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NO TITLE SPECIFIED 1: Solid Edge ST9 Release Notes

Proprietary and restricted rights notice

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ST9 Customer letter

Dear Solid Edge User:

Enclosed you'll find your copy of the latest Solid Edge release – Solid Edge ST9. This release continues to fulfill our mission to provide great productivity tools at very affordable prices to help you design better. Solid Edge ST9 delivers more than 1,100 new customer-driven productivity enhancements, which are based directly on feedback from working closely with customers like you.

Some highlights include:

- *Cloud-enabled design – on your terms:* ST9 can follow wherever you go. Your license can now be held on the cloud if you wish, which means a simple login to start designing, your personalized settings are remembered, and any software updates can be installed from the cloud, but of course only when you choose. And designs can be accessed anywhere with support for popular cloud services like Dropbox, One Drive or Google Drive.
- *Fast and Flexible 3D Modeling:* The user interface was enhanced to be more consistent with Microsoft Office behavior. Easy-to-access tabs can now be used to switch between documents. Improved support for high resolution monitors. Several enhancements were added to improve Synchronous modeling, Assembly in-context design, Drawing View performance and management, and 3D Printing.
- *Scalable Data Management:* We have included Data Management capability out of the box with Solid Edge ST9, and most importantly of all, with no need for additional IT resources, no SQL, no databases. Finding files is fast and easy - what better tool to integrate with than Windows Explorer, something every Microsoft user knows. And the Where Used capability another great searching tool to measure impact of any changes. The Teamcenter integration for Solid Edge has many improvements including embedded Active Workspace, a dedicated Teamcenter ribbon where all tools are easily accessed, improved performance, and many more.

Of course there are hundreds more customer driven enhancements that we expect will be put to great use by you designing great products. I want to encourage you to upgrade to ST9 and see how these new features can help you design better. See *What's New in ST9* for a complete list of all the great new features in ST9.

This past April Solid Edge celebrated the 20th anniversary of the initial release of Solid Edge. And for the past 20 years we have been privileged to have a large and growing community of incredible users that are willing to partner with us to help us improve Solid Edge. I am excited to continue this partnership and watch what we can achieve together over the next 20 years!

I also want to invite you to attend Solid Edge University 2016. Join Solid Edge experts, other users and industry partners as they come together to deliver more than 80 workshops and presentations, and be among the first to harness the power of Solid Edge ST9. The event will be held in Indianapolis, Indiana on October 25 – 27. I look forward to this event each year and getting the opportunity to meet with customers and see what great products you design using Solid Edge. Visit our [website](#) today to register and don't miss the early-bird special that ends July 31st.

Thanks and Design Better.

Jeff Walker

Director, Solid Edge Development

What's New in Solid Edge ST9

User interface changes

User interface enhancements

These enhancements were made to the user interface in Solid Edge ST9.

-  Application menu redesigned
-  Set your default Start page for Solid Edge
-  2D printing enhancements (Paper Print tab)
-  3D printing enhancements (Solid Print tab)
-  Tabbed document system
-  Data management for unmanaged documents
-  New command for sharing Solid Edge files
-  New command for finding Solid Edge files
-  Last command used in lists stays on top
-  New Style Palette for assembly models
-  Preview swatches for styles
-  Save face and view style changes
-  New Academic themes
-  Cloud-enabled Solid Edge
-  Support for high resolution monitors

