



new!

PTC Product Focus: A) [What's New in Sheetmetal for Wildfire 2.0](#)

B) [Change Management in PDMLink 7.0](#)

new!

Tips of the Week: A) [Alternative Approach for Modeling Sheetmetal Geometry](#)

B) [OOTB CMII Compliant Change Management Overview](#)

new!

Announcements: [Most Recent Announcements](#)

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

PTC Product Focus

What's New in Sheetmetal for Wildfire 2.0

The Sheetmetal interface in Wildfire 2.0, illustrated in Figure 1, has been given a new look. Much like the interface of standard Pro/Engineer, Sheetmetal feature creation is largely icon based. PTC has made great strides in developing each of the modules to appear much like that of the standard Pro/Engineer module. The new Sheetmetal interface is a testament to this approach.

As in standard Pro/Engineer, the dashboard will illuminate upon edit definition of a feature. Also, as in the standard Pro/Engineer module when creating features, there is a substantial amount of functionality at the right mouse button. Users are encouraged to take advantage of this ease of use functionality while minimizing mouse travel.

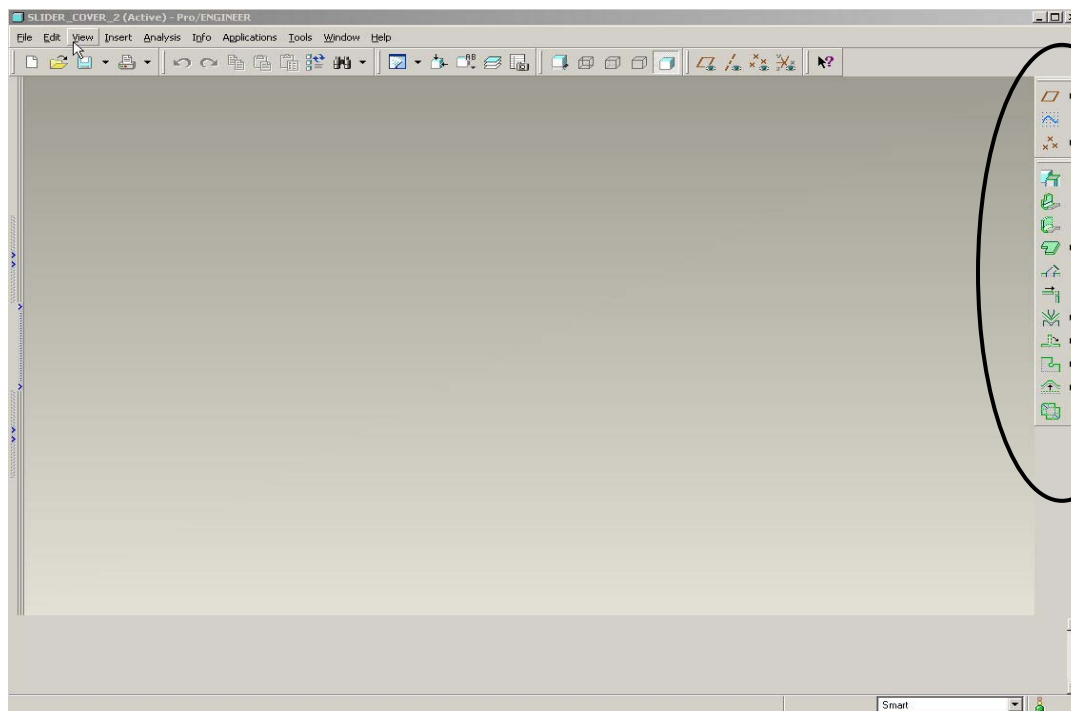


Figure 1. Sheetmetal Interface

Getting started in Sheetmetal

To begin in Sheetmetal, users must define the first wall. No other feature creation icons are activated until this first wall has been successfully generated (Refer to Figure 2). Upon selection of this icon, users are prompted with determining the sketching plane and references. From here users are brought into sketcher mode, which behaves essentially the same as it would in standard Pro/Engineer. A simple flat wall can be sketched or perhaps some unique user defined geometry. Whatever the end-users designing needs may be, Wildfire 2.0 Sheetmetal has the solution.

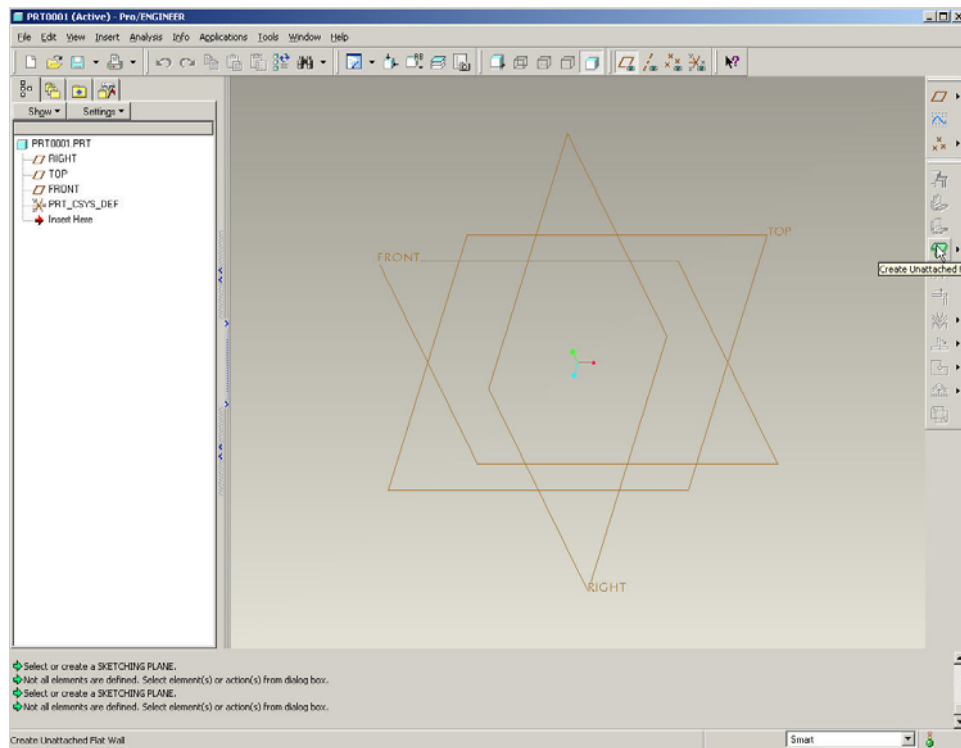


Figure 2. Initial Feature (Create Unattached Flat Wall)

Upon completion of this first feature, users may now select the additional activated icons that are available in Sheetmetal (Note: if you do not see the icons you need you can right click on the icon area and select toolbar for additional icons to add to your screen). To create basic walls the flat wall icon should be selected. As previously mentioned there is extensive functionality at the right mouse button available to users throughout the feature creation process. For example, when the dashboard is activated and a preview of the feature exists, the user may right mouse button click to find 6 additional options ranging from editing the shape (adds in additional drag handles) all the way to flipping the angle of the feature and so on.

The part illustrated in Figure 3 is an example of some fairly simple geometry created in Sheetmetal. Now that the geometry is in place, a flat pattern must be created. If the model was not designed with a flat pattern in mind, user might have to take additional steps for a flat pattern to be generated. An example of a flat pattern not working properly could perhaps be an edge that is required to be ripped.

