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## PTC Product Focus

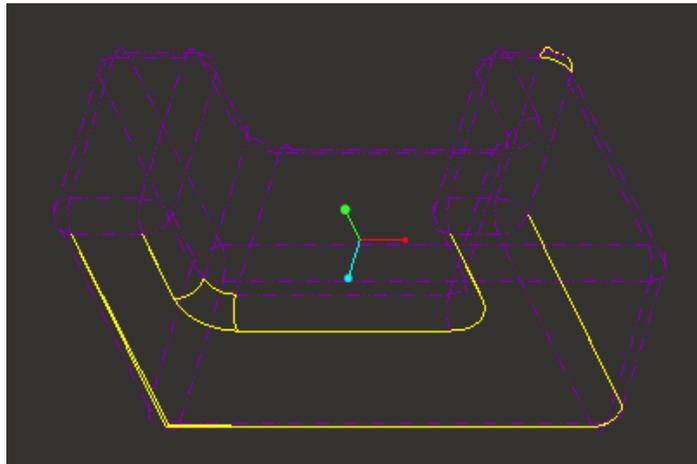
### What's New in Import Data Doctor for Wildfire 4.0

After you have imported a part model (or a part from an imported assembly model) into Pro/ENGINEER session, use Import Data Doctor to correct geometry, ensure surface consistency, add constrains, close unwanted gaps, and adjust tangencies in the imported parts.

This tutorial will use the tools in the New Import Data Doctor to repair a STEP file that has gaps, holes, and extra surfaces.

After importing an imported file, select OK in Import New Model, and then close the information window.

Notice that there are multiple edges that are yellow. This denotes one-sided edges and they need to be repaired to make this part a solid



- **RMB CLICK** on the Import Feature from the Model Tree and select **Edit Definition**.
- Select the **Import DataDoctor ...**  icon in the right-hand toolbar.

There are 3 modes available:

	<b>Repair Mode</b>	In Repair mode, you can remove gaps and sliver surfaces, add uv-curves, and fix tangency in the imported geometry.														
	<b>Modify Mode</b>	In Modify mode, you can modify individual curves and surfaces, replace one-sided edges with a uv-curve, and change surface math properties using tools and procedures such as merge, intersect, and so on.														
	<b>Featurize Mode</b>	<p>Featurize mode is the default mode when you import a model into Pro/ENGINEER. In this mode, you can convert individual surfaces or group of surfaces and quilts to analytical, spline, or procedural surfaces or quilts. You can also use surface conversion tools such as convert to plane, convert to cylinder, and so on to convert surfaces.</p> <table border="1" data-bbox="548 751 1307 1255"> <tr> <td data-bbox="548 751 630 821"></td> <td data-bbox="630 751 1307 821"><b>Repair Mode</b></td> </tr> <tr> <td data-bbox="548 821 630 890"></td> <td data-bbox="630 821 1307 890"><b>Modify Mode</b></td> </tr> <tr> <td data-bbox="548 890 630 959"></td> <td data-bbox="630 890 1307 959"><b>Featurize Mode</b></td> </tr> <tr> <td data-bbox="548 959 630 1029"></td> <td data-bbox="630 959 1307 1029"><b>Display Tangency</b></td> </tr> <tr> <td data-bbox="548 1029 630 1098"></td> <td data-bbox="630 1029 1307 1098"><b>Display Frozen</b></td> </tr> <tr> <td data-bbox="548 1098 630 1167"></td> <td data-bbox="630 1098 1307 1167"><b>Display Vertices</b></td> </tr> <tr> <td data-bbox="548 1167 630 1255"></td> <td data-bbox="630 1167 1307 1255"><b>Display Wireframe</b></td> </tr> </table>		<b>Repair Mode</b>		<b>Modify Mode</b>		<b>Featurize Mode</b>		<b>Display Tangency</b>		<b>Display Frozen</b>		<b>Display Vertices</b>		<b>Display Wireframe</b>
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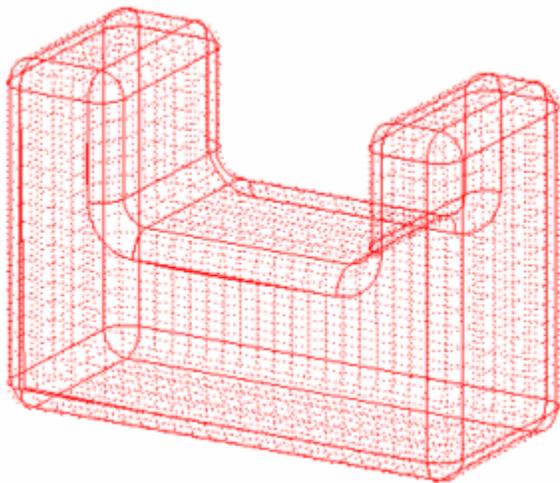
## IDD Display Toolbar

