



new!

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new!

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## PTC Product Focus

### What's New in Sketcher for Wildfire 3.0

With Pro/Engineer Wildfire 3.0 PTC has taken ease of use to the next level with several enhancements to Sketcher. With this release it is easier to set up and enter sketcher, manage your sketch plane and reference plane before or after you enter sketcher, and easier to manage the references that are used for auto-dimensioning. Below you will find information and descriptions for the following enhancements:

- Cut, Copy, and Paste in Sketcher.
- Locked/Modified Dimensions.
- Sketcher Orientation Reference.
- Sketch Setup.
- Sketched Text.
- Sketcher Palette.

### Cut, Copy and Paste in Sketcher

You can use the cut and copy operations to remove or duplicate, respectively, a portion of a section or the complete section. You can cut or copy regular sketch geometry, construction geometry, centerlines, and strong dimensions and constraints associated with the selected geometry entities. The sketch entities that you cut or copy are placed on the clipboard.

You can use the paste operation to place the cut or copied entities at a required location within an active section. When you perform the paste operation, the sketch geometry on the clipboard is not removed, allowing you to use the copied or cut sketch geometry multiple times. You can also use the cut, copy, and paste operations to move the contents of a

section across multiple sections. Additionally, you can translate, rotate, or scale the sketch geometry entities that you paste.

You can undo or redo a cut or paste operation only in the current sketcher session.

## Locked/Modified Dimensions

The locked state of the dimension is maintained when you quit or reenter the sketcher mode. Locking of dimensions allows you to avoid modifications made to the sections outside the sketcher mode by accidentally dragging a vertex or an entity.

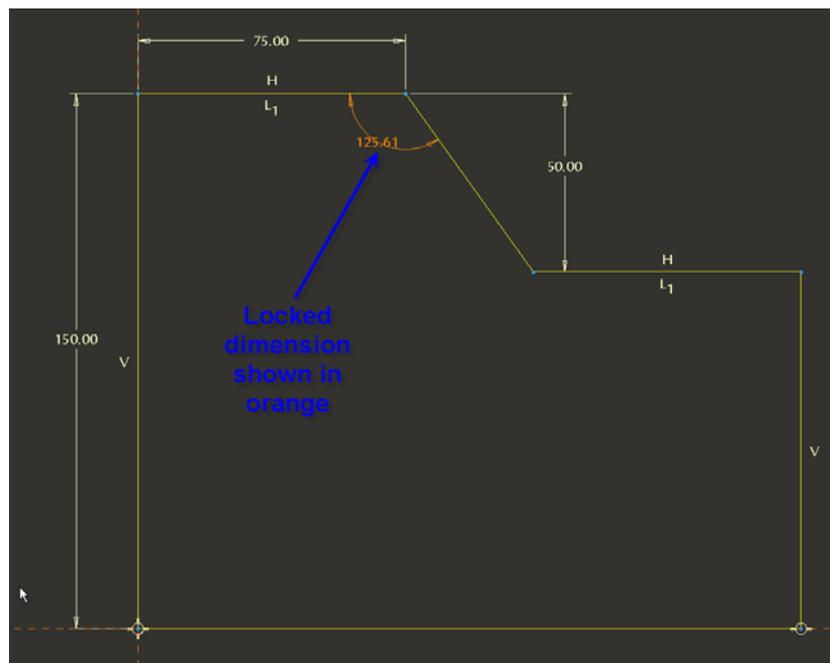
To Lock or Unlock a dimension:

1. Select the dimension that you want to lock.
2. Click **Edit > Toggle Lock**. Pro/ENGINEER locks the selected dimension or click **Lock** on the shortcut menu using the **right mouse button**.

The locked dimensions appear in orange, that is, the default color for locked dimensions. Alternatively, set the color for locked dimensions by clicking **View > Display Settings > System Colors** and selecting the required color on the **Sketcher** tabbed page.

To unlock the selected dimension, click **Edit > Toggle Lock** or click **Unlock** on the shortcut menu.

You can also automatically lock an user-defined dimension by setting the value of the `sketcher_dimension_autolock` configuration option to `yes`. Alternatively, click **Sketch > Options** and select the **Lock User Defined Dimensions** option on the **Miscellaneous** tabbed page of the **Sketcher Preferences** dialog box.



## Sketcher Orientation Reference

When you sketch a section in Sketcher, the orientation reference determines the orientation

of the sketch in the 2D environment. The orientation reference is used to define the horizontal and vertical constraints in the sketcher environment.

