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PTC Product Focus

Pro/ENGINEER ADVANCED ASSEMBLY

Pro/ENGINEER Advanced Assembly expands the power of Pro/ENGINEER-Foundation to encompass the engineering and management of medium to very large assemblies throughout an enterprise-wide product development process. It offers high-powered tools for design criteria management, top-down assembly design, large assembly management, Shrinkwrap, and process planning. These tools enhance the productivity of design teams creating and managing very large, complex product designs – and help downstream users produce accurate lifecycle documentation and assemble these complex products on the shop floor.

Key Product Features

Advanced Assembly Design

Pro/ENGINEER Advanced Assembly helps teams to design and manage large, complex products by providing tools to support top-down design, handle alternate product configurations, and manage even the largest, most complex assemblies. Its tools for top-down design include skeletons to capture and distribute critical aspects of the design, such as space claim and interface information, and reference controls to manage boundaries of design responsibility and maximize design reuse. Its interchange capabilities enable users to design and organize alternate product configurations, quickly and easily. And its assembly simplified representations and Shrinkwrap capabilities can be used to control level of detail, improving performance and visual clarity.

Specifically, advanced assembly design functionality allows designers to:

- **Use a top-down approach to design**
Working with intuitive tools, users can quickly define and populate an assembly structure with new or existing components, collect critical design criteria in skeleton models, and propagate these criteria throughout the assembly with Copy Geometry functions. Users can also manage the top-down design process with Reference Control and Reference Viewing tools.
- **Interchange assemblies**
Users can relate independent, functionally equivalent parts, so these can be easily switched in the context of a particular design – to leverage previously created designs on

new projects. Users can also create simplified interchange members. These can be substituted into a design to simplify the display, while retaining accurate mass property information.

- **Create simplified representations**

Users can create collections of components to facilitate managing a large assembly design. These collections can be based on rules, such as model size, location, name, or any other model parameter. Graphics and Geometry Only Representations can also significantly improve performance by reducing the amount of component information brought into a session.

- **Associative Shrinkwrap capabilities**

Within the Advanced Assembly, users can produce a lightweight representation that retains complete surface and mass property information. This "Shrinkwrapped" representation reduces the load on system resources – greatly improving performance. In addition, Shrinkwrapped designs can be passed to suppliers and customers without distributing the proprietary design information and Shrinkwraps are associative, allowing changes to update automatically.

Design Criteria Management

Pro/ENGINEER Advanced Assembly provides layouts for conveniently documenting, managing, and modifying high-level critical engineering criteria. The layouts present operating specifications in the form of parameters and dimensions, which can then be used to govern the design. These layouts can also be used to automate assembly of components.

Specifically, design criteria management functionality allows designers to:

- **Create graphical 2D assembly layouts**

Users can create annotations on parts and assemblies at the conceptual stage. They can also create block diagrams representing the assembly design – including notes, global parameters, relationships, and global placement information, as well as components, constraints, requirements, and descriptions of how the assembly must fit together.

- **Complete top-down descriptions**

Using the 2D-assembly layout, users can create relationships in the models, linking dimensions and parameters to features and components. This symbolic representation of the assembly can then be disseminated to all members of the design team, ensuring clear communication.

- **Facilitate automatic assembly**

Users can create a placement correspondence among multiple components in the assembly by declaring the same axis or plane in different components in the assembly to the layout. The software then can locate components with common declarations based on this knowledge – to automatically combine them in an assembly.

- **Manage symbolic information in a spreadsheet-style**

Users can create tables to document critical design information. These tables provide a top-level location for manipulating dimensional and parametric information, which, in turn, drives the assembly design.

Process Planning

The process planning functionality widely disseminates process information throughout the engineering and manufacturing organizations. It ensures clear communication of assembly procedures through either traditional process drawings or Web-viewable process plans.

Specifically, process-planning functionality allows designers to:

- **Easily create assembly process sequences**
Working with intuitive tools, users readily define assembly process steps. Drag-and-drop techniques, exploded views, and jogged explode-offset lines provide a clear, complete and accurate representation of each process step.
- **Create alternate bills of material (BOM)**
Users can create BOM's to reflect only those components assembled up to a particular process step (manufacturing BOM's) or to regroup design components based on how they are assembled during the assembly process (fabrication BOM's).