The tools on the IDD Display Toolbar are the commonly used tools in IDD.

Note the new Model Tree tab - it is the **GTS Tree** (Geometry and Topology Structure) and is similar to the standard Pro/ENGINEER Model Tree except it shows the Topology of the geometry you are attempting to repair.

## Removing Coincident Surfaces

On the GTS Tree bellow, you can see the imported model and the logical grouping of surfaces. For example, the Surface 242, lies outside the group and is not required in the model as it is a coincident surface, that is, this surface lies on top of another surface. Surface 242 can, therefore, be removed.



- Import Feature id 4
- ▶ Component 251
- ▶ Surface 242

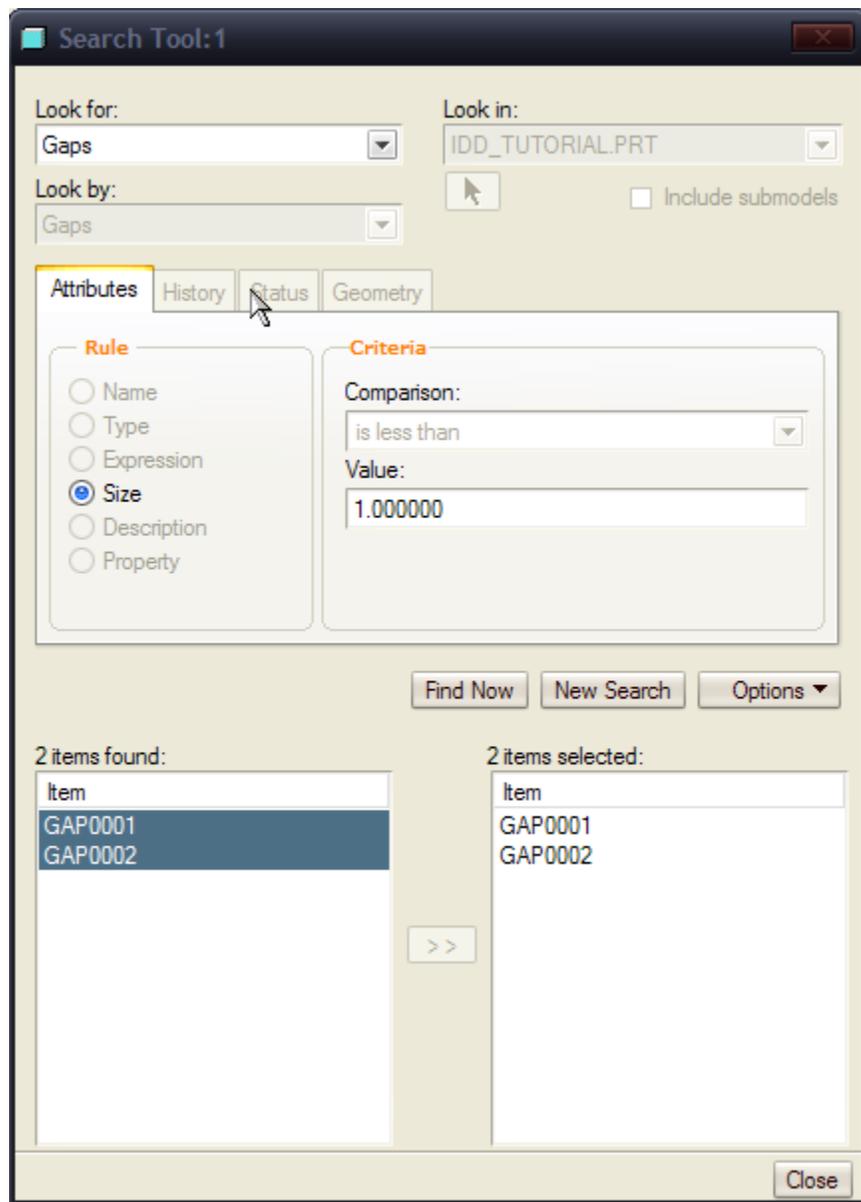
## Selecting Gaps Using the Search Tool

You can select the gaps present in the model using the Search Tool. You can define the gaps only in the Repair mode. Searching allows you to find and select gaps in your model without selecting them in the graphics window. You can specify rules for the search operation using attributes such as size value.

