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PTC Product Focus

What's New in Pro/ENGINEER Mechanics for Wildfire 4.0

With Pro/ENGINEER Mechanics for Wildfire 4.0, PTC has taken simulation to a new level. There were significant improvements throughout the entire product from simple to complex. If you haven't taken a look at Pro/ENGINEER Mechanics for a while, it is time to do so.

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

- **User Experience**
- **Functionality improvements**
- **Results**

The list that is in this document isn't the full list of enhancements in Mechanics. This is a list of the top enhancements that will have the greatest impact on users in Wildfire 4.0.

User Experience

Access to Mechanics Results with No Active Model

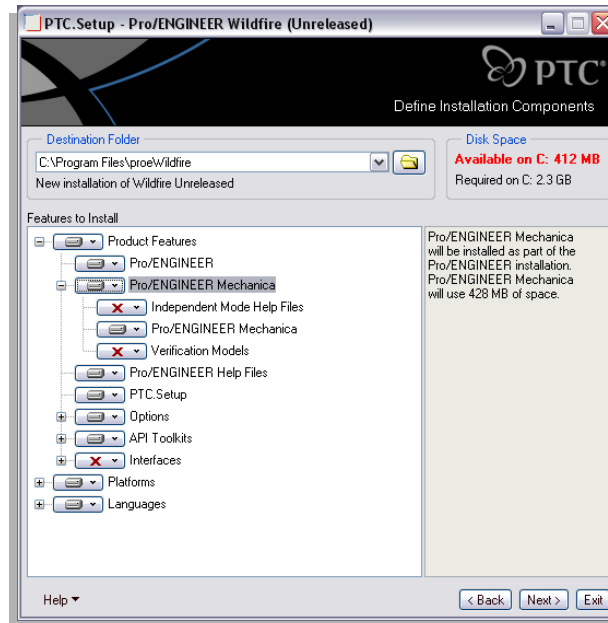
You can now enter Mechanics results to post-process analyses with no active Pro/ENGINEER model.

Merged Installation

Pro/ENGINEER Mechanics now installs as a component of Pro/ENGINEER.

Using PTC.Setup for installation, you select the Mechanics component with Pro/ENGINEER. The new installation streamlines the process of installing new or maintenance releases of

Pro/ENGINEER Mechanical. There is a 40% reduction in the number of steps to install Mechanical. No more risk of version mismatch.



Model Tolerance Enhancements

Enhancements to the model tolerance report make it easier to understand where problems are introduced.

Meshing assemblies whose components have large variances in tolerance is sometimes a problem. With the enhanced model tolerance report, you can easily view the accuracy and type for all of the components in an assembly.

Component-Component Interface Assignment

Component-Component has been added as a reference type for creating interfaces.

To simplify the workflow of setting up interfaces (bond, free, contact, thermal resistance); you can create the interface using a Component-Component reference type. Mechanical then creates contact interfaces between any two components that satisfy your specifications under Selection Filtering Tolerance.

The settings include separation distance and angle. You can also create contact between only planar surfaces. You can easily remove unwanted contacts or change them to a bonded or free interface.

Managing Assembly Connectivity

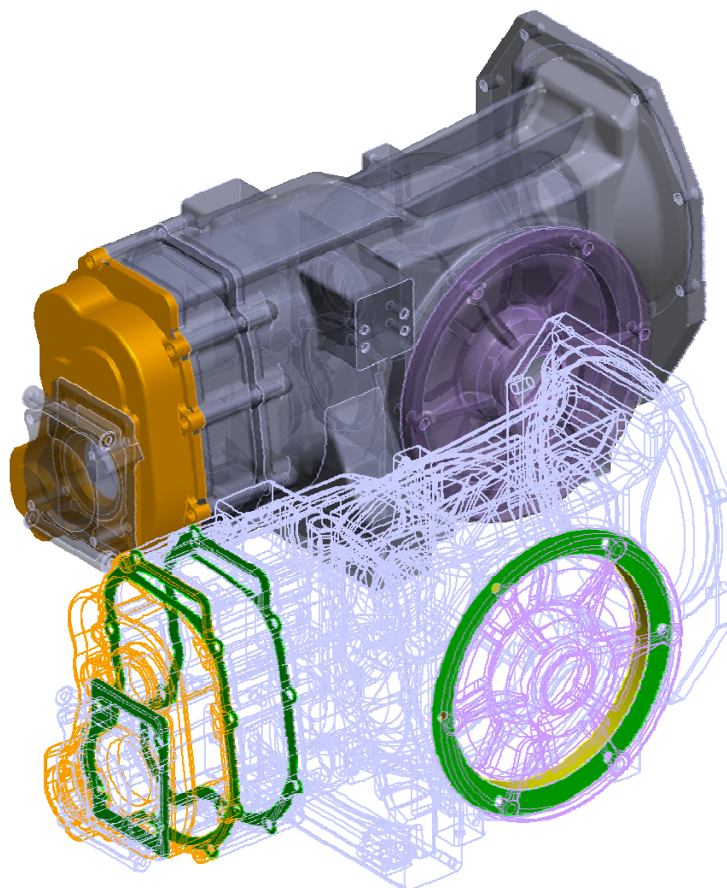
Interfaces are created through a common dialog box.

All interfaces are now created through a common user interface. With the Interface dialog box, you can assign Bonded, Free, or Contact interfaces between selected components in structural analyses, and Bonded, Adiabatic, or Thermal Resistance interfaces in Thermal assemblies.

Model Connectivity

You can preview interface types between components in an assembly.

Using the Model Connectivity command, you get a graphic preview of your assembly setup. A preview makes it easier to spot components without the correct interface type. You can easily see what components are bonded, in contact, or connected through rigid links. You can also select a color to highlight the different connections in your model.



Default Interface Type

You can select a default interface type between components in an assembly.