More Information

Follow these links for more information on Pro/ENGINEER's Advanced Assembly Extension

[Advanced Assembly Page on ptc.com](#)

[Webcast Demo Replay](#)

[Top-Down Design Task Guide](#) & [Top-Down Design Task Data Files](#)

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Working with Product Structure Views & Annotations in PDMLink?

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Flatten Quilt Features & Bend Solid Features

Introduction

There are many situations where one would like to flatten a generalized surface. Pro/Engineer provides this capability in a feature called “Flatten Quilt”. The feature can be accessed from the drop-down INSERT menu in version 2001: INSERT / ADVANCED / FLATTEN QUILT. As a compliment to the Flatten Quilt feature, the Bend Solid feature takes the transformation that flattened the surface quilt and applies its reverse to solid geometry – effectively bending the solid back into shape. It can be accessed from INSERT / ADVANCED / BEND SOLID.

Despite their “advanced” labels, the Flatten Quilt and Bend Solid features are part of the basic Pro/Engineer feature set and come with the Foundation package as well as older, basic module packages.

Capabilities and limitations

The Flatten Quilt feature is capable of flattening many, but not all, surfaces and tangent quilts including those with un-developable (compound) curvature. Be aware that this flattening does not produce darts (triangular relief cuts often used for fabric construction) in the geometry but rather it deforms the surface and the surface boundary attempting to maintain the same area and perimeter. Certain geometries provide no room for deformation at the boundaries and therefore cannot be flattened. For example, a circular patch cut out of a sphere cannot be flattened but a rectangular patch cut out of the same sphere can be flattened (because the boundaries of the rectangular patch can be curved to maintain the same perimeter but the circular patch would require an increase in the perimeter).

The Bend Solid feature bends the entire solid part geometry by reversing the process that flattened the quilt. The solid must lie in the vicinity of the flattened quilt and cannot cross its boundary. Despite these limitations, it’s still quite useful.

Additionally, though slightly misnamed, Bend Solid can also flatten datum curves using the same transform that flattened the quilt. To do this, the curve must be related to the quilt (projected, formed, etc.).

Flatten Quilt Feature

For this example, a rectangular cutout of a spherical surface will be used. [Figure 1](#) shows the base surface that will be used for this example. Besides the surface patch, a datum point on the surface is also required. The datum point determines where the flattening iteration will begin and therefore where there will be no deformation.

