



PTC Product Focus: [Progressive Die Extension \(PDX\)](#)

Tips of the Week: **A) [Imported Data w/ Downstream Applications](#)**
 B) [PDMLink : Adding New Attributes To A Part](#)

Announcements: [Most Recent Announcements](#)

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

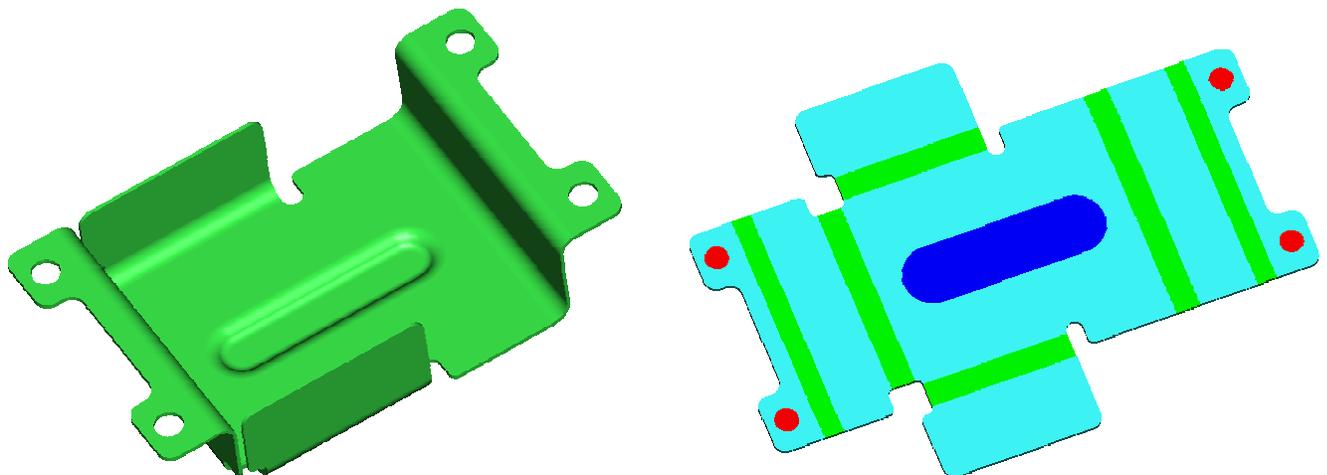
PTC Product Focus

Progressive Die Extension (PDX)

For years, Progressive Die design has been considered to be a black art. Very few designers have mastered the skill and as time has passed by, fewer individuals have been picking up the trade. The work of Progressive Die has historically been extremely tedious and time-consuming. Many efforts have been made to apply the benefits of 3D Solid modeling to this task but most users typically revert back to 2D drawings to complete their jobs. Prior attempts at automation of this task had proven lengthy, time consuming and incomplete. That was until today.

PTC has recently announced the new Progressive Die Extension otherwise known as PDX. http://www.ptc.com/community/proewf/newtools/progressive_dd_capabilities.htm By adding this module to a Foundation package, a user has the capability of designing progressive dies in significantly less time with better quality and, most of all, will be able to quickly incorporate design changes throughout the entire design through manufacturing process.

PDX is extremely easy to learn because it works the same way that a progressive die designer thinks. The process begins by preparing a part for the progressive die process. The designer determines which features on a model will be punched, formed or bent.

