What's New:
PTC Creo Simulate 3.0
Datecode M020

PTC Inc., 140 Kendrick Street, Needham, MA 02494 USA

Contents

1

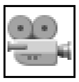
PTC Creo Simulate

New User Interface for Results in PTC Creo Simulate

The new user interface for showing results in PTC Creo Simulate includes the standard PTC Creo ribbon.

User Interface Location: In PTC Creo Simulate, click **Home ▶ Simulate Results**. In PTC Creo Parametric click **Home ▶ Utilities ▶ Simulate Results**.

Benefits and Description



Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

There is a new modern interface for displaying results in PTC Creo Simulate that you can access from PTC Creo Simulate or PTC Creo Parametric. The logically organized interface with commonly used tasks, sorted into application-specific ribbon tabs and groups, makes it easy to use and for new users, easier to learn. In addition to the ribbon interface, there are other improvements that are described in the list below:

- Search for commands—Use **Command Search** to search for and see the location of commands. Click and start to type a command in the **Command Search** box. As you type, a list of commands that match your search criteria appear. Place the pointer on a command in the list to see the command highlighted in the ribbon.
- Commands are consolidated and added—New commands are added and other commands are consolidated to improve efficiency.
  - With the displacement magnitude fringe window selected, click **View ▶ Continuous Tone** to set the tone changes to continuous. You no longer need to edit that specific results window to define tone change from within the **Result Window Definition** dialog box.
  - Under **View** the **Capping & Cutting Surfs** group is added—Functionality is consolidated to a group where you can create, edit or delete a capping or cutting surface.
- Manage the window view—There are new ways to manage the window. Click **Cascade** to cascade views. Click **Hide** to hide a view.
- Manage the Graphics window—As you could previously, you can open multiple docked windows at the same time. Although only one docked window can be a active at a time, you can double-click a different window to

easily activate it. The active window has a blue border and the Graphics toolbar appears within it. This toolbar provides access to common commands such as **Refit**, **Repaint**, **Zoom In**, **Zoom Out** and **Named Views**.

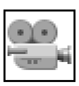
- Additional commands available from shortcut menus—You can right-click in the active window to select commands such as **Model Max**, **New Cutting/ Capping Surface**, or **Full Screen**. This makes it easier and faster to perform tasks.

Regardless of whether you are a new or advanced user, the new modern interface for results in PTC Creo Simulate improves your ability to inspect your analysis, validate your design, or identify areas in your model that need improvement.

PTC Creo Unite Technology — Working in a Multi-CAD Environment

There are multiple enhancements in the area of multi-CAD design.

Benefits and Description



Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

With PTC Creo Unite technology it is no longer necessary to install additional third-party software or licenses to work with non-PTC Creo data. CAD consolidation is easier than ever. CATIA, SolidWorks, and NX files can simply be opened and referenced in PTC Creo. Format-specific profiles and templates help to load non–PTC Creo data as easy as native PTC Creo data. Once defined, these profiles automatically control what information gets into session without additional dialog boxes or user interaction. You can start to work with non-PTC Creo assemblies immediately. PTC Creo automatically tracks changes made to the non-PTC Creo data in their native systems, and keeps the data up to date. It is also easy to distinguish the different formats in the Model Tree by their individual icons and file extensions.

In addition to new capabilities of updating and exporting SolidWorks data, PTC Creo 3.0 provides the ability to import Solid Edge parts and assemblies. During the import process of Solid Edge data, you can decide what information you want to import or can use the standard template for non-PTC Creo data. Imported geometry can easily be referenced using all available assembly constraints. PTC Creo 3.0 provides support for all major CAD formats to integrate, develop, and validate complete product designs. See article CS188705 for information about the current limitations of PTC Creo Unite technology working with PTC Creo View and PTC Windchill PDMLink 10.2.

Whether your goal is to improve collaboration between multiple formats or consolidate to a single CAD tool, with PTC Creo Unite technology it is easier than ever to work in a multi-CAD design environment.

Automatic Conversion Using PTC Creo Unite Technology

With PTC Creo Unite technology it is no longer necessary to install additional third-party software or licenses to work with non-PTC Creo data. CAD consolidation is easier than ever.

Benefits and Description

Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

CATIA, SolidWorks, and NX files can simply be opened and referenced in PTC Creo. It is easy to distinguish the different formats in the Model Tree by their individual icons and native file extensions.

PTC Creo 3.0 has enhanced functionality to modify designs in context. If you try to modify non-PTC Creo data in the context of a multi-CAD assembly, a message appears confirming that these changes will not be reflected in the native model. You can then select **Convert** or **Do not convert**. If you select **Convert** you can choose to automatically convert the non-PTC Creo data to PTC Creo data or to use advanced conversion tools. When you choose an automatic conversion, only the components that are required for the modification are converted into PTC Creo components. This ensures maximum reuse of existing non-PTC Creo files and prevents the overhead of managing multiple file formats and business objects for each component.

After converting the necessary components, PTC Creo can still maintain a link to the original non-PTC Creo files to retrieve updates based on changes made outside of PTC Creo. You also have the option to keep the newly created PTC Creo components separate from the original source files.