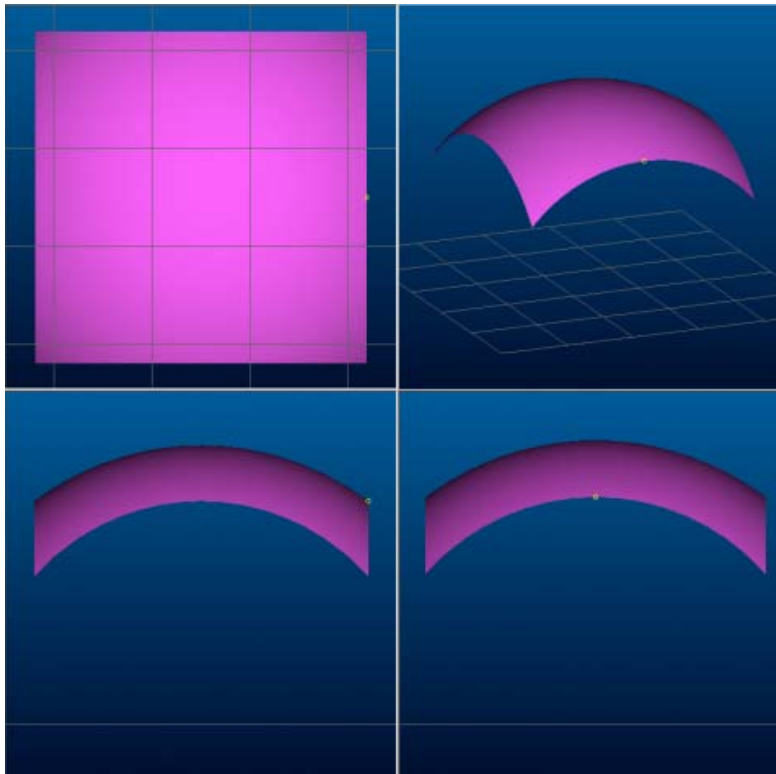


Figure 1 – Original Quilt

INSERT / ADVANCED / FLATTEN QUILT brings up the dialog box for the feature (see [Figure 2](#)). After selecting the Source Quilt and the reference Origin Point, arrows are shown which indicate the u and v directions for the surface “parameterization”. This parameterization breaks the quilt into smaller pieces that are then, iteratively, pieced back together in a flat pattern. A higher number will provide a much closer approximation of the original quilt’s surface area and perimeter length but may fail on some geometry. A lower number will be less accurate but may not fail. The range is 10-100 and the default 30 is a good place to start.

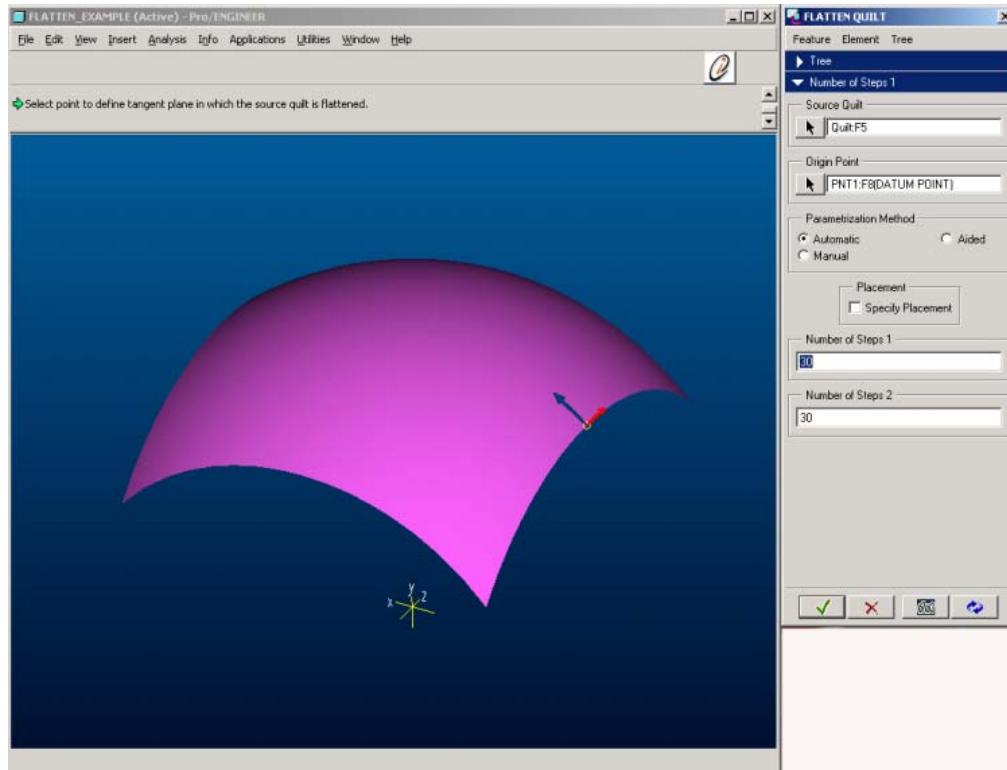


Figure 2 – Flatten Quilt Dialog

The Parameterization Method section of the dialog box is only useful for quilts (not single surfaces). “Automatic” will suffice for most geometries, however, for certain quilts, it might be necessary to approximate the quilt with a single surface. There are two methods to accomplish this: First, the “Aided” parameterization method allows for choosing 4 vertices of the quilt through which Pro/E will place a single surface that approximates the quilt geometry. Second, the “Manual” parameterization method requires a pre-built, approximate, single surface -- it still flattens the quilt, but does so using this approximate surface as a template.

Once the dialog box is completed, the flattened geometry is created. [Figure 3](#) shows the flattened example. Note that the flat surface is created tangent to the original surface at the chosen Origin Point. By changing the Origin Point’s location, a different flattened geometry will be achieved. [Figure 4](#) shows the result of the Origin Point being placed at a corner and [Figure 5](#) shows the result of the Origin Point being placed at the very center.