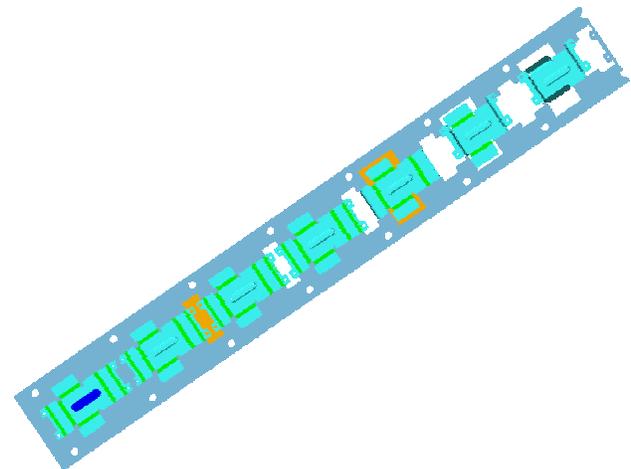
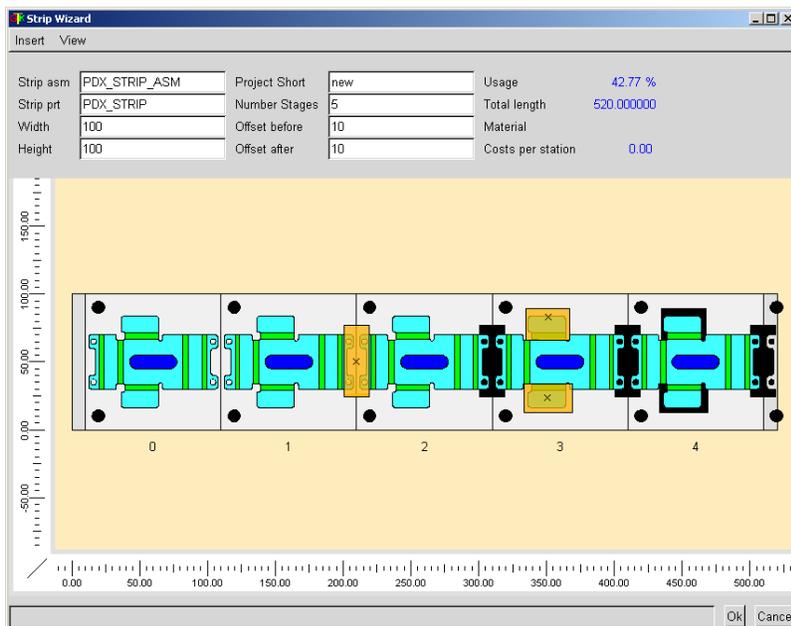


The color-coding shows that the red items will be punched, the blue items will be formed and the green items will be bent. The significance of this is even seen in the corresponding menus that have the same colors.

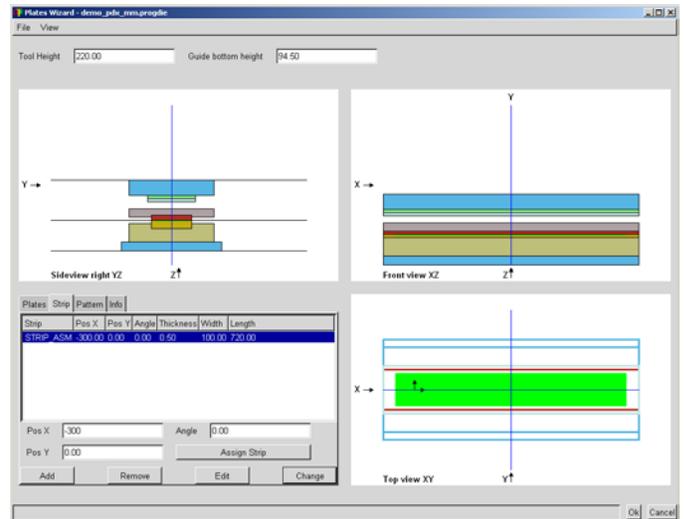
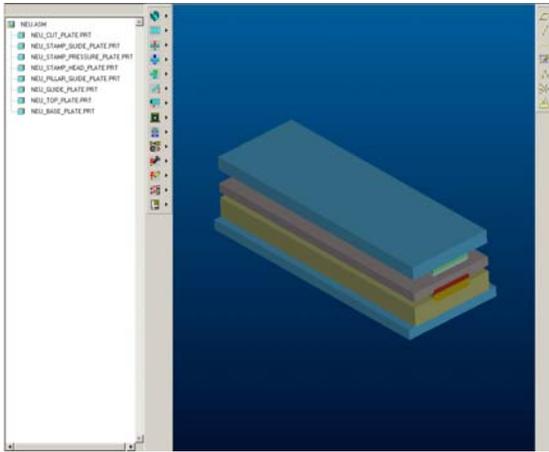


The material is then chosen and flat pattern lengths are automatically calculated.