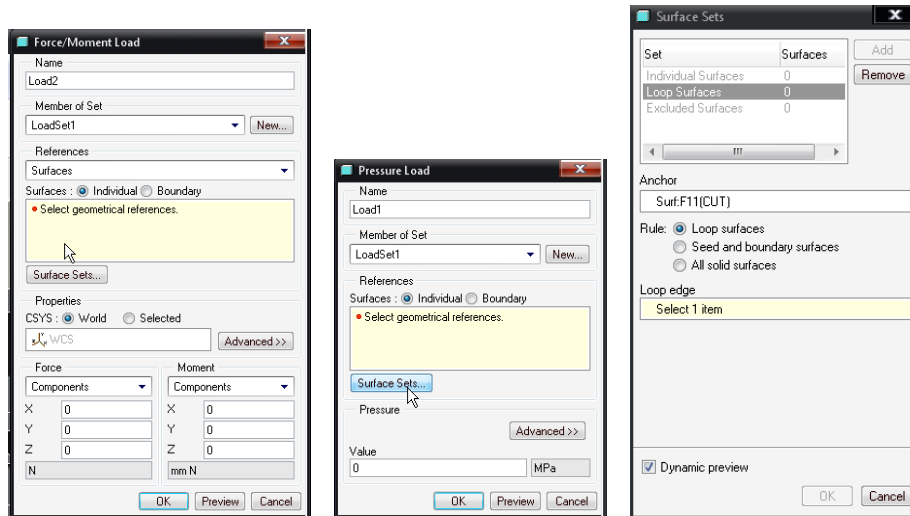
When entering Mechanica, you can select the default interface type for the coincident surfaces of components of your assembly. You can choose to merge them, leave them free, or create contacts (or thermal resistance) between them. Mechanica will automatically create interfaces to connect coincident surfaces and components while meshing or during an analysis.

Style Selection from a Collector

Reference selection in Mechanica is consistent with standard mode.

Selecting entities in Mechanica follows the same workflow as in standard mode. This simplifies selection in several ways:

- Selection is always active when you are using a dialog box.
- You can switch reference selection using the right mouse button without leaving the model.
- You can work more directly on the model.



Functionality Improvements

Enhanced Assembly Modeling

Faster, more accurate modeling of compressed assemblies is possible.

When running assemblies containing midsurface compressed components, Pro/ENGINEER Mechanical uses enhanced bonding elements instead of links to couple the structure across these gaps. These new bonding elements use less memory, solve faster, and provide more accurate results.

Improvements to Meshing for Simple Models

Curvature-based, AutoGEM mesh controls improve meshing for simple models.

The default values for the AutoGEM limits produce p-meshes for accurate results for nearly all real-world models. On very simple models, AutoGEM may be overly aggressive in minimizing the number of elements in a mesh. On rare occasions, a low mesh density in a region may lead to stress results of limited accuracy.

With the new curvature-based mesh control, you can add additional elements only where the geometry of the model has curvature. These additional elements are automatically sized by the local curvature.

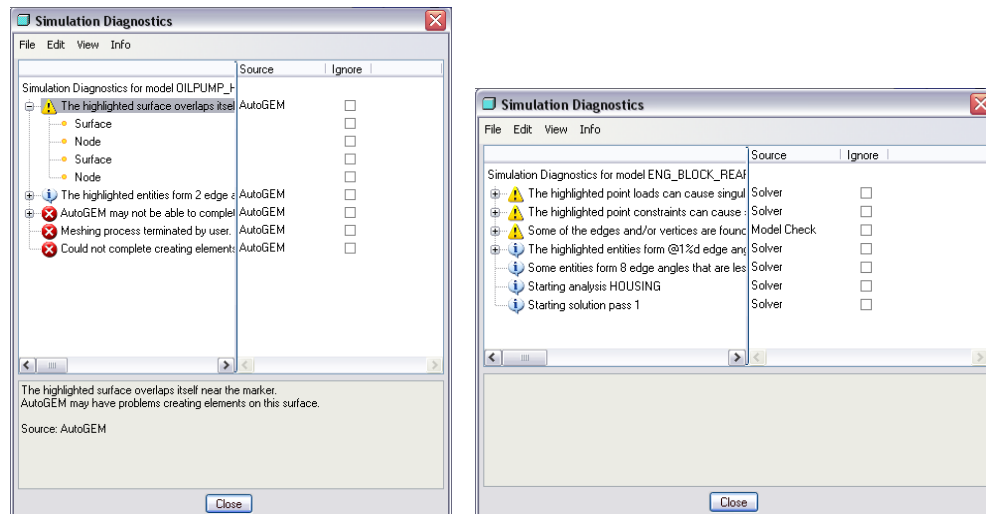
Improved Diagnostics

A Diagnostics tool helps in identifying issues with models.

The Diagnostics tool helps diagnose and resolve potential problems in your model by:

- Displaying meshing, error messages, and warnings through a new, common, and improved user interface
- Identifying clearly the differences between information, errors, and warnings
- Zooming in on the identified problem areas
- Allowing review of groups, such as knife-edge elements created during meshing process

Existing messages are more explicit and clear, and new ones have been added.



Loads Propagated from Part to Assembly Level

You can propagate loads and constraints from the part level to the assembly level.

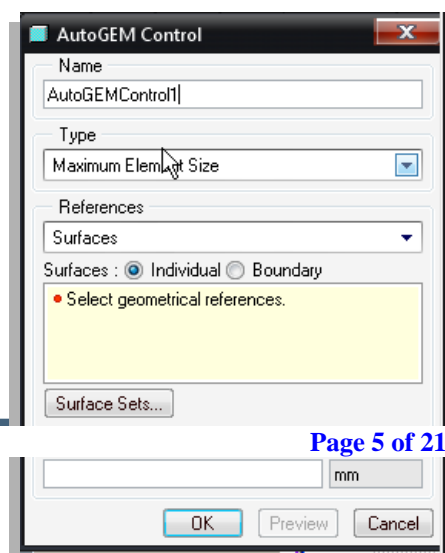
The propagation of loads and constraints is advantageous with thermal models in particular. The benefits include:

- You no longer must recreate at the assembly level heat loads and convection conditions that were already applied on individual components.
- You can assign and store heat loads in a library of components.
- For clarity, whenever you can select a load or constraint set, component names are listed with the name of the load or constraint set.
- You can control the visibility of component loads and constraints at the assembly level.