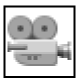
PTC Creo 3.0 provides support for all major CAD formats to integrate, develop and validate complete product designs. Whether your goal is to improve collaboration between multiple formats or consolidate to a single CAD tool, with PTC Creo Unite technology it is easier than ever to work in a multi-CAD design environment.

Creating Profiles Using PTC Creo 3.0 Unite Technology

There are multiple enhancements in the area of multi-CAD design. With PTC Creo Unite it is no longer necessary to install third-party software or licenses to work with non-PTC Creo data.

User Interface Location: Click **File ▸ Options** to open the options dialog box for your application and then click **Data Exchange**.

Benefits and Description



Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

CATIA, SolidWorks, and NX files can simply be opened and referenced in PTC Creo. Format-specific profiles and templates help to load non–PTC Creo data as easy as native PTC Creo data.

Opening and importing options and format-specific profiles are controlled by selecting **Data Exchange** in the options dialog box for your PTC Creo application. With the help of the **Import Profile Editor**, you can create and modify profiles. Once defined, these profiles automatically control what information gets into session without additional dialog boxes or user interaction. You can start to work with non-PTC Creo assemblies immediately.

PTC Creo 3.0 provides support for all major CAD formats to integrate, develop, and validate complete product designs. Whether your goal is to improve collaboration between multiple formats or consolidate to a single CAD tool, with PTC Creo Unite technology it is easier than ever to work in a multi-CAD design environment.

Solid Edge Import

You can import and then work with Solid Edge data.

User Interface Location: Click **File ▸ Open** and then set the **Type** filter to **Solid Edge**. Click **Import**.

Benefits and Description

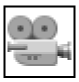
With the Solid Edge converter, you can import Solid Edge parts and assemblies. There is no need to install Solid Edge software to work with Solid Edge data.

Support for Mapkeys

You can create and execute mapkey commands directly within PTC Creo Simulate.

User Interface Location: Click **File ▶ Options ▶ Environment ▶ Mapkeys Settings**.

Benefits and Description



Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

For frequently used command sequences, mapkeys can help you to be more productive. A Mapkey is a keyboard macro that maps frequently used keyboard sequences to a certain keyboard key or to sets of keys. You can open the **Mapkeys** dialog box in the two ways described below:

- Click **File ▶ Options ▶ Environment ▶ Mapkeys Settings**.
- Click and type `mapkey` in the search box. Place the pointer on `mapkey` in the search list.

Mapkeys map frequently used commands to one or more keyboard keys. To start, click **New** in the **Mapkeys** dialog box and then in the **Key sequence** box, define a key sequence such as `ME`. When you press ME after you define the mapkey, a sequence of operations is executed. In the **Name** box, type a name for the sequence such as `Mesh`, to indicate this key sequence is for creating a mesh. In the **Description** box, provide additional details. Next, record the sequence of operations. Click **Record** to start recording your keystrokes.

For this example, click **Refine Model ▶ AutoGEM** and then click **Create** to create the mesh and to see the results in the Graphics window. When you finish reviewing the results you can close the results windows or you can first pause the macro and then add the closing of the windows to the macro. Click **Pause** in the **Record Mapkey** dialog box. The **Resume Prompt** dialog box opens. In this dialog box add some information describing the next step such as `When finished, click Resume to proceed.` Click **OK** in the **Resume Prompt** dialog box. Then, click **Close** to close all the dialog boxes that are open. In the **AutoGEM** dialog box prompt, click **No** so the mesh is not saved. Click **Stop** and **OK**. The sequence of operations you just performed is captured in the mapkey. The **Mapkeys** dialog box opens. Click **Save all** to save the macro you created to your `config.pro` file for future use. To execute the macro press ME and the mesh results appear immediately. The macro pauses so that you can review the results. Click **Resume** as prompted and the operation is completed. You can use this mapkey to perform the same action on any model or assembly. You see the elements created for the part or assembly, review them, and exit. In addition you can add mapkeys as an icon in a group or tab in the ribbon. Mapkeys are optional, however, they improve productivity by streamlining the execution of commands that are frequently used.

Support for Unicode in PTC Creo Simulate

There is support for Unicode. This makes it easier to understand and analyze models containing text in more than one language.

Benefits and Description

Unicode is now supported in PTC Creo Simulate as it has been supported in PTC Creo Parametric. All text strings appear correctly, regardless of their language even when the locale is set to a different language. As a result, sharing models is easier because text strings are legible. Any new text string is maintained and legible in all other locales. For most names, text strings must be 32 characters or less and for most descriptions, 260 characters or less. Characters can be single byte or multibyte. There is Unicode support for object names, descriptions, and other text strings that are stored inside files and appear in the user interface. Process guide templates, results window definition, and template files are not supported.

Performance Improvements in PTC Creo Simulate

A number of enhancements are implemented to improve the overall performance. These changes help solve various analyses faster so you can spend less time waiting for the analysis to converge on a solution.