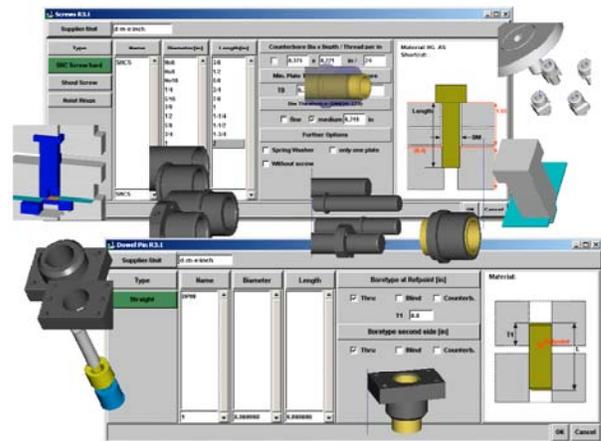
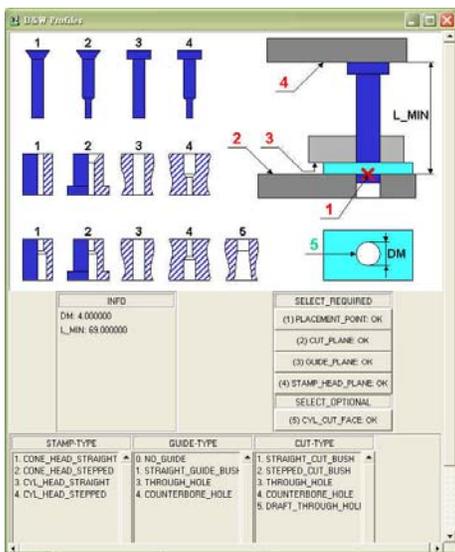
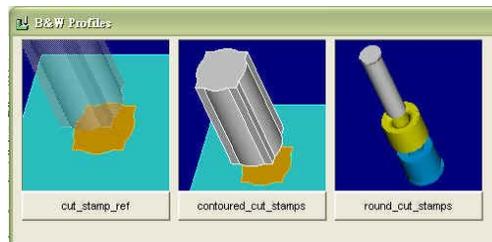
Typically, after inspecting a part, the die designer creates a strip layout. This is the step-by-step progression from the fully flattened part to the fully formed part. All of these variants are attached to a strip of sheetmetal. In the traditional process, this is time consuming because design changes have to be made for every stage of the progression. In PDX, there is a Strip Wizard which allows users to layout a strip in a typical 2D fashion while in the background, a 3D strip is automatically generated. At this point, punch references are created along with bend information. Features that are added to one stage of the progression, are automatically added to all successive stages. Changes to any one stage will automatically be reflected in all stages. All changes to the 2D Strip Layout are automatically reflected in the 3D Strip Layout and vice versa.



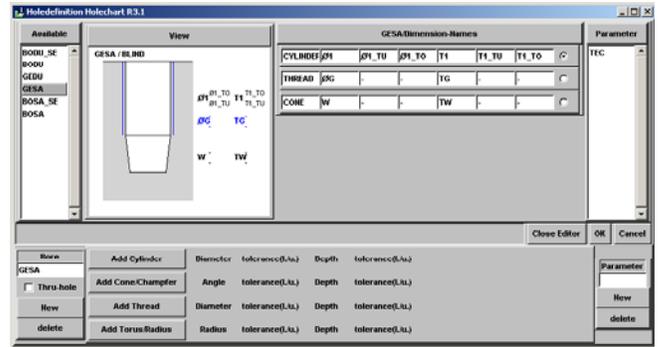
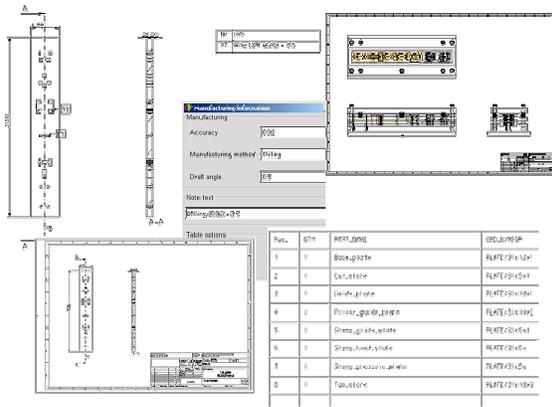
Once the strip layout portion of the design is complete, one uses the Plate Wizard to build up the necessary plates in a drag and drop fashion. The user can retrieve from a library full die sets in standard configurations with preset locations for guides, springs, etc all in the easy to use interface.



Once the plate stack has been automatically generated from the Plate Wizard, the user can now begin to place punches and forms directly relating to the strip or surrounding geometry.