When you select a sketching plane, the datum plane that is normal to the sketching plane is automatically selected as the default orientation reference. If such a datum plane does not exist, then the default orientation reference is based on an existing plane or the model default coordinate system.

**Note:** The sketch orientation is based on the model default coordinate system, regardless of the existence of a coordinate system feature.

If you select a sketching plane and the orientation reference is not selected automatically, then the sketching plane is oriented by projecting the X-axis of the default coordinate system onto the sketching plane. If the X-axis is normal to the sketching plane, then the Y-axis is projected onto the sketching plane. The projection of the axis on the sketching plane is in the horizontal direction and points to the right.

After you accept this automatically created orientation reference and proceed with sketch creation, you are prompted to select the dimensioning references irrespective of the value of the `sketcher_auto_create_refs` configuration option.

If you are not satisfied with the automatically created orientation reference, you can change it by selecting a valid orientation reference.

When you click **Info > Feature** to display information for a feature that is created using the default orientation reference, `Default` appears against **Reference** while no value is displayed for **Orientation**.

## Sketch Setup

**At anytime in the sketcher mode** you can modify the sketch setup by selecting a new sketching plane or the orientation reference, or both. Whenever you select a new sketching plane or orientation reference, the active section is regenerated. The sketcher geometry appears with the new orientation settings. Upon regeneration, a message in the message pane indicates reference failures, if any, that you must resolve.

When modifying the sketch setup, you can **undo or redo** the sketch setup operation to restore the previous or subsequent sketch setup, respectively. When you perform the undo or redo operations, the active section is regenerated and the sketcher geometry is reoriented accordingly.

When modifying the sketch setup, you can create a new datum plane that you can use as the new sketching plane or orientation reference.

When you change only the sketching plane, the existing orientation reference is retained if it is normal to the newly selected sketching plane. Otherwise, the existing orientation reference is replaced with the default orientation reference, that is, an existing plane or based on the model default coordinate system.

**Note:** The sketch orientation is based on the model default coordinate system, regardless of the existence of a coordinate system feature.

The integrity of the sketch is maintained when you modify the orientation or the orientation references. Modifying the orientation or the orientation reference is similar to a rigid rotation of the sketch geometry entities along with the model geometry. To maintain the integrity of the sketch, the constraints and dimensions behave in the following ways:

- All strong and weak constraints are preserved, except the horizontal, vertical, horizontal align, and vertical align constraint.
- Horizontal and vertical constraints are replaced with appropriate parallel and orthogonal constraints with respect to the original orientation reference. If the original orientation reference is not a dimensioning reference, then the orientation reference is automatically added so that you can add the parallel and orthogonal constraints.
- All strong or weak horizontal align and vertical align constraints are automatically deleted and replaced with weak dimensions or weak constraints or both by the sketcher solver.
- All strong or weak dimensions, except for the point to point dimensions in the horizontal and vertical directions are preserved and retain their values.
- The point to point dimensions in the horizontal and vertical directions are preserved, but their values are adjusted to reflect the new horizontal and vertical directions.

**Note:** If relations are defined at the top level and involve point to point dimensions in the horizontal, vertical, or both directions, then modifying the orientation settings is not similar to a rigid rotation. Besides a rotation, the dimensions that are used in defining relations change and modify the shape of the sketch. This change of dimensions will happen if the point to point dimension is driving the relation or the dimension is driven by the relation.

If you change the direction of sketch view, then the sketch dimensions and constraints are automatically changed in such a way that the sketch geometry goes through a rigid transformation and does not change its location relative to the model.

If changing the sketching plane causes some external references to fail, a message in the message pane indicates reference failures, if any, that you must resolve.

## Sketched Text

If you want to create text in a sketch it is done using the text option on the sketch menu.

1. Click **Sketch > Text** and select a start point on the sketching plane to set text height and orientation.
2. Click an end point. Sketcher creates a construction line between the start point and the end point. The length of the construction line determines the height of the text, while the angle of the line determines the text orientation. The **Text** dialog box opens. A green arrowhead appears at the start point of the text indicating the direction of the text.

**Note:** By default the dimension associated with the length of the construction line that defines the height of the text, is a strong dimension.

3. Under **Text line**, select **Enter text manually** or **Use parameter**.

**Note:** **Use parameter** is available only in the 3D mode.

- **Enter text manually**—Type a single line of up to 79 characters of text.
- If required, click **Text Symbol** to insert special text symbols. The **Text Symbol** dialog box opens. Select the symbol that you want to insert. The symbol appears in

the **Text line** box and in the graphics area. Click **Close** to close the **Text Symbol** dialog box.