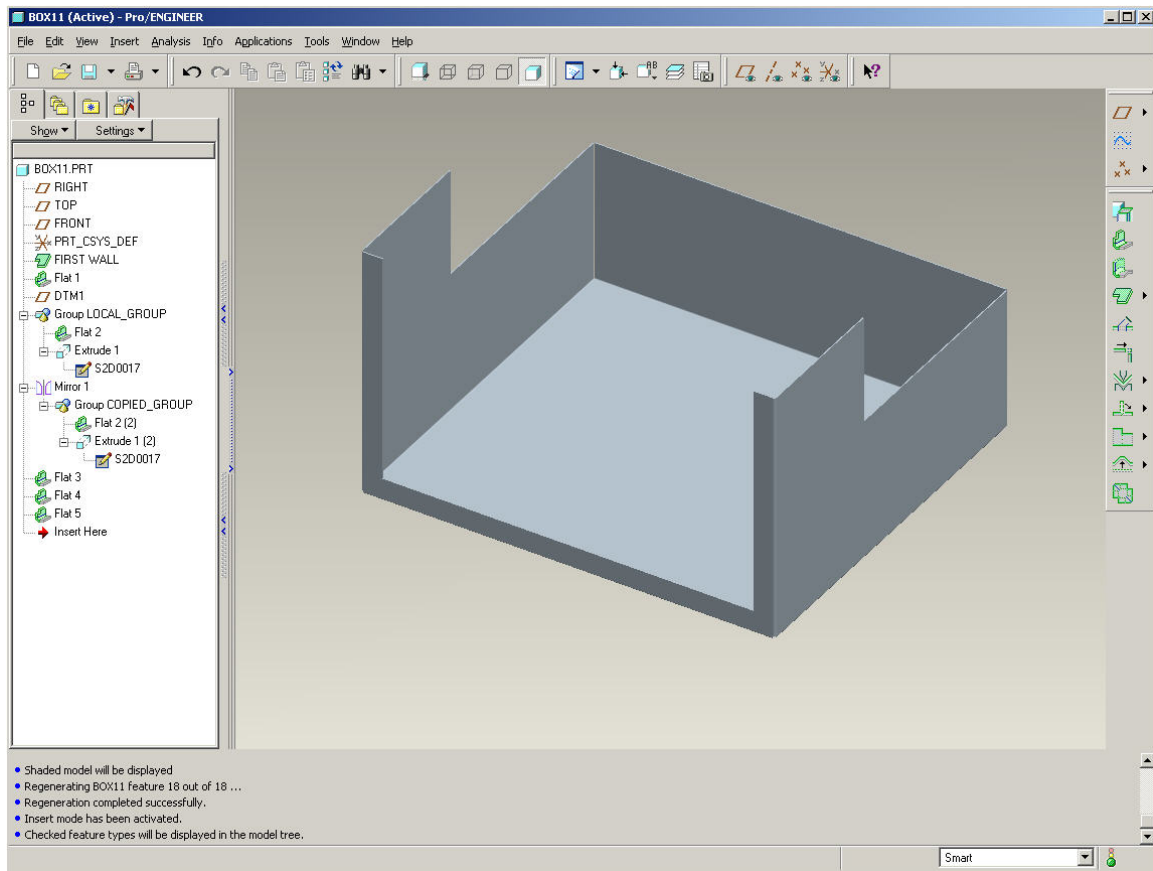


Figure 3. Basic Sheetmetal Part

Once all features within the model are defined properly, the user may select the unbend icon to generate this flat pattern of the model. Refer to Figure 4 for an illustration of this part in flat pattern form as well as the unbend icon.

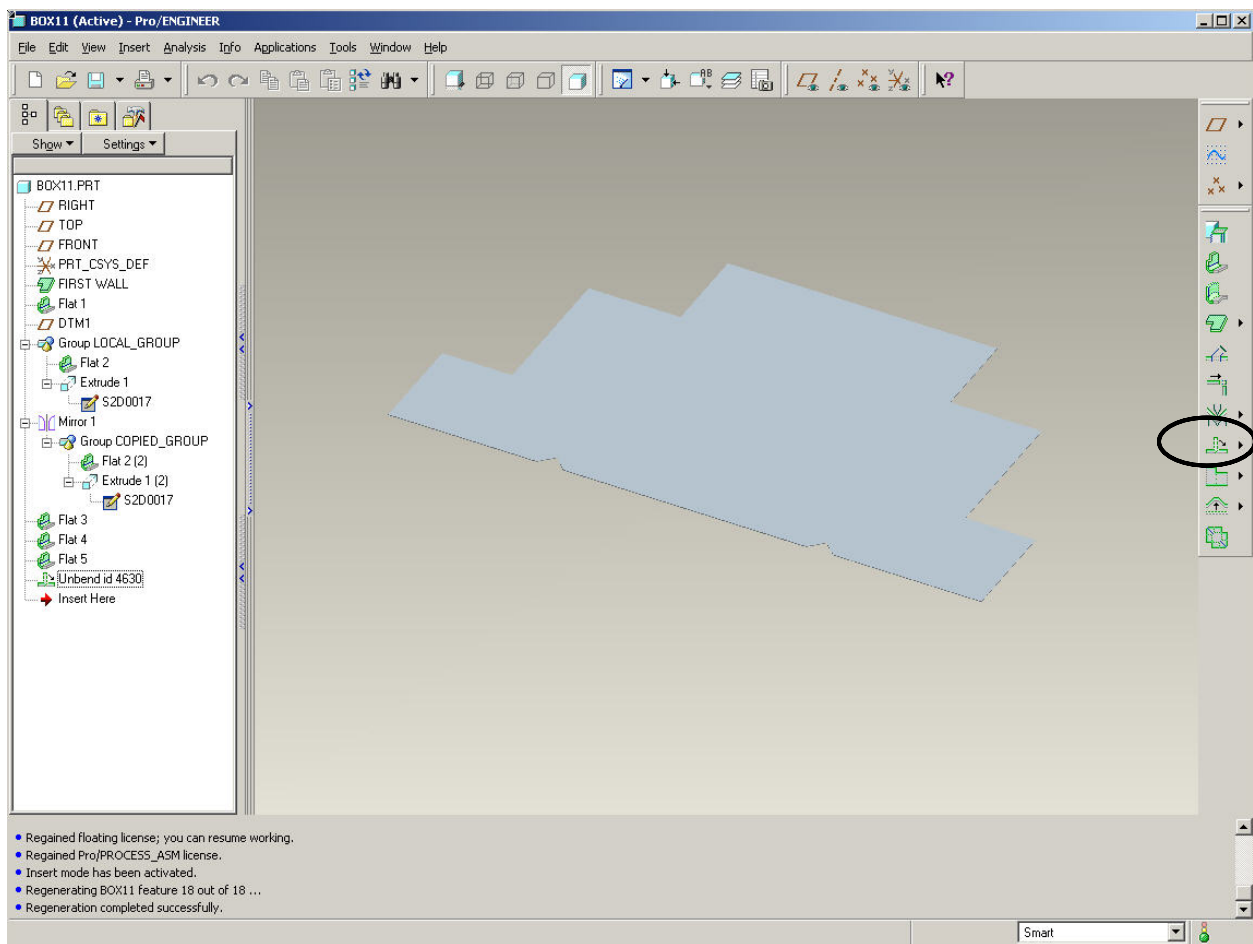


Figure 4. Flat Pattern Representation

Axis Patterning

Many feature creation icons within the Sheetmetal application are located under the Insert menu. Inserting a hole and patterning this feature is easier now than in any previous version of Pro/Engineer. Simply select Insert, hole and locate this feature with the drag handles. With the feature still highlighted, right mouse button click and hold then select pattern. The dashboard will activate, just as it would within standard Pro/E. For this pattern, select axis in the dashboard and select a central axis that this feature will be patterned about (if an axis does not exist, the user must create one). From here the user has the ability to determine how many instances of the feature will be patterned and the spacing between each. Also, the user will be prompted with a visual preview of each feature of the pattern with a black dot. If desired, the user can turn off certain features of the pattern by clicking on the black dot and turning it white (Note: all patterned features may be turned off except the original feature). Refer to Figure 5 for an illustration of axial patterning.

