**Parametric Technology Corporation**®

# What's New in Pro/ENGINEER® Wildfire™ 2.0 Preproduction

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# Foundation Advantage

Simulation (Mechanism Dynamics, Design Animation)

**Title**

## Custom Loads

**Summary**

Mechanism Dynamics has been enhanced to allow users to incorporate custom written and compiled dynamic link libraries as input forces to a mechanism for dynamic analysis

**Benefits and Description**  (200words)

There is a new method for specifying the values for a force motor or an external force, called custom loads.  A custom load is an external program developed by Pro/Toolkit, which will be used to compute load values for force motors and external loads.

When defining the value for the force motor or an external force, users will be able to select the customer load program that will evaluate the custom load at each time step of the analysis.  In addition, at each time step the user can ask the Mechanism Dynamics engine to integrate a set of StateVariables in order to simulate a control system.

This enhancement enables users to have better control on real-world situation, taking into account specific force loads unique to the defined process.

**User Interface Location**

Mechanism / Force Motors / New / Magnitude – Custom Load

# Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Copy and Paste Commands for Features

**Summary**

The Windows style of **Copy** and **Paste** commands and key combinations now manipulate features.

**Benefits and Description**  (200words)

You can copy and paste features to other parts of the model quickly and easily. You can use the familiar **Copy** (CTRL+C) and **Paste** (CTRL+V) commands and key combinations in Windows applications. During the Paste operation, you can interact with the feature using the new dashboard and see an immediate preview of the features being copied. You copy both features and geometry with the same tool through a single unified workflow.

**User Interface Location**

Edit / Copy (CTRL+C) and Edit / Paste (CTRL+V)

## Foundation Advantage

Simulation (Mechanism Dynamics, Design Animation)

**Title**

## Computed Measures

**Summary**

Mechanism Design and Mechanism Dynamics now support the ability to create and define computed measures with generalized expressions.

**Benefits and Description**  (200words)

You can define measures with generalized expressions of one or more Pro/ENGINEER parameters or other measures as variables. With generalized expressions, you can convert result data from Mechanism Design and Mechanism Dynamics into a format for graph comparison or for use in assigning design-study goals and conditions. MECHANICA Motion has the equivalent capability.

**User Interface Location**

Generate Measure Results / Create a New Measure / Type – User Defined

## Foundation Advantage

Assembly and Weld (Pro/PROCESS for Assembly)

**Title**

Assembly Component Interfaces

**Summary**

When editing a component's placement, you can select predefined component interfaces. After using the new Search Tool to find and locate the component, specify the rule-based, interface placement options.

**Benefits and Description**  (200words)

You can use an existing set of component interfaces while editing the placement of components, or you can create your own component interfaces. You can efficiently edit a component's placement and place it in a new location. The Search Tool finds Interface Placement options based on features, geometry types, names, and parameters. This tool vastly increases the usability and power of component interface definitions during drag-and-drop component placement.

**User Interface Location**
Edit / Setup / Comp Interface

# Foundation Advantage

# Part Modeling (Part and Sketcher)

<Streamline And Expand Offset Curve Creation>

### Summary

A streamlined workflow for creating offset curves is available. The Offset tool operates like the tools for other similar features. `

### Benefits and Description

You can quickly create offset curves with the Offset tool. After you select a curve and enter the tool, the surface that the curve lies on is automatically selected. This provides an immediate preview of the offset curve. You can use the dashboard attributes of the feature to create the exact curve.

### User Interface Location

Edit / Offset

# Foundation Advantage

Model Analysis (BMX, ModelCHECK, Pro/PROGRAM)

**Title**

Check for Standard Model Notes

**Summary**

ModelCHECK now analyzes and verifies that all standard model notes exist in the model and reports the number of missing notes. This report meets the growing demand for drawingless modeling.

**Benefits and Description**  (200words)

With the move toward drawingless modeling, ModelCHECK helps you search for notes added to part or assembly models and verify if these notes are per standard. You add the standard notes in the ModelCHECK start configuration file. When you run this check on the model, ModelCHECK reports on whether the notes exist in the model. This check works for part, assembly component, subassembly, and top-assembly notes.

**User Interface Location**

Analysis / ModelCHECK / MC

# Foundation Advantage

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Boundary Blend User Interface

**Summary**

The Boundary Blend tool has a more intuitive user interface similar to other features in Pro/ENGINEER.

**Benefits and Description**  (200words)

The new Boundary Blend user interface provides the most commonly used functions, such as boundary conditions, directly in the graphics window and provides enhanced visual feedback.

**User Interface Location**

You can see enhancements to the Boundary Blend user interface both dynamically in the graphics as well as in the dashboard organization.

# Foundation Advantage

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Bend Damping

**Summary**

The concept of damping has been added to the Bend tool and in the new Sculpt and Spine types of Warp.

**Benefits and Description**  (200words)

By damping a bend, the effect of the bend is allowed to "decay" or reduce on the geometry further away from the bend axis. This damping allows for finer control of a localized region of influence for the bend.

**User Interface Location**

An additional size arrow on the Bend marquee is perpendicular to both the bend axis and the pivotal direction. You can drag either the base or tip of the arrow to adjust how far the bend dampening extends in the selected direction.

# Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Automatic Annotation Element Parameters

**Summary**

As you create Annotation Elements, you can automatically assign a group of parameters to them using a user-defined, custom parameter file.

**Benefits and Description**  (200words)

Custom parameter files provide values for a group of parameters that are to be applied to Annotation Elements upon creation. These values ensure that the proper parameters are applied to all Annotation Elements.

To make the parameters automatic for Annotation Elements, you must create a specially formatted XML file. This file defines the name, type, and default value of parameters. These parameters are to be added to all Annotation Elements by default during their creation. Specify the path to this file using a configuration option.

**User Interface Location**

A custom parameter file (XML) is accessed via a configuration option.

# Foundation Advantage

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**

Support for AutoCAD 2002 on Export and Import

**Summary**

You can export and import file version 2000 of AutoCAD DWG and DXF files. For the first time, export and import to Linux and 64-bit platforms is provided.

**Benefits and Description**  (200words)

Expanded support for AutoCAD includes the export and import AutoCAD DWG and DXF files from AutoCAD 2002 (file version 2000). You can export and import AutoCAD data in DXF and DWG formats from the following file versions:

- 2000 (default in Pro/E Wildfire 2.0)
- R14
- R13
- R12

**User Interface Location**
File / Save A Copy or File / Open

# Foundation Advantage

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**

AutobuildZ Integration with Pro/ENGINEER

**Summary**

AutobuildZ, the free plug-in application for moving to 3D, is integrated into the base Pro/ENGINEER package.

**Benefits and Description**

AutobuildZ allows you to build 3D parametric, feature-based solid models from 2D drawing data. Feature creation wizards guide you through the selection of entities and references directly in the 2D drawing to create 3D Pro/ENGINEER features. The following features are supported:

- Extrusions (Protrusions and Cuts)
- Revolves (Protrusions and Cuts)
- Simple Holes
- Datum Curves
- Datum Points
- Datum Planes

The unique sketch validation and fixing tools in AutobuildZ automatically confirm the entities selected for the sketch profile and provide options to fix any errors, such as open loops, intersections, or overlapping entities. Directly from the feature wizards, you make the fixes, never leaving the 2D environment.

AutobuildZ is integrated into Pro/ENGINEER Wildfire 2.0. Using the simple, but powerful suite of tools, you can now directly reuse your legacy 2D data in a 3D context. AutobuildZ serves as an excellent training tool. With AutoBuildZ, novice 2D users can learn about 3D solid modeling concepts and about feature building tools used in Pro/ENGINEER in the familiar 2D environment. To learn more about AutobuildZ, visit the product pages on PTC.com.

**User Interface Location**
Tools / AutobuildZ (available in Drawing mode only

## Foundation Advantage

Detail Drawing

**Title**

Automatic Creation of Ordinate Dimensions for Parts

**Summary**

The automatic creation of ordinate dimensioning has been expanded to include all parts.

**Benefits and Description**  (200words)

Automatic ordinate dimensioning can now be used for more than just sheet metal. On parts, you can create ordinate dimensions quickly and easily.

To use this command, select **Insert** > **Dimension** > **Auto Ordinate**, and select the surfaces you want dimensioned. Pro/E prompts you to select a baseline. After you select the baseline, the dimension is automatically created.

**User Interface Location**

Insert / Dimension / Auto Ordinate

# Foundation Advantage

Assembly and Weld (Pro/PROCESS for Assembly)

**Title**

Assembly Feature Enhancements

**Summary**

Assembly features are widespread in the design process. Dimensions, axes, tolerances, and 3D notes of assembly cuts that intersect parts are now available in the part. Performance, memory usage, and regeneration order is improved.

**Benefits and Description**  (200words)

Dimensions, axes, tolerances, and 3D notes associated with an assembly Hole feature that intersects a part can be displayed using the **Show and Erase** command on a drawing. This command eliminates the need to create these entity types manually. In Pro/NC, assembly-Hole feature axes are part of each hole and can be used for autodrilling and other NC tool paths. This eliminates the need to recreate the axis features in the Pro/NC model and significantly decreases the time it takes for NC programming. Stable, regeneration performance improves the usability of assembly features in the design process.

**User Interface Location**

In the Intersect slide-up panel on the dashboard, select Default display level: Part Level / check mark Show Feature Properties in Sub-Models.

# Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Annotation Features

**Summary**

Annotation features can add continuously displayed detail information to your 3D models.

**Benefits and Description**  (200words)

Annotation features are "containers" where annotations (notes, dimensions, geometric tolerances (GTOLs), surface finishes, and so forth) can reside. They are made of entities called Annotation elements and are displayed parallel to a plane that is selected at the time of creation. Annotation features support the new ASME Y14.41 standard for "Digital Product Definition Data Practices."

There are several benefits to using Annotation features:

- Annotations that are specific to a role (manufacturing, quality, assembly, and so forth) can be grouped and displayed as necessary
- Annotation features, as well as annotation elements, can have parameters
- Annotation features exist in the Model Tree in the order of creation

You can control the importance of annotation references. If an annotation is critical to the design of the model, the status of its reference can be made "strong." If a strong reference is removed from the model, the Annotation feature dependent upon it will fail. If a reference is weak, the Annotation feature regenerates with the missing reference.

Annotation features can be used in simplified representations and on layers. You can also create combined view states using a defined orientation and simplified representation that can be accessed quickly and easily.

**User Interface Location**

Insert / Model Datum / Annotation

PTC

## Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Annotation Attachments in Shrinkage by Scaling

**Summary**

Shrinkage by scaling supports Annotation references. You can forward references to Shrinkage features using the **By scaling** command.

**Benefits and Description**  (200words)

When defining shrinkage by scale, you can forward references automatically to the Shrinkage feature. These references ensure that the details associated with the original part are forwarded to the new feature as necessary.

The updated interface for creating shrinkage features includes dialog boxes for both the **By scaling** and **By dimension** commands. The dialog boxes simplify the definition of the shrinkage feature. To attach Annotation references to shrinkage features, select the **Forward Reference** check box while creating a Shrinkage feature using the **By scaling** command.

**User Interface Location**
Edit / Setup / Shrinkage / By scaling

# Foundation Advantage

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Animated Playback Enhancements

**Summary**

Animated playbacks in Mechanism Design, Mechanism Dynamics, and Design Animation allow for the display of datum curves in shaded mode and in 3D notes.

**Benefits and Description**  (200words)

You can generate animations to display critical information during playback.

To support ASME Y14.41 and to allow them to be shown in graphical annotation planes, 3D notes now display while blanking, unblanking, and moving components during animated playback in Mechanism Design and Design Animation.

This enhancement allows the display of datum curves during shaded animation as well as in wireframe animation, regardless of whether the component contains solid geometry.

**User Interface Location**

N/A

## Foundation Advantage

Part Modeling (Part and Sketcher)

**Title**

Analysis User Interface

**Summary**

Curve and surface analysis tools increase usability and productivity in the design process with the user interface.

**Benefits and Description** .

The Curve and Surface analysis user interface reduces the complexity of the analysis process and makes every analysis available from a single mouse click. The result is an increase in productivity and ease of use. The Curve and Surface menus have been consolidated under Geometry analysis. The Geometry analysis tools have a more intuitive user interface with "smart defaults," last-used values, direct edits, and dynamic updates within the Style and Restyle functional areas. Each Geometry Analysis tool has its own customizable icon, so you can to customize your working environment with the most commonly used tools for the job at hand.

