Parametric Technology Corporation

Pro/ENGINEER® Wildfire™ Preproduction Release Notes

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About This Guide

Pro/ENGINEER Wildfire Release Notes highlights the new functionality for this release. A streamlined user interface provides maximum visual feedback, more accuracy with less typing, and a new dashboard for smart input for features in Pro/ENGINEER. The new Navigator provides easy access to a collaborative environment with shared connectivity for users, Web access, and integrated data management.

The release notes are organized according to the Pro/ENGINEER functional areas that appear on the Pro/ENGINEER Help Center home page.

Resources and Services

For resources and services to help you with PTC (Parametric Technology Corporation) software products, see the *PTC Customer Service Guide*. It includes instructions for using the World Wide Web or fax transmissions for customer support.

Documentation for PTC Products

PTC provides documentation in the following forms for Pro/ENGINEER Wildfire Preproduction:

• Context-sensitive Help

• *PTC Customer Service Guide*, *Pro/ENGINEER Wildfire Preproduction Installation and Administration* Guide in PDF format on the product CD-ROM. To view and print PDF books, you must have the Acrobat Reader installed.

- Pro/ENGINEER Help Center with a global search facility and Quick Links to helpful introductory information.

When your PTC software has been installed, you can drag from the Pro/ENGINEER menu bar to any dialog box to get Help or right-click on any menu.

PTC Documentation Conventions

PTC documentation uses the following conventions.

Convention	Item	Example
Bold	Menus paths, dialog box options, buttons, and other elements from the user interface	Click **File** > **Save**. **Assignee** check box **License File** dialog box
`Courier`	User input, system messages, file names	`Processing completed.`
Courier with less-than and greater-than symbols (< >)	Variables for which the user substitutes an appropriate value	`output=<LOADPOINT>`

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Fundamentals

Pro/ENGINEER Fundamentals and Design Conferencing provide basic
information about Pro/ENGINEER Wildfire Help. Highlights of new
functionality for this release follow.

Pro/ENGINEER Fundamentals

PDM Information in the Model Tree

Benefits

Better visibility of the status of the models in session keeps you
informed.

A new Database Params option has been added to the list of types of
information that can be displayed in the Model Tree. By selecting this
option **(Settings** > **Tree Columns** from the Model Tree), you can show
important PDM information for the models in session. This information
includes PDM lifecycle data and the ¿editable status¿ of the object in
session.

Menu Location

Settings > **Tree Columns** from the Model Tree

Display of Clipped Views

Benefits

The display of the clipped view can be applied to both assemblies and
parts and allows for a clearer display of the 3D models.

After you create a cross section, click the **Display** tab and change the **Visibility Options**. You can show either the front or the back of the model based on the cross section selected. A preview of the model with arrows to indicate directions shows which side is considered the front or back. You can also display the cross section in the clipped view. The clipped view remains visible even after you close the Model Sectioning tool.

Menu Location

Tools > **Model Sectioning**

Support for HTTP and FTP File Locations

Benefits

The support for HTTP and FTP file locations provides users with improved file- and data-sharing capabilities.

Built-in Web protocols extend the reach of Pro/ENGINEER from the desktop to the Internet. Internet locations (such as ftp://ftp.ptc.com) can now be registered with Pro/ENGINEER through the Server Registry command in the Tools menu. After a location is registered with Pro/ENGINEER, it is available to all Pro/ENGINEER file operations such as **File** > **Open, File** > **Backup, Component** > **Assemble**, and **Component** > **Replace**. You can also enter an HTTP or FTP address into the **Open** dialog box. Note that saving to simple HTTP locations is not supported, and simple HTTP locations are not, therefore, available through the Server Registry.

Menu Location

User Area connection on the Connections navigator

Pro/ENGINEER Wildfire Home Page

Benefits

The Pro/ENGINEER home page can increase productivity by presenting useful information on startup.

By default, Pro/ENGINEER displays the current Pro/ENGINEER home page on startup. You can customize this default behavior in two ways (1) Click **Tools** > **Customize Screen** > **Browser**. Click or clear the **Expand Browser by default when loading Pro/E** check box. The Pro/ENGINEER home page will still load, but it will not be displayed on startup; or (2) Set the `web_browser_homepage.config` configuration option to display a different home page.

Menu Location

Pro/ENGINEER browser

Console Records Server Transactions

User Interface

Benefits

Transactions with a server can execute in the background, allowing you to continue working in Pro/ENGINEER. The console records all background messages and statuses.

Many Windchill transactions run asynchronously in a Pro/ENGINEER background thread. This means that long-running Windchill transactions (such as uploading large assemblies) will not stop you from using Pro/ENGINEER while the transaction completes. At any time during or after the transaction, you can check the console for progress on uploading and downloading files.

Menu Location

Console icon on the Pro/ENGINEER status bar or from the Pro/ENGINEER menus (**Tools** > **Console**)

Browser Integration with Help

Benefits

Pro/ENGINEER has better control of the Help browser, which provides improved usability of the Help system.

Pro/ENGINEER Help is now displayed in a floating version of the built-in Pro/ENGINEER browser. The Pro/ENGINEER browser provides all of the capabilities of the Help system, but improves overall usability when compared to previous Pro/ENGINEER releases, which used an independent browser. The Help browser remains on top of the Pro/ENGINEER window and remembers its last size and location. You can also create bookmarks from the Help browser.

Menu Location

Help menu

Enhanced Drag Capabilities

Benefits

The enhanced drag capabilities make many file operations simpler and faster to use.

Pro/ENGINEER Wildfire provides enhanced file management through expanded drag capabilities. File data, such as Assemblies, Sub-Assemblies, Parts, and user-defined features, can be dragged into a Pro/ENGINEER session for retrieval; Zipped files can be dragged into a Pro/ENGINEER session or input panels and are unzipped automatically; Files can be dragged onto the Model Tree to determine exactly where in the model the data will be added; Files can be dragged from the embedded browser into the Pro/ENGINEER graphics window or Model Tree; Files can be dragged into the Pro/ENGINEER browser and processed by the browser according to the browser MIME type settings.

Menu Location

N/A

Improve and Enhance Dynamic View Manipulation Capabilities

Benefits

While using mouse interactions and modifier keys, you can focus on tasks to be completed. Expanding upon modern standards, Pro/ENGINEER Wildfire provides viewing‑related capabilities via the middle mouse button.

Pro/ENGINEER Wildfire provides two direct methods of viewing objects using the mouse basic viewing capabilities and advanced options. Basic viewing capabilities include (1) Spin ‑ Middle‑click and drag the object. (For 2D objects use Pan instead.); (2) Zoom ‑ Hold down CTRL, middle‑click, and drag the object. (Roll wheels can perform zoom also.); (3) Pan ‑ Hold down SHIFT, middle-click, and drag the object.

Advanced viewing options, such as spinning around a specific vertex, are also available. You can turn off the spin center and use the pointer to define the point at which the model will spin. You can select an entity to spin about a specific edge or vertex.

Menu Location

N/A

Using Keyboard Modifiers and Mouse Buttons

Benefits

You can use standard Microsoft keyboard functions and mouse operations in Pro/ENGINEER. For example, right-click to access shortcut menus.

Pro/ENGINEER mouse operations and keyboard modifiers (CTRL and SHIFT) provide many productivity-enhancing shortcuts using the familiar Microsoft standards. For example, use CTRL to select multiple objects, or use SHIFT to select all the features between two selections from the Model Tree.

Menu Location

N/A

Windchill PartsLink Integration

Benefits

With access to PartsLink, you can easily locate and download 3D models directly from suppliers, avoiding the need to create them yourself.

Selecting Catalogs on the Connections navigator displays a PTC Web site of catalogs that are powered by Windchill PartsLink. From this site you can browse the catalogs directly or choose to personalize the Connections navigator by adding links to the catalogs that you use most frequently. The online catalogs offer powerful search and browse capabilities, so you can quickly locate parts and download them directly into Pro/ENGINEER.

Menu Location

The Connections navigator

Selecting with Pick Boxes

Benefits

Consistent selection tools throughout Pro/ENGINEER provide familiarity. You can select multiple items at any time.

Selection has been redesigned to make it more user-friendly and consistent throughout Pro/ENGINEER. You have the same tools for selection and have the ability to select multiple objects at any time. Holding down the left mouse button and dragging the pointer creates a

pick box for selecting multiple items. Selection filters control the scope of items that can be selected.

Menu Location

Hold down the left mouse button and drag.

Undo and Redo Support

Benefits

Use Undo and Redo to reverse or redo an action, thereby avoiding the need to recreate an attribute after mistakenly deleting it.

Standard Undo/Redo support is available for feature options and settings. Your actions are recorded in a stack that you move through with top‑level icons or with standard keyboard shortcuts (CTRL + Z).

Menu Location

Edit > **Undo** or **Edit** > **Redo**

System Parameters from Mass Properties

Benefits

You can calculate system mass property parameters on each regeneration, assuring up-to-date mass properties calculations.

System parameters are created for mass properties and can be calculated on each regeneration or only when the mass properties are calculated. The new system mass property values can also be assigned through relations that are evaluated after the regeneration of the model Reporting of mass property values is expanded to provide more detailed component information in assemblies.

Menu Location

Edit > **Setup** > **Mass Props**

Connections Navigator

Benefits

The Connections navigator provides convenient access to frequently used PTC Web resources and built‑in Web applications.

The Connections navigator contains built-in connections to Projects (powered by Windchill ProjectLink); Catalogs (powered by Windchill

PartsLink); PTC User Area and User Group (PTC User Group Web site); ptc.com; and Technical Support.

Selecting any of these top-level connections takes you to the appropriate Web site, from which you can add connections to the Connections navigator. Each top-level connection on the Connections navigator holds a subscription to its target page. As long as the subscription is in effect, a change to bold type on a connection label indicates that the target page has been updated since you last visited that page. Subscriptions can be canceled or re-enabled by using the shortcut menu on the connection label.

Menu Location
The Pro/ENGINEER navigator

Screen Customization of the Pro/ENGINEER Navigator and Browser

Benefits
You can increase your productivity by customizing the Pro/ENGINEER navigator and browser.

The **Customize Screen** dialog box includes options to control the appearance and behavior of the Pro/ENGINEER browser and navigator. The new options are available from the **Toolbars** tab, the **Navigation** tab, and the **Browser** tab in the **Customize** dialog box.

Menu Location
Tools > **Customize Screen**

Access to PTC User Area

Benefits
The User Area increases productivity and Pro/ENGINEER product awareness by providing access to the latest Pro/ENGINEER information, as shared by Pro/ENGINEER users worldwide.

The PTC User Area is a Web-based resource hosted by PTC. It is designed to make product development professionals more productive by providing access to relevant tools and technical product information. Access to the The PTC User Area is a Web-based resource hosted by PTC. It is designed to make product development professionals more productive by providing access to relevant tools and technical product information. Access to the PTC User Area is built into Pro/ENGINEER through the Connections navigator. While you are connected to the Internet, click the User Area in the Connections navigator to display

the top page of the User Area in the Pro/ENGINEER browser. You can personalize the User Area by creating bookmarks to the pages that you visit most frequently. The bookmarks are added under the User Area in the Connections navigator.

Menu Location

User Area connection on the Connections navigator

Built-In Web Browser

Benefits

The built-in Pro/ENGINEER browser provides Internet access while allowing you to remain focused on the model.

The Pro/ENGINEER browser slides out from the Pro/ENGINEER Navigator. A "quick sash" provides one-touch retraction or expansion of the browser. The browser uses industry-standard browser components (Internet Explorer on Windows and Mozilla on UNIX) to deliver Web access in a user environment that works well with Pro/ENGINEER modeling tasks. Pro/ENGINEER uses the browser to display information from the Pro/ENGINEER model, such as from Feature ¿ Info, as well as information from Web services, such as Customer Support, online catalogs, Windchill Solutions, or the PTC User Area. A Web service accessed from the Connections Navigator executes in its own browsing thread, which Pro/ENGINEER creates as needed. You can switch between browsing threads using the tabs that appear above the browser toolbar. Only the general browsing thread (the first one created) has an address bar.

Menu Location

N/A

Pro/ENGINEER Navigator

Benefits

The Pro/ENGINEER navigator provides access to data from a variety of sources in a familiar and easy-to-use interface.

The Pro/ENGINEER navigator contains the following navigation tools: Model Tree - Provides navigation of and interaction with the current Pro/ENGINEER model; Folders - (Default) Provides navigation of the local file system, the local network, and Internet data; Favorites - Contains user-selected Web locations (bookmarks) and paths to Pro/ENGINEER objects, database locations, or other points of interest; Search - Provides search capability for objects in the data management

system. Note that the Search option appears when you declare a Windchill system as your primary data management system; History - Provides a record of Pro/ENGINEER objects you have opened and Web locations you have visited. Click the History icon on the browser toolbar to add the option to the Pro/ENGINEER navigator; Connections - Provides access to connections and built-in PTC solutions, such as Pro/COLLABORATE, PartsLink, and the PTC User area.

Menu Location

The left side of the user interface.

Relations Tool

Benefits

The Relations tool allows you to quickly build relations in Pro/ENGINEER.

The Relations tool allows you to switch between the levels of relation from a list. When working with multiple options, you can select a specific part, for example, from the graphics area. You can add relations that are calculated after the model geometry is regenerated using the Post Regeneration option in the Relations tool. You can use the familiar icons for cut, copy, paste, and delete to edit the relations. Another set of intuitive icons allows you to switch between dimensional values and symbols as well as insert relations-building information. Functions can be inserted from a list. Dimensions can be inserted by selecting them in the graphics area. Parameters can also be inserted from a list. You can manipulate relations. Dimensions in the graphics area can be highlighted as they are covered in the text window. Relations can be verified and sorted.

Menu Location
Tools > Relations

Send To Capability in Pro/ENGINEER

Benefits

The Send To capability providers an easy way to exchange data with remote users through e-mail.

