



The Mill and Drill

Vol 1, Issue 34 (Mar. 18, 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

FeatureCAM 2006 in on the horizon! Are you on 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 few 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	
	Apr. 6,7

FeatureCAM Basic (2 days)	May 4,5
FeatureCAM Advanced Milling	Apr. 14 May 12
FeatureCAM Turning	TBD
TIPS AND TRICKS	
<p>Hide Unselected</p> <p>Have you ever wanted to see only one or two entities in a file and hide everything else from view? Let's say you want to hide everything but two features (you have ten in all). You could Hide All, Show All Features, select all of the features that you <i>don't</i> want to see, and then Hide Selected, right? No, no, no. There's a <i>much</i> easier way. Just select what you want to see and Hide Unselected. (All of these commands are found under View > Show and View > Hide, or through the View and Show flyout menus on the Advanced Toolbar.)</p> <p>Part View v. Steps</p> <p>You may have noticed that the Toolbox window has two different "modes". When you install FeatureCAM, the Toolbox by default will show the Steps to go through to create your part. Each one of these steps is a wizard or a dialog box that guides you through the task at hand. This is a great way to learn, but before long you'll find yourself using the toolbar icons more and more, and the Steps less and less. This is where Part View becomes a valuable tool. You access it by clicking "Part View" at the top of the Toolbox. Part View is more like a history tree, and inside of it you'll find the stock, setups, features, curves, solids, and surfaces that are in your file. You can use these names to navigate through your part, edit entities, and order your operations. This is especially handy for larger files that have a lot going on graphically. To get back to the Steps mode, just click on "Steps."</p> <p>Save NC!!</p> <p>No, not North Carolina...they're just fine. This is the command that saves your NC code for your machine tool. When you use FeatureCAM, you are really working with two files: the fm file and the g-code, or NC file. The fm file is what you open with FeatureCAM, and it contains all of the graphics, features, tooling, etc. Once that is all finished and you have simulated the toolpath, the g-code has to be saved as a separate text file. To do this, go to File > Save NC. A dialog box will open, and you will have to select the destination folder for your file. You will also have several checkboxes for what files will actually be saved. We suggest unchecking every file except the NC code. All of this other information is part of your fm file anyway, so nothing will be lost, and you won't end up with 5 or 6 extra files cluttering up your disk. Now that the NC code is saved, you can take it out to the machine tool.</p>	

Save NC

NC Output Directory

Save to current directory:
C:\FeatureCAM\11.2.0.23\Examples

Save to other directory:
a: [] Browse...

NC File Name:

Save NC Program Using Short File Name

Selection

All Setups Current Setup

Operations List Tool Data

Tools List of All Setups F/S Data

Tools List of Each Setup Machining Configuration

NC Program

Create Subfolder Overwrite Existing Files

OK Cancel Help

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