



PTC Product Focus: A) [Tolerance Analysis in Wildfire 4.0](#)

B) [Windchill Export Utility : Exporting Data to Excel](#)

Tips of the Month: A) [Quick Start Tips for Pro/NC](#)

B) [Using the “Import From Spreadsheet” Action in PDMLink](#)

Announcements: [Most Recent Announcements](#)

Upcoming Events & Training Schedule: [Events & Training Schedule](#)

PTC Product Focus

Tolerance Analysis in Wildfire 4.0

Creating features with too-tight tolerances might seem like a good way to ensure a proper fit, but the results are often unnecessarily high production costs. There are very good third-party tools available to perform the necessary analysis, but these tools are often expensive and difficult to use. Don't you wish there was an easy way to determine which parts and dimensions were major contributors to a stackup? And don't you also wish there was a fast way to run “what-if” scenarios without needing to perform a new analysis from scratch?

The new Tolerance Analysis Extension in Pro/ENGINEER Wildfire 4.0 allows you to perform 1-D tolerance stackup analysis, associative to the Pro/ENGINEER dimensions and position GTOLs. Powered by Sigmetrix's CE/TOL technology, this tool is powerful and fast, yet very easy to use. It is intended for general use by all engineers and designers, regardless of Six Sigma “belt” status.

Tolerance Analysis

Tolerance Analysis Extension provides both worst-case analysis and true statistical variation analysis.

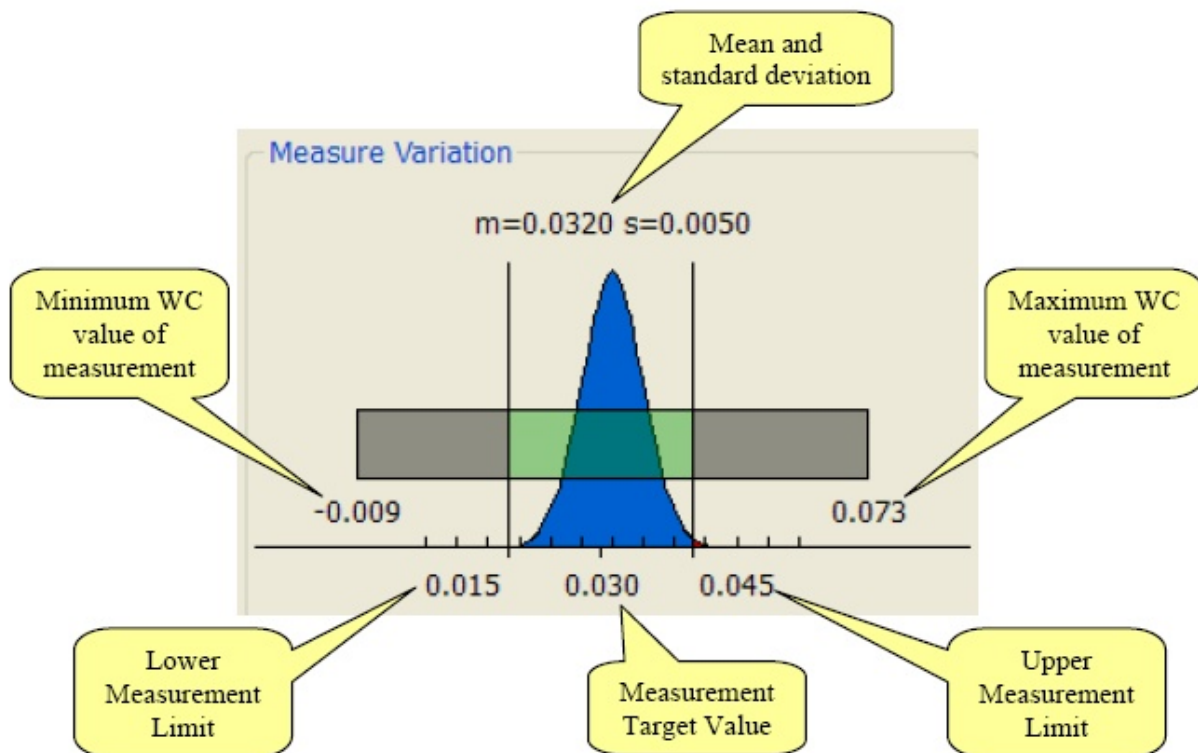
Worst-case tolerance analysis is the traditional mode of tolerance stackup calculation. Dimensions are set to their tolerance limits to make the stackup measurement as large or as small as possible. The worst-case model does not consider the statistical distribution of the individual dimensions, but rather that those dimensions do not exceed their respective specified tolerance limits. This model therefore predicts the maximum expected variation of the measurement.

