Customer PTC E-Newsletter

Date: 6/25/03

PTC Product Focus

Pro/Engineer Plastic Advisor

Plastic Advisor, an add-on to Pro/ENGINEER, simulates the plastic filling process for injection-molded parts so engineers can create perfect parts time after time. By just choosing a material type and proposed injection locations, Plastic Advisor provides an on-screen animation of the plastic fill process. Engineers can see temperature and pressure gradients, sink mark locations, heat concentrations and guidance on the overall manufacturability of the design. Plastic Advisor will also help you see potential problems and suggested corrections to your design.

Capabilities:

- Generate Web-based reports automatically within Pro/ENGINEER Wildfire embedded browser
- Access fully defined library of common plastic materials
- Select automatically from typical injection-molding machine parameters
- Identify optimal injection locations through material property and process parameters
- Pro/ENGINEER Plastic Advisor makes it possible to do more simulations in less time because achieving accurate flow simulation does not require the creation of time-consuming and difficult mid-plane geometry from complex solid models.

Benefits:

- Reduced late cycle changes
- Minimized manufacturing cycle times and material costs
- Elimination of cosmetic defects

Below, you will see examples of the Plastic Advisor workflow and also screen shots and descriptions of all of the results. For more information on Plastic Advisor, please visit HERE.

