

Product What's New: Pro/ENGINEER Wildfire 5.0

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Product What's New

Pro/ENGINEER Advanced Assembly

[Associative Solid Shrinkwrap-External Shrinkwrap Feature](#)

To easily copy geometry from a selection of models, select the Auto collect all Solid Surfaces Shrinkwrap option. You have the option to solidify the resulting geometry and to remove holes from selected surfaces.

[Default Envelope](#)

Use Default Envelope to represent an assembly with a single part.

[Granular Change Definition of Data Sharing Features](#)

The change of a copy geometry feature referencing a sketch or curve feature is determined on a more granular level. This reduces the risk of unintentionally changed objects.

[Inheritance Feature Enhancements](#)

The inheritance feature has improved support for model-based definition and toolkit.

[Simple Replace of Copied Model](#)

Quickly replace copied models by automapping the same feature IDs..

Product What's New

Pro/ENGINEER Advanced Rendering

[Advanced Rendering Overhaul](#)

There are significant improvements to Photolux.

[Appearance Overhaul](#)

The assignment, editing and managing of appearances is improved.

[New appearance and scene library](#)

New appearances and scenes have been added

[New appearance classes](#)

The appearance classes have been consolidated

[Perspective Enhancements](#)

Perspective view settings and controls are improved.

[Scene Definition is Easier to Use](#)

There are usability improvements to the scene definition.

[Shadow Catcher Support](#)

A shadow catcher appearance can now be added to the room walls.

Product What's New

Pro/ENGINEER Foundation XE

[Animated Explode and Unexplode Sequences](#)

You can animate explode or unexploded sequences.

[Assembly Cut Enhancements](#)

Switch between the simplified and advanced assembly cut environments by selecting or clearing a checkbox. Cut challenging geometry with the Check Geometry Option.

[Assembly Explode Usability Enhancements](#)

You can reposition models in the new interface for Explode, by dragging and rotating them.

[Assembly Restructure Enhanced Reference Handling](#)

Enhanced reference handling simplifies restructuring by minimizing external references and dependencies.

[Assembly Restructure Usability Enhanced](#)

Improved usability simplifies assembly restructuring.

[Associative Solid Shrinkwrap, Assembly context Shrinkwrap feature](#)

To easily copy geometry from a selection of models, select the "Auto collect all Solid Surfaces" Shrinkwrap option. You have the option to solidify the resulting geometry as well as remove holes from selected surfaces.

[Autonumbering of New models in Copied Assembly](#)

When connected to a Windchill server, you can use autonumbering to name new models generated as a result of copying an assembly,

[Component Chooser](#)

Use dynamic preview and powerful selection tools to build component subsets within the component chooser. The Component Chooser is used to define simplified representations, envelopes, and shrinkwrap features.

[Dependant Mirror Model Tree Designation](#)

Dependant mirror feature now a unique icon in the model tree.

[Dynamic Component Mirror Preview](#)

Using dynamic view, you can explore alternatives prior to generating new models.

[Enhanced Component Copy and Paste](#)

Quickly paste components using temporary interfaces.

[Envelope Enhancements](#)

With the new component chooser, envelopes are faster and easier to define and maintain. You can quickly define simplified representations based on the envelope and represent an assembly with a single part.

[Failure Tolerance](#)

You can postpone resolving assembly failures so you can open and work with the assemblies with the missing information.

[Flatten Form Enhancements](#)

Flatten form capabilities are improved.

[Flexible Component Enhancements](#)

There is improved regeneration of flexible components.

[Graphics Representation Enhancements](#)

It is easier to select graphics representation models. You can store Component placement information in the graphics representation.

[Mirror Walls in Sheetmetal](#)

You can mirror flat and flange walls in Sheetmetal.

[Model Tree for Accessory Window](#)

You can select references in the Model Tree and control layers in the Layer Tree, specific to the accessory window.

[Open Generic -Shortcut Command](#)

You can open the generic of an instance from a shortcut menu. It is easy to identify an instance in the Model Tree when its generic name is appended.

[Quick Retrieval of Rule-Based Simplified Representations](#)

It is optional to evaluate rules when retrieving a simplified representation. Retrieving the simplified representation as it was last evaluated, can sometimes result in faster retrieval.

[Repeat -Shortcut Command](#)

You can quickly repeat a component placement using a command from a shortcut menu. During a repeat sequence, you can change the number of variable constraints. Select the constraints in the Model Tree, right-click and select Repeat to repeat a component.

[Restrictions Removed from Assembly Restructure](#)

You can restructure Simplified Representations of assemblies and components that could previously not be restructured.

[Retrieval Performance of Simplified Representations with Instances](#)

Enhanced Instance dependency retrieval is extended to simplified representations. Retrieving simplified representations with Family Table instances is faster.

[Simplified Accessory Window](#)

You can dock or undock the Accessory window. The docked accessory window (default) stays in the main window so it is not hidden behind other windows. You can easily restore a minimized Accessory window by clicking a glyph.

[Simplified Representation New Default Rule](#)

When creating a new Simplified Representation, components are excluded by default.

[Simplified Representation Usability Enhancements](#)

Use extended preview and information to easily create or select the correct simplified representation.

[Snap Single Constraint](#)

Snapping one constraint at a time simplifies drag and snap.

Product What's New

Pro/ENGINEER Interactive Surface Design

[General Improvements in Style](#)

There is improved usability in the Style feature.

[New N-sided surface](#)

A new n-sided surface is added.

[Style Curve Enhancements](#)

Style curve creation and editing tools are enhanced.

[Style Surface Enhancements](#)

The Style surface creation and editing tools are enhanced.

[Surface Edit Enhancements](#)

The Style Surface Edit functionality is enhanced.

Product What's New

Pro/ENGINEER Reverse Engineering

[Enhanced Surface Modify Command](#)

The Restyle Surface modify command is easier to use and more powerful.

Product What's New

Assembly

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Pro/ENGINEER

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The assignment, editing and managing of appearances is improved.

[Perspective Enhancements](#)

Perspective view settings and controls are improved.

[Realism Enhancement](#)

A transparent floor option is added to realtime rendering.

[Scene Definition is Easier to Use](#)

There are usability improvements to the scene definition.

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Rendering

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Sheetmetal Design and Manufacturing

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Surfacing - Restyle

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The Restyle Surface modify command is easier to use and more powerful.

Product What's New

Detailed Design

[3D Contact Connections](#)

There are 3D contact connections with material properties, and static and kinematic friction between planes, spheres, cylinders and vertices.

[Analyze Effect of Changed Nominal Dimensions](#)

Edit Nominal Dimensions in the Tolerance Analysis Tree List.

[Animated Explode and Unexplode Sequences](#)

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generated as a result of copying an assembly,

[Belt Connections](#)

You can route closed belts around pulleys in a mechanism for use in drag and all kinematic and dynamic analysis types.

[Column Control in MODS Analysis Result Table](#)

You can configure column display in MODS Result tables.

[Component Chooser](#)

Use dynamic preview and powerful selection tools to build component subsets within the component chooser. The Component Chooser is used to define simplified representations, envelopes, and shrinkwrap features.

[Datum Points and Coordinate Systems from Mathcad Matrices](#)

Create datum points and coordinate systems from Analysis features that reference matrices in Mathcad.

[Default Envelope](#)

Use Default Envelope to represent an assembly with a single part.

[Dependant Mirror Model Tree Designation](#)

Dependant mirror feature now a unique icon in the model tree.

[Design Studies of Parameters in Footer Features](#)

You can use parameters from Features in the Footer in Design Studies

[Draft Analysis Enhancements](#)

Color tools for Draft Analysis are enhanced.

[Dynamic Component Mirror Preview](#)

Using dynamic view, you can explore alternatives prior to generating new models.

[Enhanced Component Copy and Paste](#)

Quickly paste components using temporary interfaces.

[Enhanced Excel analysis Feature](#)

You can store an updated Excel file on regeneration.

[Enhanced Torsion Spring](#)

There are torsion springs with user-selectable endpoints, representing the attachment

points for the torque arms of a torsion spring.

[Enhanced Workflow in Tolerance Analysis](#)

Delete and substitute components and dimensions in an existing analysis.

[Envelope Enhancements](#)

With the new component chooser, envelopes are faster and easier to define and maintain. You can quickly define simplified representations based on the envelope and represent an assembly with a single part.

[Failure Tolerance](#)

You can postpone resolving assembly failures so you can open and work with the assemblies with the missing information.

[Flatten Form Enhancements](#)

Flatten form capabilities are improved.

[Flexible Component Enhancements](#)

There is improved regeneration of flexible components.

[Forces Depending on Forces](#)

User-defined forces consider existing force and torque measures.

[Gap Close Element](#)

You can magnetically close gaps in the model.

[Granular Change Definition of Data Sharing Features](#)

The change of a copy geometry feature referencing a sketch or curve feature is determined on a more granular level. This reduces the risk of unintentionally changed objects.

[Graphics Representation Enhancements](#)

It is easier to select graphics representation models. You can store Component placement information in the graphics representation.

[Inheritance Feature Enhancements](#)

The inheritance feature has improved support for model-based definition and toolkit.

[Keyframe Sequences Based on Explode States](#)

There is a new animation type for animating between explode states.

[Mirror Walls in Sheetmetal](#)

You can mirror flat and flange walls in Sheetmetal.

[Model Tree for Accessory Window](#)

You can select references in the Model Tree and control layers in the Layer Tree, specific to the accessory window.

[New Generic Gear Type](#)

There is a new gear type that enables any type of rotational and translational motion coupling.

[Open Generic -Shortcut Command](#)

You can open the generic of an instance from a shortcut menu. It is easy to identify an instance in the Model Tree when its generic name is appended.

[Performance Monitor](#)

Monitor critical design measurements and safety margins to design requirements.

[Quick Retrieval of Rule-Based Simplified Representations](#)

It is optional to evaluate rules when retrieving a simplified representation. Retrieving the simplified representation as it was last evaluated, can sometimes result in faster retrieval.

[Reaction Measures on Ball and Bearing Joints](#)

Reaction measures can be expressed in the X,Y,Z coordinates of the involved bodies.

[Repeat -Shortcut Command](#)

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[Simplified Representation New Default Rule](#)

When creating a new Simplified Representation, components are excluded by default.

[Simplified Representation Usability Enhancements](#)

Use extended preview and information to easily create or select the correct simplified representation.

[Slot Motor](#)

You can apply servo motor profiles to Slot connections.

[Snap Single Constraint](#)

Snapping one constraint at a time simplifies drag and snap.

[Spur, Bevel, Worm and Rack and Pinion Gear Types](#)

There are new gear types with entry fields for gear teeth manufacturing angles. The impact of manufacturing angles on gear loads are considered in the reaction load calculation

[Statistical Design Studies](#)

Statistical attributes can be applied to dimensions and parameters, enabling the calculation of resulting statistical parameters for any measurable goal in the model.

[Tolerance Analysis Saved as Feature](#)

You can save Tolerance Analysis measurement as a feature, with feature parameters representing Tolerance Analysis results.

[User-Defined Dimensions](#)

Create annotation dimension on the fly if there are no existing dimensions or annotations

[Virtual Components](#)

Create placeholder components to simulate dimension properties that are not accounted for in the CAD model.

[Volume Analysis on Closed Quilts](#)

You can analyze volume and mass properties on closed quilts.

Product What's New

Associative Solid Shrinkwrap-External Shrinkwrap Feature

To easily copy geometry from a selection of models, select the Auto collect all Solid Surfaces Shrinkwrap option. You have the option to solidify the resulting geometry and to remove holes from selected surfaces.

Product Information

Product	Pro/ENGINEER Advanced Assembly
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	In a part, click Insert > Shared Data > Shrinkwrap and then select or open model from which geometry will be copied. Define placement and then select Auto collect all solid surfaces in list next to Subset. Click Subset to select contributing geometry and click OK. Click the Options panel to fill contours and solidify the geometry.
Processes and Initiatives	Detailed Design

Benefits and Description

The Auto collect all solid surfaces shrinkwrap feature option copies the all the solid surfaces from a model of your choice to a part feature.

When you define the Shrinkwrap you can:

- Specify what parts of an assembly contributes to the resulting Shrinkwrap geometry
- Solidify the resulting Shrinkwrap geometry
- Fill contours (holes) on selected surfaces
- Display filled (removed) holes with a yellow curve

Filling contours such that a cavity is enclosed, removes the cavity from the Shrinkwrap geometry.

Product What's New

Default Envelope

Use Default Envelope to represent an assembly with a single part.

Product Information

Product	Pro/ENGINEER Advanced Assembly
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	No preexisting default envelope: Right-click an assembly and select Representation > Default Envelope. Provide a name and select a template. In the Shrinkwrap dash board, click Subset to define a subset contributing to the geometry and then click OK. Click the Options tab to fill contours and solidify the geometry. Preexisting default envelope: Right-click an assembly and select Representation > Default Envelope. The default envelope represents the assembly in a temporary simplified representa
Processes and Initiatives	Detailed Design

Benefits and Description

The Default Envelope automatically substitutes all the components of the assembly with a single new part. You control what components contribute to representing the geometry of this part and whether to solidify and remove holes from the geometry.

A specific simplified representation holds the default envelope for easy substitution in higher-level assemblies.

You can create Default envelopes:

- Bottom up. Create the default envelope in the subassembly prior to using in a higher-level assembly
- Top down. If no default envelope of the selected subassembly is present, you can create a default envelope when required in a higher-level assembly. The default envelope of the subassembly is created in the subassembly and used simultaneously in the higher-level assembly.

Product What's New

Granular Change Definition of Data Sharing Features

The change of a copy geometry feature referencing a sketch or curve feature is determined on a more granular level. This reduces the risk of unintentionally changed objects.

Product Information

Product	Pro/ENGINEER Advanced Assembly
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	N/A
Processes and Initiatives	Detailed Design

Benefits and Description

Change of a copy geometry feature referencing datum curves (sketch) is determined at the entity level rather than the curve feature level. If the master curve feature (sketch) is changed, only copy geometry features containing changed entities, and parts containing these copy geometry features, are considered changed.