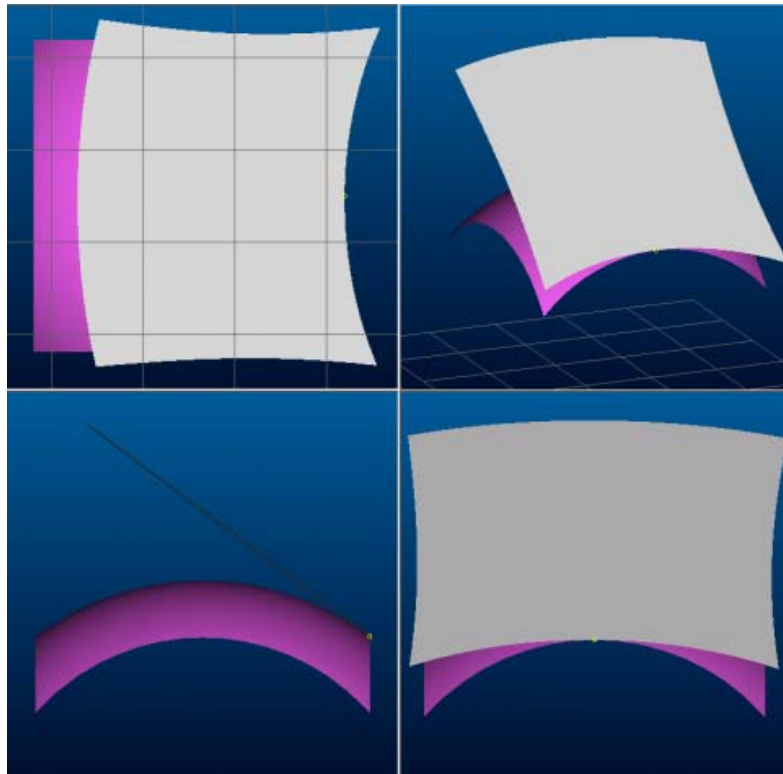


Figure 3 – Flatten Example 1

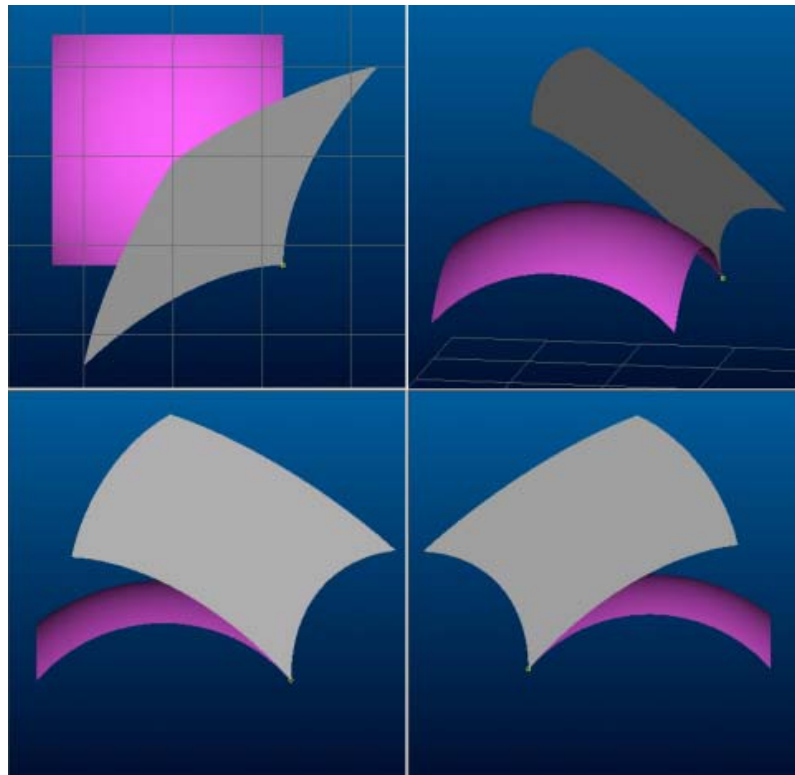


Figure 4 – Flatten Example 2

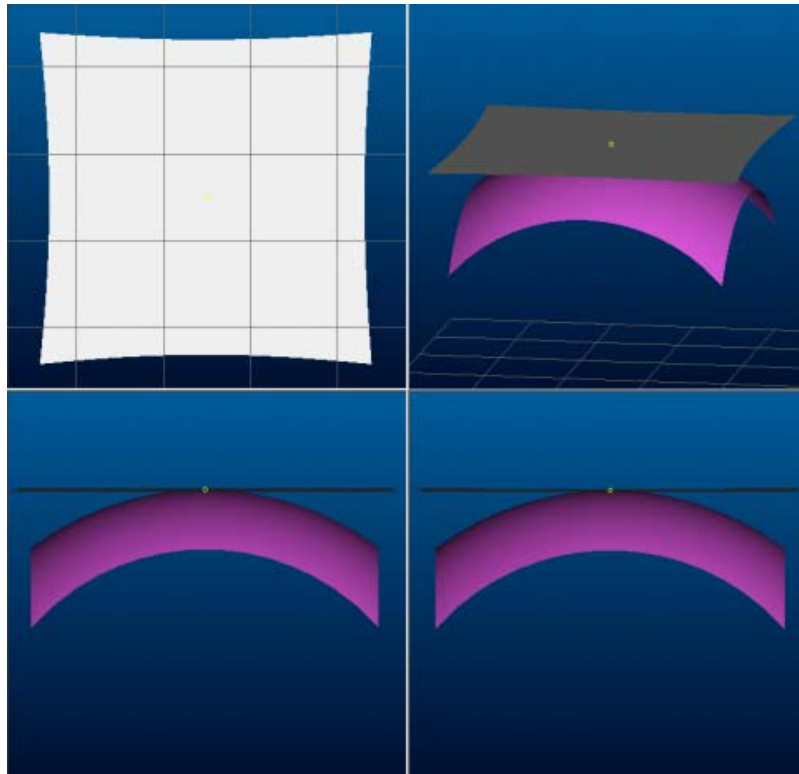


Figure 5 – Flatten Example 3

If it is desired that the flattened geometry be placed elsewhere than at the Origin Point, the “Specify Placement” box can be checked and a coordinate system chosen (See [Figure 6](#)). The flattened quilt will be placed in the X-Y plane with its Origin Point centered on the coordinate system chosen. An additional point, the X Direction Point, on the quilt is also needed to define the rotation of the surface about the Origin Point. A surface vector drawn from the Origin Point to this X Direction Point will be aligned with the coordinate system’s X-axis. [Figure 7](#) shows an example of this. Notice how the X-axis of the coordinate system passes through both the Origin Point and X Direction Point of the flattened quilt.