- Click  on the IDD Display Menu.
- Click **Find**  on the Edit toolbar to open the **Search Tool**.
- Under **Look for**, select **Gaps**.
- Click **Find Now**.

A list of found items and an area to transfer selected items are displayed.

- Set the **Value** and click **Find Now**. The Gaps appears in the list of found items.
- Select the items and click to  transfer them to the selected items area.



## Closing Gaps

Before closing the gaps you must add the edges (gaps that you have found using the Search Tool) to wireframe. When you add edges to the wireframe, Pro/ENGINEER automatically creates a wireframe piece that forms two closed loops and closes the gap.

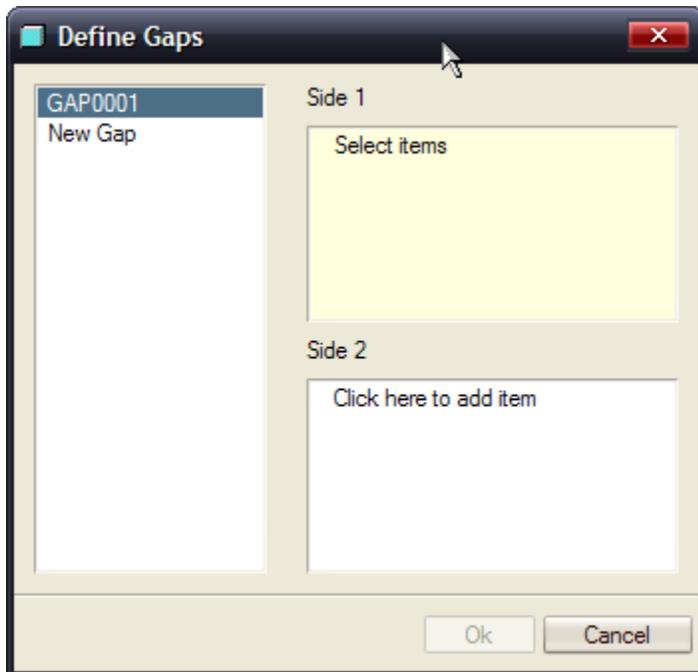
- Click  on the IDD Repair Tools toolbar to add the edges to the wireframe.  
Alternatively, click **Edit > Add to Wireframe**.
- Click  on the IDD Repair Tools toolbar. The repair dashboard appears. The new surfaces that will be created are highlighted in the graphics window.

