

PTC Product Focus: A) [What's New in Pro/ENGINEER Mechanical for Wildfire 5.0](#)

B) [Windchill QMS - CAPA](#)

Tips of the Month: A) [Creating Shell Elements in Wildfire 5.0](#)

B) [Improving the Performance When Browsing Windchill Through the Embedded Browser in Wildfire 5.0](#)

Announcements: [Most Recent Announcements](#)

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

PTC Product Focus

What's New in Pro/ENGINEER Mechanical for Wildfire 5.0

Pro/ENGINEER Mechanical for Wildfire 5.0 is another significant release packed with new functionality and usability enhancements. You'll find in this article some incredible new features that are going to make your experience with Mechanical easier and more powerful than before.

The What's New in Pro/ENGINEER Mechanical can be broken down into five categories which will be used to describe the enhancements:

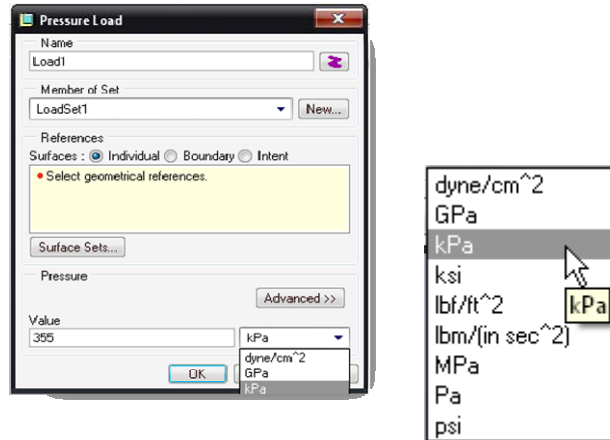
- Usability and Workflow
- New Modeling Tools and Features
- Weldment and Midsurface Modeling Enhancements
- Enhanced Solution Capability
- Post-processing and Results

The list that is in this document isn't the full list of enhancements in Mechanical. This is a list of the top enhancements that will have the greatest impact on users in Wildfire 5.0.

Usability and Workflow

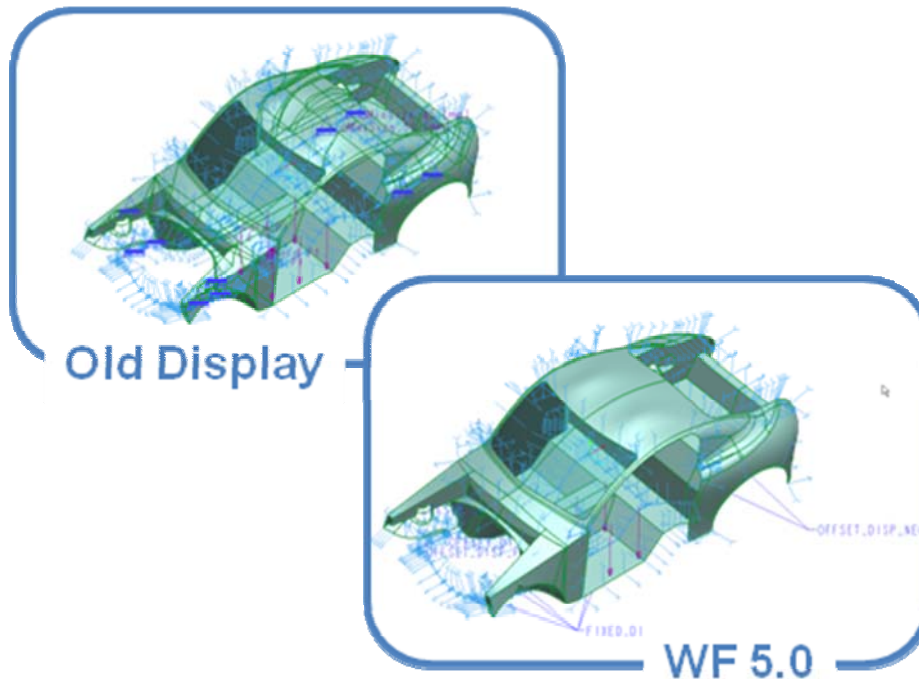
Support for Heterogeneous Units

The units for all modeling entities may be now set on the fly. There is also a built in conversion tool for converting from one unit to another. You can also see results in any type of units you wish to. This enhancement makes life much easier.



