



new! **PTC Product Focus:** **A) [What's New in Interactive Surface Design for Wildfire 2.0](#)**

B) [Windchill DynamicDesignLink at a Glance](#)

new! **Tips of the Week:** **A) [On the Fly Representations in Pro/ENGINEER Wildfire](#)**

B) [Dynamic Documents in Windchill DynamicDesignLink](#)

Announcements: [Most Recent Announcements](#)

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

PTC Product Focus

What's New in Interactive Surface Design for Wildfire 2.0

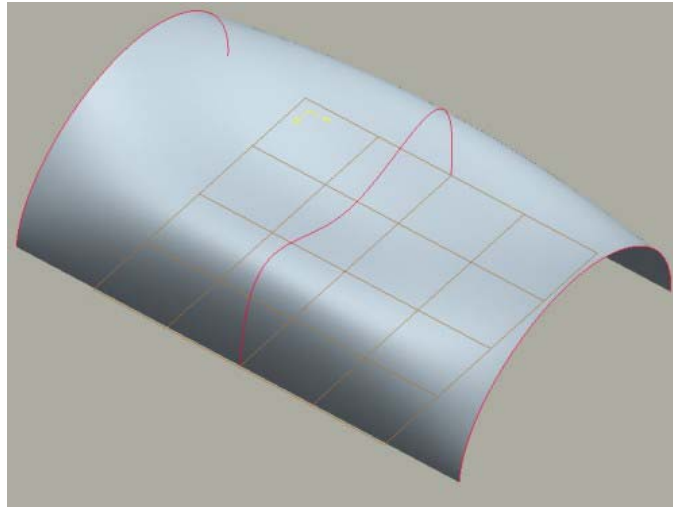
With [Pro/ENGINEER Interactive Surface Design](#) (ISDX), PTC has taken surfacing to the next level of flexibility. ISDX creates a freeform modeling environment inside Pro/ENGINEER to give you more dynamic control in creating surfaces. Instead of having to input dimensions as you do with parametric modeling, you push and pull on curve points with a mouse to get the exact complex shape you are looking for.

ISDX introduces a new superfeature inside Pro/ENGINEER called STYLE, which allows you to build curves and surfaces together in the same feature, with controls for continuity and relationships between all of these elements. This all happens in a dynamic environment so you can see results real-time as you make changes in your design.

With the introduction of Wildfire 2.0, PTC continues to expand on the already powerful capabilities found in ISDX. Read on to learn what's new in ISDX for Wildfire 2.0.

Lofted Surface capability

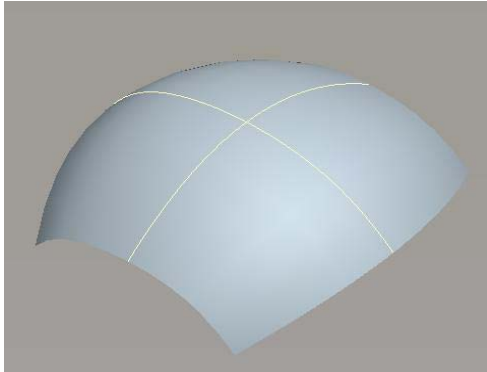
Loft surface allows a Style surface to be created with a number of curves (two or more), defining the shape in one direction and no curves in the secondary direction.



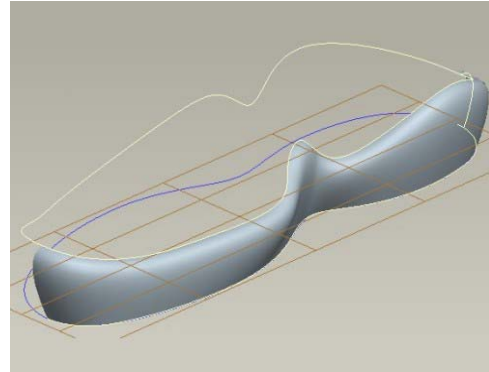
Loft Surface with 3 curves

Blend Surface capability

The Blend surface allows the creation of a surface from one or two guide curves and any number of profiles.