Benefits and Description

Performance is improved in a number of areas including those described in the list below:

- **Thin Solid**—The existing **Thin Solid** AutoGEM option makes it easier to mesh thin regions with wedge and brick elements. **Thin Solid** is enhanced to minimize the p-level of the element edges that are perpendicular to the top and bottom surfaces. The p-level defaults to a maximum of p=3. This enhancement reduces the complexity of the solution to be solved and shortens the time to converge on a solution.
- Dynamic Analysis—You can already quickly calculate the stresses in dynamic analyses, based on the superposition of modal stresses that are calculated in the associated modal analysis. The calculation of stresses in dynamic analyses is further improved by calculating the superposition of modal displacement for

displacement, velocity, or acceleration results in dynamic analyses. To take advantage of this enhancement, ensure that the **Rotations** check box in the **Output** tab of the **Modal Analysis Definition** dialog box is selected. This makes it easier to perform dynamic analyses and faster to gather results.

- Modal Analysis Improvements—In modal analysis you can output the modal participation factors and effective masses, to help determine whether additional modes are needed to accurately capture the response for the dynamic analysis. While these factors vary with applied loads, it is assumed that these factors are computed for translations along the WCS X, Y, and Z directions and for rotations about the WCS origin in the X, Y, and Z directions. To take advantage of this enhancement, click the **Mass Participation Factors** check box in the **Output** tab of the **Modal Analysis Definition** dialog box. This improves your ability to perform a more accurate dynamic analysis.

Support for ANSYS 14.5 in PTC Creo Simulate

There is support for the latest release of ANSYS.

Benefits and Description

There is support for ANSYS 14.5. As a result you can run your FEM analysis with the latest ANSYS solver.

Support for MSC Nastran 2012 in PTC Creo Simulate

There is support for the latest release of MSC Nastran.

Benefits and Description

There is support for MSC Nastran 2012. As a result, you can run your FEM analysis with the latest MSC Nastran solver.

Performance Tuning for Solver Engine I/O Is Improved

The solver engine is tuned to improve the I/O buffering to make better use of the large cache of modern hard drives.

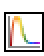
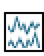
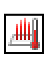
Benefits and Description

When running an analysis, the process of transferring data between memory and disk (Disk I/O), especially during the solver engine postprocessing step, can be time consuming. When using older hard drives, these results files appear to be written in small blocks at a time. For larger models with large results files, this is inefficient and time consuming. The solver engine I/O performance is improved and tuned for modern magnetic hard disks. These hard disks typically have larger onboard cache and solid state disk drives. As a result of this tuning, processing time is faster when creating results files.

Icons are Updated

New icons help to provide a clearer understanding of the functionality.

Benefits and Description

- New icons in the Model Tree under **Analyses** display the following analyses types found in the model:
    - —Static
    - —Modal
    - —Buckling
    - — Prestress Static
    - —Prestress Modal
    - —Dynamic Time
    - —Dynamic Frequency
    - —Dynamic Shock
    - —Dynamic Random
    - —Steady State Thermal
    - —Transient Thermal

  These new icons are also used to define analyses in the **Analyses and Design Studies** dialog box:
- —Icon for the new **Hard Surface** type of mesh control.
- —Icon for the new **Crack** type of idealization.

Changing Entity Colors

You can easily change the colors for entities and use these colors in future sessions.

User Interface Location: Click **File ► Options ► System Colors** and then make a selection under **Simulation Entity Colors**.

Benefits and Description

Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

You can change the colors of the PTC Creo Simulate entities. Entities are grouped into the types listed below:

- **Modeling Entity**
- **Load and Constraint**
- **AutoGEM and FEM Mesh**
- **Miscellaneous**
- **Simulate Results**

Under **Modeling Entity**, for example, you can change the colors for **Shell**, **Beam**, **Mass**, **Crack**, **Weld**, and so on.

In this example, if you mesh an object, you will see the colors representing the solid mesh elements in the Graphics window. Click **File ► Options ► System Colors** and then under **AutoGEM and FEM Mesh**, you can see the colors that currently represent **Wedge Element** and **Brick Element**. If you change these colors in the **PTC Creo Simulate Options** dialog box, the colors in the model change immediately. To keep the new colors and to see these colors in future sessions, click **OK**. You can customize your environment based on your preferences.

Usability Improvements to Fasteners with Defined Preloads

Fasteners with preloads to automatically scale the preload force applied to a deformed structure to be equal to the specified preload force.

User Interface Location: Click **Refine Model ► Fastener** and then in the **Fastener Definition** dialog box, select **Account for Stiffness**.

Benefits and Description

You may want to apply a preload to a fastener which simulates the degree to which a bolt or a screw is tightened and then compresses the components. This is controlled by the preload force which defines the tensile force in the fastener. The tensile force results from tightening the bolt or screw.

When a fastener is applied with a preload, the preload force defines the load that the fastener applies on the undeformed geometry of the model. As the model deforms, the compressive force of the fastener on the structure relaxes, causing the actual preload force applied to become smaller than the specified value defined in the **Fastener Definition** dialog box. If you want to specify the preload force which will be exerted on the deformed geometry, you must scale the specified value to account for the force decrease. This is performed by running an initial analysis to determine the decreased preload force, scaling the specified preload force to account for the decreased force, and running the final analysis.