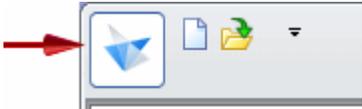
Application menu redesign

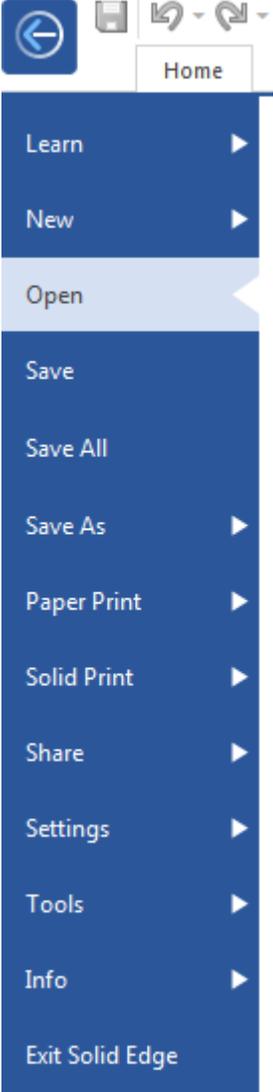
The **Application** menu was redesigned to use the touch-friendly, Microsoft backstage layout and functionality. This is the same interface that Microsoft uses in its Office products, such as PowerPoint, Excel, and Word. The backstage is where you manage your files and the data about them—creating, saving, adding custom file properties and materials, and setting options.

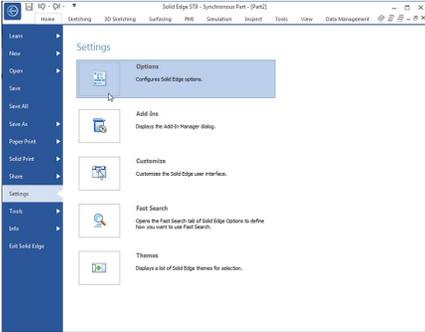
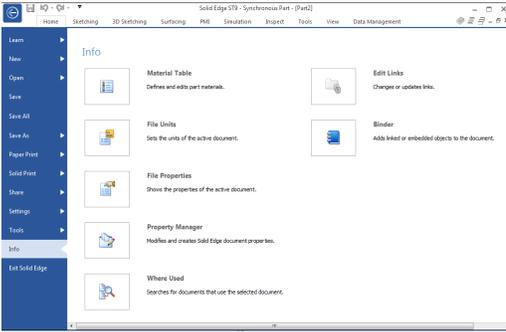
The following enhancements were made:

- In ST9, the default color scheme is **White**. Other options are **Light Gray**, **Dark Gray**, or **Black**. You can change the color scheme using the **Application color scheme** list on the **Helpers** tab in the **Solid Edge Options** dialog box.

- You now can add commands accessible from the **Application** menu to other areas of the user interface, such as the command ribbon, the radial menu, or the Quick Access toolbar. To do this, open the **Customize** dialog box, and after selecting the **Environment to customize**, set **Choose commands from:=System Menu**.
- When you select the **Application Button**, all of the functions that pertain to what you can do with the active document are displayed as tabbed pages along the left side of the application window. The commands and options that are available on the backstage pages are based on the document type and whether it is managed or unmanaged. If there are no documents open, then the options are more limited.

To	Do this
Display the Application menu, which provides access to all file-level functions.	Click the Application Button . 
Return to the active document in the tabbed document view.	Click this:  Or click a tab on the ribbon, such as the Home tab.
Display a page of related functions in the	On the Application menu, hover over or click a tab with an arrow indicator, for example, Learn , New , Open , and Save As .