Blend Surface using 1 Guide Curve and 1 Profile Curve

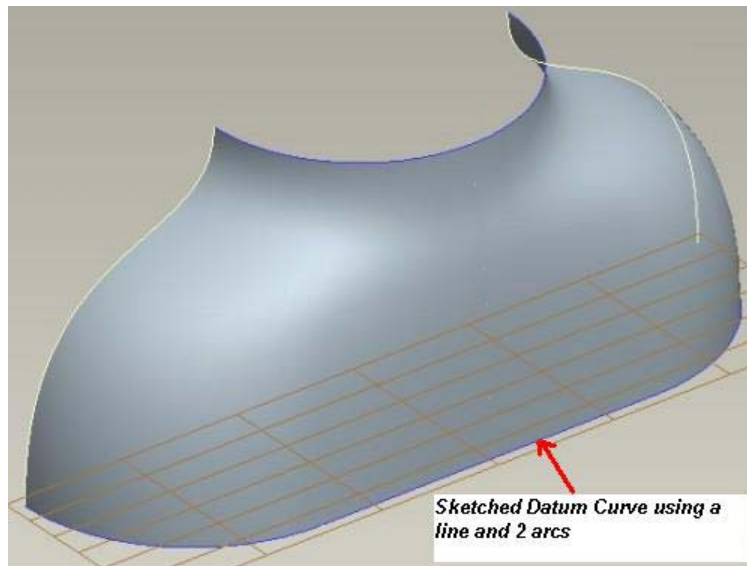


Blend Surface using 2 Guide Curves and 1 Profile Curve

Composite Surface Creation

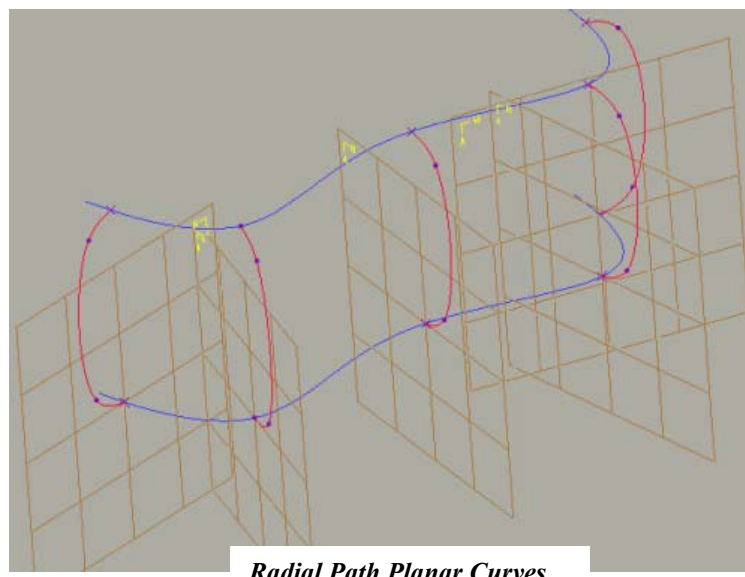
You can now select compound strings of curves to be treated as a single boundary edge during surface creation within Style.

All Style surfaces can use compound curves as boundaries. This ability extends the use of Style surfaces by allowing increased use of sketched curves as well as compound strings of Style curves. The compound curve elements, however, must be at least tangent continuous. To create a surface using a compound curve, hold down SHIFT while selecting the desired curve segments.



Radial Path Planar Curves

An option to build radial-path, planar curves has been added to the Planar Curve tool. Within Style, you can construct a planar curve on a plane that remains normal to the tangent vector of a previously created curve at a specific location to build a radial path. The new plane and curve maintain their history, so that when you modify the path curve the geometry updates. You can also drag the plane along the path curve.



Planar on Endpoints

With the Planar on Endpoints command, you can make 3D curves temporarily planar without constraining them to a particular reference.