Icon and Label Display Improvements

We've added a z-buffering capability to Mechanica for display of icons that represent different aspects of the model setup. This greatly reduces visual clutter on the screen. You can also control the distribution density for different types of loads. In addition, you can now set colors individually and highlight referenced entities.



Dashboard UI for Surface and Volume Regions

We've now updated the dashboard UI for these two definitions. This includes a geometry preview, a simplified definition UI and some new options like 'surface region split by chain' and 'volume region depth options.'

Smaller enhancements for Usability and Workflow

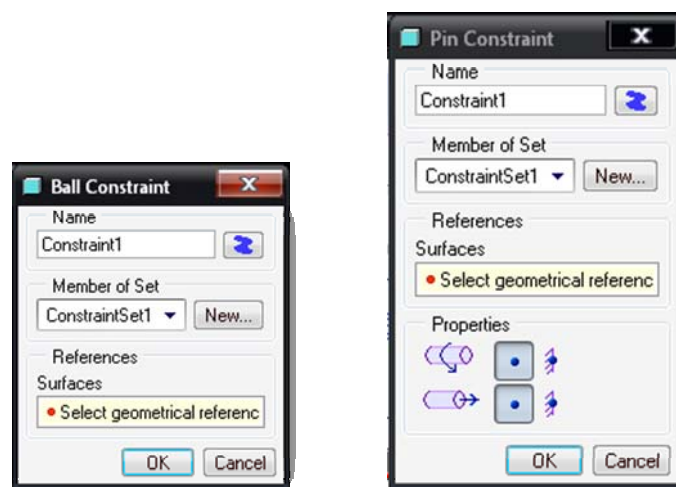
Here are additional enhancements around usability and workflow:

- Material limits consolidated and no longer coupled to specific failure criteria
- Persistent display of Csys triad
- Zoom into command available for any selected simulation entity
- New tabbed interface for displaying solution status files.

New Modeling Tools and Features

Pin, Ball and Planar Constraints

It is now easier to specify engineering constraints with these new features. They can be applied to any corresponding geometry.



Advanced Reference Selection

You now have the ability to use intent references and the ability to use datum reference features. This enables a user to 'tag' geometry for simulation entities without entering Mechanica. For example, add the internal surfaces of a valve body to a datum reference feature and in Mechanica apply a pressure load to the surfaces in this feature.