The Send To capability allows you to send models from your current session by e-mail. You can choose an export format for the files, such as IGES or STEP, and compress the files into a Zip file before sending them.

Menu Location

Click **File** > **Send To** or right-click **Send To** from the shortcut menu on the Model Tree.

HTML Display of Text-based Information Files

Benefits

These text-based information files are reformatted and contain hyperlinks that allow interaction between the reports and the Pro/ENGINEER model.

The following reports, which are accessible through the Info menu, have been reformatted and are now output in the Pro/ENGINEER browser: Feature, Model, Relations and Parameters, Feature List, and Audit Trail. The new format is more readable and provides additional functionality that allows you to interact with the model in session. For example, from the Information report you can highlight dimensions or navigate up and down parent-child hierarchies by clicking hyperlinks. For backward compatibility, the original text-based format is still available and can be reactivated by setting the info_output_format config.pro option to text (The default value is html).

Menu Location

Info menu

Enhanced Surface Collection User Interface

Benefits

You can quickly transition from a single surface to a set of surfaces without having to navigate through menus or cancel actions.

To begin, select a solid or quilt surface and press SHIFT to indicate that you want to use the selected surface as the anchor of a parametric surface set. While pressing SHIFT, you can only select surface sets that can be created from the anchor. The type of surface you construct depends on the type of anchor specified and the current target (the item over which the pointer is positioned). You can choose to construct loop faces, seed and boundary faces, or all solid faces surface sets. While you gather sets of surfaces, you are provided with valuable feedback.

Menu Location

N/A

Hyperlinks on Pro/ENGINEER Text

Benefits

Models can reference data that is outside of Pro/ENGINEER through hyperlinks. Models are more self-describing, which allows you to add extensive and reusable annotations.

You can apply hyperlinks to 3D notes (as in previous releases) and 2D notes (including table notes). When a text item has an associated hyperlink, visual feedback is now given in the form of underlined text. In addition, when the pointer hovers over the note, a ToolTip displays the value of the hyperlink. Clicking the text while pressing CTRL activates the hyperlink. A context-sensitive pointer appears when you press CTRL while the pointer is on a note with a hyperlink.

Menu Location
Setup Note > **Insert Hyperlink**

Advanced Selection and Definition

Benefits

You can now focus on desired results rather than on how to accomplish the task. By receiving continuous feedback on pending results you can achieve what you want on the first attempt.

You can create extensive, easily expandable rule-based selections in all applications with a high degree of customizability using the Search Tool. The new dialog box provides easy access to items previously in the **Select By Menu** dialog box. In addition, you can collect and filter items for selection, search for items based on rules, and store your search results into layers.

Menu Location
Edit > **Find**

Snapping Drag Handles

Benefits

You can quickly add definition to features by using consistent handle functionality while staying focused on the model or interacting with the dashboard.

You can drag a handle to a specific value. You can also control handle values using other methods. You can move a handle to a specific reference by pressing SHIFT and moving the pointer to the desired

reference. To unsnap a handle from a reference, press SHIFT and drag the handle out of the snapping zone. The handle is no longer snapped to reference, and the dashboard displays the updated dimensions.

You can also specify the dimension interval when dragging handles using the configuration option `zzz_grid_interval`, where `zzz` represents the different units that can be set on the model, such as `inch_grid_interval` when using inches. This option can be set to a value, such as `.100`, which will force all dragging to snap to dimensional values divisible by `.100`.

You can also place the handle at an exact location by double-clicking the displayed dimension in the graphics area and entering the value.

Menu Location

N/A

Enhancements to the Model Tree

Benefits

You can quickly perform feature operations by selecting features on the Model Tree rather than navigating through menus or dialog boxes.

You can rename features easily by double-clicking the name in the Model Tree or by pressing F2.

Menu Location

N/A

Working with Zip Files

Benefits

Sending and retrieving Zip files is an easy way to informally share Pro/ENGINEER data.

Zip files that contain Pro/ENGINEER data can be opened in Pro/ENGINEER in one of the following ways(1) Click **File** > **Open** in Pro/ENGINEER and open the Zip file; (2) Drag the Zip file onto the Pro/ENGINEER graphics window. The **Open** dialog box opens; or (3) Drag the Zip file onto the Pro/ENGINEER browser. This action uses the browser's MIME setting and opens the Zip file using the currently configured application.

Menu Location

File > **Open** and drag the Zip file onto the graphics window or the browser

Designation Features and Geometry as Objects

Benefits

The ability to have features or geometry represent components enables faster assembly creation. Features and geometry can be designated as components and will be visible in Bills of Material and Pro/REPORT tables.

You can designate features or geometry as objects in the **Designate** dialog box For example, a cosmetic feature can be designated as label and assigned a part number and cost. In the Bill of Materials, the feature would appear as a label part. In Pro/REPORT, the label part could also show values for cost and part number. Designating a Shrinkwrap feature will show that the shrinkwrap feature represents all of its reference models

Menu Location

Edit > **Setup** >**Designate**

Design Conferencing

Design Conferencing for Pro/ENGINEER Users

Benefits

Design Conferencing allows people to work together, reducing the need to travel to meet with one another. It also eases the flow of information between people.

In Design Conferencing, one user creates a conference and invites others to join. These other users must be connected to the Internet. The leader has control of everyone's Pro/ENGINEER session.

Users can also work offline. When they reconnect, the shared spaces are automatically synchronized. A shared space is an XML database and binary file stored on your machine, which is synchronized with all other machines participating in the same conference. The initiator can store models in the shared space. Everyone who accepts the invitation to the space can retrieve the model and make changes to it. When the model is saved back to the shared space, it is also copied to everyone else¿s shared space. Therefore, all participants always have the latest copy of a design.

Machines can be configured to automatically log into the Groove relay server, which is hosted by Groove Networks. Users can then see and search for each other online.

Menu Location
Tools > **Conference**

ProjectLink Integration

Benefits

ProjectLink provides an excellent way for you to manage the exchange of project-related information with others.

Windchill ProjectLink allows you to create a project on the ProjectLink server, upload data to the project directly from Pro/ENGINEER, and then invite other people to join the project. Only the people that you invite to your project can see and access your project data. Of course, ProjectLink provides much more than this, and the best way to explore the ProjectLink potential is to try it. To access the ProjectLink evaluation server, click **Projects** on the Connections navigator and follow the instructions on that Web page. The ProjectLink server contains a brief tutorial to get you started. After you have successfully created a project and registered it with Pro/ENGINEER, sending data to ProjectLink is as simple as saving it to another disk (clicking **File** > **Backup**). Retrieving data from ProjectLink is as easy as clicking **File** > **Open**.

Menu Location
Navigator > **Connections tab** > **Projects**

Favorites Navigator

Benefits

The Favorites navigator increases productivity by allowing you to quickly return to frequently used sites.

You can add the currently displayed Web page to your list of favorites either by using the shortcut menu or by using the Add to Favorites command on the Favorites navigator. The list of favorites is specific to the Favorites navigator and is different from the list of favorites that you may have created with your general browser.

Menu Location
Pro/ENGINEER navigator

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Part Modeling

Part and Sketcher make up the part modeling functional area for
Pro/ENGINEER Wildfire Help. Highlights of new functionality for this
release follow.

Part

Using the Extrude Feature

Benefits

One tool for extrude combines capabilities previously offered
separately. This tool provides instant graphical preview with drag
handles, during creation and redefinition

After defining a sketch, you immediately get dynamic preview geometry
based on intelligent defaults. Drag handles allow you to freely modify
the depth to a desired value or snap the handle to a reference. With
dashboard settings or shortcut menus, you can quickly change between
options such as solid or surface, protrusion or cut, and solid or thin.

Menu Location
Insert > **Extrude**

Working with Revolve Features

Benefits

The Revolve tool consolidates the capabilities of several features. It also
provides instant graphical preview with drag handles during creation
and redefinition.

After defining a sketch, you immediately see dynamic preview geometry based on intelligent defaults. Drag handles allow you to freely modify the angle of revolution to a desired value or snap the handle to a reference. With dashboard settings or shortcut menus, you can quickly switch between options such as solid or surface, protrusion or cut, and solid or thin.

Menu Location

Insert > Revolve

Using the Trim Tool

Benefits

The new Trim tool gives you the ability to cut back quilts or surfaces using other quilts, surfaces or planes. You can perform the operation using graphical direct manipulation or dashboard-style controls.

After you select the objects to trim, you immediately preview the resulting geometry. By specifying other options in the dashboard or in context-sensitive menus, you can progress to more complex trim geometry. For example, you can set attributes to keep the trimming surface or create a thin trim operation.

Menu Location

Edit > Trim

Working with Rounds

Benefits

There is no distinction in the interface between simple and advanced rounds. All rounds are created with drag handles and interactive dynamic preview.

Every round feature contains a least one set. By editing the set, or creating additional sets, you can make the round geometry more complex. For variable rounds, simply select a drag handle and right-click to add a radius. Drag the base of the handle along the edge chain to control the position of the handle. Use a Shortcut menu to create full rounds and transitions.

Menu Location

Insert > Round

Pattern User Interface and Workflow

Benefits

The dashboard of the Pattern interface provides easy access to all the information necessary when creating or redefining patterns of parts or features.

On the dashboard you can enter all the pertinent information when creating patterns of parts or features. You can enter the pattern type, number of instances and spacing directly on the dashboard. With the dashboard you can both create and redefine patterns. All the previous functionality for patterns is available on the pattern dashboard.

Menu Location
Edit > Pattern

Working with the Move Tool

Benefits

The Move tool allows you to translate or rotate quilts, datum curves, and axes using graphical, direct manipulation, or dashboard-style controls.

Select quilts, curves, or axes, and then activate the Move tool. Use the Move tool to translate or rotate by interacting with the dashboard or by using right mouse button actions. After you select a direction reference, an interactive geometric preview appears in which you can move the item to the desired location.

Menu Location
Edit > Move

Separating Sketcher

Benefits

You can quickly redefine a feature's shape by changing only the sketch feature.

Sketch features now appear in a model's Feature tree. In Pro/ENGINEER Wildfire, sketches can be redefined by simply expanding the feature group to change the sketch itself. Sketches can also be created prior to feature creation. This allows you to replace sketches easily by redefining the feature and selecting a new sketch as a reference. (In the past, sketches were absorbed by the feature and could only be changed by redefining the feature.) You can also continue

to create sketches within a feature tool by selecting the Sketch icon in the dashboard. After a sketch is used and the feature is completed, the sketch will automatically be hidden to clean up the display of the graphics area. NoteIf external sketches are referenced in a feature tool, the feature tool will create an independent copy of the original sketch.

Menu Location
Insert > Model Datum > Sketched Curve

Working with Datum Coordinate Systems

Benefits
All the ways to create datum coordinate systems are consolidated and controlled by your selections on the model and from the options available in the streamlined dialog box.

The Datum Coordinate System feature has dynamic preview and drag handles so that you can quickly create datum coordinate systems. All create options in the dialog box are also available when redefining the datum coordinate system.

Menu Location
Insert > Model Datum > Coordinate System

Working with Datum Planes

Benefits
Using the datum planes interface, you can create, name, and size datum planes. You can construct planes using graphical direct manipulation or the dialog box.

The Datum Plane feature has dynamic preview and drag handles so you can quickly create datum planes. All Datum Plane creation methods are consolidated and controlled by your selections and by the options available in the new streamlined dialog box.

Menu Location
Insert > Model Datum > Plane

Using the Draft Tool

Benefits

The Draft tool provides instant graphical preview with drag handles during creation and redefinition, which allows you to focus on the model or interact with the dashboard.

The Draft tool consolidates all surface and solid draft features. By using drag handles or interacting with the dashboard, you can create simple or complex geometry. You can access advanced attributes through slide-up panels on the dashboard or by using the shortcut menu. As any attribute is defined or modified, the dynamic geometry preview updates, showing how those changes alter the model.

Menu Location
Insert > **Draft**

Working with Extend

Benefits

The Extend tool provides instant graphical preview with drag handles during creation and redefinition so you can focus on the model or interact with the dashboard.

The Extend tool allows for the use of drag handles and the dashboard to define extensions on surfaces. Adding a drag handle can create variable extensions. All extend types are available through interaction with the dashboard. As any of the extend definitions are modified, the geometry preview updates immediately.

Menu Location
Edit > **Extend**

Feature Toolbar

Benefits

Intuitive icons provide quick access to the most commonly used features in Pro/ENGINEER.

The **Feature** toolbar organizes the most common feature creation and edit tools into three toolbars. Creation tools such as extrude, revolve, sweep, style, and boundary blend are located in one toolbar. Standard engineering features such as holes, shell, draft, rounds and chamfers are located in a second toolbar. The third toolbar holds edit features such as move, mirror, pattern, and merge.

Menu Location

N/A

Creating Field Datum Points

Benefits

You can create field datum points and redefine the attributes for field datum points using one dialog box.

The **Field Datum Point** dialog box allows you to quickly define placement planes. You can also set properties such as feature names using the **Properties** panel in the dialog box.

Menu Location
Insert > **Model Datum** > **Point** > **Field**

Working with Fill

Benefits

The Fill user interface provides dynamic preview and a streamlined dashboard that is consistent for all features.

The Fill tool allows you to quickly create a flat surface after selecting or sketching a closed section.

Menu Location
Edit > **Fill**

Folder Navigator

Benefits

The Folder navigator provides quick access to Pro/ENGINEER data from a wide range of local or network-based locations.

The Folder navigator provides a uniform and familiar way to access any data location that has been registered with Pro/ENGINEER. Some locations, such as the local file system, the current Pro/ENGINEER session, and the Network Neighborhood, are built in. Other locations, such as Windchill servers, a Pro/INTRALINK server, FTP locations, or Conference Spaces, can be added to the Folder navigator by registering these locations through Tools > Server Registry. Using the Folder navigator, you can browse registered locations. After finding a Pro/ENGINEER file or assembly of interest, the file or assembly can be loaded into Pro/ENGINEER.