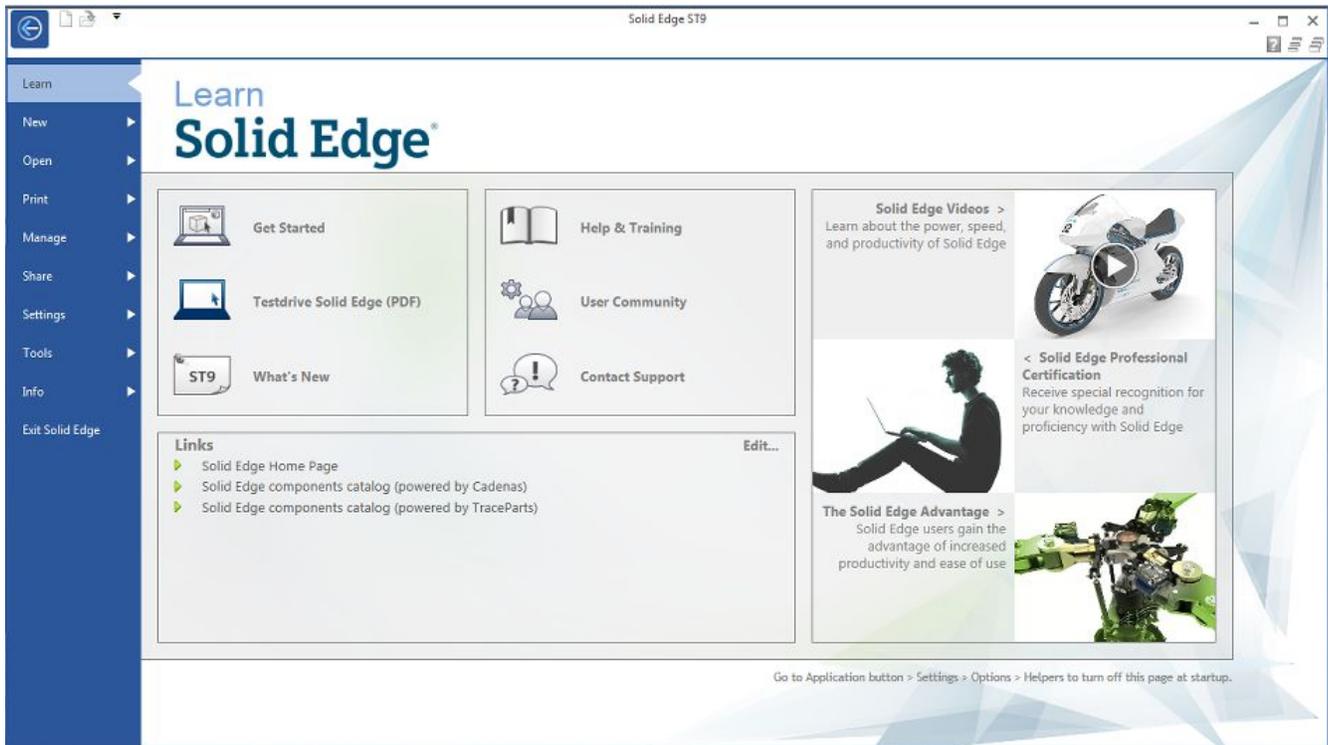
To	Do this
backstage area.	<p data-bbox="610 352 1451 516">Note Application menu items without the arrow indicator launch a command directly, for example, Save, Save All, and Exit Solid Edge.</p>  <p>The screenshot shows the application menu in Solid Edge. At the top left is a blue square button with a white left-pointing arrow. To its right are icons for a document, a refresh/cancel icon, and a redo/undo icon. Below these is a white button labeled 'Home'. The main menu is a vertical blue bar with white text and right-pointing arrowheads. The items are: Learn, New, Open (highlighted with a white arrowhead), Save, Save All, Save As, Paper Print, Solid Print, Share, Settings, Tools, Info, and Exit Solid Edge.</p>

To	Do this
<p>Open the Solid Edge Options dialog box to specify the type of tooltip you want to see.</p> <p>Open the Customize dialog box to customize the default arrangement of tools, commands, and keyboard functions.</p> <p>Change the Solid Edge Theme.</p>	<p>Click the Settings tab.</p> 
<p>Open these frequently used dialog boxes:</p> <ul style="list-style-type: none"> • Material Table • File Properties. • Property Manager. 	<p>Click the Info tab.</p> 