**User Interface Location**

Analysis / Geometry

# Foundation Advantage

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**

Efficient Import of 2D Data

**Summary**

With the 2D Import dialogs (**Import DXF, Import DWG, and Import IGES for DXF, DWG, and IGES respectively)** boxes, you can configure and control the import of 2D into Pro/ENGINEER with a single user interface.

**Benefits and**

Besides configuring and controlling imported 2D data, you can map fonts from the 2D file being imported or appended to available Pro/ENGINEER fonts. The user interface for the import of 2D data is context-sensitive to data types. You can control import options such as:

- 2D views
- Dimensions as associative dimensions
- Variable-sized sheets

The **Import DXF, Import DWG, or Import IGES** dialog box opens when opening a DXF, DWG, or IGES file respectively or when appending a DXF, DWG, or IGES file into a drawing, layout, diagram, or other format in Pro/ENGINEER.

**User Interface Location**

File / Open or Insert / Shared Data / From File (if a drawing, format, layout or diagram is active)

## Foundation Advantage

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**

Improved Multi-Line Text Support

**Summary**

Improved control over the export and import of multi-line text expands and improves interoperability of Pro/ENGINEER with AutoCAD.

**Benefits and Description**  (200words)

You can create a single multi-line text note in Pro/ENGINEER from a multi-line text in a 2D interface file. In addition, when you export a Pro/ENGINEER drawing to an AutoCAD DWG or a DXF file, you can create a single MTEXT (AutoCAD entity) entity for any multi-line text note in the Pro/ENGINEER drawing.

**User Interface Location**

File / Open

## Foundation Advantage

Assembly and Weld (Pro/PROCESS for Assembly)

**Title**

Flexibility during Drag-and-Drop Component Assembly

**Summary**

When assembling a component using drag-and-drop operations, you can specify the part flexibility without having to redefine the component later.

**Benefits and Description**  (200words)

Specifying flexibility during component assembly dramatically increases the workflow and ease-of-use during component placement. Additionally:

- Dimension boundaries and surface finishes are available as varied items.
- Changes in the flexible model can be propagated back to the original model.

**User Interface Location**

Edit / Setup / Flexibility

## Foundation Advantage

Part Modeling (Part and Sketcher)

**Title**

Directional and Axial Pattern Enhancement

**Summary**

The Pattern tool now has directional and axial pattern types for more flexibility in establishing directions for patterns.

**Benefits and Description**  (200words)

You can pattern features around an axis or in any direction, regardless of how the original pattern was created. Previously, a Make Datum feature was required for axial patterns to incorporate the angular dimension for patterning. Additionally, you can pattern any feature in any direction without having to select any dimensions as defined in the original feature. Pattern preview is also available for directional and axial patterns. The preview consists of dots to represent the center of the pattern member. You can select these dots to turn pattern instances on or off during the definition of the pattern. Pattern dimensions have drag handles to allow for on-screen manipulation of the pattern.

**User Interface Location**

Edit / Pattern

# Foundation Advantage

Assembly and Weld (Pro/PROCESS for Assembly)

**Title**

Real-Time Dragging of Packaged Components

**Summary**

In Assembly mode, you can kinematically drag packaged components in the remaining degrees of freedom, based on existing placement constraints.

**Benefits and Description**  (200words)

Design assemblies no longer require a skeleton or the reconstruction of assemblies in Mechanism to evaluate mechanism behavior. Basic dragging allows full interrogation of the assembly through its full range of motion, based on the degrees of freedom in the placement constraint. For a component with no placement constraint, you can move it into the vicinity of the assembly for placement. This capability vastly improves the workflow and usability of assemblies and mechanisms.

**User Interface Location**

During component placement: Hold down CRTL+ALT and middle-click.

# Foundation Advantage

Electrical Design (Diagram, Cabling, Harness,ECAD)

**Title**

**ECAD Importing Improved**

**Summary**

The ability to control the importing of hole information using IDF 3.0 data has been improved.

**Benefits and Description**  (200words)

When directly importing an assembly during the board creation process, you have a choice of hole filters. You can now automatically filter PIN holes when creating a board. During the import of IDF 3.0 data, the **Hole Import** list displays PIN, VIA, MTG, TOOL, and other information.

By clicking **Options** on the **ECAD – Import** dialog box (available from the **Import New Model** dialog box), you can modify the selection of holes to import as required.

There are two new configuration options:

- `ecad_pin_hole_import` controls the default settings for the hole filter of type PIN. When the option is set to `no` (default), PIN holes within the IDF 3.0 file are not imported.
- `ecad_via_hole_import` controls the default settings for the hole filter of type VIA. With the option is set to `yes` (default), VIA holes within the IDF 3.0 file are imported.

**User Interface Location**
N/A

# Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

New Feature Move Tool

**Summary**

This new enhancement implements the Windows style Copy and Paste Special functionality to move and rotate features.

**Benefits and Description** (200words)

Users can now quickly and easily move, translate and rotate features to other parts of the model using the already familiar Copy <CTRL-C> and Paste Special functions found in a majority of Windows based software applications. Multiple transformations are now possible during the operation. The moved features are now Editable allowing the user to redefine the transformation without having to recreate the moved features from scratch. During the Paste Special operation, users have the ability to interact with the feature using the new dashboard and seeing immediate preview of the features being moved.

**User Interface Location**

Edit > Copy <CTRL>-c  and  Edit > Paste Special

## Foundation Advantage

Detail Drawing

**Title**

Moving Dimensions While in the Show/Erase Dialog Box

**Summary**

The ability to adjust the location of dimensions without exiting the **Show/Erase** dialog box has been reinstated.

**Benefits and Description** (200words)

Now you can pause the **Show/Erase** dialog box operations while showing dimensions. You can complete the task of showing dimensions without interrupting your workflow.

To use this functionality, on the **Show/Erase** dialog box, show the dimensions. Right-click to pause the show and erase tool. Adjust dimensions as necessary, and right-click again to resume operations.

**User Interface Location**

While inside the Show/Erase dialog box, right-click to select the command to pause or resume the show and erase process.

# Foundation Advantage

Model Analysis (BMX, ModelCHECK, Pro/PROGRAM)

**Title**

ModelCHECK Detailed Report (HTML)

**Summary**

ModelCHECK provides a new, static detailed report in HTML for Pro/ENGINEER data. The detailed report lists errors and warnings, and you can configure it to include other pertinent information from the data being checked.

**Benefits and Description**  (200words)

You can use the static, detailed report in HTML in other downstream applications. The different report types that can be generated follow:

- Single part, assemblies, drawing summary report
- Bill of materials (Assembly Component Models) summary report
- Drawing and model summary report

These static reports are generated from the ModelCHECK XML files generated when you run ModelCHECK. These XML files are filtered through a custom report template. You can edit this template to display additional data information in the report.

**User Interface Location**

N/A

## Foundation Advantage

Model Analysis (BMX, ModelCHECK, Pro/PROGRAM)

**Title**

ModelCHECK Configurator Simplifies the Configuration Process

**Summary**

The ModelCHECK Configurator streamlines the configuration process of ModelCHECK. You can easily find, edit, and create new configurations and text files.

**Benefits and Description**  (200words)

Accessed through Pro/ENGINEER, the ModelCHECK Configurator makes it easy to create, find, and edit ModelCHECK configurations and files to meet company standards and best practices. The new user interface makes configuration easy with direct links to online Help. You can also scale the configurations, adding things like new checks and conditions, to conform to the current environment.

**User Interface Location**

Analysis / ModelCHECK / Configure ModelCHECK

## Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

New Mirror Tool

**Summary**

This new enhancement provides a new workflow to mirror geometry and features.

**Benefits and Description**  (200words)

You can now mirror geometry and features for multiple transformations using one tool. The mirrored features can be created independently and the sketch of sketched-based features can be unlinked from the original. During the mirroring operation, you define the mirror definition using the already familiar dashboard.

**User Interface Location**

Edit / Mirror

## Foundation Advantage

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Joint Axis and Slot Friction

**Summary**

Mechanism Dynamics incorporates the ability to apply frictional forces on joint axes and slot connections. Additions to measure types facilitate the inspection of applied friction force values.

**Benefits and Description**  (200words)

For more accurate dynamic modeling of mechanisms, friction must be taken into account. You can now incorporate frictional influences within mechanism joints to appropriately simulate real-world behavior. Each editable joint axis has a friction coefficient specification enabled. With friction properties, joint friction forces are computed and applied to the joint axis in a direction opposite to the direction of motion in the joint. Joint axis friction is enabled for all editable joint axes, but does not consider coupling effects between multiple degrees of freedom in a particular connection.

**User Interface Location**

Application / Mechanism

# Foundation Advantage

Assembly and Weld (Pro/PROCESS for Assembly)

**Title**

New Layer Functionality

**Summary**

You can add items from lower-level subassemblies, collect layer items using the Layer Tree, and define rule-based layers to add and to remove layer items.

**Benefits and Description**  (200words)

- Assembly layers can now contain components from lower levels, eliminating the need to create same-named layers for all levels of subassemblies to control the display of subassembly components.
- When selecting items using the object-action workflow, you can select layer items using the Layer Tree, which makes it easier to identify and select components from layers.
- When defining rules for layers, use the template models to define the rules. Using rule-based layers previously defined in a model (part, assembly, or drawing), gives you additional flexibility with standards-based layer definition and assignment.

**User Interface Location**

From the Model Tree, click Show / Layer Tree.

# Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Annotations Support on Layers

**Summary**

Three-dimensional annotations, Annotation features, and Annotation Elements can be placed on layers.

**Benefits and Description**  (200words)

You can manage 3D annotations, Annotation features, and Annotation Elements by layers, using manual placement or by creating rules.

To use this functionality, create a 3D annotation or an Annotation feature with a graphical element. Select a layer from the Layer Tree and edit the properties. You can add an annotation (or an Annotation Element) by selecting it from the graphics window or the Model Tree. To add an Annotation feature, select it from the Model Tree or select the Annotation Element from the graphics window, and then right-click and select **Select Feature**. You can also define a rule to automatically add the annotation to a layer.

**User Interface Location**

From the Layer Tree, select the layer, right-click and select Layer Properties from the shortcut menu. Select the item to be added or create a rule to automatically place the annotation on a specific layer.

## Foundation Advantage

Other Functional Areas

**Title**

PTC.Setup Workflow Enhancements

**Summary**

PTC.Setup has been redesigned to improve the installation workflow, to simplify the installation process, and to improve the speed of installation. The modern user interface provides additional ease of installation.

**Benefits and Description**  (200words)

PTC has redesigned PTC.Setup to consolidate screens and to remove unnecessary screens. The modern user interface simplifies the steps to install applications. From PTC License Manager to PTC Help, the new workflow in PTC.Setup makes installation easier.

**User Interface Location**
N/A

PTC

## Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

ProductView Express and Annotations

**Summary**

Three-dimensional annotations created on an Annotation plane are viewable in ProductView Express.

**Benefits and Description**  (200words)

The ProductView Express preview browser shows all annotations in a model created using an Annotation plane. You can see the model, along with any annotations that are attached to it, in the preview window before you open or assemble the part.

To use this functionality, browse to the folder where your model resides and select the model. The preview window will show your model, including all visible annotations. You can also see annotations in the preview window when you open a part.

**User Interface Location**
N/A

## Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

ProductView Express and UDFs with Stored Reference Models

**Summary**

In the preview window of the ProductView Express browser, you can view User-defined Features (UDFs) with their stored reference models.

**Benefits and Description**  (200words)

Now you can see UDFs "as created" with their reference models in the preview window before you place them in the model. The preview window enables visual verification of the selected UDF before placement.

To use this functionality, browse to the folder in which your UDF resides and select the GPH file. A preview of the UDF and the reference model appear in the window.

**User Interface Location**
ProductView Express preview window

# Foundation Advantage

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**

Multiple Sheet Sizes in the Import of DXF and DWG Drawings.

**Summary**

The import of DXF and AutoCAD DWG files with multiple sheet sizes expands and improves the interoperability of Pro/ENGINEER with AutoCAD.