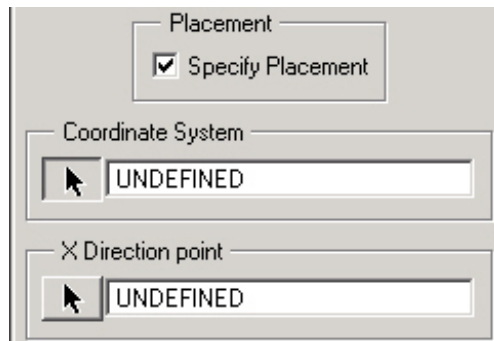


Figure 6 – Placement Dialog

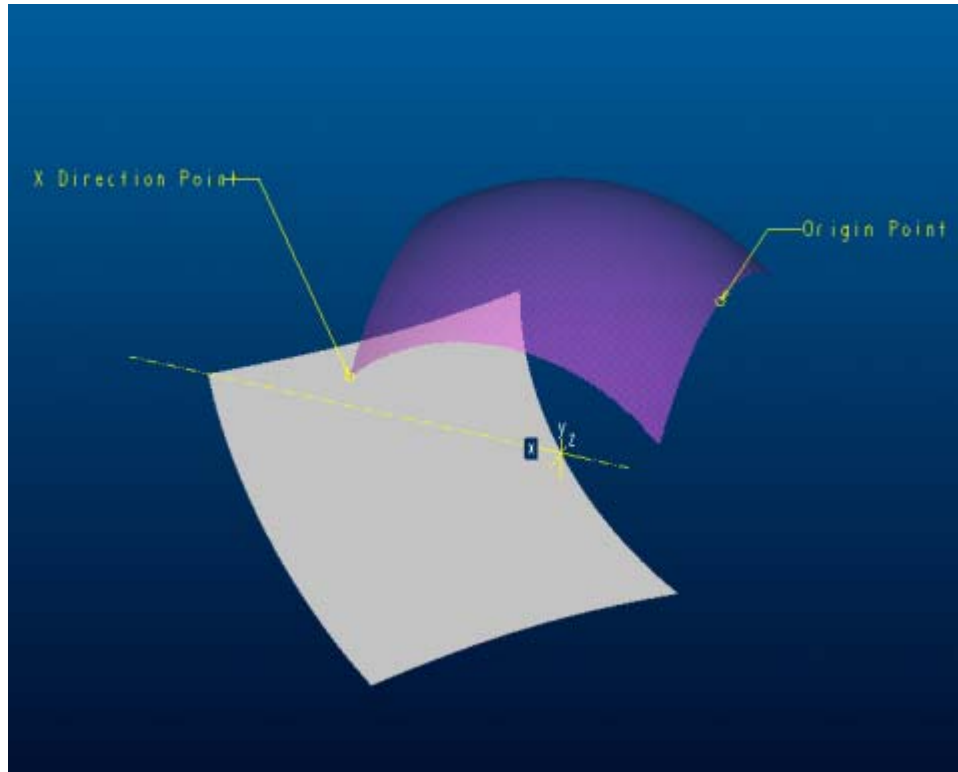


Figure 7 – Example of Specifying Placement

Bend Solid Feature

The Bend Solid feature provides two options, Flatten Curves or Bend Solid. For either option, first select an existing flattened quilt. This provides the feature with the necessary transformation information. See [Figure 8](#)

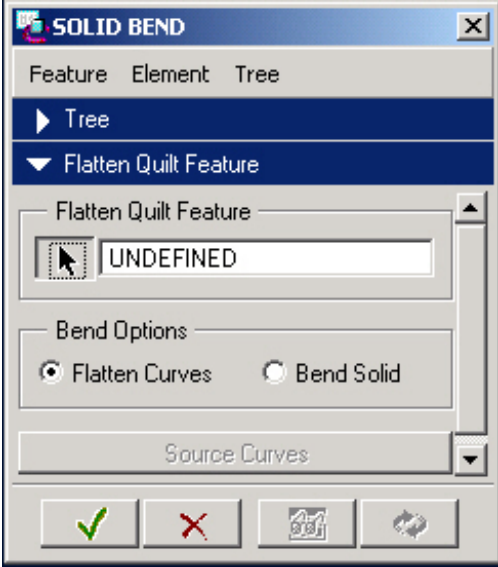


Figure 8 – Bend Solid Dialog Box

For the Flatten Curves case, select curves associated with the bent quilt. [Figure 9](#) shows the result of this feature when applied to a curve formed from the intersection of the S-shaped surface and a plane at a 12-degree angle.