Designing to worst-case tolerance requirements guarantees that all of the parts will assemble and function properly, regardless of the actual component dimensional variation and stackup combination. The major drawback is that the worst-case model often requires very tight

individual component tolerances. The obvious result is expensive manufacturing and inspection processes and/or high scrap rates.

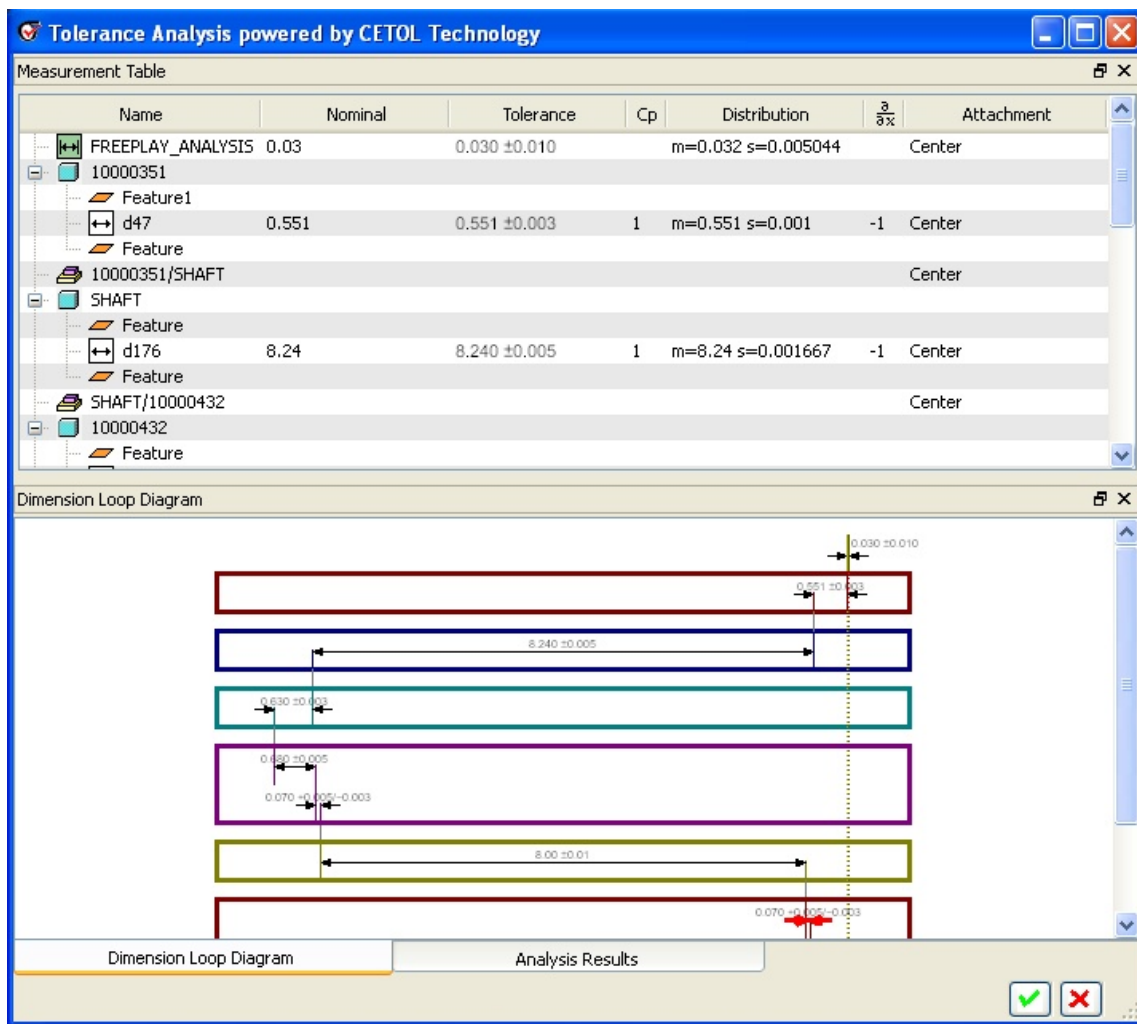
The statistical variation analysis model uses the principles of statistics to relax the component tolerances without sacrificing quality. Each component's variation is modeled as a statistical distribution and these distributions are summed to predict the distribution of the assembly measurement.