Product What's New

Inheritance Feature Enhancements

The inheritance feature has improved support for model-based definition and toolkit.

Product Information

Product	Pro/ENGINEER Advanced Assembly
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Click Insert > Shared Data > Merge/Inheritance. Select or open a model to inherit from and click Define Placement. Click the Toggle Inheritance icon in dashboard. In the Options pane, click Varied Items and then click 3D Note.
Processes and Initiatives	Detailed Design

Benefits and Description

You can vary 3D annotations within an inheritance feature the same way you vary other items. This simplifies variant design of models with 3D drawings.

There is Toolkit support for varied properties.

Product What's New

Simple Replace of Copied Model

Quickly replace copied models by automapping the same feature IDs..

Product Information

Product	Pro/ENGINEER Advanced Assembly
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Right-click a model and select Replace from the Shortcut menu. In the Replace dialog box click Unrelated Components, and then select new components. Click Edit Ref Table > (Evaluation Rules) > Evaluate.
Processes and Initiatives	Detailed Design

Benefits and Description

A Same ID (and type) evaluation rule is included among the evaluation rules of Unrelated replace.

By using the Same ID (and type) evaluation rule, references of the same ID and type are automatically matched Therefore, you can quickly replace a model with a copy of the same, or two otherwise unrelated models with the same entity IDs.

Product What's New

Advanced Rendering Overhaul

There are significant improvements to Photolux.

Product Information

Product	Pro/ENGINEER Advanced Rendering
PTC Support Release	Wildfire 5.0
Product Functional Area	Rendering
User Interface Location	View > Model Setup
Processes and Initiatives	

Benefits and Description

The rendering engine for the Advanced Rendering Extension has been changed to Mental Ray. This produces more realistic and representative images.

New and improved functionality is added including support for:

- Physically correct lighting with presets for standard light types
- Final gathering
- Global Illumination
- Caustics

Product What's New

Appearance Overhaul

The assignment, editing and managing of appearances is improved.

Product Information

Product	Pro/ENGINEER Pro/ENGINEER Advanced Rendering
PTC Support Release	Wildfire 5.0
Product Functional Area	Pro/ENGINEER Rendering
User Interface Location	Directly from the toolbar
Processes and Initiatives	

Benefits and Description

When working with appearances there are three tasks:

- Applying
- Editing
- Managing

To apply an appearance, click the Appearance Gallery icon. The Appearance Gallery lists all appearances in My Appearances, the Model, and the Library. You can select the appearance first and then apply it to the model or select the model and then apply the appearance. To edit an appearance right-click the appearance in the Model palette and select Edit from the shortcut menu. After setting appearances, right-click and choose Select Objects from the shortcut menu to replace appearances. Use the Appearance Manager to define and save your appearance libraries.

Product What's New

New appearance and scene library

New appearances and scenes have been added

Product Information

Product	Pro/ENGINEER Advanced Rendering
PTC Support Release	Wildfire 5.0
Product Functional Area	Rendering
User Interface Location	
Processes and Initiatives	

Benefits and Description

A new library of appearances have been added accurately simulating real world materials.
New scene files have been created taking advantage of the new functionality.

Product What's New

New appearance classes

The appearance classes have been consolidated

Product Information

Product	Pro/ENGINEER Advanced Rendering
PTC Support Release	Wildfire 5.0
Product Functional Area	Rendering
User Interface Location	
Processes and Initiatives	

Benefits and Description

The appearance classes and interface has been significantly consolidated, improving the workflow and usability.

Product What's New

Perspective Enhancements

Perspective view settings and controls are improved.

Product Information

Product	Pro/ENGINEER Pro/ENGINEER Advanced Rendering
PTC Support Release	Wildfire 5.0
Product Functional Area	Pro/ENGINEER Rendering
User Interface Location	Top-level toolbar. Settings available through View > Model Setup > Perspective Settings
Processes and Initiatives	

Benefits and Description

Perspective incorporates preset lens types and from the top-level menu you can toggle perspective mode.

Product What's New

Scene Definition is Easier to Use

There are usability improvements to the scene definition.

Product Information

Product	Pro/ENGINEER Pro/ENGINEER Advanced Rendering
PTC Support Release	Wildfire 5.0
Product Functional Area	Pro/ENGINEER Rendering
User Interface Location	View > Model Setup > Scene Editor
Processes and Initiatives	

Benefits and Description

The Room, Light, and Effects dialog boxes are consolidated into the Scene Editor dialog box. Multiple tabs are consolidated, improving the definition and usability of these options.

Product What's New

Shadow Catcher Support

A shadow catcher appearance can now be added to the room walls.

Product Information

Product	Pro/ENGINEER Advanced Rendering
PTC Support Release	Wildfire 5.0
Product Functional Area	Rendering
User Interface Location	
Processes and Initiatives	

Benefits and Description

A new appearance class has been added allowing users to apply an appearance to any wall of the room. This appearance will catch shadows and reflections only. This is beneficial when rendering with a background image. The rendered object will then appear as if it is sat on the background image.

Product What's New

Animated Explode and Unexplode Sequences

You can animate explode or unexploded sequences.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Click View > Explode > Explode View. ; Click View > Explode > Unexplode View. ; Click View > Display Settings > Model Display. Under Animation while exploding click Enable.
Processes and Initiatives	Detailed Design

Benefits and Description

When exploding or unexploding an assembly you can choose to:

- Not animate
- Animate the shortest distance
- Animate following the explode sequence

Use the Model Display dialog box or set the following configuration options to specify preferences and the duration of the animation:

animate_explode_states

explode_animation_max_time

Product What's New

Assembly Cut Enhancements

Switch between the simplified and advanced assembly cut environments by selecting or clearing a checkbox. Cut challenging geometry with the Check Geometry Option.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Select the Advanced Intersection check box in the Intersect panel of the Assembly Extrude dashboard. ; Select the Check Geometry Option check box in the Intersect panel of the Assembly Extrude dashboard.
Processes and Initiatives	Detailed Design

Benefits and Description

You can now easily switch between the basic and advanced intersection environment by selecting or clearing a check box. Set the default with the configuration option `advanced_intersection`.

The basic intersection environment is simplified. You can simultaneously set the display level on all components, while the advanced environment continually provides component-by-component display level control.

With the Check Geometry Option there is a more stable component intersection. This makes it possible to create challenging intersection geometry caused by:

- Components with different accuracy than that of the assembly
- Assembly cuts intersecting other assembly cuts

If you select Check Geometry Option, regeneration time may increase.

Product What's New

Assembly Explode Usability Enhancements

You can reposition models in the new interface for Explode, by dragging and rotating them.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Click View > Explode > Edit position. ; Click View > View Manager and click the Explode tab. Click New and specify a name. Click Properties and then click the Edit position icon.
Processes and Initiatives	Detailed Design

Benefits and Description

Assembly explode functionality is migrated to the dashboard.

You can use drag handles and CTRL to multiselect and simplify explode definitions.

You can rotate and translate components when defining an explode state.

Use the explode drag handles to:

- Quickly reposition components without defining motion references.
- Provide three drag directions based on placement constraint where possible.
You can if you like, specify alternative motion references in the Reference tab of the dashboard.

Product What's New

Assembly Restructure Enhanced Reference Handling

Enhanced reference handling simplifies restructuring by minimizing external references and dependencies.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	N/A
Processes and Initiatives	Detailed Design

Benefits and Description

As a result of the enhanced reference handling, there are significantly fewer external references and restructure dependencies that were previously associated with restructuring assemblies:

- When restructuring multiple components from a parent assembly to a subassembly, the parent assembly is removed from the reference paths of direct references between the restructured components.
- References to the parent assembly, or components in subassemblies of the parent assembly, still include the parent assembly in the reference path.
- When restructuring components from the parent assembly to the subassembly, a dependency to the parent assembly is not created.

Product What's New

Assembly Restructure Usability Enhanced

Improved usability simplifies assembly restructuring.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Select a component or components and then drag to new location. ; Right-click one or more components and select Move to New Subassembly from the shortcut menu. Specify a name, choose a template, and specify assembly constraints. ; Select components to restructure and click Edit > Restructure > Select target assembly Click Edit > Restructure, and then select the components to restructure and select the target assembly.
Processes and Initiatives	Detailed Design

Benefits and Description

You can restructure completely without menus or the Restructure dialog box.

Enhancements include:

- Drag and drop restructure.
- Restructuring multiple components at a time.
- The Move to New Subassembly command on a shortcut menu, for on the fly creation of a new target assembly. The component selected prior to selecting this command, determines the Model Tree placement of the new target assembly.

The Restructure dialog box supports both the Object-Action and Action-Object workflows.

Product What's New

Associative Solid Shrinkwrap, Assembly context Shrinkwrap feature

To easily copy geometry from a selection of models, select the "Auto collect all Solid Surfaces" Shrinkwrap option. You have the option to solidify the resulting geometry as well as remove holes from selected surfaces.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	In an assembly, click Insert > Shared Data > Shrinkwrap > select Auto collect all solid surfaces in the drop down menu next to Subset > click Subset to select contributing geometry > OK > Expand the Options tab to fill contours In a part activated in an assembly, click Insert > Shared Data > Shrinkwrap > select Auto collect all solid surfaces in the drop down menu next to Subset > click Subset to select contributing geometry > OK > Expand the Options tab to fill contours and solidify the geomet
Processes and Initiatives	Detailed Design

Benefits and Description

The Auto collect all solid surfaces shrinkwrap feature option copies all the solid surfaces of components included in an assembly to an assembly feature, or to a feature in a part activated in the assembly.

Solid surfaces in a part feature can be solidified.

When defining the Shrinkwrap you can:

- Specify what parts of an assembly contribute to the resulting Shrinkwrap geometry
- Solidify the resulting Shrinkwrap geometry (part feature)
- Fill contours (holes) on selected surfaces
- Display filled (removed) holes with a yellow curve

Filling contours such that a cavity is enclosed removes the cavity from the Shrinkwrap geometry.

Product What's New

Autonumbering of New models in Copied Assembly

When connected to a Windchill server, you can use autonumbering to name new models generated as a result of copying an assembly,

Product Information

Product	Pro/ENGINEER Foundation XE Windchill Windchill ProductPoint
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	In the Assembly Save a Copy dialog box, click Automatic Name/Number generation.
Processes and Initiatives	Detailed Design

Benefits and Description

When connected to a Windchill server with autonumbering enabled, you can automatically name new parts created as a result of copying an assembly.

Product What's New

Component Chooser

Use dynamic preview and powerful selection tools to build component subsets within the component chooser. The Component Chooser is used to define simplified representations, envelopes, and shrinkwrap features.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	N/A
Processes and Initiatives	Detailed Design

Benefits and Description

The Component Chooser consists of a structure tree and a window in which you can dynamically preview the assembly and quickly define component subsets. Preview does not require loading components into memory.

A new Derived status shows components with driven statuses. The derived status changes when the driving components status is changed.

Check boxes indicate component and assembly statuses. An assembly with components of different statuses is Mixed. If you select a Mixed check box, you can reset statuses of components in that assembly.

In the Structure tree you can set Components' statuses by:

- Clicking a component status and selecting a new status from the list
- Right-clicking a component name and selecting a status from the Representation shortcut menu
- Selecting and clearing check boxes to toggle statuses

To locate a component prior to setting its status you can:

- Use the search tool in the upper left corner of the chooser.
- Select the component in the preview window to expand the structure tree and highlight the component.

Product What's New

Dependant Mirror Model Tree Designation

Dependant mirror feature now a unique icon in the model tree.

Product Information

Product	Pro/ENGINEER Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Sheetmetal Design and Manufacturing
User Interface Location	NA
Processes and Initiatives	Detailed Design

Benefits and Description

The Model Tree icon used to designate dependant mirror features is updated to differentiate it from independent mirror features.

Product What's New

Dynamic Component Mirror Preview

Using dynamic view, you can explore alternatives prior to generating new models.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Preview icon in mirror dialog
Processes and Initiatives	Detailed Design

Benefits and Description

While mirroring components you can preview the result prior to creating any models. You can, for example, explore the influence of different mirroring planes or make decisions on whether to reuse or copy components.

Product What's New

Enhanced Component Copy and Paste

Quickly paste components using temporary interfaces.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Select the model, click Edit > Copy and then click Edit > Paste. Select the model, press CTRL+C, and then press CTRL+V.
Processes and Initiatives	Detailed Design

Benefits and Description

A temporary interface is extracted and used during the copy and paste of a model. The Component Placement dashboard contains this interface so you can autoplace the model. Alternatively, you can place the model manually.

Product What's New

Envelope Enhancements

With the new component chooser, envelopes are faster and easier to define and maintain. You can quickly define simplified representations based on the envelope and represent an assembly with a single part.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Click View > Envelope Manager > New and then specify envelope component name. Select models to be substituted by the envelope (definition), click Envelope Part and select an envelope method: ; Create and Envelope Part , specify a part name and click OK . Choose a template and click OK. Specify placement > Envelope geometry and create geometry or copy geometry from assembly models.
Processes and Initiatives	Detailed Design

Benefits and Description

The Component Chooser simplifies :

- Envelope Definition and enables dynamic selection of all components to be substituted (included) in the envelope.
- Envelope Part and geometry creation by consolidating the tools.

With the appropriate license you can create and populate the envelope part using an external Shrinkwrap feature.

Right-click the envelope and select Use Envelope to create a simplified representation based on the envelope.

When you use the simplified representation in higher level assemblies the assembly is represented by the envelope part.

The envelope definition and, therefore, the simplified representation, is updated when

components are added to the assembly so that you can create the envelope early in your design process.

Product What's New

Failure Tolerance

You can postpone resolving assembly failures so you can open and work with the assemblies with the missing information.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Right click a failed model. Select an appropriate command to resolve the failure.
Processes and Initiatives	Detailed Design

Benefits and Description

An assembly with missing references, models missing references or missing models ,no longer requires immediate resolution.