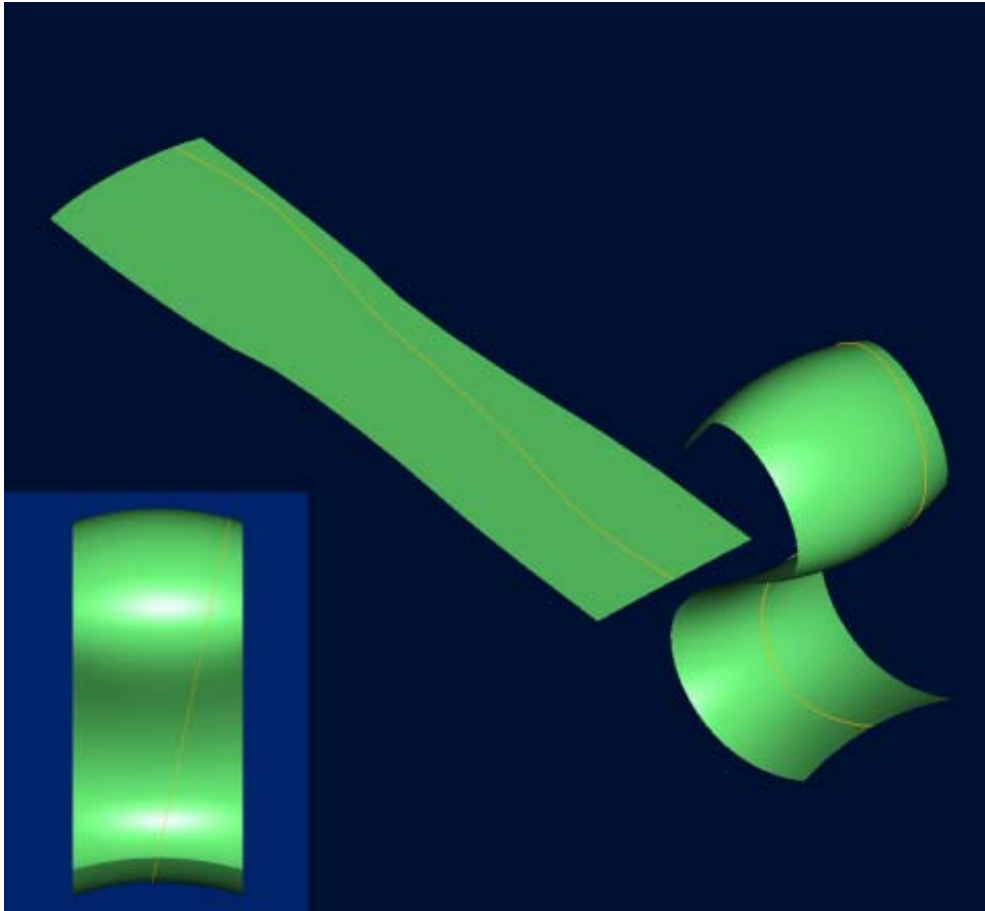


Figure 9 – Bend Solid - Flatten Curves Option

For the Bend Solid case, no additional selections are required. All solid geometry is bent back to match the original quilt. [Figure 10](#) shows a before and after shot of the feature's effects. Notice how only the solid geometry was affected.