Thus, statistical variation analysis predicts a distribution that describes the assembly measurement variation, not the extreme values of that variation. This analysis model provides increased design flexibility by allowing the designer to design to any quality level, not just 100 percent.



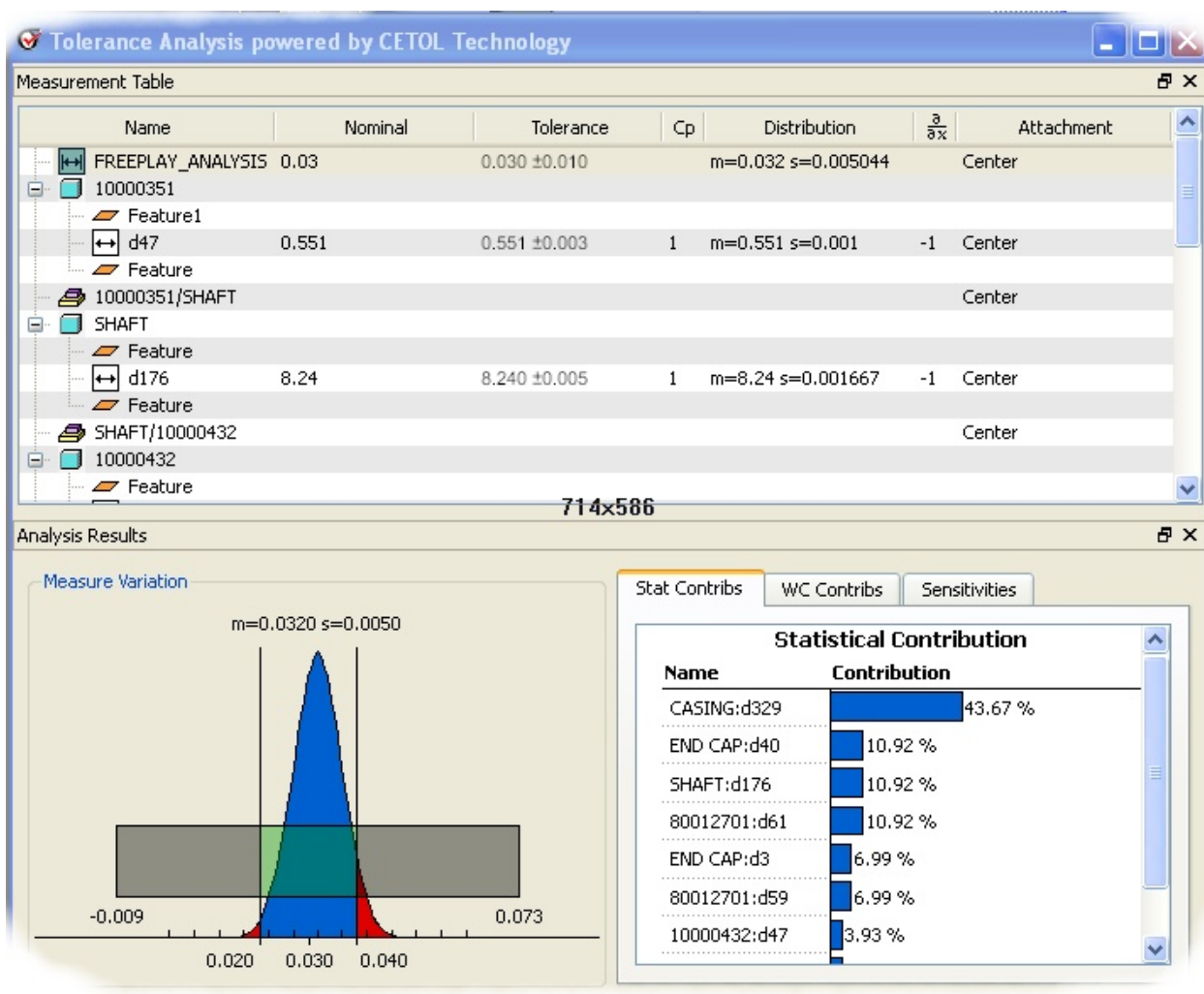
A Tolerance Analysis Tool that is Easy to Use