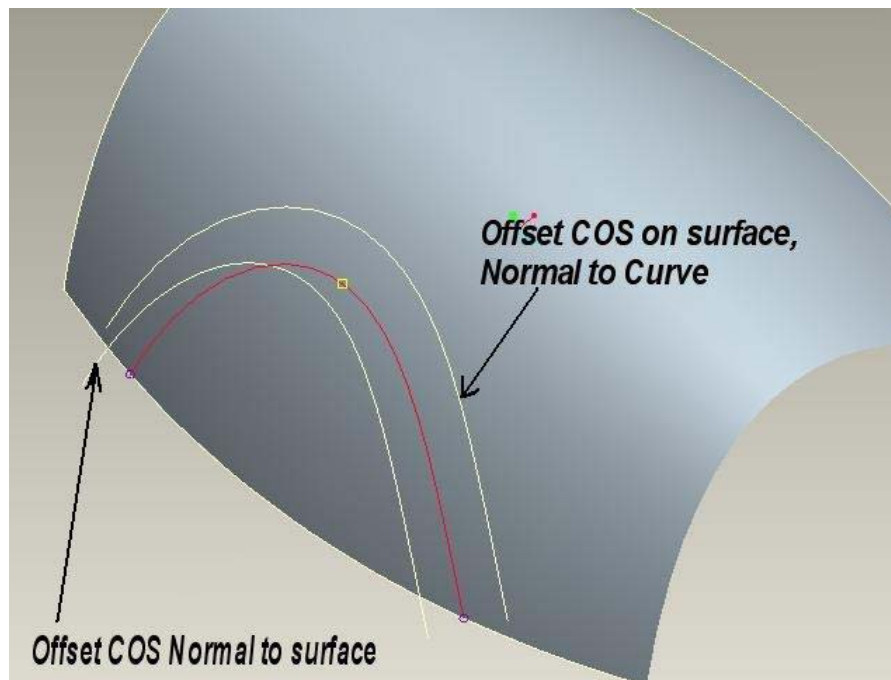
The curve becomes planar between its endpoints normal to the active datum plane. This functionality is only valid for free curves. Free tangents at the endpoints are adjusted if necessary. If any point or tangent constraints exist, the command does not affect the curve.

To access this option; select a curve, select the Edit tool, right-click the curve, choose Planar on Endpoints from the popup menu.

Offset COS (Curve On Surface)

Curve On Surface (COS) curve offsets are now supported inside the Style feature.

COS curves can be offset either on the surface or on the normal to surface, making this a powerful function. Offset curves are history-based and update whenever the original COS is modified and can be redefined to alter their direction or dimension. You can set offset dimensions via the dashboard for use in other areas of Pro/ENGINEER.

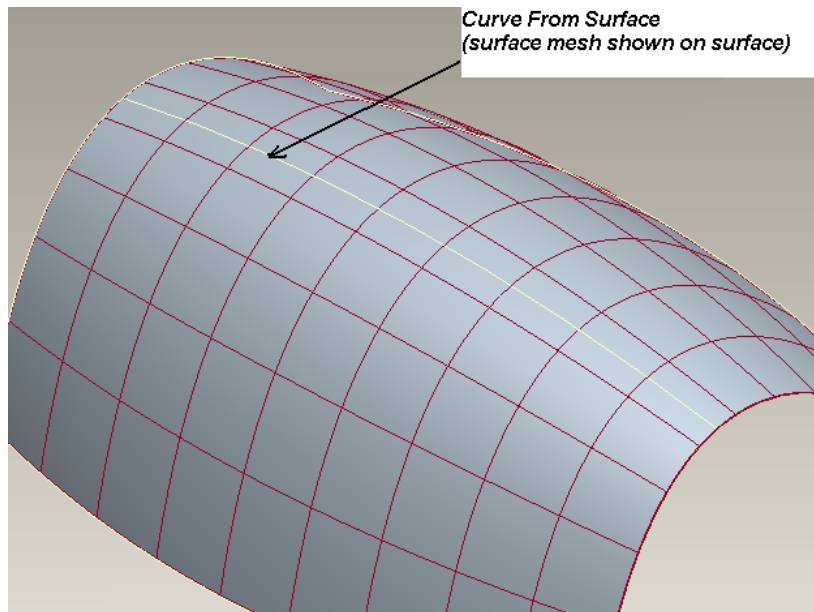


Curve from Surface

With the Curve from Surface command added to the Style module, you can create a free or COS type curve at any point on an existing surface that follows the isoparametric line of the surface.