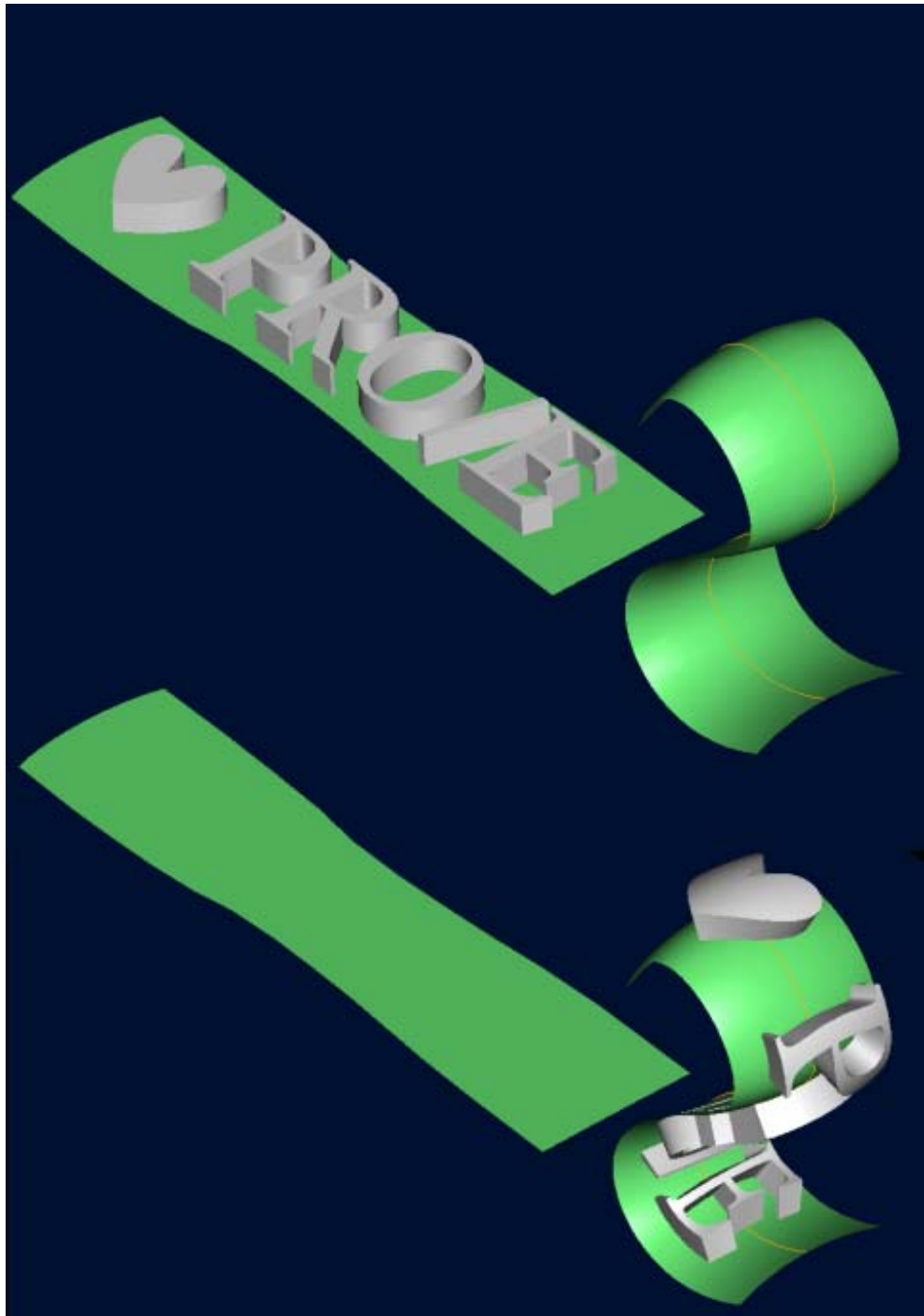


Figure 10 – Bend Solid - Bend Solid Option

Helpful Hints

Be aware that these features do not provide a tremendous amount of error diagnostics, usually just a “failed regeneration” message. Here are a few helpful hints for using these features:

- 1) Practice with simple quilts before tackling more complex ones.

- 2) Try to place the Origin Point as close to the center of area as possible. This limits the amount of deformation required at the extremities. If the feature fails, try moving the Origin Point around.
 - 3) Avoid small details in your surface. These tend to be stretched out-of-whack during the flattening process, often just failing.
 - 4) Unlike Pro/Sheetmetal, Flatten Quilt does not account for material deformation.
 - 5) Quilts made without compound curvature flatten very easily.
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Summary

Although they probably won't be used on a daily basis, Flatten Quilt and Bend Solid features can be a nice addition to your feature toolbox. They can assist in creating geometry that might be very difficult, if not impossible, otherwise.

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Tips of the Week

Part Definitions

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Announcements

Customer PTC E-Newsletter Survey

For the past 2 years I've been sending you our Customer PTC E-Newsletter such as this one.

In an attempt to deliver the highest quality and create the most value we would like to use this survey to further our "Learning" Relationship with you, our Customers. Even if you are a first time recipient of the newsletter, please feel free to share your comments on this one you're reading.

It will not take longer than a few minutes to fill out this quick survey as your input is invaluable and we greatly appreciate your input. Please click on the following link to begin:

[To Enter Survey, Please Click](#)

Please note it may be impossible to "Customize" the newsletter to meet every individual request however we will attempt to integrate as many ideas as possible.

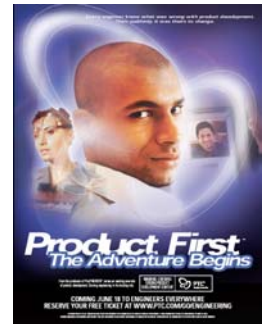
Thank-you very much for your time and feedback as we try to improve this value-added service.

PTC Sponsored Events

Click below to see PTC sponsored events:

<http://www.ptc.com/company/news/events/index.htm>

Click on the poster to gain access to the Product First Virtual Event resource center!! See the software and pick up a screen-saver or two.



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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/>

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Upcoming Events & Training Class Schedules

Upcoming, 2003 Your local Pro/Engineer User Groups
<http://www.prouser.org/rugs/>

Upcoming, 2003 Pro/USER International Conference
<http://www.prouser.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/E 2001 Student Edition!

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

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 products 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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