

new!

- PTC Product Focus:** A) [What's New in Detail Drawings for Wildfire 3.0](#)
B) [Windchill Supplier Management Solution](#)

new!

- Tips of the Month:** A) [Tricks with Text in Pro/ENGINEER](#)
B) [Windchill PDMLink Folder Level Access Control](#)

Announcements: [Most Recent Announcements](#)

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

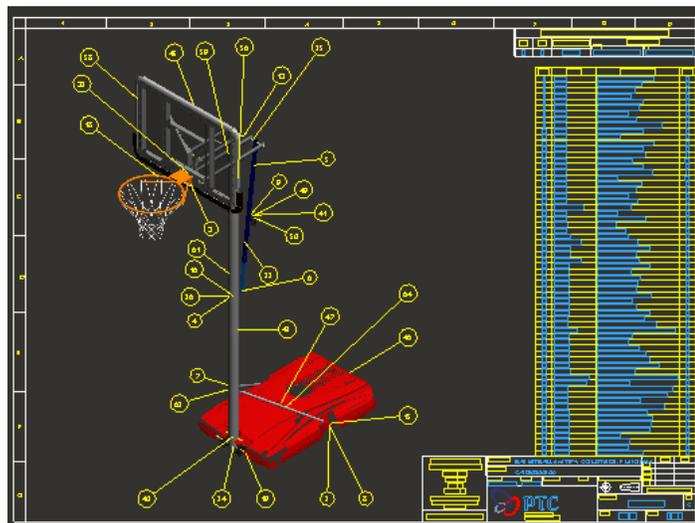
PTC Product Focus

What's New in Detail Drawings for Wildfire 3.0

Wildfire 3.0 brings with it many great new enhancements. This section will highlight some of the most notable new functionality that has been added to detailing/drawing mode. Feel free to download the latest copy of Pro/ENGINEER and try these things out for yourself.

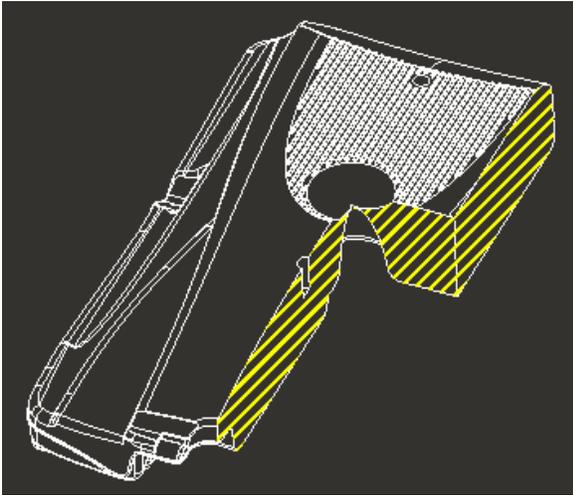
Shaded Views in Drawings

Possibly the most noticeable change in the drawing mode, Wildfire 3.0 now allows users to include shaded views of models in drawings. Simply edit the view properties and set the view to "Shaded". Using shaded views in drawings gives more options for communicating critical design information. Colors can provide visual cues to help describe designs. By using shading, drawings can include more detail. Updated plotter drivers support plotting drawings with shaded views and improve plotting of drawings with embedded objects.



Create Snap Lines Offset 2D Entities

Now you can create snap lines that are offset from draft entities in drawings. This provides the ability to create a snap line based upon a user sketched line to help maintain the position of draft and model annotations.



3D Section Display in Drawings

In Pro/ENGINEER Wildfire 3.0, drawing views can display 3D sections (zones) from models. This allows even better re-use of 3D functionality in drawings.

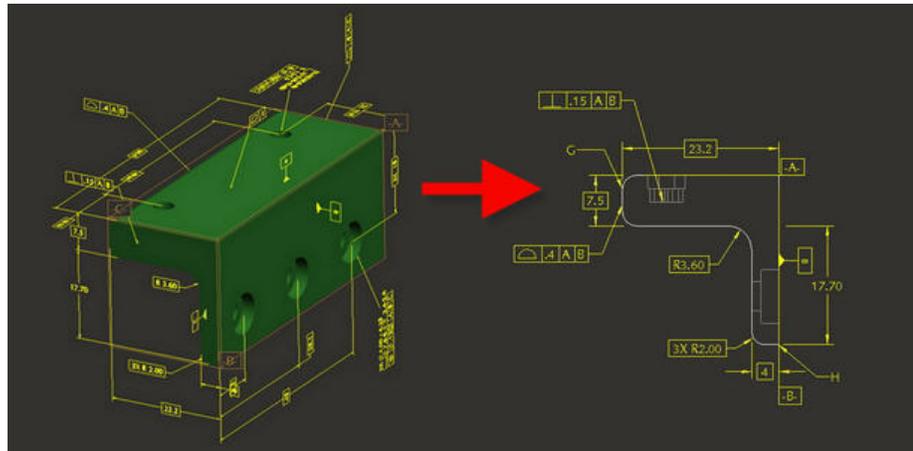
Activate Layer

Many of our users use layers in drawings to manage their display as well as to facilitate drawing creation and modification. We've created a better, faster way to add items to layers. Simply select the layer in the layer tree, RMB > Activate.