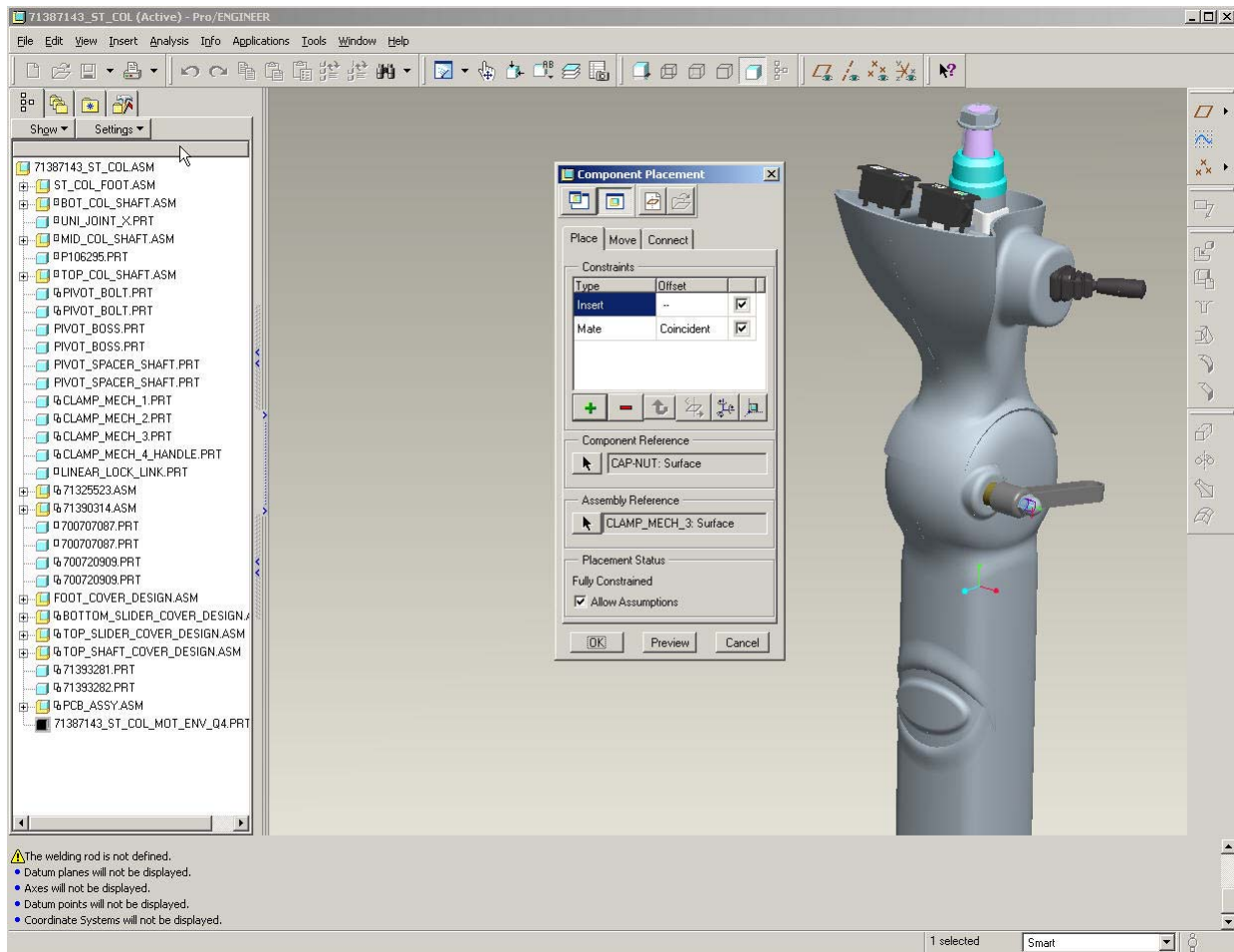


Figure 6. Weblink Component Drag and Drop Functionality

Another example of the Weblink functionality would be dragging and dropping of features. For this example, a formed feature will be dragged from the library into a Sheetmetal part. In order to place this feature a datum point must first be established in the appropriate location in the part. From here, the user can simply find the appropriate formed feature within the library and select this feature and drag it into the user interface. Pro/Engineer will then prompt the user for references to locate this feature (select a placement plane, datum point and horizontal reference). Figure 7 is an illustration of the Weblink Feature library.