Amazingly, when a stamp, punch or component is placed, all associated cutouts are generated on related plates automatically. These cutouts can have prescribed clearances. Each plate could have a different clearance if desired. All plates are automatically documented using drawing templates. Advanced holechart capability automatically detects hole geometry and documents accordingly.



Full associativity allows design changes to the original part be reflected in the flattened part, the strip layout, all related tools and surrounding plates. The 2D documentation will automatically update and so will manufacturing toolpaths.

[Back To Top](#)

Imported Data W/ Downstream Applications

More and more, companies are being asked to incorporate imported data into designs that they are currently working on. In earlier years, the difficulty was first in importing data and then in trying to assimilate these files into a current design. Early transfer of data was made difficult because 3D surface modeling systems were imprecise. Such a surface model was only as good as the author wanted it to be. In that case, all edges of surfaces did not need to meet to have a “valid” surface model.

When a surface model was imported into a newer system like Pro/ENGINEER, it appeared that the solid modeling system was having trouble with the imported data. In most cases, Pro/ENGINEER was only importing the data in the form that it was given from the authoring tool. Everyone blamed Pro/ENGINEER or IGES but in many cases (not all), it was the original 3D surface model that was the culprit.

Today with 3D Solid Modeling being much more mainstream, data transfer is significantly easier than it had been in the past. The Solid Model, by definition, is a closed model. Unless there were unattached features on the model, all surfaces had to meet in order for that model to be solid in the first place. IGES transfers today seem to be working much better than they did in the past but much of this can be attributed to the higher quality of the original models. In other words, the solid modeling system, in many cases, was forcing the user to build a closed surface model in order for it to be solid in the first place.

Now with the advent of STEP, there is a transfer of not only clean surface information, but also solid information. Using STEP, you can import a whole assembly from a single file. Still, physical data transfer can defy logic. For example, IGES might work where STEP might not. You could have a perfectly symmetric model and one side of the model imports cleanly while the other does not. The list goes on. This is why PTC offers the Import Data Doctor as functionality above and beyond the import capability offered in the Foundation package of Pro/ENGINEER.

What we are assuming for this tip is that the user is able to get over the first hurdle of importing the geometry and is able to assemble the object into a given assembly. This object may have references made to it by other assembly members or there may be manufacturing toolpaths applied. There may even be an analysis performed on it by products like Pro/MECHANICA or CE/TOL. Everything may be going along great until there is a design change made in the original authoring product and this design change needs to be seen in the imported model and all downstream applications. In an ideal case, all the user would have to do is Redefine the import feature on the changed model, read in a new STEP or IGES file and all related objects should update. This can occur if the original model changes in only dimension and the shape remains intact. In other words, there were no additional cuts, rounds, or protrusions added to the model and no features were removed.