From that point forward, all newly created items will automatically be added to that layer. Activating a different layer will de-activate any currently active layer enabling detailers to quickly switch to a different layer.

Automatic Annotation Display in Drawings

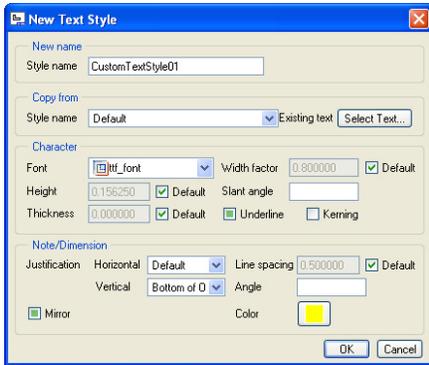
Maximize the value of detailing your 3D models. When you create a drawing view of a model that has 3D detailing items that use an annotation plane, those annotations are automatically shown. Specifically the items that will be shown are those that have the proper orientation and viewing direction. In this way the drawing view matches the annotation plane.



Automatic Clipped Dimensions

We've added a new way to create clipped dimensions in Pro/ENGINEER Wildfire 3.0. This type of dimension calls out twice the measured value between the first entity and the centerline. Automatic clipped dimensioning helps to reduce the amount of time it takes to dimension partial and half drawing views where only one of the references required for the dimension and the centerline is available.





Detailing 'On-Item' Note Position Enhancements

When creating custom text styles users are now able to and define both the vertical and horizontal justification for the style. They can even define the style of text to be mirrored. The resulting note boundary box will be tight against the text string, providing additional control on its exact position in the drawing.

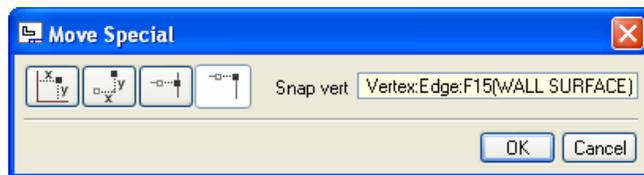
Additionally, users will be able to allow font kerning in the text style allowing Pro/ENGINEER to interpret the kerning values incorporated into the font characters. This is controlled by a new option check box in the text style dialog or note properties dialog.

Drawing Template Improvements

Drawing templates now support 3D sections and combination states, but a major functionality improvement that the view tool now provides advanced scaling options. You can drag the view boundary during placement to define the space on the drawing sheet the view should occupy. When the template is used, the model is automatically scaled to fit inside the space defined by the template view boundary. This prevents users from needing a variety of differently scaled templates for different size models.

Exact Placement Updates

We've added a new button to the Move Special dialog to enable easier access to relative coordinates. Move views symbols and other drawing entities cleanly and exactly at absolute drawing coordinates or at the vertex of a selected object.



Export Drawing Tables as CSV Files

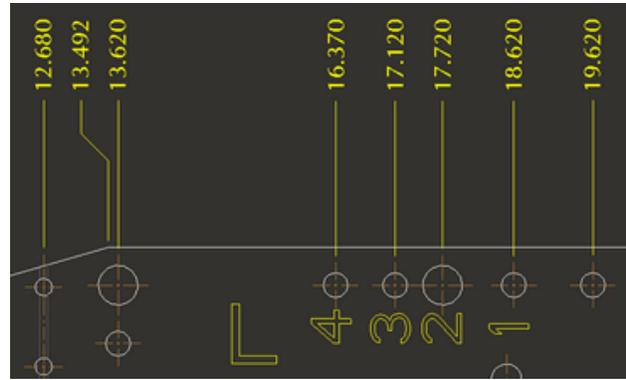
Many of our customers use Microsoft Excel or other spreadsheet programs along with Pro/ENGINEER as part of their normal processes. To enable better information re-use, we've added a new "Save As" option that's compatible with most spreadsheet programs providing the tabular information as a CSV file.

'On Item' Text Placement Improvements

Many detailers use the 'On Item' text placement option to position notes parametrically on model geometry. The most common attachment entity is a model datum point. Now users can use 3 horizontal and 3 vertical alignment options to achieve the desired text position. Horizontal alignment options include Left, Center, and Right. Vertical alignment options include Top, Middle, and Bottom. When these options are use in conjunction with stored fonts, company logos, etc can be exactly positioned as needed.

Improved Ordinate Dimensioning

Ordinate dimensioning has been greatly improved. With the direct creation of ordinate dimensions, you no longer must create linear dimensions that share a common reference and toggle them to ordinate. Faster dimensioning in drawings is the result. You have two creation methods to choose from:

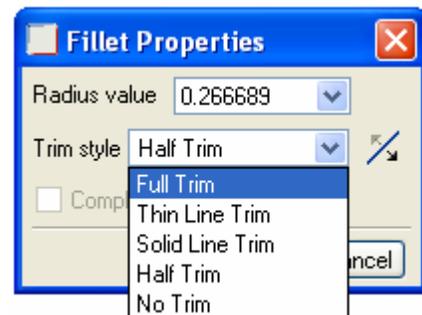


1. Select the baseline, select the items to be dimensioned, then place all of the dimensions at once.
2. Select the baseline, select an attachment point, then place the dimension, select another attachment point, place the dimension and so on.