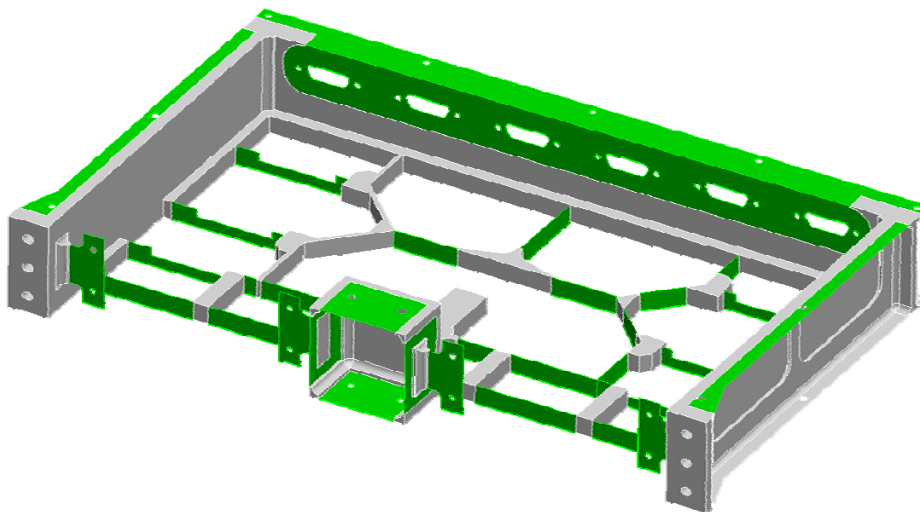
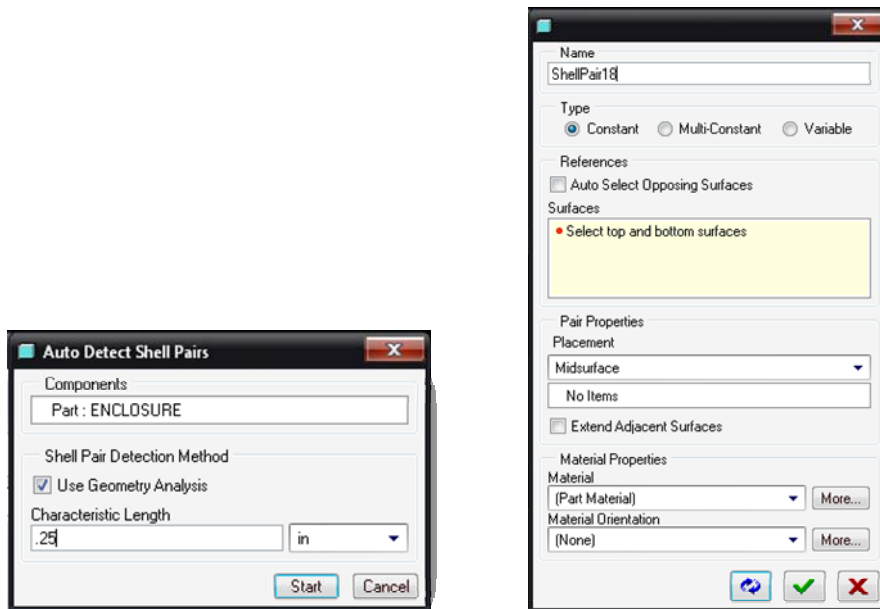
Here are additional enhancements new modeling tools and features:

- Total bearing load at a point
- Total volumetric heat load can be specified as a total load
- Reaction measures at a point
- Beam releases on curves
- Preview of imported loads

Weldment and Midsurface Modeling Enhancements

New Shell Pair Definition Workflow

Wildfire 5.0 has an updated and streamlined UI and workflow for creating shell elements. You now have access to a quick repeat and automated selection of opposing surfaces. This includes a 'geometry based' shell pair auto-detection and not a modeling method detection like previous releases. In addition, the compression of pairs has a tangent surface option. This works for both assembly and part modes. You'll find this functionality very useful you'll be amazed how much time you'll save.



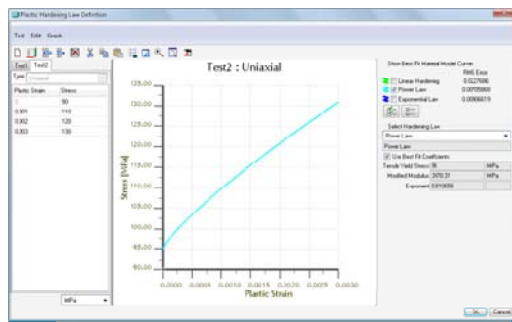
What about my Pro/WELD information?

Now with Wildfire 5.0, you can reuse all your Pro/WELD work that you did in assembly mode. You simply select the Pro/E welds that you'd like to include and they are then added into your assembly as shell elements. You can also define separate material properties for the weld features.

Enhanced Solution Capability

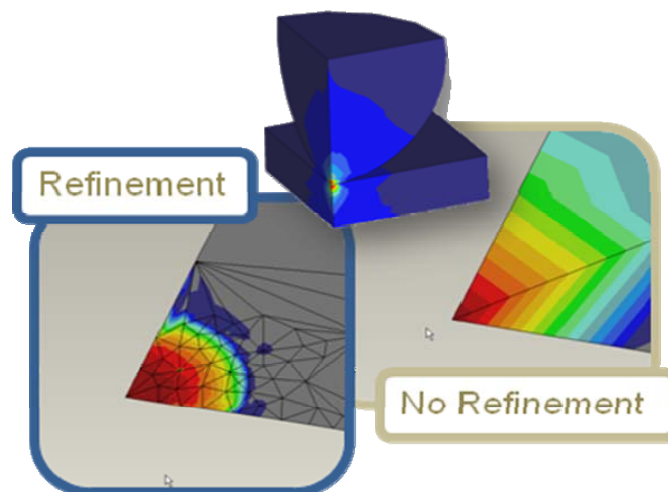
Nonlinear Elasto-plastic Materials

For the first time ever, Pro/MECHANICA has entered into the material plasticity world with new functionality in Wildfire 5.0! We can now support nonlinear elasto-plastic materials. The three most common material laws can be applied, which are linear hardening, power law and exponential law. The material definition window is very easy to use and has material fitting capability so that it selects the most appropriate material law or allows you to select the one you wish to use. You can now understand the post-yield behavior and both loading and unloading are supported.