Menu Location

Pro/ENGINEER navigator

History Navigator

Benefits

The History navigator increases productivity by allowing you to quickly return to a previously visited site.

The History navigator, which appears in the Pro/ENGINEER navigator only when you click the History icon, manages the list of Web sites that you have visited through the Pro/ENGINEER browser. The History navigator shows the browser history as a list, which you can sort and manage. The History navigator is specific to Pro/ENGINEER and differs from the history list that is created and maintained by your general browser.

Menu Location

Pro/ENGINEER navigator

Using the Intersect Tool

Benefits

You can use one tool to create curves at surface intersections or when projecting two separate curves.

The Intersect tool consolidates two previous ways of creating curves. If you select two surfaces, the tool creates a curve at the intersection of those surfaces. If two sketched curves are selected, then a curve is created at the projection intersection of both sketches.

Menu Location

Edit > Intersect

Working with Merge

Benefits

You can use direct modeling concepts and shortcut menus to toggle options such as Intersect or Join when creating merge features.

The Merge tool provides a dashboard interface consistent with that of other features. The Merge tool collects two quilts and displays arrows indicating the sides of the quilt to keep. By default, the operation uses the Intersect function, but you can change this option to Join either by interacting with the dashboard or by using the shortcut menu.

Menu Location

Edit > Merge

Working with Patch

Benefits

The Patch tool provides instant graphical preview with drag handles during creation and redefinition so you can focus on the model or interact with the dashboard.

The Patch tool consolidates features that add or remove material using quilts or surfaces and includes functionality for replacing solid surfaces with quilts. The Patch tool uses a dashboard and shortcut menus for advanced options. As any changes are made to the definition of the feature, graphical preview updates, showing how those changes affect the model.

Menu Location

Edit > Patch

Using the Project Tool

Benefits

You can use direct modeling and advanced options on shortcut menus when projecting curves.

The Project tool includes a dashboard and instant preview as you are working to project curves on surfaces or solids. Use the Dashboard or right-click to define or modify attributes. Geometry Preview updates immediately.

Menu Location

Edit > Project

Working with Rib

Benefits

The Rib tool provides instant graphical preview with drag handles during creation and redefinition.

The Rib tool allows you to interactively flip the material side and the width of the rib feature. All modifications can also be made in the dashboard.

Menu Location

Insert > **Rib**

Working with Sketched Datum Points

Benefits

You can create sketched datum points and redefine the attributes of all sketched points in the same dialog box.

The **Sketch Datum Point** dialog box lets you quickly define sketching planes and references. You can also set properties such as feature names using the properties panel in the dialog box.

Menu Location

Insert > **Model Datum** > **Point** > **Sketched**

Using the Sweep Tool

Benefits

One tool provides the capabilities of several features. The Sweep tool also provides instant graphical preview with drag handles during creation and redefinition.

The Sweep tool consolidates the creation of sweep and variable section sweep features into one tool. After specifying a trajectory and defining a sketch, you get dynamic preview geometry of the sweep feature. Using CTRL, you can quickly add another trajectory and progress from a simple sweep to a variable section sweep. Dashboard settings, or shortcut menus, allow you to quickly change between options such as solid or surface, protrusion or cut, and solid or thin. Advanced attributes, such as defining a pivot direction, are available in slide-up panels on the dashboard.

Menu Location

Insert > **Variable Section Sweep**

Using the Thicken Tool

Benefits

Use the new Thicken tool to produce solid geometry by providing a thickness for surfaces or quilts using graphical, direct manipulation, or dashboard-style controls

The Thicken tool is an object-action tool. After you select a surface and enter the thicken tool environment, you are given preview geometry

with a drag handle. With a shortcut menu and the dashboard, you can add or remove material and exclude surfaces from thickening.

Menu Location
Edit > **Thicken**

Using the Wrap Tool

Benefits
You can use direct modeling and advanced options on shortcut menus to wrap curves on surfaces or solids.

The Wrap tool includes a dashboard and instant preview as you are working to create formed datum curves on surfaces or solids. You can also wrap a section with or without a sketcher coordinate system.

Menu Location
Edit > **Wrap**

Creating Datum Points Offset from a Coordinate System

Benefits
You can both create and redefine points in the progressive user interface for datum points. You can use direct modeling concepts and shortcut menus to toggle placement options.

The dialog box for creating datum points offset from a coordinate system allows you to enter explicit x-, y-, and z-values or drag points along the three axes. You can enter additional points or import a file containing the location of the points. Using a familiar tabular interface, you can easily edit point values and names in the dialog box.

Menu Location
Insert > **Model Datum** > **Point** > **Offset Coordinate System**

Streamlined Datum Point Feature

Benefits
You can both create and redefine datum points. You can also use direct modeling concepts and shortcut menus to toggle placement options.

The enhanced datum point feature consolidates all creation tools, such as On Surface, Offset Surface, Curve X Srf, On Vertex, Three Srf, At

Center, On Curve, and Offset Point, in one dialog box. The tool provides graphical preview of points based on the references selected.

Menu Location
Insert > **Model Datum Point**

Working with Boundary Blend Surface

Benefits

The Boundary Blend tool provides instant graphical preview with drag handles during creation and redefinition so you can focus on the model or interact with the dashboard..

The new Boundary Blend tool provides all the functionality needed to create surfaces through a set of curves. The dashboard allows you to have full control over the order in which curves are passed through, tangency conditions, and control points for the surface. The dynamic geometry preview updates as the surface attributes are modified or defined in the dashboard.

Menu Location
Insert > **Boundary Blend**

Working with Datum Axes

Benefits

Using the datum axes interface, you can create, name, and size datum axes. You can construct axes using graphical direct manipulation or using a new dialog box.

The new Datum Axis feature has dynamic preview and drag handles so you can quickly create datum coordinate systems. All create options in the dialog box are also available when redefining the datum coordinate system.

Menu Location
Insert > **Model Datum** > **Axis**

Working with Sketched Datum Curves

Benefits

You can both create and redefine sketched datum curves through the same dialog box. You can also modify the attributes of all sketches using this dialog box.

The new **Sketch Datum Curve** dialog box allows you to quickly define sketching planes and references. You can also set properties such as cross-hatching and section names using the **Properties** panel in the dialog box.

Menu Location

Insert > **Model Datum** > **Sketched Curve**

Working with Offset

Benefits

You can use one tool to create surface and curves offsets while leveraging direct modeling concepts and shortcut menus to toggle advanced options such as excluding individual surface patches.

The new offset tool combines eight features and incorporates Object-Action capability to provide ease-of-use and intuitive feature creation. This tool can be used on both surfaces and curves. You can quickly create the desired feature by interacting with drag handles and dynamic preview or by interacting with the dashboard. The eight features that have been consolidated are Offset curve, From curve, Offset from surface (curve), From boundary, Surface offset, Tweak offset, Draft offset, and Area offset.

Menu Location

Edit > **Offset**

New Dimension Type for Rounds

Benefits

You create conical rounds with drag handles and interactive dynamic preview, which allows you to quickly preview the geometry prior to completing the feature.

To access the new dimension type, click the Sets slide-up panel and change the round type from **Circular** to **D1** x **D2 Conic**. Drag handles to control both leg dimension values and the conic handle appear. If you change a value on the dashboard, the preview geometry updates immediately.

Menu Location

Insert > **Round**

Enhanced Fill Pattern Capability

Benefits

The new fill pattern functionality saves significant time when replicating many objects in a specific area and is more intuitive than previous techniques.

Use fill patterns to populate a region by defining a sketch or to fill a region using various spacing types. You can also define the distance between instances and the distance between the instances and the sketched boundary. If a circular spacing type is defined, use fill patterns to define the rotation of the pattern and radius. To remove unwanted instances, select directly on the preview for each pattern.

Menu Location
Edit > **Pattern**

Working with the Mirror Tool

Benefits

When you use the Mirror tool, you can use direct modeling concepts and shortcut menus to toggle advanced options, such as keeping the original feature.

Using an object-action approach, you can select quilt, curves, or axes prior to opening the Mirror tool. Once in the tool, as soon as you select a plane, a graphical preview appears. You can interact with the dashboard or use shortcut menus for options, such as keeping the original quilt, curve, or axes.

Menu Location
Edit > **Mirror**

Using the Shell Feature

Benefits

You can leverage direct modeling concepts to drag thickness handles and to use shortcut menus to toggle advanced options, such as defining special thickness surfaces.

When using the Shell tool, you can use either an object-action or action-object approach. With the drag handles you can pull dimensions or enter a value directly on the screen. Press and hold CTRL to select multiple surfaces to exclude. You can define advanced attributes, such

as special thickness surfaces, by using a shortcut menu or by interacting with the dashboard.

Menu Location

Insert > **Shell**

Working with the Hole Feature

Benefits

A compact and functional user interface provides enhanced hole functionality.

You can interact with preview geometry for hole placement, size, and depth. By snapping drag handles to appropriate references, you can quickly define hole features. You can also interact with the dashboard for other hole types, such as standard holes or sketched holes.

Menu Location

Insert > **Hole**

Working with the Copy Feature

Benefits

With one tool you can create surface copies and composite curves.

You can leverage direct modeling concepts and shortcut menus to toggle advanced options, such as creating an approximate or exact curve. The Copy feature consolidates the SURF COPY, SURF COPY BY TRIM, and Composite Curve features. You can work with your model using direct graphical manipulation and dashboard-style interaction. You receive immediate preview display feedback whenever possible and before you must commit to the feature.

Menu Location

Edit > **Copy**

New User Interface for Chamfers

Benefits

You can quickly create chamfers by selecting edges and dragging a handle. Using chamfer sets and transitions, you can create complex chamfers using one chamfer feature.

The chamfer tool operates similarly to the round tool. Users can add edges to one chamfer set by pressing CTRL while selecting the edges. Users can create new sets by selecting alternate model edges. There also is a transition mode that allows users to define the geometry that intersects multiple chamfer sets. Users can interact with the dashboard to control the dimensioning scheme needed for the chamfer desired.

Menu Location

Insert > **Chamfer** > **Edge Chamfer**

Sketcher

Enhanced Spline Creation and Modification

Benefits

You can modify splines using the same tools as used for Style features.

You can create and modify freeform splines in Sketcher using tools that are similar to other curve tools in Pro/ENGINEER, for example, those in Style and Restyle.

Menu Location

Sketcher > **Modify**

3

Detailing

Pro/DETAIL includes tools to prepare drawings of parts and assemblies. Highlights of new functionality for this release follow.

Using 2D Drawing Symbols

Benefits

You can capture more information in the 3D model using drawing symbols.

Drawing symbols can be displayed in 3D parts and assemblies. They are placed on annotation planes that rotate with the model. Annotation planes are similar to datum planes but are not visible to the user. This functionality supports the new ASME Y14.41 drawing standard currently under review but not yet released.

Menu Location
Setup > Symbol

Support for Drawing Standards

Benefits

Enhanced support of the major drawing standards such as ASME, ISO, and JIS makes it easier to create the desired drawing.

A number of improvements have been made to enhance support of the major drawing standards. Enhancements include (1) Datum axes can be attached to a cylindrical surface shown as a silhouette; (2) New options have been added for datum targets, including rectangular and circular target shapes; (3) For ISO, the leader line now attaches to the

center of the stack of geometric tolerances instead of the dimension for cases where gtols have been placed in a dimension; (4) For ISO, multiple leaders can be created for geometric tolerance control frames; and (5) Radius dimensions can be displayed according to the JIS standard.

Menu Location

N/A

Enhancements to the Drawing User Interface

Benefits

The number of mouse clicks and mouse movements is reduced. You can initiate all commands from one toolbar.

All commands can be accessed from the main toolbar or from shortcuts menus.

Menu Location

Reference Designators as 3D Notes

Benefits

Locating unique components is faster and easier because you can display component reference designators as 3D notes.

Component reference designators are shown automatically as 3D notes in your ECAD assemblies. You can quickly and easily locate any component by name. To activate this functionality, click **Tools > Environment** and click the **Reference Designators** check box. Customize your toolbar to display the reference designators icon so you can quickly turn on or off reference designators.

Menu Location

Tools > **Environment** > **Display Reference Designators**

Improved BOM Balloon Behavior

Benefits

You can save time with the improved initial placement of balloons. You can display desired balloons for multiple occurrences of the same component.

Improvements to BOM balloons include (1)Improved placement - BOM balloons are placed more intelligently around a view to avoid

overlapping; (2) Automated cleanup - New functionality allows balloons to be positioned relative to view borders; (3) Multiple BOM balloons for the same component - You can show additional balloons for the same component using the same index number; and (4) Highlight balloons from table - You can find corresponding balloons by selecting the row in the table. Select a balloon to highlight the table row.

Menu Location

Edit > **Cleanup** > **BOM Balloons** and **Table** > **BOM Balloons** > **Add Ref Balloon**

Sending a Drawing by E-mail

Benefits

Pro/ENGINEER saves you time by creating a Zip file of the active drawing and its required model files.

Pro/ENGINEER can package and send a drawing file and its required model files to another e‑mail recipient. You have the option of creating a Zip file or sending the files as attachments.

Menu Location
File > **Send To**

Notes on Bend Lines

Benefits

A 3D note is automatically created on any bend, resulting in increased productivity. The note increases your understanding of the part while looking at the Flat Pattern.

The Note on each bend of the side and the angle of bending makes looking at a Flat Pattern much more intuitive. You can use the Note in Drawing mode to increase your understanding of the part, thus increasing your productivity.

Menu Location
N/A

Enable True Type Fonts for Non-European Languages

Benefits

The ability to modify text using True Type fonts provides flexibility when creating drawing documents and can improve the appearance of these documents.

The ability to use industry standard True Type fonts with non-European languages has been added to Pro/ENGINEER. You can modify the text font and use True Type fonts.