- **Use Parameter**—Allows you to select a parameter that you have defined. The value of the parameter is displayed on the screen.
- Click **Use Parameter**. The **Select Parameter** dialog box opens.
- Under **Look In**, select **Part**, **Feature**, **Annotation Element**, **Surface**, or **Edge** to determine the object type. The name of the object type you select, appears in the **Look In** box.
  - Select a parameter from the **Parameters Table** and click **OK**. The name of the selected parameter appears in the **Text line** box and cannot be modified. The value of the parameter is displayed on the screen.

4. Specify the following under **Font** in the **Text** dialog box.

- **Font**—Select a type face from a list of PTC-supplied fonts and TrueType fonts.
- **Position**—Select a combination of horizontal and vertical positions to place the start point of the text string.

**Horizontal**—Select the horizontal position from **Left**, **Center**, or **Right**. **Left** is the default.

**Vertical**—Select the vertical position to place the dimension and the start point on the construction line that defines the height and orientation of the text string. Select from **Bottom**, **Middle**, or **Top**. **Bottom** is the default.

You can use any combination of the horizontal and vertical positions to snap the start point of the text on to the construction line that defines the height and orientation of the text.

- **Aspect ratio**—Use the slide bar to increase or decrease the aspect ratio of your text.
- **Slant angle**—Use the slide bar to increase or decrease the slant angle of your text.

5. Click **Place along curve** to place the text along a curve and select the curve on which you want to place the text. Select a combination of horizontal and vertical positions to place the start point of the text string along the selected curve. The horizontal position defines the start point on the curve.

**Note:** When specifying the horizontal position for the start point of the text string, you can select the **Center** only if the selected curve is a linear curve.

6. If required, click **Flip** to change the direction in which you want the text to flow.

**Note:** When you click **Flip**, the construction line and the text string are placed at the other end of the opposite side of the selected curve. This placement is based on the start point of the text.

7. Click **Kerning** to enable font kerning for the text string. This controls the space between certain pairs of characters, improving the appearance of the text string. Kerning is a characteristic of the particular font. Alternatively, set the `sketcher_default_font_kerning` configuration option to automatically enable kerning for all the new text strings that you create.

- Click **OK** to create the text. If you have selected **Enter text manually**, then the text is created on the sketching plane and if you have selected **Use parameter**, then the value of the parameter is displayed in the current sketcher window.

**Note:** If you select a parameter that does not have a value, then the graphics area displays \*\*\* (three asterisks) as the value of the parameter.

## Sketcher Palette

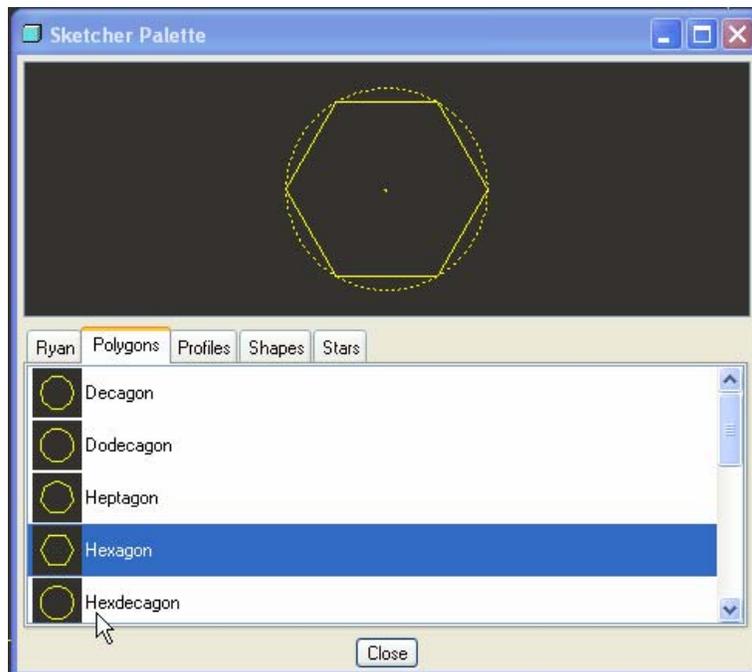
Sketcher provides you with a customizable library of predefined shapes that you can readily import into an active sketch. These shapes are presented in a palette. You can resize, translate, and rotate the shape when using it in an active sketch.