- Click ✓ on the repair dashboard to close the gaps

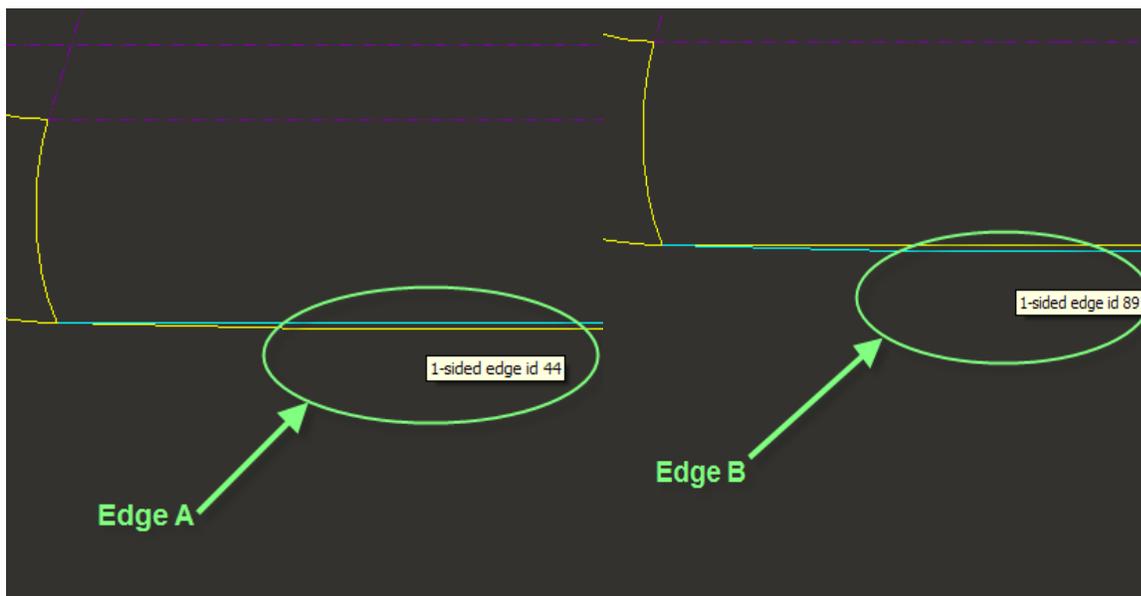
## Defining Gaps Manually

You can Define the gaps in the model by selecting them in the graphics window.

- Click **Define Gap**  (**IDD > Define Gap**) on the IDD Repair Tools toolbar. GAP0001 is displayed on the left hand side of the dialog box.



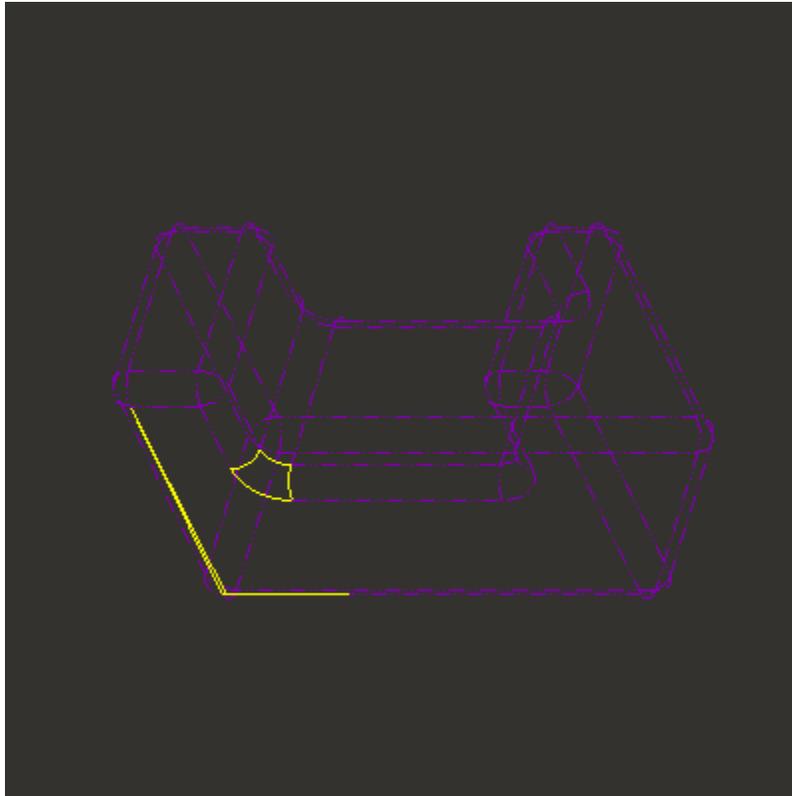
- Click the Side 1 collector box in the **Define Gaps** dialog box and select Edge A in the graphics window as shown and then click the Side 2 and select Edge B



## Adding Selected Edges to the Wireframe and Closing the Gaps

Now that you have defined the gaps to be closed, you must add the edges to the wireframe to enable Pro/ENGINEER to close the gaps.