AutoGEM - Max Element Size for Mesh Control

You can control the maximum size of elements for mesh created by the AutoGEM mesh generator in particular areas of a model.

Limiting the size of the elements created is useful in areas of concern, such as stress concentration. You can define the



maximum element size for AutoGEM controls on the following types of geometry:

- Components
- Volumes
- Surfaces
- Edges and curves

Nonlinear materials

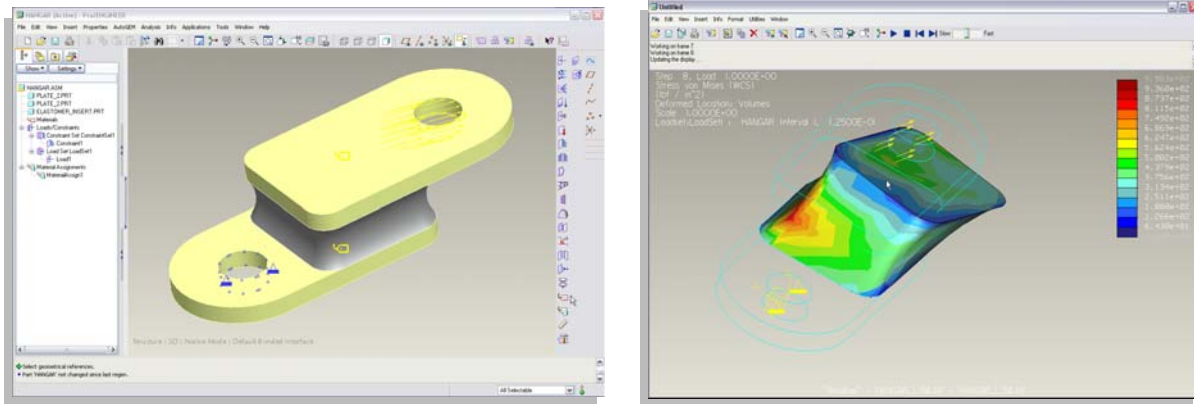
Support for nonlinear elasticity

Nonlinear Elasticity – Where the Stress/Strain relationship is nonlinear but when the load is removed the component returns to its original shape; i.e. there is no permanent (plastic) deformation. This addresses plastic part design, elastomers, etc.

Industry leading materials definition UI brings complex nonlinear solution capabilities to non-specialist users. Material properties can be defined by experimental data or using the following materials laws:

- Arruda-Boyce
- Mooney-Rivlin
- NeoHookean
- Yeoh
- Polynomial Order 2
- Reduced Order Polynomial

Users can input test data, we fit the appropriate material law. Material laws fit to experimental data in real time. Nonlinear materials participate in large deformation analyses.



Contact with Infinite Friction in Mechanics

More high-end solution capability brought down to engineer level capability. Users have the ability to add friction to individual contact regions.

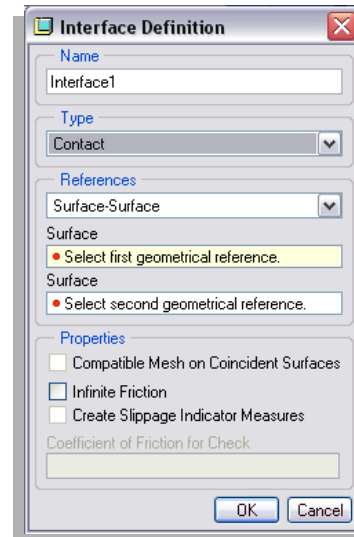
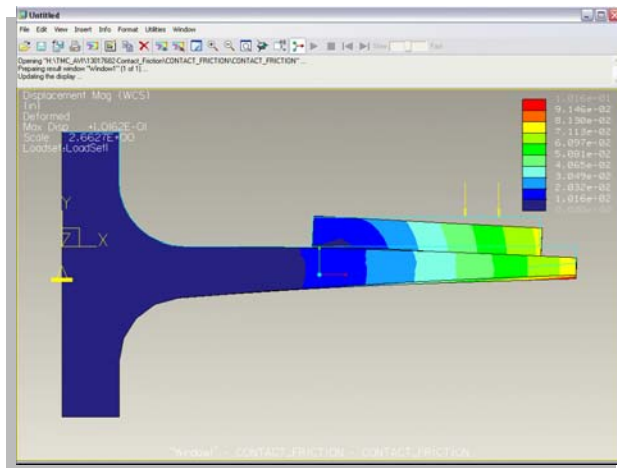
Infinite Friction – When there is any normal force (contact pressure) a very high stiffness is applied tangentially; i.e. there is no tangential motion in the contact region.

Once contact is enabled, users may also define “slippage measures.” The measure will let users know if the normal force with the friction coefficient specified is enough to keep the parts from moving.

Friction can be used to keep contact models from being under constrained in static analyses.