**Benefits and Description**  (200words)

For the imported DXF or DWG drawing, you can either maintain the size of each sheet or use a standard Pro/ENGINEER sheet size (A4, A, C, and so forth) that is closest to the original sheet size.

**User Interface Location**

File / Open

## Foundation Advantage

Part Modeling (Part and Sketcher)

**Title**

Offset Analysis

**Summary**

The Pro/ENGINEER suite of analysis tools includes an Offset Analysis tool for greater analysis of how curves and surfaces will offset downstream in the design process.

**Benefits and Description**  (200words)

Offset Analysis, a new analysis type, provides dynamic visualization of the effects of an offset on curves and surfaces within Pro/ENGINEER. With Offset Analysis, you can clearly identify issues relating to shells, thicken operations, and so forth, early in the design process. Early identification of issues helps eliminate costly rework of offset issues late in the design process.

**User Interface Location**

Analysis / Geometry / Offset, and select the desired curve or surface for display.

# Foundation Advantage

Sheetmetal (Design, NC)

**Title**

Flat Wall Feature

**Summary**

The Flat Wall feature consolidates and simplifies the creation of sheetmetal flat walls.

**Benefits and Description**  (200words)

This feature offers the quickest and easiest way to create flat walls in sheetmetal using the object-action and action-object paradigm. It comes with industry-standard, predefined shapes for the walls (rectangular, trapezoid, L, and so forth). You can define your own section using standard Pro/ENGINEER sections.

Direct manipulation of all standard shapes and types of wall relief is available through handles. Whenever necessary, automatic wall relief is added.

Using the dashboard, you can flip the section side and the section thickness side. There are no constraints in the selection of the edge from which to generate any wall.

**User Interface Location**

Insert / Sheetmetal Wall / Flat or the new icon  in Sheetmetal mode

# Foundation Advantage

Sheetmetal (Design, NC)

**Title**

Flange Wall Feature

**Summary**

The Flange Wall feature consolidates and simplifies the creation of sheetmetal extruded, swept, and hem walls.

**Benefits and Description**  (200words)

This feature offers the quickest and easiest way to create flange walls along an edge on a sheetmetal part using the object-action and action-object paradigm. It comes with industry-standard, predefined shapes for the walls. You can define your own section using standard Pro/ENGINEER sections.

Direct manipulation of all standard shapes and types of wall relief is available through handles. Wherever necessary, automatic wall relief is added.

Using the dashboard, you can flip the thickness of the sections. There are no constraints in the selection of the edge from which to generate any wall.

**User Interface Location**

Insert / Sheetmetal Wall / Flange or click  in Sheetmetal mode

# Foundation Advantage

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Net Load Measure

**Summary**

Net Load Measures have been enhanced in Pro/ENGINEER Wildfire 2.0 to provide greater control and reporting for total loads and loads applied by a force or torque.

**Benefits and Description**  (200words)

These new enhancements address the inconsistencies of the Connection Reaction Measure and the shortcomings of the Load Reaction measure.

- These new measures will now calculate the total load acting at a joint. When a joint axis is subjected to more than one load (such as force motor, damper, spring etc), it is not possible to query the total load at that axis.
- **Connection Reaction Measure**: Report the correct reaction value for axial force measures (on translational axes) and axial moment measures (on rotational axes) - This value should take into consideration the constraints (i.e. locking) created on the concerned joint axis and friction force if any. Any force entity present on the joint axis should not be transferred to the axis as a reaction as this will be measured as a Net Load and not as a reaction.
- **Net Load Measure**: This measure was formerly called the Load Reaction measure. Accept Force, Torque and Joint Axis in the selection for a Net Load measure (in addition to Spring, Damper and Motor).

**User Interface Location**

Generate Measure Results / Create a New Measure / Type – Net Load

# Foundation Advantage

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Multiple-Measure Support

**Summary**

Profiles of a servo motor, force motor, and force/torque have been enhanced to include more than one measure variable in the user-defined function definition.

**Benefits and Description**  (200words)

User-defined functions are used to define the profile of a servo motor, force motor, and force/torque. The functions consist of one or more expression segments and a domain associated with each of the expression segments. These functions can be defined by including arithmetic operators, mathematical functions, constants, and a variable.

User-defined functions for servo motors are always a function of time. In contrast, user-defined functions for force motor and force/torque can be defined either as functions of time or measure variables. With multiple-measure support, you can build force motor and force/torque profiles with more than one measure variable in the user-defined function definition. These functions enable complex force motor and force/torque profiles, which are dependent on multiple measures in a mechanism, to exist.

**User Interface Location**

Mechanism / Force Motors or Mechanism / Force/Torque

# Foundation Advantage

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Spine Command

**Summary**

The **Spine** command provides a method of creating curve-based deformations to parts and sets of surfaces from native geometry to imported IGES files.

**Benefits and Description**  (200words)

The **Spine** command provides an excellent means to make specific and highly complex deformations to a model. It is a powerful addition to the set of Warp tools. The spine deformation is a global, character-line reshaping and is based on a curve or a spine. When the spine is edited (via control points, tangents, and so forth), it influences the model around it. The spine can act in one of three directions (linear, cylindrical, or radial) and damping regions can be applied to localize the deformation.

**User Interface Location**

Warp / Spine

# Foundation Advantage

Other Functional Areas

**Title**

PTC.Setup Workflow Enhancements

**Summary**

PTC.Setup has been redesigned to improve the installation workflow, to simplify the installation process, and to improve the speed of installation. The modern user interface provides additional ease of installation.

**Benefits and Description**  (200words)

PTC has redesigned PTC.Setup to consolidate screens and to remove unnecessary screens. The modern user interface simplifies the steps to install applications. From PTC License Manager to PTC Help, the new workflow in PTC.Setup makes installation easier.

**User Interface Location**
N/A

## Foundation Advantage

Part Modeling (Part and Sketcher)

**Title**

Reflection Analysis

**Summary**

The **Reflection** command for analysis provides visualization of the surface quality during the design process. You can visualize surface flow lines and identify problem areas early in the design process.

**Benefits and Description**  (200words)

Reflection Analysis (or Zebra Striping) is a new analysis type for geometry analysis. Reflection Analysis shares the user interface for Curve and Surface Analysis. The command is also available from a shortcut menu. This dynamic method of analysis checks surface and connection quality within Pro/ENGINEER.

**User Interface Location**

Analysis / Geometry / Reflection. In the Reflection dialog box, select the surface patches of interest and the reflection analysis is displayed**.**

# Foundation Advantage

Rendering

**Title**

## Reflection of the Environment on a Model

**Summary**

Environment mapping allows the environment, that is, room appearances, to be reflected in the model.

**Benefits and Description**  (200words)

Environment mapping not only provides an increased level of realism in realtime, but also allows you to quickly set up proper reflections before performing a higher quality static rendering. To use environment mapping, first apply desired textures to the room walls. After setting up the room, select **View** > **Display Settings** > **Model Display**, select the **Shade** tab from the **Model Display** dialog box, and select **Realtime Rendering** from the **Enable** area of the dialog box. **Environment Mapping** becomes available. The option to hide the environment (hide the room) also becomes available, so you can display model reflections without seeing the room reflections.

**User Interface Location**

Environment Mapping is located under the Shade tab in the Model Display dialog box.

# Foundation Advantage

Model Analysis (BMX, ModelCHECK, Pro/PROGRAM)

**Title**

Redesign of the RuleCHECK User Interface

**Summary**

RuleCHECK, a component of ModelCHECK, has been redesigned to take advantage of the Pro/ENGINEER browser and to make RuleCHECK easier to use. RuleCHECK allows companies to easily document and enforce important engineering rules.

**Benefits and Description**  (200words)

With RuleCHECK, design information is made available in a proactive fashion and is integrated into the Pro/ENGINEER browser. The new user interface not only makes RuleCHECK compatible with ModelCHECK, but it also provides the ability to take advantage of dynamic HTML links to other pertinent information and makes it easy to use.

As a design advisor, RuleCHECK offers immediate access to your company's design knowledge. Often this information is stored in design manuals, industry design codes, and in the minds of senior designers. Engineers must know when and where to search for this information. Today, products are becoming more complex, design time is reduced, and there is considerable turnover and reorganization of engineering staff. Immediate access to correct and up-to-date engineering knowledge is critical to the success of many projects.

**User Interface Location**

Analysis / ModelCHECK / RuleCHECK

# Foundation Advantage

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Sculpt Command

**Summary**

The **Sculpt** command provides a method for deforming a part or set of surfaces via a control mesh, greatly expanding the functionality within the Warp feature.

**Benefits and Description**  (200words)

The control mesh for sculpted geometry can be located on any of the six faces of an imaginary bounding box that surrounds the area to be warped. You have full control over the density of the mesh via input to the dashboard. The sculpting action pulls on rows, groups, or individual mesh points. You can control the movement of the points to achieve the smoothest deformation and desired result and preserve symmetry while using the **Sculpt** command. You can view and edit the sculpt parameters outside of the feature, as with all other Warp types.

**User Interface Location**

Warp / Sculpt

## Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Reduced Regeneration Time with a Progress Bar

**Summary**

Regeneration times can be as much as 80% less for assembly models. You can view the progress of the regeneration with the Regeneration Progress bar.

**Benefits and Description**  (200words)

Using Smart Regeneration technology within Pro/ENGINEER Wildfire, only features and components requiring regeneration because they are children of modified features are regenerated. In Part mode, shell, draft and surface Copy features are now checked and only regenerated as required. This smart regeneration saves time regenerating large, complex parts and assemblies. During regeneration, a progress bar shows the remaining time for the regeneration. In addition, scrolling text above the progress bar identifies the model and counts down the number of features left to be regenerated.

**User Interface Location**

Edit / Regeneration

## Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Reduced Regeneration Time with a Progress Bar

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**Benefits and Description**  (200words)

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**User Interface Location**

Edit / Regeneration

# Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Regeneration Footer

**Summary**

Entities that are always created or evaluated at the end of the feature list are automatically placed into the regeneration footer.

**Benefits and Description** (200words)

Some entities are evaluated at the end of regeneration regardless of their position in the Model Tree. A new section in the Model Tree clarifies the organization of information. Items with no real dependency on their position within the Model Tree are automatically placed into the regeneration footer, for example, a zone. Zones are always evaluated at the end of regeneration and cannot be referenced by other features.

Some features in the active Model Tree can be moved to the footer. For example, if you want a published geometry, data-sharing feature to always be evaluated at the end of regeneration, you can move it to the footer after creation.

To use this functionality, select the item to be moved into the footer, right-click and select **Move to footer** from the shortcut menu. To remove an item from the footer, expand the footer, select the item, and right-click and select **Move from footer**.

**User Interface Location**
Model Tree beneath Insert Here

## Foundation Advantage

Part Modeling (Part and Sketcher)

**Title**

Completed the Curve Wrap Tool

**Summary**

The curve Wrap Tool now provides a streamlined workflow and additional functionality for creating wrapped curves.

**Benefits and Description**  (200words)

The Wrap Tool for datum curves now prevents the feature failure when the parent geometry is redefined from a surface to a solid.  The Preview of the geometry during curve creation has been dramatically improved particularly with sketches contain text.  The additional functionality includes being able to wrap separated curves on connected surfaces and the portion of the curve that cannot be wrapped has the option to be trimmed.

**User Interface Location**

Edit / Wrap

## Foundation Advantage

Assembly and Weld (Pro/PROCESS for Assembly)

**Title**

View Manager Includes Cross Sections and Zones

**Summary**

The View Manager is a powerful tool for managing and easily switching between simplified representations, explode states, orientation, component display states, and now zones and cross sections.

**Benefits and Description**  (200words)

The consolidation of all the viewing states of a model into one centralized location dramatically improves the workflow and usability of the various viewing tools. Within the View Manager, it is now possible to capture views in one saved "All State." This All State represents the combination of a simplified rep, in a specific orientation with some components shaded and others in wireframe, showing a cross section and several components exploded. This All State is available for design reviews, product documentation, or vendor meetings.

**User Interface Location**

View / View Manager

## Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Feature Verify Improvements

**Summary**

Feature Verify is now easier to use.