You have the option to postpone resolving issues. As failed models are frozen, however, the assembly may not update accurately. A restricted set of actions applies to failed models. In the model tree, failed models are highlighted in red. You can also view the specific status of a model by adding the status column to the model tree.

Failure statuses include:

- Failed. A model that cannot be retrieved or placed.
- Child of failed. A models referencing a failed model.
- Child of external failed.

You can use a shortcut menu to select a command for resolving a failed model.

Product What's New

Flatten Form Enhancements

Flatten form capabilities are improved.

Product Information

Product	Pro/ENGINEER Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Sheetmetal Design and Manufacturing
User Interface Location	Click Format > Legend .
Processes and Initiatives	Detailed Design

Benefits and Description

You can flatten forms that cross over multiple surfaces as in corner gussets. The flattened form area is visible in the unbent model state.

Product What's New

Flexible Component Enhancements

There is improved regeneration of flexible components.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	N/A
Processes and Initiatives	Detailed Design

Benefits and Description

The improved regeneration of flexible components minimizes undesired revision bumping of flexible components.

Product What's New

Graphics Representation Enhancements

It is easier to select graphics representation models. You can store Component placement information in the graphics representation.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Right-click components and select Backup References. ; Click the Backup References check box in the Properties tab of the Component Placement dashboard. ; Click Tools > Assembly Settings > On Demand > Un check Retrieve backed up references
Processes and Initiatives	Detailed Design

Benefits and Description

Preselection highlights the boundary box of models in a graphics representation. This makes it easier to see and select graphics rep models.

You can store placement references in the assembly, resulting in component placement updates in the absence of component geometry.

This enhances the use of light weight graphics representations by minimizing the need to retrieve geometry representations of components whose placements are impacted by a design change.

You can store placement references for selected components using a command from a shortcut menu. You can also store placement references while placing or redefining component.

Set the configuration option `auto_backup_new_placemnt_refs` to automatically create a component reference backup.

Avoid On Demand retrieval of Geometry representations by un checking Retrieve backed up references.

Product What's New

Mirror Walls in Sheetmetal

You can mirror flat and flange walls in Sheetmetal.

Product Information

Product	Pro/ENGINEER Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Sheetmetal Design and Manufacturing
User Interface Location	Click Edit > Mirror.
Processes and Initiatives	Detailed Design

Benefits and Description

Designing symmetric parts in Sheetmetal is easier with the ability to mirror flange and flat walls. You need to specify a plane and select walls to be mirrored. The mirrored walls are created with dependant sections.

Product What's New

Model Tree for Accessory Window

You can select references in the Model Tree and control layers in the Layer Tree, specific to the accessory window.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	N/A
Processes and Initiatives	Detailed Design

Benefits and Description

In a docked Accessory window, the main Model Tree or Layer Tree is split. The lower half of the tree contains Accessory window specific information.

The undocked Accessory window contains a separate Model Tree or Layer Tree specific to the Accessory window.

Product What's New

Open Generic -Shortcut Command

You can open the generic of an instance from a shortcut menu. It is easy to identify an instance in the Model Tree when its generic name is appended.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Right-click the instance and select Open Generic from the shortcut menu.
Processes and Initiatives	Detailed Design

Benefits and Description

Right-click an instance in the Model Tree or Graphics window and select Open Generic. This command opens the top most Generic of that Instance.

The configuration option `modeltree_show_generic_of_instance`, controls the display of the Generic name extension to the instance name in the Model Tree.

Product What's New

Quick Retrieval of Rule-Based Simplified Representations

It is optional to evaluate rules when retrieving a simplified representation. Retrieving the simplified representation as it was last evaluated, can sometimes result in faster retrieval.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Click the Auto Evaluate Model Rules check box in the Open Rep dialog box.
Processes and Initiatives	Detailed Design

Benefits and Description

If the configuration option `auto_evaluate_simprep_rules` is set to yes, the newly created simplified representation is always updated on retrieval or regeneration.

Click the Auto Evaluate Model Rules check box to override the default setting during retrieval.

Product What's New

Repeat -Shortcut Command

You can quickly repeat a component placement using a command from a shortcut menu. During a repeat sequence, you can change the number of variable constraints. Select the constraints in the Model Tree, right-click and select Repeat to repeat a component.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Right-click the component and select Repeat from the shortcut menu. Right-click the constraints to vary in Model Tree and select Repeat from the shortcut menu.
Processes and Initiatives	Detailed Design

Benefits and Description

The Edit > Repeat command is also available on a shortcut menu.

For greater flexibility, you can change the number of variable assembly references during a repeat sequence.

You can repeat a component by selecting one or more references from the component placement folder in the Model Tree, and selecting Repeat from the shortcut menu. The selected references are the variable assembly references. However, within the component placement environment any reference can be varied, providing greater flexibility.

Product What's New

Restrictions Removed from Assembly Restructure

You can restructure Simplified Representations of assemblies and components that could previously not be restructured.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	
Processes and Initiatives	Detailed Design

Benefits and Description

You can restructure:

- The first component of an assembly.
- Components intersected by assembly features
- Components in a Simplified Representation of an assembly. To evaluate and resolve reference changes, some excluded components may be retrieved during the restructure operation.

Product What's New

Retrieval Performance of Simplified Representations with Instances

Enhanced Instance dependency retrieval is extended to simplified representations. Retrieving simplified representations with Family Table instances is faster.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	N/A
Processes and Initiatives	Detailed Design

Benefits and Description

Retrieving simplified representations of assemblies with family tables and family table components is faster due to:

- Less regeneration. Regeneration of intermediate generics of nested family tables is not required.
- Fewer models being retrieved. Components exploded by the family table are not retrieved during the evaluation of the family table.

Product What's New

Simplified Accessory Window

You can dock or undock the Accessory window. The docked accessory window (default) stays in the main window so it is not hidden behind other windows. You can easily restore a minimized Accessory window by clicking a glyph.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	N/A
Processes and Initiatives	Detailed Design

Benefits and Description

It is easier to use the Accessory window for:

- Component placement
- Data Sharing
- Sheet metal form features

While the docked accessory window can be repositioned and resized to your preference, it remains confined within the main window, ensuring it is not hidden behind other windows. A minimized Accessory window appears as a glyph that you can click to open the window.

Using the configuration option `accessory_window_display`, you can undock the Accessory window. If the component to be displayed in the undocked Accessory window is opened in a main window, a new Accessory window opens.

You can use the window's menu or the task bar to locate a undocked accessory window.

Product What's New

Simplified Representation New Default Rule

When creating a new Simplified Representation, components are excluded by default.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	N/A
Processes and Initiatives	Detailed Design

Benefits and Description

The default simp rep rule is changed from Master rep to Exclude. This simplifies large assembly management by:

- Eliminating unintentional retrieval of a new representation where all parts are set to Master Rep.
- Not forcing preview of all components in the component chooser.

You can change the default rule of a new simplified representation by selecting the top assembly in the component chooser and changing its status to Master.

Product What's New

Simplified Representation Usability Enhancements

Use extended preview and information to easily create or select the correct simplified representation.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	In Open Rep dialog box, place the pointer on a user-defined representation. ; In Simp Rep tab of the View Manager, click Display and select Show Preview. Right-click a simplified representation in the View Manager and select Redefine from the shortcut menu.
Processes and Initiatives	Detailed Design

Benefits and Description

Opening or switching to a Simplified Representation is easier because:

- Descriptions are provided in the Open Rep tooltip.
- A preview is provided in the View Manager.

Redefine is added to the Simplified Representation shortcut menu in the View Manager.

Using the dynamic preview and selection tools in the component chooser, you can easily define and edit simplified representations without loading models into memory.

The configuration option new_wf5_simp_rep_ui is set to Yes by default.

Product What's New

Snap Single Constraint

Snapping one constraint at a time simplifies drag and snap.

Product Information

Product	Pro/ENGINEER Foundation XE
PTC Support Release	Wildfire 5.0
Product Functional Area	Assembly
User Interface Location	Click Tools > Assembly Settings > Drag Preferences. In the Drag Preferences dialog box, click the Snap to single constraint check box.
Processes and Initiatives	Detailed Design

Benefits and Description

You can investigate placement options when placing a component, by dragging it and seeing it snap to matching references. This is useful when pasting a component with predefined constraints. Snapping requires one or more constraints and the selection of the Active Snapping Drag option.

You can now snap one constraint at a time rather than snapping all constraints at the same time. Snapping one constraint at a time simplifies dragging and snapping by allowing you to progressively place the component.

By unselecting the Snap to single constraint checkbox in the Drag Preference dialog box, you can snap all constraints at once.

You can also change the drag preference using the configuration option `snap_single_constraint`.

Product What's New

General Improvements in Style

There is improved usability in the Style feature.

Product Information

Product	Pro/ENGINEER Interactive Surface Design
PTC Support Release	Wildfire 5.0
Product Functional Area	Surfacing - ISDX
User Interface Location	Click the Surface Display or Curve Display icons on the Style toolbar. Click Styling > Preferences.
Processes and Initiatives	

Benefits and Description

The following general enhancements have been made to the Style feature:

- In the Style feature, you can hide or unhide the visibility of all curves or surfaces for that feature.
- All options in the Style Preferences dialog box are also available as configuration options.
- Intent planes can be referenced in the Style feature.

Product What's New

New N-sided surface

A new n-sided surface is added.

Product Information

Product	Pro/ENGINEER Interactive Surface Design
PTC Support Release	Wildfire 5.0
Product Functional Area	Surfacing - ISDX
User Interface Location	Click Styling > Surface or the Surface icon on the dashboard. Then, click Access Shape on the Dashboard.
Processes and Initiatives	

Benefits and Description

The following new capability has been added for creating surfaces:

- You can create a surface from more than 4 boundary curves.
- This surface can have positional, normal, or tangent constraints.
- Three-sided surfaces can be built as an n-sided surface. This removes singularities generated from a 3-sided patch.
- Control the surface shape through a Shape Control parameter.

Product What's New

Style Curve Enhancements

Style curve creation and editing tools are enhanced.

Product Information

Product	Pro/ENGINEER Interactive Surface Design
PTC Support Release	Wildfire 5.0
Product Functional Area	Surfacing - ISDX
User Interface Location	Click Styling > Curve Edit.
Processes and Initiatives	

Benefits and Description

The following curve enhancements have been added:

- When dragging a curve, the natural length of its tangent is maintained. This generates smoother curves that are scalable and is the default for all new curves. For curves created prior to Wildfire 5.0, you can manually change from Fix Length to Relax length.
- When editing a curve, Right-click and select Delete from the shortcut menu to delete multiple points on the curve.
- Incrementally move a curve point in a given direction or use the arrow keys.

Product What's New

Style Surface Enhancements

The Style surface creation and editing tools are enhanced.

Product Information

Product	Pro/ENGINEER Interactive Surface Design
PTC Support Release	Wildfire 5.0
Product Functional Area	Surfacing - ISDX
User Interface Location	Click Styling > Surface or Surface icon on the dashboard. Access Reparam mode from the Dashboard.
Processes and Initiatives	

Benefits and Description

The following enhancements have been added to Surfaces:

- Chain collectors are added to surface creation. You can drag, snap and limit boundary curves during their creation.
- Use the Reparam Curve function to redistribute the parameterization of the created surface.

Product What's New

Surface Edit Enhancements

The Style Surface Edit functionality is enhanced.

Product Information

Product	Pro/ENGINEER Interactive Surface Design
PTC Support Release	Wildfire 5.0
Product Functional Area	Surfacing - ISDX
User Interface Location	Use the Style toolbar, or click Styling > Surface Edit.
Processes and Initiatives	

Benefits and Description

With the enhancements to Surface Edit you can:

- Snap the alignment extent when aligning a surface boundary.
- Merge or replace knots from the neighboring surface during an align operation.
- Display the knots of any surface using the new Surface Knots analysis.
- Add a Normal and a Normal Position constraint to a boundary.
- Hide the base surface or display it as a point cloud. Controls are provided for the density of the point cloud.

Product What's New

Enhanced Surface Modify Command

The Restyle Surface modify command is easier to use and more powerful.

Product Information

Product	Pro/ENGINEER Reverse Engineering
PTC Support Release	Wildfire 5.0
Product Functional Area	Surfacing - Restyle
User Interface Location	Insert > Restyle
Processes and Initiatives	

Benefits and Description

The Surface Modify command in Restyle is redesigned to be consistent with the Style Surface Edit command.

You can:

- Use mesh selection
- Add or remove segments
- Preserve boundary conditions
- Incrementally tap the control mesh using the arrow keys
- Work with the improved controls for dragging

Product What's New

Realism Enhancement

A transparent floor option is added to realtime rendering.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Pro/ENGINEER Rendering
User Interface Location	View > Realtime Rendering > Transparent Floor, or from the toolbar
Processes and Initiatives	

Benefits and Description

The new realtime rendering option, Transparent Floor, generates a soft shadow and reflection on an artificial floor, located at the lowest part of the geometry. You can control the orientation of this floor.

Product What's New

3D Contact Connections

There are 3D contact connections with material properties, and static and kinematic friction between planes, spheres, cylinders and vertices.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Mechanism Design & Dynamics
User Interface Location	Click Insert > 3D Contacts
Processes and Initiatives	Detailed Design

Benefits and Description

You can create 3D contact connections that work during drag and any mechanism analysis. You can:

- Define contact between planar surfaces, cylinders and spheres
- Apply material properties from material files
- Include impact effects of Young's modulus, damping, static and kinematic friction
- Measure contact force, area, slip, and pressure angle
- Create Vertex contact and apply a user-defined radius around the selected vertex

3D contact enables greater flexibility in creating highly accurate dynamic analysis of your designs.