Additional results quantities include:

- Tangential Traction – The tangential load in the contact region
- Slippage Indicator – If Red (greater than zero) the normal force and friction coefficient are not great enough to overcome tangential load



AutoGEM Excluded Element Mesh Control

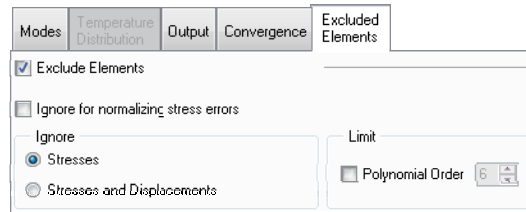
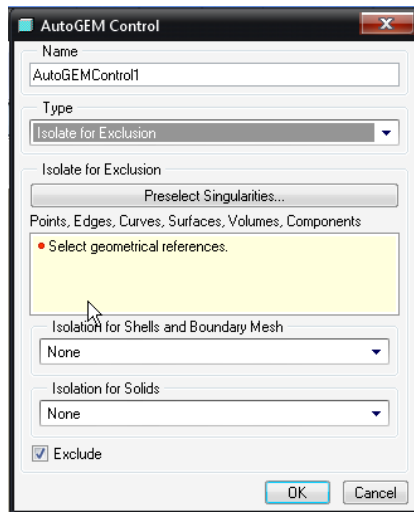
This is a Standalone capability now in integrated mode. This gives the user the ability to exclude singularities from convergence. This can be used to Isolate and/or exclude elements from convergence in both MPA and SPA analyses

Entities may be selected for isolation either by Auto selection of singularities or any points, edges, surfaces, volumes, or components

A maximum element size may be specified for these entities. (For example - any solid element touching this vertex must be smaller than 2.0 mm)

Unselecting the “Exclude” checkbox will only influence the mesh, even if excluded elements are turned on in analysis definition. Below are some additional information on this capability.

- Exclusion elements may be reviewed in AutoGEM
- When Excluded in analysis definition these elements are ignored when checking convergence
- When “Ignore for Normalizing” in analysis definition the stresses in these elements are not used for errors



Contact thermal resistance

In the real world, parts don't conduct heat perfectly between them for 2 primary reasons, surface roughness and some intermediate material (Thermal greases, etc...)

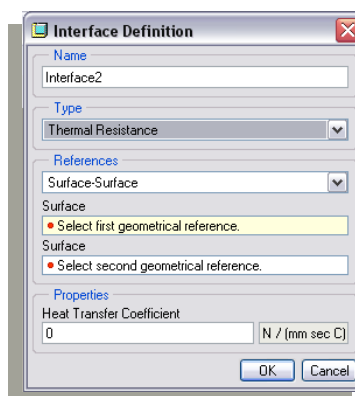
This gives more accurate modeling electronics such as PC boards.

Enter the heat transfer coefficient.

- “the quantity of heat that passes in unit time through unit area of a plate of particular thickness when its opposite faces differ in temperature by one degree”
- heat transfer coefficient = Thermal Conductivity/Length = k/L

For example

- k for thermal grease = $0.7 \text{ W}/(\text{m } ^\circ\text{K})$; Length = $.1 \text{ mm}$
- $k/L = 7000 \text{ Kg}/(\text{s}^3 \text{ } ^\circ\text{K}) = 7000 \text{ N}/(\text{mms } ^\circ\text{K})$



Results

Location in Dynamic Query

You can view coordinates for the position of the dynamic query.

When using dynamic query in results, you can display or not display the coordinates of the dynamic query along with the current value.

Results Legend Editing

Greater control over customizing the results legend significantly improves your ability to format your results to match your own specifications and needs.

You now have more control over customizing and interacting with the legend in results. Added capability includes:

- Defining the colors used for each legend level
- Associating specific values to each legend level
- Saving the customized legend spectrum
- Setting a config.pro option to point to a legend directory or to point to the default legend

AVI Format Available in Movie Export

AVI format is now available for the export of movies.

You can now export the movie of your Mechanica analysis in AVI format in addition to the previously available MPEG format.

Enhanced Viewing of Results Geometry

Greater control over area of viewing in Mechanica results.

In Mechanica results, you can select specific geometry to review. You can use the complete range of Pro/ENGINEER selection and display capabilities, including the smart filter and query select.

Dynamic Query on Capping and Cutting Surfaces

Dynamic query is available with capping and cutting surfaces.

You can dynamically query the results windows containing capping and cutting surfaces or query the capping and cutting surfaces themselves. You can also display the location of the query point.

Resultant Measures

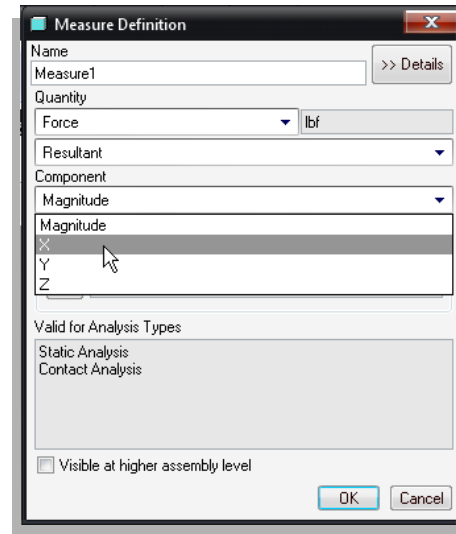
Resultant measures, previously only in standalone Mechanica, are available in integrated mode.

In integrated mode, you simply "cut" the model by using volume regions and extract the reaction force through this cut. The new, more accurate algorithm computes the resultant measures for static and steady-state thermal analyses.

Reaction forces through any slice in the model. Users simply “cut” the model using volume regions and can extract forces and moments through these slices.

Resultant added to Force and Moment measures. All quantities available, X, Y, Z, and Magnitude.

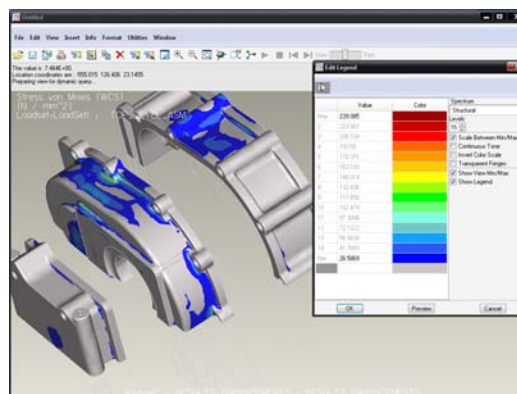
Note: The surfaces selected should slice “through” the model (I.e. they should not form a “crack” that does not completely extend through the model)



Exploded Views in Results

You can view results of assemblies in an exploded state in Pro/ENGINEER Mechanical.