Menu Location

Double-click a note and click **Font** from the **Text Style** tab.

Breaking Crosshatch Lines

Benefits

Breaking crosshatch lines around dimension text and notes improves the clarity of the drawing.

The crosshatch lines of a section view can be broken around dimension text and notes to improve clarity. To enable this functionality, click the option **Break Crosshatching** under the **Text Style** tab of the **Properties** dialog box.

Menu Location

Double-click a dimension or note or click the **Break Crosshatching** option under the **Text Style** tab.

Working with the Repeat Region Parameter

Benefits

The repeat region parameter reduces the number of required tables by listing multiple configurations of an assembly in one table using one parts list. The repeat region parameter lists multiple configurations of an assembly in one table using a single parts list. The parameter can specify any other assembly model, including family table instances or generic assemblies. A detail setup option, sort_method_in_region, provides flexibility for sorting regions alphabetically and numerically.

Two new repeat region parameters have been added to locate assembly components on a drawing using the drawing grid. The parameter asm.mbr.pos_loc.all_views lists the locations of the component in all views on the drawing. The parameter asm_mbr. pos_loc.view name lists the locations of the component in a specified view.

Menu Location

Table > Repeat Region > Column Model/Rep

Enhancement to Parametric Drafting

Benefits

The improved snapping to axes makes it easier to create 2D circles and arcs. This is especially helpful when creating full and partial bolt-hole circles. Associative Use Edge and Offset Edge allows easier creation of intelligent 2D entities on drawing views.

While creating a 2D arc or circle, the arc center or arc will snap to axes. The 2D commands Use Edge and Offset Edge can be made associative to the model. This means that the 2D entities created using these tools will update if the model is modified.

Menu Location

Sketch > Edge > Use and Sketch > Edge > Offset

Modifying Angular Dimensions

Benefits

You can directly input angular dimensions in degree-minute-second format when modifying dimensions in a drawing. You can modify angular dimensions directly in degree-minute-second format instead of as decimal input. The **Properties** dialog box for dimensions displays dimension values as degree-minute-second. You can input new values in the chosen format.

Menu Location

Double-click an angular dimension. Select a format from the list under **Angular Dimension Units**.

Creating Hyperlinks in Drawings

Benefits

The support of hyperlinks expands flexibility within the drawing environment and adds more information to the drawing. You can now create hyperlinks on drawing sheets.

Menu Location

Double-click a note. Select **Hyperlink** from the **Text** tab

4

Assembly and Welding

Assembly and welding applications for Pro/ENGINEER Wildfire Help include the following software: Pro/ASSEMBLY, Advanced Assembly Extension (AAX), Assembly Performance Extension, Pro/PROCESS for Assembly, and Pro/Welding. Highlights of new functionality for this release follow.

Pro/ASSEMBLY

Enhanced Reference Control

Benefits
More restrictive exceptions to global reference rules can now be enforced, which allows wider use of the top-down design methodology.

Reference control options are accessible for a single, specific component. If a global reference rule and a component-specific reference-control rule conflict, the more restrictive rule is used. For example, if a specific component has reference control to none and the global reference control is set to any, then any component can be referenced except the component with the reference control. On the other hand, if the global reference control is set to none and a specific component has reference control set to any, then no component can be referenced.

Menu Location
Edit > **Model** > **Reference Control** (for component reference control)

Edit > **Assembly Settings** > **Reference Control** (for global reference control**)**

Component Replacement

Benefits

The simplicity of this new method of replacement benefits both new and existing users. New users can easily explore the capabilities, and existing users can work faster.

The method for replacement falls into two categories: By **Model** and **By Copy**. To replace a component **By Model**, Pro/ENGINEER offers options such as **Family Table**, **Interchange**, **Reference Model**, **Layout**, and **Manual Replacement**. Pro/ENGINEER recognizes the appropriate options. For example, the **Family** Table option is unavailable unless the component has family table instances. If you select **By Copy**, a copy of the component replaces the original. The new, copied component is automatically assembled in place.

Menu Location
Edit > **Replace**

Shrinkwrap Components in Bills of Material

Benefits

Using a shrinkwrap to represent a set of components can dramatically reduce the system requirements for high-quality visualization and regeneration of the model. Shrinkwrap features can now be represented in Bills of Material with their reference components.

Any shrinkwrap feature that is designated as an object will be identified with its parent components. Click **Edit** > **Setup** > **Designate** to open the **Designate** dialog box. Choose the **Objects** tab, and select any shrinkwrap feature. Any referenced objects of the shrinkwrap feature appear in the designation list. Create the Bill of Materials using **Info** > **Bill of Materials**, and check the **Designated Objects** box. The Bill of Materials will list the components represented by the shrinkwrap feature in any assembly that contains that feature.

Menu Location
N/A

Angle-Offset Constraint for Component Assembly

Benefits

You can now create an angle offset constraint between two planar surfaces to specify the rotation of a cylindrical component.

After you create an axis alignment or an insert constraint, the angle-offset constraint is available in the **Component Placement** dialog box.

Menu Location

Insert > **Component** > **Assemble**

Automatic Component Placement

Benefits

After you define assembly constraints and references within a component interface, you can simply drag the component into an assembly.

Similar references are identified as they are brought into an assembly. As a result, assembly constraints are automatically satisfied. The system locates matching references in the assembly for the constraints and references defined in the component.. Multiple interfaces can be created for an individual component. Within each interface, multiple constraints, such as mate and align, are defined along with a corresponding component reference. After these interfaces are defined, they can be used for assembly. Any one interface set, which represents multiple constraints, can be selected for assembly. Automatic placement also allows multiple occurrences of one component to be placed simultaneously.

Menu Location

Edit > **Setup** > **Component Interfaces**

Tools > **Assembly Settings** > **Component Placement**

User-Defined Feature Enhancement - Subassembly Inclusion

Benefits

UDFs can now be used on a larger scale for automation and standards promotion.

When creating a UDF, you can select subassembly components created in the assembly along with the created part components and features.

Menu Location

Tools > **UDF Library**

Working with Flexible Models

Benefits

Flexible components allow the same component to be shown in different geometric representations and still report one item on the Bill of Materials.

You can make a component flexible after you place it by selecting a component or subassembly and clicking **Edit** > **Make Flexible**. You can define the variation in geometric states manually, or it can be derived as a dimensional value from an analysis result, such as distance. After the value is determined, there are four options to define the flexibility.

You can also predefine flexibility in a component by clicking **Edit** > **Setup** > **Flexibility**. You can use this flexibility definition each time the component is added to an assembly.

Menu Location
Edit > **Make Flexible**

Bulk Item Templates and Measure Parameters

Benefits

Bulk items have been enhanced for all associative measurement parameters and encourage the reuse of data by providing Bulk Template parts.

You can create Bulk Template parts by clicking **File** >**New** >**Bulk** to open the **Bulk Item Definition** dialog box. From this dialog box, you can create bulk item parameters, relations, and family tables of the bulk part. Bulk parameters have been enhanced to include measure parameters. For example, you can create an Area Measure type parameter called PAINT. When this bulk item is added to an assembly, you will be prompted to select a surface as a reference for the PAINT parameter. The surface area will be calculated and used as the value of PAINT. Other new parameter types are Length Measure, Distance Measure, Diameter Measure, Angle Measure, and Parameter Sum. The Parameter Sum type will add the values of all parameters of the same name.

Menu Location
File > **New** > **Bulk**

Advanced Assembly Extension (AAX)

Working with Inheritance Features

Benefits

The expansion of the inheritance feature furthers the ability to create inheritance features and to manipulate derivative models for manufacturing purposes.

During the creation of an inheritance feature, you can choose options to inherit individual parameters and geometric tolerances. You can also define new values for all inherited items. The derived object (target part of the inheritance feature) can be retrieved and modified without retrieving the original model. Inheritance features can be created within an assembly in the same manner that copy geometry features can be created. Inheritance features can be used to remove material from a part. While replacing a component, you can replace a reference model with its derived object or vice versa. This capability uses the reference-forwarding capabilities to automate the replacement. You can create inheritance features in sheet metal components.

Menu Location
Insert > Data Sharing > Inheritance Feature

Assembly Performance Extension

Symbolic Simplified Representation

Benefits

Symbolic simplified representation, or lightweight simplified representation, dramatically reduces the system requirements for high-quality visualization and quick regeneration times.

By default, symbolic simplified representations replace components with a datum point. This representation can be customized to any 3D symbol, including the orientation of the annotation plane.

Menu Location
View > Representation > Symbolic Only

External Simplified Representation

Benefits

External representation dramatically reduces the system requirements for high-quality visualization and quick regeneration times by more widely proliferating its use. External simplified representations are created without modifying the master assembly.

The **Create Assembly** dialog box provides all of the new and existing simplified representation options. The creation process generates a new assembly file with a dependency on the original assembly. You can also create External simplified representations by copying an internal simplified representation in the **View Manager** dialog box.

Menu Location

File > **New** > **Assembly** > **External Simplified Representation**

View Manager

Benefits

The View Manager provides the capability for managing and using various states.

The basic workflow for creating, editing, and switching between simplified representations, exploded states, display states, and orientation states has changed. In Pro/ENGINEER Wildfire, the model is constantly in a working state. Components can be switched to hidden line or excluded, or a component can be moved to represent an exploded state without using any dialog boxes. Changes are then stored using the View Manager.

Menu Location
View > **View Manager**

Pro/Welding

Creating Lightweight Edge Preparation Features in Welding

Benefits

You can quickly create edge preparation features with minimal impact on assembly performance.

For welds requiring edge preparation, you can create a lightweight edge preparation that captures the information about the edge preparation but displays it in the model similar to a datum curve.

Menu Location

From the **Weld Creation** dialog box, click **Environment** and specify **Light** for **Edge prep geom** type.

Creating Automatic Weld Notches

Benefits

You can save time by automatically creating weld notches to provide clearances for welds.

You can create weld notches automatically when defining a weld. Weld notch features are cuts that create clearance in intersecting geometry for the weld feature.

Menu Location

From the **Weld Creation** dialog box, click **Notch**.

Working with Edge-Surface Fillet Welds

Benefits

Welding supports use cases where edge-to-surface welds are required.

Welding now supports additional use cases where edge-to-surface welds are required. A new weld type has been added in addition to the existing fillet types edge-edge and surf-surf.

Menu Location

From the **Weld Creation** dialog box, select the fillet weld type and **Edge** > **Surf** from the **REF OPTIONS** menu

5

Pro/ENGINEER Interface

Pro/ENGINEER Interface for Pro/ENGINEER Wildfire provides direct data exchange and industry standard translators. Highlights of new functionality for this release follow.

ProductView Express 2.0

Benefits

ProductView Express is a Web browser plug-in that lets you view Pro/ENGINEER models outside of Pro/ENGINEER.

This version of ProductView Express is shipped with Pro/ENGINEER Wildfire and is installed automatically. It is used for previewing Pro/ENGINEER models from the Pro/ENGINEER embedded browser or from outside of Pro/ENGINEER.Performance is improved and spin, pan, and zoom controls are now similar to those in Pro/ENGINEER. You can now use ProductView Express to view Pro/ENGINEER section (.sec) and manufacturing (.mfg) files.ProductView Express is also available as a free download from www.ptc.com.

Menu Location

N/A

6

Electrical Design

Electrical design in Pro/ENGINEER Wildfire Help includes the following applications: Pro/DIAGRAM, Pro/CABLING, Pro/HARNESS, and Pro/ECAD. Highlights of new functionality for this release follow.

Harness Family Tables

Benefits

The creation of family tables of harness parts accommodates multiple configurations of harnesses or similar harnesses of different lengths.

Harness wiring is an addition to the Pro/ENGINEER family table functionality. With Pro/ENGINEER Wildfire, you can create family tables of assemblies and vary the configuration of the harness parts in them.

Menu Location
Family tab

User-Defined Terminator Parameters

Benefits

You can report terminator parameters like manufacturer, material, and internal part numbers, making your BOMs more complete and accurate.

Terminator files are similar to spool information files. When the terminator table functionality is used, the terminator is applied to the wire ends as specified. The terminator then has all of the parameters stored in the new terminator files. This terminator information can be

reported in wire lists, BOMs, and connector loading tables, as necessary.

Menu Location

Terminators

Improved Bundle Alignment

Benefits

A 3D harness looks more realistic. Assembly drawings for downstream documentation applications can be created more easily with less modification.

When you create a harness in Pro/ENGINEER Wildfire, the enhanced appearance of bundle junctions affects all your harnesses. Harnesses created in previous versions may be updated using the Align Bundle command on the Location menu.

Menu Location

Location > **Bundle Alignment**

Improved Subconnector Functionality

Benefits

Managing subconnectors in a top-level assembly eliminates the need to create assembly coordinate systems as ports to which you route your wires. You can use the entry ports stored in the parts.

With the improved Subconnector functionality, you can create complex, top-level assemblies with their own parameters. You can also assign parameters to the individual parts that make up the assemblies. The BOM will accurately reflect the top-level assembly instead of each individual part; which is perfect for prefabricated assemblies. When a harnesses is flattened, the top-level assembly is automatically assembled to the shortest attached wire segment. You can also show the parts individually attached.

Menu Location

Logical Reference

Modify Network Tangency

Benefits

The ability to control network tangency at network branches makes auto-routing wires easier and faster. This option reduces the amount of time it takes to create a functional network.

You can control the tangency of a network branch from the user interface with no workarounds required.

Menu Location
Network Ops > **Tangency Operations**

Using the Copy Network Functionality

Benefits

The Copy Network functionality significantly reduces the amount of work required for routing through cable trays or sections of geometry where the harness route needs to be spread out.

You can copy sections of the network at specified intervals using Copy Network. The offset can be modified after creation as necessary.

Menu Location
Network Ops > **Route** > **Offset**

Allow References to ECAD Cosmetic Sketches

Benefits

Using the ECAD cosmetic area edges as a reference, you can sketch a new ECAD area next to an existing ECAD area.