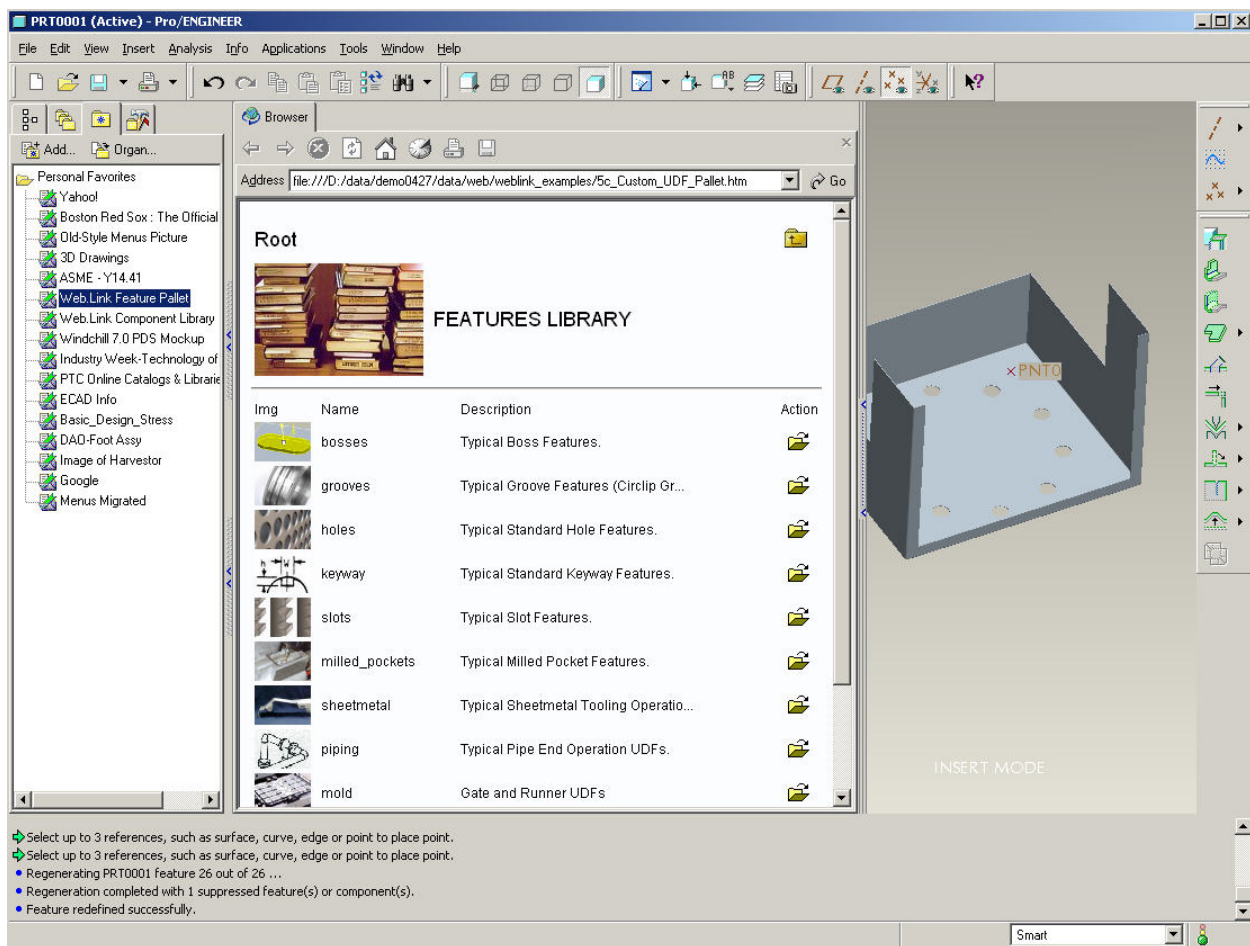


Figure 7. Weblink Feature Library

Directional Patterning

As a natural continuation of inserting a single formed feature into a part, users may wish to next pattern this feature. For this situation, a directional pattern will be used. With the feature highlighted, right mouse button click and hold and select pattern. The dashboard will illuminate prompting the user to determine the type of pattern desired. A directional pattern will be used, as could a dimensional pattern. From here the user has the ability to determine how many instances of the feature will be patterned and the distance between each by manipulating the drag handles and dashboard as shown in Figure 8. Also, the user will be prompted with a visual preview of each feature of the pattern represented by a black dot.

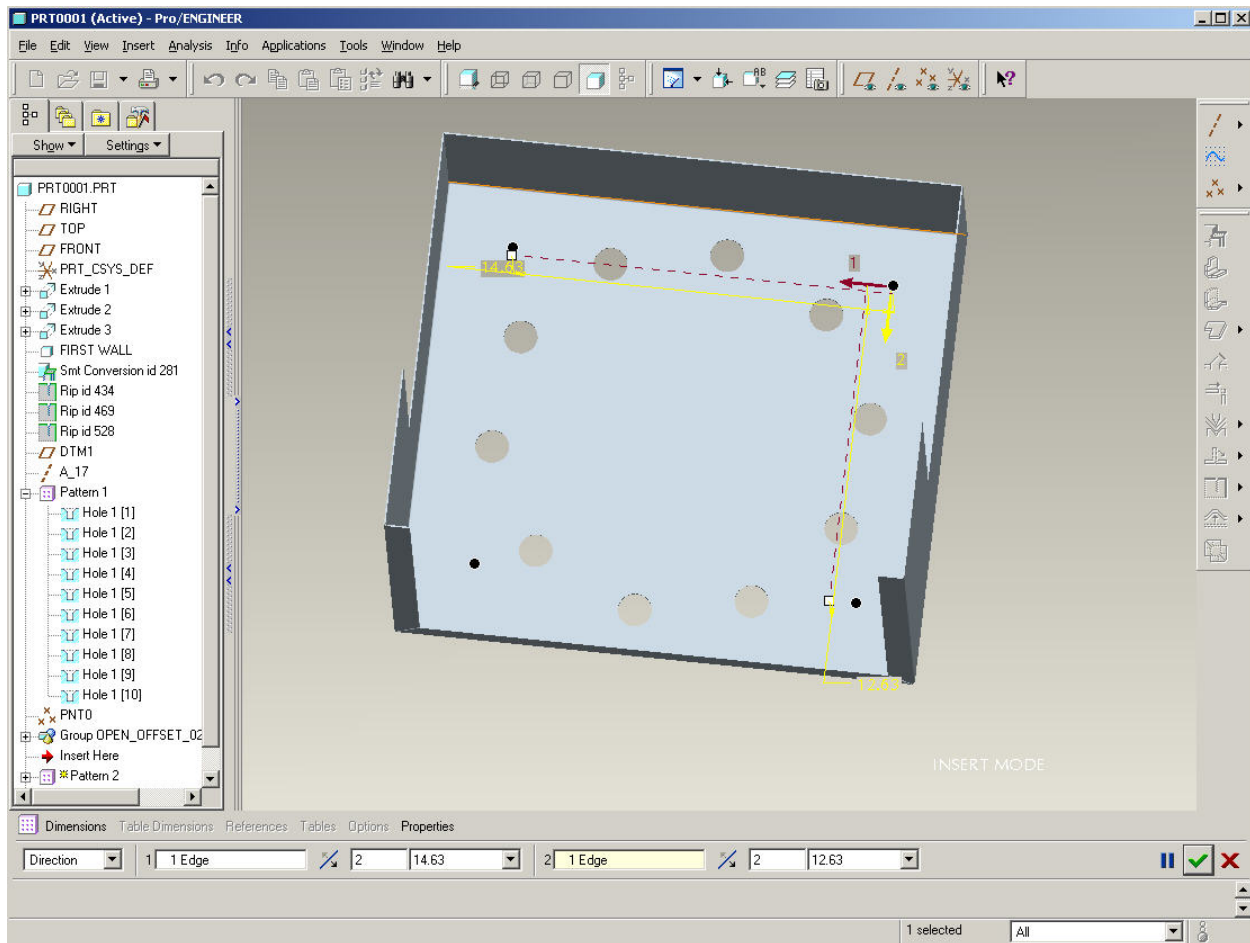


Figure 8. Directional Pattern

If desired, the user can turn off certain features of the pattern by clicking on the black dot and turning it white (Note: all patterned features may be turned off except the original feature).

Undo/Redo

Undo and Redo exists within the Sheetmetal application at both the sketcher level as well as the feature level.

Copy/ Paste

Within the Sheetmetal application, users have the ability to copy features (walls, holes, etc.) and paste (or paste special) those features to a new location in the part. Much like within the Microsoft Office Programs, Control-C and Control-V also work for copy/paste. Refer to Figure 9 for illustration of copy/paste for a wall.