Product What's New

Analyze Effect of Changed Nominal Dimensions

Edit Nominal Dimensions in the Tolerance Analysis Tree List.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Other Functional Areas
User Interface Location	Click Analysis > Tolerance Study.
Processes and Initiatives	Detailed Design

Benefits and Description

You can change nominal values of dimensions and analyze the effect on the goal.

You may not be able to change tolerances because they are inherent properties of the manufacturing process. In these cases you may have to change nominal dimensions to achieve the overall tolerancing goal.

Product What's New

Belt Connections

You can route closed belts around pulleys in a mechanism for use in drag and all kinematic and dynamic analysis types.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Mechanism Design & Dynamics
User Interface Location	Click Insert > Belts.
Processes and Initiatives	Detailed Design

Benefits and Description

You can route a closed Belt that connects rotational pulleys. With this new connection you can:

- Use geometry references, or enter user-defined pulley diameters and adjust with drag handles
- Use current or user-defined belt length
- Use self-adjusting tensioner pulleys based on belt length
- Create a part with associative belt trajectory curve
- Apply stiffness of belt (for dynamic analysis)
- Measure belt tension (dynamic analysis)
- Measure belt slip (dynamic analysis)

Belt connections simulate routed rotational coupling of pulleys, and can be used to simulate the kinematic effect of closed chains.

Product What's New

Column Control in MODS Analysis Result Table

You can configure column display in MODS Result tables.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Behavioral Modeling
User Interface Location	Click Analysis > Multi-Objective Design Study.
Processes and Initiatives	Detailed Design

Benefits and Description

You can set which Parameters and Goals appear in the MODS result table. The columns can be reordered and resized.

Product What's New

Datum Points and Coordinate Systems from Mathcad Matrices

Create datum points and coordinate systems from Analysis features that reference matrices in Mathcad.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Behavioral Modeling
User Interface Location	Click Analysis > External Analysis > Mathcad Analysis. Load a worksheet that contains one or more Nx3 or 3x4 result matrices that in Mathcad have Properties - Tag set to "mc2proe". Select them as Outputs. Select Add Feature.
Processes and Initiatives	Detailed Design

Benefits and Description

You can create points from data in Mathcad by creating a Mathcad Analysis feature that references a worksheet with an Nx3 matrix. You can also create Coordinate systems from 3x4 matrices.

Product What's New

Design Studies of Parameters in Footer Features

You can use parameters from Features in the Footer in Design Studies

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Behavioral Modeling
User Interface Location	Use any Design Study and select parameters from Analysis Features in the footer, if available.
Processes and Initiatives	Detailed Design

Benefits and Description

You can use any feature parameters from features placed in the footer in Design Studies such as:

- Sensitivity
- Feasibility/Optimization
- Multi-Objective
- Statistical design

You can also use the Performance Monitor with parameters from features placed in the footer.

Product What's New

Draft Analysis Enhancements

Color tools for Draft Analysis are enhanced.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Part Modeling
User Interface Location	Click Analysis > Geometry > Draft
Processes and Initiatives	Detailed Design

Benefits and Description

With the Draft Analysis color tools you can:

- Set the color that indicates zero draft
- Show positive and negative draft in subsections with unique colors
- Set customized colors and save settings to a file
- Move the pointer over the plot and see a tooltip of the draft value
- See dynamic correlation between mouse pointer and graph value
- Reset to default values

Product What's New

Enhanced Excel analysis Feature

You can store an updated Excel file on regeneration.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Behavioral Modeling
User Interface Location	Click Analysis > External Analysis > Excel Analysis. In the Result Datums step, select Create: Yes at the option "Output file name with current values of inputs and outputs"
Processes and Initiatives	Detailed Design

Benefits and Description

You can update the Excel file parameters after regenerating the model, and then automatically store a copy of the Excel file with current parameter values.

Product What's New

Enhanced Torsion Spring

There are torsion springs with user-selectable endpoints, representing the attachment points for the torque arms of a torsion spring.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Mechanism Design & Dynamics
User Interface Location	Click Insert > Springs.
Processes and Initiatives	Detailed Design

Benefits and Description

You can select user-defined attachment points for the torsion arms of a simulated torsion spring. Some of the capabilities are:

- Defining current angle without having to define joint axis settings
- Selecting points or planes as torque arm attachments
- Applying current angle as unstretched angle to create a neutral spring at the current configuration

You can create more realistic torsion spring properties with attachment points similar to an actual installation.

Product What's New

Enhanced Workflow in Tolerance Analysis

Delete and substitute components and dimensions in an existing analysis.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Part Modeling
User Interface Location	Click Analysis > Tolerance Study. Delete dimensions or components in an existing Tolerance study.
Processes and Initiatives	Detailed Design

Benefits and Description

You can delete components and create voids in the tree view of Tolerance Analysis. To close the voids, select alternate component and dimension paths until the void is filled.

You do not have to roll back the analysis from the bottom up if something in the middle of the loop needs replacing.

Product What's New

Forces Depending on Forces

User-defined forces consider existing force and torque measures.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Mechanism Design & Dynamics
User Interface Location	Click Insert > Force Motors and then define the force motor or click insert > Force/Torque and then define the force or torque.
Processes and Initiatives	Detailed Design

Benefits and Description

You can create user-defined force motors and force and torques that are based on other measures that capture force or torque results.

Product What's New

Gap Close Element

You can magnetically close gaps in the model.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Other Functional Areas
User Interface Location	Click Analysis > Tolerance Study.
Processes and Initiatives	Detailed Design

Benefits and Description

You can close gaps in the model such that the analysis considers a selected void to have zero size. The gap is added to the size of the goal.

Product What's New

Keyframe Sequences Based on Explode States

There is a new animation type for animating between explode states.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Design Animation
User Interface Location	Click Animation > Animation. Select New > Explode.
Processes and Initiatives	Detailed Design

Benefits and Description

You can create keyframe sequences based on explode states, while applying all Animation tools such as View at time, Transparency at Time and so on.

Product What's New

New Generic Gear Type

There is a new gear type that enables any type of rotational and translational motion coupling.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Mechanism Design & Dynamics
User Interface Location	Click Insert > Gears. Select Generic.
Processes and Initiatives	Detailed Design

Benefits and Description

The Generic gear type enables easy setup of motion relationship between rotational and translational joint axes such as:

- Rotation - Rotation
- Rotation - Translation
- Translation - Translation

Generic gears can relate any joint axes in your model, enabling almost any joint axis relationship you need.

Product What's New

Performance Monitor

Monitor critical design measurements and safety margins to design requirements.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Behavioral Modeling
User Interface Location	Click Analysis > Performance Monitor.
Processes and Initiatives	Detailed Design

Benefits and Description

You can monitor a list of critical design measurements, such as Analysis Feature parameters, to ensure they are within design specifications. A Green, Yellow, or Red indicator appears, depending on whether your measure is safe, getting close to, or violating a design limit.

Product What's New

Reaction Measures on Ball and Bearing Joints

Reaction measures can be expressed in the X,Y,Z coordinates of the involved bodies.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Mechanism Design & Dynamics
User Interface Location	Click Analysis > Measures. Select Create New Measure > Connection Reaction. Select a ball or bearing connection.
Processes and Initiatives	Detailed Design

Benefits and Description

You can create measures that extract load reactions on ball and bearing joints, based on body coordinate systems of the connected bodies.

- For ball joints you can measure X,Y,Z and total reaction forces
- For bearing joints you can measure X, Y, Z, total, radial, radial X, radial Y and axial forces

Product What's New

Slot Motor

You can apply servo motor profiles to Slot connections.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Mechanism Design & Dynamics
User Interface Location	Click Insert > Servo Motors. Select a Slot Connection.
Processes and Initiatives	Detailed Design

Benefits and Description

You can apply any servo motor profile on a Slot connection for full control of the motion profile along the Slot connection trajectory.

Product What's New

Spur, Bevel, Worm and Rack and Pinion Gear Types

There are new gear types with entry fields for gear teeth manufacturing angles. The impact of manufacturing angles on gear loads are considered in the reaction load calculation

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Mechanism Design & Dynamics
User Interface Location	Click Insert > Gears. Select Spur, Bevel, Worm or Rack & Pinion.
Processes and Initiatives	Detailed Design

Benefits and Description

The new Spur, Bevel, Worm and Rack & Pinion gear types provide an enhanced workflow for fast and accurate gear connections. Some of the capabilities of these gears are:

- Automatic calculation of pitch diameter and pitch circle location of the driven gear
- Automatic calculation of bevel angles for bevel gears
- Pressure and helix angle gear properties
- Screw angle and number of worm spirals for Worm gear
- Reaction loads between gears calculated based on the torque transferred between the gears and the teeth manufacturing angles (Dynamic Analysis)
- Gear teeth loads shown as animated vectors where pitch diameters meet (Dynamic Analysis)
- Symbolic 3D preview of gear diameters and helix angle

With greater fidelity in defining gear interaction properties, these enhancements increase your productivity and provide more accurate results when working with gear-enabled mechanisms.

Product What's New

Statistical Design Studies

Statistical attributes can be applied to dimensions and parameters, enabling the calculation of resulting statistical parameters for any measurable goal in the model.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Behavioral Modeling
User Interface Location	Click Analysis > Statistical Design Study.
Processes and Initiatives	Detailed Design

Benefits and Description

You can analyze the statistical properties of almost any CAE measurement in your model, based on statistical distributions assigned to model dimensions or parameters.

In conjunction with other PTC CAE tools, use Statistical Design Studies to answer questions such as:

- How can statistical dispersion of material properties affect the stress and deformation of my design?
- How are the clearances between mechanism parts in motion affected by statistical effects on the size and shape of the parts?
- What is the standard deviation of the mass and volume of my design, given the statistical dispersion of some critical dimensions?
- What is the best combination of dimension values that will minimize the standard deviation of a desired design property?

Analyzing statistical effects on CAE measurements is an important asset in support of Six-Sigma initiatives, Strategic Design Planning, and Root Cause Analyses.

Product What's New

Tolerance Analysis Saved as Feature

You can save Tolerance Analysis measurement as a feature, with feature parameters representing Tolerance Analysis results.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Other Functional Areas
User Interface Location	Click Analysis > Tolerance Study. Select Feature as result type.
Processes and Initiatives	Detailed Design

Benefits and Description

You can define a Tolerance Analysis and choose if you want to save it as a Saved Analysis or as a feature.

Saving the analysis as a Feature enables you to place the analysis in the Model Tree and extract and monitor resulting parameters as required.

Product What's New

User-Defined Dimensions

Create annotation dimension on the fly if there are no existing dimensions or annotations

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Other Functional Areas
User Interface Location	Click Analysis > Tolerance Study.
Processes and Initiatives	Detailed Design

Benefits and Description

You can create annotation dimensions as needed from within the Tolerance Analysis tool.

This is useful if there are no parametric dimensions from which to build or continue building the Tolerance Analysis loop.

Product What's New

Virtual Components

Create placeholder components to simulate dimension properties that are not accounted for in the CAD model.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Other Functional Areas
User Interface Location	Click Analysis > Tolerance Study.
Processes and Initiatives	Detailed Design

Benefits and Description

You can create Virtual Components representing objects or properties that claim space and have dimensions, but are not modeled as 3D elements in the model.

Virtual components are useful when you want to include the effects of ceramic coating, paint, or simply a component of known size that is not yet placed in the assembly. You can create user defined Annotation dimensions that define the size of the Virtual component.

Product What's New

Volume Analysis on Closed Quilts

You can analyze volume and mass properties on closed quilts.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Simulation - Behavioral Modeling
User Interface Location	1. Click Analysis > Measure > Volume. Select the quilt. 2. Click Analysis > Model > Mass properties. Select the quilt.
Processes and Initiatives	Detailed Design

Benefits and Description

You can reference closed quilts while measuring volume and mass properties of your models.

Product What's New

Pro/ENGINEER Advanced Assembly

[Associative Solid Shrinkwrap-External Shrinkwrap Feature](#)

To easily copy geometry from a selection of models, select the Auto collect all Solid Surfaces Shrinkwrap option. You have the option to solidify the resulting geometry and to remove holes from selected surfaces.

[Default Envelope](#)

Use Default Envelope to represent an assembly with a single part.

[Granular Change Definition of Data Sharing Features](#)

The change of a copy geometry feature referencing a sketch or curve feature is determined on a more granular level. This reduces the risk of unintentionally changed objects.

[Inheritance Feature Enhancements](#)

The inheritance feature has improved support for model-based definition and toolkit.

[Simple Replace of Copied Model](#)

Quickly replace copied models by automapping the same feature IDs..

Product What's New

Pro/ENGINEER Advanced Rendering

[Advanced Rendering Overhaul](#)

There are significant improvements to Photolux.

[Appearance Overhaul](#)

The assignment, editing and managing of appearances is improved.

[New appearance and scene library](#)

New appearances and scenes have been added

[New appearance classes](#)

The appearance classes have been consolidated

[Perspective Enhancements](#)

Perspective view settings and controls are improved.

[Scene Definition is Easier to Use](#)

There are usability improvements to the scene definition.

[Shadow Catcher Support](#)

A shadow catcher appearance can now be added to the room walls.

Product What's New

Pro/ENGINEER

[3D Contact Connections](#)

There are 3D contact connections with material properties, and static and kinematic friction between planes, spheres, cylinders and vertices.

[3D Import Wizards and Data Exchange Profiles](#)

New wizards control import options for most 3D formats, providing you with immediate access to most commonly used import settings including use templates, enable ATB, and import as facet. Casual users can quickly choose a predefined profile to apply. Advanced users can tweak the details of a chosen profile or the current session settings.

[Actual Radius Dimensions in 2D Drawings](#)

When the actual or true radial dimension cannot be shown in a given drawing view, you can choose from a list of radii for the selected feature, and show it as a note.

[Analyze Effect of Changed Nominal Dimensions](#)

Edit Nominal Dimensions in the Tolerance Analysis Tree List.

[Annotation Out-of-plane Movement Options](#)

There are new options to control the planar location of an annotation element relative to its references.