**Benefits and Description**  (200words)

During feature creation, when selecting feature Verify (eyeglasses in the dashboard), Pro/ENGINEER will regenerate the feature and display it on the model.  At this point, the user can interrogate the current feature in the model using the Analysis tools to measure any geometry item of interest.  This makes it easy for the user to know that the feature is correct and right before finishing the feature creation and regenerating the remainder of the mode.  To exit Verify, the option Exit Verify is now available using the Right Mouse Button (RMB).

**User Interface Location**
Press the Eyeglasses Icon in the Dashboard during Feature Creation

# Foundation Advantage

Part Modeling (Part and Sketcher)

**Title**

Improved Workflow for Variable Section Sweep Features

**Summary**

Users can now create Variable Section Sweep features using an improved and more streamline workflow within the tool.

**Benefits and Description**  (200words)

The improved workflow for Variable Section Sweep features is now easier to understand and provides a smoother workflow.  Feature drag handles are now available for use during feature creation.  The feature node in the Model Tree has also been updated with a new icon and a new feature type description to make it easier for the user to recognize.  Graphical highlighting for tangent trajectories provides the user with immediate feedback during feature creation.  The option to split surfaces using trip, allowing the user to keep both sides of the feature adds to the power of the Variable Section Sweep feature.

**User Interface Location**

Insert > Variable Section Sweep

# Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Annotations in Inheritance Features

**Summary**

With various annotations and Annotation elements in inheritance features, you can vary the surface-finish values. You now have greater control over the status of copied inherited annotations.

**Benefits and Description**  (200words)

Improved definition of inheritance features provides additional options and more control over annotations. Inheritance features now support variable surface-finish values. Additional controls are provided for the copy status of geometric tolerances (GTOLs), surface finishes, notes, and symbols. Annotation features are also supported in inheritance features with the same controls for Annotation Elements:

- Variable surface-finish values–Specifies a new value for surface finishes included in inheritance features
- Additional copy status control–Controls the copy status of the individual annotations from the source model. You can copy all annotations, exclude all annotations, or select the annotations you would like copied into the inheritance feature.

To use these functionality improvements, create an inheritance feature and select **Var Annotations** to configure the annotations. The new options are located in the **Var Annotations** section.

**User Interface Location**
Insert / Shared Data / Inheritance from other model / Var Annotations

# Foundation Advantage

Model Analysis (BMX, ModelCHECK, Pro/PROGRAM)

**Title**

ModelCHECK as a Gatekeeper for Windchill

**Summary**
Windchill now uses ModelCHECK as the gatekeeper to the Windchill database. ModelCHECK helps verify that the checked-in data is of good quality and meets company specifications.

**Benefits and Description**  (200words)

ModelCHECK promotes the use of company standards and best practices to improve the effectiveness of downstream design reuse. With this new enhancement to Windchill, the ModelCHECK Gatekeeper is used to help filter out data that is being check-in into the Commonspace that does not meet company standards or best practices.

Companies can configure a unique ModelCHECK Gatekeeper filter to define the requirements for a successful check-in into the Commonspace. For each lifecycle state, companies can control the density of the filter by selecting the Gatekeeper to check for various ModelCHECK read-only parameters or for modification of the data.

During the check-in process, ModelCHECK Gatekeeper completes the check-in for data that meets the established requirements, or limits the check-in if any of the components do not pass the check. If the check-in fails, Windchill reports back the data's particular errors and warnings.

**User Interface Location**

File / Check-in

# Foundation Advantage

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Specification of Units for Parameters

**Summary**

Units can be specified for user-defined parameters during creation and when using parameters in relations. Parameter units are recognized and supported by Pro/INTRALINK and PDMLink.

**Benefits and Description**  (200words)

Specify parameters as unitless or with units during their creation or when using them in relations. If of differing units, parameters in relations are scaled correctly based on calculations. For example, if the model units are in millimeters the relation TOTAL_LENGTH = 1 [in] + 5[mm] would evaluate TOTAL_LENGTH = 30.4[mm]. This scaling dramatically increases the flexibility and power for defining custom units. You can mix units when writing relations without having to convert the parameter values before using them. Scalar, or unitless parameters, are still supported and can be used with parameters with units in relations.

**User Interface Location**

Tools / Parameters / Properties

# Foundation Advantage

Detail Drawing

**Title**

Undo/Redo Functionality for Drawings

**Summary**

Undo/Redo functionality is now supported in Pro/ENGINEER. Users will have the ability to quickly undo or redo selected commands in drawings.

**Benefits and Description**  (200words)

Undo/Redo functionality in drawings gives you the ability to quickly undo (or redo) accidental changes to drawings. This eliminates the need to manually re-create items that may have been accidentally deleted or moved. The first implementation of undo/redo functionality will be limited to selected commands. The commands in drawings are divided into 3 categories: Supported commands, unsupported commands that clear the stack, and unsupported commands that do not clear the stack. Supported commands are actions that can be undone and redone. Examples of these commands are: insert>dimension and moving a drawing view. To prevent the user from unintentionally undoing an action that was executed before several unsupported commands, there are some actions that clear the stack. Examples of these types of commands are File>Open and Format>Text Style. Unsupported commands that do not clear the stack are essentially invisible to undo/redo. These commands are those that have no real impact to the drawing itself, like changing the size of the working window and viewing commands. Full details of the supported commands can be found in the help documentation.

**User Interface Location**
Edit>Undo OR Edit>Redo

# Foundation Advantage

Part Modeling (Part and Sketcher)

**Title**

Expanded Trim Tool Functionality

**Summary**

Additional functionality has been added to the Trim Tool to allow for a better, more streamlined workflow when trimming surface and curve geometry.

**Benefits and Description**  (200words)

The Trim Tool now provides access to full chain selection when selecting geometry to use in the trim operation.  Additionally, the user now has the silhouette trim functionality as part of the Trim feature dashboard.  The feature preview visibility has also been enhanced to provide a better visual of the feature during definition.

**User Interface Location**
Edit > Trim

# Foundation Advantage

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Transparency at Animation Time

**Summary**

Design Animation has been improved so you can define the transparency for assembly or its components at various animation times.

**Benefits and Description**  (200words)

Design Animation supports the display of components using various display styles, such as wireframe, shaded, hidden-line, and so forth, at various animation times. You can now set the transparency of the components to visible or invisible. This becomes valuable when the motion of the components inside another component must be animated. To better visualize the motion of the inner components, you can fade the outer components.

**User Interface Location**

Animation / Trans @ Time

# Foundation Advantage

Simulation (Mechanism Dynamics, Design Animation)

**Title**

System and Body Measures

**Summary**

System measures and Body measures have been added to Mechanism Design and Mechanism Dynamics. These measures generate additional information from either a dynamic or kinematics analysis.

**Benefits and Description** (200words)

After you define the System and Body measures, you can monitor behavior throughout the dynamic or kinematics analysis.

- **Body**—Measures several quantities that describe the behavior of a selected body. You can monitor the behavior of a body within the overall mechanism system.

- **System**—Measures several quantities that describe the behavior of the entire system. You can use the results as objectives in design studies or include them in more complex expressions or in computed measures.

**User Interface Location**

Generate Measure Results / Create a New Measure / Type – System or Body

# Advanced Assembly Extension (AAX)

Assembly and Weld (Pro/PROCESS for Assembly)

**Title**

Inheritance Feature Enhancements

**Summary**

Building on its powerful and robust features, the enhanced Inheritance Features (IF) functionality provides easier access and more flexibility in a design.

**Benefits and Description**  (200words)

You access the subfeatures within an Inheritance feature by selecting them from the graphics window or by using the new Search Tool. You no longer must choose the features from the Model Tree. When creating external Inheritance and external Merge and Cutout features, you can use placement constraints expanded from CSYS to CSYS to all existing placement constraints found in Assembly.

**User Interface Location**
Insert / Shared Data / Inheritance from Other Model

## Interactive Surface Design Extension II (ISDXII)

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Copy Proportional Command

**Summary**

An enhancement to the **Copy** command inside Style allows copied curves to be scaled proportionally.

**Benefits and Description**  (200words)

The **Copy Proportional** command operates on free and planar Style curves in the current feature. During the operation, you have dynamic, visual feedback with respect to the size and shape of the geometry.

**User Interface Location**

Select a curve, select Edit / Copy Proportional, and arrows connect the start and end points of the original and copied curves. Drag the arrowheads to achieve the desired shape.

## Interactive Surface Design Extension II (ISDXII)

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Composite Surface Creation

**Summary**

You can select compound strings of curves to be treated as a single boundary edge during surface creation within Style.

**Benefits and Description**  (200words)

All Style surfaces can use compound curves as boundaries. This ability extends the use of Style surfaces by allowing increased use of sketched curves as well as compound strings of Style curves. The compound curve elements, however, must be at least tangent continuous.

**User Interface Location**

Hold down SHIFT while selecting the desired segments during the boundary curve selection while creating a surface.

## Interactive Surface Design Extension II (ISDXII)

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

4-View Enhancements in Style

**Summary**

In 4-view mode in Style you can dynamically resize four panes, allowing for a more usable work area while designing.

**Benefits and Description**

Changing the size and aspect ratio of the panes allows you to have one relatively large "primary" pane and use the other three as references from time to time.

**User Interface Location**  (10-20

To resize the panes, drag the horizontal or vertical sashes, or drag the splitter where these two sashes meet. Double-click the sashes to reset them.

## Interactive Surface Design Extension II (ISDXII)

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Loft and Blend Surfaces

**Summary**

Loft and Blend surfaces in Style provide an additional and powerful tool set for creating complex forms.

**Benefits and Description**  (200words)

Loft surface allows a Style surface to be created with a number of curves (two or more), defining the shape in one direction and no curves in the secondary direction. The Blend surface allows the creation of a surface from one or two guide curves and any number of profiles.

**User Interface Location**

To create a loft curve, select the Surface tool, select the segments in one direction and complete the loft surface. To create a blend surface, follow the steps for creating a loft, and then select Cross curve from the dashboard, collect the desired cross curves, and complete the blend surface.

# Interactive Surface Design Extension II (ISDXII)

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Planar on Endpoints Command

**Summary**

With the **Planar on Endpoints** command, you can make 3D curves temporarily planar without constraining them to a particular reference.

**Benefits and Description** (200words)

The curve becomes planar between its endpoints normal to the active datum plane. This functionality is only valid for free curves. Free tangents at the endpoints are adjusted if necessary. If any point or tangent constraints exist, the command does not affect the curve.

**User Interface Location**

Select a curve, select the Edit tool, right-click the curve, and choose Planar on Endpoints from the shortcut menu.

## Interactive Surface Design Extension II (ISDXII)

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Offset COS Command

**Summary**

Curve On Surface (COS) curve offsets are now supported inside the Style functional area.

**Benefits and Description**  (200words)

COS curves can be offset either on the surface or on the normal to surface, making this a powerful function. Offset curves are history-based and update whenever the original COS is modified and can be redefined to alter their direction or dimension. You can set offset dimensions via the dashboard for use in other areas of Pro/ENGINEER.

**User Interface Location**

Select the curve to offset, select Styling / Offset COS and set the options in the dashboard.

# Interactive Surface Design Extension II (ISDXII)

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Radial-Path Planar Curves

**Summary**

An option to build radial-path, planar curves has been added to the Planar Curve tool.

**Benefits and Description**  (200words)

Within Style, you can construct a planar curve on a plane that remains normal to the tangent vector of a previously created curve at a specific location to build a radial path. The new plane and curve maintain their history, so that when you modify the path curve the geometry updates. You can also drag the plane along the path curve.

**User Interface Location**

Select the curve creation tool, toggle the curve type to planar, select the Refs tab from the dashboard, select reference and pick a point on the path curve. A plane will be constructed and its grid displayed. Click points on the plane to create the planar curve. When editing the curve, you can drag the plane along the path curve by holding down ALT, as you would to offset any planar curve reference plane dynamically.

# Behavioral Modeling Extension (BMX)

Model Analysis (BMX, ModelCHECK, Pro/PROGRAM)

**Title**

User defined input for multi-objective design studies

**Summary**

Users can now explicitly specify the locations at which experiments will be performed in multi-objective design studies.

**Benefits and Description**  (200words)

In past releases BMX determined the design space for multi-objective design studies (MODS) by uniformly distributing experiments between the users specified maximum and minimum values.