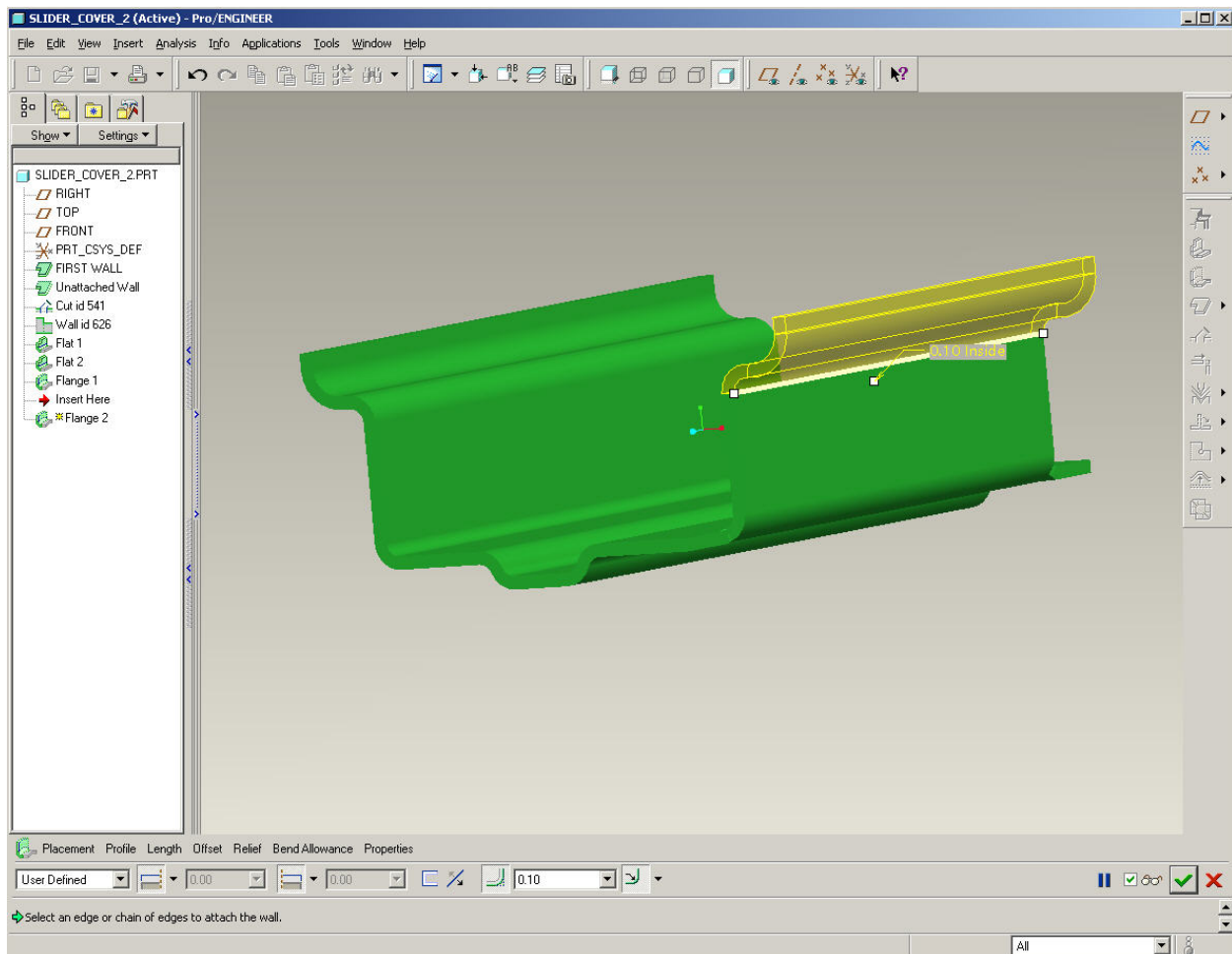


Figure 9. Copy/Paste Functionality

Another added benefit of the copy/paste functionality is that the copied feature is independent of the original feature in the model tree. Future modifications to the original feature will not impact the copied feature. However, if the opposite outcome is desired, users can select the paste special functionality so that the copied feature is dependent on the geometry of the original feature.

Reliefs

Four standard relief options exist within the Sheetmetal application. Users will have the ability to modify some of the basic dimensions of those reliefs if desired. Also, users have the ability to define opposite ends of the same wall with different types of reliefs adding to the flexibility of the tool as shown in Figure 10.

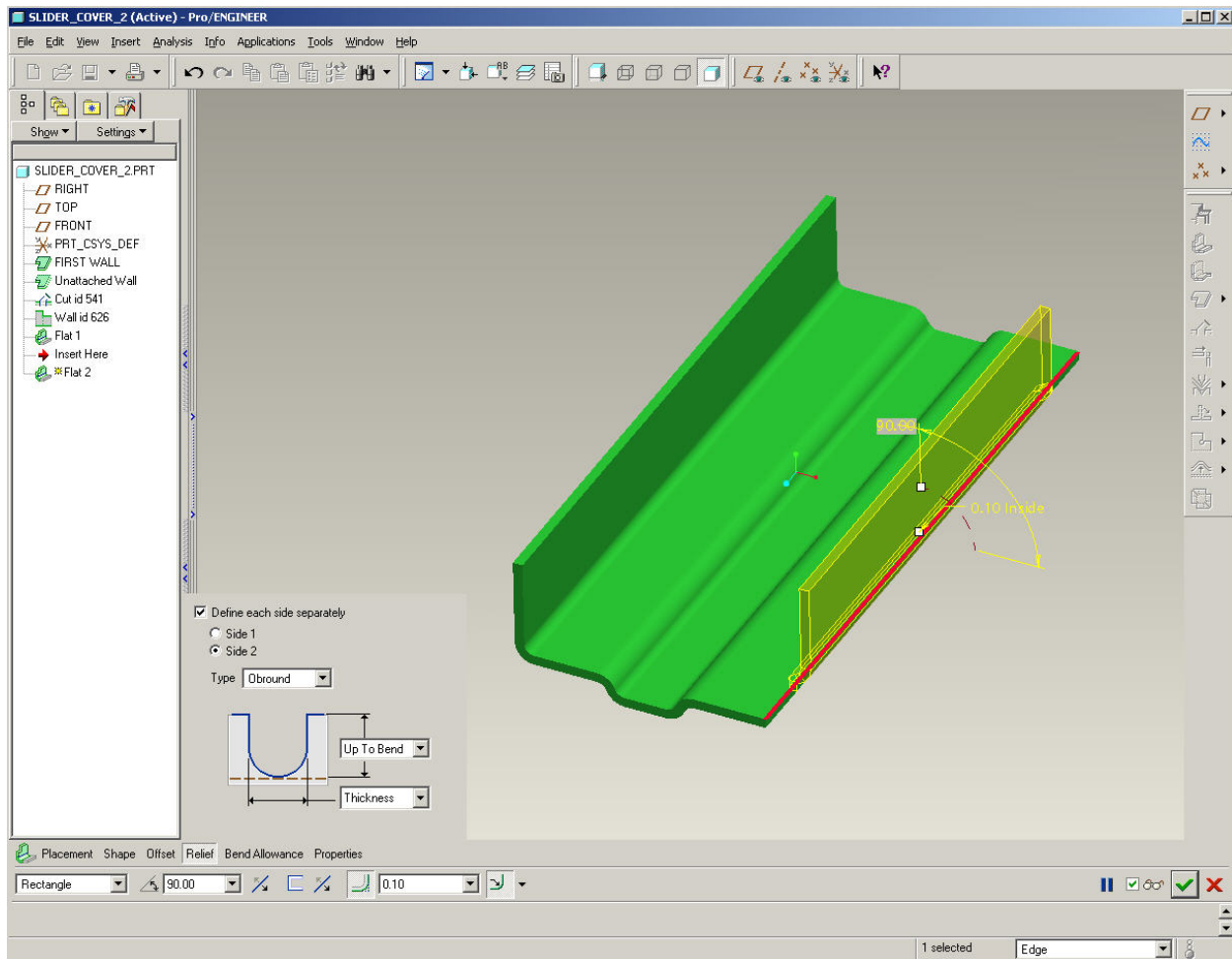


Figure 10. Reliefs in Sheetmetal

Family Table - Sheetmetal Part

Users have the ability to make a family table instance of the flat pattern for later use within a drawing. Select Tools, Family Table and the add/delete table columns icon. Select feature in the Family Items menu and within the model tree select the unbend feature. Now select the insert new instance icon in the Family Items menu and set the unbend feature to yes. This flat pattern instance can now be inserted as a to-be dimensioned view in the drawing that will be used by manufacturing when fabricating the part.