Greater Solution Accuracy with:

- New algorithm for mesh refinement in contact
- Convergence/Error Plots for Single Pass Adaptive (SPA)
- Convergence target selection for SPA
- Large deflection Analysis now supports rigid links



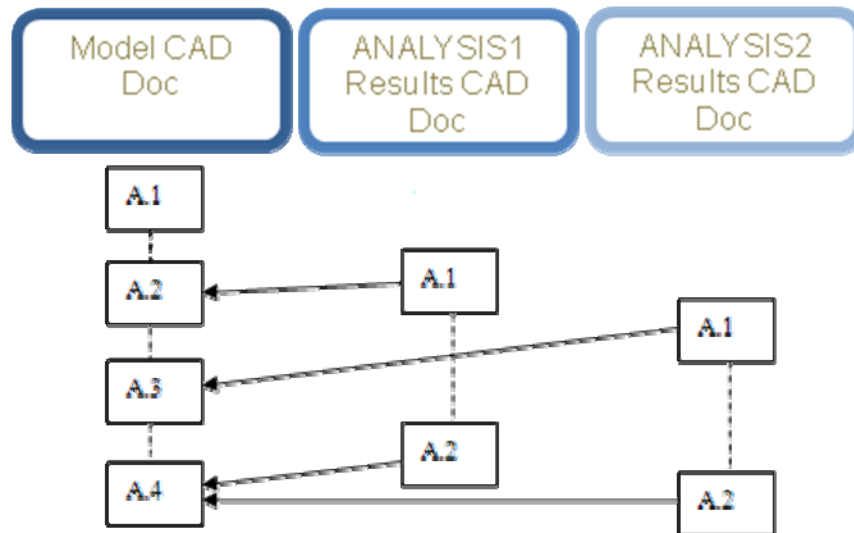
Post Processing and Results

Many additional capabilities for post processing:

- Principle stress vector results
- Relative paths for the .rwd file
- Beam thickness display control (adds much more realism to beam results)

Mechanica results in PDMLink

Both 'raw' study directories as well as HTML reports are supported in PDMLink. A new option has been added in Wildfire 5.0 in the RUN dialog box which enables you to 'vault results.' This creates a zip archive of the study directory and moves it to the workspace as a separate document. It references the last checked in version of the model as a derived link. You can browse to these files directly. The HTML reports are also auto associated to the model in session. The HTML report is saved to the workspace upon save. The following image helps you understand the link of the results with the corresponding files:



[Back To Top](#)

PTC Product Focus

Windchill QMS - CAPA

[Click Here To View](#)

[Back To Top](#)