Using a shape from the palette is similar to importing the corresponding section in an active section. Each shape in the palette appears as a thumbnail with the name of the defining section file. These thumbnails appear with the default line style and color for the sketcher geometry.

You can use an existing section created in standalone Sketcher mode or a section created in Part or Assembly mode, to represent the user-defined shape.

The sketcher palette has tabs representing categories of sections. Each tab has a unique name and contains at least one section of a certain category. There are four predefined tabs with predefined shapes:

- Polygon—contains regular polygons
- Stars—contains regular star shapes
- Profiles—contains common profiles
- Shapes—contains miscellaneous common shapes



You can add any number of tabs to the sketcher palette and place any number of shapes within each such defined tab. You can also add or remove shapes from the predefined tabs.

The shape available in the sketcher palette is defined by section file present in the sketcher shape directory. Use the `sketcher_palette_path` configuration option to specify the path to the sketcher shape directory. If you are working in a PDM environment, you can use a PDM location as the path to specify the sketcher shape directory.

**Note:** You must have adequate access rights to the PDM location to access the tabs and shapes.

The label of the tab in the sketcher palette corresponds to the tab subdirectory that you define within the sketcher shape directory.

**Note:** If you do not set the `sketcher_palette_path` configuration option, the current working directory is set as the sketcher shape directory, and only one user-defined tab appears in the sketcher palette. The label for this tab corresponds to the name of the working directory. The section files (files with `.sec` extension) within the working directory appear as the available shapes in the sketcher palette.

If you change the contents of the sketcher shape directory, for example, add or remove a tab subdirectory or shape, then you must reload the sketcher palette to see these changes. To reload the sketcher palette, close and reopen the sketcher palette.

The sketcher palette also has a preview pane. When you select a shape thumbnail in the sketcher palette, all the entities that will be imported into an active section appear in the preview pane. These entities can be sketcher geometry, construction geometry, internal strong dimensions, and constraints.

When you use a shape from the sketcher palette in the current sketch, the following entities of the section that defines the shape are imported:

- Geometry of the section
- Construction geometry of the section
- Strong constraints internal to the section (other than the vertical and horizontal constraints)
- Strong dimensions internal to the section
- Section relations

**Note:** The locked state of the dimensions of the shape is retained when the shape is imported into an active section.

## Add a Tab to the Sketcher Palette

Create a subdirectory within the sketcher shape directory. The subdirectory name appears as a tab label in the sketcher palette.

**Note:** The subdirectory name appears as a tab label in the sketcher palette only if the subdirectory contains at least one section file, that is, a file with a `.sec` extension.

## Add a Sketch to the Palette

1. Create a new section or retrieve an existing section.
2. Save the section to the subdirectory corresponding to the sketcher palette tab within the sketcher shape directory. The file name of the section appears as the name of the shape along with the thumbnail of the shape within the sketcher palette tab.

## To Use a Shape from the Sketcher Palette

1. In the Sketcher mode, click **Sketch > Data from File > Palette**. The **Sketcher Palette** dialog box opens.

**Note:** Alternatively, click  on the **Sketcher** toolbar.

2. Select the required tab on the sketcher palette. The thumbnail and the label corresponding to the shapes available within the selected tab appear.
3. Click the thumbnail or the label corresponding to the required shape. The section corresponding to the selected shape appears in the preview pane.
4. Double-click the same thumbnail or the label again to import the selected shape into the active section.

The pointer changes to include a plus sign (+) indicating that you must select a location to place the selected shape.

5. Place the selected shape into the active section in one of the following ways:

### By retaining the default size of the selected shape

- a. Click anywhere in the graphics window to select a location to place the shape.

The shape with its default size, that is, one-fourth the graphics window, is placed at the selected location with the center of the shape coinciding with the selected location. The entities that define the shape remain selected.

The **Scale Rotate** dialog box opens. Additionally, the Scale, Rotate, and Move handles appear on the imported shape. The Move handle coincides with the selected location.

- b. Scale, rotate, or move the shape as required.

### By resizing the selected shape

- a. Click and hold the left mouse button to specify the location of the shape. The imported shape appears in a very small size at the location you select.
- b. Drag the handle until the required size of the shape is achieved.
- c. Release the left mouse button to confirm the size of the shape.
- d. You can further scale and rotate the shape by specifying respective values in the **Scale Rotate** dialog box.
- e. Scale, rotate, or move the shape as required.

**Note:** By default, the imported shape does not snap when you place it. To enable snapping, move the Move handle on to the imported shape. To move the Move handle, right-click it and drag it to the required location on the imported shape.