[Appearance Overhaul](#)

The assignment, editing and managing of appearances is improved.

[Approximate Surface Offset](#)

Surfaces that cannot be offset, can be approximated.

[Belt Connections](#)

You can route closed belts around pulleys in a mechanism for use in drag and all kinematic and dynamic analysis types.

[Column Control in MODS Analysis Result Table](#)

You can configure column display in MODS Result tables.

[Curvature Continuous Rounds](#)

There is support for the creation of Curvature continuous rounds.

[Data Exchange of Nongeometric Content](#)

Data exchange of nongeometric content includes parameters, 3D notes, and annotations.

[Datum Points and Coordinate Systems from Mathcad Matrices](#)

Create datum points and coordinate systems from Analysis features that reference matrices in Mathcad.

[Datum Target Annotation Features](#)

You can create Datum targets in the 3D model as annotation features.

[Dependant Mirror Model Tree Designation](#)

Dependant mirror feature now a unique icon in the model tree.

[Design Studies of Parameters in Footer Features](#)

You can use parameters from Features in the Footer in Design Studies

[Dimensional Alignment Improvements](#)

In 3D you can align Dimensions. In 2D and 3D you can use dimensional alignment in more cases.

[Dimensional Tolerances Improvements](#)

The updated dimension properties user interface gives you more control over the tolerance.

[Draft Analysis Enhancements](#)

Color tools for Draft Analysis are enhanced.

[Draft Geometric Tolerances Created During Copy](#)

You can copy and paste geometric tolerances.

[Drawing Print Preview](#)

You can get a true preview of how your drawing will print or plot before committing to printing or exporting.

[Drawing Tree](#)

Selectable items in a drawing are enumerated on a Drawing Tree, improving usability.

[Drawing User Interface is Task-Based](#)

The Drawing user interface is task-based, improving workflows and efficiency.

[Edit Text Style of Multiple Annotations](#)

In 3D, you can change the text style of multiple annotations at the same time.

[Enhanced Excel analysis Feature](#)

You can store an updated Excel file on regeneration.

[Enhanced Torsion Spring](#)

There are torsion springs with user-selectable endpoints, representing the attachment points for the torque arms of a torsion spring.

[Enhanced Workflow in Tolerance Analysis](#)

Delete and substitute components and dimensions in an existing analysis.

[Expose Combined States at the Top-Level User Interface](#)

You can easily navigate between the combined states of a model without opening the View Manager. Combined or All states appear as tabs, each with a thumbnail preview, in the Graphics window.

[Fixed Location for Marquee](#)

There is a control for fixing the marquee location.

[Flatten Form Enhancements](#)

Flatten form capabilities are improved.

[Forces Depending on Forces](#)

User-defined forces consider existing force and torque measures.

[Foreshortened Radial Dimensions in 2D](#)

When a radius on a drawing is large, it is difficult to show an internal radial dimension. In 2D drawings, an internal radial dimension can appear as foreshortened.

[Gap Close Element](#)

You can magnetically close gaps in the model.

[Geometric Tolerances Can Reference Set Datums in Inheritance Features](#)

You can use set datum planes and axes contained in an inheritance feature, for referencing within a geometric tolerance.

[Hole Table Updates](#)

Hole tables in drawings are easier to use, and you can use hole tables to report the placement of additional features.

[Import DataDoctor Diagnostics Wizard](#)

The Diagnostics wizard extends the Troubleshooter dialog box with a series of new error, warning, and informational geometry checks unique to the Import DataDoctor (IDD) environment.

[Import DataDoctor Match Tool](#)

The Match tool is a new Repair mode tool that replaces poor quality n-sided imported surfaces with good quality n-sided surfaces.

[Import DataDoctor Workflow and Selection Improvements](#)

Multiple enhancements to Import DataDoctor modes, selection, filters, and features help streamline, simplify, and shorten the data repair process.

[Import DXF Files into Sketcher](#)

You can import DXF files into Sketcher

[Import Logs Update](#)

3D data exchange import log files are now in an XML format. They are rendered in the Pro/ENGINEER web browser and can be interactively interrogated.

[Improved Movement and Appearance of Angular Dimensions](#)

There are improvements to the appearance and movement of angular dimensions in 3D and 2D.

[Include Dimensions in Draft Groups](#)

You can include dimensions in 2D drawings in draft groups.

[Keyframe Sequences Based on Explode States](#)

There is a new animation type for animating between explode states.

[Layer States in View Manager](#)

You can create layer visibility states from the View Manager, and toggle the display of all layer-assigned content.

[Mirror Walls in Sheetmetal](#)

You can mirror flat and flange walls in Sheetmetal.

[Model Tree Improvements in Drawings](#)

The appearance of the Model Tree in drawings is updated and commands from shortcut menus are available. There is better interaction between the Model Tree, the drawing, and the Drawing Tree.

[New Generic Gear Type](#)

There is a new gear type that enables any type of rotational and translational motion coupling.

[Parameter Enhancements](#)

Parameters are more visible and it is easier to understand how to use them correctly. New system-level parameters are created for geometric tolerances.

[Performance Monitor](#)

Monitor critical design measurements and safety margins to design requirements.

[Perspective Enhancements](#)

Perspective view settings and controls are improved.

[Printing Improvements](#)

True Type fonts are correctly handled during printing. The string and font definition are passed from Pro/ENGINEER instead of rasterizing the output.

[Pro/ENGINEER Interface for Inventor](#)

The Pro/ENGINEER Interface for Inventor is an import-only processor you can use to read parts and assemblies for Autodesk Inventor 2009 and earlier.

[QuickPrint Flat-to-Screen Annotations](#)

You can output flat-to-screen content when using QuickPrint. The position of the content in the output is based on the relative x- and y-positions of the content in the Graphics window.

[Reaction Measures on Ball and Bearing Joints](#)

Reaction measures can be expressed in the X,Y,Z coordinates of the involved bodies.

[Realism Enhancement](#)

A transparent floor option is added to realtime rendering.

[Reference Model Cross-Section Edges for Annotations](#)

You can reference the temporary edges of a cross section as they appear in the model, for annotation attachment.

[Rights Management Improvements](#)

Digital Rights Management (DRM) is improved with a more flexible and efficient application of security policies and in providing a better awareness of protection for recipients.

[Rounded Dimension Display](#)

Dimensions appear with a different number of significant digits from the actual value of the dimension.

[Scene Definition is Easier to Use](#)

There are usability improvements to the scene definition.

[Sheet Tabs in Drawings](#)

Sheets appear in drawings as tabs.

[Single Arrow Option for Auxiliary Views](#)

When creating an auxiliary view, you can select a single arrow to appear in the parent view.

[Sketcher Constraints Improvements](#)

Sketcher constraints and workflows are more flexible. There are shortcut menus, object-action workflow, and a consolidated user interface. A new constraint type, equal dimension, is introduced.

[Sketcher Creation Tools Enhancements](#)

You can create new entity types in Sketcher, providing flexibility and speed in feature creation.

[Sketcher Dimensioning Improvements](#)

Several new dimension enhancements in Sketcher improve usability and flexibility.

[Sketcher Points and Coordinate Systems Improvements](#)

Sketcher points, centerlines, and coordinate systems are consolidated to simplify their application, and make them more intuitive and flexible.

[Slot Motor](#)

You can apply servo motor profiles to Slot connections.

[Spur, Bevel, Worm and Rack and Pinion Gear Types](#)

There are new gear types with entry fields for gear teeth manufacturing angles. The impact of manufacturing angles on gear loads are considered in the reaction load calculation

[Statistical Design Studies](#)

Statistical attributes can be applied to dimensions and parameters, enabling the calculation of resulting statistical parameters for any measurable goal in the model.

[Store and Retrieve PDF Configuration Files](#)

PDF generation from drawings is simplified because you can store and retrieve configuration files.

[Surface Finish Enhancements in 3D](#)

You can apply surface finishes annotation elements to more objects and there are new movement options.

[Surface Merge Configuration Option](#)

A new configuration option is added for merging surfaces.

[Target Areas for Annotations](#)

You can create target areas (supplemental geometry) in 3D for annotation referencing.

[Tolerance Analysis Saved as Feature](#)

You can save Tolerance Analysis measurement as a feature, with feature parameters representing Tolerance Analysis results.

[User-Defined Dimensions](#)

Create annotation dimension on the fly if there are no existing dimensions or annotations

[Virtual Components](#)

Create placeholder components to simulate dimension properties that are not accounted for in the CAD model.

[Volume Analysis on Closed Quilts](#)

You can analyze volume and mass properties on closed quilts.

[Warp Feature Enhancements](#)

There are general enhancements to the Warp feature.

[Witness Line Improvements in 2D Drawings](#)

Dimension witness lines in 2D drawings are improved to provide automatic clipping for created dimensions.

Product What's New

Pro/ENGINEER Foundation XE

[Animated Explode and Unexplode Sequences](#)

You can animate explode or unexploded sequences.

[Assembly Cut Enhancements](#)

Switch between the simplified and advanced assembly cut environments by selecting or clearing a checkbox. Cut challenging geometry with the Check Geometry Option.

[Assembly Explode Usability Enhancements](#)

You can reposition models in the new interface for Explode, by dragging and rotating them.

[Assembly Restructure Enhanced Reference Handling](#)

Enhanced reference handling simplifies restructuring by minimizing external references and dependencies.

[Assembly Restructure Usability Enhanced](#)

Improved usability simplifies assembly restructuring.

[Associative Solid Shrinkwrap, Assembly context Shrinkwrap feature](#)

To easily copy geometry from a selection of models, select the "Auto collect all Solid Surfaces" Shrinkwrap option. You have the option to solidify the resulting geometry as well as remove holes from selected surfaces.

[Autonumbering of New models in Copied Assembly](#)

When connected to a Windchill server, you can use autonumbering to name new models generated as a result of copying an assembly,

[Component Chooser](#)

Use dynamic preview and powerful selection tools to build component subsets within the component chooser. The Component Chooser is used to define simplified representations, envelopes, and shrinkwrap features.

[Dependant Mirror Model Tree Designation](#)

Dependant mirror feature now a unique icon in the model tree.

[Dynamic Component Mirror Preview](#)

Using dynamic view, you can explore alternatives prior to generating new models.

[Enhanced Component Copy and Paste](#)

Quickly paste components using temporary interfaces.

[Envelope Enhancements](#)

With the new component chooser, envelopes are faster and easier to define and maintain. You can quickly define simplified representations based on the envelope and represent an assembly with a single part.

[Failure Tolerance](#)

You can postpone resolving assembly failures so you can open and work with the assemblies with the missing information.

[Flatten Form Enhancements](#)

Flatten form capabilities are improved.

[Flexible Component Enhancements](#)

There is improved regeneration of flexible components.

[Graphics Representation Enhancements](#)

It is easier to select graphics representation models. You can store Component placement information in the graphics representation.

[Mirror Walls in Sheetmetal](#)

You can mirror flat and flange walls in Sheetmetal.

[Model Tree for Accessory Window](#)

You can select references in the Model Tree and control layers in the Layer Tree, specific to the accessory window.

[Open Generic -Shortcut Command](#)

You can open the generic of an instance from a shortcut menu. It is easy to identify an instance in the Model Tree when its generic name is appended.

[Quick Retrieval of Rule-Based Simplified Representations](#)

It is optional to evaluate rules when retrieving a simplified representation. Retrieving the simplified representation as it was last evaluated, can sometimes result in faster retrieval.

[Repeat -Shortcut Command](#)

You can quickly repeat a component placement using a command from a shortcut menu. During a repeat sequence, you can change the number of variable constraints. Select the constraints in the Model Tree, right-click and select Repeat to repeat a component.

[Restrictions Removed from Assembly Restructure](#)

You can restructure Simplified Representations of assemblies and components that could previously not be restructured.

[Retrieval Performance of Simplified Representations with Instances](#)

Enhanced Instance dependency retrieval is extended to simplified representations. Retrieving simplified representations with Family Table instances is faster.

[Simplified Accessory Window](#)

You can dock or undock the Accessory window. The docked accessory window (default) stays in the main window so it is not hidden behind other windows. You can easily restore a minimized Accessory window by clicking a glyph.

[Simplified Representation New Default Rule](#)

When creating a new Simplified Representation, components are excluded by default.

[Simplified Representation Usability Enhancements](#)

Use extended preview and information to easily create or select the correct simplified representation.

[Snap Single Constraint](#)

Snapping one constraint at a time simplifies drag and snap.

Product What's New

Windchill

[Autonumbering of New models in Copied Assembly](#)

When connected to a Windchill server, you can use autonumbering to name new models generated as a result of copying an assembly,

Product What's New

Windchill ProductPoint

[Autonumbering of New models in Copied Assembly](#)

When connected to a Windchill server, you can use autonumbering to name new models generated as a result of copying an assembly,

Product What's New

Pro/ENGINEER Interactive Surface Design

[General Improvements in Style](#)

There is improved usability in the Style feature.

[New N-sided surface](#)

A new n-sided surface is added.

[Style Curve Enhancements](#)

Style curve creation and editing tools are enhanced.

[Style Surface Enhancements](#)

The Style surface creation and editing tools are enhanced.

[Surface Edit Enhancements](#)

The Style Surface Edit functionality is enhanced.

Product What's New

Pro/ENGINEER Reverse Engineering

[Enhanced Surface Modify Command](#)

The Restyle Surface modify command is easier to use and more powerful.

Product What's New

Simulation - Mechanism Design & Dynamics

[3D Contact Connections](#)

There are 3D contact connections with material properties, and static and kinematic friction between planes, spheres, cylinders and vertices.

[Belt Connections](#)

You can route closed belts around pulleys in a mechanism for use in drag and all kinematic and dynamic analysis types.

[Enhanced Torsion Spring](#)

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Product What's New

Other Functional Areas

[Analyze Effect of Changed Nominal Dimensions](#)

Edit Nominal Dimensions in the Tolerance Analysis Tree List.