The **Curve from Surface** command is a tool not based on history with which you can create a free curve or COS that follows the isoparametric line of a surface.

You select Curve from Surface from the **Styling** menu, select the type of curve (free or COS) from the dashboard, and select a point on the surface that the curve will pass through. If you hold down the left mouse button during selection, you can drag the curve along the surface. Hold down CTRL while selecting a point on the surface to “flip” the U- and V-direction for the curve. After you create the curve, you can edit and manipulate it like any other style curve.



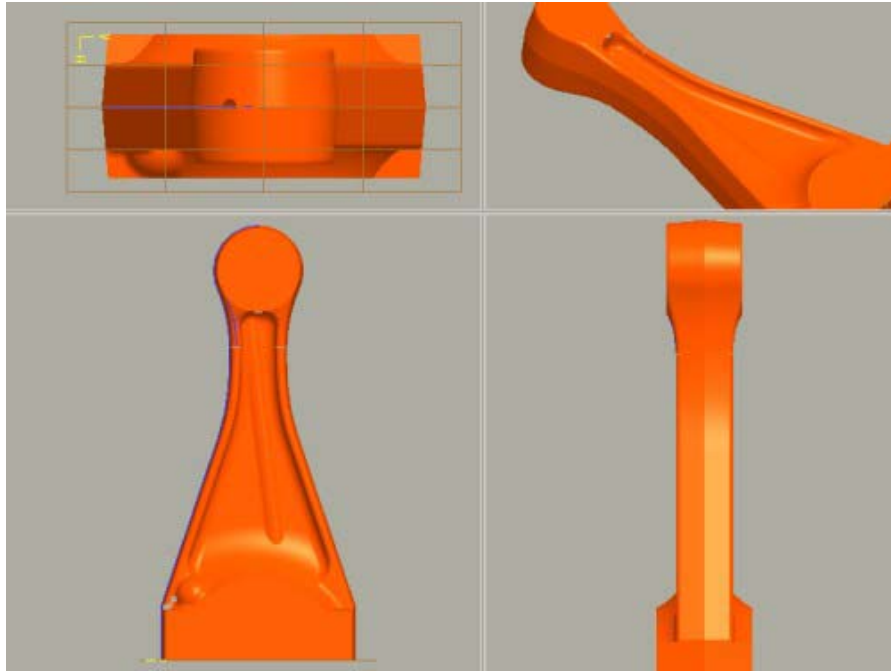
Copy Proportional

An enhancement to the Copy command inside Style allows copied curves to be scaled proportionally.

The **Copy Proportional** command operates on free and planar Style curves in the current feature. During the operation, you have dynamic, visual feedback with respect to the size and shape of the geometry.

4-View Enhancements in Style

In 4-view mode in Style you can dynamically resize four panes, allowing for a more usable work area while designing.



4 view mode with resized window panes

More Information on Interactive Surface Design

To find out more about Pro/ENGINEER's [Interactive Surface Design](#) visit our Website.

[Back To Top](#)

PTC Product Focus

Windchill DynamicDesignLink at a Glance

[Click Here To View](#)

[Back To Top](#)

Tips of the Week

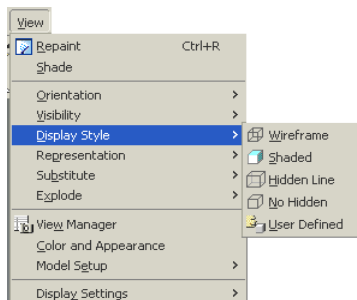
On the Fly Representations in Pro/Engineer Wildfire

Pro/Engineer Wildfire now gives you the ability to create Simplified Reps, Display Styles, and Explode States on the fly using an object/action technique. This means you can build the representation as you work and not have to build the rep ahead of time. If you decide later that you want to save the rep with the model for use later, you can do that also.