You can continue to follow this process or automate it by selecting **Account for Stiffness** in addition to selecting **Include Preload** in the **Fastener Definition** dialog box. As a result, the two analyses described below are performed:

- The initial analysis determines all the scaling factors required for each fastener with defined preloads and scales the forces exerted on the deformed geometry to meet the defined preload force value. The preload force value is specified in the **Fastener Definition** dialog box.
- A second analysis automatically accounts for the scaled force values and provides the final results. You can then view the initial forces and the scaled forces in the status report for the study. In the results window only the measure values from the final analysis are available.

Improvements to FEM Mode

Functionality is added to FEM mode to make it more like Native mode.

User Interface Locations: When in FEM mode use any of the paths in the list below:

- Click **Home ► Force/Moment**. In the **Force/Moment Load** dialog box click **Advanced** and then in the **Spatial Variation** box, select **Interpolated Over Entity**.
- Click **Home ► Mesh ► Settings**. In the **FEM Mesh Settings** dialog box, click the **Create Solid-Shell Links** check box.
- Click **Home ► Force/Moment**. In the **Force/Moment Load** dialog box click **Advanced** and then in the **Distribution** box, select **Total Load at Point**.
- Click **Home ► Force/Moment**. In the **Force/Moment Load** dialog box click **Advanced** and then in the **Distribution** box, select **Total Bearing Load at Point**.

Benefits and Description

The enhancements listed below are added to help close the functional gap between FEM Mode and Native Mode:

- Interpolated Load Variation—You can interpolate loads in FEM Mode to vary the defined load linearly, quadratically, or cubically along a selected entity. Interpolated loads in FEM mode apply to the items listed below:
  - **Force/Moment** for surface and edge
  - Pressure
  - Heat for surface and edge
  - Prescribed temperature for surface and edge
- Solid-Shell Links—Depending on the model, the mesh created in FEM Mode automatically creates solid-shell links for all solid-shell interfaces where there is a bond connection and the nodes are merged. This is controlled through a new **FEM Mesh Settings** option which is active by default. To achieve a realistic solid-shell interface, an automatic mesh control is applied along the affected edges to create element sizes close to half the shell thickness on both sides of the edges. The shell and solid elements are connected by weighted links.
- **Total Load at Point**—You can apply a force or moment load and control the distribution as **Total Load at Point** in FEM Mode. This is available under **Distribution** when you click **Advanced** in the **Force/Moment Load** dialog box. This option applies a distributed load on a curve or surface that is statically equivalent to a load applied to a single point.
- **Total Bearing Load at Point**—You can control the distribution of a force or moment as **Total Bearing Load at Point** in FEM Mode to represent the force and moment that one cylindrically shaped part exerts on another. **Total Bearing Load at Point** is an advanced bearing load distribution on cylindrical surfaces or along circular edges and curves that is defined by a resultant force and moment at any selected point in the model.

Weighted Links in 2D Analysis

Support for weighted links is extended to 2D plane stress, plain strain, and axisymmetric models.

User Interface Location: To select a 2D mode of operation click **Home ▶ Model Setup** and then select **2D Plane Stress (Thin Plate)**, **2D Plane Strain (Infinitely Thick)**, or **2D Axisymmetric**. To create a weighted link click **Refine Model ▶ Weighted Link**.

Benefits and Description

Following the same process for applying weighted links for 3D models, you can apply weighted links to any type of 2D model. With this extended functionality you can apply the mass or load from a single-source dependent point and distribute it to a collection of targeted geometric entities which are independent references within the 2D model type. These geometric entities can be a combination of points, edges, and surfaces. As is the case with weighted links for 3D models, you control the translational degrees of freedom in the X and Y axes in 2D models.

Simple Fracture Mechanics Is Added

You can create simple fracture mechanics to determine if initial cracks in your model will grow under specified loading conditions.

User Interface Location: Click **Refine Model ► Crack**.

Benefits and Description

A new **Crack** idealization is available in both Structure and Thermal modes. This new idealization requires a license for PTC Creo Advanced Simulation Extension. With the new **Crack** idealization, you can select curve references for 2D models and individual surfaces for 3D models. When running AutoGEM, these references are treated as hard curves and hard surfaces.

Simulate checks are introduced to evaluate if there is a crack that cuts through the entire model. If so, a warning appears in the **Diagnostics** dialog box for the analysis.

The measure type, **Stress Intensity Factor**, is available in the **Measure Definition** dialog box. When selected, you can choose from one of the measure components listed below:

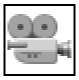
- **Mode I (Opening)** for 2D models
- **Mode II (Sliding)** for 2D models
- **Mode III (Tearing)** for 3D models

For **Stress Intensity Factor** you can select two spatial evaluation references, a crack and a point on the boundary of the crack. Using **Crack** idealization, you can easily perform simple fracture mechanics simulations on your models and determine if an initial crack could result in a failed design.

Improved Process to Determine the Best Analysis

It is easier to determine which type of analysis to run.

Benefits and Description



Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

The solver is enhanced to help you decide what kind of analysis is best to run for your model. In this example the part is constrained based on the four holes in the component and there is an applied force load depicting a downward force of 100 newtons. A material is defined and assigned to the component. It is this material and its defined properties that the solver uses to determine if the analysis you choose is accurate for the given component.