[Gap Close Element](#)

You can magnetically close gaps in the model.

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Create annotation dimension on the fly if there are no existing dimensions or annotations

[Virtual Components](#)

Create placeholder components to simulate dimension properties that are not accounted for in the CAD model.

Product What's New

Simulation - Behavioral Modeling

[Column Control in MODS Analysis Result Table](#)

You can configure column display in MODS Result tables.

[Datum Points and Coordinate Systems from Mathcad Matrices](#)

Create datum points and coordinate systems from Analysis features that reference matrices in Mathcad.

[Design Studies of Parameters in Footer Features](#)

You can use parameters from Features in the Footer in Design Studies

[Enhanced Excel analysis Feature](#)

You can store an updated Excel file on regeneration.

[Performance Monitor](#)

Monitor critical design measurements and safety margins to design requirements.

[Statistical Design Studies](#)

Statistical attributes can be applied to dimensions and parameters, enabling the calculation of resulting statistical parameters for any measurable goal in the model.

[Volume Analysis on Closed Quilts](#)

You can analyze volume and mass properties on closed quilts.

Product What's New

Part Modeling

[Approximate Surface Offset](#)

Surfaces that cannot be offset, can be approximated.

[Curvature Continuous Rounds](#)

There is support for the creation of Curvature continuous rounds.

[Draft Analysis Enhancements](#)

Color tools for Draft Analysis are enhanced.

[Enhanced Workflow in Tolerance Analysis](#)

Delete and substitute components and dimensions in an existing analysis.

[Import DXF Files into Sketcher](#)

You can import DXF files into Sketcher

[Sketcher Constraints Improvements](#)

Sketcher constraints and workflows are more flexible. There are shortcut menus, object-action workflow, and a consolidated user interface. A new constraint type, equal dimension, is introduced.

[Sketcher Creation Tools Enhancements](#)

You can create new entity types in Sketcher, providing flexibility and speed in feature creation.

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Several new dimension enhancements in Sketcher improve usability and flexibility.

[Sketcher Points and Coordinate Systems Improvements](#)

Sketcher points, centerlines, and coordinate systems are consolidated to simplify their application, and make them more intuitive and flexible.

[Surface Merge Configuration Option](#)

A new configuration option is added for merging surfaces.

Product What's New

Simulation - Design Animation

[Keyframe Sequences Based on Explode States](#)

There is a new animation type for animating between explode states.

Product What's New

3D Import Wizards and Data Exchange Profiles

New wizards control import options for most 3D formats, providing you with immediate access to most commonly used import settings including use templates, enable ATB, and import as facet. Casual users can quickly choose a predefined profile to apply. Advanced users can tweak the details of a chosen profile or the current session settings.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Import Data Doctor
User Interface Location	3D Wizards launch when opening supported nonnative 3D file type. To edit a profile, click Tools > Data Exchange > Import Profile Editor.
Processes and Initiatives	

Benefits and Description

Data exchange import profiles (*.dip) are in industry standard XML format. Profiles supplant many of the data-exchange related configuration options. Using the Profiles Editor, you can easily migrate from existing import-related configuration options to profiles. Make as many profiles as you need and create multiple profiles per file format.

Profiles are designed for easy administration and to be part of the company standards. To specify a default profile, set the configuration option `intf_in_profile_default`. To specify a profile directory, set the configuration option `intf_profile_dir`. Distributed Pro/BATCH also supports the use of profiles.

This functionality benefits:

- Administrators--Centrally configure, manage, and control profile settings for users. Use profiles for flexibility when configuring file format, source software, or software version.
- Casual users--Follow a simple workflow and apply company recommended settings from a list.

- Advanced users--Get immediate access to most commonly used 3D import options and can change settings on-the fly, during import operations.

Product What's New

Actual Radius Dimensions in 2D Drawings

When the actual or true radial dimension cannot be shown in a given drawing view, you can choose from a list of radii for the selected feature, and show it as a note.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	In drawings, click the Annotate tab and select Actual Radius.
Processes and Initiatives	

Benefits and Description

In certain drawing views, a feature or object that was defined using one or more radial dimensions, may not be positioned so the radius can be measured or identified. When this is the case, you can create an Actual Radius dimension. Then, select from a list of available radii for the feature selected, place as an attached note.

Product What's New

Annotation Out-of-plane Movement Options

There are new options to control the planar location of an annotation element relative to its references.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	Various
Processes and Initiatives	

Benefits and Description

You can use the following options to move annotations out-of-plane from their original location:

- Force Annotations to Plane--When setting the model's Active Annotation Orientation, forces the resulting annotation to the planar reference, regardless of the location of the references.
- Move Dims to Plane--When multiple dimensions are selected, forces all the dimensions to be positioned in the planar space of the first-selected dimension. This command is on a shortcut menu.
- Move to Plane--When one or more annotations are selected, prompts you for a new planar reference, resulting in the movement of all annotations to the selected reference. This command is on a shortcut menu.

Z-extension lines (dashed, gray lines) are created when the attachment point moves off its reference to maintain a clear relationship between the two.

Product What's New

Approximate Surface Offset

Surfaces that cannot be offset, can be approximated.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Part Modeling
User Interface Location	Edit > Offset
Processes and Initiatives	

Benefits and Description

The special handling capabilities of surface offset are improved:

- In addition to excluding patches during the offset, you can choose to approximate these patches.
- When you cannot offset an internal patch of a quilt due to minimum radius restrictions, Pro/ENGINEER attempts to approximate that patch with a single surface.
- You can choose which patches to exclude and which to approximate, and whether or not to attach the approximated patches.

Product What's New

Curvature Continuous Rounds

There is support for the creation of Curvature continuous rounds.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Part Modeling
User Interface Location	Insert > Round... > Sets > C2 Continuous or D1XD2 C2
Processes and Initiatives	

Benefits and Description

The following enhancements have been made to Rounds:

- You can create Rounds with curvature continuity along their boundaries.
- Curvature continuous rounds can be constant or variable.
- A shape factor adjusts the curvature influence.
- You can define Constant curvature continuous rounds with different leg lengths on each side.

Product What's New

Data Exchange of Nongeometric Content

Data exchange of nongeometric content includes parameters, 3D notes, and annotations.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Interface
User Interface Location	Click File > Save a Copy and then in the Type list select a supported 3D file type and click OK.
Processes and Initiatives	

Benefits and Description

Annotation support includes notes, dimensions, surface finish, geometric tolerances, and set datums, and symbols. Annotations and 3D notes are exported and imported as graphical polyline representations. The level of parameter support varies per supported file type:

- Neutral supports model and entity parameters only
- STEP does not support parameters
- ProductView, JT, and PDF support model parameters only

Import of nongeometric content is supported for Neutral, STEP, ProductView and JT formats. Export of nongeometric content is supported for Neutral, STEP, ProductView, and PDF formats.

Data exchange of nongeometric content:

- Expands the amount of data available for long-term archival
- Provides the support required to implement model-based definition with customers and suppliers using different CAD systems

Product What's New

Datum Target Annotation Features

You can create Datum targets in the 3D model as annotation features.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	Click Insert > Annotations > Datum Target Annotation Feature.
Processes and Initiatives	

Benefits and Description

You can annotate your models with datum targets. A new annotation feature type is available, containing only the relevant elements of the datum target structure:

- Symbolic target annotations
- A Single datum feature annotation.

In conjunction with new target areas, you can assign target annotations to datum points, datum curves, surfaces, edges, axes, and target areas. You can use the top portion of a target symbol to display a dimension representing the area. In the symbol definition you are prompted for this dimensional assignment.

Because the symbol annotations and the datum feature annotation are in the same annotation feature:

- If the datum feature is selected, the corresponding targets that define it are highlighted.
- If a target is selected, the other related targets are also highlighted.

Product What's New

Dimensional Alignment Improvements

In 3D you can align Dimensions. In 2D and 3D you can use dimensional alignment in more cases.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	In 2D and 3D. select dimensions right-click and select Align Dimensions from the shortcut menu.
Processes and Initiatives	

Benefits and Description

You can align linear and angular dimension annotations in 3D. Select a number of dimensions to align and the alignment occurs based on the position of the first-selected dimension. This alignment in 3D works for dimensions that are in same or parallel annotation planes.

In 2D and 3D angular dimensions, there is no dimensional overlap when in the same plane.

Product What's New

Dimensional Tolerances Improvements

The updated dimension properties user interface gives you more control over the tolerance.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	Right-click a dimension and select Dimension Properties from the shortcut menu.
Processes and Initiatives	

Benefits and Description

You can better control tolerances with:

- More upper and lower tolerance values during adjustment.
- Positive and negative signs that appear in the upper and lower tolerance boxes, simplifying entry, such as with $+/+$ tolerances.
- Separate decimal place settings for the tolerance and dimension.
- Configuration options that control the default tolerance digits and lead trail zeros.
- Showing only the tolerance value. You can use additional text such as a prefix, to appear as an overridden dimension with tolerance.

Product What's New

Draft Geometric Tolerances Created During Copy

You can copy and paste geometric tolerances.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	Right-click a geometric tolerance and select Copy from the shortcut menu.
Processes and Initiatives	

Benefits and Description

You can copy and paste draft geometric tolerances. When pasting, the GTOL is placed with a free attachment, and you can reattach it to any model edge or draft entity.

Product What's New

Drawing Print Preview

You can get a true preview of how your drawing will print or plot before committing to printing or exporting.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	Click the Publish tab and select the type of output. Click Settings and then click Preview.
Processes and Initiatives	

Benefits and Description

You can save paper by first confirming your prints using Print Preview functionality.

When previewing, the following attributes defined in the current printer configuration, are considered:

- Line weight and style
- Line priority
- Color
- Pen tables
- Scale and sheet position

When previewing, the Graphics window is redrawn to represent the output type selected (Print/Plot, DXF, IGES, and so on). When set to Print/Plot, a white-background paper-space appears, representing margins and resulting drawing plot. In Preview mode, you can pan and zoom in the Graphics window.

The previous Print and Printer Configuration dialog boxes are merged, renamed Settings

and are located in the Publish tab.

Product What's New

Drawing Tree

Selectable items in a drawing are enumerated on a Drawing Tree, improving usability.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	The drawing tree appears with the Model Tree in the Model Tree area.
Processes and Initiatives	

Benefits and Description

Selectable drawing objects appear a tree hierarchy. This Drawing Tree helps make you better aware of the following drawing items:

- Sheets
- Views
- Created (drawing) annotations
- Shown (model) annotations
- Datums
- Snap lines
- Groups
- Sections
- Draft entities
- Objects
-

Overlays

-

Tables

The content of the drawing tree varies depending on the tab selected, simplifying the tree structure. Objects are highlighted in the Graphics window when you select them from the Graphics window or the Drawing Tree. You can right-click in the Drawing Tree to select commands from a shortcut menu.

Icons in the tree indicate whether the drawing item is shown or hidden.

Product What's New

Drawing User Interface is Task-Based

The Drawing user interface is task-based, improving workflows and efficiency.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	
Processes and Initiatives	

Benefits and Description

Drawing commands are reorganized into a ribbon-style interface. Drawing-specific top-level commands are in logically organized tabs within the ribbon. Only commands and workflows appropriate for the current task are available, making the interface easier to use.

The selection filter and selectable objects also change depending on the current task. There is a drawing tree you can use to identify the selectable drawing objects in a logical tree form.

Tasks are grouped under the following tabs:

- Layout
- Table
- Annotate
- Sketch
- Autobuildz
- Review
- Publish

Product What's New

Edit Text Style of Multiple Annotations

In 3D, you can change the text style of multiple annotations at the same time.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	Right-click an annotation in the Graphics window or the Model Tree and select Text Style from the shortcut menu.
Processes and Initiatives	

Benefits and Description

You can select multiple annotation types and from different sources and alter their text style simultaneously. This helps to quickly create a consistent presentation when, for example, the style of component annotations differs from the style of other annotations in an assembly.

Product What's New

Expose Combined States at the Top-Level User Interface

You can easily navigate between the combined states of a model without opening the View Manager. Combined or All states appear as tabs, each with a thumbnail preview, in the Graphics window.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	Click View > View Manager and then click the All tab. Select the Display combined views check box.
Processes and Initiatives	

Benefits and Description

After you create a combined state from the View Manager, it appears as tab at the bottom of the Graphics window. For quick accessibility, a new tab appears for each combined state you create..

If you place the pointer on a tab, a small graphical preview of the combined state appears. Right-click a tab to select commands from a shortcut menu, such as Rename. Renaming a tab changes the name of the combined state. You can reorder tabs by dragging them.

Product What's New

Fixed Location for Marquee

There is a control for fixing the marquee location.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Surfacing - WARP
User Interface Location	Insert > Warp
Processes and Initiatives	

Benefits and Description

The marquee for Stretch, Bend, Twist and Sculpt can be fixed to existing geometry. The Warp feature can change while the overall intent of the design is maintained.

Product What's New

Foreshortened Radial Dimensions in 2D

When a radius on a drawing is large, it is difficult to show an internal radial dimension. In 2D drawings, an internal radial dimension can appear as foreshortened.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	In a drawing, click the Annotation tab and then click the icon for Z- Radius Dimension.
Processes and Initiatives	

Benefits and Description

When creating a foreshortened radial dimension, the resulting structure is Z shaped with drag handles for the placement of the different line segments and the virtual center of the arc. You can attach the virtual center to existing entities, or it can be free-floating. You can toggle the symbol for the virtual center between a cross, a filled circle, or none.

Product What's New

Geometric Tolerances Can Reference Set Datums in Inheritance Features

You can use set datum planes and axes contained in an inheritance feature, for referencing within a geometric tolerance.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	For 3D, click Insert > Annotations > Geometric Tolerance. For 2D, click the Annotate tab. On the Insert panel, click the Geometric Tolerance icon.
Processes and Initiatives	

Benefits and Description

When setting the datum references for a geometric tolerance (GTOL), you can select inheritance feature set datums from the Graphics window or the Model Tree. After identifying the set datum, it appears in the Datum list on the GTOL user interface. This is consistent with the referencing of a part set-datum in an assembly.