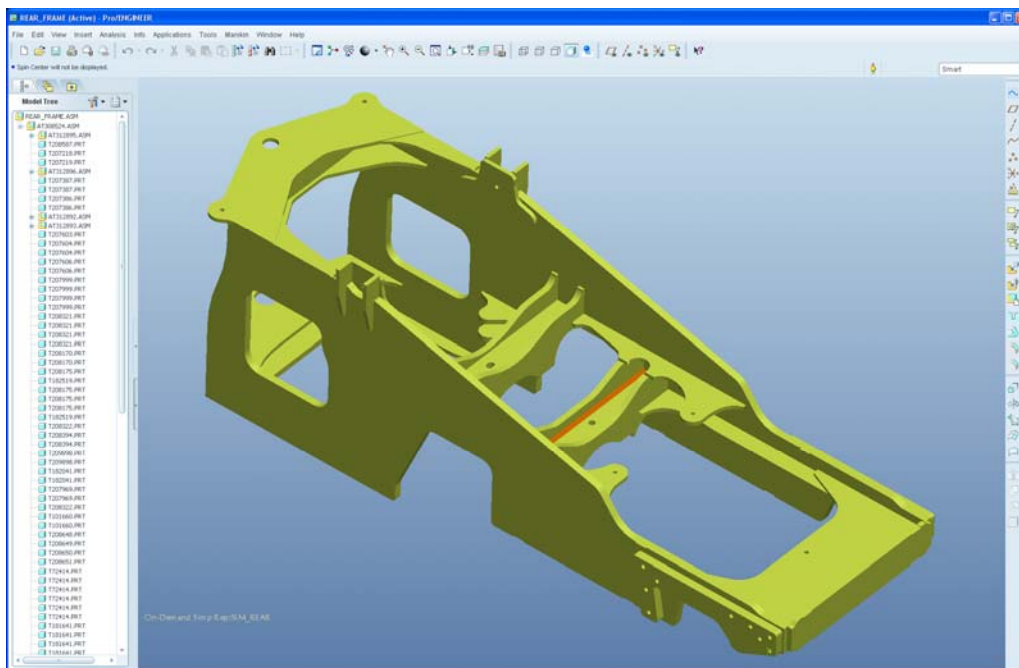
Tips of the Month

Creating Shell Elements in Wildfire 5.0

This is one of the biggest time savers in all of Pro/ENGINEER Wildfire 5.0, if not the biggest! Creating shell elements manually has always been a tedious task when the geometry was imported or you were not able to auto pair most of an assembly. But the fact remained, you had to pair everything at the part level and could not do any of the work at the assembly level. Well, not anymore! You can do the entire assembly at the assembly level with just a few mouse clicks. And you can create these 'geometry based' pairs in either part or assembly level. Here is how this is done and how you can save yourself hours of work by creating shell elements in assembly mode!

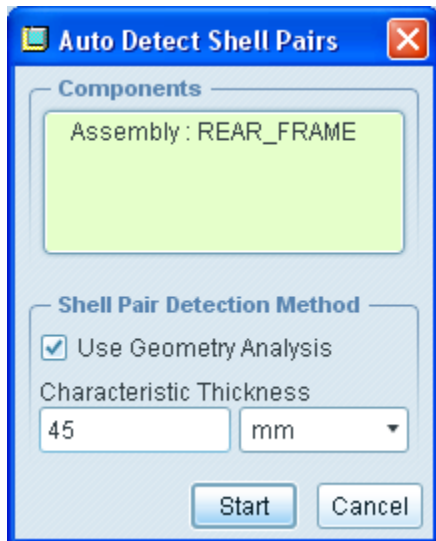
Step 1 – Open the Assembly and enter Mechanica Mode

Open any assembly inside of Pro/ENGINEER. We'll use this weldment as shown. After you open it, go into Mechanica by selecting APPLICATIONS>MECHANICA.



Step 2 – Create the shell pairs

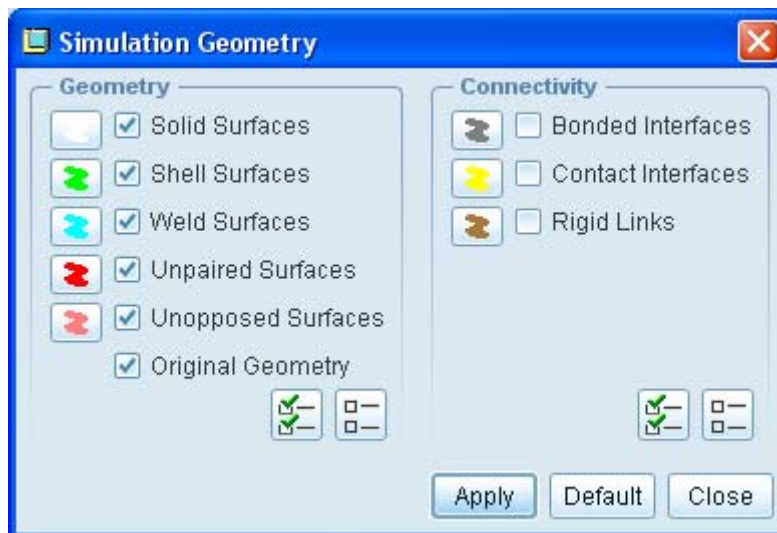
You need to understand the thickness of the geometry that you are dealing with. Go ahead and make some measurements to understand the thickness of the plates or geometry that you are working with. This is simply done with the Analysis>Measure menu. Next thing to do right now is click on INSERT>MIDSURFACE>AUTO DETECT SHELL PAIRS. The following dialog box appears:



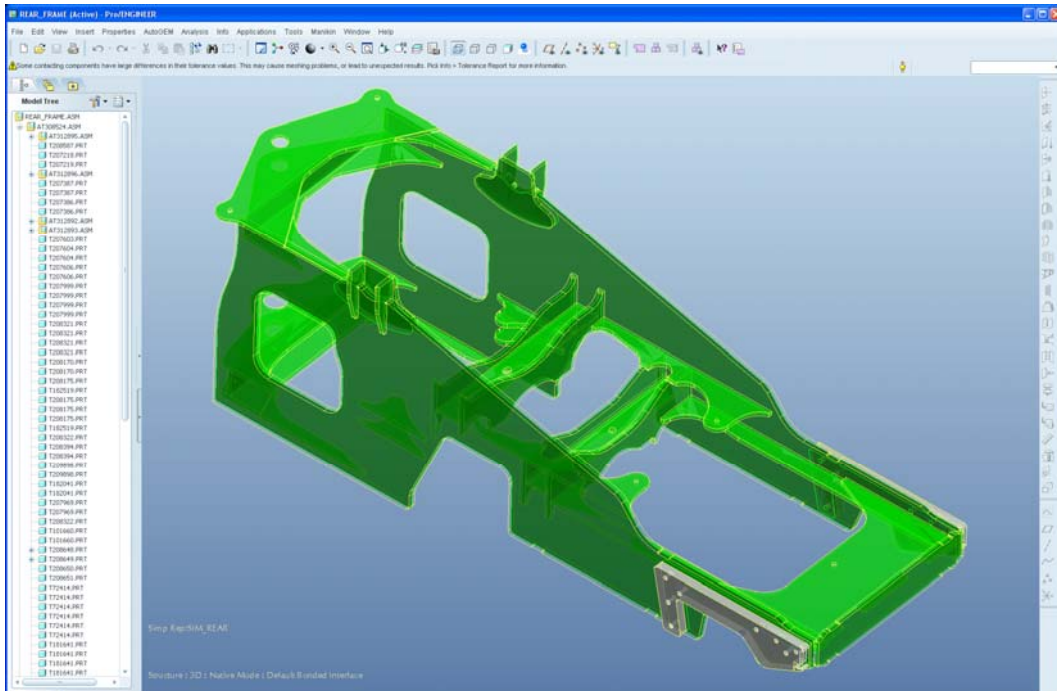
Select the top level assembly node or the assembly name that you want to mesh from the model tree. Enter the value of the thickness you'd like to pair geometry based on. Mechanica will now pair any geometry that is equal or less than this value. Then click START! In just a second, all your pairs are done! Really, they are complete! In this example, 34 shell pairs were created.

Step 3 – Review the shell pairs

Next, let's see how we did! Select AUTOGEM>REVIEW GEOMETRY. This is a new dialog box in Wildfire 5.0.

