



The Mill and Drill
Vol 1, Issue 11 (Sept. 10, 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

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.

FeatureCAM at IMTS

If you're going to IMTS this year, stop by and see the latest that FeatureCAM has to offer at booth **D3037**. Be sure to tell the folks from EGS that you're a TriMech customer!

Speaking of IMTS...

We are offering a 10% discount on FeatureCAM base modules purchased by the end of the show. This offer applies to all customers, whether they attend the show or not. Orders must be received by **September 15, 2004**. Call (540) 949-7703 for more details.

TriMech Manufacturing is Growing...

...to serve you even better. Initial plans are under way to add Manufacturing personnel to the Pittsburgh area and North Carolina. We are still early in the recruitment stage for Sales Representatives/Applications Engineers, so we will be making more announcements as details become finalized.

Tell a friend!

Don't feel like you have to keep our newsletter all to yourself! If you have friends and associates who aren't TriMech customers, but would benefit from our information or services, feel free to forward our newsletter to them.

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)	Oct. 5,6
	Nov. 9, 10

	Dec. 7,8
FeatureCAM Advanced 2.5D Milling and FeatureRECOGNITION	Sept. 16
FeatureCAM Turning	Sept. 23
TIPS AND TRICKS	
<p>Maximum Tap RPM</p> <p>Sometimes, FeatureCAM can generate tap RPMs that are <i>way</i> too fast. Well, in FeatureCAM 2005 there is a way to avoid these crazy speeds - and a bunch of snapped taps! Go to your Machining Configurations and edit My Configuration. Access the Drilling tab, find the field called Max. tap spindle RPM, and set the speed to the desired value. From now on, no tap will run faster than this speed, and your speeds and feeds will remain proportional.</p> <p>Multiple Profiles in a Feature</p> <p>Did you know that you can use more than one profile curve in a feature? For example, if you have several pockets, you can create one feature that cuts all of the pockets instead of having a separate feature for each one. When you are at the Curves page of the New Feature Wizard, just select all of the curves you want to use and add them to the list. Using this method, you can change or manage all of the pockets with one step instead of several.</p> <p>Now, there are a couple of rules here: All of the profiles have to be created at the same Z level, and they have to be the same depth. Another rule is that you obviously can't combine different <i>types</i> of features such as pockets and bosses into one feature.</p> <p>But what if you have a boss and an open side feature that are on the same level? If you create them in separate features, the boss will wipe out the side, or the side will wipe out the boss. In this case, you <i>have</i> to combine them so that they "see" each other, even though they are different feature types. Select <u>both</u> curves, and create a side feature. Remember, you can use open <i>and/or</i> closed curves in side features...just make sure your machining sides are correct.</p> <p>Ordering Multiple Profiles in a Feature</p> <p>OK, you followed that last Tech Tip, and you have all of the pocket profiles combined into one neat, clean feature. How do you control the order in which the feature cuts the profiles? (First of all, you upgrade to FeatureCAM 2005, because this is a new enhancement!) After you finish creating the feature, open the feature Properties and click the Curves button. (Note: some features label this button Boundaries.) You will notice a new Ordering section in the lower right-hand corner of the dialog box. Automatic Ordering will be selected by default, but you can switch to Manual Ordering and the profiles will be machined in the order that they appear on the left. You can change the order of the curves by selecting curves and clicking the Up and Down arrows in the Ordering box.</p>	

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