In Wildfire 2.0, in addition to the current method of defining the design space, users may also specify a table of values at which they want experiments to be performed. The values for the design variable can be manually entered, or they can be imported from a text file.  You then have the option of running experiments on all of the combinations of design variables, or one experiment per "row" of design variable values.

This new functionality makes MODS vastly more powerful than before, and opens the door to discrete optimization (for example optimizing a sheet metal assembly varying between fixed gauges of material) as well as using more advanced distribution functions, such as a Gaussian distribution.

**User Interface Location**
Analysis->Multi-Objective Design Study

# Cabling Design Extension

Electrical Design (Diagram, Cabling, Harness,ECAD)

**Title**

XML Data and Routed Systems Designer

**Summary**

With a new menu item, Pro/CABLING provides for import of XML data from Routed Systems Designer.

**Benefits and Description**  (200words)

The selection of logical data created in Routed Systems Designer has been made simpler. On the new menu, you can set the import filter automatically to *.XML to filter out other file types and reduce the number of mouse clicks.

**User Interface Location**

Logical Ref / Import / RS Designer

# Cabling Design Extension

Electrical Design (Diagram, Cabling, Harness,ECAD)

**Title**

Flattening of Closed Loops

**Summary**

Automation for the flattening of closed loops has been simplified.

**Benefits and Description**  (200words)

During the flattening process, any closed loop is highlighted and you are prompted for a preferred path. After the main harness has been flattened, you can choose to go back and close the loops. Select two locations, and the loops are closed.

**User Interface Location**

Flatten / Lay Out / Close Loop

# Cabling Design Extension

Electrical Design (Diagram, Cabling, Harness,ECAD)

**Title**

Significant Locations Only and Flattened Harnesses

**Summary**

Stability and performance has improved for the flattening of harnesses with the consideration of only significant locations.

**Benefits and Description**  (200words)

Considering only significant locations during the flattening of 3D harnesses reduces risk of failures, decreases rework after modifications, and improves stability and performance. Pro/HARNESS for manufacturing considers the following locations as significant:

- Connector locations–Locations at the entry port of connectors
- Transitional locations–Locations where the wire(s) and/or bundles are not the same on both sides
- Branch locations–Locations where wires or bundles (one or more) do not share the same adjacent location

**User Interface Location**

N/A

# Piping Design Extension

Piping (Spec Driven, Non-Spec Driven)

**Title**

Routing of Flexible Pipes

**Summary**

You can route flexible pipes using 2D logical information in Spec-Driven Piping.

**Benefits and Description**  (200words)

This routing operation uses schematic information generated in Routed Systems Designer for the creation of the 3D flexible pipes within spec-driven piping. Use of this information reduces errors often caused by the manual interpretation of 2D schematic data.

To use the schematic data from Routed Systems Designer, you must set the `config.pro` option `piping_schematic_driven` to `yes`.

**User Interface Location**
Pipeline / Create

## Piping Design Extension

Piping (Spec Driven, Non-Spec Driven)

**Title**

Improved Pipe Weight Calculation

**Summary**

You can specify the density units of a pipeline independent of an assembly's density units.

**Benefits and Description**  (200words)

Accurate mass property calculations for pipelines consisting of multiple materials are now possible.

**User Interface Location**

N/A

## Piping Design Extension

Piping (Spec Driven, Non-Spec Driven)

**Title**

Enhanced Fitting Libraries

**Summary**

Over 12,000 library fittings, both JIS and ANSI, have been updated.

**Benefits and Description**  (200words)

The updating of the JIS and ANSI fitting libraries takes advantage of the improvements made to Pipe and Fitting weight calculations. If you use these fitting libraries, you are saved any rework.

**User Interface Location**

N/A

# Piping Design Extension

Piping (Spec Driven, Non-Spec Driven)

**Title**

Improved Weight Calculation for Fittings

**Summary**

You can obtain accurate mass property information without having to accurately model fittings.

**Benefits and Description**  (200words)

To reduce library setup time and improve performance, user-defined parameters allow mass property information from vendor's data sheets, which are added and stored with a fitting.

Set `pro_mp_source` to `parameters` and add any of the user parameters to a fitting. This process provides enough information so that the Model Analysis functionality can calculate the user value for mass property calculations.

**User Interface Location**

N/A

# Design Collaboration Extension (DCX)

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Design Conferencing via a Data Management Tool

**Summary**

During a real-time design session with Pro/ENGINEER, peer-to-peer Design Conferencing provides a data management system as the data source. This new option extends the capability of design conferencing and provides the security and control of the data management system.

**Benefits and Description**  (200words)

As a Design Conferencing participant, you can work with information stored in data management systems as well in the shared space. You can retrieve, save, and upload models, drawings, and so forth directly to and from the Commonspace or Cabinets during a conference. The conference host, called the PDM Master, grants access and privileges that can be updated at any time for security.

As in traditional conferencing, an observer can get control from the PDM Master and become the current presenter. When others are presenters beside the host (PDM Master), all PDM transactions are performed on the PDM Master's computer. All participants can observe the PDM transaction during the shared session, and the data is transferred to the participants' shared space via the PTC Conference Center.

**User Interface Location**

N/A

# Design Collaboration Extension (DCX)

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Design Conferencing User Account Login

**Summary**

The PTC Conference Center has an improved method of logging in through a list of accounts for a specified Conference server.

**Benefits and Description**  (200words)

To use Design Conferencing, you must have a Design Conferencing account on the specified Conference server to log in to the PTC Conference Center on the local machine.

When you start the PTC Conference Center for the first time on the specified Conference server, you must create a Design Conferencing account. If you already have an account on the local Conference server, you can log in to the Conference Center using your account name and password. To improve the login process, a **User Name** list provides all the accounts for the specified Conference server. The first account name on the list is the last person who had logged into the PTC Conference Center.

**User Interface Location**

Standalone Conference Center or Applications / Conference

# Design Collaboration Extension (DCX)

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Design Conferencing Participant License Package

**Summary**

Globally dispersed product development teams must collaborate from many locations. With the Pro/ENGINEER Design Conferencing Participant license package, even those without a license can participate in live design sessions.

**Benefits and Description**  (200words)

With more companies outsourcing jobs to minimize costs, time to market, and so forth, more people off site get involved in the design process. Powered by Groove Networks, Pro/ENGINEER Design Conferencing Participant is fast, flexible, and secure.

With this license package, even those who do not own or have access to a license of Pro/ENGINEER can still participate in live design sessions. Once connected, a Design Conferencing Participant can perform all critical functions, like sharing control of live design sessions with their team, no matter where team members are located.

**User Interface Location**

N/A

# Design Collaboration Extension (DCX)

Fundamentals (Fundamentals, Design Conferencing)

**Title**

Pop-up Menu Enhancements during Design Conferencing

**Summary**

During a real-time design session with Pro/ENGINEER, pop-up menus allow observers to follow more closely with the steps and selections of the presenter.

**Benefits and Description**  (200words)

Peer-to-peer Design Conferencing provides the ability to conduct real-time, peer-to-peer, multiuser design conferences with other Pro/ENGINEER users via the Web. Any number of participants can join the conference. The leader starts the shared session and can pass control over Pro/ENGINEER to any participant, who becomes the presenter. When the presenter performs an action in Pro/ENGINEER, everyone's session follows along accordingly.

Conferencing can be used for many types of situations, like training and holding design reviews. With these new enhancements, observers are able to follow every move of the presenter, from menus selected or commands executed from the Model Tree, to commands executed via the right-mouse button. This brings observers one-step closer to feeling as if they were in the same room as the presenter, working on the same machine.

**User Interface Location**

N/A

# Reverse Engineering Extension (REX)

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Automatic Surfacing

**Summary**

Automatic surfacing in the Restyle functional area provides a simple, progressive means for creating surfaces. This functionality is ideal when working with extremely complex, organic shapes.

**Benefits and Description**  (200words)

The Auto-Surfacing tool automatically generates the curve-patch structure as well as the surfaces directly on a facet model. The Auto-Surfacing dashboard includes three phases:

- Include or exclude geometry for surfacing.
- Select or create the curve patch structure.
- Generate surfaces based on the selected curve network.

This dashboard-driven functionality provides an extremely quick method to create a surface representation of complex facet models and requires little experience.

**User Interface Location**

Select the Auto Surface icon to open a dashboard.

# Reverse Engineering Extension (REX)

Surfacing (ISDX, Restyle, Warp, Facet Modeling)

**Title**

Surface Connections for Curvature (G2)

**Summary**

The ability to create surface connections for curvature (G2) within the Restyle module greatly enhances workflow and flexibility while designing with surfaces.

**Benefits and Description**  (200words)

You can create a surface connection for surfaces that are tangent continuous across the boundary and share curvature along the common boundary. The user interface for this new constraint type streamlines workflow. Connection constraints in the Pro/ENGINEER window are designated by a dashed line (Positional), a single line (Tangent), and a double line (Curvature continuous). To access and modify connection information, right-click the selected surface, and choose the display constraint from the shortcut menu. Select the yellow indicator, right-click, and choose the desired constraint type from the shortcut menu.

**User Interface Location**

Select the desired Surface, right-click and choose display constraints, select the yellow indicator, and choose the desired constraint type from the shortcut menu.

# Tool Design Option (TDO)

Mold Design and Casting

**Title**

Inherited Model as a Reference Part

**Summary**

As a Mold Designer, you can select an inherited model as a reference part for the mold assembly. The inherited model provides additional flexibility and control over the reference model.

**Benefits and Description**  (200words)

Using an inherited model as a reference model for a mold assembly, you can apply shrinkage on this reference model without affecting the initial model. In addition, you have access to all the flexibility of the inherited model in terms of the control of features and dimensions and the propagation of changes from the initial part.

**User Interface Location**

Mold Model / Assemble / Ref Model / …select and assemble the reference model … select Inherited

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Coordinate System Display in Result Windows

**Summary**

Coordinate systems used in the creation of your model can now be displayed in result windows.

**Benefits and Description**  (200words)

The coordinate system display icons (Cartesian, cylindrical, and spherical) used in the definition of your model can now be seen in results. The coordinate system icon is displayed in the results window for fringe, vector, and model type. The icons can be toggled on or off using the **Format** > **Result Window** dialog box.

The display of coordinate systems in result windows makes it easier for you to document and communicate your analysis model with others.

**User Interface Location**

Format / Result Window

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Combined point, edge, and surface force/moment load definition

**Summary**

Point, edge, and surface force/moment load definition commands have been merged into a common dialog.

**Benefits and Description**  (200words)

The three different force/moment load commands for point, edge, and surface types have been merged into a single dialog.  This simplifies the user interface by reducing the number of toolbar commands and menus.  It also makes it possible to edit a load from one reference type to another; this was not possible in previous releases.

**User Interface Location**

Insert -> Force/Moment Load

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Combined point, edge, and surface displacement constraint definition

**Summary**

Point, edge, and surface displacement constraint definition commands have been merged into a common dialog.

**Benefits and Description**  (200words)

The three different displacement constraint commands for point, edge, and surface types have been merged into a single dialog.  This simplifies the user interface by reducing the number of toolbar commands and menus.  It also makes it possible to edit a constraint from one reference type to another; this was not possible in previous releases.

**User Interface Location**

Insert -> Displacement Constraint

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Bolt fasteners in Structural and Thermal

**Summary**

Bolt fasteners have been added to Structural and Thermal simulation.

**Benefits and Description**  (200words)

Modeling bolt used to be a long and tedious process that required a lot of experience and patience to master.  In Wildfire 2.0, a new modeling entity has been introduced which makes this much easier, the bolt fastener.

It is now possible to define bole fasteners in assemblies simply by selecting the holes that will be held together by the fastener, and specifying a material and bolt diameter.  Advanced fasteners may have their stiffness specified by spring properties, as well as a bolt pre-load value.  Three types of bolt fasteners are supported.  These include:

- Through bolts in solid models
- Through bolts in shell models
- Blind (screw) fasteners in solid models

**User Interface Location**

Insert -> Connection -> Fastener

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

AutoGEM meshing over small geometry

**Summary**

You may now specify a minimum edge length below which AutoGEM will ignore geometry features, simplifying your model.

**Benefits and Description** (200words)

When performing analysis of large, complex models it is often desirable to make the model mesh with fewer elements. In the past, you accomplished this by de-featuring the geometry of the model. This was time consuming and difficult.