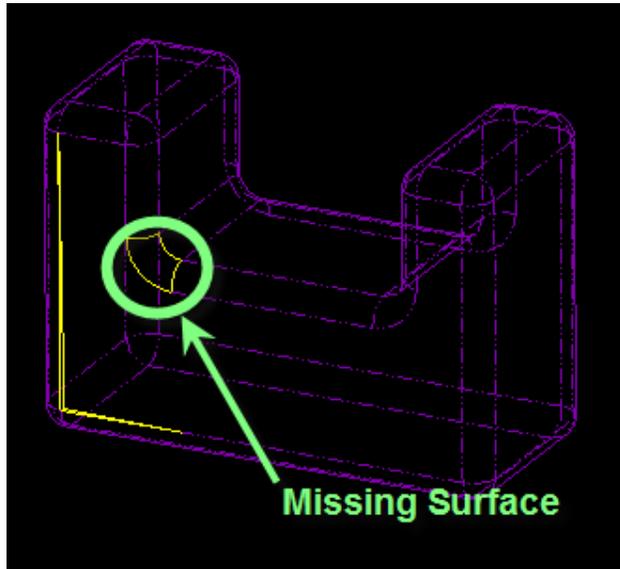
- Click **Add to WireFrame**  on the IDD Repair Tools toolbar to add the edges to the wireframe.
- Click **Repair**  on the IDD Repair Tools toolbar. The repair dashboard appears. The new surfaces that will be created are highlighted in the graphics window.
- Click  on the repair dashboard to close the gaps.



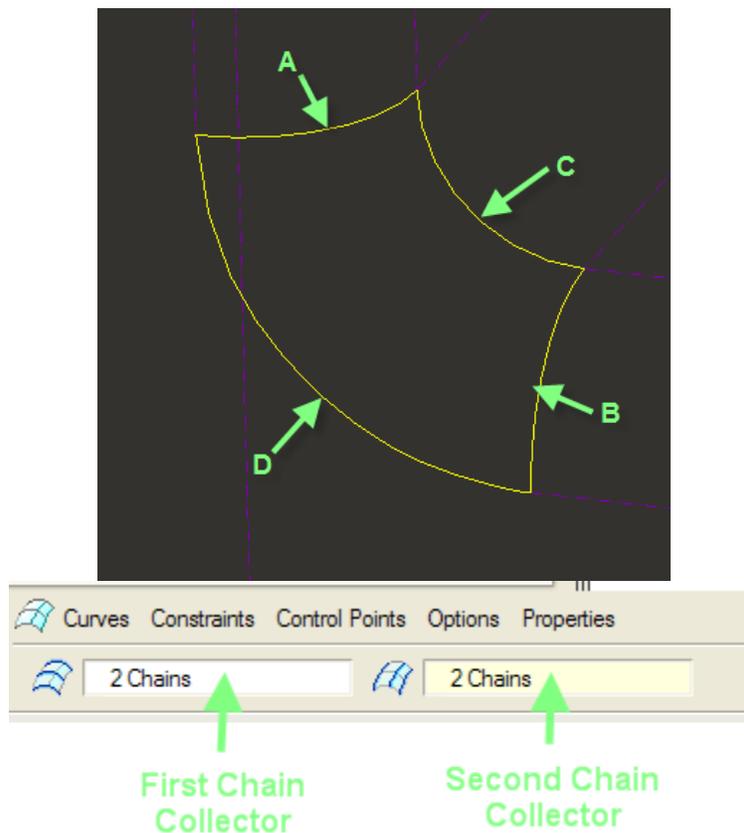
## Adding a Boundary Blend Surface

You can close missing surfaces as shown in the following figure, using the boundary blend tool. With the boundary blend tool, you can create a boundary blended feature between reference entities that defines the surface in one or two directions. The first and last entities selected in each direction define the surface boundary.

- Click **Featurize**  on the IDD Display toolbar to change the IDD mode.



- Click **Boundary Blend**  on the IDD Featurize Tools toolbar (**IDD > Boundary Blend Surface**). The boundary blend dashboard appears and the **first direction chain collector** is active.
- Select curves A and B for the first direction of the surface, as shown in the following figure.
- Once you select curve A, hold down CTRL and select curve B.