To create an analysis, one must simply add a new "Tolerance Study" from the Analysis menu. The interface then guides the user to select the features of interest, and the selection of dimensions, annotations, and GTOLs progresses until a closed loop is formed. Each step of the setup may be redefined at any time. Parts may be renamed for ease of understanding the report, without affecting the Pro/ENGINEER model.



Reporting and What-if Studies

The report is generated automatically. Both the worst-case analysis and statistical distributions are displayed in a single report. Best of all, the user may tweak individual tolerances to play “what-if” scenarios, optimizing the design for the best quality and lowest manufacturing cost. When the analysis is saved, any tolerance changes are made in the Pro/ENGINEER model!



This is one more reason to take a good look at Pro/ENGINEER Wildfire 4.0 today!

[Back To Top](#)

PTC Product Focus

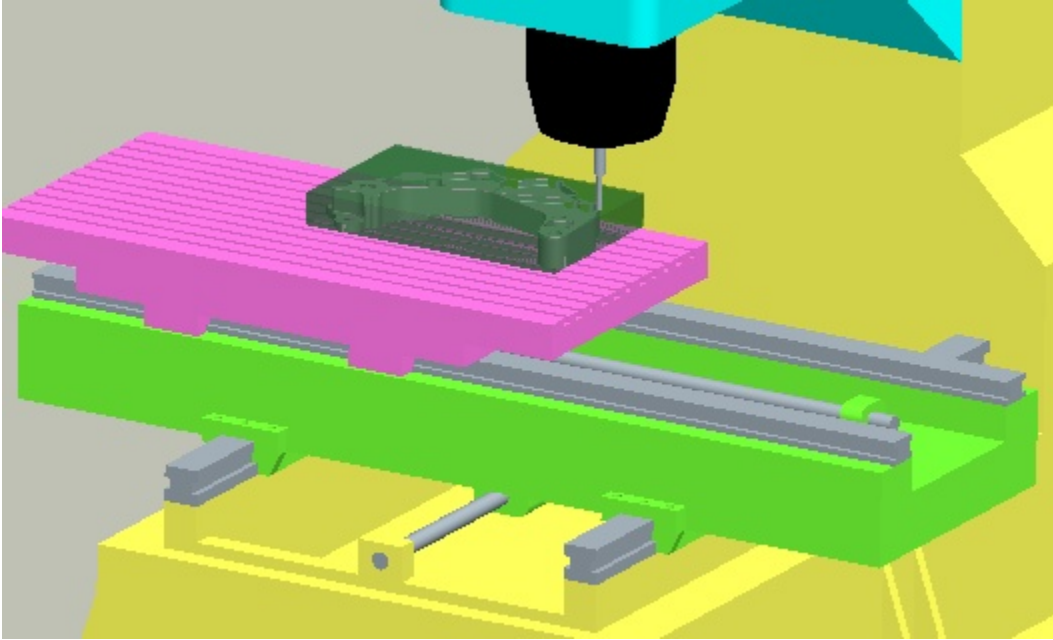
Windchill Export Utility : Exporting Data to Excel

[Click Here To View](#)

[Back To Top](#)

Tips of the Month

Quick Start Tips for Pro/NC



Many people know that PTC's Pro/NC (or Pro/MANUFACTURING, as some call it) is one of the most powerful and capable CAM systems available. Unfortunately, many users fail to take a little time to set it up properly, and therefore spend too long creating programs because they are forever starting from scratch. This article will review several important Pro/NC setup tips that will help get you making chips in no time at all.