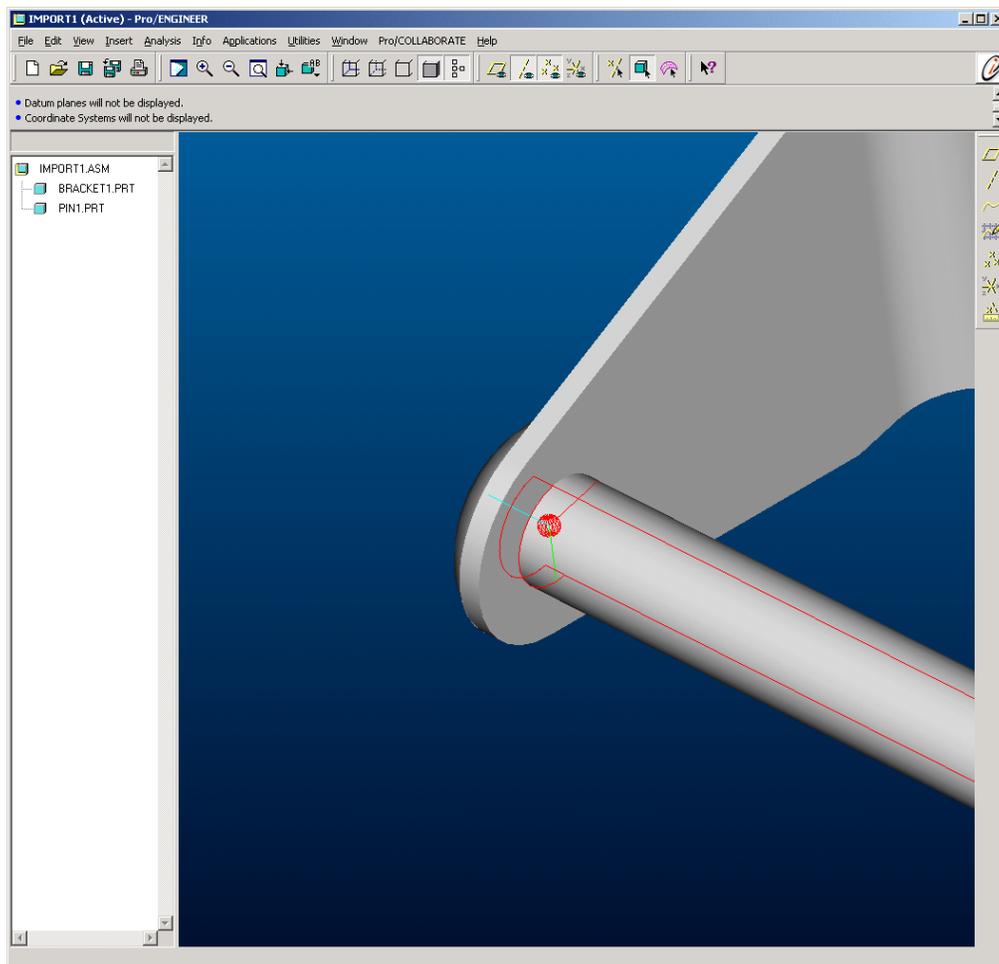
In the reality, it would be asking a great deal to never have the topology of an imported model change over time. When the topology does change, the surface identifications (id's) that were used as references in the assembly process or by other applications will most likely change making these downstream applications no longer valid. With some products (CATIA, CADD5, and Pro/DESKTOP), Pro/ENGINEER offers the Associative Topology Bus (ATB) where

these surface id's are remembered during topology changes and downstream applications should all update correctly.

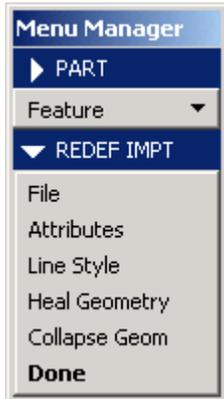
In the cases where the ATB has not been implemented yet, the user can manually map the corresponding surface id's to obtain much of the same end result. I recently had to do this when I was importing IDEAS geometry into Pro/ENGINEER for (among other things) the purpose of using CE/TOL. The following outlines possible issues with using imported geometry where surfaces are referenced by another application and how to solve the problem. Keep in mind that while this was seen with CE/TOL, the same problem could have occurred in other Pro/ENGINEER downstream applications.

The following assembly has a CE/TOL analysis already performed on it. A change has been made in IDEAS and we need that change to be reflected in CE/TOL. If we are able to maintain the joints and measurements, then the analysis will also be retained. There were just two joints placed on this model for simplicity.

The first thing that needs to be done is to interrogate the joints in the CE/TOL model. Simply by highlighting the joints one by one in the upper left window of CE/TOL, you can see highlighted the corresponding surfaces in the Pro/ENGINEER model that are used as references. See below:



These reference surfaces are the surfaces that must be picked when redefining the import feature on the respective parts.



At this time, we open each of the parts in the assembly that have had changes made to them in the original CAD system. We choose the Import Feature from the Model Tree and with the Right Mouse button, we choose Redefine.

We then choose File to bring in a new import file by browsing the harddrive.

Verify the Coordinate System.

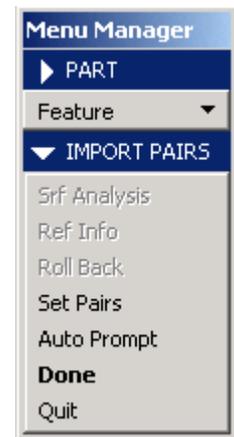


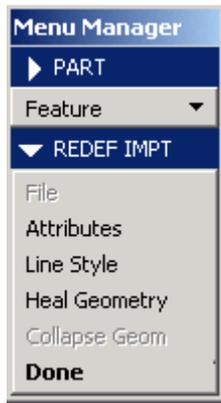
Choose Replace All for Surfaces and Done.



When the system asks if you want to accept default pairs choose "NO."