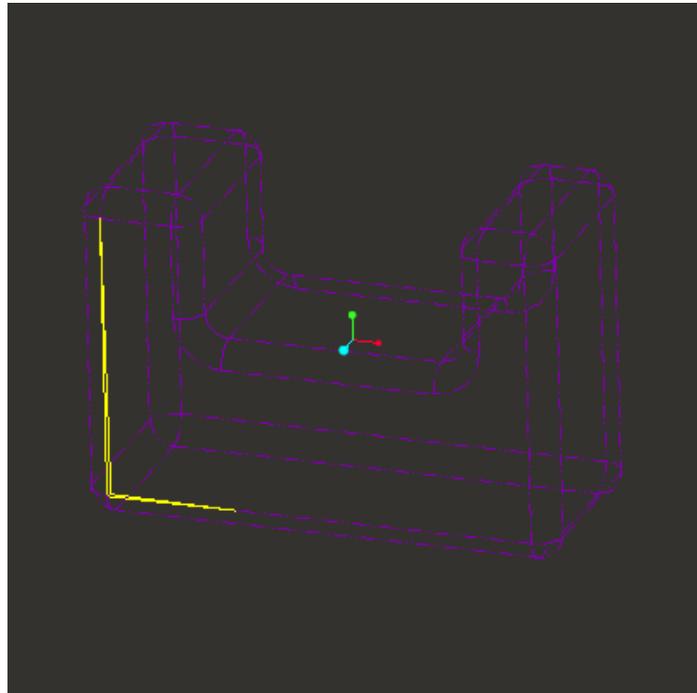
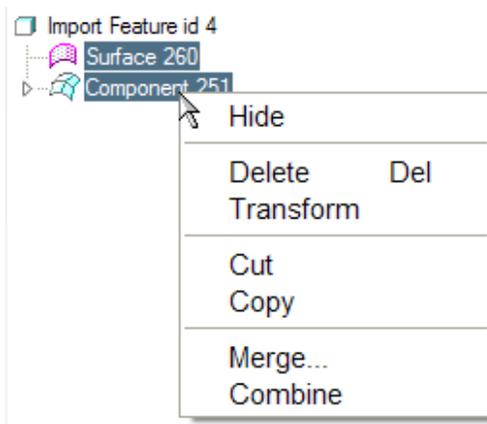


- Click **second direction chain collector** on the boundary blend dashboard and select curves C and D in the graphics window as shown in the previous figure.

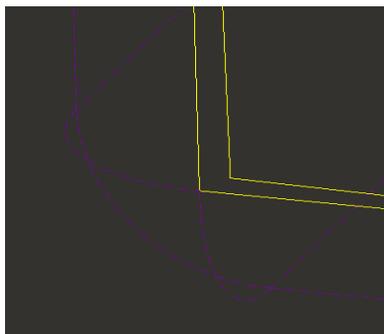
## Combining the Boundary Blend Surface

You must combine the newly created boundary blend surface with other surfaces in the imported model. When you combine the boundary blend surface with other surfaces, Pro/ENGINEER creates a combine node.

- Select the nodes as shown, click **RMB > Combine**.

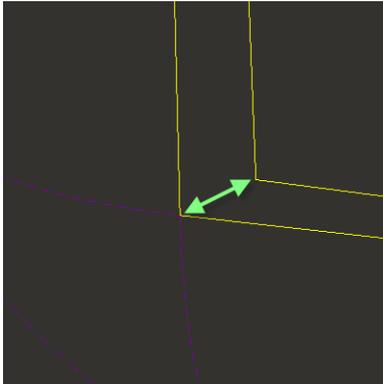


## Advanced Repair Mode



You will can repair the model by moving a vertex using the Modify Mode.

- Click **Modify Mode**  on the IDD Display toolbar to change the IDD mode.
- Click **Move Vertex**  on the Modify Mode Toolbar.
- Select the vertex with the **LMB** and **hold** to drag the vertex to the correct corner.

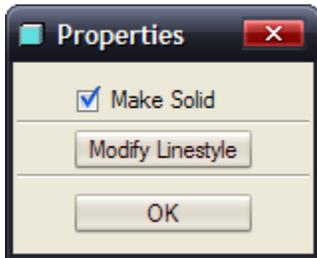


Notice that it automatically zips the gaps. If there are still open edges (i.e., magenta) then zoom in and repeat the above steps.

- Click  on the IDD Featurize Tools toolbar.

The model is fully repaired at this point. The next steps will convert the quilt to a solid feature.

