



*The Mill and
Drill*

Vol 1, Issue 37 (Apr. 8,
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

Are you on FeatureCAM maintenance?

If so, great! If not, you really should consider it. Here's what you get with an annual maintenance contract:

Software updates - You can get the latest and greatest updates from EGS including one major version upgrade.

Post processors - We will either customize what you have, or we will provide new posts for any additional machinery that you might purchase.

Technical support - Phone, fax, or e-mail us M-F from 8-5.

Plus, you won't feel left out in a couple months when we start raving about all the awesome new stuff in **FeatureCAM 2006!**

Prices vary depending on what modules you own and their current maintenance status, so give John Kanney a call for detailed pricing info.

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)	May 4,5 June 8,9
FeatureCAM Advanced Milling	May 12 June 16
FeatureCAM Turning	TBD

TIPS AND TRICKS

Machining Attributes

One of the great things about feature-based CAM software is that it is so automatic. Once you describe the feature, you hit Finish, and everything else is done for you; tools are selected, stepovers and pecks are established, feeds and speeds are set, etc., etc.

How does FeatureCAM make all of those decisions? How does it know to rough *and* finish a pocket? Why is the tool ramping back and forth into the material? The answer to these questions and many more are found in the **Machining Attributes**.

Go to **Manufacturing>Machining Attributes**, and take a look at the dialog box that appears:

The first thing that you will notice is that this set of attributes has a name, or

configuration, and the name matches your part name. That means two things: settings in this configuration will affect this *entire* part and this part only; these settings are embedded in the part file, so that if this part is opened on another computer, its settings stay intact.

Browse through the tabs and look at the settings that are available. (Your tabs may look different than these, depending on what FeatureCAM modules you own.) A lot of these will look familiar because they appear in the feature wizards, usually on the Strategy page. If you have any questions about the parameters, hit **Help** and the context-sensitive help will provide you the answers you need. Once you become familiar with these pages, you'll have a better idea of how a feature is constructed.

Any changes you make to the Machining Attributes will carry through the entire file unless they are overridden at the feature level. So if you were to uncheck the Spot Drill box, none of the drill features in this part would include a spot drill operation.

Next week, we'll take the setting one step further with Machining Configurations.

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