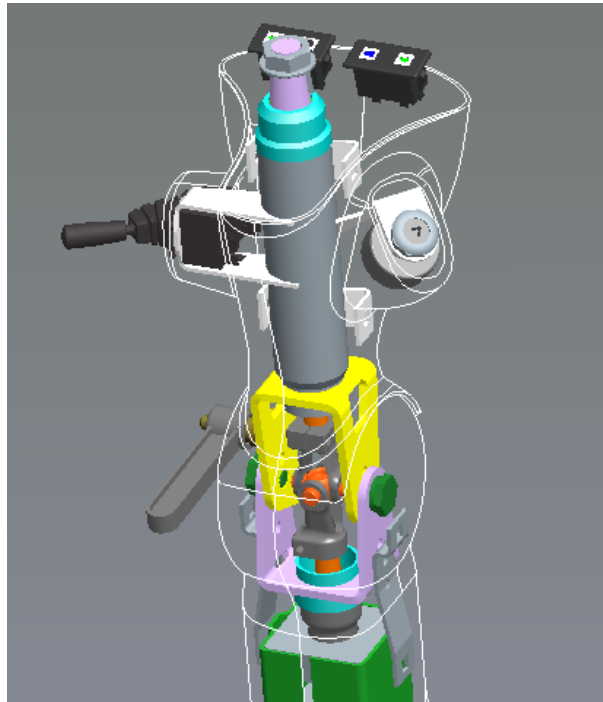
First we'll consider Display Styles. Let's say you're working on an assembly that has some external shields that you would like to view in wireframe display so you can see into your design without completely turning the components off.



Select the components you would like to be shown as wireframe. Use your control key to select multiple objects. Once the objects are selected, choose **View** from the tool bar and then **Display Styles**. From the **Display Styles** fly out menu, select the desired display style to be applied.

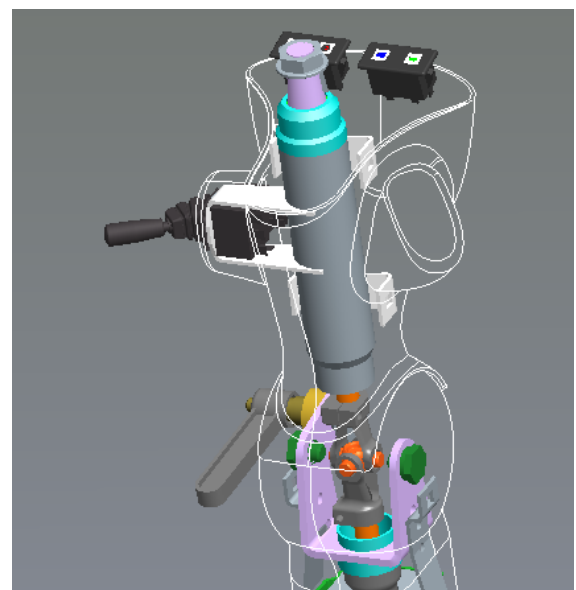
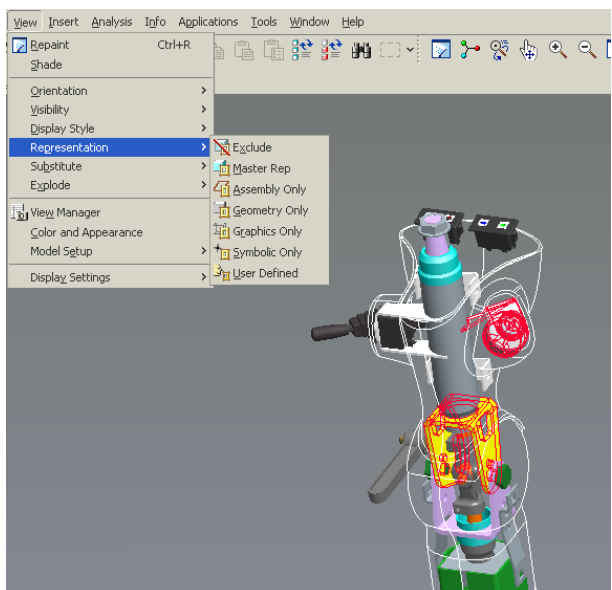


The model should now show the selected components in the wireframe display style as shown below.