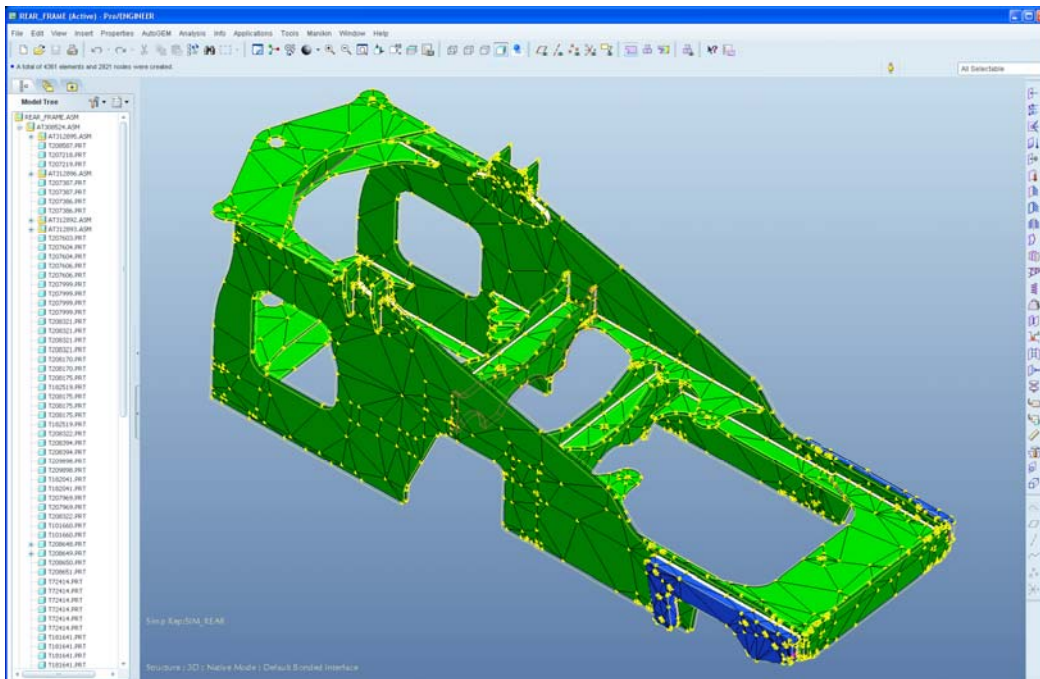


Make certain that 'Shell Surfaces' is selected. After you see this dialog box, select 'APPLY' to review your shell surfaces. You'll be prompted that this may take a considerable amount of time, but just say OK to that dialog box. Now take a look at the shell surfaces – this is a huge time saver! After you've reviewed the shell pairs, close the REVIEW GEOMETRY dialog box.



Step 4 – Review the mesh

If you have material properties already applied to your assembly, then you can mesh your model. This step is just to verify that you have valid shell elements and to get an idea of how many elements you are dealing with when you solve your analysis later on. Click on the ‘Create Elements’ icon or select **AUTOGEM>CREATE**. Then click on the ‘Create’ button in the dialog box. After a few seconds, my assembly now looks like this:



Wow, from start to finish, this is a 2 minute process. Compare this to the process in previous revisions where it would take a substantially more time to get this far.

I hope you enjoyed this tutorial and can apply it to your use of Mechanical!

[Back To Top](#)

Tips of the Month

Improving the Performance When Browsing Windchill Through the Embedded Browser in Wildfire 5.0

[Click Here To View](#)

[Back To Top](#)

Announcements

The PTC/USER Portal is getting an updated, fresh user interface.



The functionality is remaining the same but the site is being redesigned to allow for easier navigation by users.

You will also begin to see blog articles being posted by the PTC/USER Board of Directors highlighting different aspects of your community and other relevant info.

The four major areas in which our site design has been updated are:

1. Redesigned Header Area
2. Simplified Navigation
3. New Site Wide Right Side Bar
4. Other Design Elements

For more details on these changes you can read about them in the [What's New](#) guide.

Enjoy the redesigned site and we look forward to your feedback.

Evan Caille

PTC/USER Board of Directors

PTC Tips & Techniques Newsletter Archives

Did you 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.

[Customer Care Zone](#)

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!](#)

Special Hardware offers for PTC Customers

- <http://www.hp.com/go/ptc>
- <http://www.hp.com/go/ptcworkstation>

PTC Sponsored Events

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

Explore what is new with the Pro/ENGINEER Wildfire family!

<http://www.ptc.com/go/showcase>

Connect with PTC using the latest Social Networking resources:

facebook



LinkedIn

YouTube

Also visit <http://social-product-development.blogspot.com/>

[Back To Top](#)

Upcoming Events & Training Class Schedules

Upcoming, 2010 Your local Pro/Engineer User Groups
<http://www.ptcuser.org/rugs/>

June 12 – 15, 2011 Las Vegas, NV 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.

[You're Invited to Attend...](#)

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.

[Back To Top](#)