## Creating a Solid Feature



- Click **Edit > Feature Properties**. The **Properties** dialog box opens.
- Click **Make Solid** to convert the closed volume into a solid.
- Click **OK** button in the Properties dialog box.
- Click  to exit the Edit Definition Mode.

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## PTC Product Focus

### Arbortext Publishing Engine Administration Overview

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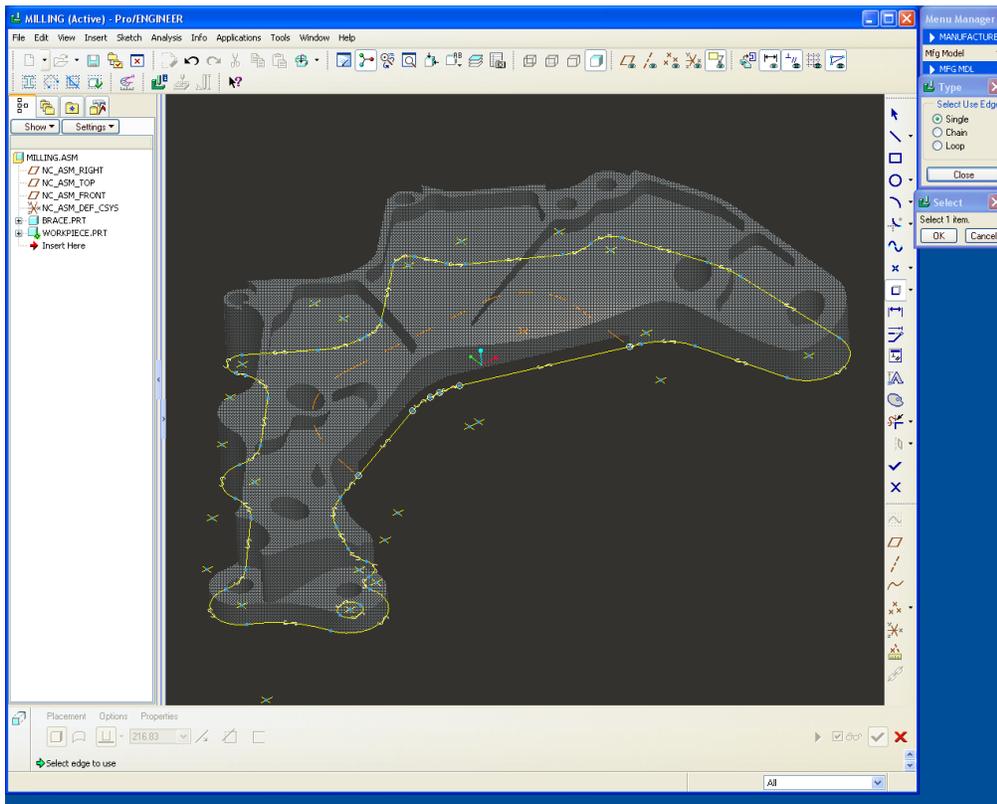
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## Tips of the Month

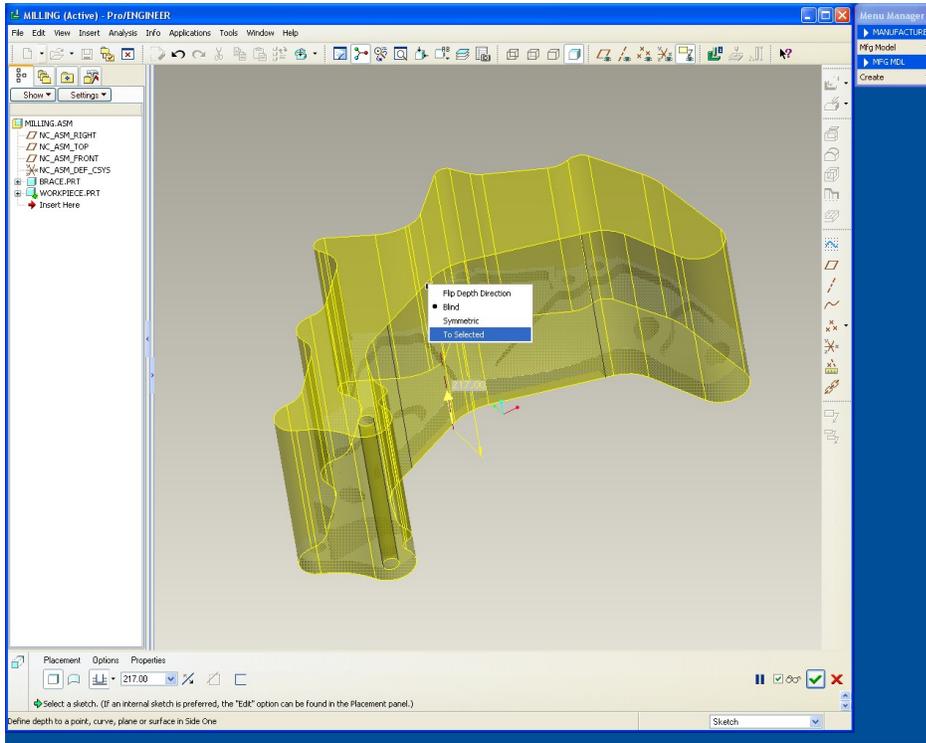
### Creating a Casting Workpiece Model in Pro/NC