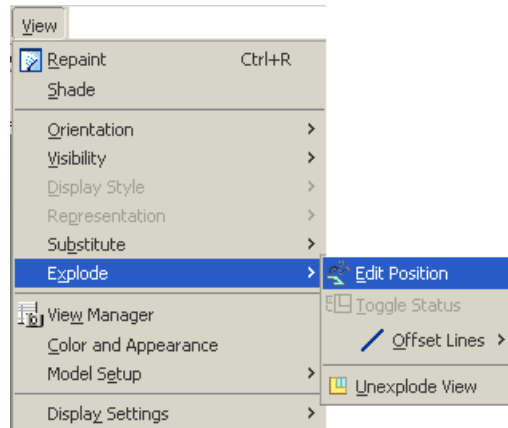


Now that you have the shields switched to be wireframe, let's consider that you may also want to simplify the assembly by excluding some parts. We will simply follow the same steps as above by first selecting the part or parts you would like to turn off. In this case, I'll select the yellow bracket and the ignition switch.

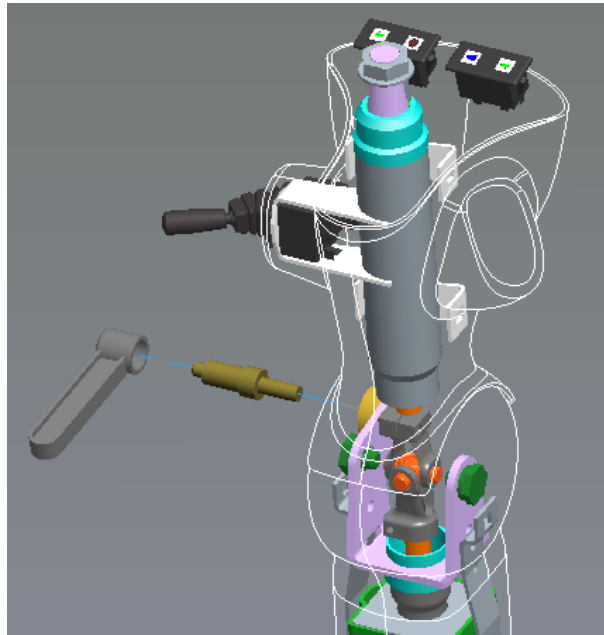
This time we will select **Representation** from the **View** pull down menu and **Exclude** from the **Representation** fly out menu.



The last representation change we will make will be to explode the handle from the assembly. To do this, we don't need to pre-select anything. We will simply choose the **View** pull down menu and select **Explode** and then **Edit Position** from the fly out menu.

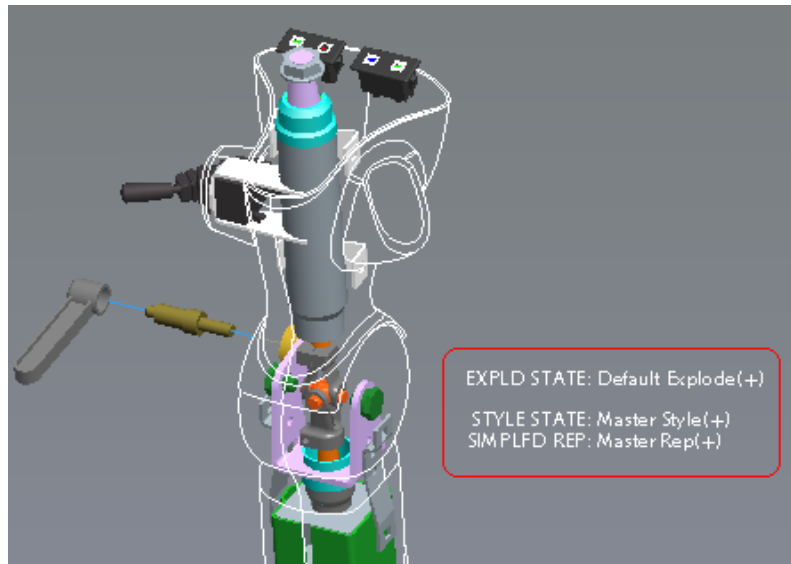


From this point you can explode the components, as you would like. You can also add you offset lines at this time if desired.



So now that everything looks like you want, you can continue working on the design. However, you may also decide that you would like this information saved with your assembly so that you can retrieve it later. Right now, everything has been created “on the fly” and none of the representations have been given names. Let's take a look at how you can save this info for later use.