6. Middle-click to accept the position, orientation, and the size of the imported shape. The imported dimensions and constraints are created as strong dimensions and constraints.

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## **PTC Product Focus**

### **Flex PLM Attribute Viewing and Updating**

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### Quick Print in Pro/ENGINEER Wildfire 3.0

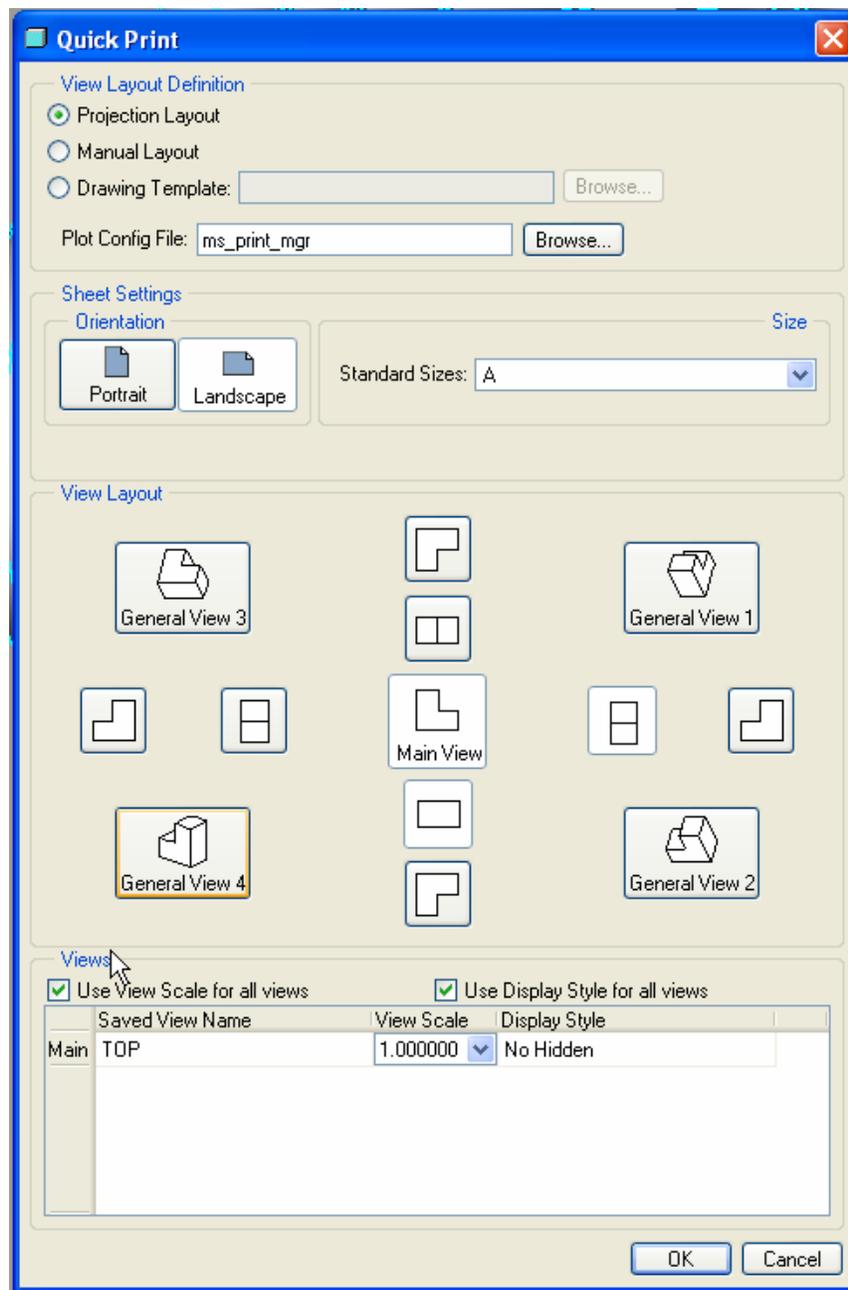
#### Making a Quick Print of a Part or Assembly

While using the **Standard** application (Part or Assembly), use the **Quick Print** command (**File > Quick Print**) to directly plot views for the active model, without manually creating a drawing.

By specifying one of the following layouts and a printer configuration file, you can create the plot:

- **Manual Layout**—Specifies the number of views per sheet, named views and a scale for each named view, and a display style for each view.
- **Projection Layout**—Selects the projection layout based on a named view.
- **Drawing Template**—Uses a template to define all aspects of the layout, except for the printer configuration.

