



The Mill and Drill
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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 Manufacturing now offers sheet metal fabrication software

That's right, TriMech is now a value-added reseller for **Computes**. **Computes** offers a powerful, integrated solution for programming punches, lasers, waterjets and plasma cutters. It also has **MasterNEST** - an advanced system for single- or multi-sheet nesting. Call John Kanney for details. (540) 949-7703

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)	Dec. 7,8
FeatureCAM Advanced Milling	Dec. 16
FeatureCAM Turning	TBA

TIPS AND TRICKS

Something to watch out for with SolidWorks files...

Lots of our CAM customers import SolidWorks CAD files into their FeatureCAM. The files come in great, but there is one thing that we should bring to your attention: Even though you might be importing a single **.sldprt** file, FeatureCAM "sees" all of the solids that were used to create the part. If it is an assembly that was saved as a part, FeatureCAM will still list of all of the solids in the original assembly. That shouldn't come as a big surprise, but if the **.sldprt** was created by importing and modifying an

existing solid body, FeatureCAM will see the final result *plus the original imported body even though there is only one solid body in the SolidWorks file*. This isn't a big deal as long as you understand what's going on. Just delete or hide any solids that you don't need.

Transforming Multiple Solids

Speaking of multiple solid bodies, here's a related topic that came up this week: *Suppose I do bring in an assembly. How can I transform all of the solids together? I can't seem to select more than one solid at a time.*

First, hide anything that you don't want to move. (Ideally, you should get all of the moving out of the way before you create features anyway.) Go to **Edit > Select All**. Then, you can go to **Transform** and move the whole assembly as one unit. (Thanks, Mike.)

Macro of the Week

MoveSetupToUL.bas This is another handy little macro that comes loaded with FeatureCAM that will move your current setup to the upper left-hand corner of the top of the stock with a single mouse click. Why? Well, a lot of machinists like to take zero off that corner because it is against a stop and the solid jaw of the vise. So, you can make your part with zero located where the print has it (that way you can take numbers straight off the print), then you can run the macro before you post your code.

Remember, to turn on one of these macros, go to **Options > Add-ins** and check the box beside the desired macro. An icon for the macro will appear on your interface.

QUESTIONS/COMMENTS?

Please send any tips/tricks, feedback (regarding this newsletter) or requests to be added/removed from our distribution list to...

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