When inheritance features with set datums are added to a model, and the model contains datums with the same name, then the added datums are renamed within the context of the model.

Product What's New

Hole Table Updates

Hole tables in drawings are easier to use, and you can use hole tables to report the placement of additional features.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	In a drawing, click the Table tab and then click Hole Tables.
Processes and Initiatives	

Benefits and Description

There is a new user interface for setting up the rows and columns of a Hole Table. Hole tables are enhanced to support:

- Automatic inclusion of extrude and revolve cuts in the table
- Automatic inclusion of SMT cuts with punch axis points in the table
- Customization of the standard table columns
- Deletion of features from existing tables

Product What's New

Import DataDoctor Diagnostics Wizard

The Diagnostics wizard extends the Troubleshooter dialog box with a series of new error, warning, and informational geometry checks unique to the Import DataDoctor (IDD) environment.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Import Data Doctor
User Interface Location	In Import DataDoctor click Info > Geometry Checks. From the Troubleshooter dialog box, click Edit > IDD Settings.
Processes and Initiatives	

Benefits and Description

In the IDD environment, the Troubleshooter is asynchronous. You specify which geometry checks to perform and in some instances threshold values. The analyses and diagnostics are performed or updated when the Troubleshooter starts, settings are modified, or imported geometry is changed in the IDD environment. The diagnostics correspond to the active geometry in the following ways:

- The troubleshooter analyzes the imported geometry and the list of resulting geometry checks appears in the Troubleshooter dialog box.
- When you select an item in the Troubleshooter dialog box, the corresponding geometry highlights and there is a zoomed in view of the problem. In the Troubleshooter dialog box there is a description of the problem and suggestions on how to resolve it.
- You can right-click the identified problem in the Troubleshooter dialog box, to select the problem geometry.

Diagnostics that you can perform are:

1. Bad surfaces (concave domain or boundaries with parallel or almost parallel boundaries) (error)
2. Bad two-sided edges (error)
3. Bad wireframe curves (error)
4. Poorly tessellated edges (warning)
5. Small loops of one-sided edges (warning)
6. Short one-sided edges (warning)
7. Bad vertices (warning)
8. Unsatisfied wireframe(info)
9. Unsatisfied tangency (info)
10. Gaps not added to wireframe (info).
11. Almost tangent edges (info)
12. Small surfaces (info).
13. Narrow surfaces (info)

Diagnostics 5, 6, 10, 11, 12 and 13 require that you specify some distance. Default values based on model accuracy are provided.

The Troubleshooter in Import DataDoctor helps you to focus on important geometry issues. Repair time is significantly reduced by providing immediate visual and instructional feedback for repairing the geometry check.

Product What's New

Import DataDoctor Match Tool

The Match tool is a new Repair mode tool that replaces poor quality n-sided imported surfaces with good quality n-sided surfaces.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Import Data Doctor
User Interface Location	In Import DataDoctor click Repair mode, select surfaces or wireframe, and click Edit > Match.
Processes and Initiatives	

Benefits and Description

The user interface and workflow for the Match tool is similar to that of the Repair tool. The Repair tool requires that surfaces with more than 4 sides be replaced by multiple 4-sided patches before they can be repaired. With the Match tool, you can replace poor quality n-sided surfaces with good quality n-sided surfaces in imported geometry without the need to split the surface in 4-sided patches.

The Match tool simplifies repair of surfaces with more than four sides by eliminating the requirement to split n-sided surfaces into multiple 4-sided surfaces. This shortens the repair process for n-sided surface patches.

Product What's New

Import DataDoctor Workflow and Selection Improvements

Multiple enhancements to Import DataDoctor modes, selection, filters, and features help streamline, simplify, and shorten the data repair process.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Import Data Doctor
User Interface Location	After selecting an imported feature, right click and select Edit Definition > Geometry > Import DataDoctor.
Processes and Initiatives	

Benefits and Description

There is more consistency between Repair and Modify modes. Repair mode includes the commands Move Vertex, Extrapolate, and Modify Curve/Surface. Selection filters include the topological conditions satisfied wireframe, unsatisfied wireframe, and unsatisfied tangency options. Using shortcut menus you can select a parent and switch between modes. All object-action commands in the active mode are available on the shortcut menu. You can select by crossing the boundaries of selection boxes, in addition to selecting within the box. The active component or surface set limits the scope of edit operations that act on vertices, edges and UV curves to: Move Vertex, Modify Edge, Replace, Modify UV curve, Split Crv/Wfm, and Merge Crv/Wfm.

Improvements to Import DataDoctor features include:

- Merging multiple segments in a single operation using the Merge Curve/Wireframe tool
- Optional tangency repair using the Repair and Match tools
- Availability of most recently used value lists when searching for slivers and gaps
- Acceptance of edges of other surfaces or 3D curves as input when replacing one-sided edges. This results in the automatic projection of these edges or 3D curves onto the surface
- Repair command accepts Wireframe curves as input.

Improvements simplify the data repair workflow reducing time to repair data.

Product What's New

Import DXF Files into Sketcher

You can import DXF files into Sketcher

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Part Modeling
User Interface Location	Sketcher
Processes and Initiatives	

Benefits and Description

You can import DXF files into Sketcher. When the option to create sketch groups on import is set to yes, importing a DXF file creates a single group, regular, or draft.

DXF files in the working directory or in the Sketcher shape directory appear in the sketcher palette and are available for preview and import from the palette.

Product What's New

Import Logs Update

3D data exchange import log files are now in an XML format. They are rendered in the Pro/ENGINEER web browser and can be interactively interrogated.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Interface
User Interface Location	Click File > Open and open a supported format file. Click Info > Data Exchange > Import Log File.
Processes and Initiatives	

Benefits and Description

During import operations data exchange import logs are written in an XML format (*.xml). For most import operations you can select one of the following options:

- none - No log file (default)
- short - Header information, errors, and warnings sections only
- long - All recorded log information

Set the configuration option `intf3d_generate_log_file` to short, long, or no, to change the default log file.

If a log file is not generated during import, the log information is stored in the Pro/ENGINEER model for the session. You can generate a log file after import is complete. Subsequent imports into the same model overwrite this temporary information. Logging information is not saved in the model and is discarded when you exit Pro/ENGINEER.

You can view and dynamically interact with import logs in the embedded Pro/ENGINEER browser. The XML content is transformed and appears in HTML with icons to identify import successes, errors, and warnings. You can dynamically expand or collapse sections of the log and sort by columns within the sections of the log. At any time you can save the display of log file to a static HTML file visible in a standalone browser.

Logging options in the import user interface and logging information stored for the Pro/ENGINEER session provide a higher level of workflow flexibility and forgiveness while streamlining the import process. Log files conform to industry standard XML format and enable dynamic interrogation of the information. The HTML display draws attention to import errors and warnings, directs you to areas of concern, and shortens the log analysis process.

Product What's New

Improved Movement and Appearance of Angular Dimensions

There are improvements to the appearance and movement of angular dimensions in 3D and 2D.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	
Processes and Initiatives	

Benefits and Description

The appearance of angular dimensions is improved in 3D and 2D. The dimension leader arc is fully broken around the dimension so there is no overlap. The witness lines break around overlapping dimension text, even if additional text is used in the dimension properties. These enhancements also apply to linear dimensions.

You can use the Move Text command in 2D and 3D with linear dimensions. The Move Text command keeps the witness line and a leader arc position fixed, and repositions the dimension text.

Product What's New

Include Dimensions in Draft Groups

You can include dimensions in 2D drawings in draft groups.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	Click Edit > Group > Draft Group.
Processes and Initiatives	

Benefits and Description

Just as you can with draft entities, you can add or remove all types of dimensions from groups. When dimensions are in a group, they move with the group as much as possible. For example, dimensions with references that are not in the group, move only their text.

Product What's New

Layer States in View Manager

You can create layer visibility states from the View Manager, and toggle the display of all layer-assigned content.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	Click View > View Manager and click the Layers tab.
Processes and Initiatives	

Benefits and Description

You can create layer visibility states from the View Manager and retrieve the states from the Layer Tree or the View Manager. You can assign a layer state to a combined or All state during definition, with corresponding retrieval when the combined state is selected.

Layer states set and restore the layer visibility status within parts, and within assemblies at the assembly and component levels. Layers are used to more efficiently organize annotation content.

When you save a model you also save the state of all the layers. Layer states do not interfere with this functionality.

Product What's New

Model Tree Improvements in Drawings

The appearance of the Model Tree in drawings is updated and commands from shortcut menus are available. There is better interaction between the Model Tree, the drawing, and the Drawing Tree.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	The Model Tree with the Drawing Tree is located on the left side of the Graphics window.
Processes and Initiatives	

Benefits and Description

Improvements to the Model Tree when you are working in drawings are:

- Slider bar for simultaneous adjustment of the Drawing and Model trees
- Quick collapse and expansion with horizontal and vertical scroll bars
- Quick access to drawing models and available representations
- A glyph indicating which model annotations are shown in the drawing
- Removal of the Use Model in Tree command
- Addition of the Active Model View command for any given view. When set, the Model Tree reflects the selected view and any views that reflect the active model are highlighted.

Product What's New

Parameter Enhancements

Parameters are more visible and it is easier to understand how to use them correctly. New system-level parameters are created for geometric tolerances.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	Click Tools > Parameters.
Processes and Initiatives	

Benefits and Description

You can run a conflict report between the parameter definition in the object and the parameter definition in the external file. If the parameter type of the two is consistent, you can choose Update Restriction Definition to update your nonrestricted parameters with restrictions. This guarantees the definition in the model matches one of the acceptable predetermined values for the parameter.

You can see and export a list of all parameters within the chosen scope. In the Parameter dialog box, use a new filter option to see and act on all subfeature parameters, from the selected object level and below. You can create and store custom parameter filters.

All extensible values from a geometric tolerance (GTOL) automatically create system-level parameters. As a result, you can set and alter the GTOL through parameter setting, or through toolkit interaction.

While editing parameters, you can regenerate the model when the Parameters dialog box is open.

Product What's New

Printing Improvements

True Type fonts are correctly handled during printing. The string and font definition are passed from Pro/ENGINEER instead of rasterizing the output.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	In a drawing, click the Publish tab and then click Print.
Processes and Initiatives	

Benefits and Description

When True Type and Open Type fonts are used with the Windows printer driver or postscript driver, the resulting quality of the output is improved and the plot files are smaller.

A new configuration controls this during output.

The supported device list is externalized in an XML file so local administrators or PTC can add additional devices (along with their device settings) in the future. Several obsolete pen-plotters are removed from this list, and a generic HP-GL device type is added.

Product What's New

Pro/ENGINEER Interface for Inventor

The Pro/ENGINEER Interface for Inventor is an import-only processor you can use to read parts and assemblies for Autodesk Inventor 2009 and earlier.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Interface
User Interface Location	Click File > Open and then in the Type list select Inventor Part (*.ipt) or Inventor Assembly (*.iam).
Processes and Initiatives	

Benefits and Description

This interface is included in the base package of Pro/ENGINEER. It requires an installation and licensing of Autodesk Inventor 2008 or 2009 on the same computer as Pro/ENGINEER. If Autodesk Inventor 2008 is the latest installed version, you cannot open Autodesk Inventor 2009 files. You can import into Pro/ENGINEER Wildfire 5.0, earlier releases of Autodesk Inventor files that are upward compatible with the configured Autodesk Inventor version.

Pro/ENGINEER Interface for Inventor supports import of exact geometry, datums (planes, axes and points), curves, colors, and layers. Batch import of Inventor files is supported using Distributed Pro/BATCH.

The benefits of Pro/ENGINEER Interface for Inventor are:

- Data exchange is simplified by eliminating the need for Autodesk Inventor users to generate a non-native format.
- Quality of imported data is improved.
- Color and layer support provides additional clarity and control of data organization.
-

Distributed Pro/BATCH support provides better use of resources, with automated bulk import processing without manual intervention.

Product What's New

QuickPrint Flat-to-Screen Annotations

You can output flat-to-screen content when using QuickPrint. The position of the content in the output is based on the relative x- and y-positions of the content in the Graphics window.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	Click File > QuickPrint and then select the Print Flat to Screen Annotations check box.
Processes and Initiatives	

Benefits and Description

From the QuickPrint dialog box, you can select to print the currently-visible flat-to-screen annotation content. The position of the output is derived from the ratio of the x and y positions of the annotation to the overall x and y widths of the Graphics window.

Product What's New

Reference Model Cross-Section Edges for Annotations

You can reference the temporary edges of a cross section as they appear in the model, for annotation attachment.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	Click View > View Manager and then click the Xsec tab.
Processes and Initiatives	

Benefits and Description

When a section is created in the model, you can use the temporary edges that appear when the section is visible, for annotation attachment. Edges are available when the section appears, regardless of whether the section is active.

When the active or inactive section is not visible, referenced annotations are not visible.

Product What's New

Rights Management Improvements

Digital Rights Management (DRM) is improved with a more flexible and efficient application of security policies and in providing a better awareness of protection for recipients.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Digital Rights Management
User Interface Location	Click File > Security
Processes and Initiatives	

Benefits and Description

The Apply Policy dialog box includes new options. When applying a security policy, the publisher can:

- Generate or choose not to generate a new key on the policy server
- Change which policy is protecting the object
- Protect or secure originally unprotected components of an assembly
- Assign a server key to the entire assembly or to individual components
- Immediately view and alter the members of a policy, and their rights

A save operation provides a minor save of the object, so the same server key controls it and its previous iteration.

Recipients are made more aware of their session restrictions with:

- A DRM lock in the status bar
- An Offline Synchronization indicator, which establishes whether they can safely take

their Pro/ENGINEER model offline and continue to work.