Use the Explode command for fringe, vector, or model plotting to view an exploded state of the assembly. Using the exploded state, you can more easily investigate results at the connection point between different components.



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Tips of the Month

Integrating Behavioral Modeling with a Mechanism Analysis

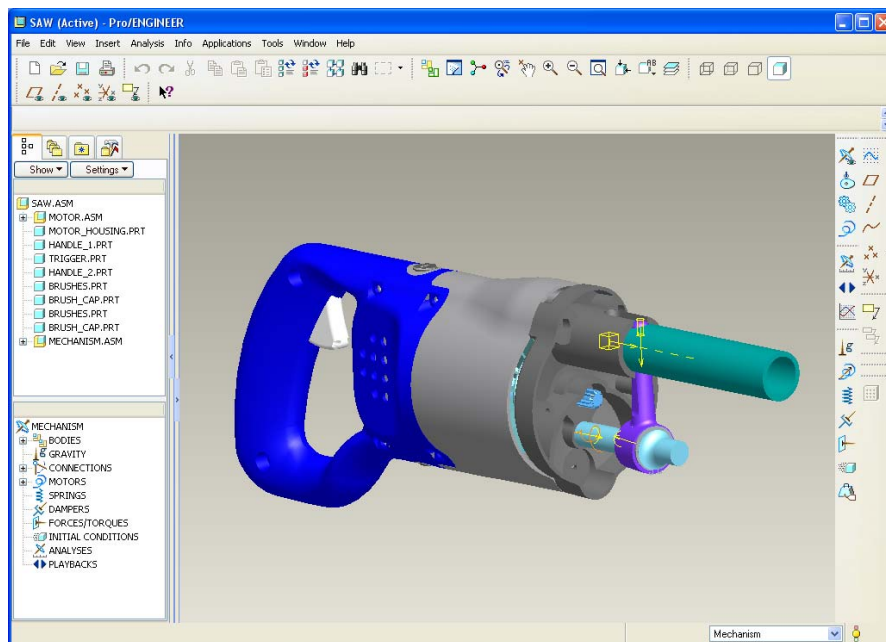
Many people are aware of how to use Behavioral Modeling (BMX) with a static part. There is also a very powerful technique that can be used to integrate BMX with Mechanism Design and Mechanism Dynamics. My example here will only show Mechanism Design integration, but keep in mind that any Mechanism Dynamics problem can be integrated as well.

The benefits of using BMX and Mechanism are huge. You can optimize any mechanism for any type of measurable item. So, think of that for a second. Any measurable angle or distance can be measured during a mechanism analysis, and then you can optimize that measurement to what you desire. No more guessing, no more differential equations, only letting BMX do the heavy lifting and solving your problem.

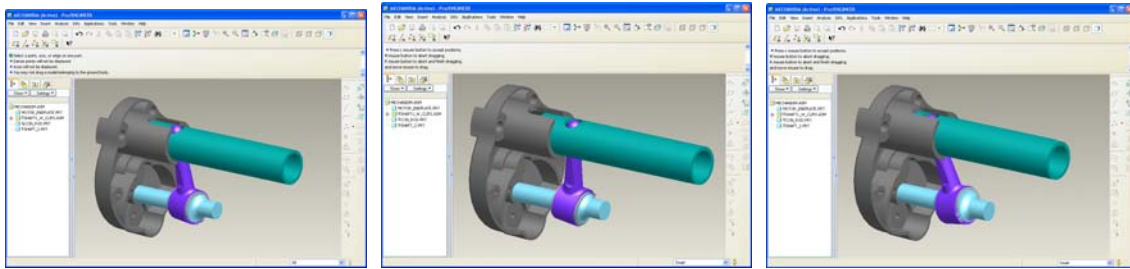
This example is illustrated here and this concept can be used for any mechanism problem. You can view the entire demonstration on YouTube by clicking [here](#).

Step 1 – Defining the Mechanism

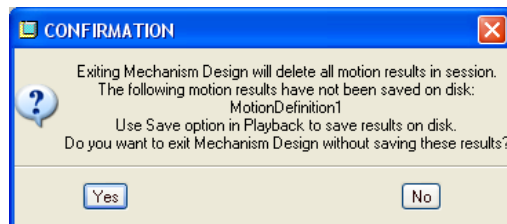
We are going to use a reciprocating saw example. The saw is already defined with mechanism joints that enable it to drag smoothly and a servo motor has been defined for it the driving motor to do one revolution. We are only going to focus on the subassembly at the front of the saw.



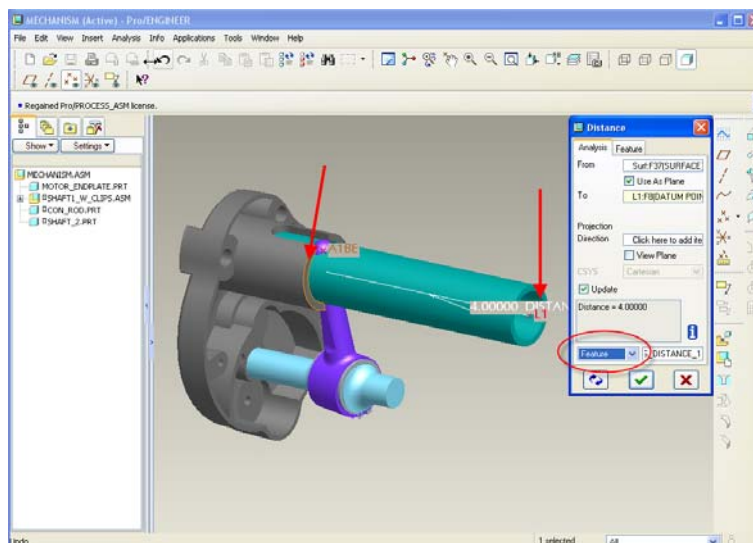
The mechanism is complete and moves like the next few screen shots.