Tip #1: Important config.pro options

Set up your Pro/NC environment with these config.pro options:

pro_mf_workcell_dir (path to where workcells are stored)
pro_mf_tprm_dir (path to where tool parameters are stored)
pro_mf_param_dir (path to machining parameters and site files)
postpp_dir (path to post processor library)
pro_library_dir (path to standard libraries, including Tooling library)
start_model_dir (path to the location of start models and drawings)
search_path (path to any folders that contain required models such as fixtures and solid tools)
cl_arrow_scale (0 means no arrows show up on the tool path – helps when spinning the model with a tool path displayed)
rename_drawings_with_object (a value of “both” will automatically create drawings when creating a new mfg model; think automatic setup sheets)

Tip #2: Use site files for default parameter settings

Are you tired of turning the coolant on for every NC sequence because the default is “off”? Use a site file to specify your own default parameter settings.

To create a site file:

Pick **Mfg Setup > Param Setup > Site > Create > enter a name > select a type**. Then set the parameters as desired, and save it (it is saved as a .sit file in pro_mf_param_dir).

To use a site file:

Method 1: Mfg Setup > Param Setup > Site > Create (Retrieve) > Activate > choose site.

Method 2: In the Machine Tool Setup (workcell) dialog box, pick the **Defaults** button and choose a site file to activate for that workcell.

Method 3: During NC sequence creation, select **Site** from the Mfg Params menu, and select which site file to use.

Tip #3: Save and reuse your workcells



Don't create that same 3-axis mill workcell named “MACH01” and populate it with cutting tools every time you start a new program – retrieve it instead!

To save the workcell after you've created it, pick **File > Save** in the Machine Tool Setup dialog box. The workcell is saved as a *.gph file in the pro_mf_workcell_dir folder.

Tip: When you create the workcell, selecting “Defaults” allows you to select a default site file to use with this workcell. The active associated site file will be saved in the workcell.

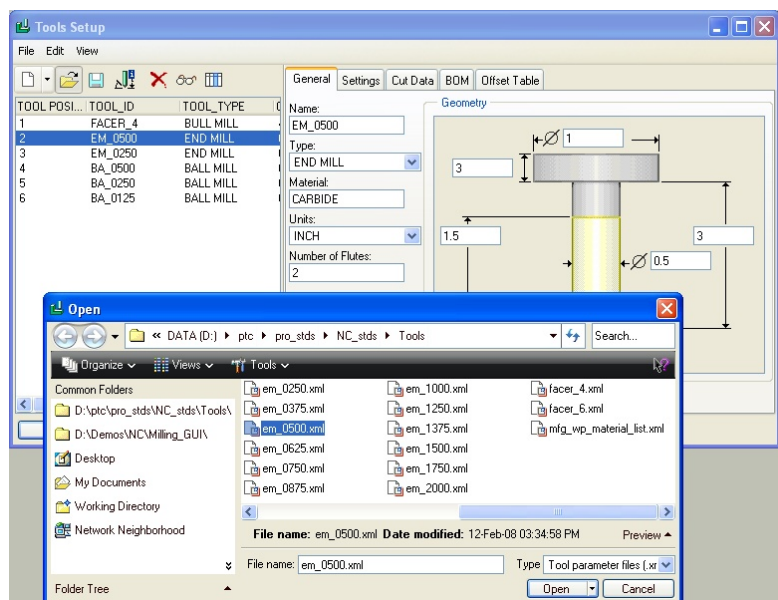
Tip: It is also a good idea to create or retrieve any standard cutting tools that are always loaded in the machine (more on cutting tools in tip #4). Any cutting tools defined with the workcell remain with it when it is saved.

Tip: Add PPRINT settings as desired; these too are saved in the workcell.

Tip #4: Tools

There once was a programmer who would create a new half-inch end mill for every new program. Don't be “that guy.” Spend a few minutes or an hour to create a cutting tool library, and then use it.

To add a tool to your library:



Open the Tools Setup window by picking **Mfg Setup > Tooling > select a workcell**. (*Tip: you must first create a workcell – even a “dummy” one – before creating cutting tools.*)

Name each tool, and set parameters, comments, etc. as needed. (*Tip: you may also use **File > Open Tool Library** to retrieve solid tool parts or assemblies.*)

To associate material cutting data with the tool, see Tip #5.

