3	center_2	0.7 IPM	868 RPM
↑	drill	pattern1.hole1	TD_01250_1 8:J	0.7 IPM	764 RPM

If you have any shortcuts, tips, or tricks that you would like us to put in the *Mill & Drill*, send them to us. We'd love to hear your ideas.

QUESTIONS/COMMENTS?

Please send any tips/tricks, feedback (regarding this newsletter) or requests to be added/removed from our distribution list to...

ccole@trimech.com