In Pro/NC environment you can create the Workpiece on the fly by sketching the solid part and then applying the Offset Expand Feature to give the uniform stock material.

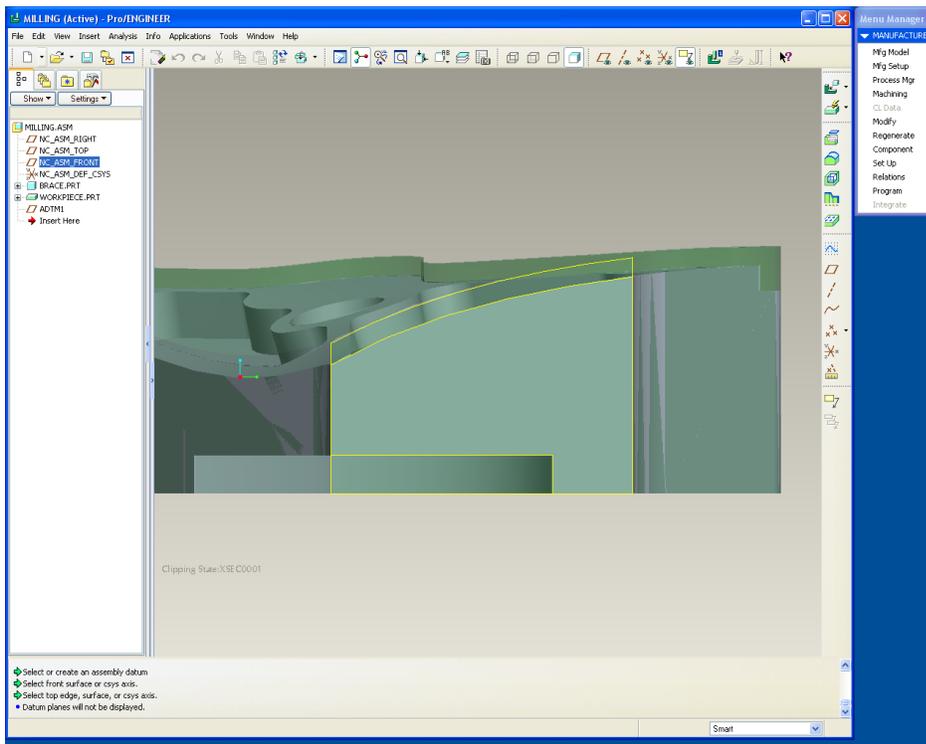
First, create a manufacturing model, select Reference Part, Default position. Select MFG Model, Create, Workpiece, type the name and select the first feature, for example, Protrusion.



After sketch the feature you need to define the blind. Select the small square, RMB and choose To Select option, and then click the top of the part.



RMB in the workpiece part in Model Tree, Activate model, In Modify Part menu, select Feature, Create, Tweak, Offset. Change the Offset option to Expand Feature. Select the top of the workpiece part and type a value. Done Return, Done. When defining the Manufacturing parameters to a Volume milling toolpath, change the Trim to Workpiece to Full Trim.



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**Tips of the Month**

**Editing a DCF file with Architect**

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## Announcements

### Educational Resource Library

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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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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, 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.

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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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