As you create the ECAD areas on a board part, you can reference the edge of another area as needed.

Menu Location
Sketcher

Missing Component Configuration Option

Benefits

The ability to set the default placement option for missing components to "keep missing" or "delete missing" reduces the amount of time it takes to confirm changes to board components.

To investigate missing components, add the configuration option `ECAD_MISSING_COMPONENT_STATUS` to your config.pro file. Set the values as "keep missing" or "delete missing." After the new option is activated, missing components will have the status you have preset in the `config.pro` file.

Menu Location

`config.pro` file

7

Manufacturing

Manufacturing applications for Pro/ENGINEER Wildfire Help include Pro/NC, Expert Machinist, Pro/PROCESS for MFG, Pro/CMM, G-Post, and FIL. Highlights of new functionality for this release follow.

Automatic Mill Window from Silhouette of Reference Part

Benefits

Creating a mill window by silhouette minimizes the number of steps required to define the window curve. It also minimizes references, improves regeneration time, and facilitates automation.

A mill window is one of the most efficient ways to define an area to be machined in Pro/NC. Sketching the references for the window can be time-consuming and may create many references. Pro/ENGINEER Wildfire speeds up the process by enabling the user to create a mill window based on the silhouette of a part (user selected, but from reference model by default) viewed along the z-axis of the mill window coordinate system.

You have the option to keep or remove inside loops created by through cuts (or holes).

All Boundary rules (ON, TO, PAST, and Offset) apply to windows created by silhouette.

Menu Location

Mfg Setup > Mfg Geometry > **Mill Window** > **Create Window**

Using NC Job Manager for Remote Tool Path Processing

Benefits

You can increase productivity by not having to wait for the tool path computation to complete. You can also take advantage of powerful servers and processing time at night.

As quality finishing standards increase, so does the time it takes to generate the tool paths for cutting materials. The Pro/ENGINEER NC Job Manager allows you to select the time and the workstation to be used for tool path processing. Computation of tool path sequences does not prevent continued use of Pro/ENGINEER by blocking processing. You can optionally perform computations out-of-process, so that work on subsequent tool path sequences or parts can continue uninterrupted, either remotely or on the local machine.

Menu Location

Submit Path (during NC Sequence creation)

Transparent Workpiece Display in Manufacturing Mode

Benefits

It is now easier to see the details of the components to be machined

For assembly machining, the new Pro/ENGINEER Wildfire graphics library allows you to display, by default, your workpiece in transparent mode. The transparency is controlled by the config.pro option mfg_workpiece_transparency, with a default value of .5 for 50% transparency.

Transparent mode applies only to models created in Pro/ENGINEER Wildfire.

Menu Location

N/A

Manufacturing File Preview

Benefits

You can select the models that you want to work with by previewing them before you open Pro/ENGINEER.

In Pro/ENGINEER Wildfire, you can preview your manufacturing models using **File** > **Open** > **Preview**.

Manufacturing file preview is also available when you select files in the Pro/ENGINEER browser.

Menu Location
File > **Open** > **Preview**

Import and Export of Manufacturing Process in XML Format

Benefits

Exporting Pro/ENGINEER Wildfire manufacturing processes in XML format provides access to all the manufacturing parameters used in a specific tool path by using a Web-based standard format.

Pro/ENGINEER Wildfire exports and imports manufacturing processes in XML format. You can use the XML files for multiple tasks, such using style sheets to create custom process documentation.

During the import of the XML files, manufacturing references such as coordinate systems, retract planes, and manufacturing geometry are automatically assigned, based on feature type and the name stored in the XML file. For instance, if the initial tool path was defined using a mill window named Outline, and the new manufacturing model includes a mill window with the same name, you won't be prompted for the reference. This process is similar to a Manufacturing User-Defined Feature (MUDF) operation, but with automatic assignment of references based on name and type.

Menu Location
N/A

Roughing and Re-Roughing Tool Paths

Benefits

The combination of techniques from CADDS 5 (slice calculation) and Pro/NC increases tool path¿processing performance and reliability in mold machining.

Pro/ENGINEER Wildfire includes a new NC sequence for roughing and re-roughing with the following features (1) An internal tessellated model to improve processing performance; (2) Complete support for native and non-native data (IGES); (3) Inclusion of all Pro/NC high-speed machining scan typesconstant load, follow hard wall, and so on; (4) Inside/outside tolerances to improve performance and machining accuracy; (5) Combination of cutting strategies for both open and close areas; and (6) Continuous analysis of the model.

Another unique feature, extra slice calculation, allows you to calculate an intermediate re-roughing process. When the roughing tool completes its normal rough-milling routine, it goes back and reduces height and width of the steps it left behind. Therefore, the material is optimally prepared for the next tool, and the roughing tool has removed as much material as possible.

Menu Location

Machining > **NC Sequence** > **Roughing** or **Re-roughing**

Automatic Solid Tool Export to VERICUT for Pro/ENGINEER

Benefits

You no longer need to redefine solid tools in VERICUT for Pro/ENGINEER to get an accurate Pro/NC simulation.

Solid tool assemblies used in Pro/ENGINEER Wildfire for the creation of tool paths are exported directly to the VERICUT tool database for Pro/NC simulation. By default, the complete tool assembly will be exported as a single tool component.The component parameter VERICUT_TYPE allows you to control individual part components of the tool assembly to be exported as a TOOL or HOLDER.

For turning-tool assembly, the tool outline is exported. For milling-tool assembly the revolution envelope of each tool component is exportedThe interface between Pro/ENGINEER and VERICUT for Pro/ENGINEER has been modified to handle multiple fixtures, blanked layers, and simplified reps.

What you see in the Pro/ENGINEER Wildfire window is what you get in the VERICUT for Pro/ENGINEER window.

Menu Location

CL Data > **NC Check**

Leading Edge Machining in Trajectory

Benefits

Machining with the leading edge maximizes the tool¿s efficiency because the center of the tool cannot cut. The LEADING_EDGE_MACHINING parameter guarantees this cutting condition..

The LEADING_EDGE_MACHINING parameter provides a means to control flat and corner radius cutters so that they always cut with the

leading edge of the tool in trajectory milling. This is particularly important for areas of high curvature.

Menu Location

N/A

Tool Axis Definition by Location

Benefits

Tool axis definition by location provides more flexibility in the definition of a tool-axis control point directly on the geometry of the reference part.

Tool axis definition by location for trajectory milling allows the user to place a datum axis at a physical location on the part instead of on a cut motion. The tool axis is then applied to the cutter path based on its proximity to the cutter path. Interpolation is based on proximity of the position on the cutter path to one or more datum axes.

Menu Location
Trajectory > **Customize** > **Axis Def**

Integration of VERICUT 5.2 in Pro/ENGINEER Wildfire

Benefits

The integration of VERICUT for Pro/ENGINEER makes available the latest functionality of VERICUT from CGTECH for PTC customers.

VERICUT for Pro/ENGINEER offers the following features
(1) Constant gauge allowing you to dynamically detect potential tool path gauges; (2) Tool path review enables you to easily replay all or a section of a tool path over the cut stock; (3) Curve fitting improves finishes on surfaces and reduces machining cycle time; (4) DXF profile import in tool manager; (5) Zoom window provides a zoom view in a separate window; (6) Threading simulation; (7) Boring bar and facing head support; and (8) Improved simulation performance.

Menu Location
CL Data > **NC Check**

Optimized Retract and Connection in Trajectory Milling

Benefits

This time-saving functionality minimizes or removes unnecessary retract motions in multipass trajectories. You have complete control on the connection move between slices.

When creating a trajectory sequence in Pro/NC using multiple slices (start height and step depth), the tool retracts, moves back to the start location, and plunges to the depth of the next slice. In Pro/ENGINEER Wildfire, you can now control the connection motions with the CONNECTION_TYPE parameter using the following options: RETRACT - Functionality as currently implemented in Pro/ENGINEER 2001; Z LAST - Tool moves from the end point to start point in the xy-plane and then plunges in the z-plane; Z FIRST - Tool moves from the end point to the new slice depth with a plunge in the z-plane followed by a positioning move in the xy-plane to the start point; SIMULTANEOUS - All three axes move simultaneously from the end point to the start point of the next slice.

Menu Location

Machining > **NC Sequence** > **Trajectory** > **Parameters**

Selection of Surfaces for Machining

Benefits

The new surface selection mechanism simplifies the manipulation of external data for direct machining.

When machining non-solid surface data that may have been imported to Pro/ENGINEER from an external source, the machining algorithms need to know which side of each surface should be machined.

For each quilt an arrow is displayed, and you are required to select whether it is the displayed side or the flip side that should be machined.

In Pro/ENGINEER Wildfire, you will be prompted to select the surface side for a seed surface, and Pro/ENGINEER will automatically align the others surfaces based on the initial selection.

Menu Location

Any surface selection in an NC Sequence

Mill Window User-Definable Offset

Benefits

The Offset option for mill windows minimizes references and number of steps, simplifies sketching (if used to define the window), improves regeneration time, and facilitates automation.

When you enter a supplemental value for the Offset option, the value is applied to the boundary of the mill window. The offset is applied to the default window curve, regardless of the tool condition at the boundary (ON, TO, PAST). You can set the direction of the curve offset in the sketching plane using a flip arrow. After the offset value is set, the tool will offset the boundary by the tool side (ON, TO, PAST) condition and the specified value. The default value of the offset is zero.

Menu Location

Mfg Setup > **Mfg Geometry** > **Mill Window** > **Create Window**

Pro/NC-GPOST in Pro/ENGINEER Wildfire

Benefits

The integration of Pro/NC-GPOST into Pro/ENGINEER Wildfire simplifies post-processor development and reduces the need of FIL macro development. The top ten FIL macros have been integrated in the Pro/NC-GPOST questionnaire (Optfile). The latest version of GPOST from Intercim Austin NC has been integrated into Pro/ENGINEER Wildfire.

Pro/NC-GPOST brings direct support for five-axis milling kinematics, including complete "Nutator axis" support, new NURBS interpolation support for Bezier format (MAHO), extended registers definition, integrated calculator, HTML-based documentation, user-definable tape file extension, support for G99/G98, improved **Open** dialog box, and improved FIL macro functions. Many of these enhancements are based on the top ten FIL macro requests received by PTC Technical Support.

The Pro/NC-GPOST page on the PTC Technical Support Web site includes a large database of post-processors for download and tutorials for Pro/NC-GPOST.

Menu Location

Applications > **NC Post** > **processors**

8

Model Analysis

Model analysis applications for Pro/ENGINEER Wildfire Help include Behavioral Modeling Extension (BMX), Pro/ENGINEER ModelCHECK Extension, and Pro/PROGRAM. Highlights of new functionality for this release follow.

Behavioral Modeling Extension (BMX)

Behavioral Modeling and Pro/MECHANICA Integration

Benefits

The integration of analysis types enables Behavioral Modeling to improve designs based on a broader range of problems and addresses multidisciplinary problems.

After you set up an analysis in Pro/MECHANICA, you can turn the analysis into an Analysis feature. During the definition of the Analysis feature, you can tag the list of quantified results from the Mechanica analysis as a computed parameter. Use these computed parameters for relations and improvement activities such as Global Optimization and Multi-Objective Design Studies.

Menu Location
Analysis > **Mechanica Analysis**

Using the Graphing Tool

Benefits

You have one tool for all graphing in Pro/ENGINEER. Exact formatting control allows you to customize the graphs for your needs.

The new graphing tool provides consistent formatting control for results from Pro/MECHANICA, Mechanism, and Behavioral Modeling. Consistent formatting includes representations of the data, background colors, axis labels, tick marks, colors and many more capabilities.

Menu Location

N/A

New Global Optimization

Benefits

The common issue with gradient-based optimizations of settling into a local minimum or maximum goes away. Global Optimization has a higher chance of finding the global minimum or maximum.

To set a Global Optimization, you first define the number of experiments to populate the design space. After the number of experiments has been filled, Behavioral Modeling uses a Paretto capability to identify the best candidates of the populated space. These best candidates are then the starting points of the gradient-based optimization.

Menu Location

Analysis > **Feasibility/Optimization** > **Options** > **Preferences**

Integration of Behavioral Modeling and Mechanism Analysis

Benefits

The integration of analysis types enables Behavioral Modeling to improve designs based on a broader range of problems and also addresses multidisciplinary problems.

You can use Pro/ENGINEER parameters to define Mechanism entities. The results of mechanism analyses can be used in improvement activities, such as optimizations and multi-objective design studies.

Menu Location

Analysis > **Motion Analysis**

Pro/ENGINEER ModelCHECK Extension

ModelCHECK - Check for Circular References

Benefits
Circular References should generally be removed from assemblies.

Circular references are created when one feature (A) references a given feature (B) and B in turn references A. When this happens, Pro/ENGINEER does not know how to treat each feature and the results are unpredictable.

Menu Location
CIRCULAR_REFS

ModelCHECK - Duplicate Model Information Storage in Pro/INTRALINK

Benefits
It is now easier to administer Shape Indexing and to find duplicate models.

Menu Location
N/A

ModelCHECK - Check for Items Outside the Drawing View Boundary

Benefits
With this check, you can detect drawings that have items outside of the boundaries of the drawing.

Menu Location
This is an enhancement to the existing check BOUND_INFO.

Improved ModelCHECK User Interface

Benefits
Reports are now easier to read and provide more information. You can highlight and fix more things from the report.

The ModelCHECK report appears in the Pro/ENGINEER Web browser. The full, interactive report appears immediately. All reports, including printable and batch reports, have a consistent look and feel. You can filter what types of checks are displayed by selecting check boxes for **E**

(errors), **W** (warnings), **I** (info), and checks that passed. The `config_init.mc` **options** NETSCAPE **and** BROWSER **have been replaced with** USE_EMBEDDED_BROWSER**. The acceptable values for this option are** Y **(yes) and** N **(no). Setting the option to** Y **will use the Pro/E embedded browser for displaying the report. Setting the option to** N **will use the default browser designated during PTCSetup for displaying the report.**

Users can filter what types of checks are displayed by selecting check boxes for **E** (errors), **W** (warnings), **I** (info) and checks that passed.

