



The Mill and Drill

Vol 1, Issue 27 (Jan. 28, 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

TriMech is more than just CAM/DNC software!

Did you know that **TriMech Manufacturing** is only one of the TriMech Companies? We also have divisions that can handle all your CAD needs (**TriMech Solutions**) as well as engineering consulting and staffing (**TriMech Services**). Please check out www.trimech.com and explore the many services and solutions that the TriMech Companies offer.

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)	Feb. 9,10 Mar. 9,10
FeatureCAM Advanced Milling	Feb. 17 Mar. 17
FeatureCAM Turning	TBD
TIPS AND TRICKS	
Zero Speeds in Turn	

A few customers discovered that when they upgraded to FC2005, some of their turning posts started giving them speeds of 0. We're not exactly sure why this is happening, but we know how to fix it. Go to **Manufacturing > Post Process**. Verify that the correct post is active, and click **Edit**. X-Build (the post editor) will launch in a separate window. Click on **File > Save CNC** in the X-Build window. You will get a warning that the post will be converted from an older version and will become unusable by older versions. No problem...this is what we're after, so go ahead and save the post. Now check out your code; the speed values should be restored.

Trochoidal Toolpath in Grooves

This rather large word describes a specialized strategy in groove and slot features that moves the tool in an orbiting motion. This only applies to Simple grooves and slots, and is enabled with a checkbox on the **Milling** tab of the operation:



The feature will then take a smaller endmill and spiral down the centerline, essentially "nibbling" away the material.



This strategy enables the user to cut grooves without taking 100% of the tool width, which can be great for harder materials.

Exporting Configurations

Do you have a Machining Configuration that you have customized and want to share with another user in your company? You can export it to them. Go to **Manufacturing > Machining Configurations** and click the **Export** button. Select the destination path and choose the configuration(s) that you want to send out. Click the **Export** button, and off they go. The user on the other end follows a similar process to **Import** the configuration into his or her system. This is also a good way to save customized configurations off to a safe place so that they won't get overwritten when you reinstall or upgrade.

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