Before beginning, in the Model Tree, right–click the material that is assigned and select **Edit Definition**. In the **Material Assignment** dialog box, you see that the material is thermoplastic. Click **More** next to **Material**. In the **Material Definition** box you can see under **Failure Criterion** that the failed criterion is set as **Distortion Energy (von Mises)**. For this failure criterion you need to enter the defined **Tensile Yield Stress** for this material. It is this information that the solver uses to determine if the stress results in the material exceed the yield stress for the material that is assigned. To start, run a static analysis with a small deformation. Click **Analyses and Studies** to open the **Analyses and Design Studies** dialog box. Click **New ► File ► Static Analysis**. In the **Name** box type a name such as `Static_sd` for a small deformation. Select **Single-Pass Adaptive** as the convergence method and click **OK** and then run the analysis from the **Analyses and Design Studies** dialog box. After the analysis is complete the diagnostics appear.

Review the warning messages in the diagnostics window. In this case, the following warning appears: **The strains are large during this analysis, which may lead to inaccurate results. You should consider using the Calculate Large Deformations option, if applicable.** This helps you to determine if you chose the correct analysis. In this case the correct analysis was not run, based on the conditions of the model such as the constraints, the load, and the material assigned. The following warning also appears: **The yield condition has been exceeded for the material(s) listed below. You should consider using an elasto-plastic material, if applicable.** This helps you to determine if you chose the correct material, based on the analysis that you are trying to perform.

Based on the information you received, you can update these scenarios to resolve the problems brought to your attention from the warning messages. First, in the Model Tree, right-click the material and in the **Material Assignment** dialog box, click **More**, next to **Material**. In the **Materials** box select an elasto-plastic material such as `steel.mtl`. Click the arrow button to move the selected material to the **Materials in Model** list. In the **Materials in Model** list select `STEEL` and then click **OK**. In the **Material Assignment** dialog box **Material** changes to `STEEL`. Click **OK** and then click **Analyses and Studies** to run an analysis. In this case click **File ►**

New Static and in the **Static Analysis Definition** dialog box type a name for the analysis such as `Static_LDA`. Click the **Calculate large deformations** and **Plasticity** check boxes. Select the **Single-Pass Adaptive** method and **OK** and then from the **Analyses and Design Studies** dialog box run the analysis. After running the second analysis the two previous warning messages received during the static analysis with small deformation no longer appear in the diagnostics. This gives you confidence that this analysis with the defined elasto-plastic material has not exceeded the yield conditions based on the failure criterion. In addition, since you have set the calculations of large deformations, you know that the material has not exceeded the yield stress for the material assigned. You can have confidence that results displayed are accurate.

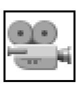
Whether you are a designer, a design engineer, or an analyst, the easy-to-use interface and robust solver helps you to obtain high-quality results.

New Workflow for Creating Notes

The workflow for creating notes is similar to the workflow in PTC Creo Parametric.

User Interface Location: Click **Tools** and then select a type of note from the **Notes** group.

Benefits and Description



Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

You can continue to create a part, assembly, or feature note by right-clicking the item in the Model Tree and selecting **Create a Note**. You can also create notes from the ribbon. Click **Tools** and then in the **Notes** group you can select from the types of notes listed below:

- **Unattached Note**
- **Tangent Leader Note**
- **Leader Note**
- **On Item Note**
- **Normal Leader Note**

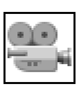
You enter the content of any note from the **Note** dialog box. In this dialog box you create the note and define information about the model or feature, without defining its placement. In this example, right-click the assembly and select **Create a Note ▶ Assembly**. In the text area of the **Note** dialog box, you may want to provide the name of the actual component. You can click **Insert** to insert text **From File** or **From Note** and you can click **Symbols** to select from text symbols to include in your note. After you define your note click **OK**. **Annotations** appears in the Model Tree. If you expand **Annotations** you can see the note that you created. The note is unattached and does not appear in the Graphics window. To change the display, placement, or other attributes of the note, right-click the note, select **Note Type**, and then select **On Item**, **Leader**, **Normal Leader**, or **Tangent Leader**. You can also right-click the note and select to **Delete**, **Rename**, or get the model **Information** pertaining to the selected note. You can repeat this operation to create a note on a particular part. For example, select the first component in the assembly, right-click and select **Create a Note ▶ Part**. The **Note** dialog box opens. You can for example, type the name of the material. Click **OK** and you can see the second annotation appears under **Annotations** in the Model Tree. It is grayed out and unattached. Right-click the note and select **Note Type ▶ Leader**. In the Graphics window, use the pointer to define the placement of the note associated to the component. For example, you can select an edge and then drag the pointer to preview placement of the note relative to the edge. Middle-click to place the note. This enhancement makes it easy to create notes and keep them in the model without cluttering the display of the model. You can capture important information at the assembly or the component level without being forced to place the note in the Graphics window.

Analysis Studies are Listed in the Model Tree

Saved analyses are listed in the Model Tree.