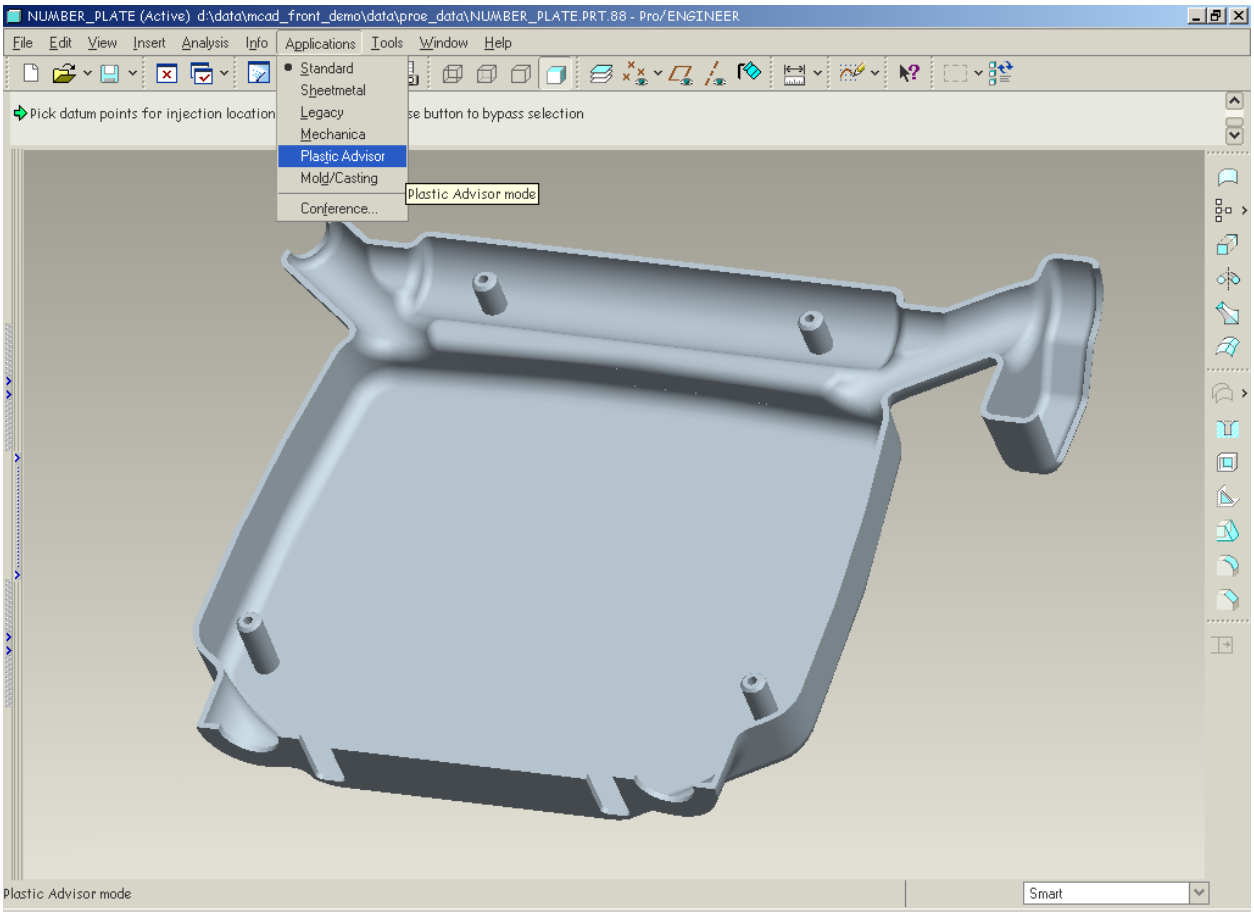


Figure 1: Pro/ENGINEER Wildfire Model

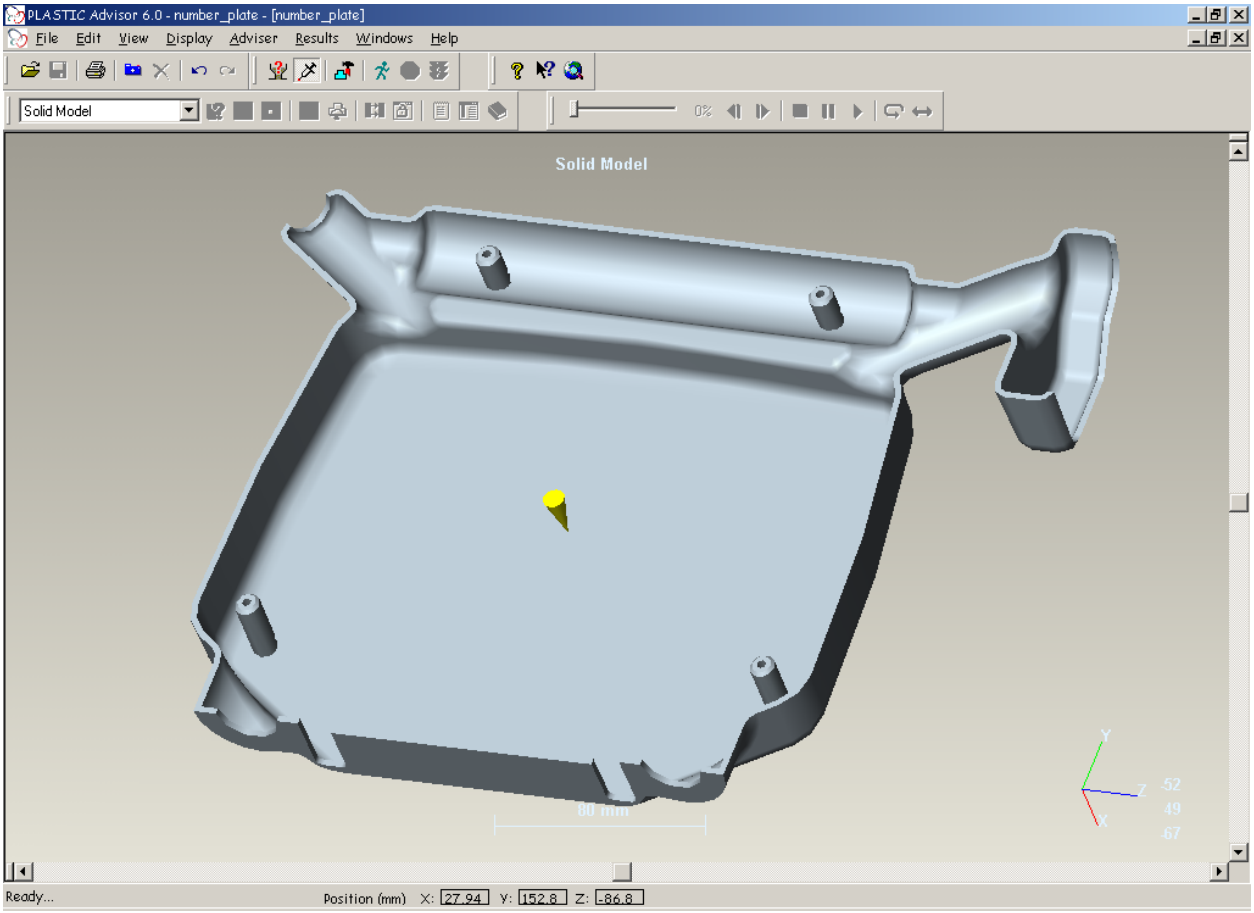


Figure 2: Pro/ENGINEER Plastic Advisor Model with Gate Location Defined

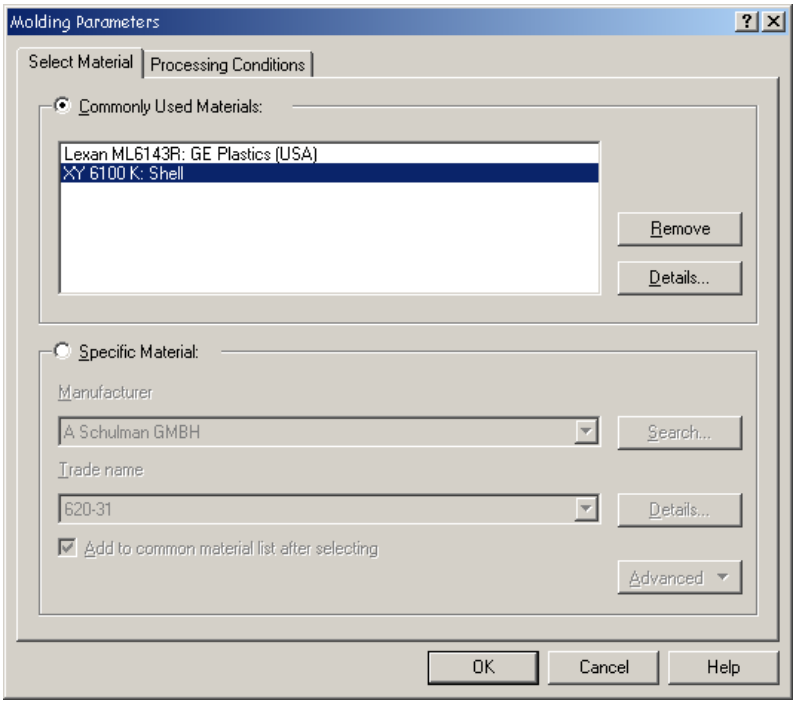


Figure 3: Material Database

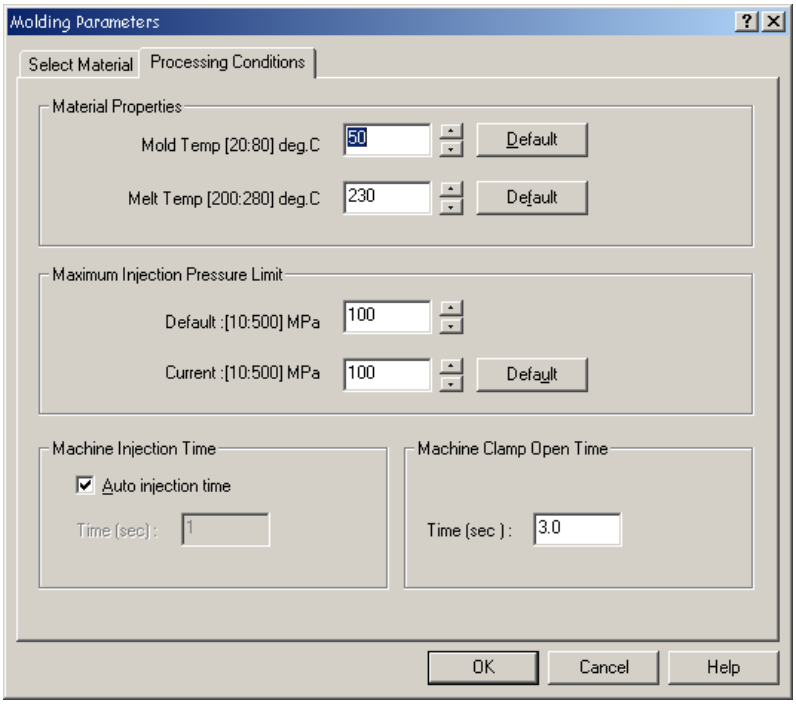


Figure 4: Control of the Processing Conditions

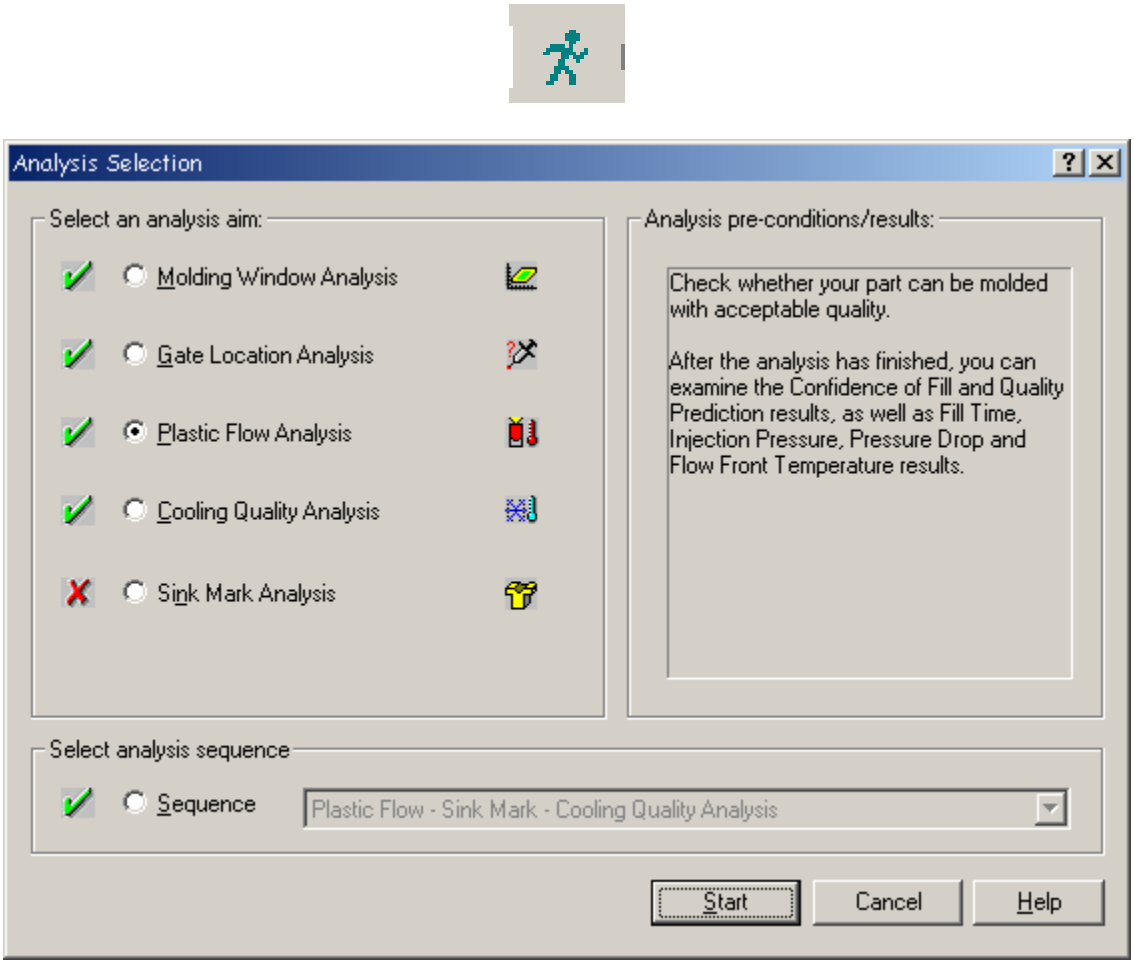


Figure 5: Analysis Options

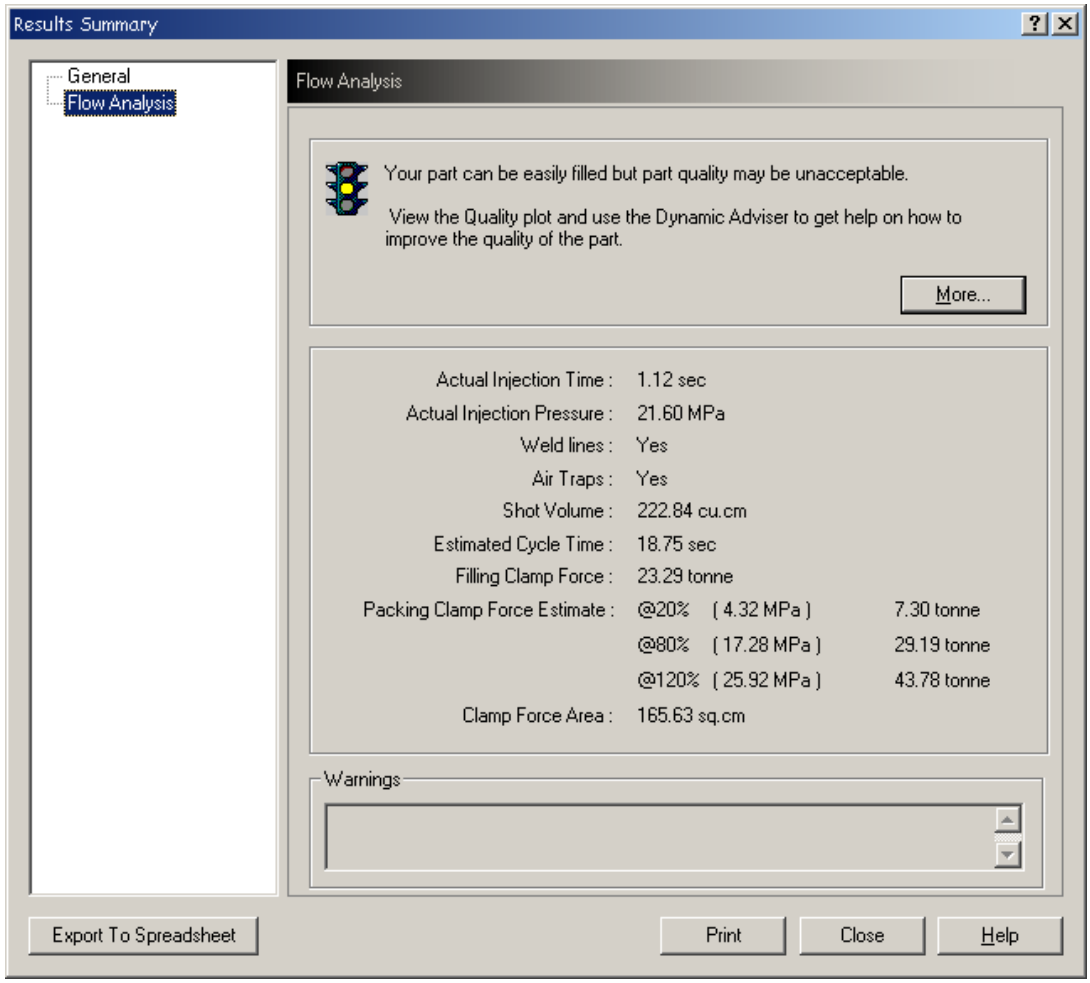


Figure 6: Results Summary

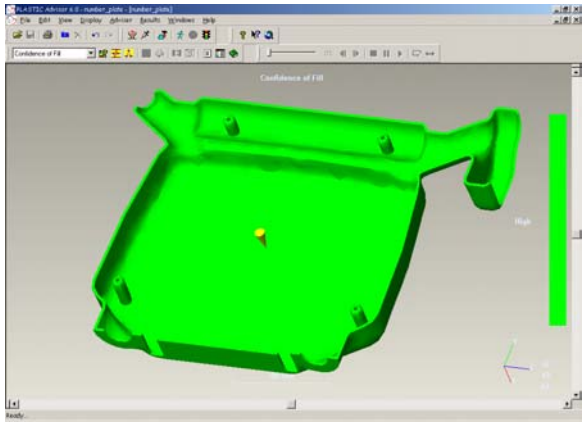


Figure 7: Confidence of Fill Results

The confidence of fill result displays the probability of a region within the cavity filling with plastic at conventional injection molding conditions. This result is derived from the pressure and temperature results.

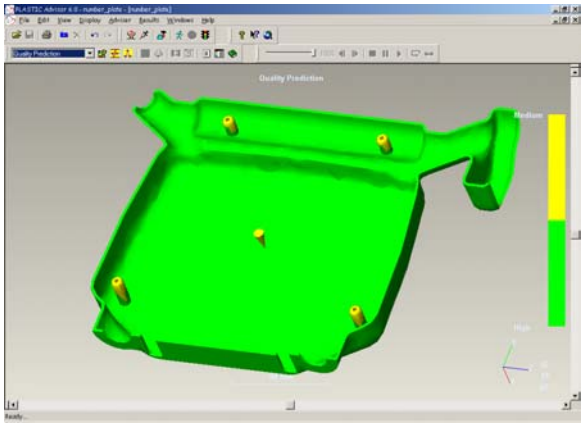


Figure 8: Quality Prediction Result

The Quality display is derived from combinations of the five results listed below. These five results are each divided into ranges - unacceptable (red), acceptable (yellow) and preferred (green). The five results are:

- flow front temperature
- pressure drop
- cooling time
- shear rate
- shear stress

For each area of the cavity, the five results are evaluated. If all five results in an area are acceptable, the area is green. If there is at least one unacceptable result, the area is red. If there are both acceptable and preferred results, the area is yellow.

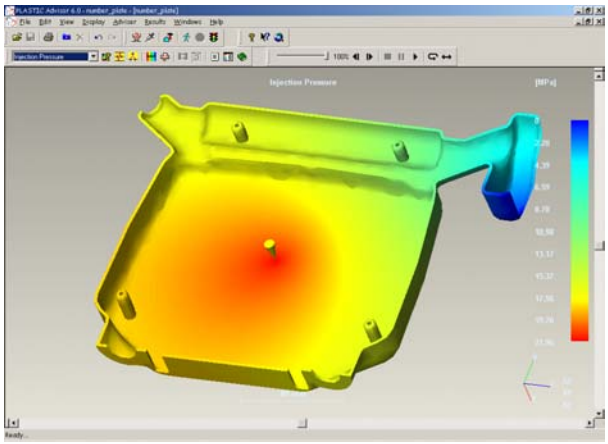


Figure 9: Injection Pressure Result

The injection pressure result uses a range of colors to indicate the region of lowest pressure (colored blue) through to the region of highest pressure (colored red). The color at each place on the model represents the pressure at that place on the model, at the moment the part is filled

completely. This is a 'snapshot' result, that is, it shows the pressure through the whole part at the end of fill.

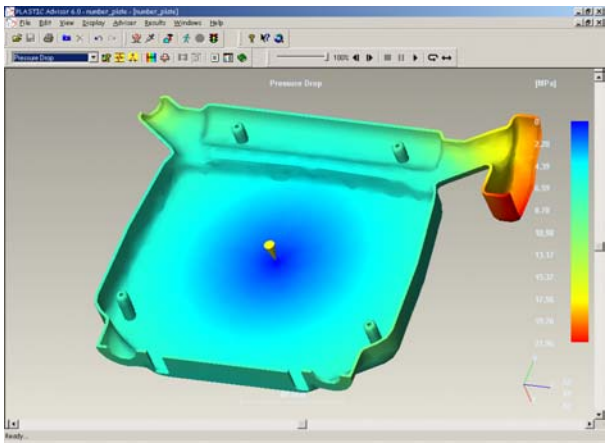


Figure 10: Pressure Drop Result

The pressure drop result uses a range of colors to indicate the region of highest pressure drop (colored red) through to the region of lowest pressure drop (colored blue). The color at each place on the model represents the drop in pressure from the injection location to that place on the model, at the moment that place was filled. That is, the pressure required to force material to flow to that point.