Save the tool (parameter tools are saved into pro_mf_tprm_dir). Saved tools may be retrieved at any time into other workcells.

To retrieve a saved tool, open the Tools Setup window and select **File > Open Parameter File**.

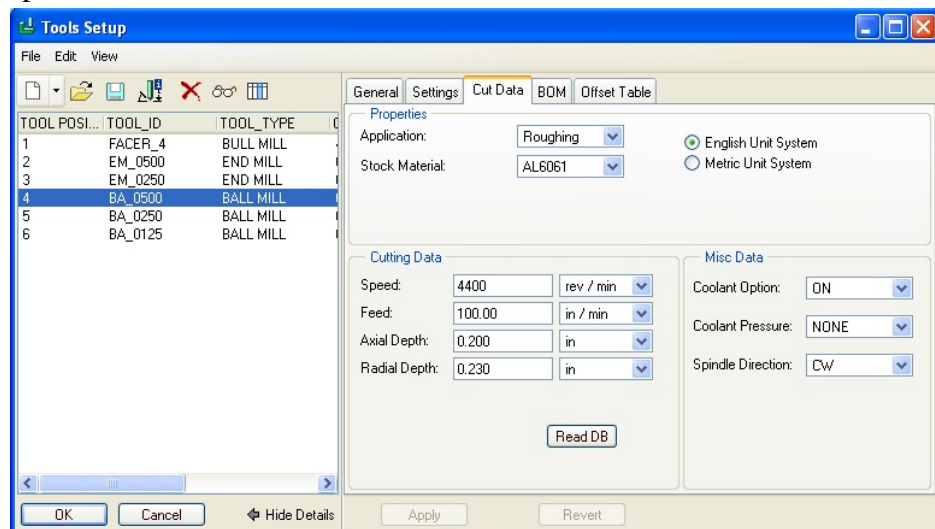
Tip #5: Associating machining parameters with cutting tools and workpiece materials

Capture and reuse your machining knowledge by storing material cutting data with your cutting tools. Here's how:

Define your materials by creating a folder structure:

- Start Windows Explorer and navigate to your pro_mf_tprm_dir.
 - In that folder, create a folder called “materials”
 - In the materials folder, create a folder for each material type.
 - Add the cutting data to each tool for each material and cutting condition
- Open the Tool Manager in Pro/NC
 - a. Create or retrieve the tool and open the Speeds & Feeds tab
 - b. Select a stock material (your materials will be in the pulldown list).
 - c. Select an Application (Roughing or Finishing).
 - d. Supply speed, feed, axial depth, radial depth data.
 - e. Repeat step c for the other Application, if desired.
 - f. Save the tool.
 - g. Repeat steps a-e for each material, as desired.

Perform all of the above for each tool as needed.



To use these parameters in an NC sequence, use either of the following methods:

(Tip: it helps to first specify the Stock Material in the Operation Setup dialog box.)

- In the Parameter Tree in an NC sequence, select **Edit, Copy From Tool, All/Spindle/Feed/Depth, Roughing/Finishing** -- the speed/feed/cut depth parameters for the current stock material will be read from the tool file.
- Or use relations in the parameters to call out the cutting data. The parameter names for use in relations are shown in the table that follows.

The use of relations applies to individual parameter files, and also to site files, which makes this a very powerful technique. For example, if in your active site file, STEP_DEPTH is set to TOOL_ROUGH_AXIAL_DEPTH, then the STEP_DEPTH will always be set for you to the proper value (assuming that the tool contains that information for the material).

Cutting Data	Roughing	Finishing
Speed (rpm)	TOOL_ROUGH_SPINDLE_RPM	TOOL_FINISH_SPINDLE_RPM
Speed (length/min.)	TOOL_ROUGH_SURFACE_SPEED	TOOL_FINISH_SURFACE_SPEED
Feed (/min.)	TOOL_ROUGH_FEED_RATE	TOOL_FINISH_FEED_RATE
Feed (/tooth, e.g. ipr)	TOOL_ROUGH_FEED_PER_UNIT	TOOL_FINISH_FEED_PER_UNIT
Axial Depth	TOOL_ROUGH_AXIAL_DEPTH	TOOL_FINISH_AXIAL_DEPTH
Radial Depth	TOOL_ROUGH_RADIAL_DEPTH	TOOL_FINISH_RADIAL_DEPTH

Tip #6: Tie it all together into a manufacturing template

The fastest way to begin a new job is to use a manufacturing template (available in Wildfire 2.0 and up). You can create and use any number of template models, also known as start models. You may, for example, have a different template for each machine, or even for each material or product.