Sheetmetal Drawings

Users can easily add views of their Sheetmetal part within drawing mode for detail purposes. As with parts generated in standard Pro/Engineer, the default three views (front, right and top) will automatically be added to the drawing. For a Sheetmetal part it is important to show dimensions in the flattened state. Figure 11 displays a flat pattern instance of the part with ordinate dimensions. When placing ordinate dimensions users can specify whether it is desirable to have the dimensions called out to the center of a hole or to a tangent point. Also included is a hole table which calls out dimensions relative to a central coordinate system that the user must specify.

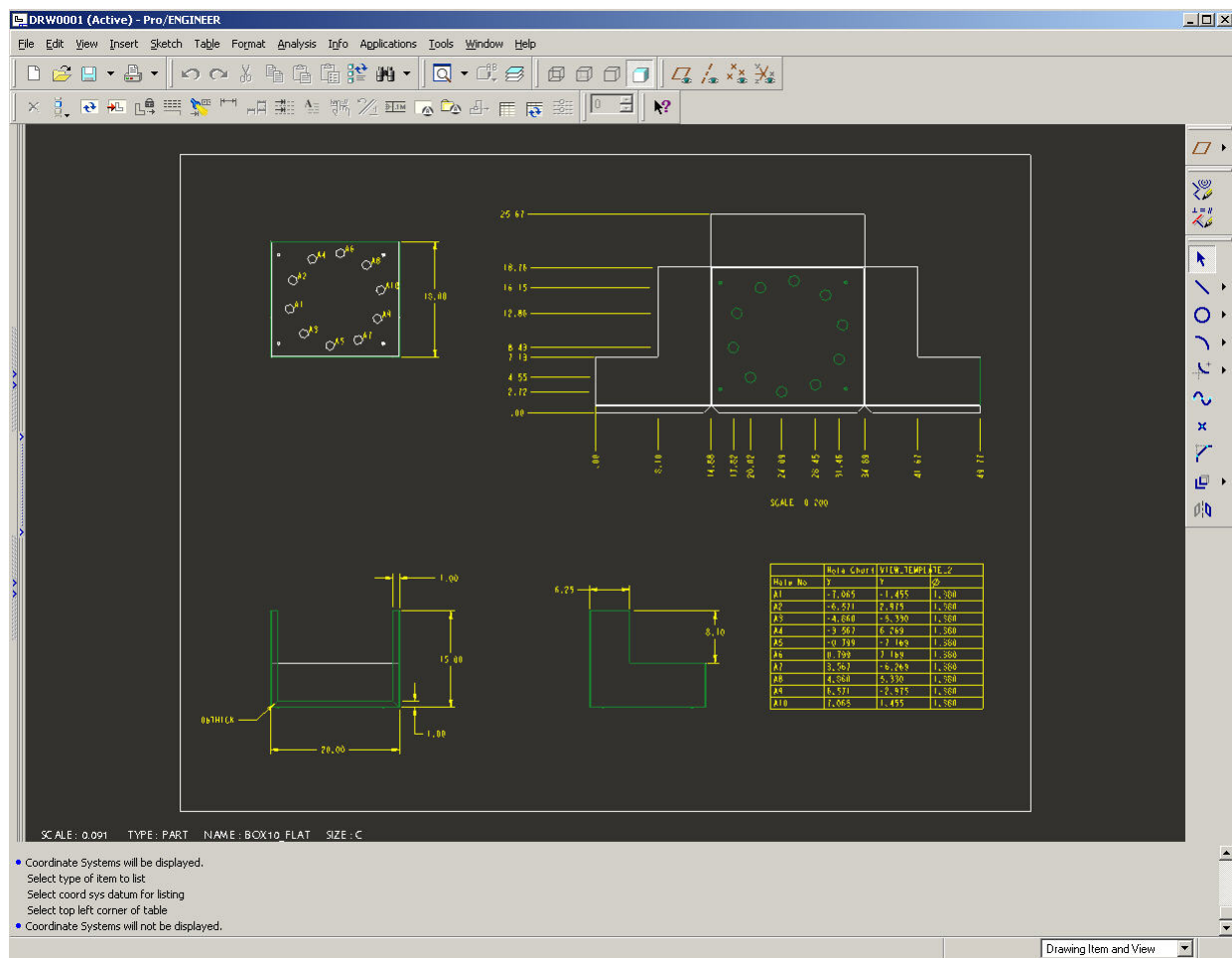


Figure 11. Detailed Sheetmetal Part

More Information on Sheetmetal in Wildfire 2.0

To find out more about Pro/ENGINEER's Sheetmetal Application visit our Website.

[Back To Top](#)

PTC Product Focus

Change Management in PDMLink 7.0

[Click Here To View](#)

[Back To Top](#)

Tips of the Week

Alternative Approach for Modeling Sheetmetal Geometry

There are several ways in which to get started creating a Sheetmetal part. As an alternative approach to starting in the Sheetmetal application, users could model their geometry in standard Pro/Engineer and then transfer this geometry to the Sheetmetal application. Choosing which way to start your Sheetmetal part is completely up to the individual user. As experience is gained with standard Pro/Engineer and the Sheetmetal application, it will become easier to determine which approach will be more efficient. Also, users will be able to firmly understand whether all required design intent for the model is being incorporated into the part when starting in Standard Pro/Engineer.

User must first start by modeling the general geometry, then select Sheetmetal under the Applications tab as shown in the Figure 1.