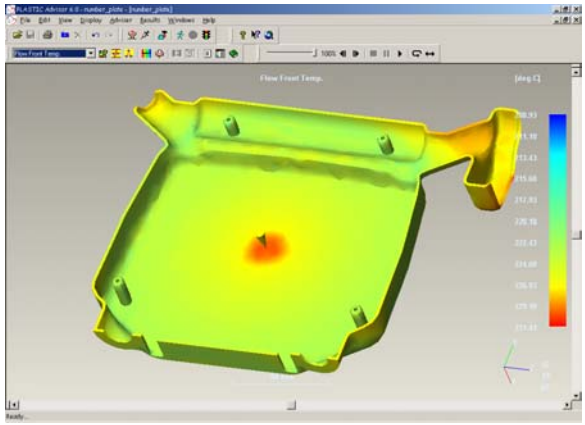


Figure 11: Flow Front Temperature Result

The flow front temperature result uses a range of colors to indicate the region of lowest temperature (colored blue) through to the region of highest temperature (colored red). The colors represent the material temperature at each point as that point was filled. The result shows the changes in the temperature of the flow front during filling.

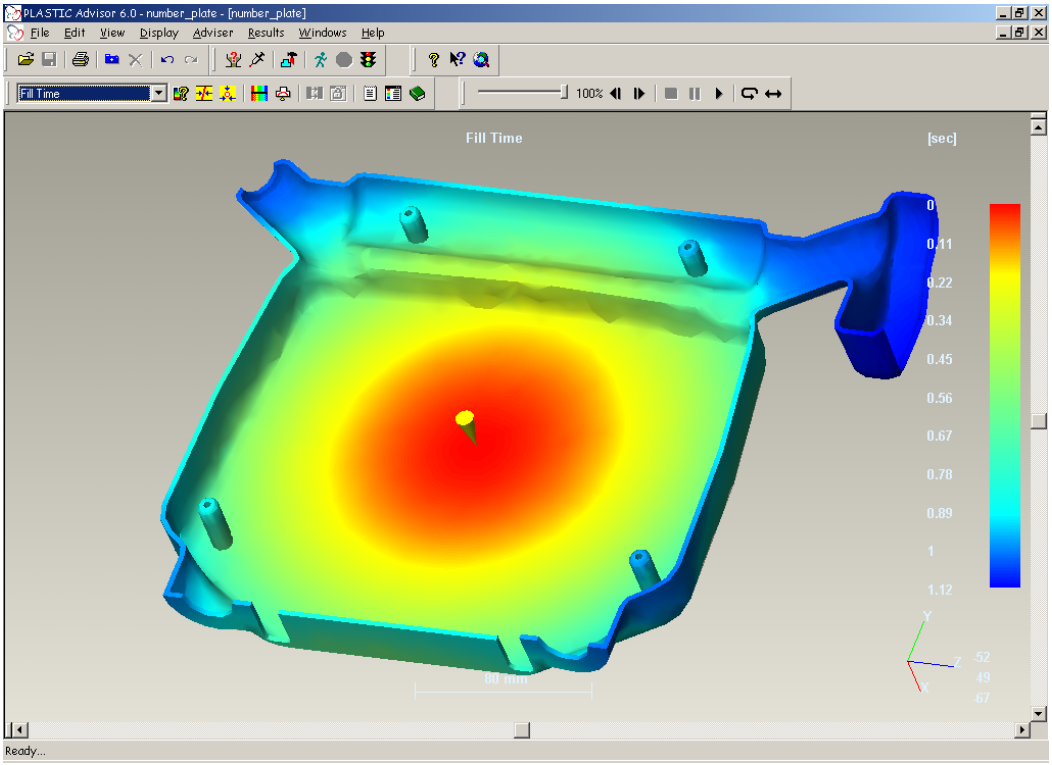


Figure 12: Fill Time Report

This result shows the flow path of the plastic through the part by plotting contours which join regions filling at the same time. These contours are displayed in a range of colors from red, to indicate the first region to fill, through to blue to indicate the last region to fill. A short shot is a part of the model that did not fill, and will be displayed as translucent. By plotting these contours in time sequence, the impression is given of plastic actually flowing into the mold.

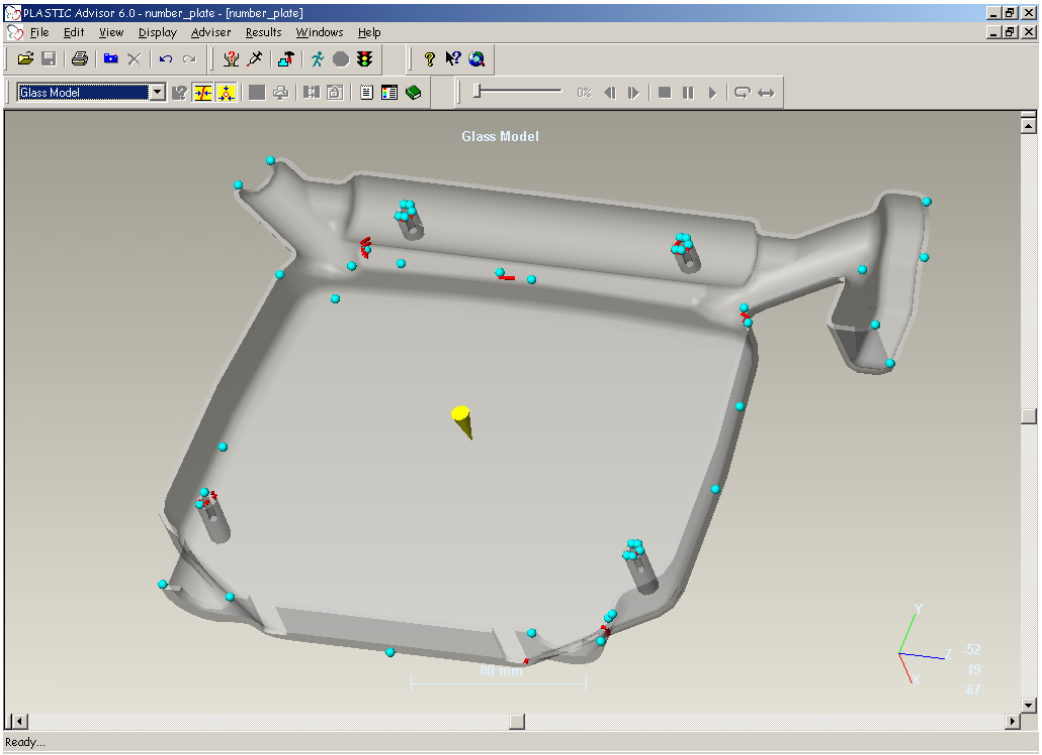
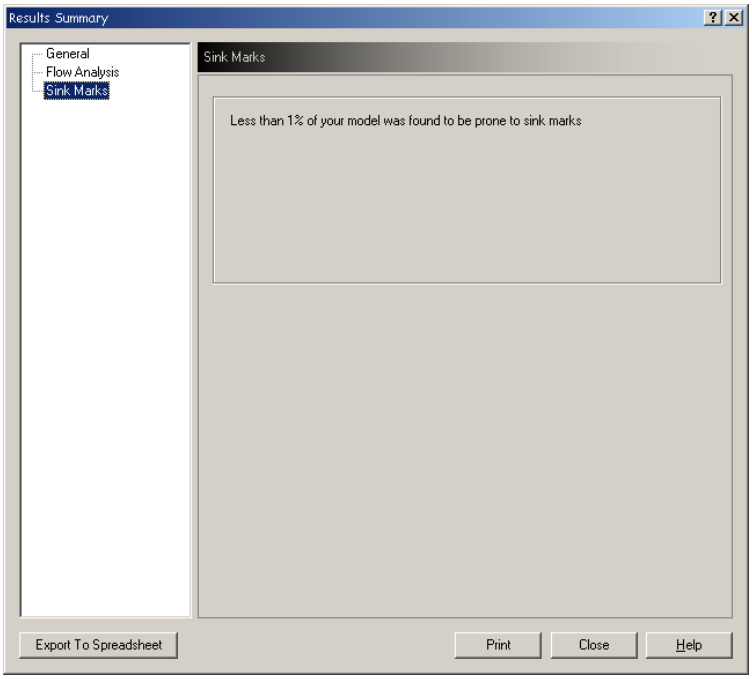


Figure 13: Weld Lines and Air Traps

Air Traps - The result shows the regions where the melt stops at a convergence of at least 2 flow fronts or at the last point of fill, where a bubble of air becomes trapped.

Weld Lines - This result indicates the presence and location of weld and meld lines in the filled part model.



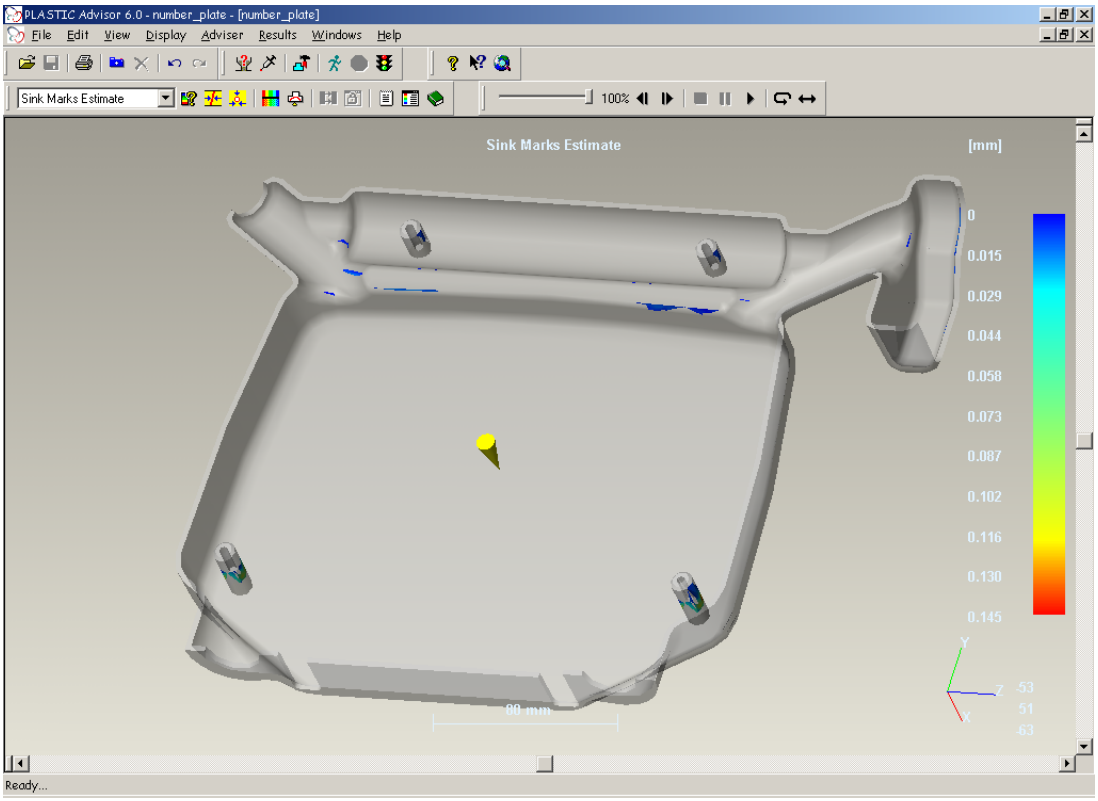
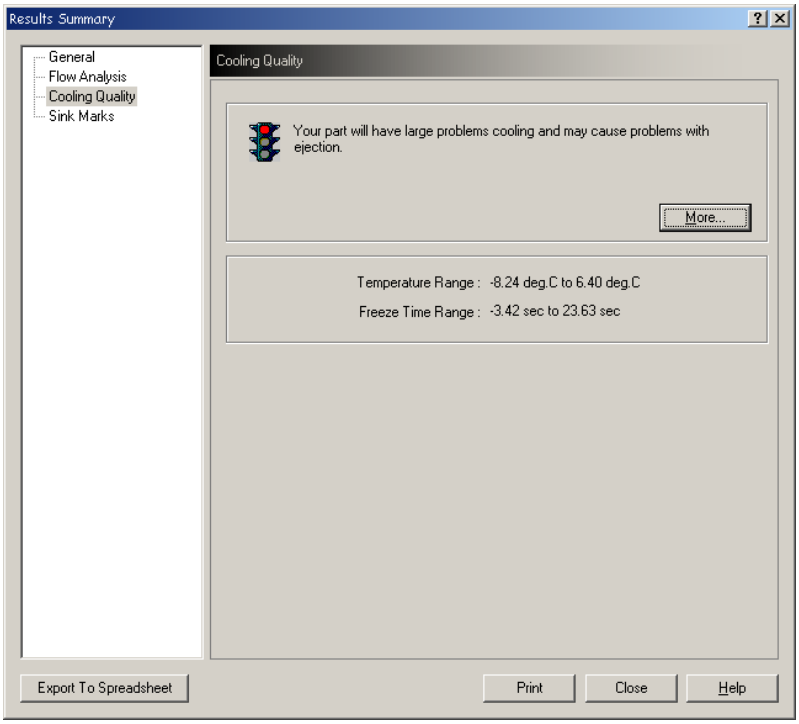