In Wildfire 2.0, it is now possible to define a minimum edge length AutoGEM control. In defining this mesh control you to specify a minimum length by either a dimensional value or percentage of the model. When the AutoGEM mesh is being created, any geometry edges that fall below that specified in the mesh control will be ignored. This gives you the ability to significantly reduce the number of elements required to mesh complex models such as castings and forgings.

Using this new mesh control you will find it easier and quicker to simplify your models, which is useful when looking for global responses such as displacements or modal frequencies.

**User Interface Location**
AutoGEM -> Control

asdfasdfasfd

asdfasdf

# What's New for Pro/ENGINEER Wildfire 2.0
Formerly Interactive Release Notes

## Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

AutoGEM nodes on edge mesh control

**Summary**

You may now specify the minimum number of nodes created on an edge in AutoGEM using the new AutoGEM mesh controls.

**Benefits and Description**  (200words)

Experienced users occasionally want to exert control over the mesh that is created in AutoGEM.  In past releases the only option was to modify the AutoGEM settings (which impact the model globally) or modify the geometry by adding points or curves.  In Wildfire 2.0, you may now simply create a Nodes on Edge mesh control where you specify the number of nodes AutoGEM will create on a given curve/edge.  This makes it much easier to create a refined mesh you are happy with.

**User Interface Location**

AutoGEM -> Controls

## Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

AutoGEM performance and robustness improvements

**Summary**

AutoGEM meshing performance has been improved by up to 25%.  Thin model meshing robustness has also been improved.

**Benefits and Description**  (200words)

AutoGEM performance has been improved by up to 25% for solid meshing. Additionally, the robustness of solid meshing large thin models has been greatly improved.

**User Interface Location**

AutoGEM -> Create

## Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Display of AutoGEM Mesh in Integrated Mode

**Summary**

More options have been added to control the display of the AutoGEM mesh in integrated mode. You can improve the display performance of large models.

**Benefits and Description**  (200words)

The performance and display options for viewing the AutoGEM mesh in integrated mode have been improved. You now have control over the display quality (fine, medium, and coarse). Additionally, a shell element can be displayed as in Pro/ENGINEER with its thickness or with a zero thickness (as in the standalone).

**User Interface Location**

View / Simulation Display

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Object and Model Information in HTML

**Summary**

Information summaries for Simulation modeling entities now display in HTML using the Pro/ENGINEER browser.

**Benefits and Description**  (200words)

The information summaries in the Pro/ENGINEER browser are easier to read and contain more information, such as hyperlinks to the referenced entities, other simulation modeling objects, coordinate systems, and geometry. The HTML browser makes it easier to interrogate your model and to share it with others.

**User Interface Location**

None

## Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Free Connection Type More Robust

**Summary**

The robustness of the **Free** connection type for interfaces in Structural and Thermal Simulation is improved.

**Benefits and Description**  (200words)

Using **Free** connection, you can model press-fit models without using standalone mode.

**User Interface Location**

Insert / Connection / Interface

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

FEM Meshing Robustness and Performance

**Summary**

FEM meshing performance has been improved for assembly, multi-volume part, and mixed mesh models. The meshes are more robust.

**Benefits and Description**  (200words)

Solid meshing in FEM mode for the following model types has been greatly improved:

- Multi-volume parts (volume regions, and so forth)
- Assemblies
- Mixed mesh models (solids and shells)

Besides performance improvements, FEM meshing provides more robust meshes. This increased quality of the meshes reduces the time to create a mesh to your satisfaction.

**User Interface Location**

Mesh / Create

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

FEM Meshing Quality

**Summary**

The FEM mesher now takes curvature and geometry proximity into account when creating the solid mesh.

**Benefits and Description**  (200words)

Enhancements to the FEM mesher make it easier and quicker to create high-quality meshes for use with third-party solvers. When you create a solid mesh, surface curvature and geometric proximity are taken into account. Curvature refinement creates more, better-shaped elements, in areas where your model geometry is curved such as in rounds and around holes. Proximity refinement creates more elements with improved aspect ratios in areas of the model that are thin, such as where a hole is near the surface of a model.

**User Interface Location**

Mesh / Create

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Shape Optimization Improvements in FEM Mesh

**Summary**

For FEM mesh elements, the optimization step in the meshing process has been improved to increase element quality.

**Benefits and Description**  (200words)

After you create a FEM mesh, you can perform steps to optimize the element shape before you complete the mesh. The algorithm for the optimization results in better quality for elements in your solid meshes.

**User Interface Location**

Mesh / Improve

## Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Create FEM Mesh Dialog Box

**Summary**

The **Create FEM Mesh** dialog box provides all commands for the element types to generate the mesh. This concise dialog box merges all the Menu Manager commands for FEM mesh creation.

**Benefits and Description**  (200words)

In addition to being easier to use, the **Create FEM Mesh** dialog box retains the last settings that were used to create a mesh (solid, mixed, mid-surface). Consequently, you can more easily re-create your mesh at a future date.

**User Interface Location**

Mesh / Create

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Support of family tables in Structural and Thermal

**Summary**

You can now define family table instances of your simulation models. Additionally, simulation-modeling objects may be manually suppressed and resumed.

**Benefits and Description**  (200words)

Simulation modeling objects may now be suppressed by family table instance. In standard mode of Pro/ENGINEER, if you define an instance, upon entry to Structural and Thermal you will be able to add simulation-modeling objects (loads, constraints, mesh controls, idealizations, etc…) to the family table.

Additionally, you are also now able to manually suppress and resume simulation-modeling objects either through the edit menu, or directly from within the model tree using the right mouse button pop-up menu.

These capabilities make it easier than ever for you to build complex simulation models that meet your requirements.

**User Interface Location**

Tools -> Family Table

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Enhanced Selection

**Summary**

Selection in simulation now supports preselected references and provides an improved workflow.

**Benefits and Description**  (200words)

It is easier and quicker to build your simulation models. A selection filter has been added in the lower-right corner of the Pro/ENGINEER makes it easier to interact with complex models. Additionally, you can use preselected references when defining modeling objects. For example, if you select a surface, and then open the **Pressure Load** dialog box, the selected surface becomes the reference. That surface is "active" in related dialog boxes, so you no longer must pick the references.   Another selection enhancement implemented is that selection is "active" as soon as a dialog is invoked; you no longer need to select the pick arrow to select the references.

**User Interface Location**

N/A

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Enhanced HTML Result Reporting

**Summary**

HTML result reports have been enhanced so you can document and communicate your analysis results quickly and easily.

**Benefits and Description**  (200words)

You can now include modeling information in your result report, including data such as loads, constraints, properties, and any idealizations used. Other additions include a user-configurable title block and an improved workflow for creating your report. The simplified format of the output report provides for easier editing with standard word processing software.

With the HTML report, you can document your analysis results more completely and in less time.

**User Interface Location**

In results File / Export / HTML Report

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Display of the Nodal Solution Coordinate System in FEM Mode

**Summary**

You can now display the nodal solution (displacement) coordinate system in FEM mode.

**Benefits and Description**  (200words)

When creating models in FEM mode, many modeling entities can impact the nodal solution (displacement) coordinate system when the mesh is output to a solver. With this new review tool, you can display the nodal coordinate system without leaving Structural and Thermal Simulation. The display of the nodal solution coordinate system in FEM mode makes it easier to better understand your FEM model. To use the new tool select to review nodes once you have output your model to the screen.

**User Interface Location**

Mesh / Review

## Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Robustness and performance improvements in simulation geometry creation

**Summary**

Simulation geometry creation robustness and performance has been dramatically enhanced.

**Benefits and Description**  (200words)

When you select to interactively AutoGEM a model or start an analysis in integrated mode, the geometry of the Pro/ENGINEER model is processed to be suitable for Mechanica.   This project improves the performance of this step 10 to 15 times over that of previous releases.    This can dramatically reduce the amount of time it takes to mesh and solve model in integrated mode.

**User Interface Location**

None

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Mirror Symmetry Constraints

**Summary**

Mirror symmetry constraints are implemented in Structural and Thermal Simulation.

**Benefits and Description**  (200words)

You can define mirror symmetry constraints in structural analysis models by selecting the geometry that lies on the mirror symmetry plane (point, edges, and surfaces). This method is fast and easy. These new constraints are displayed on the model and in the Model Tree.

**User Interface Location**

Insert / Symmetry Constraint

## Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Mesh Control Visibility

**Summary**

Mesh controls are now visible whenever you are in simulation mode.

**Benefits and Description** (200words)

The display of mesh controls can be toggled on or off in the **Simulation Display** dialog box.

**User Interface Location**

View / Simulation Display

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Display of Loads and Constraints in Results

**Summary**

In results, you can display load and constraint icons. These icons are superimposed on the results window as they are on the preprocessor.

**Benefits and Description**  (200words)

The display of the load and constraint icons makes it easier to document and to communicate how you set up the model. You can create results windows for fringe, vector, and model display with the load and constraint icons on the model. You can toggle the display of the icons on or off in the **Format Result Window** dialog box.

**User Interface Location**

Select Format / Result Window

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Mechanism Design Loads Transfer

**Summary**

You can automatically transfer loads from Mechanism Design and apply them to your structural analysis model.

**Benefits and Description**  (200words)

With load transfer, you are ensured that the load values and referenced coordinate transformations are accurate. This accuracy of information simplifies the task of performing structural analyses on models simulated in Mechanism Design.

After running a motion dynamic analysis, you must pick a reference Mechanism Body and save the results. Next, you pick a component (part, assembly, or subassembly) to which the results will be saved. Reaction forces and inertial acceleration data are then written to the specified component part or assembly file. This data is then available when you open this component (or an assembly containing it) in Structural and Thermal Simulation.

To use saved loads, you must import the stored Mechanism loads once in Mechanica. After import, the load information saved in the component is used to automatically create an equivalent and unassociated total load. This load is applied at a point and placed into a new Mechanica load set. You can then select the geometry on which the loads will be acting.

**User Interface Location**

Insert / Mechanism Load

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Workflow for Model Types

**Summary**

The workflow for entering Structural and Thermal Simulation and specifying the model type has been streamlined.

**Benefits and Description** (200words)

When you enter Structural and Thermal Simulation for the first time with a new model, the **Model Type** dialog box opens so you can specify the type of model to work on. On subsequent entries, these settings remain so you begin in the same mode as your previous model type. To edit the model type, click **Edit** > **Mechanica Model Type**.

This improved workflow for model types reduces the amount of time it takes to switch between standard and Structural and Thermal Simulation. Thus, it is easier to make iterations to your design.

**User Interface Location**

Edit / Mechanica Model Type

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Redesigned measures

**Summary**

Measures functionality in Structural and Thermal Simulation has been greatly enhanced in Wildfire 2.0 to improve usability and functionality.

**Benefits and Description**

Measures have been redesigned in the Wildfire 2.0 release in order to enhance both the usability as well as the functionality of measures.

When you select to create a new measure, you will see that the user interface for defining them is more streamlined than in previous releases. You will also notice that as measures are created, they appear both on the model, as well as in the model tree. Some of the other enhancements to measures include the following:

- Measures from a lower level model can be made visible in higher-level assemblies
- The ability to select geometry for evaluation location (surfaces, curves, components, etc…)
- The ability to define computed measures in Structural and Thermal
- Multi-edit capability to quickly place multiple instances of point measures at different locations in a model.

This new functionality will make is quicker and easier for you to build and interpret the results of your analysis models.

**User Interface Location**

Insert -> Simulation Measure

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Results Templates

**Summary**

Results templates can be defined and used to dramatically reduce the amount of time it takes to generate results in Structural and Thermal Simulation.

**Benefits and Description**  (200words)

With results templates, you can quickly format and display results from different analyses, automating this repetitive task. Saving a set of results windows that you prefer to a template is easy (**File** > **Save as Template**). You can choose whether to save the legend values, model orientation, annotations (if present), and the deformed scale to the template. By default, the template is saved to the working directory. You can, however, specify a template directory using the `config.pro` file.

When you define new results windows from a template, select the template that you want to use and the study directory to which to apply it. If the study directory contains multiple load sets or modes, you can either repeat the template for each load set or mode, or you can combine them.

**User Interface Location**

Select Insert / Results Window from Template

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

New Search Tool in Simulation

**Summary**

The Pro/ENGINEER Search tool for advanced selection now works in simulation mode.