In addition, you can add a new dimension to an existing ordinate dimension group by inserting the new dimension and referencing any dimension already in an ordinate dimension group. If an existing ordinate dimension is no longer valid in a modified drawing, you can now edit its attachment instead of re-creating it. Ordinate dimension groups can also be deleted by selecting only the baseline and deleting it.

Parametric Draft Fillets and Chamfers

Users can now create parametric draft fillets and chamfers within drawing mode. This means users will be able to add fillet or chamfer draft entities referencing model edges and Pro/E will automatically trim the edges in parametric views. There is a sketcher preference that allows model edges to be automatically hidden and various trim options for chamfers. Since these draft entities are parametric to the model geometry, they will update automatically when changes are made to the model. We've also added a new fillet type - 3 tangent. This allows users to create fillets that are tangent to 3 edges. This new functionality reduces the amount of time it takes to make revision changes to drawings where chamfers and fillets have been added as drawing documentation.

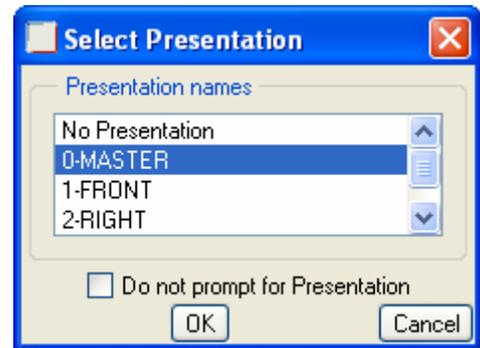


Part Simplified Representations in Drawings

In response to an overwhelming number of enhancement requests, we've added support for part simplified reps in drawing views. When a drawing is created using a part model with simplified representations, the user will be prompted to select the rep to add as the active drawing model. To place a view of a part simplified rep in a drawing, the user must set the desired rep as the active model before placing the view. A drawing view of a part simplified rep cannot be changed to a different rep after definition. This functionality eliminates the need for a "dummy" assembly or family table instances to create a drawing view representing a simplified version of a part.

View Manager States for Drawing Views

You can reuse all states created in 3D models to help configure drawing views. The View Manager provides for faster creation of drawing views with a greater degree of design reuse. You define orientation, explode state, cross sections, and so forth and store them in the 3D model using the View Manager. While creating or modifying drawing views, you can select a presentation in a model. Many of the configured options are automatically used to help define the drawing view when you choose the Presentation State command.



More Information...

With the above items, we've only scratched the surface. There are many more enhancements for you to discover and use. To find out more about Pro/ENGINEER Wildfire 3.0 visit PTC's Website.

[Pro/ENGINEER Wildfire 3.0 Resource Center](#)

[Back To Top](#)

PTC Product Focus

Windchill Supplier Management Solution

[Click Here To View](#)

[Back To Top](#)

Tricks with Text in Pro/ENGINEER

Creating a Box to Enclose Text, Characters, or Symbols.

A drawing note can be created that encloses any desired text, characters, or symbols in a box attached to the leader line.

Typing “@[“ in front and “@]” following the text, characters, or symbols while entering or editing the note will surround the item with a box. For example, if a note were created with the text: “This note includes a @[box@]”

The result would look like this:

This note includes a box

See how only the word “box” is included in the rectangle as it’s 3 characters were in between the “@[“ and the “@]”. Now you can easily box any text on a drawing.

Modifying the Text Height of a Single Character Within One Note

The text height of a single word or character within one note can be modified manually. This process will allow breaking the note into different pieces so the text height of specific characters can be changed. However, the position of the note as a single entity can still be moved. Select the note and use the right mouse button or a double-click to access the note’s Properties dialog box.

Using the “Text” tab, modify certain characters on a single line to act as their own notes. The syntax is the following:

```
{0:This is line one.}
```

```
{1:This is line two.}
```

```
{2:This is line three.}
```

To change the text height of the "T's" in "This" do the following.

```
{0:T}{1:his is line one.}
```

```
{2:T}{3:his is line two.}
```

```
{4:T}{5:his is line three.}
```

Now select “OK” in the Note Properties dialog box and select any of the individual text sections you just created. (You may need to select twice as the first selection grabs the note as a whole). Using the Right Mouse Button access the Text Style dialog box for each section and change the

style as much as you like. Notice that you can still move the note as one item and access the whole note's properties but selecting it with a single click.

[Back To Top](#)

Tips of the Month

Windchill PDMLink Folder Level Access Control

[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

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

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, 2006 Your local Pro/Engineer User Groups
<http://www.ptcuser.org/rugs/>

June 4 - 7, 2006 Dallas, Texas
PTC/USER International Conference
<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.

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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](#)