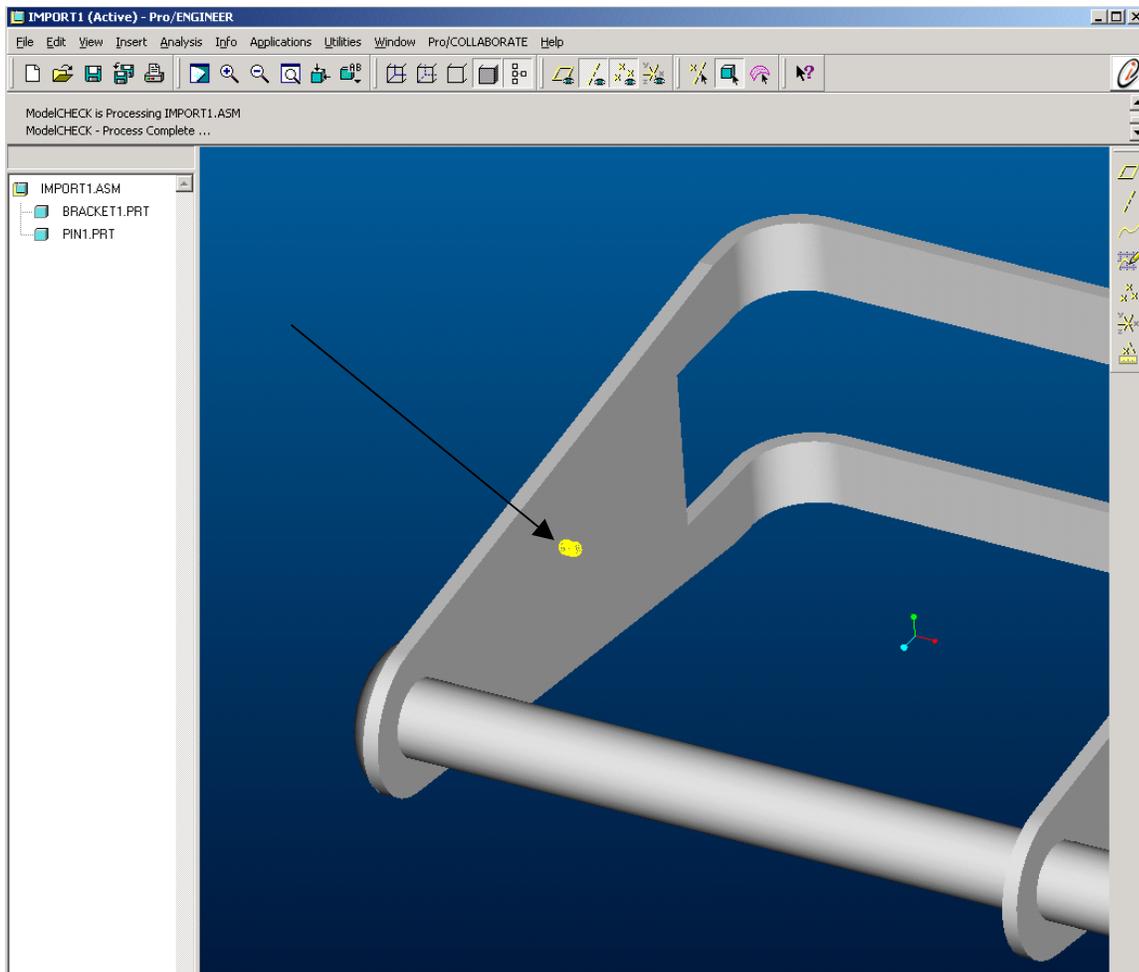
When we choose SET PAIRS, the system now allows us to pick corresponding critical surface pairs. Pro/ENGINEER now brings up a small, overlapping window containing the original part covering over the new part window. We simply pick one critical surface on the Original Part and pick the corresponding surface on the New Part (Pro/ENGINEER automatically brings forward the correct part window for surface picking). We do this for each surface pair until all critical surfaces have been mapped. Once complete, we choose Done from the Import menu.