**Benefits and Description**  (200words)

With the new Search tool, you can search by the following attributes:

- Name
- Type (loads, beams, and so forth)
- Property (materials, shell property, beam sections, and so forth)

In addition to advanced selection capabilities, the Search tool can also be used to build queries and define layers by rule (for example, a layer that contains all shells with the material property Steel assigned to them).

**User Interface Location**

Edit / Find

## Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Coordinate System Control for Shell Elements

**Summary**

The **Shell Element** coordinate system mesh control option provides control of the coordinate system direction area for shell element.

**Benefits and Description** (200words)

You can control the shell element coordinate system. With this mesh control, you can align the shell element coordinate system determined by the numbering of the nodes to a selected coordinate system. This alignment provides more precise control of the solver input deck that is generated in FEM mode.

**User Interface Location**

Mesh / Control

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Master Control for Display of Simulation Entities

**Summary**

You can now toggle the display of all simulation modeling entities with a single control.

**Benefits and Description**  (200words)

The display of large simulation models can become cluttered. With the master control (the **Show Simulation Entities** check box) on the **Simulation Display** dialog box, you can temporarily override the other settings and toggle the display of the simulation modeling objects.

**User Interface Location**

View / Simulation Display

## Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Saving sketched beam sections to a library

**Summary**

Sketched beam sections may now be saved to a library.

**Benefits and Description**  (200words)

It is now possible for you to save sketched beam sections to the beam section library. Additionally, a config.pro option has been added where the location of the beam section libraries directory may be specified (sim_beamsection_path).

**User Interface Location**

Properties -> Beam Sections

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Weld definition dialog

**Summary**

A new dialog has been implemented for perimeter and end weld definition.

**Benefits and Description**  (200words)

End and perimeter weld definition has been simplified through the implementation of a new definition dialog.

**User Interface Location**

Insert -> Connection -> Weld

## Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Two point rigid connections

**Summary**

Rigid connections may now be defined between 2 points.

**Benefits and Description**  (200words)

Rigid connections in Structural and Thermal Simulation may now be defined between 2 points.  In past releases, the minimum number of points required to define a rigid connection was three non-collinear points.

**User Interface Location**

Insert -> Connection -> Rigid Connection

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Enhanced Total Load Review

**Summary**

Total load review has improved functionality and usability.

**Benefits and Description**  (200words)

Total load review is now in the **Load Resultant** dialog box rather than in the Menu Manager. In addition to improved usability, you can:

- Select non-WCS coordinate systems
- Select the point about which the load resultant is computed
- Access total load review in FEM mode

**User Interface Location**

Info / Review Total Load

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Migration from the Menu Manager to Top-Level Menus

**Summary**

Menus in Structural and Thermal Simulation have been migrated from the Menu Manager to the top-level menus.

**Benefits and Description** (200words)

Commands in simulation mode have migrated from the Menu Manager to the top-level menus in Structural and Thermal Simulation. This simplifies the user interface by making it much easier and faster to locate commands. This change makes it possible to expand the graphics window.

**User Interface Location**

N/A

# Structural and Thermal Simulation

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Tolerance report

**Summary**

The tolerance report simplifies the process of understanding why assembly models may fail to mesh.

**Benefits and Description**  (200words)

Large variations in model accuracy can cause problems during meshing.  The tolerance report provides a quick way in which you can see the model accuracy of all of the components in an assembly, spotting which model may be causing problems by having too large an accuracy setting.

**User Interface Location**

Info -> Tolerance Report

# Application Programming Toolkit

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**

Expanded and Improved Programmatic Access

**Summary**

Expanded programmatic access is provided to features, sections, and sketches in Pro/ENGINEER. In addition, improvements to section and sketcher related functionality in Pro/TOOLKIT are significant.

**Benefits and Description**

The feature element trees of the following Pro/ENGINEER features have been published:

- Round
- Draft
- Shell
- Move
- Mirror
- Trim
- Thicken
- Merge
- Solidify (formerly Patch)
- Sheetmetal flat wall
- Sheetmetal flange wall

Included in Pro/TOOLKIT are utility type functions for increased coverage to the section interface. For example, you can calculate the number and type of loops for a section. This output can help validate the section before trying to use it to create a Pro/ENGINEER feature. Expanded coverage and utilities for programmatic feature creation are part of the commitment of PTC to data interoperability and openness.

**User Interface Location**

N/A

# Application Programming Toolkit

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**

Pro/Web.Link (JavaScript API) on UNIX

**Summary**

Pro/Web.Link (Pro/ENGINEER browser version), the JavaScript interface to Pro/ENGINEER, is now supported on UNIX platforms. The API is the same as the Microsoft Internet Explorer-based Pro/Web.Link introduced in Pro/ENGINEER Wildfire.

**Benefits and Description**  (200words)

The new architecture of Pro/Web.Link runs in the Pro/ENGINEER browser (Microsoft Internet Explorer on Windows and Mozilla using XPCOM on UNIX).

Pro/Web.Link links the World Wide Web to Pro/ENGINEER, so you can use the browser and the Internet as tools to automate and streamline parts of your engineering process. You can load Pro/Web.Link pages directly into the Pro/ENGINEER browser. The new version of Pro/Web.Link offers a number of benefits:

- Easy setup (no separate plug-ins required, installed by default with Pro/ENGINEER)
- Extensive Pro/ENGINEER functionality covered
- Ability to call into Pro/TOOLKIT or J-Link task libraries
- Single interface (Pro/ENGINEER browser, so no additional windows are required) to interact with Pro/ENGINEER.

**User Interface Location**
N/A

PTC

# Application Programming Toolkit

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**User Interface Location**
N/A

# Application Programming Toolkit

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**: Feature Element Trees in XML

**Summary**

The XML feature engine in Pro/TOOLKIT helps construct Pro/ENGINEER features based on XML definitions for the feature element tree.

**Benefits and Description**

Pro/TOOLKIT now includes functions to write the contents of a feature element tree to XML format as well as to read an XML file to construct a feature element tree. Schema files as templates for some feature types help validate the construction of the XML files used in creating the feature element tree.   Pro/TOOLKIT now provides direct conversion between feature element trees and standard XML format files. This mechanism supports the following types of element value:

- Integer / enumerated type
- String
- Wide string
- Boolean
- Transform
- Reference
- Reference array ("multivalued" element)
- Model (component model pointer)
- Curve collection
- Surface collection

Easier representation of the element trees in the form of XML documents simplifies the feature creation procedure. By using XML as a neutral file format for exchanging feature element trees, you can effectively separate data from the Pro/TOOLKIT application for the creation or modification of various features. You modify the XML file and not the application.

**User Interface Location: N/A**

# Application Programming Toolkit

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**

User Interface Improvements in Pro/TOOLKIT

**Summary**

Pro/TOOLKIT contains a number of user interface enhancements for seamless integration of interfaces developed with Pro/TOOLKIT.

**Benefits and Description**  (200words)

Significant improvements in usability continue to support introduction of object-action and action-object tools. Programmatic access has been provided to many of these individual areas of improvements, such as the ability to:

- Access the contents of selection buffers and tools.
- Add context-sensitive buttons to the shortcut menus (right-click) in the graphics window and in the Model Tree.
- Assign pop-up menus to custom dialog boxes.
- Display the **Parameter** dialog box and allow a flexible context for selection.

With these enhancements, you can develop custom user interfaces with a similar look and feel to Pro/ENGINEER yet integrated with other applications for a streamlined user experience..

**User Interface Location**
N/A

# Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

NC Sequence for 2-Axis Trajectory Milling

**Summary** .

New 2-axis trajectory milling supports milling or drilling tools, and automatic material removal in expert machining.

**Benefits and Description**

The NC sequence for 2-axis trajectory milling answers requests for specific machining techniques. It allows the selection of a drilling tool to drive along a curve to create, for example, a "break edge" NC Sequence. This is a very common practice in high-production machining where the same tool that was used to drill a hole is used to cut a small chamfer to reduce the tool change time.

This new NC sequence, based on the freehand tool path, supports automatic material removal. It also supports Arc Fitting, allowing a spline to be output as circular interpolation (using the parameter `approximate_splines` set to `yes`).

**User Interface Location**

MACHINING / NC Sequence / NC Sequence / Trajectory / 2 Axis in Pro/NC

# Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Tool Adaptors with Fixed Orientation

**Summary**

Support for tool adaptors (for example, right-angle head) in Pro/NC provides a mechanism to assign a fixed, nonvertical position for a cutting tool.

**Benefits and Description**

The adaptor for milling NC sequences can support right-angle or any fixed orientation head. Both tool and adaptor are simulated in Shaded mode, so you can check visually for collision. A SET/HOLDER command with CSYS offset information is generated in the CL file, allowing the post-processor to compensate for the adaptor orientation.

Adaptors are defined using a Pro/ENGINEER solid model with two coordinate systems:

- TOOL_ATTACH_POINT—Defines the tool axis orientation and tool location.
- SPINDLE_CONTROL_POINT—Defines the spindle axis and location.

**User Interface Location**

Machining / NC Sequences / select any milling toolpath / Seq Setup / Adaptor

# Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Finishing Tool Path for Surface Machining

**Summary**

The new finishing tool path provides optimum surfacing for steep, shallow, and flat surface machining, including crosscut surfaces. Surface slope rather than boundaries determines the machining strategies.

**Benefits and Description**  (200words)

With the new Finishing tool path, you can automate a procedure for creating an optimized, surface machining tool path based on the threshold surface angles. Steep surfaces are machined using high-speed machining, Z-profile cuts using an "S connection." Shallow and flat surfaces are machined either with straight (conventional) cuts or the new, high-speed spiral cuts. You have complete control on which area needs to be machined and in which order.

This new tool path requires minimum input to cut cost by reducing manufacturing lead-time. The optimized tool path produces a quality surface finish.

**User Interface Location**
MACHINING / NC Sequence / Finishing

## Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Step-Depth Control for Engraving

**Summary**

The engraving tool path has been improved to offer step-depth control and multiple groove selection.

**Benefits and Description**  (200words)

As an NC Programmer, you can now machine cosmetic grooves with multiple passes using the STEP_DEPTH parameter in the engraving tool path. In addition, using arc output can minimize the toolpath length and improve quality.

By using multiple groove selection, you can avoid the definition of additional NC sequences.

**User Interface Location**

MACHINING / NC Sequence / Engraving / Seq Setup / Parameters

# Design Collaboration Extension (DCX)

Fundamentals (Fundamentals, Design Conferencing)

**Title**

E-mail of Invitations from the PTC Conference Center

**Summary**

From the PTC Conference Center through electronic mail, you can directly invite participants outside or inside the local domain.

**Benefits and Description**  (200words)

The invitation process has been streamlined, speeding the process and making it easy to use. Directly from within the Conference Center, you can follow the standard invitation process and include any e-mail addresses. The conference invitation is attached to the e-mail. All the recipient need do is accept the invitation and dynamically join the conference.

**User Interface Location**

From within a conference in the PTC Conference Center, use the Extend invitation to join conference icon and the Invite a participant to a conference by email icon.

# Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Support for Multi-tip Tools

**Summary**

Pro/NC supports multiple control points for the existing tools and a new type of "multi-tip " or "stage" tool with multiple control points and diameters.

**Benefits and Description**  (200words)

For series or production machining, special tools are often required. With one tool, different diameters or faces can be manufactured. To simplify the definition and the programming of these tool types in Pro/NC, the definition and the control of multiple control points has been added to tool definition. A new type of "multi-tip " or "stage" tool, with multiple control points and diameters, is available for trajectory and holemaking sequences.

As an NC programmer, you can now select during a tool path which tool tip is driven. Combined with a custom cycle, multi-tip tools provide programmers for series machining with a powerful tool to cover special production needs for holemaking.

**User Interface Location**

Tool Manager

## Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Roughing, Reroughing, and Finishing of Imported STL Data

**Summary**

As Pro/NC user, you can machine directly imported STL data. Three tool paths are available for roughing, reroughing, and finishing.

**Benefits and Description**  (200words)

Stereolithography (STL) refers to the representation of 3D forms as a boundary representation of solid models that are constructed entirely of triangular facets. It is a format used mainly during the concept phase or prototyping.

You can directly machine an STL model or a combination of STL, surfaces, and solids, using three toolpath types: **Roughing**, **Re-roughing**, and **Finishing**.