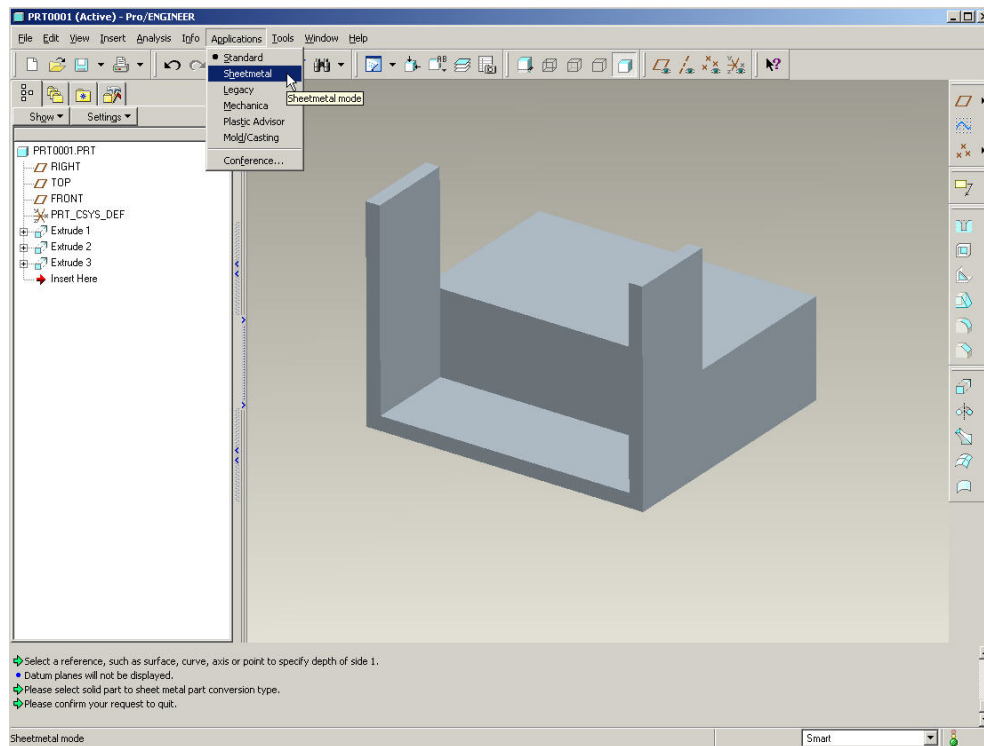


Figure 1. Geometry Created in Standard Pro/Engineer to be Imported Into Sheetmetal

At this point, Pro/Engineer will prompt the user to shell the part by selecting the surfaces you wish to shell followed by specifying a material thickness. Upon completion of this task, the part now exists in the Sheetmetal application and the user will need to take appropriate next steps to create a flat pattern of this part.

Since all edges are fully attached, appropriate edges will need to be ripped. By selecting the create conversion icon as shown in Figure 2 users can rip the back 2 edges needed to create a flat pattern and add a bend to the front 3 edges if desired.

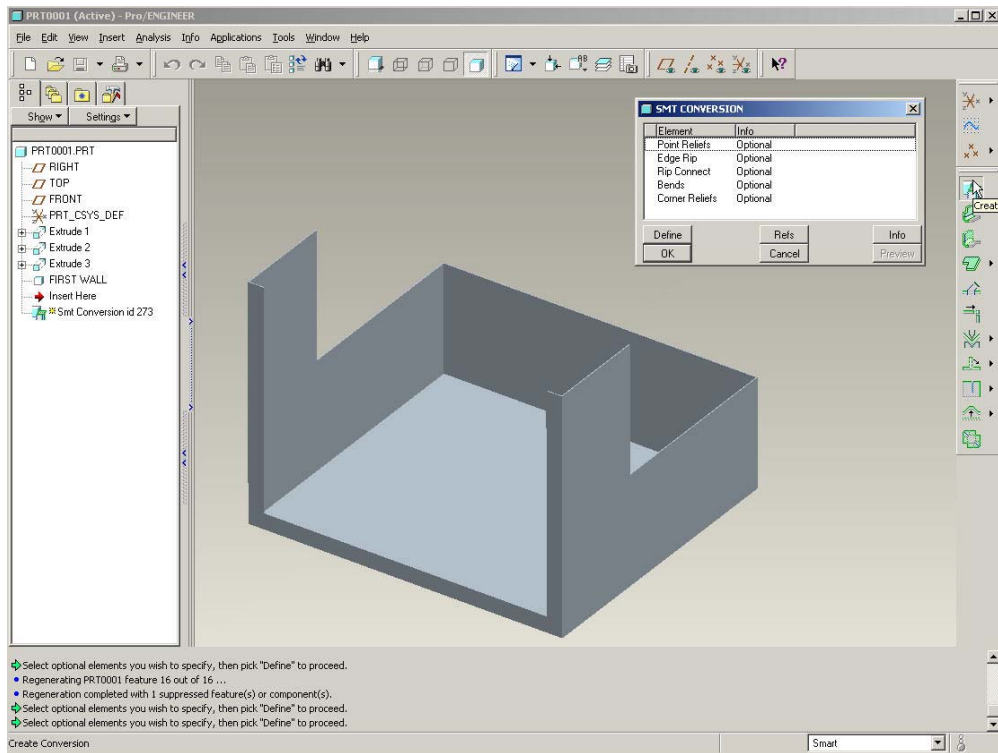


Figure 2. Create Conversions Functionality Used to Rip Edges

From here, the part now requires two surface rips to remove material at the front two lower edges as seen in Figure 3.

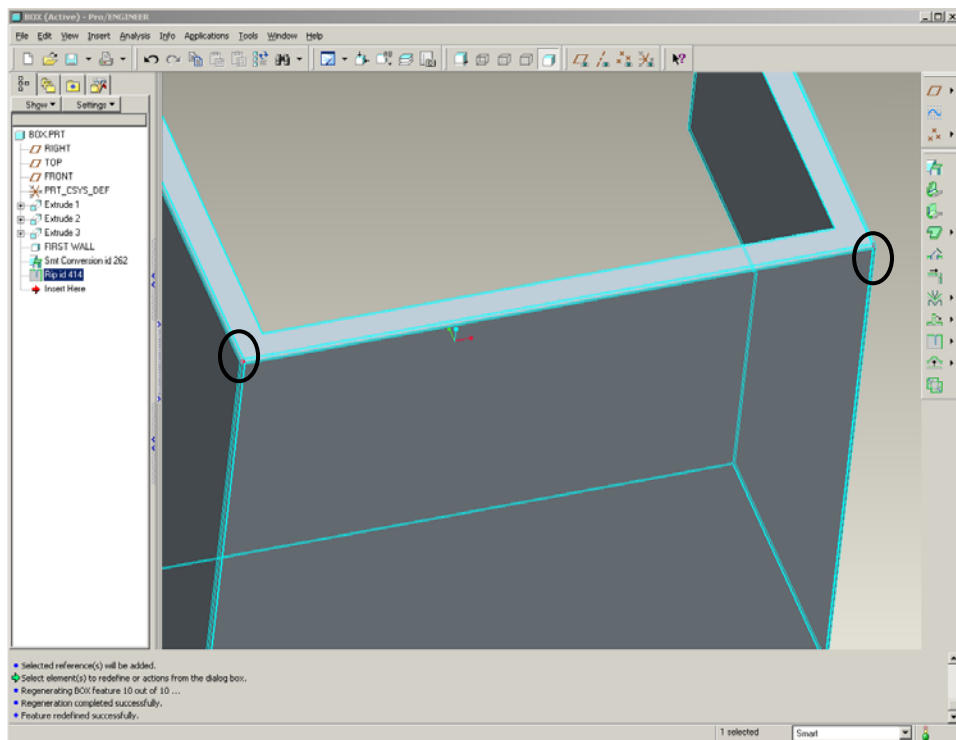


Figure 3. Surface Rips of Lower Front Edges

Lastly, the user is required to create a regular rip to break the remaining three front faces into individual sections. Refer to Figure 4 for applicable surfaces.

