



The Mill and Drill
Vol 1, Issue 18 (Oct. 29, 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.

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)	Nov. 3,4
	Dec. 7,8
FeatureCAM Advanced Milling	Nov. 18

TIPS AND TRICKS

Special Curves

FeatureCAM has tools for creating all kinds of "special" curves, such as text, ellipses, gears, etc. To access these, go to **Construct > Curve > Other Methods**. Each selection will take you to a simple dialog box where all you have to do is fill in the blanks with the information the curve needs. The result can then be used for a variety of different features.

Tool Selection Part I: Spot Drills v. Center Drills

When you spot drill holes, do you prefer to use a spot drill, or a center drill? FeatureCAM can actually record that preference so that the drill style you want is selected automatically. Go to **Manufacturing > Machining Configurations > My Configuration > Edit**. Click on the **Tool Selection** tab and select which drill type you

prefer (upper left-hand corner).

Tool Selection Part II: Counter bores v. Endmills

Since you we are talking about the **Tool Selection** tab, check out the *lower* left-hand corner. There is a section where you can dictate what kind of tool is used for counter bores as well, and here you have three choices. "**Use counter bore**" will go to your tool crib for the correct counter bore. If that tool is in your crib, fine, but if it's not, you will get an error when you try to run a simulation. "**Use an endmill**" will automatically select an endmill that is smaller than the bore size, and it will circle-mill the counter bore. "**Automatic**" will search your crib for the correct counter bore; if it can't find one, it will use an endmill to circle-mill instead.

Check this out: Let's say you need to cut a .5 counter bore and the software has selected a .4375 endmill to cut it. The tool will mill around the bore, and you can control it like any other milling feature. If you change to a .5 endmill and hit **Apply**, the software is smart enough to switch over to a drilling cycle such as a G82. Slick, huh?

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