



The Mill and Drill
Vol 1, Issue 21 (Nov. 19, 2004)

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

No Newsletter Next Week

"*The Mill and Drill*" will be taking a little break next week for the Thanksgiving holiday. We at TriMech Manufacturing would like to take this opportunity to thank *all* of you for your business, and we look forward to serving you in the future.

****FLASH - TriMech Manufacturing now offers sheet metal fabrication software**

That's right, TriMech is now a value-added reseller for Computes. Computes offers a powerful, integrated solution for programming punches, lasers, waterjets and plasma cutters. It also has MasterNEST - an advanced system for single- or multi-sheet nesting. Call John Kanney for details. (540) 949-7703

Did you miss a newsletter?

Don't worry, the newsletters are posted, every issue, in PDF format on the TriMech Web Site (www.trimech.com). Just hover over TriMech Manufacturing, Technical Support and you will see "The Mill & Drill" letters to the right side.

Is there anybody else at your company who would like to receive the newsletter?

Send us their name and e-mail address, and we'll add them to the list.

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)	Dec. 7,8
FeatureCAM Advanced Milling	TBA
FeatureCAM Turning	TBA
TIPS AND TRICKS	
FeatureCAM training...it's not just for machinists anymore.	

Get your managers and designers involved with FeatureCAM. What better way is there to get everybody on your manufacturing team to understand the potential of this powerful CAM tool? We are also starting to see more designers who take our classes in order to better understand how their solid models transition into the manufacturing environment with **FeatureRECOGNITION**. Here's what one of our recent attendees had to say:

"I would like to take this time to thank you for the basic and advanced classes in Feature Cam. I got quite a bit out of the class with respect to the cad/cam interface. The software is a powerful tool that will provide us here at [our company] the ability to go from cad to cam in a seamless manner. I strongly recommend any manager that runs a machine shop or any designer that is involved with the production cycle of a product to take your courses."

Jose M. Bouza II
Manager, Manufacturing

Thanks, Jose!

Macro of the Week

Last week, we mentioned that FeatureCAM can be customized with the use of macros, and we promised to highlight some of the ones that EGS offers. Here's the first one:

MakeHexagon.bas is a handy little macro that comes loaded in FeatureCAM and it simply draws a hex. Sure, you can draw one manually, but it's a pain. Here's how it works: Go to **Options > Add-ins**, check the box beside MakeHexagon, and hit OK. See the new icon on your toolbar? Don't hit it yet! First you have to draw a circle or arc that the macro can use for size and location. The hex will be created around that entity (inscribed circle). So, select the circle, *then* hit the Hex icon. If you need to change the angle of the hex, select all six sides and **Transform > Rotate** around its center. Give it a try.

Controlling Z Increments

FeatureCAM uses default values to control the depth increments of milling cuts. The stock values are 100% of the tool diameter for rough operations, and full depth (within limits of the flute length) for finish operations. One way to change these values is through your **Machining Attributes** and **Configurations**, but you can also control these values on a feature-by-feature basis. Go to the feature properties and click on the milling operation that you want to change. Click on the **Milling** tab, and look for the Z increment parameter. It is called **Rough pass Z increment** in rough operations, and **Finish pass Z increment** in finish operations. Highlight the parameter, fill in the desired value, and hit Set.

QUESTIONS/COMMENTS?

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

jkanney@trimech.com