Figure 14: Sink Mark Analysis Results

This result indicates the presence and location of Sink Marks (and Voids) likely to be caused by features on the opposite face of the surface. Sink Marks typically occur in moldings with thicker sections, or at locations opposite ribs, bosses or internal fillets. The result does not indicate Sink Marks caused by locally thick regions.



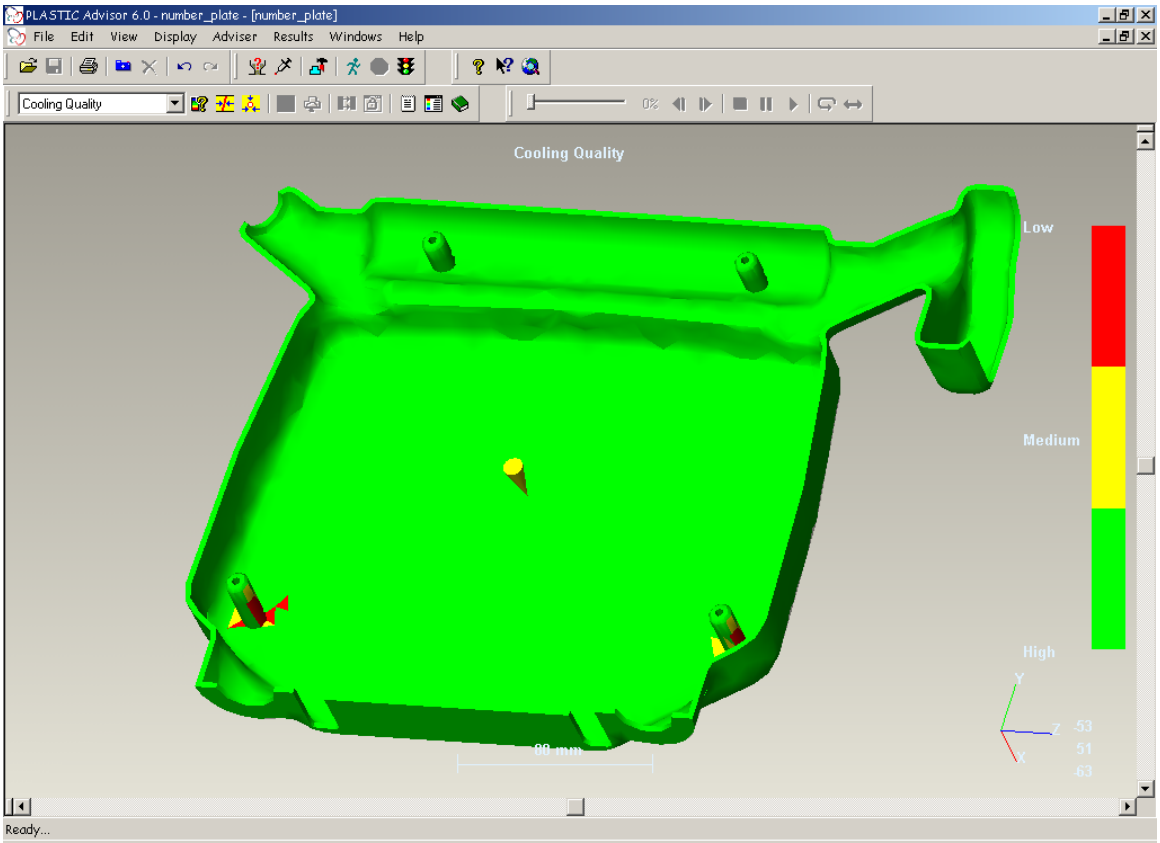


Figure 15: Cooling Quality Results

The Cooling Quality Analysis Adviser highlights areas in which the freeze time variance is greater than normal.

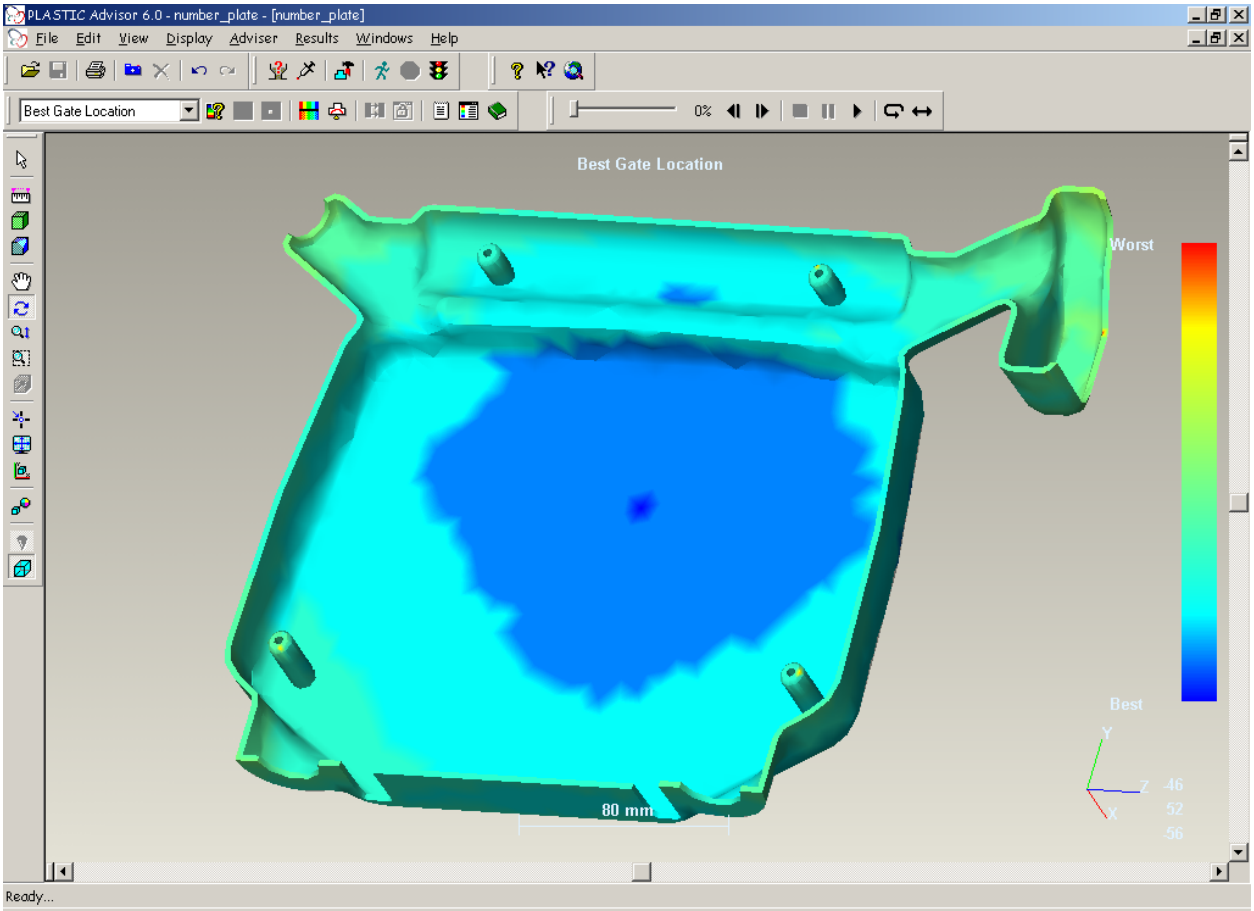


Figure 16: Best Gate Location Results

The Gate Location result rates each place on the model for its suitability for an injection location. The most suitable areas, colored blue, are rated as best, and the least suitable areas of the model, colored red, are rated as worst. A rating of blue (best) does not necessarily mean that the part can be filled from this location.

Back To Top

PTC Product Focus

ProductView Realizer : Constraint Concepts

Click Here To View

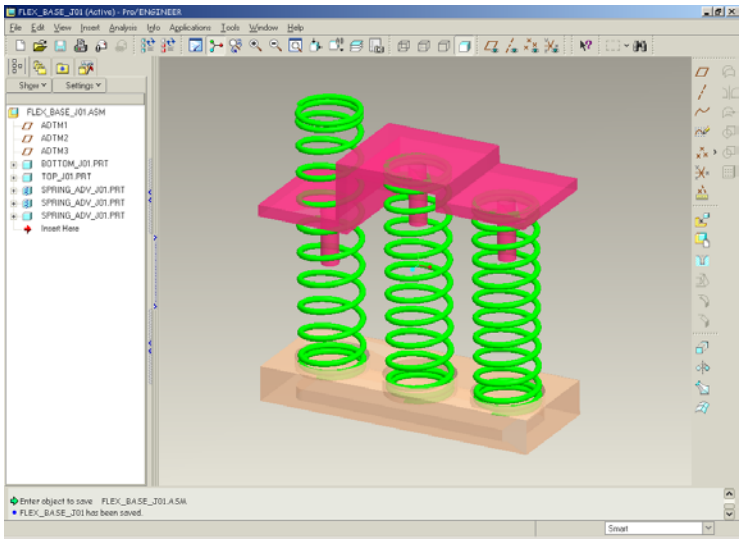
Back To Top

Tips of the Week

How to Use Flexible Components in Wildfire

Flexible components allow a model to be presented within the assembly in different states. For example a spring can be used in different compression conditions. The solid model geometry and structure of the reference model is not modified when a component is flexible. You can define flexible components:

- While placing the component
- After placing the component



Varied items can be predefined for any standard flexible part or assembly. The definition can be used each time you place the component or assembly. You can define the following varied items of the component to become flexible:

- Values for dimensions, tolerances, and parameters
- Suppress or resume state of features and components (for subassemblies)

Flexible components contain the following properties:

- Flexible component name remains the same as the original component even though the shape or structure is different.
- Flexible components refer to the same original model. The original model must be present in the Pro/ENGINEER session.
- Common properties are items not selected as varied. These common properties are associatively shared between the original model and all related

flexible components. Modifying a common property of a flexible component means modifying the original model.