User Interface Location: In the Model Tree, under **Analyses**.

Benefits and Description



Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

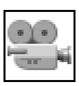
You can view and access saved analyses studies from the Model Tree. When you save an analysis, you can see that analysis at the bottom of the Model Tree under **Analyses**. If you expand **Analyses**, you can see the analysis listed with an icon representing the analysis type. Previously you accessed analyses through the **Analyses and Design Studies** dialog box. With this enhancement you can quickly interrogate, copy, and edit existing studies all from the Model Tree. If you right-click an analysis, you can **Edit Definition**, **Copy**, **Delete**, **Rename**, or get additional **Information** such as **Simulation Entity**. For example, you can create a **New Fatigue** analysis. It is important to give the analysis a unique and meaningful name. This is the name you see in the Model Tree. After you name the analysis and click **OK**, you can see it listed in the Model Tree. Run the analysis and save the model. The analyses remain in the Model Tree and are available when you open the model later. This enhancement is available in both Structure and Thermal modes. In the next example, a model is open in **Thermal Mode** and has a Thermal analysis. Note how that the analysis is listed in the Model Tree. Depending on whether you are running in Thermal Mode or in Structure Mode, you may want to see both analyses in the Model Tree. In the Model Tree under **Settings**, click **Tree Filters**. In the **Model Tree Items** dialog box, click **Simulate** and then click the **Thermal entities in Structure** and **Structure entities in Thermal** check boxes. You then see the analyses from both modes when you are in **Structure Mode** or **Thermal Mode**. You can view all the analyses types without switching between modes or opening a dialog box.

Improved Workflow for Interrogating Results

In the results windows there is a new object-action workflow. You can select common commands directly within the Graphics window and interrogate the results in the results window.

User Interface Location: Click **Home ▶ Simulate Results**.

Benefits and Description



Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

The results windows are enhanced to support the object-action workflow for many common operations. Previously you had to first select the tool from the ribbon. Now you can right-click to access these tools or you can select objects directly in the Graphics window. For example, you can edit the legend directly in the Graphics window. If you select `117.736` in the legend, you can immediately enter a new value. Type `100` and you are prompted with the question, **Do you want to redistribute levels linearly from first to last?** If you click **Yes** the legend updates and the corresponding fringe plot is updated on the model. In addition if you right-click on the legend the following commands are available for selection:

- **Continuous Tone**
- **Show View Min Max**
- **Scale Between Min Max**
- **Reset**
- **Invert Color Scale**
- **Hide**

Alternatively, you can click **Format** and then select **Edit** from the **Legend** group. When you have the pointer in the Graphics window you can right-click to access the commands listed below:

- **Dynamic Query**
- **Model Max**
- **View Max**
- **Edit**
- **Full Screen**
- **New Cutting/Capping Surface**
- **New Annotation**
- **Show Legend**

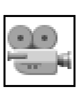
For example, right-click in the Graphics window and select **Model Max**. The **Model Max** is immediately displayed in the Graphics window. You also can perform the same actions by using commands in the ribbon. If you need to edit the **Result Window Definition**, you can double-click in the Graphics window to open the **Result Window Definition** dialog box instead of clicking **Edit** in the ribbon. Notice the two lists that are available in the top left of the Graphics window. You can use these lists to select the different quantity types for the model results. When you make changes to your selections in these lists, the updates appear immediately in the Graphics window. This makes interrogating your model easier. You no longer need to edit the display in the **Result Window Definition** dialog box. These enhancements improve the usability of the **Results** window so that you can efficiently inspect your analysis, validate your design, and identify areas in your model that may need improvement.

Defining Contact Interfaces with Finite Friction

You can define contact interfaces with finite friction. As a result, you can determine if sliding has occurred between components and the tangential force load transferred across each interface.

User Interface Location: Click **Refine Model ► Interface**.

Benefits and Description



Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

You can perform a large deformation contact analysis with finite friction in both 2D and 3D models. In this example you go through the process of defining a contact interface on an assembly. Click **Refine Model ► Interface** to open the **Interface Definition** dialog box. In the **Type** box select **Contact** and in the **Reference** box select **Surface-Surface**. After making a selection in the Graphics window you can select the **Properties** of the interface. In the **Friction** box you can select from the types of friction described in the list below:

- **None**—Defines a frictionless contact interface so that the components can freely slip relative to each other.
- **Infinite**—Defines a contact interface where the two components cannot slip relative to each other.
- **Finite**—You define a static coefficient of friction in addition to a dynamic coefficient of friction between the two references.

For this example, select **Finite**. You can now define the **Static Coefficient of Friction**, as `0.15`. You can also define the **Dynamic Coefficient of Friction** or for this example, click the **Same as static** check box. After applying finite friction to the contact interface, you can determine if sliding has occurred during a contact analysis. You can also determine how much sliding has occurred at each interface by using the new measure for tangential force, from which you can determine the tangential force load transferred for each of these interfaces. A user-defined measure, `Reaction_Y` is also defined. This represents the overall force required to push and snap this component into place.