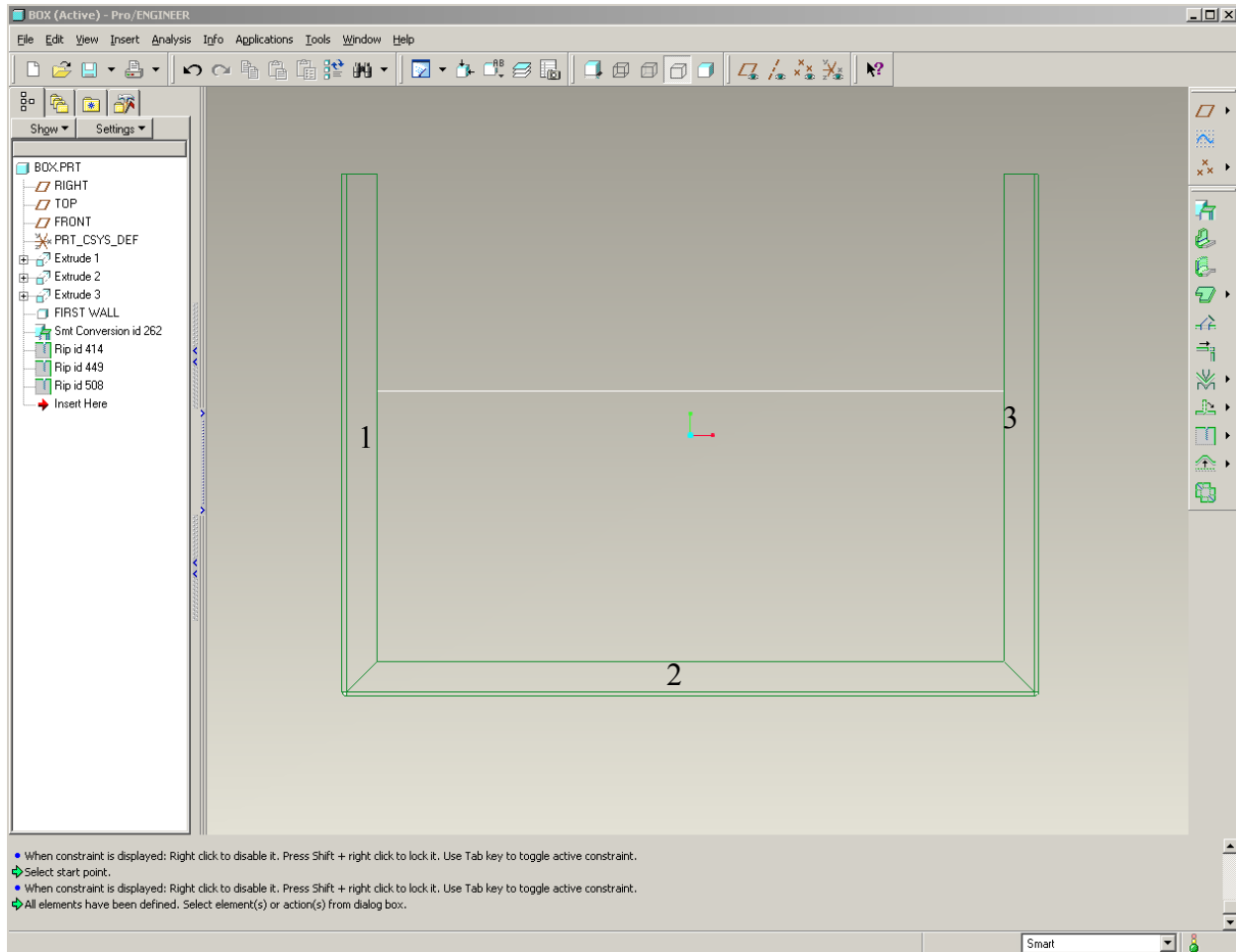


Figure 4. Regular Rips Creating Three Individual Sections

Upon completion of the above steps, this Sheetmetal part can now be flattened. Figure 5 represents this part in its flattened form.

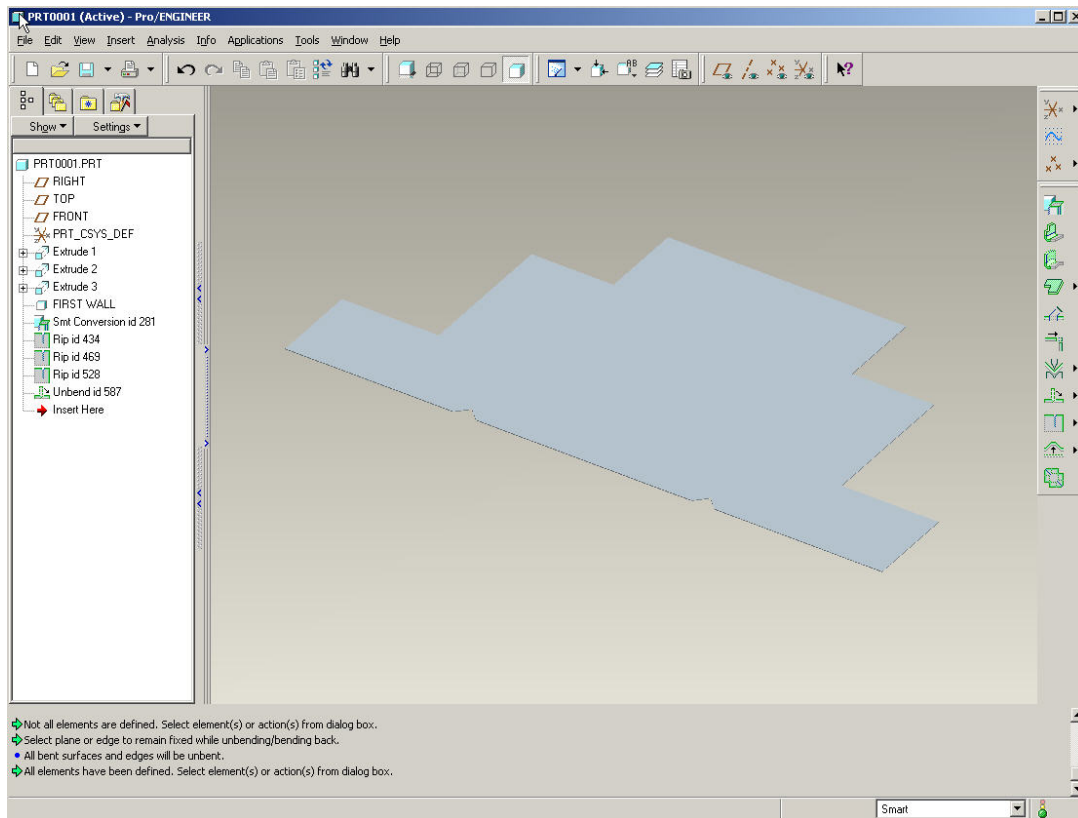


Figure 5. Flat Pattern

[Back To Top](#)

Tips of the Week

OOTB CMII Compliant Change Management Overview

[Click Here To View](#)

[Back To Top](#)

Announcements

Live Instructor-Lead Virtual PTC Training Courses

PTC will be offering a series of Virtual Classrooms this quarter for you looking to save money (15% off ILT prices) and cut down on travel.

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 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.

[Click Here To Access](#)

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!](#)

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

<http://www.3dlabs.com/PTC/>

PTC Sponsored Events

Click below to see PTC sponsored events:

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

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

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](#)