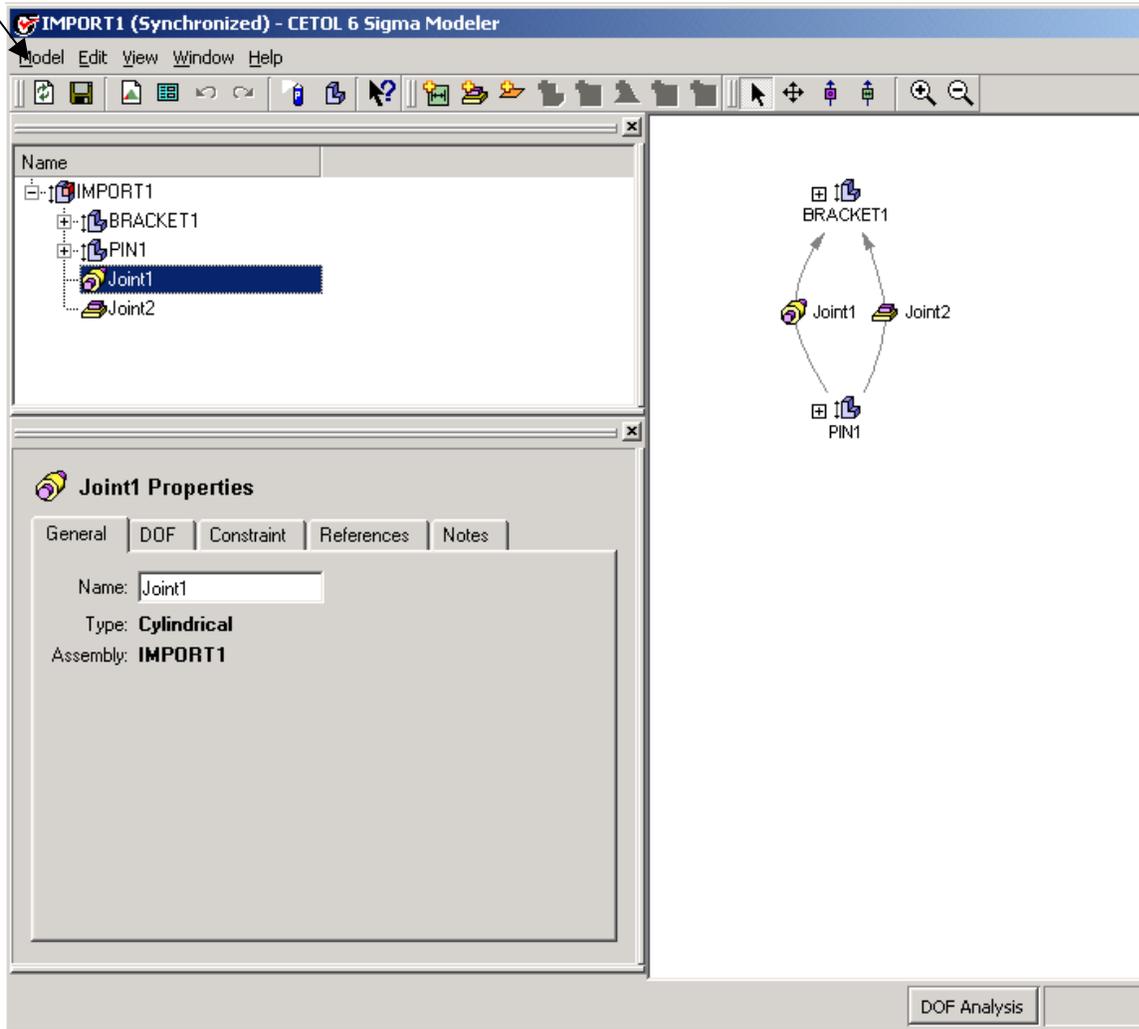


We perform this task on each and every modified part in the assembly.

After the assembly has been correctly Regenerated, we may notice that the joints are not being displayed where they should be. This is because the CE/TOL info has not been updated yet.



To update the CE/TOL model, we toggle to the CE/TOL window and choose the “Synchronize with Pro/ENGINEER” icon.



If everything was done correctly, all joints and measurements will remain and no error message will come up. The downstream application of CE/TOL has been updated to work with the new imported geometry.

[Back To Top](#)

Tips of the Week

PDMLink : Adding New Attributes to A Part

[Click Here To View](#)

[Back To Top](#)

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.

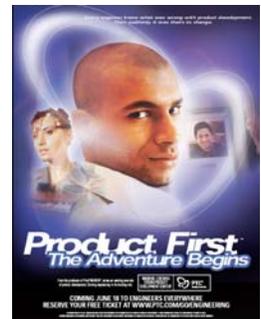
Thank-you very much for your time and feedback as we try to improve this value-added service.

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.



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, 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.

[Back To Top](#)