Click **Home ► Analyses and Studies** to start the analysis process. In the **Analyses and Design Studies** dialog box, click **File ► New Static** and in the **Static Analysis Definition** dialog box, type a name for the analysis. Click the **Nonlinear / use load histories**, **Calculate large deformations** check boxes. Because contacts are represented with interfaces in the model, the **Contacts** check box is automatically selected. On the **Convergence** tab, click the **Press fit (initial interpenetration)** check box for the defined analysis. On the **Output** tab, under **Output Steps**, select **User-defined Output Steps**. In the **Number of Master Steps** box type `11` and then click **Space Equally** and **OK**. In the **Analyses and Design Studies** dialog box, run the analysis and display the study status. Review the report as you run the analysis. When the report is complete, scroll up to see when the slippage first occurred. In this example it occurred after step 7. You can now review the results. There are three results windows. On the left there is a graph for `Reaction_Y`. `Reaction_Y` is a user-defined measure created to represent the force required to snap the component into place. On the right you can see the tangential force across `Interface2`. In the middle there is an animation of the overall stress applied to the assembly as the components are snapping into place.

With this enhancement you can determine how much sliding occurs at each interface and the tangential force load that is being transferred for the interface.

Mapped Mesh Option for 2D Models

You can use the control option **Mapped Mesh** for 2D models.

User Interface Location: Click **Refine Model ▶ Control ▶ Mapped Mesh**.

Benefits and Description

Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

Click **Refine Model ▶ Control ▶ Mapped Mesh**. The **Mapped Mesh** option is helpful for the conditions listed below:

- If you have 2D models with contact interfaces where the accuracy of the calculated pressure is dependent on the refinement of the mesh in the contact area
- When performing a large deformation analysis
- On models with elasto-plastic material

In this example there is a 2D assembly of a shock absorber shim stack. This shim stack controls the flow of dampening oil within the shock absorber. During the compression or rebound stroke, the increased pressure of the shim stack deforms, allowing the oil to flow. In this example, you want to determine the total deformation of the stack and then tune the shims by changing the thickness, diameter, or the number of shims. The given interfaces between the shims are already defined and the given pressure load is applied. Click **Refine Model ▶ AutoGEM** to review the mesh. You can see that the resulting mesh is automatically generated. Since this analysis will be run with large deformation calculations enabled, you should try to refine the mesh at the given contact regions. Click **Control ▶ Mapped Mesh** to open the **Mapped Mesh Control** dialog box. Next select the geometry. The first area in which to map the mesh is the top shim. As you select from the surface geometry it automatically creates a quad mapped mesh in that region. You can see QUAD1 appears under **Meshing Regions**. Notice that this is a default three by three mesh. In the **Default** box, you can control the density of the mesh, or under **Edge Sets** you can select a given edge and subdivide that mesh along that reference. Note that only the **Quad** and **Tri** shapes are available under **Region Shape**.

Create another region to complete the operation. Click **AutoGEM** to create the mesh again. Where the mapped mesh regions were defined, the mesh is more refined. Now, when you run the analysis, the run time is shorter.

Display of Shells, Beams, and Fasteners is Improved

Shells, beams, and fasteners can be displayed as **Wireframe**, **Shaded**, or **Transparent**.

User Interface Location: Click the **Simulation Display** icon-.

Benefits and Description

Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

In this example, there is a model of an aircraft wing built using surface geometry. To analyze this, you need to create shell elements. Click **Refine Model ▶ Shell** to open the **Shell Definition** dialog box. In the Graphics window, select specific geometry. As you make your selections you can see the **Surfaces** listed in the dialog box. Define the thickness and click **OK**. The shell is created and is displayed as a solid. Click to open the **Simulation Display** dialog box. On the **Settings** tab in the **Shells** box, you can choose from **Outline**, **Wireframe**, **Shaded**, or **Transparent**. Select **Transparent** and then click **OK**. The shell updates and appears as transparent. If you select the shell, you see notes that indicate the thickness of the shell including its units at a specific location.

In the next example, you see the display enhancement for beams. In the Graphics window is a sketch feature that represents a frame structure. Click **Refine Model ▶ Beam** to open the **Beam Definition** dialog box. For references, select the curve and add it to the collector. For the **Beam Section** use the predefined Beam 1 and click **OK**. You can see the beam on the model as a shaded object. Click to open the **Simulation Display** dialog box. On the **Settings** tab in the **Beam** box, select **Transparent**, and click **OK**. The beam updates, and appears as transparent.

In the next example, first place a fastener in the model. Click **Refine Model ▶** **Fastener** to open the **Fastener Definition** dialog box. Next, select references. Based on the reference selected, the **Diameter** box is automatically populated. Choose your material and define the **Fastener Head and Nut Diameter**. Click **OK** and the fastener is created and is displayed as solid. You can change the display of the fastener from the **Simulation Display** dialog box.

This enhancement helps you to better visualize the shape, size, and location of shells, beams, and fasteners in your model.

Analyze Models with Failed Features

You can analyze a model with a failed feature.

Benefits and Description

Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

With this enhancement, models containing failed features can be opened in PTC Creo Simulate and analyzed. It is a good practice to resolve failed features in a model. In some cases, however, in the process of simplifying the model for analysis, features may be impacted due to child and parent relationships. In this case, moving a feature may result in the failure of other features.