An analysis is set up and ran inside of Mechanism mode. This analysis plays a key role in the following steps. Once the mechanism analysis runs, you can now exit Mechanism mode. You will be warned with the following dialog box, but this is okay. Simply click 'Yes,' and you are then back in normal assembly mode.

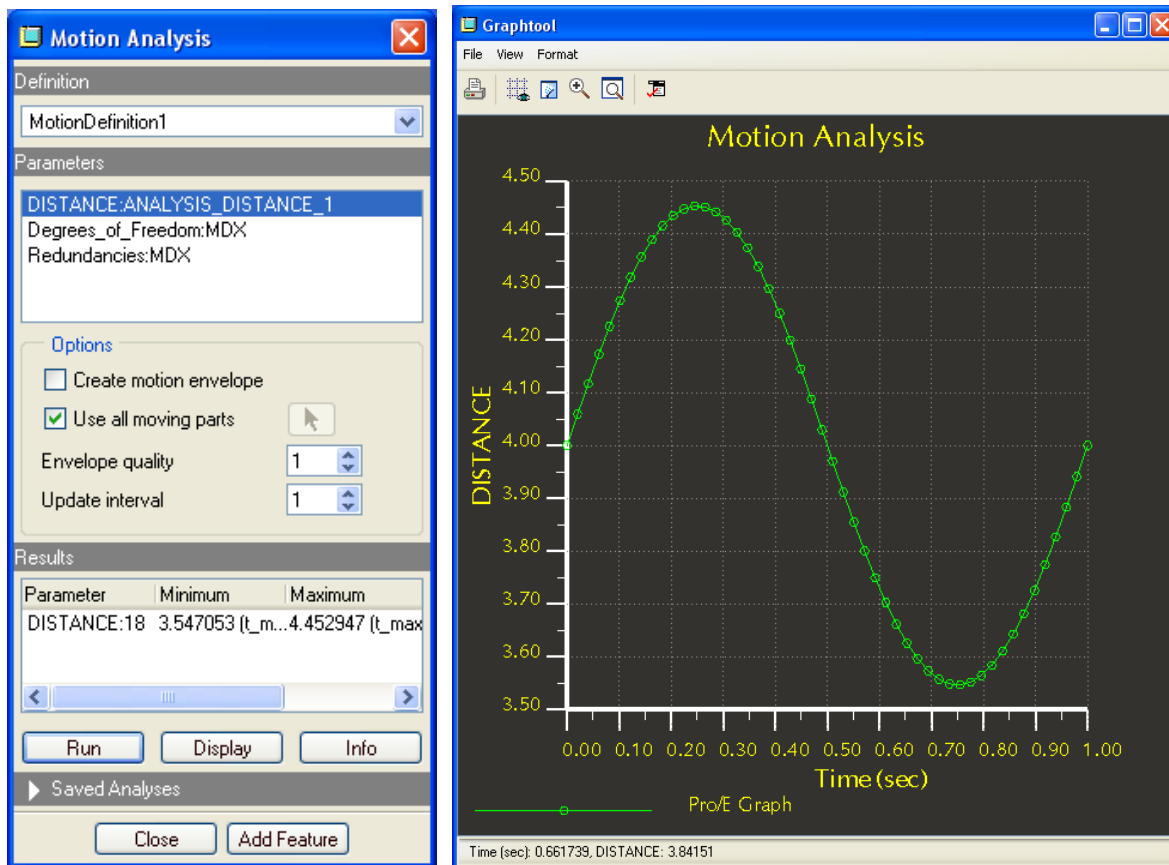


The first thing that you need is an analysis feature in Pro/ENGINEER. This feature is the key ingredient to setting up the BMX work. In this case, we are going to measure the distance between the endpoint of the shaft with a stationary surface on the motor endplate.




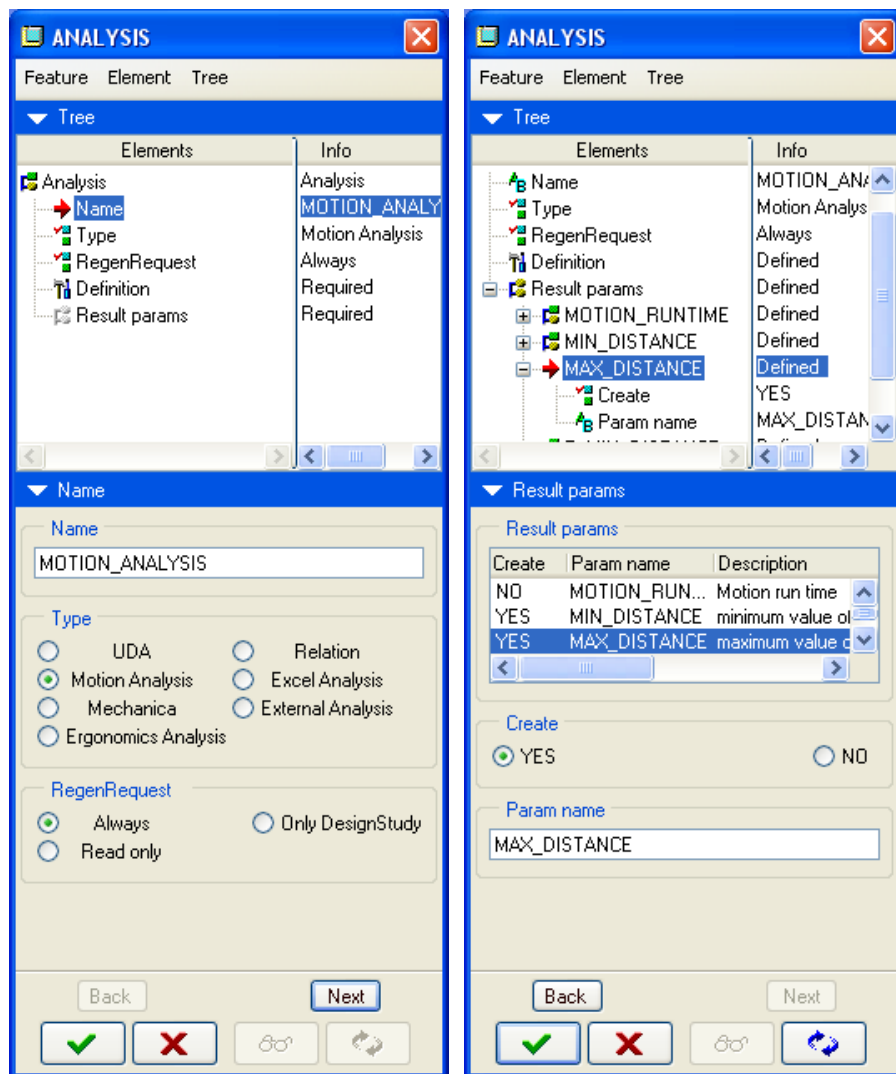
Before you close the measure dialog box, make sure you change the type of measure from quick to feature. Then hit the check mark and the analysis feature will now be shown on your model tree. (You may have to show features on your model tree first.)