Product What's New

Rounded Dimension Display

Dimensions appear with a different number of significant digits from the actual value of the dimension.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	In the Dimension Properties dialog box, select the Rounded Displayed Value check box. In Sketcher, right-click and select Round Displayed Value from the shortcut menu..
Processes and Initiatives	

Benefits and Description

You can fine-tune the significant digits that appear while modeling, to a different level of precision. This is available in Sketcher, the model, or the drawing. Select the Round Displayed Value check box so the dimension maintains its actual value, but with the number of decimal places set in Decimal Places,

Sketcher does not have its own setting for default decimal places. The configuration option `default_dec_places` controls the default decimal places throughout the application.

New options control the default (rounded) appearance of dimensions and whether Sketcher strengthens dimensions to the weak dimension's true significant digits, or to the setting for default decimal places.

Product What's New

Sheet Tabs in Drawings

Sheets appear in drawings as tabs.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	Drawing sheet tabs appear at the bottom of the Graphics window in Drawing, Format, Layout, and Diagram modes.
Processes and Initiatives	

Benefits and Description

Drawing sheets appear as tabs across the bottom of the Graphics window. Right-click a tab to select commands such as New Sheet, Rename, Update Sheets, Setup, and Delete, from a shortcut menu. Two new commands, Move or Copy, and Select All are added.

Product What's New

Single Arrow Option for Auxiliary Views

When creating an auxiliary view, you can select a single arrow to appear in the parent view.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	In the View Properties for the auxiliary view, under View Type select Single.
Processes and Initiatives	

Benefits and Description

To better support worldwide drawing standards, an option is available for auxiliary view arrows. When showing the projection line on the parent view, you can select no arrows, a single arrow, or double arrows.

Detail setup options control the format of the auxiliary view note, and the positioning of the projection arrows relative to the selected reference during creation.

Product What's New

Sketcher Constraints Improvements

Sketcher constraints and workflows are more flexible. There are shortcut menus, object-action workflow, and a consolidated user interface. A new constraint type, equal dimension, is introduced.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Part Modeling
User Interface Location	Sketcher
Processes and Initiatives	

Benefits and Description

You can use either of these workflows for creating constraints:

- Object-action—Select the entities, right-click, and select available constraints from the shortcut menu. Available constraints depend on the entities selected.
- Action-object—Select the constraints first and then select the entity.

While sketching and dragging sketched entities, new key sequences and mouse clicks speed up constraint creation. During sketching, use consecutive right-clicks to toggle through locking, disabling, or enabling the constraint. Press and hold down SHIFT to disable all constraints. During dragging, press and hold down SHIFT to enable the offering and accept constraints.

You can apply the new constraint type, equal dimension, to any dimensions of the same type. Applied in the same way as equal length or diameters, an E constraint (E for equal dimension) appears.

Product What's New

Sketcher Creation Tools Enhancements

You can create new entity types in Sketcher, providing flexibility and speed in feature creation.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Part Modeling
User Interface Location	Sketcher
Processes and Initiatives	

Benefits and Description

You can create the following Sketcher entity types:

- Slanted ellipse created by either establishing the major axis endpoints or by defining the center and one end of the major axis.
- Slanted rectangle
- Parallelogram
- Chamfer with or without intersecting construction lines

In addition to using or offsetting an existing model edge in a sketch, you can thicken an edge resulting in two parallel entities a stated distance apart and a stated distance from the reference.

Use commands from shortcut menus to quickly access sketcher options and references.

Product What's New

Sketcher Dimensioning Improvements

Several new dimension enhancements in Sketcher improve usability and flexibility.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Part Modeling
User Interface Location	Sketcher
Processes and Initiatives	

Benefits and Description

With enhanced Sketcher dimensioning you can:

- Create a total included angle about an axis of revolution
- Create an arc length dimension, with appropriate symbols
- Toggle between a diameter, radius, and linear diameter dimension
- Create a linear dimension tangent to two arcs in a specified direction
- Toggle between a reference and driving dimension
- Create a new perimeter dimension for a loop or chain
- Create any type of dimension (Normal, Perimeter, Reference, or Baseline) by clicking the dimension icon on the toolbar

Product What's New

Sketcher Points and Coordinate Systems Improvements

Sketcher points, centerlines, and coordinate systems are consolidated to simplify their application, and make them more intuitive and flexible.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Part Modeling
User Interface Location	Sketcher
Processes and Initiatives	

Benefits and Description

Construction points are used only within the sketch. Depending on how the sketch is used, geometry points can produce a datum point or axes in the resulting model. Geometry points can create axes during extrusion, therefore, axis points are eliminated. Geometry points are saved with the section, improving reuse of common designs, such as the palette.

There is a single workflow for centerline creation and you can toggle between geometry and construction. Use construction centerlines only within the sketch. Depending on how the sketch is used, geometry centerlines can produce datum axes in the model. For consistency in revolved features, you can use only geometry centerlines as axes of revolution.

A new type of coordinate system (CSYS) is introduced, and the existing horizontal-vertical (HV) CSYS inside sketches is updated. You can create a new rotatable geometry CSYS (rotatable around the z-axis only; the x- and y-axes remain in the sketch plane) within a sketch, resulting in a true CSYS within the resulting model. The existing HV CSYS, available only within in a sketch is updated so it defaults to HV, but can be rotated.

Product What's New

Store and Retrieve PDF Configuration Files

PDF generation from drawings is simplified because you can store and retrieve configuration files.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	In a drawing, click the Publish tab.
Processes and Initiatives	

Benefits and Description

PDF generation from a drawing is an exporting operation. You can save to an XML-based PDF configuration file from the PDF settings dialog box. Settings in the configuration file do not need to be reestablished for every output, making it easier to store and retrieve.

Set the configuration option `intf_profile_dir` to specify a default profile folder. The profile folder stores import or export configuration files.

Product What's New

Surface Finish Enhancements in 3D

You can apply surface finishes annotation elements to more objects and there are new movement options.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	Click Insert > Annotations > Annotation Feature and then click Surface Finish.
Processes and Initiatives	

Benefits and Description

You can drag surface finishes with an entity or normal attachment within the extents of their attachment reference. If the reference is planar, you can drag the surface finish accompanied by the automatic creation of extension lines.

You can attach surface finishes to assembly-level quilts. As with notes attached to surface features, surface finishes propagate through user-defined features and you can see them in drawings.

Product What's New

Surface Merge Configuration Option

A new configuration option is added for merging surfaces.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Part Modeling
User Interface Location	Edit > Merge
Processes and Initiatives	

Benefits and Description

The configuration option, `merge_type`, sets the default merge calculation in a Surface Merge feature. Set `merge_type` to intersect (the default) or join.

Product What's New

Target Areas for Annotations

You can create target areas (supplemental geometry) in 3D for annotation referencing.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	3D Drawings
User Interface Location	Click Insert > Cosmetic > Designated Area.
Processes and Initiatives	

Benefits and Description

There are a few ways to create target areas. The area created must divide the model by specifying a closed-loop chain that resides on the model, a sketch plane, or a quilt.

After establishing the target area, you can apply different appearances and assign parameters to it and you can run queries to identify them. Annotations can reference a target area truly as a different reference, even though it may be coincident with another surface.

The most common use for target areas for annotations is to establish a surface finish for the target area that is different from the surface finish assigned to the rest of the surfaces.

Product What's New

Warp Feature Enhancements

There are general enhancements to the Warp feature.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	Surfacing - WARP
User Interface Location	Insert > Warp
Processes and Initiatives	

Benefits and Description

The jack inside Warp follows the same practice as Style, resulting in improved control over placement and manipulation operations.

The internal mechanism for Warp references uses intent objects and alternate reference controls, resulting in more robust reference handling of post warp feature references.

Product What's New

Witness Line Improvements in 2D Drawings

Dimension witness lines in 2D drawings are improved to provide automatic clipping for created dimensions.

Product Information

Product	Pro/ENGINEER
PTC Support Release	Wildfire 5.0
Product Functional Area	2D Drawings
User Interface Location	Right-click a dimension and select Properties from the shortcut menu. Select the Enable intersection witness lines check box.
Processes and Initiatives	

Benefits and Description

As dimensions are created and moved in drawings for model views, the witness lines are automatically trimmed to the model edges they reference, even if they were originally overlapping. This saves time when cleaning the appearance of drawings, when created dimensions are used.

For dimensions created using intersection points, you can create intersection witness lines to convey the point of intersection. They can be resized, turned off at the same time or individually, and they assume the properties, such as the color, of the dimension.

Product What's New

Import Data Doctor

[3D Import Wizards and Data Exchange Profiles](#)

New wizards control import options for most 3D formats, providing you with immediate access to most commonly used import settings including use templates, enable ATB, and import as facet. Casual users can quickly choose a predefined profile to apply. Advanced users can tweak the details of a chosen profile or the current session settings.

[Import DataDoctor Diagnostics Wizard](#)

The Diagnostics wizard extends the Troubleshooter dialog box with a series of new error, warning, and informational geometry checks unique to the Import DataDoctor (IDD) environment.

[Import DataDoctor Match Tool](#)

The Match tool is a new Repair mode tool that replaces poor quality n-sided imported surfaces with good quality n-sided surfaces.

[Import DataDoctor Workflow and Selection Improvements](#)

Multiple enhancements to Import DataDoctor modes, selection, filters, and features help streamline, simplify, and shorten the data repair process.

Product What's New

2D Drawings

[Actual Radius Dimensions in 2D Drawings](#)

When the actual or true radial dimension cannot be shown in a given drawing view, you can choose from a list of radii for the selected feature, and show it as a note.

[Dimensional Tolerances Improvements](#)

The updated dimension properties user interface gives you more control over the tolerance.

[Draft Geometric Tolerances Created During Copy](#)

You can copy and paste geometric tolerances.

[Drawing Print Preview](#)

You can get a true preview of how your drawing will print or plot before committing to printing or exporting.

[Drawing Tree](#)

Selectable items in a drawing are enumerated on a Drawing Tree, improving usability.

[Drawing User Interface is Task-Based](#)

The Drawing user interface is task-based, improving workflows and efficiency.

[Foreshortened Radial Dimensions in 2D](#)

When a radius on a drawing is large, it is difficult to show an internal radial dimension. In 2D drawings, an internal radial dimension can appear as foreshortened.

[Hole Table Updates](#)

Hole tables in drawings are easier to use, and you can use hole tables to report the placement of additional features.

[Include Dimensions in Draft Groups](#)

You can include dimensions in 2D drawings in draft groups.

[Model Tree Improvements in Drawings](#)

The appearance of the Model Tree in drawings is updated and commands from shortcut menus are available. There is better interaction between the Model Tree, the drawing, and the Drawing Tree.

[Printing Improvements](#)

True Type fonts are correctly handled during printing. The string and font definition are passed from Pro/ENGINEER instead of rasterizing the output.

[Rounded Dimension Display](#)

Dimensions appear with a different number of significant digits from the actual value of the dimension.

[Sheet Tabs in Drawings](#)

Sheets appear in drawings as tabs.

[Single Arrow Option for Auxiliary Views](#)

When creating an auxiliary view, you can select a single arrow to appear in the parent view.

[Store and Retrieve PDF Configuration Files](#)

PDF generation from drawings is simplified because you can store and retrieve configuration files.

[Witness Line Improvements in 2D Drawings](#)

Dimension witness lines in 2D drawings are improved to provide automatic clipping for created dimensions.

Product What's New

3D Drawings

[Annotation Out-of-plane Movement Options](#)

There are new options to control the planar location of an annotation element relative to its references.

[Datum Target Annotation Features](#)

You can create Datum targets in the 3D model as annotation features.

[Dimensional Alignment Improvements](#)

In 3D you can align Dimensions. In 2D and 3D you can use dimensional alignment in more cases.

[Edit Text Style of Multiple Annotations](#)

In 3D, you can change the text style of multiple annotations at the same time.

[Expose Combined States at the Top-Level User Interface](#)

You can easily navigate between the combined states of a model without opening the View Manager. Combined or All states appear as tabs, each with a thumbnail preview, in the Graphics window.

[Geometric Tolerances Can Reference Set Datums in Inheritance Features](#)

You can use set datum planes and axes contained in an inheritance feature, for referencing within a geometric tolerance.

[Improved Movement and Appearance of Angular Dimensions](#)

There are improvements to the appearance and movement of angular dimensions in 3D and 2D.

[Layer States in View Manager](#)

You can create layer visibility states from the View Manager, and toggle the display of all layer-assigned content.

[Parameter Enhancements](#)

Parameters are more visible and it is easier to understand how to use them correctly. New system-level parameters are created for geometric tolerances.

[QuickPrint Flat-to-Screen Annotations](#)

You can output flat-to-screen content when using QuickPrint. The position of the content in the output is based on the relative x- and y-positions of the content in the Graphics window.

[Reference Model Cross-Section Edges for Annotations](#)

You can reference the temporary edges of a cross section as they appear in the model, for annotation attachment.

[Surface Finish Enhancements in 3D](#)

You can apply surface finishes annotation elements to more objects and there are new movement options.

[Target Areas for Annotations](#)

You can create target areas (supplemental geometry) in 3D for annotation referencing.

Product What's New

3D Interface

[Data Exchange of Nongeometric Content](#)

Data exchange of nongeometric content includes parameters, 3D notes, and annotations.

[Import Logs Update](#)

3D data exchange import log files are now in an XML format. They are rendered in the Pro/ENGINEER web browser and can be interactively interrogated.

[Pro/ENGINEER Interface for Inventor](#)

The Pro/ENGINEER Interface for Inventor is an import-only processor you can use to read parts and assemblies for Autodesk Inventor 2009 and earlier.

Product What's New

Surfacing - WARP

[Fixed Location for Marquee](#)

There is a control for fixing the marquee location.

[Warp Feature Enhancements](#)

There are general enhancements to the Warp feature.

Product What's New

Digital Rights Management

[Rights Management Improvements](#)

Digital Rights Management (DRM) is improved with a more flexible and efficient application of security policies and in providing a better awareness of protection for recipients.