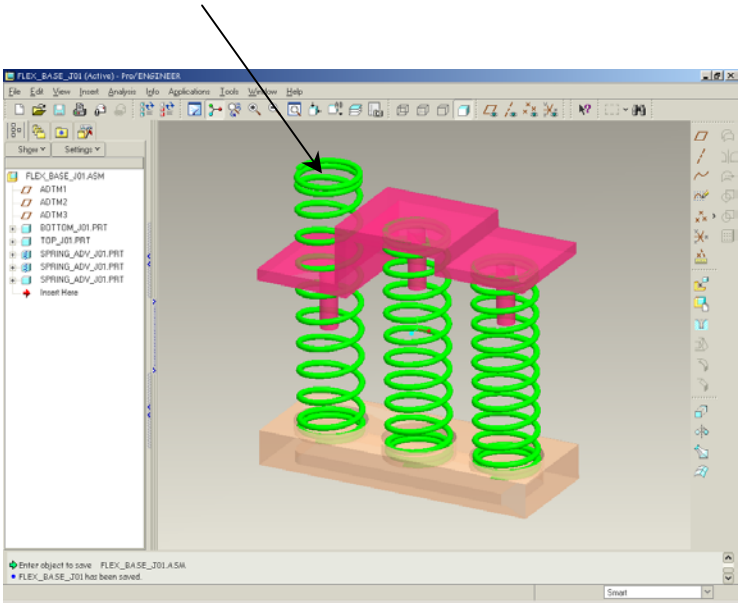
- Creating or modifying a varied item affects the flexible component, not the original model.
- Varied dimensions can be associated with a measurement in the assembly context.
- Varied items can be driven by an assembly relation, program, or family table using corresponding associated parameters.
- Varied items of a flexible subassembly can directly affect components at any level of the subassembly. Such components are known as "Affected by Flexible".
- Children of flexible components and "Affected by Flexible" components are known as "Driven by Flexible".
- When auto placing single components with predefined flexibility using drag and drop from a file browser, the component is placed automatically without prompting for flexible definition. The component must be redefined to add flexible attributes. Additionally, auto placement of multiple components does not work with flexible definition during placement.

Note: Flexible parts or parts and assemblies driven or affected by flexible components are identified in the Model Tree with .

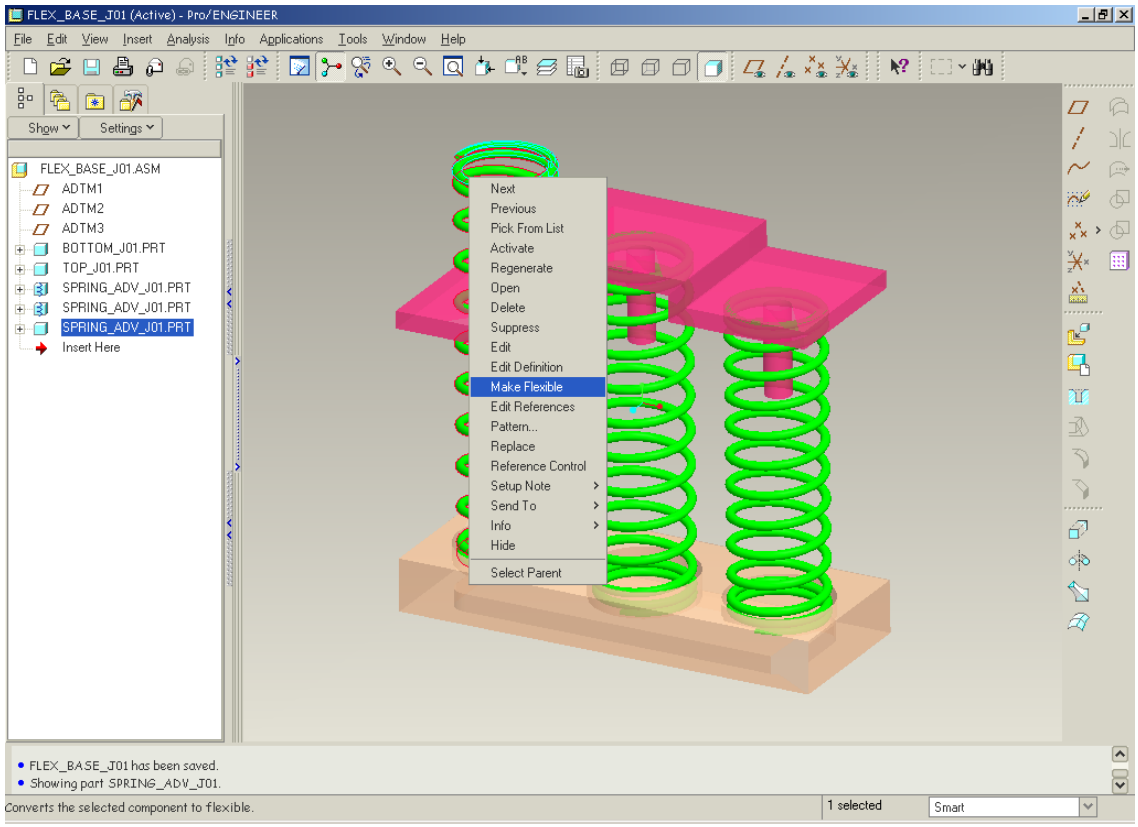
Example

Let's walk through an example of how to define a spring as a flexible component. And we will also leverage Behavioral Modeling functionality to build in even more intelligence to the spring to determine whether or not it has reached its maximum compression height.

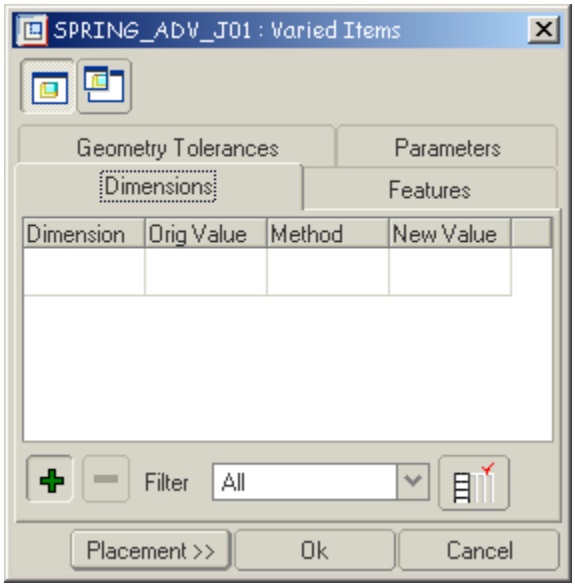
In this assembly, we will make the left most spring flexible by driving the profile height of the spring to fit within the top and bottom components in the assembly.



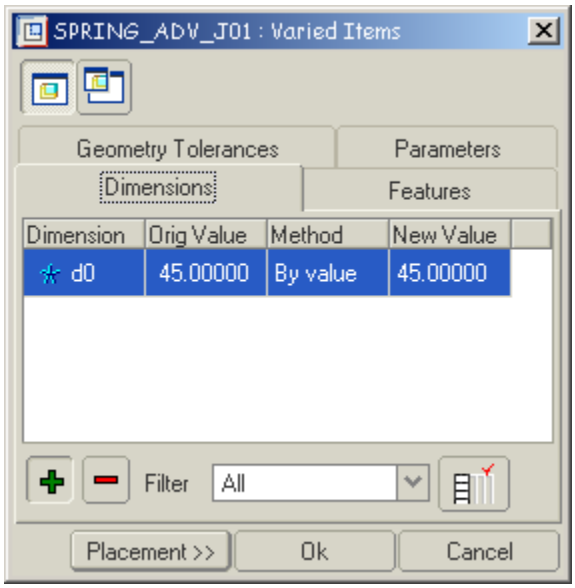
First, select the part with the left mouse button. After the part is selected, hold down your right mouse button to display the pop up menu and select Make Flexible.



The varied items dialog box will pop up, which will allow you to start the selection of dimensions, parameters, tolerances and features to manipulate for this flexible component.



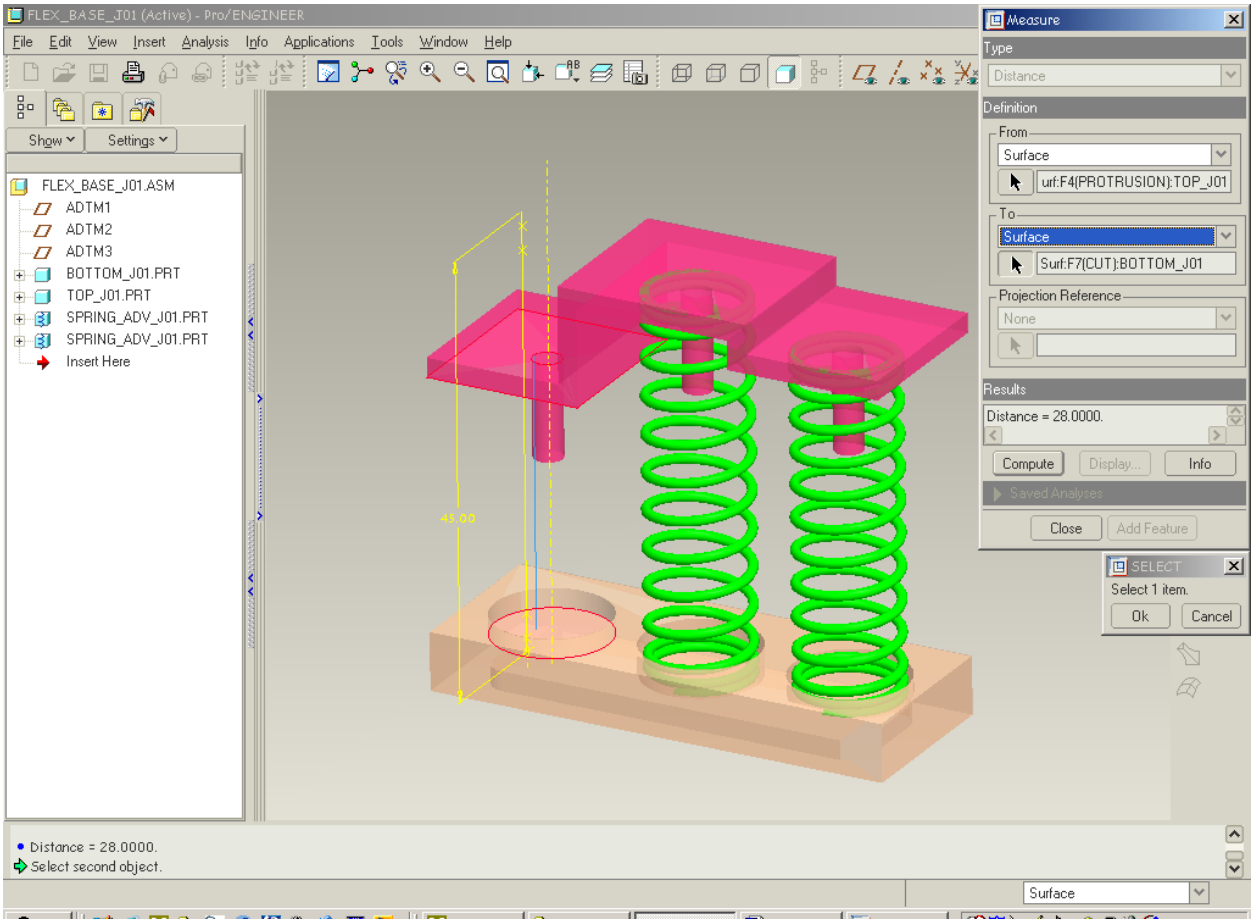
Select the green add button and select the dimension in the spring that will allow it to be compressed. In this case, we will select the profile height dimension of the spring. The selected dimension will then appear in the varied items dialog box.