The plot reflects the layer settings, simplified representation state, explode state, and zone clipping for the current model, and appears as if a drawing were created and plotted. The file used to create the plot is automatically erased from your Pro/ENGINEER session.



**Note:**

- In Part, **Quick Print** is not available for simplified representations.
- In Assembly, **Quick Print** is available for geometry and graphics representations but not for symbolic representations.

**Configuring for Direct Plotting**

Use the following configuration options when you set up for direct plotting:

- **quick\_print\_plotter\_config\_file**—Specifies the path for the default [plotter configuration](#) file (.pcf) when using the **Quick Print** command.

If `plotter` is set to `ms_print_mgr`, no other options are necessary. If it isn't set to `ms_print_mgr`, adding to the plotter configuration file the values for `plot_file_dir` and `plotter_command` is recommended.

If you do not set `quick_print_plotter_config_file`, it defaults to `<Pro/E_loadpoint>/text/plot_config/postscript.pcf` on UNIX and `<Pro/E_loadpoint>/text/plot_config/ms_print_mgr.pcf` on Windows.

**Note:** The `.pcf` defined in the **Quick Print** dialog box overrides the configuration option setting.

- **quick\_print\_drawing\_template**—Specifies the path for the default drawing template file when using the **Quick Print** command with **Drawing Template** selected as your **View Layout** definition.
- Some settings in the **Quick Print** dialog box may conflict with settings in the plotter configuration file. For example, plotter configuration file settings for `plot_access` and `allow_file_naming` may be overwritten by settings in **Quick Print**.
- Some settings in the plotter configuration file are ignored by **Quick Print** because they do not apply to **Quick Print**. Settings in the plotter configuration file that are ignored by **Quick Print** include `paper_size` and `paper_size_allowed`. Plotter configuration options that work with **Quick Print** include `plot_label`, `plot_sheets`, `plot_file_dir`, and `plot_segmented`.

## Making a Quick Print of a Part or Assembly Using a Projection Layout

After you click **File > Quick Print**, the **Quick Print** dialog box opens. Follow the procedure below when you select **Projection Layout** as your **View Layout Definition**.

In the **Plot Config File** box, type the path to your plotter configuration `.pcf` file or click **Browse** and select the file.

**Note:** If you have set the configuration option `quick_print_plotter_config_file`, the path to the plotter configuration file is automatically defined. You can override the configuration option by typing a path to or browsing to another `.pcf` file.

Click **Portrait** or **Landscape** to specify the orientation of the sheet, and click the **Standard Sizes** box to select the size of the sheet.

To use the same view scale and display style for all views, click the **Use View Scale for all views** and **Use Display Style for all views** check boxes. Note the **View Scale** and **Display Style** defaults for the **Main View** are listed in the **Views** area of the dialog box.

**Note:** To assign unique view scales and display styles, clear these check boxes.

Click the **Saved View Name** box for the **Main** view and select another name, or keep the default **Standard Orientation** name.

Click in the **View Scale** and **Display Style** boxes to select a new values or style, or accept the defaults for the **Main** view.

In the **View Layout** area click the projection views you want. Projection views determine view layout and positioning.

**Note:** Projection views are the unlabeled boxes in the **View Layout** area.

Click a general view if needed. Each general view is listed in the **Views** area of the dialog box.

- If you selected **Use View Scale for all views** or **Use Display Style for all views**, the values for **View Scale** and **Display Style** initially default to the values for **Main**. If you change a value for **View Scale** or **Display Value** for the **Main View** or any general view, then that value changes for all views listed.
- If you cleared **Use View Scale for all views** and **Use Display Style for all views** check boxes, then you can make changes for **Main** or any general views, without affecting other values.

Click **OK** to plot the part or assembly.

**Note:**

- If the plotter configuration file cannot be found, is missing required values, or is generating plotter errors, then an error dialog box opens after you click **OK**. Depending on the error, you can redirect to the correct plotter configuration file or edit the plotter configuration file from the dialog box.
- If you have defined a model grid, and you are plotting a **Project Layout View** or **Manual Layout View**, the model grid is automatically plotted for all views with the correct orientation. For information on defining a model grid, see Detailed Drawings help.

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## Tips of the Month

### Line Plan “Views” Interface & Updating Data

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## 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.

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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!](#)

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### **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/>

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## Upcoming Events & Training Class Schedules

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

June 3 - 6, 2007                      Tampa, Florida  
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.

## 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.

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