Menu Location
FAMILY_INFO **configuration option**

ModelCHECK - Check If the Family Table Has Been Verified

Benefits

It is important that models with family tables are verified before the model is released to a Product Data Management system.

The report displays the status for nested family tables in addition to unregenerated instances or failed regenerations.

Menu Location
FAMILY_INFO configuration option

ModelCHECK - Check for External Dependencies

Benefits
Models with external dependencies may behave unexpectedly if they are used in multiple assemblies

Menu Location

DEPENDENT_FEATURE

9

Mold Design and Casting

The Pro/ENGINEER Tool Design Option for Pro/ENGINEER Wildfire Help includes the following applications: Pro/MOLDESIGN and Pro/CASTING. Highlights of new functionality for this release follow.

Automated Mold/Casting Accuracy Setting

Benefits

You no longer must set the absolute accuracy of the Mold/Cast cavity model. Time is saved and splitting failures are reduced later in the core/cavity creation due to inaccurate geometry.

After placing the reference part, you are prompted if the reference part and the mold/cast cavity (.mfg) accuracies do not match. You can choose whether or not you want to change the mold accuracy.

Menu Location

N/A

Automated Undercut Check and Insert Geometry Creation

Benefits

When undercut areas in the mold are identified, slider geometry is automatically created. This ensures that no undercuts exist and die lock conditions are avoided.

Based on the pull direction indicated for the mold, undercuts are detected and identified. The geometry that is necessary to eliminate the undercut problem is created automatically. This geometry can be easily

modified, extended, and trimmed to create appropriate insert geometry, such as a sliding insert.

Menu Location

N/A

New Tangent Draft User Interface

Benefits

The streamlined user interface for Tangent Draft simplifies the creation of draft geometry for cast and molded parts.

Graphics in the user interface make selecting and creating the correct type of tangent draft more intuitive.

Menu Location

N/A

New Type Tangent Draft Feature

Benefits

The process for creating draft for cast and molded parts is simplified because you can now remove material for draft creation.

The surface is automatically drafted and a radius is created that is tangent to the adjacent surface.

Menu Location

N/A

10

Piping

Spec-driven and non spec-driven piping are optional applications for Pro/ENGINEER Wildfire Help. Highlights of new functionality for this release follow.

Pro/ENGINEER ISOGEN Enabled Piping Design (Spec-Driven)

Pipe-Bend Information on ISOGEN Spool Drawings

Benefits

With this enhancement you can generate spool drawings instantaneously from ISOGEN. These spool drawings will include all of the pipe-bend information necessary for production.

Critical pipe-bend information is automatically reported to the .pcf file that Piping Design produces to drive ISOGEN drawing creation. You can create drawing formats that access and report this bend information. The resultant drawings are instantaneously created with the correct bend tables and information.

Menu Location

N/A

Pro/ENGINEER Piping Design (Non Spec-Driven)

Assemble Fittings from Either End of a Pipe Segment

Benefits

You can define which end of a pipeline segment the fitting references. This choice allows more flexibility during fitting insertion so you can add the necessary constraints.

After you insert a fitting into a pipeline segment, you can toggle the end of the segment that is referenced. For example, you insert a check valve into a pipeline segment. Then you can select to reference the insertion of the fitting to the beginning of the pipe segment or the end of the pipe segment. You enter the length constraint as necessary.

Menu Location
Fitting > Insert

Pipeline Assembly Insertion

Benefits

Productivity increases on pipeline assemblies with many identical pipeline runs. You can insert entire pipelines or sections of pipelines from stored models or from within the assembly.

Pipeline assembly insertion allows you to reuse entire pipelines. You can select the entire pipeline or a section of a pipeline, specify the start and end segments, and redefine the orientation and angle.

Menu Location
Route > Insert

Route Pipelines from the Free Port on Valid Fittings

Benefits

Pipeline segments that are routed from the free port of a valid fitting are automatically "joined" to the adjacent ports. This ensures proper propagation of the pipeline segments.

Routing a pipeline from the free port of a tee fitting will automatically create an intersection of the centerlines. It is as if you had routed a branch off of the main pipeline. If the fitting is replaced, the pipe segment that was attached to it will propagate to the new size of the fitting. If the pipeline segment is modified, such as changing the angle, the fitting will update accordingly.

Improved Extend Feature

Benefits

The consolidation of the routing tools in piping provides different routing operations from one dialog box, reducing the time it takes to route pipeline assemblies.

Routing is easy when you use one dialog box and can select to route by coordinate systems or coordinate system axes and can also define references. The EZ router and **Extend** dialog box in Piping Design have been combined into a single, user-friendly tool.

Menu Location
Route > **Extend**

Consolidated Routing Tool

Benefits

The **Extend** dialog box contains all of the routing tools you need in one place. This consolidation reduces the number of selections necessary to route pipelines and speeds up routing.

The EZRouter and the extend features have been combined to enable you to quickly and easily move from one extend function to another without having to exit and enter menus.

Menu Location
Route > **Extend**

Enhanced Pipe Routing

Benefits

You can create pipelines that have a parametric relationship to other pipes or planes.

This functionality allows you to parametrically route pipelines with respect to the top, bottom, left, or right of other pipelines. Pipelines can be routed adjacent to other pipes or you can specify an offset distance. You can also reference planes.

Menu Location

Route > **Set Start** > **Define Pipe Edge**

Working with Schematic-Driven Piping

Benefits

You can reuse data from your system schematic to recreate the 3D pipeline assembly and reduce the number of errors caused by misinterpretation.

To use the schematic data from Routed Systems Designer, you must set your `config.pro` **option** `piping_schematic_driven` **to** `yes` **and** create your pipeline using the pipeline information contained in the XML file.

Menu Location

Pipeline > **Create**

Merging Pipelines

Benefits

Multiple users can individually model segments of a single pipeline and then merge them when they are done. This makes top-down design much easier.

You can create two sections of a pipeline simultaneously and merge them into a single pipeline. This type of design makes it easy for companies to split 3D pipeline design work and combine the efforts at the end.

Menu Location

Pipeline > **Merge**

Automatically Assigning Nuts and Bolts

Benefits

Nuts and bolts are automatically called out in the BOM.

You can define nuts and bolts as bulk items and automatically assign them based on a code specified in the specification database.

Menu Location

Specification database

User-Defined, Combined Size Codes

Benefits

With the `combined_size_code` parameter, you can define formats to accurately represent a company's naming conventions.

You create a Defined Size Code table in the `pipeod` folder in the specification database that sets the `combined_size_code` parameter. This parameter is based on existing size parameters in the fitting specification. You can then use the `combined_size_code` parameter to define the stock number format or material classification code system (MCCS) number.

Menu Location

Specification database in the `pipeod` folder

Specification Break for Portions of Pipelines

Benefits

Because you can change a portion of pipeline specification, you can create a pipeline assembly from a single specification and easily modify selected sections.

You select the section of the pipeline that needs to be driven by a different specification and change the current specification. All of the fittings along the selected section will automatically be updated to the new specification.

Menu Location
Pipeline > **Spec Break**

11
Rendering

Pro/PHOTORENDER provides advanced rendering tools in
Pro/ENGINEER Wildfire. Highlights of new functionality for this
release follow.

Advanced Rendering Extension

Benefits

Photorendering, a powerful tool, aids in effectively communicating
design intent and helps companies market products before they are
manufactured. Companies can gather critical marketing and sales
information without having to invest money in production.

PhotoLux images offer the highest degree of photorealism for a model.
Combine real-world lighting, accurate reflections and shadows, and the
fastest render times, and you have the leading rendering software in
the industry. Advanced features include volumetric light scatter, lens
flare, depth of field, fog, and more.

Menu Location

View > **Model Setup** > **Render Control** > **Modify** rendering
configuration options

Real-Time Rendering

Benefits

While visualizing and manipulating the model, you can have accurate
wall reflections and shadow casting onto room walls.

Real-time rendering provides valuable graphical feedback and visualization in a dynamically rendered environment. Room reflections and shadows add to the realism of the Real‑Time Rendering environment.

Menu Location

View > **Display Settings** > **Model Display**. In the **Model Display** dialog box, click **Display** tab > **Realtime Rendering**.

12

Sheetmetal

Sheetmetal applications for Pro/ENGINEER Wildfire include Pro/SHEETMETAL and Pro/NC-SHEETMETAL. Highlights of new functionality for this release follow.

Pro/SHEETMETAL

Form Feature Placement

Benefits

Form feature placement is the easiest and fastest way to place the Form features into sheet metal parts.

Form feature placement has been enhanced and uses the same user interface as Assembly mode uses. The result is a clear and easy way to place the Form features.

Menu Location

N/A

Inheritance Feature for Sheetmetal Parts

Benefits

When building one-way associative parts related to a master model, parts using an inherited feature do not modify the original design. Original design changes are in the inherited parts.

The Inheritance feature allows for the multiple handling of materials, toolings, and part shapes while you focus on the final model. For example, you may tool a sheet metal part with progressive die,

however, while you are waiting for the tooling to complete, you may create the laser cut or punched parts with different materials. The Inheritance feature helps you to focus on the final part. Any modification to this final part is reflected in inherited parts. Modification to inherited parts is not reflected in the final part.

Menu Location

Insert > **Shared Data** > **Inheritance from Other Model**

Hem Feature Creation

Benefits

You can create a full or partial hem on an edge with one feature. Because you can keep the initial wall height, the number of required actions is reduced.

The functionality in the Hem feature lets you keep or extend the initial wall height with the hem size. Because you can define the Hem feature on a partial edge, it includes a relief handling at each side of the hem in the same way, or for each side, separately. Required user actions are decreased, there are fewer features, and graphical feedback is instantaneous.

Menu Location

Hem feature

Convert Extensions and Cut Backs

Benefits

The availability of drag handles and the consolidation of create extend and create cut walls speeds up your work.

While performing a conversion, you now have more options regarding Edge Rip interaction. In addition to the Open and Overlap options, the Blind option is now available.Two drag handles are available on the part so you can directly drag the wall lengths. Right-click to see the most used values, **Thickness extension dimensions** and **2*Thickness extension dimensions**.

Menu Location

Convert feature

Pro/ENGINEER Wildfire Preproduction Release Notes

Datum Splitting

Benefits

In 3D displays you see additional, smaller datums. In 2D drawings, you eliminate unnecessary rework.

Through All cuts now generate a different axis at each wall intersection. When you create a Flat Pattern, any hole on the part has a center axis that you can use. This would be useful, for example, for a production drawing.

Menu Location

N/A

Pro/NC-SHEETMETAL

Parameters Tool

Benefits

The Parameter tool allows you to manage parameters.

You can create, edit, and designate parameters from this dialog box. You can also get information about read only parameters. Certain parameters, such as those created in Pro/INTRALINK or other PDM systems, or driven by relations or Pro/PROGRAM, cannot be modified from the **Parameter** dialog box. These parameters will be shown as locked, and will provide the source of the locked status. Parameters can also have a status of Unlocked (can be modified anywhere) or Restricted (can only be modified from the **Parameter** dialog box).

Menu Location

Tools > **Relations**

13

Surfacing

For surfacing tasks in Pro/ENGINEER Wildfire, the following
applications are available in Help: Pro/SURFACE, Style (ISDX) ,
Restyle, Facet Modeling, Pro/SCANTOOLS, and Warp. Highlights of
new functionality for this release follow.

Interactive Reverse Engineering Extension

Benefits

The transformation of scan data into a facet model is highly automated.
You can modify and customize the resultant with simple and complex
geometry that is not dependent on any data.

Cutting-edge Restyle tools refine point cloud and polygonal data for
optimum results. These tools reduce noise or the total points to speed
up processing without sacrificing detail. You can delete or refine
triangles to eliminate any error and not compromise surface integrity
or detail. The transformed data captures the design intent and
maintains critical geometry for use later. On a facet model, you can
quick-pick simple surfaces or complex surfaces. You can project a
surface to fit facet data or to fit boundary surfaces of curves that you
have sketched. Pro/ENGINEER provides a full set of tools to analyze
the surface for any deviation from the facet model. You can edit surface
to change such properties as surface type (NURBS/BEZIER), U/V
parameters, and so forth. You can manipulate the surface in a free-form
or numerical manner to fine-tune the model. The result is a modifiable
and customizable surface representation of a product that fits right the
first time.

Menu Location

Select > **Insert** > **Restyle**

Primitive Shape Recognition for Surfaces

Benefits

Primitive shape recognition highly automates the creation of Pro/ENGINEER surfaces on facet data for simple geometry such as cylinders, cones, flats, revolutions, and extrusions.

Shape recognition provides you with an extremely efficient way to recognize and create primitive geometry from a facet model. Current functionality provides for the recognition of cylinders, cones, flats, revolutions, and extrusions. Shape recognition allows the engineer or designer the unique opportunity to focus more time on complex areas of the model.

Menu Location
Insert > **Restyle**

Parametric Texture Mapping

Benefits

With parametric texture mapping, you can achieve a uniform look in color and appearance. Previous placement options did not provide this result.

Appearances, or textures, assume a normal placement at various areas of the surface to which they are being applied. Thereby, you can place a texture in the optimal position on compound or complex curved surfaces.

Menu Location
View > **Color and Appearance**

Triangular Surface Creation

Benefits

Without leaving the Style feature, you can create a boundary surface when only three bounding curves are available.

Boundary surfaces containing only three bounding curves can now be created within a Style feature. Triangular surfaces may have internal curves. Internal curves that are perpendicular to the natural edge must pass through the vertex of the surface patch. You can reposition the vertex manually via the dashboard.

