



The Mill and Drill
Vol 1, Issue 17 (Oct. 22, 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

TIPS AND TRICKS

Halt!! Who goes there?

E-mail is a great tool for us at TriMech. We can send you example programs and *The Mill & Drill*. You can send us sales and tech support questions. However, if you are going to send us e-mail, please send it with a name and/or subject that will help us identify your e-mail as valid e-mail. There is a lot of trash floating around in cyberspace, and if we get e-mail that we don't recognize, we probably won't open it. In fact, our virus protection might weed it out before it even makes it to our inboxes. We'd hate to miss any communication from you, so please clearly identify your e-mail.

Reading Graphics Files into FeatureCAM

We are often asked if it is possible to open graphics files such as BMPs and JPGs with FeatureCAM. You can't open them directly, but there are programs available that *can* read those files in and convert them to DXF files which FeatureCAM will open. One of

those programs is called **WinTopo**, and it available for free through the FeatureCAM website. Go to www.featurecam.com , click on **Support**, and scroll to the bottom of the page. The link is on the right.

OK, what did they do with the Maximum RPM setting?

*I used to have to set the maximum RPM for my machine in the Machining Attributes **and** the Post Options screen. Now it's in neither place.*

Maximum RPM is now part of your post processor, so you have to edit you post in order to change the setting. This actually makes better sense, considering the maximum RPM is a function of your machine tool, not the part file. To change it for a particular post, go to **Manufacturing > Post Process**. Make sure the correct post is active and click **Edit**. This will launch your post editor, **X-Build**. Once inside X-Build, click on **CNC-Info > Speeds and Feeds**. There it is! Set your max RPM, save your post, and exit X-Build.

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