Because the mathematical representation of the faceted model is quite simple, toolpath computation is optimized. Direct machining of STL data provides a cheap solution (no need for an STL machine) for rapid prototyping during the concept phase.

**User Interface Location**
MACHINING / NC Sequence / Roughing, Re-roughing, or Finishing

# Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Job Manager

**Summary**

Job Manager for batch toolpath computation for NC sequences and operations can be started from the Model Tree.

**Benefits and Description**  (200words)

Directly from the Model Tree, you can start the toolpath computation using Job Manager on a complete operation, on a single NC sequence, or on a multiple selection of NC sequences. Right-click an item in the Model Tree and select **Job Manager** from the shortcut menu.

The shortcut menu improves productivity by giving an easy way to start the Job Manager to compute the selection from the Model Tree. The Job Manager automatically calculates any selection, such as one or more operations, or continuous or noncontiguous NC sequences. It creates all necessary settings on the fly.

**User Interface Location**

In the Model Tree, right-click an NC sequence or operation. Select Job Manager from the shortcut menu.

# Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Merging Drilling Steps

**Summary**

Using the Process Manager, you can merge consecutive drilling steps using the same tool into a single, optimized NC step.

**Benefits and Description** (200words)

In addition to methods to create optimized tool paths for drilling NC sequences, the Process Manager offers the ability to merge independent but consecutive drilling steps using the same drilling tool. You can prepare the NC process while thinking feature by feature. Later in the process, you can optimize the tool path between holes; which gives better flexibility in the process definition.

**User Interface Location**

In the Process Table, select consecutive drilling steps using the same tool. Right-click and select Merge from the shortcut menu, or use Edit / Merge in the Process Table.

# Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Templates in Process Manager

**Summary**

NC templates can be used to automate the NC process creation.

**Benefits and Description**  (200words)

From the Process Manager's Process Table, you can store any selection of one or more steps as a template. A template is an XML file that includes the definition of the selected steps, which can be reused to create NC steps in the Process Table. You can easily store and reuse best practices without creating manufacturing UDFs.

**User Interface Location**

Select the steps to include in the template, and then click the icon in the Process Manager's Process Table.

## Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Process Manager for Pro/NC and the Process Table

**Summary**

A Process Manager extends Pro/NC capabilities in terms of viewing and editing an NC process.

**Benefits and Description**  (200words)

The new Process Manager in Pro/NC allows viewing of any manufacturing files created with Pro/NC in a table environment: the Process Table. Using this table, you can define custom views such as the coordinate system, retract planes and references, any parameter, and the cutting time.

Directly from the Process Table, you can edit the steps guided by the dashboard. By right-clicking a selection, you can play a single step or multiple steps, and play selected contiguous or noncontiguous steps.

**User Interface Location**

Process Manager button in the Menu Manager in a Pro/NC session or the dedicated icon in a Pro/NC session

# Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Process Export for Pro/NC from the Process Table

**Summary**

By exporting a manufacturing process from the Process Table (steps) in CSV format, you can use it in a Microsoft Excel spreadsheet.

**Benefits and Description**  (200words)

Any information based on a view layout (see Views in Process Manager) can be exported into a CSV file for use directly by Microsoft Excel. With the CSV file, you can generate any kind of report using the standard Microsoft Excel functions.

**User Interface Location**

File / Export Table (CSV) in the Mfg Process Table dialog box

## Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Start Model for Pro/NC

**Summary**

Pro/NC sessions can be started by using a "start model" similar to the parts and assemblies model "start part."

**Benefits and Description**  (200words)

As an NC Programmer, you can now start a new NC session from an existing environment. Any existing NC file can be reused as a start model. A typical start model can include components for the fixture, operations, machine tools, probes, and so forth.

**User Interface Location**

File / New / select Use Default Template / set the two `config.pro` options:

`mfg_start_part` **set to** `yes`
`start_model_dir` **set to the directory where the start model is stored**

## Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Referencing a Previous NC Reroughing Tool Path

**Summary**

The reroughing NC Sequence allows the selection of a previous reroughing tool path as a reference.

**Benefits and Description**  (200words)

As an NC Programmer, you can now select a previous reroughing NC sequence as a reference for a new reroughing operation. Material left by the previous roughing and reroughing parent sequences are machined by the new reroughing tool path.

**User Interface Location**

MACHINING / NC Sequence / Re-roughing / Select NC sequence from sequence list

# Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Shaded Tool for NC Toolpath Simulation

**Summary**

Toolpath simulation in Pro/NC displays the tool shaded, allowing dynamic rotation and zooming during simulation. It automatically performs line removal on the tool path motions.

**Benefits and Description**  (200words)

To improve simulation and facilitate toolpath verification, the NC Sequence **Play Path**, by default, shades the tool during the simulation. This shading applies to standard, sketched, and solid tools.

Line removal is automatically processed for the toolpath motions, allowing better visibility of the tool path. Dynamic model view manipulations (zoom, pan, rotate) are also allowed during toolpath simulation, giving complete control of the simulation.

To simplify access to toolpath simulation, Play Path (CL Player) is directly available in the Model Tree using a shortcut (right-click) menu. You can select an individual NC sequence, a set of sequences, and an operation for toolpath simulation from the Model Tree.

**User Interface Location**
MACHINING / NC Sequence / Play Path / Screen Play /  …then select View / Shaded Tool

## Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Views in Process Manager

**Summary**

You can create custom views in Process Manager.

**Benefits and Description**  (200words)

By formatting the Process Table, you can display any information related to the machining process. You can also choose what information to include. These custom views allow you to select which step information should be displayed, in which order it should be presented, and eventually which way to filter it. When exporting the table into a CSV file, this exact content is exported.

**User Interface Location**

The icon for View Builder in the Process Table

# Complete Machining Option

Manufacturing (NC, Expert Machinist, GPOST, FIL)

**Title**

Toolpath Computation

**Summary**

Toolpath computation for NC sequences and operations can be done from the Model Tree. The simulation is done with a shaded tool for a single-step display.

**Benefits and Description**  (200words)

Directly from the Model Tree, you can start the toolpath computation on a complete operation, on a single NC sequence, or on a multiple selection of NC sequences. Right-click an item in the Model Tree and select **Play Path** from the shortcut menu.

The shortcut menu improves productivity by providing an easy way to start the CL player to play the selection from the Model Tree. The **Play Path** command provides automatic simulation of any selection, such as one or more operations, or continuous or noncontiguous NC sequences. It creates all necessary settings on the fly.

**User Interface Location**

In the Model Tree, right-click an NC sequence or operation. Select Play Path from the shortcut menu.

# Mechanism Design Extension (MDX)

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Mechanism's 6 Degrees of Freedom Connection

**Summary**

A 6 Degree of Freedom (DOF) connection provides you with the flexibility to specify connection constraints to real-world problems.

**Benefits and Description**

With a 6 DOF connection, you can model connections unaccounted for with currently available joint and connection types. This ability makes it easier and quicker to complete a mechanism model. You can easily:

- Model and drive spatial, 3D motion of a free body, such as a falling ball.
- Model and apply servomotor-driven profiles to the rotational DOF of a bearing-joint ball. Previously, joint axes were not available for these joint types.
- Model an unlimited number of general joints that require various combinations of the three translations and three rotational DOF of a body.

**User Interface Location**

Insert / Component / Assemble / Connect / 6DOF

# Mechanism Design Extension (MDX)

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Cam Measures for Mechanism

**Summary**

**Curvature**, **Pressure Angle**, and **Slip Velocity** have been added to the measures commands to expand the results available for examining cam connection, kinematic behavior.

**Benefits and Description**  (200words)

You can generate expanded results for Cam connections for the following measure types to match the functionality of Mechanica Motion:

- Curvature—Measures the curvature (1/r) at the point of contact for the specified cam surface.
- Pressure Angle—Measures the angle (degrees) between the normal on the specified cam and the velocity of the contact point.
- Slip velocity—Measures the tangential velocity of the contact points on each cam surface relative to one another.

**User Interface Location**

Mechanism / Measures / New / Type / Cam

## Mechanism Design Extension (MDX)

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Evaluation at Time and at Intervals

**Summary**

Design Studies in Mechanism design (MDO) can measure the performance of a mechanism at a particular instance in time and at intervals.

**Benefits and Description**  (200words)

Evaluation of a measure at a specified time allows you to run evaluation analysis at a particular time during the motion run. In this fashion, design studies can query performance measurement at desired and discrete intervals throughout the time domain of an analysis. Dynamic analyses can be run with 0 duration to query the value of a measure at time = 0.

**User Interface Location**

In the Measure Definition dialog box, select either At Time or Each Time Step from the Evaluation Method menu.

## Mechanism Design Extension (MDX)

Simulation (Mechanism Dynamics, Design Animation)

**Title**

Point-to-Point Separation Measure and Forces

**Summary**

Mechanism Dynamics supports the ability to define point-to-point separation measures and forces, which match the functionality of Mechanica Motion.

**Benefits and Description**  (200words)

You can generate expanded results for Mechanism Dynamics for accurate measurement of the separation distance, speed, and dependent forces, such as nonlinear springs and dampers.

Point-to-Point Separation Measure allows you to capture separation distance, velocity, and acceleration between two points. You can specify force expressions dependent upon point-to-point separation, velocity, and acceleration. Currently, the creation of a point-to-point distance measure is only provided through BMX.

Forces can now be defined at a point relative to a coordinate system, on a joint axis, or as linear springs. Point-to-point forces enable generalized separation distance for dependent forces, such as nonlinear springs, nonlinear dampers, and magnetic forces that accept both table input and user-defined expressions.

**User Interface Location**

Application / Mechanism

## Mechanism Design Extension (MDX)

Model Analysis (BMX, ModelCHECK, Pro/PROGRAM)

**Title**

Subassembly Mechanism Editing

**Summary**

All Mechanism entities, such as motors, springs, and joints, are accessible from the top-level assembly for creating, editing, or deleting.

**Benefits and Description**  (200words)

You can work directly with any mechanism entity from within the top-level assembly. During direct modeling workflow, use preselection highlighting, Model Tree selection, or context-sensitive shortcut menus (right-click) to activate a subassembly to create, edit, or delete mechanism entities. This improved workflow within mechanism models decreases the time to manipulate a mechanism model, improves the ability to visualize the work being completed, and provides behavior consistent with Pro/ENGINEER assemblies.

**User Interface Location**

Right-click menu / Activate

## Distributed Applications

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**

ATB Operations and New Data Interfaces in Distributed Pro/BATCH.

**Summary**

Distributed Pro/BATCH now supports the new data interfaces introduced in Pro/E Wildfire 2.0. Also, you can now perform Associative Topology Bus (ATB) update and verify operations in batch mode on remote network machines.

**Benefits and Description**  (200words)

In Pro/E Wildfire 2.0, you can use Distributed Pro/BATCH to perform import and export of new data interfaces introduced in this release. Batch processing templates (TTD) are included for the following new formats:

- CATIA V5 CATPart (Import)
- I-DEAS (Import)
- Unigraphics (Import and Export)
- Parasolid (Export)

Distributed Pro/BATCH allow allows you to offload the time-consuming and resource intensive ATB operations to be processed in batch mode on one or more remote machines on your network. The following types of ATB operations can be performed using dBATCH:

- Verify
- Update

**User Interface Location**

N/A

# Distributed Applications

Data Exchange (ATB, DataDoctor, Interfaces)

**Title**

Distributed Computing Toolkit to truly leverage your network resources.

**Summary**

Distributed Computing technology is Wildfire 2.0 is greatly enhanced with the introduction of a C (C++ compatible) and Java Application Programmers Interface (API) for developing custom distributed applications.

**Benefits and Description**  (200words)

In the Pro/E Wildfire release, PTC introduced the 'Distributed Services Manager' and 'Distributed Pro/BATCH', a client/server application that allowed you to perform common Pro/E operations like plotting, import/export, etc. on remote network machines in a distributed computing environment. With Pro/E Wildfire 2.0, you can develop your own custom clients/server applications that will utilize the distributed processing capabilities of the DSM in performing the desired tasks on remote networked machines.

When you install the DSM, you can also choose to install the C and Java toolkits that will allow you to build custom plug-in components (distributed clients and services).

**User Interface Location**

N/A