The next step that you can look at is the ‘motion analysis.’ This is a quick feedback tool that shows how your BMX feature that you just made can interact with the mechanism analysis that you previously ran while in Mechanism mode. To get to the motion analysis, you simply click Analysis>Motion Analysis. A dialog box pops up and then you can select the distance feature that you just created as a BMX feature and then click RUN. A graph like the following appears.




This graph tells me the measurement value during the mechanism analysis. This by itself is very valuable, but let's keep going to see what other features we can make and eventually optimize our assembly. Close the dialog box and follow the next step.

We now would like to capture the high and low parts of the graph. Create a pure analysis feature by clicking the icon , or selecting Insert>Model Datum>Analysis. Change the type to Motion Analysis as shown in the next picture, and then click Next. As before, you select the Distance measurement and then click Run. Then close the graph dialog box and click close again. This time another dialog box appears. Make the changes shown. What you are trying to do here is to have BMX capture a maximum and minimum value for the entire mechanism run. After you are complete with the changes, click the check mark to go on to the next step.



Only one more step is now required in this problem. You need to find out what the entire stroke length is for the saw so that you can optimize on this value. A BMX relation feature works well for this.

Create a pure analysis feature by clicking the icon , or selecting Insert>Model Datum>Analysis. Change the type to 'Relation,' then click Next. Now fill in the dialog box with a new parameter called 'stroke' and put this logic into it.

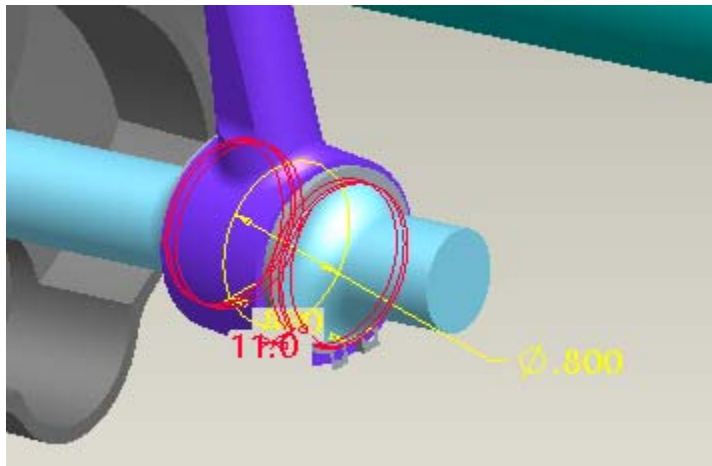
stroke=MAX_DISTANCE:FID_19-MIN_DISTANCE:FID_19

Select OK and then finish the feature. You should be able to see the new feature in the model tree and if you do an Info>feature on it, you'll find that the value is .906 inches. Let's try and get the assembly into a state where the stroke is 1.05 inches.

Relation Table		
Relation	Parameter	New Value
Feature relations:		
stroke=MAX_DISTANCE:FID_MOTION_ANALYSIS-MIN_DISTANCE:FID_MOTION_ANALYSIS	STROKE	9.058940e-01

Local Parameters						
Symbolic constant	Current value	TYPE	SOURCE	ACCESS	DESIGNATED	DESCRIPTION
STROKE	9.058940e-01	Real Number	Relation	Locked	NO	

The next thing is to run a sensitivity analysis to see what will make an impact on the assembly to get to our desired answer. Next, click on Analysis>Sensitivity Analysis. Select something that you'd like to change. In this case, select the angle of the wobble joint, which is currently at 11 degrees. Then fill out the remainder of the sensitivity dialog box with the parameter you'd like to see as a result (stroke) and the limits of the angle. Then click 'Compute.'