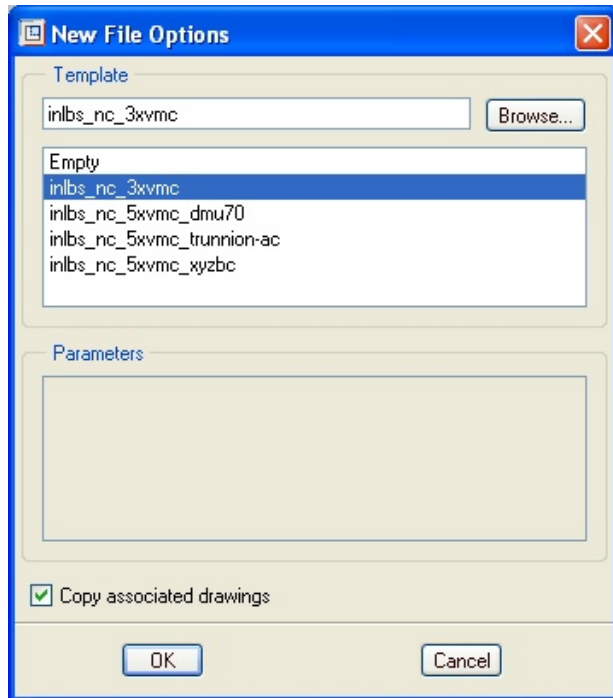
Here's how to create a template. Only the first and last steps are required. However, the more items you include in your template, the greater the benefit.

1. Create a new manufacturing model, and give it a name such as "inlbs_nc_3x_haasvf8").
(Tip: use the default mfg start model to save yourself step 3 below.)
2. Set the units (**Setup > Units**)
3. Create a default coordinate system and datum planes, if necessary.
4. Create layers to hide items such as datum planes (don't forget to "save status"!)
5. Set up named views as desired (for example, an XZ view for a lathe; or top, front, and side views for a mill; or a view for each operation).
6. Create one or more operations, setting the default stock material if desired (see tip #5).
7. Retrieve/create a workcell for each operation. Retrieve/define cutting tools for the workcells if needed.
8. Create/retrieve and activate site files as desired.
9. Make any other settings as needed – basically, anything that will never (or rarely) change, and that you don't want to set up every time. An example here might be a fixture setup such as the table, tombstone, clamps, etc.
10. Create a drawing (e.g., setup sheet or process drawing) of the start model, using the same name as the start model, using the manufacturing .asm as the "default model". Set it up as desired, then save it.

11. Save the manufacturing model; the filename.mfg, and filename.asm will be saved (along with any fixture components that you created). Put these files (and the setup drawing if you made one) into your start_model_dir. It is recommended that these files be made Read Only.

To use the template, select it from the list when creating a new mfg model (you must first uncheck the “use default template” box).. If you have followed tips 1-5 above, then all you need to do is assemble your reference model, create or assemble a stock model if needed, and begin creating NC sequences!

(Tip: if a drawing file (e.g., a setup sheet) with the same name as the mfg template exists, a drawing of your new mfg model will be created at the same time! Set the config option rename_drawings_with_object to “both” in order to do this automatically.)



Bonus Tip: Helpful web pages

Although some of these PTC web pages – available to customers on active maintenance – are updated only infrequently, all of them contain useful information <http://www.ptc.com> :

- [Pro/NC Central](#)
- [GPOST Central](#): -- includes links to post processor downloads.
- [Vericut Central](#)
- [Educational Resource Library](#) (tutorials, demos, tips & techniques):
- [Software Update Downloads](#) (can download Tooling library, for example):

[Back To Top](#)

Tips of the Month

Using the “Import From Spreadsheet” Action in PDMLink

[Click Here To View](#)

[Back To Top](#)

Announcements

Educational Resource Library

Learn things you always wanted to do - but didn't know you could.