In this example there is a model containing a number of failed features. You can still open this model in PTC Creo Simulate. After opening the model you receive a warning about the failure of the model to regenerate and information about what analyses types can be run for the model. The failed features are listed in the Model Tree but you can still define many of the traditional PTC Creo Simulate features such as forces, displacements, idealizations, connections, surface or volume regions and so on. The only limitation is when running an analysis or design study. In the **Analyses and Design Studies** dialog box, click **File**. You can see that **New Sensitivity Design Study** and **New Optimization Design Study** are not available.

This enhancement lets you continue analyzing your data, even with failed features.

Updated Process for Creating Linearized Stresses

You can easily create linearized stress plots for your model.

User Interface Location: In the **Results** Window, click **Linearized Stress**.

Benefits and Description

Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

In this example there is a simple pipe assembly. To generate the linearized stress you need to run a static analysis. In this example, a static analysis has been run for a pipe. Previously, to view the linearized stress results you would open the **Result Window Definition** dialog box, then select **Model** in the **Display Type** box, and **Linearized Stress** on the **Quantity** tab. The process has changed. Now, in the **Display Style** box select **Fringe** and on the **Quantity** tab select **Stress**. You can then set **Display Options** and click **OK and Show**. In the Graphics window you can see the **von Mises** stress in your model. In the ribbon click **Linearized Stress** to open the **Linearized Stress Report** dialog box. From this dialog box it is easy to select points to display the linearized stress. Click the arrow next to **Point 1** and then select a point on the model. In the **Points Selection** dialog box, click **OK**. For the **Point 2** select **On Opposite Surface** and you can see the results in the Graphics window. The detail of the linearized stress looks at the nodal data for the complex stress pattern found along the line and breaks it down into the following components:

- **Membrane**—Overall average stress
- **Bending**—The difference in stress from the inside point to the outside point
- **Mem+Bend**—The sum of the **Membrane** and **Bending** numbers
- **Peak**—The highest stress found along the line

If you zoom into the model you can see the **Point 1** and **Point 2** and the resulting data that is represented in the graph. From the **Component** box, you can choose a component such as **von Mises** and the graph and corresponding numbers update. This information defines what the stress will be along that line between those two points. Click **Done**.

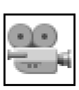
With this improved workflow you can quickly determine the linearized stress across critical areas of your data.

Fatigue Analysis with Multiple Load Sets

You can perform a fatigue analysis with multiple load sets.

User Interface Location: In the **Analyses and Design Studies** dialog box click **File ► New Fatigue**.

Benefits and Description



Watch a video on PTC University Learning Exchange, demonstrating the enhancement described below.

Depending on the type of load being applied, you may prefer to group these loads separately rather than put them into a single load set. In addition you may want to analyze certain loads but not others.

In this example, you are working with a gearbox component. In the Model Tree you can see three loads. As is the case with any fatigue analysis, you need to define a material that has a fatigue criteria set. Right-click `Material Assignment1` and select **Edit Definition** to open the **Material Assignment** dialog box to see what material is assigned. Click **More** to open the **Material** dialog box. Under **Materials in Model** select the material. In this example select `GRAY_CAST_IRON` and click **OK** to open the **Material Definition** dialog box. In the **Fatigue** area of this dialog box define a **Unified Material Law (UML)**. In the **Material Type** box **Other**, is added to the list of materials. In the **Surface Finish** box, the list of finishes is updated. When working with a model from an earlier release, a conversion is required. If this is the case, a warning appears indicating that some of the surface finishes defined in the model are no longer supported. You are prompted to provide the **Failure Strength Reduction Factor** to take these finishes into account. After the materials are defined and applied click **OK** to close multiple dialog boxes.

The first analysis you need to run is the static analysis. Click **Analyses and Studies** to open the **Analyses and Design Studies** dialog box. For this example, you can see that the analysis, `Gearbox`, is completed. Click **File ► New Fatigue** to run a fatigue analysis. In the **Fatigue Analysis Definition** dialog box there are some changes. The **Previous Analysis** tab is first, followed by the **Load History** tab. On the **Previous Analysis** tab, you can see that the load sets from the model are listed. For this example, select the three load sets and click the **Load History** tab where you define the **Desired Endurance**. For this example, **Desired Endurance** is set to `2e+5`. Next, define the load criteria for each load set. In the **Load Set** box, select the load. Select an **Amplitude Type** such as **Zero-Peak**. Complete this operation for each of the load sets in the model. Click **OK** and then from the **Analyses and Design Studies** dialog box, run the analysis. After the analysis is completed, you can view the results. In this example you review the fatigue fringe plot for **Log Life**. In the **Results** window you can see that most of the model has a **Log Life** of 20. This means ten to the power of twenty cycles will pass before the crack initiates. You can review the model and see areas that have a lower defined **Log Life**. To get a better perspective, you can select in the legend and type a value such as 15. This redistributes the levels from first to last so that you can get a better idea of where there are potential issues in the model.

This enhancement improves your ability to analyze a model with multiple loads and to determine the life expectancy of your design.