Pro/ENGINEER Wildfire Preproduction Release Notes

Menu Location

Styling > **Surface**

Curve and Surface Auto-Regeneration

Benefits

With auto‑regeneration, you can perform unprecedented visualization and design iterations. Exploration of design possibilities takes a fraction of the time required by conventional methods.

Curves and surfaces will now automatically regenerate in real time during curve modification. Child curves and surfaces also will update in real time as their parents are modified. With auto‑regeneration, an engineer or designer can experiment with literally thousands of design possibilities in seconds.

Menu Location

Styling > **Curve Edit**

Style Super Feature and Internal Datum Plane Creation

Benefits

Internal datum creation provides increased workflow while working within a Style feature.

Internal Datum Plane Creation functionality provides additional references for curve and surface creation and eliminates the need to leave the Style feature to accomplish this task. Internal Style Datum Plane creation is consistent with traditional datum planes in Pro/ENGINEER. Internal datums are displayed only while in the Style feature and are not visible when the feature is completed

Menu Location

Styling > **Internal Plane**

Selectively Exposed Style Parameters

Benefits

The ability to expose style parameters provides flexibility to use the information in BMX analysis, Relations, Family Tables, and so forth. Increased workflow and productivity result.

Style parameters can be selectively exposed outside of the Style feature. When you edit the Style feature, the selected parameters

become visible in a graphics window for modification or for use in other functions within Pro/ENGINEER. Parameters are typically valuable in the context of Relations or Family Tables. With style parameters exposed, you are relieved from having to redefine the style feature when a change is necessary.

Menu Location

On the Style dashboard, click the check box next to the desired parameter.

Multiple Curve Edits

Benefits

The character within a network of curves is preserved, while you manipulate multiple curves. In a single operation, you can assign a single value for multiple control points.

You can edit multiple curves by clicking the Edit Curves icon, then holding down CTRL, and selecting the desired curve entities. You can then select one or more control points and make the necessary modifications.

Menu Location

Click the Edit Curves icon. Hold down CTRL while selecting the desired curve entities.

Style Feature and Surface Trimming

Benefits

You do not have to exit Style to trim a surface and create desired geometry. Workflow and the Model Tree are simplified.

You can trim surfaces within the Style feature in the same manner as within the standard Pro/ENGINEER environment.

Menu Location

Trim Selected Quilts icon or **Styling** > **Trim**

Global Modeling Feature (Warp)

Benefits

The Warp feature provides quick visual, global modeling without your having to consider the construction of the model. With Warp you can

use imported data such as IGES in family tables, Relations BMX Analysis, and so forth.

The Pro/ENGINEER Warp feature provides for universal, user‑defined deformations of selected geometry within a 3D marquee. Selected geometry includes facets, curves, surfaces (native and imported), and solids. Functionality includes the ability to dynamically scale, reorient, taper, stretch, bend, and twist geometry. The resultant Warp feature behaves as any other Pro/ENGINEER feature, including the ability to make quasi‑parametric modifications.

Menu Location
Insert > **Warp**

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Simulation

Simulation applications in Wildfire Help include Pro/MECHANICA, Mechanism Design Extension, and Mechanism Dynamics Option. Highlights of new functionality for this release follow.

Pro/MECHANICA

Improved Workflow for Defining and Solving Analyses and Studies

Benefits

Many of the menus and dialog boxes you use to define and solve analyses are now merged into a new, concise, easier-to-use run form.

Defining and solving analyses and design studies now takes less time and fewer menu selections.

Description

The workflow for defining analyses and design studies is streamlined in Pro/MECHANICA Wildfire, enabling you to define, edit, run, and view the progress of analyses and studies using the same, concise dialog box. This new dialog box replaces the old analysis and design study finder forms, as well as the old run dialog box. This consolidated dialog box makes it much simpler to define analyses. Additionally, the run summary and log now share a common window, enabling you to quickly toggle between .rpt and .stt file views. This shared window is also resizable.

Menu Location
Analyses and Studies

New Simulation Display Dialog Box

Benefits

The simulation visibilities and simulation display dialog boxes are now merged into a new dialog boxes that is simpler to use and has more functionality.

You will have greater control over model visualization and interact more easily with the new simulation display dialog box.

Description

The simulation display settings and display dialog boxes have been merged into a better-organized, more fully featured simulation display user interface. Some of the new capabilities include the ability to display simulation-modeling icons during mode rotations and to fully configure the display settings in the config.pro file. This new simulation display dialog box is also accessible from a toolbar button.

Menu Location
View > **Simulation Display**

New Shell Properties Definition Dialog

Benefits

A new shell-properties user interface streamlines advanced shell definition (by laminate layup or by stiffness).

Improvements to the shell properties user interface make defining advanced shells (either by laminate layup or stiffness) easier and quicker.

Description

The new shell properties form offers functionality improvements like output of laminate layups and stiffnesses to a file, as well as enhancements like use of the right mouse button for editing the table you use to define laminate plies.

Menu Location
Model > **Idealizations** > **Shells** > **Properties – New**

No More .mda or .mdb Files for Integrated Mode Models

Benefits

All Pro/MECHANICA modeling data for integrated models is now stored in Pro/ENGINEER part and assembly files.

You will have fewer files to track, saving time and minimizing data management overhead.

Description

All integrated mode simulation modeling data is now stored in Pro/ENGINEER part and assembly files. This means you no longer have to keep track of .mdb and .mda files, making it easier to manage model files and reducing the likelihood of losing modeling data.

Menu Location

None

Pro/MECHANICA and Pro/ENGINEER on the Same CD

Benefits

Pro/ENGINEER and Pro/MECHANICA are now installed from the same CD-ROM.

Enjoy simpler, quicker installation as well as fewer release incompatibilities between Pro/ENGINEER and Pro/MECHANICA.

Description

You now install Pro/MECHANICA and Pro/ENGINEER from the same CD-ROM. The process improvements inherent in this approach results in faster installation and end date code mismatches between the two products.

Menu Location

None

Pro/MECHANICA Configuration Options Merged into Pro/ENGINEER config.pro

Benefits

Pro/MECHANICA configuration options from the config.mech file have been merged into the Pro/ENGINEER preferences file.

If you use integrated mode, you save time by no longer needing to manage separate configuration files for Pro/MECHANICA and Pro/ENGINEER.

Description

All integrated mode Pro/MECHANICA simulation settings in the config.mech file are now merged into the Pro/ENGINEER config.pro preferences file. This lets you escape the overhead of separately managing your simulation-modeling environment. It also gives you access to the Pro/ENGINEER preference editor's improved usability and search capabilities.

Menu Location
Tools > **Options**

New Assembly Modeling Capabilities

Benefits
Pro/MECHANICA features new methods of intelligently joining midsurface compressed solid parts, making it easier than ever to model assemblies that include such parts.

This new capability significantly reduces the amount of time it takes to set up and solve assemblies that contain midsurface compressed solids.

Description
Pro/MECHANICA Wildfire offers dramatic improvements in modeling assemblies that contain midsurface compressed parts, such as sheet metal assemblies. In the past, such assemblies were painstaking to model partly because, once compressed, the components were often no longer in contact. You then needed to modify part locations (artificially interpenetrate parts by half the material thickness), define connections such as welds, or modify the shell pair placement. With these assembly-modeling enhancements, Pro/MECHANICA detects where components are in contact. The software automatically ties the displacements and rotations of the contacting surfaces or edges together in the compressed model. If you do not want particular surfaces joined, you can either leave a small gap in the assembly or define a "Free" connection (also new in Pro/MECHANICA Wildfire) between these surfaces. You can view the automatically created links in AutoGEM for both integrated and independent modes.

Menu Location
None

Pro/MECHANICA Integration into Behavioral Modeling (BMX)

Benefits

Behavioral modeling users can now include Pro/MECHANICA measures and analysis in their BMX design studies.

Pro/MECHANICA integration into BMX gives both BMX and Pro/MECHANICA users new, exciting capabilities for optimizing and better understanding the behavior of their designs.

Description

You can now define analysis features that report measures from a specified Pro/MECHANICA analysis. These Pro/MECHANICA analysis features can be included in all BMX studies, including feasibility and multi-objective design studies. For example, now you can optimize a model for center of gravity, stress, and surface area at the same time.

Menu Location

Insert > Model Datum > Analysis

Mass Idealizations Defined from Component

Benefits

You can now specify the mass values for mass idealizations directly from an assembly component, maintaining the correct value and orientation.

Specifying the mass properties for a mass idealization directly from a component minimizes the likelihood of typing errors and incorrect orientation.

Description

You can now specify the mass of a mass idealization in an assembly by selecting a "source" component. This enables you to define the mass idealization without having to worry about orientations or the specific values of the inertia tensor. "From Component" mass idealizations also handle any offset from the application point and the center of gravity about which the mass is computed. To define a "From Component" mass idealization, you start by creating a simplified representation of the assembly that excludes the component to be idealized. Next, you move to Pro/MECHANICA and define a mass idealization in the model, then select the "source" component for the mass values from the model tree. This mass value is associative and updates whenever the source component is regenerated.

Menu Location

In an assembly, select Structure > **Model** > **Idealizations** >
Masses > **New (**Type "**Component**")

Import of Convection Coefficients and Bulk Temperatures

Benefits

You can now import bulk temperatures and convection coefficients and
apply them to thermal analysis models.

You can create more complex analyses involving heterogeneous
analysis programs such as computational fluid dynamics (CFD).

Description

If you use Thermal, you can now import boundary conditions from
computational fluid dynamics (CFD) software packages.
Pro/MECHANICA imports the file in a manner similar to that used for
external temperature loads in the Structure module. The import file
uses FNF (finite neutral format). This functionality enables you to
define more complex, multi-physics analyses in Pro/MECHANICA.

Menu Location

Thermal > **Model** > **Bndry Conds** > **New** > **Conv Cond** > **Surface**
(Variation set to external)

Full AutoGEM in Integrated Mode

Benefits

Integrated mode users can now use the full interactive AutoGEM
meshing capabilities featured in independent mode.

If you encounter meshing problems, you no longer have to go to
independent mode to diagnose your model.

Description

One of the most significant projects introduced in Pro/MECHANICA
Wildfire is full access to AutoGEM in integrated mode. You can now
modify mesh settings and interactively create a Pro/MECHANICA
mesh by selecting volumes or surfaces, or by simply telling AutoGEM to
mesh all model geometry that has properties (shell, material, and
beam). As AutoGEM meshes, it displays full information and warnings
for you to review. Reviewing capabilities include element validation,
boundary face highlighting, and edge highlighting. Once you are
satisfied with your mesh, you can save it for use in an analysis.

Creating and reviewing the mesh before your run is an optional step; you can still run your model without first creating a mesh.

Menu Location

Structure > Model – AutoGEM

Volumetric Heat Loads

Benefits

You can now define volumetric heat loads in integrated mode for use in thermal analyses.

Defining volumetric heat loads in integrated mode saves you the effort of transferring to independent mode for this functionality.

Description

You can now define volumetric heat loads in integrated mode. These can be applied to components in an assembly.

Menu Location

Thermal > Model > Heat Loads > New > Component

2D Pressure Loads

Benefits

You can now define pressure loads for 2D models such as axisymmetric 2D models or plane stress 2D models.

Defining 2D pressure loads in integrated mode saves you the effort of transferring to independent mode for this functionality.

Description

You can now define pressure loads on curves for 2D models. Unlike independent mode where you apply pressure loads to element edges instead of curves, you can use the new integrated mode approach more robustly in sensitivity and optimization studies.

Menu Location

Structure > (select 2D model type) > Loads > New > Pressure

P-level Plots

Benefits

Integrated mode users can now define P-level results plots.

Reviewing P-level plots in integrated mode saves you the effort of transferring to independent mode for this functionality.

Description

P-level plots can now be defined and viewed in integrated mode results. Prior to Pro/MECHANICA Wildfire, this capability was only accessible in independent mode.

Menu Location

Results > **Insert** > **Result Window**
(Fringe > **Quantity to be P-level)**

AutoGEM Meshing Performance and Business Improvements

Benefits

Pro/MECHANICA Wildfire heralds significant improvements in AutoGEM performance and robustness.

AutoGEM improvements pay dividends by lowering run times for every analysis and design study.

Description

Pro/MECHANICA Wildfire brings you substantial improvements in AutoGEM performance and robustness. You will experience mesh time reductions of two to four times, with even greater gains for large models and thin solid models.

Menu Location

None

Pro/MECHANICA Solver Performance Improvements

Benefits

Pro/MECHANICA solver speed and memory usage have undergone dramatic improvements.

The solver improvements pay dividends by lowering run times for every analysis and design study.

Description

Pro/MECHANICA solution performance has been improved by up to three times, and memory usage sees reductions of up to 40%.

Menu Location

None

New FEM Tetrahedral Mesher

Benefits

A new tetrahedral mesher has been implemented for single volume parts.

The new tetrahedral mesher is much faster, produces higher quality elements, and is more robust than the one it replaces.

Description

In addition to AutoGEM improvements, Pro/MECHANICA Wildfire features an entirely new tetrahedral mesher for FEM mode. At this time, the new mesher is available for single volume parts. You will see performance improvements of up to an order of magnitude, along with much better element quality. Setting the Pro/ENGINEER configuration option FEM_NEW_TETRA_MESHER to "yes" activates this option.

Menu Location

None

Redesigned Pro/MECHANICA Results Workflow

Benefits

The user interface for defining Pro/MECHANICA results has been completely redesigned for improved usability and functionality.

The new user interface and workflow for defining results windows saves you time by making this task much more straightforward and gives you many new capabilities.

Description

Pro/MECHANICA Wildfire offers tremendous usability and capability enhancements in results. You will experience improvements ranging from streamlined result window definition to MPEG export for results animations. The number of dialog boxes you use to define and display a result window has been reduced from four (or more!) to one. In addition to a simpler definition user interface, you will notice many other

enhancements, including the ability to animate all result types, overlay undeformed wireframe or shaded representations of a model on a deformed results plot, and add up to 15 color levels to the legend. This summary highlights only a few of the enhancements in this area.