Sensitivity

File Options

Study Name
Name: SENS1

Variable Selection
Dimension Parameter
angle:SHAFT_1

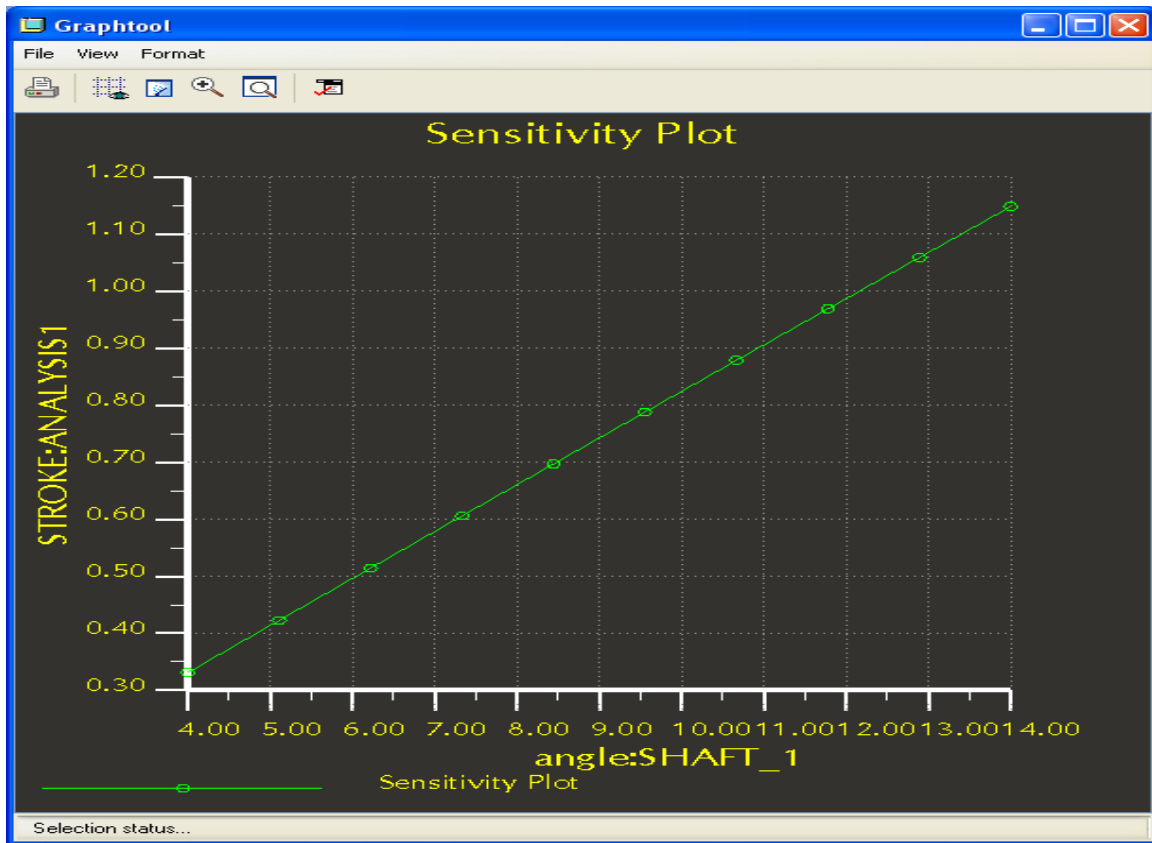
Variable Range
Minimum: 4
Maximum: 14

Parameters To Plot
STROKE:ANALYSIS1

Steps: 10

Compute Close

The following graph shows us that the goal of 1.05 inches is attainable. But what is the value exactly? That is what will be covered next.

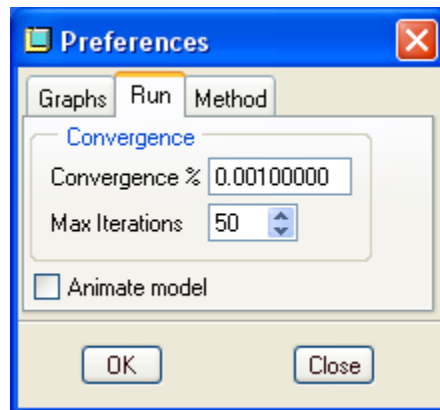
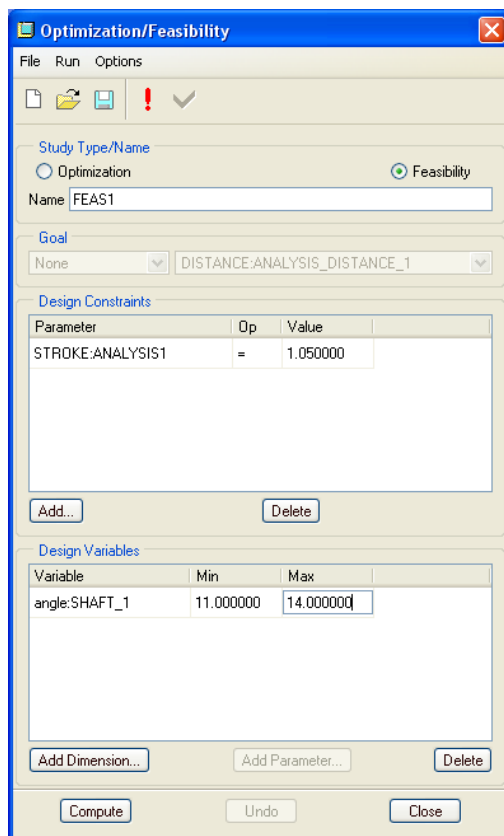


The final step in the process is to use the optimization feature in BMX. This can be done one time, or can be created as an optimization feature that is part of the model tree. To create this feature, click Analysis>Feasibility/Optimization.

Change the dialog box to be a feasibility (because we know the answer!). Then add the parameter 'stroke' to the design constraint area and set it to 1.05. Then choose the design variables. You can select as many as you wish, but we only have one in this case and it is the wobble angle. We know from the sensitivity study that the answer will be very close between 11 and 14, so that is what we enter as limits.

It is also a good idea to change the preferences – Options>Preferences. You can get a more exact answer if you change your convergence from .5 to .001. This way you'll get exactly 1.05 for the answer instead of something 'just close enough.'

The following shows us the dialog boxes for this step.



Now click Compute and it finishes with the answer you want in just a few seconds. The magic answer for the wobble angle was 12.8. If we do another Info>feature on the relation feature, it will show that the stroke is now exactly 1.05.

So the next time you have yourself in a situation where you are optimizing a mechanism, use BMX to help you get the perfect answer. All the tools and features are there to use for you to let Pro/ENGINEER do the solving for you.

Remember, you can watch this [demo on YouTube](#).

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Tips of the Month

Creating Profiles to Support Classification or Portion Marking

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Announcements

Educational Resource Library

Learn things you always wanted to do - but didn't know you could.

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

Check out the [Educational Resource Library](#) today.

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It's better than finding the Ark of the Covenant!

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!](#)

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.

[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.

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