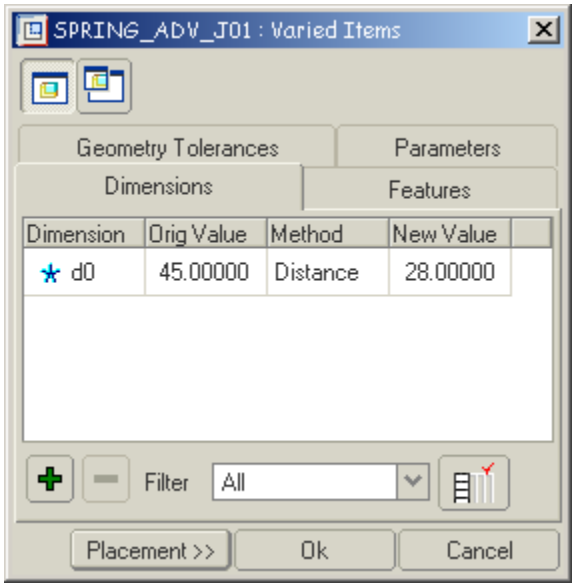
Now we can determine what method we want to use to drive the new value of the spring profile height. Here are the available options that you have.

- **By Value** (default) – Enter the new value in the New Value cell.
- **Curve Length** – Opens the **Measure** dialog box. Select a curve or edge, which drives the new value.
- **Distance** – Opens the **Measure** dialog box. Select two entities, which drives the new value.
- **Angle** – Opens the **Measure** dialog box. Select two entities, which drives the new value.
- **Area** – Opens the **Measure** dialog box. Select that surface which drives the new value.
- **Diameter** – Opens the **Measure** dialog box. Select a round surface or edge, which drives the new value.

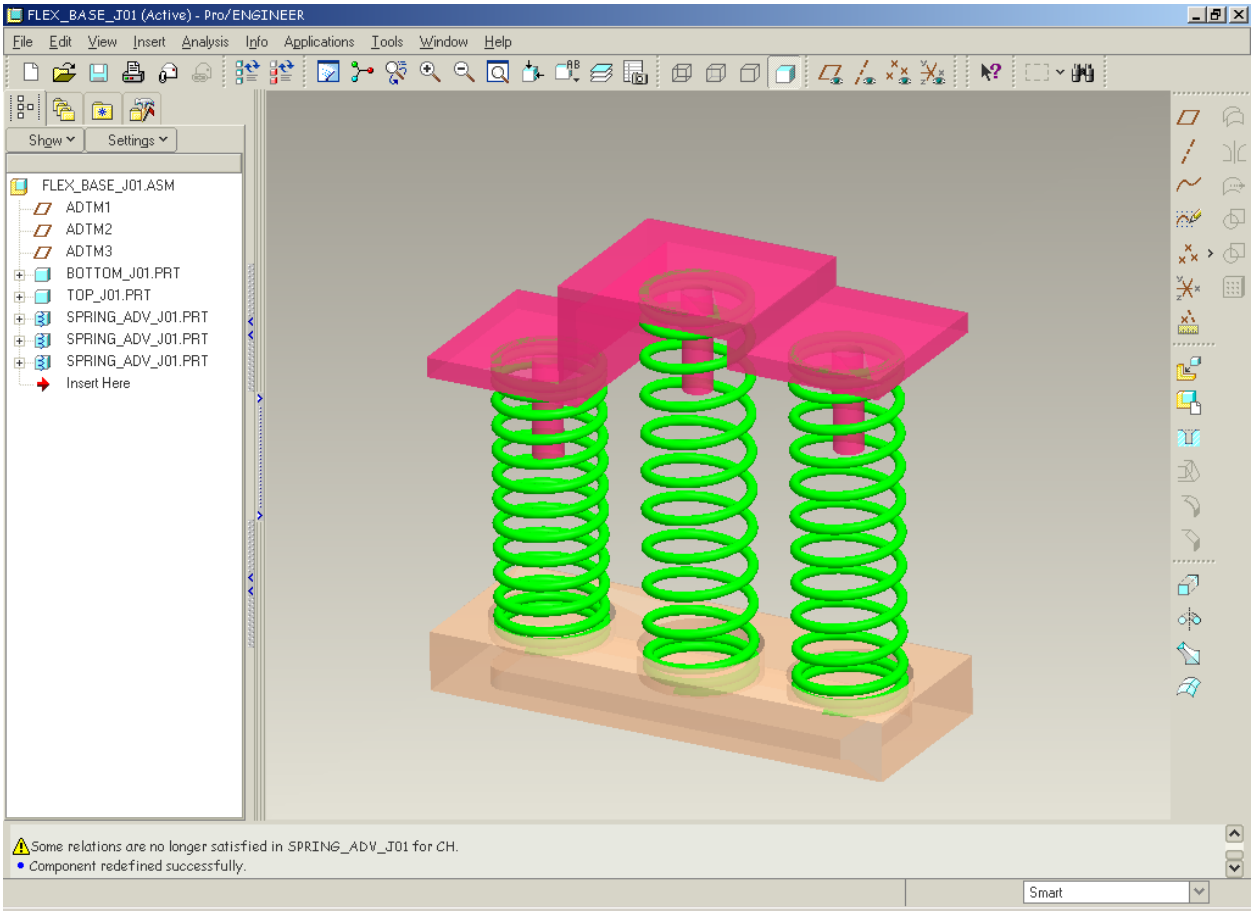
In our example we will drive the new value using the **Distance** option and measure the distance from the lower mating surface on the top part to the upper mating surface on the bottom part.



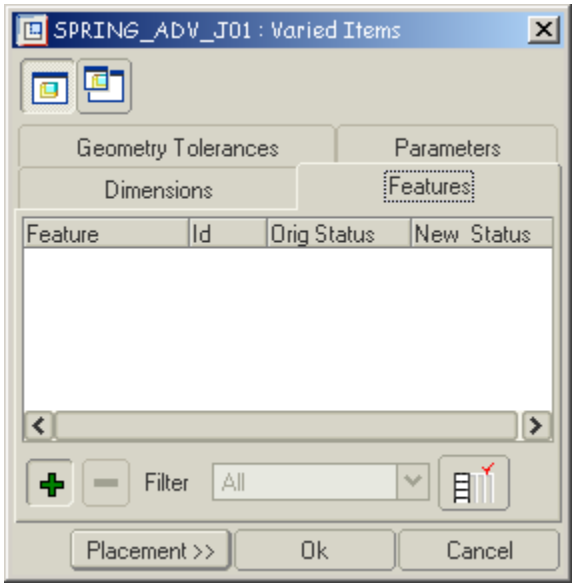
Our varied item dialog box now looks like this.



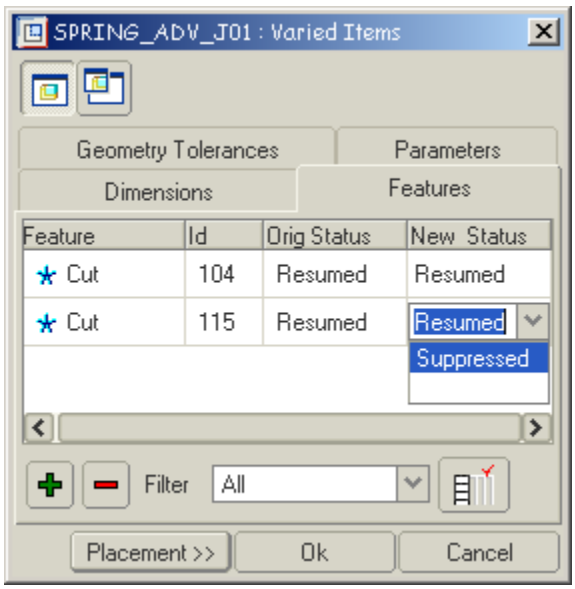
Select Ok to complete the definition of this flexible component.



You can also define varying features as part of the flexible component definition. Let's take a look at how to do that. In the varied items dialog box, select the features tab.



Select the green add button to select features in the component that you would like to suppress or resume. In this example, I will select the ground ends (cuts) of the spring to suppress.



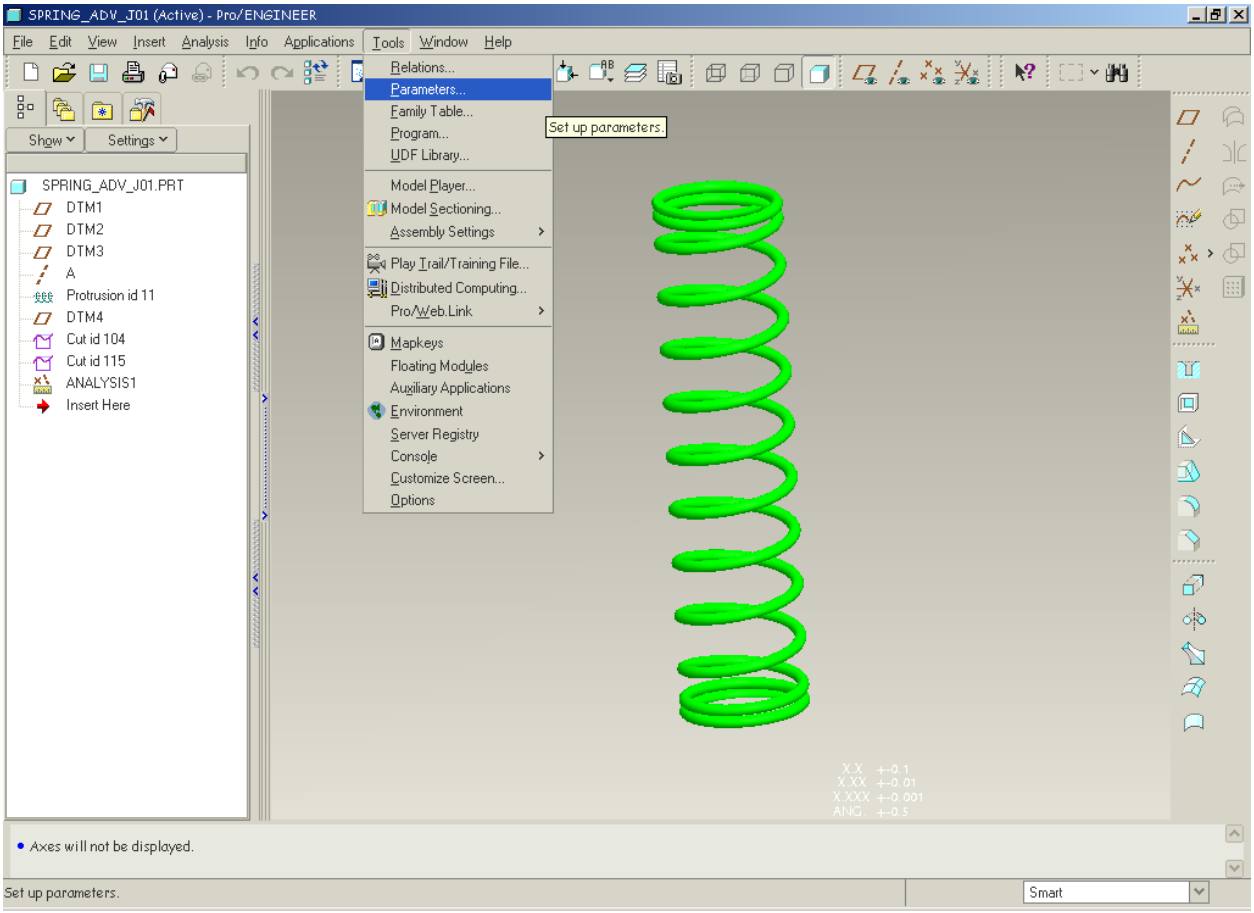
Now by using the pull down menu under the New Status column, I can select that the features new status will be suppressed. Select Ok and the two cut features will be suppressed.