This one stop educational resource library will help you learn more about PTC Solutions and provide you with technical materials developed by the product experts to help you become more productive.

Get tutorials, how-to videos and expert advice for:

- Pro/ENGINEER
 - Conceptual and Industrial Design
 - Detailed Design
 - Simulation/Analysis
 - Production
 - Design Collaboration
- Windchill PDMLink
- Windchill ProjectLink
- Pro/INTRALINK
- PTC Online Tools

Check out the [Educational Resource Library](#) today.

PTC Tips & Techniques Newsletter Archives

Miss an issue! Can't find that awesome technique you read about? Fear not, you can click on the link below and go through our Customer PTC E-Newsletter archives.

[Click Here To Access](#)

It's better than finding the Ark of the Covenant!

PTC Tips & Techniques Webcasts: Work Smarter. Not Harder.

Click below to see regularly scheduled Tips & Techniques technical Webcasts that are designed to provide you with the most popular time-saving tricks that Pro/ENGINEER users of all skill levels will find useful. Get more out of your maintenance dollars!

[Tips & Techniques: Work Smarter Not Harder!](#)

E-PROFILES IS HERE!!

We have been eagerly anticipating the debut of the new electronic version of Profiles Magazine and now it is here! This new web site will supplement the print edition of the magazine and will

provide new useful features not feasible with paper media. e-Profiles will provide you with 24x7, worldwide access to key information previously available exclusively in the print version. "Tips & Tricks," a popular feature pioneered by Pro/USER, has also moved to the web and will be expanded as the site matures.

Please take a few minutes to check out this new web site. We don't think you will be disappointed.

<http://profilesmagazine.com/>

[Back To Top](#)

Upcoming Events & Training Class Schedules

Upcoming, 2008 Your local Pro/Engineer User Groups
<http://www.ptcuser.org/rugs/>

June 1 – 4, 2008 Long Beach, CA USA
PTC/USER World Event
<http://www.ptcuser.org/>

Events

Our seminars and conferences seek to provide you with relevant information regarding product development trends in your industry as well as innovative software learning experiences. Think of them as a constructive day off where you can share experiences and swap ideas with your peers.

If you can't manage to get away, we'll bring it to you. Check back often for regularly scheduled live webcast events.

[You're Invited to Attend...](#)

Please visit the [PTC Education Services](#) website for the latest training information including course descriptions, schedules, locations, and pricing.

- Attend a course at any PTC Center and receive a **free** copy of Pro/ENGINEER Wildfire Student Edition!

<http://www.ptc.com/services/edserv/index.htm>

Live Instructor-Lead Virtual PTC Training Courses

Virtual Classrooms provide interactive learning with a trained PTC instructor in convenient and manageable sessions that last approximately 4 hours over a series of days. It's easy to join a class right from your desk using a phone or voice-over IP technology.

Sessions are performed just like a traditional ILT (including interactive exercises where you and the instructor can work on lab exercises together) and feature some of our most popular ILT courses. These sessions cover the exact same material as the traditional ILT in-center courses. Also look for some of our most frequently requested mini-topics delivered in the same format that are only an hour - two hours in duration.

If you have any questions about these sessions or would like to see getting other courses, not on this list, on the schedule please feel free to contact me for more details. They are a great way to bring training to you without you having to worry about location or being out from work for long stretches.

You can register for these sessions just as you would for any normal ILT class either by:

1. calling order admin at <http://www.ptc.com/services/edserv/training/registra.htm> or
2. you can go to PTC University directly at <http://www.ptc.com/learning> and submit a registration request directly. All you have to do is search the catalog by typing in “virtual” in the search field and you will see a listing.

PTC

Note: This PTC E-Newsletter will continue to be used for the following:

- 1) Inform you on events related to PTC products (user groups, conferences, training schedules, etc.)
- 2) Educate you on solutions that are available at PTC
- 3) Tips & Techniques using PTC Products

Note: These messages are compiled in the local PTC office and will be distributed via e-mail.

[Back To Top](#)