Menu Location

Results > **Insert** > **Result Window**

Enhancements to Cutting and Capping Planes

Benefits

The Pro/MECHANICA results user interface provides improved cutting and capping plane definition and functionality.

The cutting and capping plane enhancements save you time by reducing the amount of menu interaction and make it easier to probe your results by adding new capabilities.

Description

Pro/MECHANICA Wildfire enhancements have made cutting and capping plane functionality easier to use and more precise. In addition experiencing the benefits of a new, simplified user interface, you can now define cutting and capping plane location by specifying a results value (either a percentage or a specific number). This provides a quick means of isolating areas of high or low stress or displacement for further investigation.

Menu Location

Results > **Insert** > **Capping** > **Cutting Surf**

Named Views in Results

Benefits

You can now access named views from your Pro/ENGINEER model in results. Additionally, you can define and save new views.

Reusing named views from your Pro/ENGINEER model makes it quicker and easier for you to display results exactly the way you want.

Description

You can now access views from Pro/ENGINEER in Pro/MECHANICA results. Additionally, you can define new views (orientation and zoom state) and save these to the results file.

Menu Location

Results > **View** > **Saved View**

Viewed Results by Selecting Components or Layers

Benefits

You can now define results in Pro/MECHANICA by selecting which Pro/ENGINEER components or layers to display.

By defining results using Pro/ENGINEER components or layers, you save time because you can reuse these definitions from within the standard modeling mode.

Description

This feature enables you to select the location for which to display results by selecting either components or layers from a display tree. During layer selection, Pro/MECHANICA only displays layers containing simulation model idealizations such as shells or beams. You have all the same tools you would for Pro/ENGINEER layers. These include hide, show, and isolate.

Menu Location

Results > **Insert** > **Result Window (fringe or vector)** > **Display Location** (select **Component** > **Layer**)

Annotations in Results

Benefits

Pro/MECHANICA now offers results markup capability. You can type notes, add leader lines, and draw shapes directly in the results window.

Sharing and documenting concerns directly in result windows improves communication and eliminates the overhead of going to a separate package for this functionality.

Description

You can now annotate and mark up results windows from within the results user interface. You simply type in a note, select its location in the result window, and specify whether you want to add a leader line pointing to a location in the window. If you select a model location for the leader line, Pro/MECHANICA updates the "attachment point" as you rotate the model. You can format your annotations by defining text style, color, leader style, and so forth. You can also save your

annotations. Pro/MECHANICA supports annotations in printed reports, exported images, and HTML reports.

Menu Location

Results > Insert – Annotation

Graphing over Multiple Curves or Edges

Benefits

You can now use multiple continuously connected curves and beams to define the location for graph results.

You will find it much simpler and quicker to define graphs of results quantities where the location traverses multiple curves or beams.

Description

You can now select curve and beam chains when defining a graph in Pro/MECHANICA results. You can also toggle the start point (left side) of the graph.

Menu Location

Results > Insert > Result Window (graph) > Display Location

Principle Stress Vector Results for Solid Models

Benefits

You can now define principle stress vector results for solid models in Pro/MECHANICA.

Principle stress vector results for solid model provide you with a better understanding of how the model responds to applied loads or constraints.

Description

You can now define principle stress vector results for models containing solid elements. In prior Pro/MECHANICA releases, you could only define this type of result for shell models.

Menu Location

Results > Insert > Result Window (vector - quantity to be principle stress)

Multiple Entity Selection for Results Location

Benefits

Enhanced display location selection in results enables you to select multiple surfaces, curves, or volumes.

You can now more easily select the display location for Pro/MECHANICA results.

Description

You can now select multiple volumes, surfaces, or edges for the display location in results. In prior Pro/MECHANICA releases, you could only select single volumes, surfaces, or edges.

Menu Location

Results > **Insert** > **Result Window (fringe or vector)** > **Display Location**

New Graphing Tool

Benefits

A new, greatly enhanced graphing tool has been implemented. This tool is common to Pro/MECHANICA, Behavioral Modeling, and Mechanisms Design.

The new graphing tool offers significant usability improvements, saving you time. Its enhanced capabilities also reduce the need to pass data to external graphing programs.

Description

Not only does Pro/MECHANICA feature a greatly enhanced results user interface, but it provides an entirely new graphing implementation too. Pro/MECHANICA, Mechanisms Design, and Behavioral Modeling all share this new graphing tool. The graphing tool provides such improvements as increased formatting, editing, and display capabilities. You will also appreciate the ability to output graphs and data directly to MS Excel worksheets.

Menu Location

Results > **Insert** > **Result Window (graph)**

Hierarchical FEM Modeling

Benefits

Hierarchical FEM modeling lets you store FEM meshes for component models and elevate these meshes to higher-level assemblies, incorporating them into the overall system model.

Your design team saves time because individual contributors can work concurrently on the definition of system level analysis models. Plus, you experience greater reuse of FEM modeling data.

Description

Hierarchical FEM modeling gives analysts the tools to model complex assemblies in much the same way that designers model them in Pro/ENGINEER. It does this by letting you associate a mesh with a specific component, and allowing this mesh to be used in a higher-level assembly containing the component. Before you mesh assembly components, you define interfaces between these components and copy these interfaces into each of the models sharing a particular interface. Thus, you can partition large models so multiple teams can work concurrently, bringing the system level model together wherever the assembly is opened. Another benefit is the ability to reuse previously defined component meshes in future assemblies. With hierarchical FEM modeling, the mesh persists during modeling so you can see the mesh when adding loads, constraints, and materials. You can enable this option for models you plan to output to NASTRAN. You do so by setting the FEM_MESH_PRESERVE config.pro option to "yes".

Menu Location

None

Enhanced NASTRAN Modeling Entity Support

Benefits

You can now define rigid and weighted links in FEM mode for use in NASTRAN.

Enhanced NASTRAN modeling entity support allows NASTRAN users to create more complex structural analysis models.

Description

NASTRAN users of FEM mode will see additional support for more advanced NASTRAN modeling entities. You can now define rigid and weighted links in FEM mode. These modeling entities output RBAR

and RBE3 cards to the NASTRAN deck. You can also control the degrees of freedom specified in their definition.

Menu Location

Model > **Connections** > **Rigid or Weighted Links**

Distributed Masses in FEM Mode

Benefits

You can now define masses in FEM mode that are distributed over surfaces or curves.

This new functionality lets you model nonstructural mass (such as paint or a wiring harness) in a structural analysis assembly.

Description

FEM mode users can now define distributed masses. You apply the mass to curves or surfaces and specify the value as either a total value or a flux (applied per unit length or per unit area). You can also derive the total value from a component mass.

Menu Location

Model > **Idealizations** > **Masses** > **New** (select **curve** or **surface** reference)

Merged Mechanica Mode and FEM Mode Results

Benefits

FEM mode results have been merged into the new Pro/MECHANICA post-processor.

As a result of merging the workflow and capabilities of FEM mode and Mechanica mode results, FEM mode users will see enhanced functionality and usability.

Description

FEM mode results have been merged into the Pro/MECHANICA results user interface. In addition to improved usability, FEM mode users will be able to access Pro/MECHANICA capabilities such as dynamic query, annotations in results, cutting and capping planes, and much more. New results quantities include beam stresses and strains.

Menu Location

Results. Ansys results are unchanged.

Mechanism Design Extension (MDX)

Mechanism > Component Motion Envelope

Benefits

By efficiently isolating the capture of swept volumes of interest to selected components, you improve envelope quality and reduce creation time.

After running an analysis, you can directly create motion envelopes from the **Playback** dialog box. Select the desired components and quality level. Then preview the envelope or save it as a part.

Menu Location

Applications > **Mechanism** > **Mechanism** > **Playback**

Mechanism > Parametric Motor Profiles

Benefits

You can specify motor profile definitions that respond to change. By leveraging BMX design studies, you can optimize mechanism performance over a range of input characteristics.

Create and assign values for numeric Pro/ENGINEER assembly parameters. When creating a Servo Motor, select the user-defined magnitude type and define an expression. Activate the desired parameters and specify the mathematical expression for the magnitude profile.

Menu Location

Applications, Mechanism, Mechanism, Servo Motors

Mechanism > Graph Tracking Bar

Benefits

You can pinpoint key performance events, such as acceleration spikes and dwell periods, by simultaneously tracking a graphed measurement while observing the mechanism in motion.

For a particular analysis, you can graph desired measures of interest. With the graph window displayed, you can animate the mechanism by starting the analysis playback. A vertical tracking bar in each graph window changes position as the playback progresses.

Menu Location

Applications > Mechanism > Mechanism > Measures

Mechanism > Servo Motor Velocity & Acceleration Graphs

Benefits

Before running an analysis, you can review, examine, and compare the position velocity or acceleration characteristics of a Servo Motor profile.

Mechanism Design Extension performs the necessary integration and derivative calculations to compute and graph position, velocity, and acceleration. When defining the profile of a Servo Motor, select one or more graphing optionsposition, velocity, or acceleration. You can plot the individual profiles on the same graph or in separate graphs that are stacked vertically for comparison.

Menu Location

Applications > Mechanism > Mechanism > Servo Motors

Mechanism - True Kinematic Analysis

Benefits

You can evaluate the kinematic performance of a mechanism without having to assign mass properties to assembly components.

Pro/ENGINEER Foundation alone represents a comprehensive kinematic simulation tool for mechanism design without requiring any additional modules. First, create measures for the velocity and acceleration of a point or joint. Then create and run a kinematic analysis. The velocity and acceleration measure data can be graphed or displayed as animated vector arrows during playback of the analysis.

Menu Location

Applications > Mechanism > Analysis > Kinematic

Mechanism - Object Action for Direct Interaction

Benefits

Quickly and easily, you can create, review, delete, or edit mechanism features such as connection properties, Servo Motors, or gears through direct interaction with the assembly model.

Select a mechanism icon and right-click to reveal a shortcut menu with pertinent actions. Select a joint axis to edit its settings or to create a

motor. Select a Servo Motor to review its current properties in an information window, edit its profile characteristics, or simply delete it. Each mechanism feature icon provides access to its own associated actions.

Menu Location

Applications > Mechanism

Mechanism - Gear Connections

Benefits

Gear connections provide a generalized method of modeling spur, bevel, helical, rack and pinion, and worm gear combinations, as well as idealized pulley pairs.

Select two joint axes and either define appropriate pitch circle diameters to allow Mechanism Design to assign a gear ratio, or define your own ratio to be enforced between the moving bodies.

Menu Location

Applications > Mechanism > Mechanism > Gears

Mechanism - Enhanced Animation Capture

Benefits

New capture options ensure the creation of quality animation files in an expanded range of file types.

The aspect ratio of captured animation files can now be locked to the aspect ratio of the displayed model, helping to ensure quality images free of distortion. A new setting allows adjustment of frame rate for MPEG animations. Export options now include the TIFF and Windows BMP file types.

Menu Location

Applications > Mechanism > Mechanism > Playback > Capture

Mechanism Dynamics Option (MDO)

Mechanism - Measure Dependent Forces

Benefits

To a mechanism you can apply forces of angular orientation, separation distance, velocity, and so forth, which cannot be described easily as functions of time.

Create appropriate position or velocity measures. When creating a Force Motor, a Point Force, or a Body Torque, select the user-defined magnitude type. Then select the desired measure from the variable list, and specify a mathematical expression and domain for the magnitude profile.

Menu Location

Applications > Mechanism > Mechanism > Force Motors

Mechanism - Static Analysis Progress Graph

Benefits

Observe and track the solution progress of a static analysis by monitoring the updated graph of maximum system acceleration as it approaches zero.

When running a static analysis using the Mechanism Dynamics Option (MDO), the mechanism animates, and a graph of maximum system acceleration appears. The graph updates with each solution step.

Menu Location

Applications > Mechanism > Analysis > Static

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Distributed Applications

Distributed applications for Pro/ENGINEER Wildfire Help include Distributed Pro/BATCH and Distributed Services Manager (DSM). Highlights of new functionality for this release follow.

Distributed Pro/BATCH

Data Exchange

Benefits

With Pro/BATCH, you can offload tasks to remote servers to boost productivity and broaden organizational access to Pro/ENGINEER.

Distributed Pro/BATCH performs batch jobs in a distributed environment by leveraging hardware and software resources on your network. Using Distributed Pro/BATCH, you can perform large volumes of routine, time-consuming, and computer-intensive tasks, such as plotting, importing and exporting of 2D and 3D data, and batch execution of ModelCHECK. By using Distributed Services Manager, a distributed resource and services management software system, jobs are queued if resources are unavailable. In addition, system administrators can schedule the processing of batch type jobs over the network at low traffic times. A Web interface provides clients with the current status of pending jobs from any location. Distributed Pro/BATCH and the Distributed Services Manager are built from the Web up by incorporating open standards and Web protocols, such as SOAP, XML, and HTTP.

Menu Location

N/A

Distributed Services Manager (DSM)

Data Exchange

Benefits

With DSM, you can schedule the processing of distributed tasks and queue the tasks when resources are unavailable. A centralized hub connects all distributed clients and services on the network.

Distributed Services Manager (DSM) is new for release with Pro/ENGINEER Wildfire. With DSM, engineers or CAD Administrators can set up and manage a server farm of network resources. Remote clients can request tasks from the DSM, which assigns these tasks to available service resources or queues the tasks if necessary. The service resources host one or more server-side applications, such as Distributed Pro/BATCH, to perform the requested task. Distributed Pro/BATCH is the first distributed service developed by PTC to provide distributed resource management.

Companies can leverage DSM technology in the scheduling the availability of services in response to requests from remote clients. A Web-portal provides you with online access to monitor the progress of jobs from any location. DSM provides the latest Web-based distributed computing protocols and open standards, namely SOAP, XML, and HTTP.

Menu Location

N/A