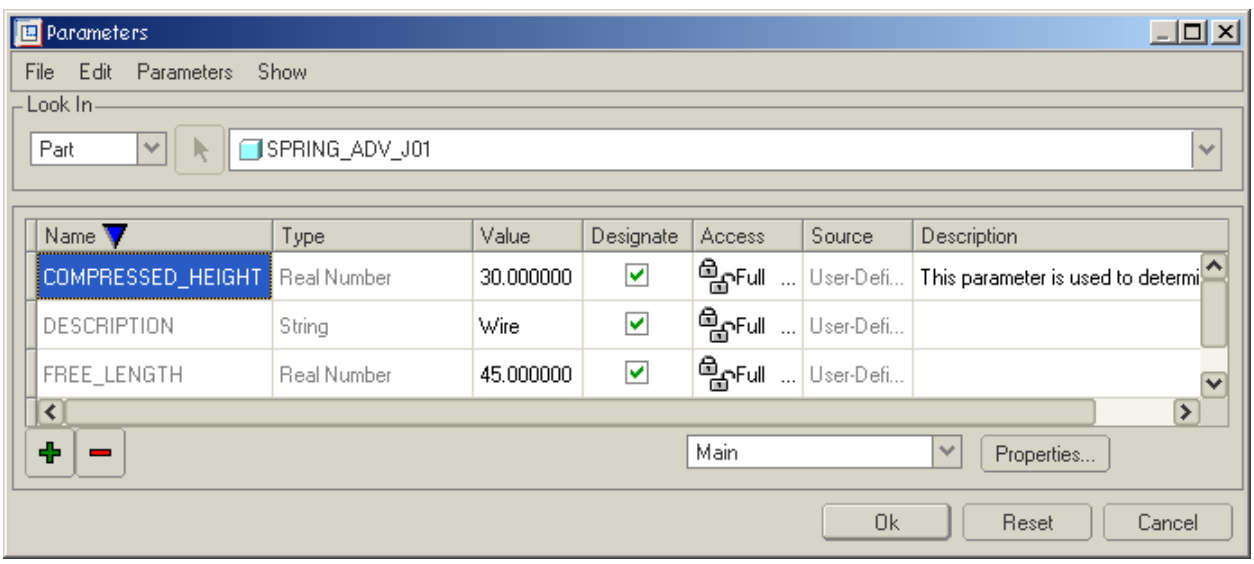
Parameters and Geometry Tolerances can be varied in the same fashion as the dimensions and features.

<u>Adding Additional Intelligence to our spring</u>

We will now look at how we can add in some additional intelligence to our spring model, so that when we do make it flexible in the assembly, it will tell us if we have violated the maximum compression height of the spring.

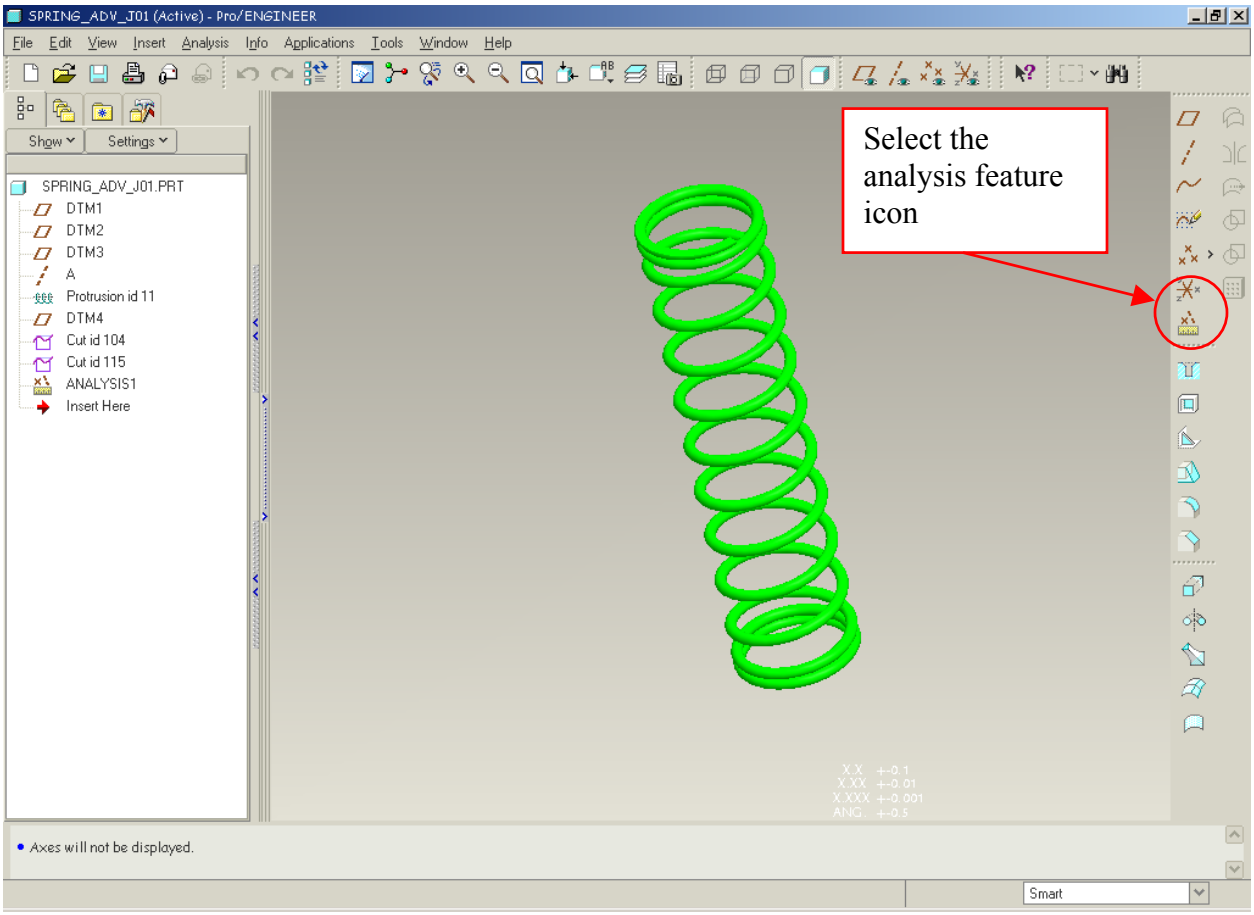
First, open up the spring model. Next, we need to create a Real Number parameter named 'compressed_height'. To do this, select the Tools menu, and Parameters from the pull down.

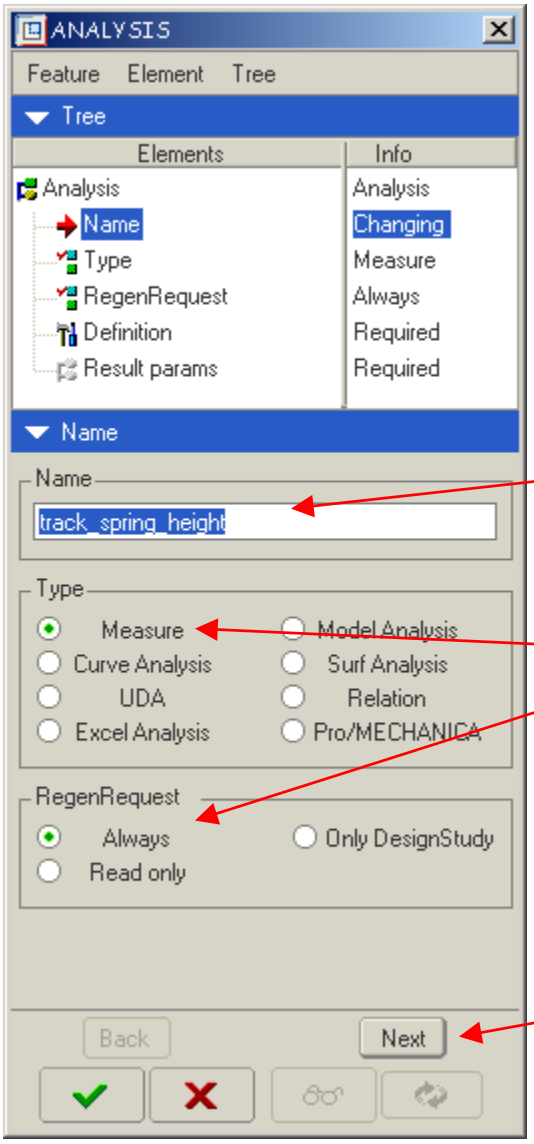




To add a new parameter, select the green plus button. Rename the parameter to 'compressed_height', set the type of the parameter to Real Number and then enter in the value of the compressed height for that spring. (typically this is found in the spring specs)

Now we are going to create an analysis feature to track what the height is between the two ground surfaces (cuts) of the spring that would mate against the other parts in the assembly. Here is how to do this.

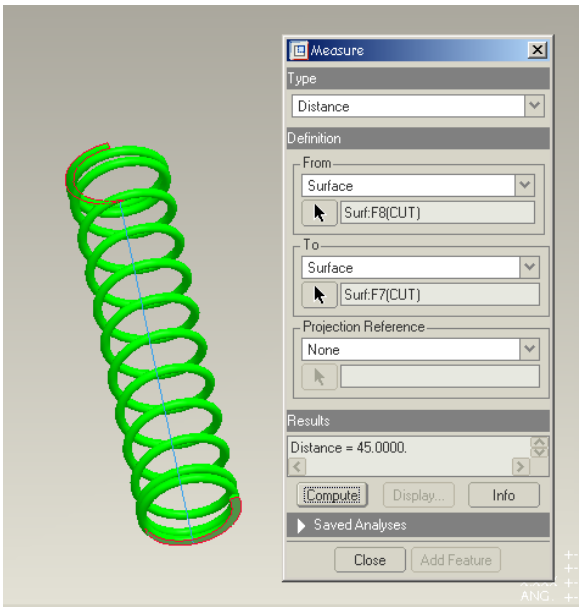




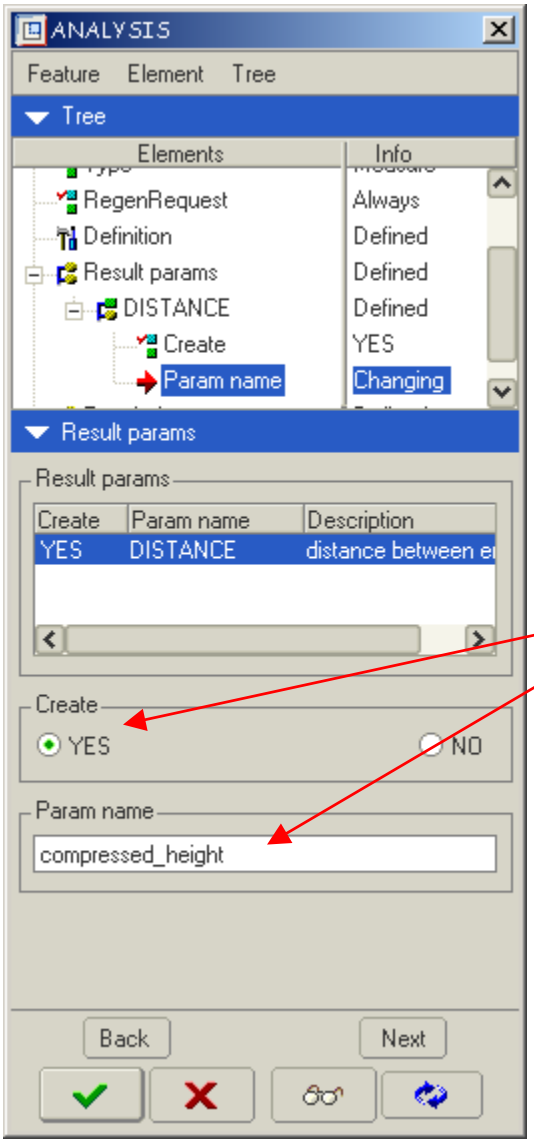
Name the feature. Make sure you hit enter after typing in the name.

Set the Type to Measure and the RegenRequest to Always

Select Next



Measure the Distance from the ground surfaces (cuts) of the spring, then hit close.