You may have noticed that each state we have created so far has been temporarily identified by the system placing a (+) next to the representation name in the graphics area as shown below.

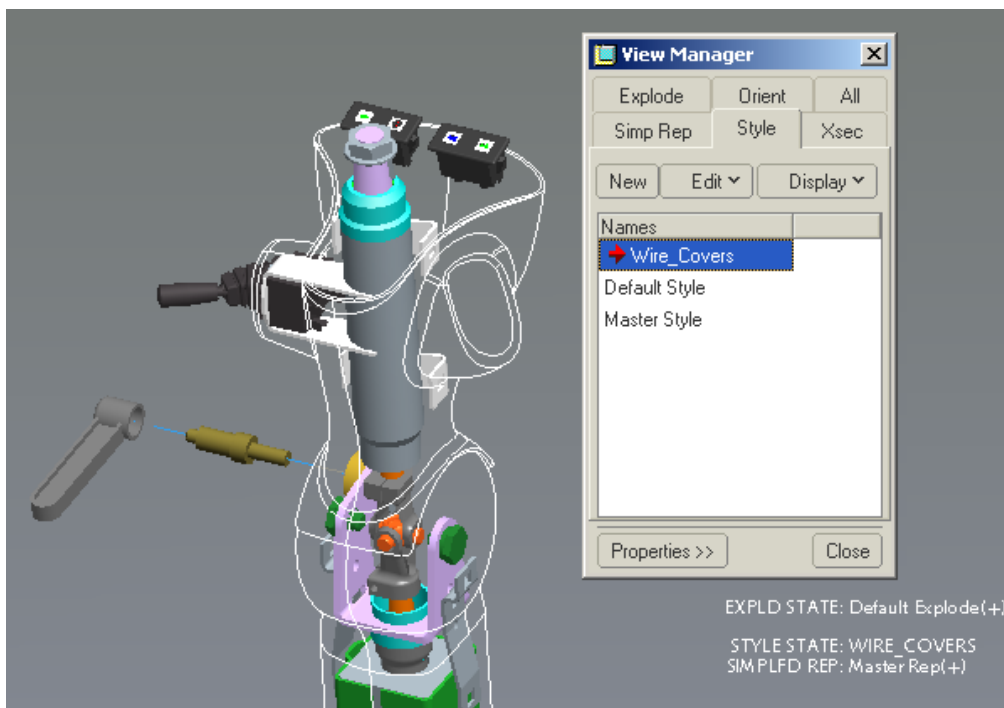


To save the temporary states, we will open the **View Manager**



and continue to work from there.

Again we'll start with the **Style** Tab first. Notice that the Master style has the same (+) sign next to it. To save the current temporary state we will simply create a new Style state. Just select the **New** button and give the state the appropriate name.



Notice that the (+) sign is removed once a new state is created.

That's it; you just need to repeat the naming steps for each of the states you have created. Don't forget, Wildfire now allows you to create a representation that combines all of the options found in the View Manager.

[Back To Top](#)

Tips of the Week

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[Click Here To View](#)

[Back To Top](#)

Announcements

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Hands-On Workshops

Experience and receive the next generation of CAD - Pro/ENGINEER Wildfire. During these workshops you can try it yourself to experience this breakthrough in simple, powerful, and connected in CAD software.

http://www.ptc.com/appserver/it/icm/cda/template_lib/events/online.jsp?im_dbkey=17625&im_language=en

Special Hardware offers for customers updating to Pro/ENGINEER Wildfire

http://www.ptc.com/partners/hardware/current/wildfire_tlo.htm

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Thinking About Pro/ENGINEER Wildfire? Check this out.



<http://www.ptc.com/go/engineering/index.htm>

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. Future plans include several foreign-language editions of Profiles for our many international readers. Currently, Profiles is printed in English and Japanese.

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, 2004 Your local Pro/Engineer User Groups
<http://www.ptcuser.org/rugs/>

June 5 - 8, 2005 Orlando, FL
PTC/USER International Conference
<http://www.ptcuser.org/>

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>

PTC

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- 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](#)