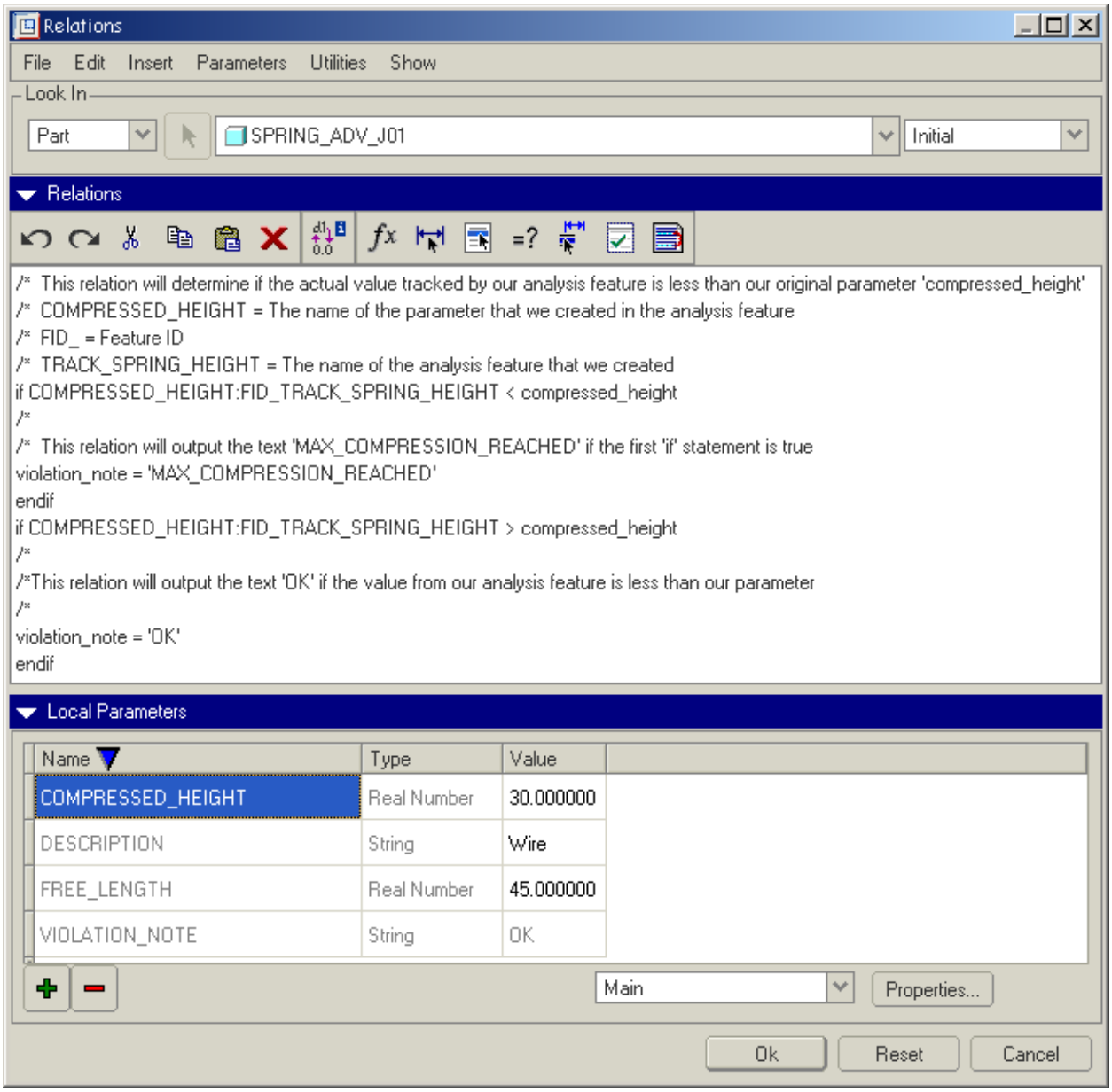


Select YES to create parameter and give it a name. Hit enter after typing in the name to accept the change. Write down the name of this parameter. We will need to use this later. Hit the green check mark.

Now that we have the parameter created to set what the actually compression height is of the spring per the spec and the analysis feature created to track what the compression height of the spring is at all time, we just need to develop a means to let us know when we violated the compression height. Here is what I did.

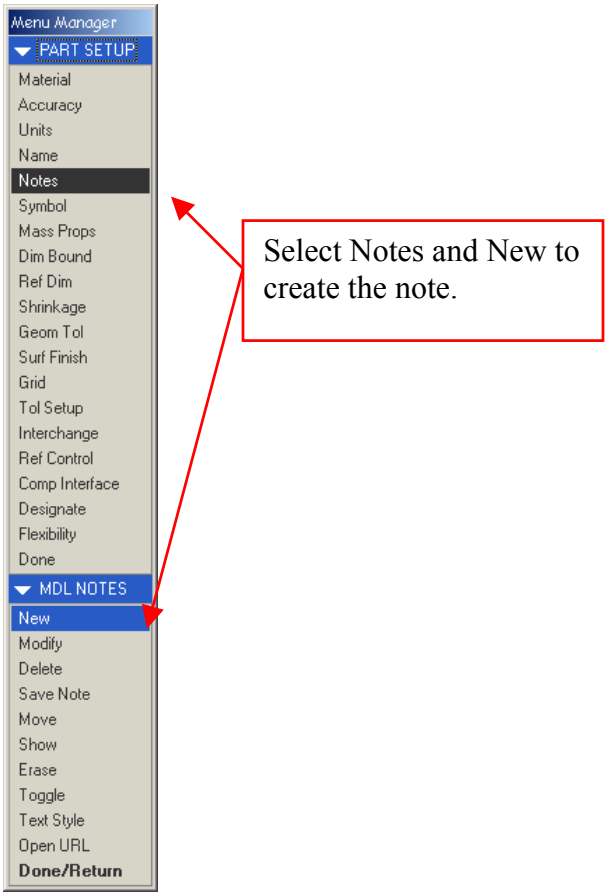
We will create a relation and a note to check to see if we violated the compression height and have the note change accordingly to let us know.

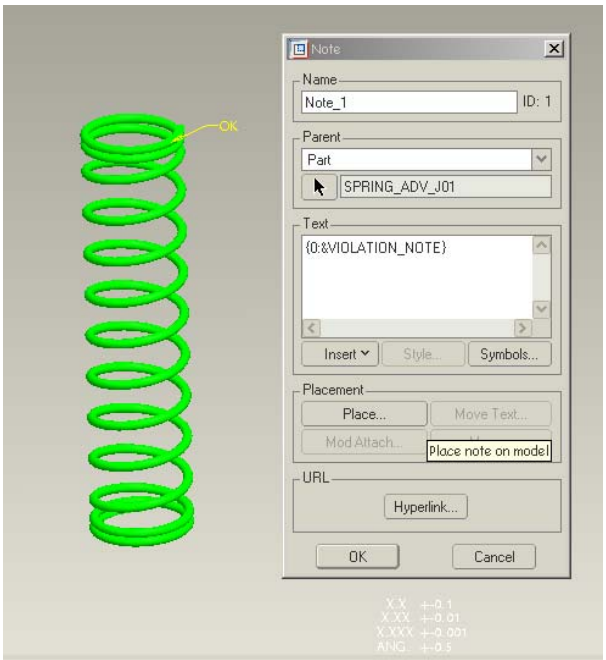
First, access the relations by selecting Tools and Relations from the pull down menu. Enter in the relations as shown below. Be careful to make sure you enter in the correct names of the features and parameters that we created in our analysis feature.



Hit Ok when done entering the relations.

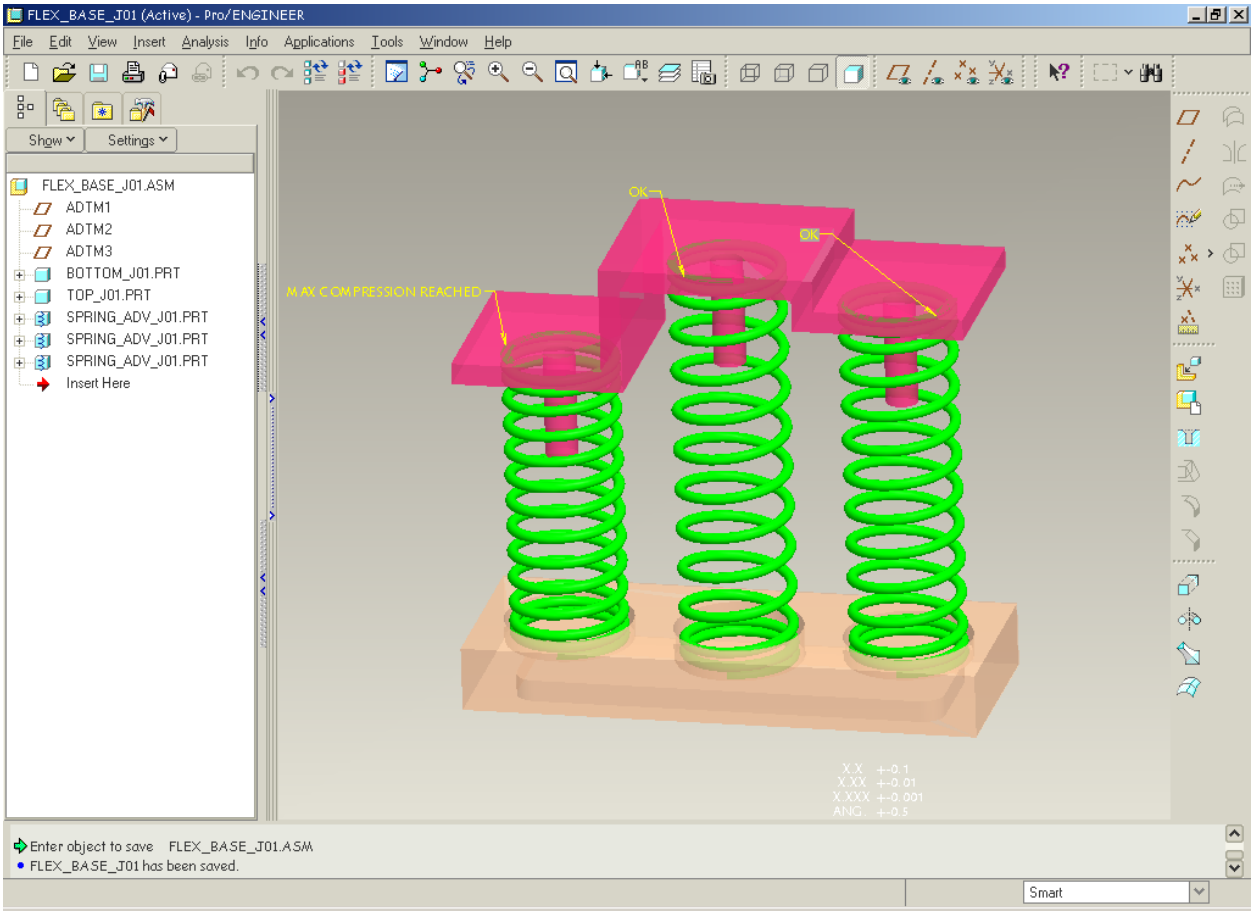
The last step in this process is to create a 3D note to display the results. Here is what to do. In Wildfire, select Edit / Setup and then select Notes from the Menu Manager. Select New to create a new note.





Enter in the text field **&vilolation_note.** This will display the result of the note after the relations have been evaluated. Place the note with a leader attached to spring model.

Now we are done building in the added intelligence to the spring. Let's see how it works. If we revisit the assembly, each instance of this spring will have the note display. After we make the part flexible and regenerate the assembly, the noted will change to let us know immediately if we violated the maximum compression height for the spring.



Good Luck!

Back To Top

Tips of the Week

Creating a Mechanism in ProductView

Click Here To View

Back To Top

Announcements

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, 2003	Your local Pro/Engineer User Groups http://www.prouser.org/rugs/
June 8 - 11, 2003	Orlando, FL 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