Changes to the Solid Edge startup screen

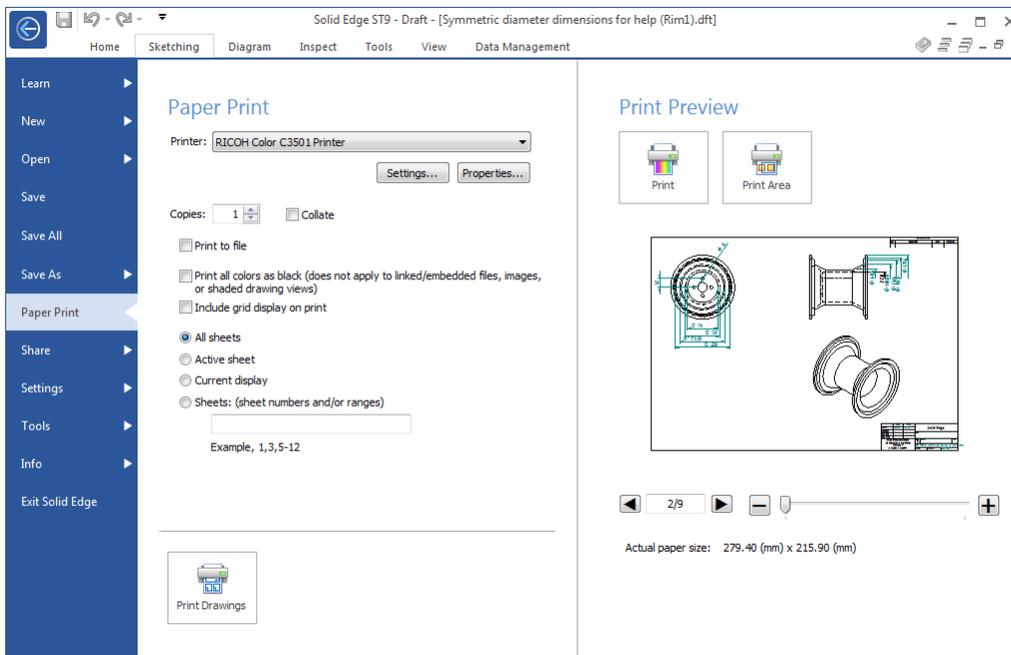
In conjunction with moving the file-related functions of the **Application** menu onto the new Microsoft backstage area, the Solid Edge startup screen was redesigned as a start page that you can change to meet your current design needs. Now you can choose one of several page or document options to display when Solid Edge opens, so that you can begin creating or editing faster. The **Learn** page (shown below) hosts links to Solid Edge Help and Training. This is the default start page when you first install Solid Edge.

To change your start page option, go to **Application Button**→**Settings** tab→**Options** command→**Solid Edge Options** dialog box→**Helpers** tab.



2D printing enhancements (Paper Print)

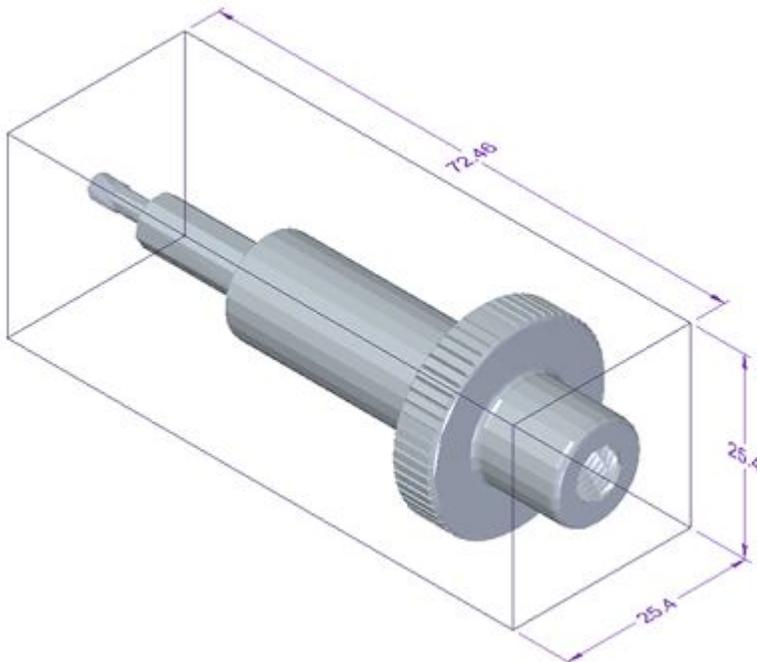
The new layout of the **Application** menu has the advantage of grouping similar functions and making them visible and available without having to navigate additional menus. The new **Paper Print** page on the **Application** menu contains all of the 2D print commands and related functions for printing any Solid Edge document to paper or to a file.



3D printing enhancements (Solid Print)

A new **Solid Print** page on the **Application** menu groups the following 3D print-related commands and other functions that you need to generate a 3D prototype of your part, sheet metal, or assembly model:

- **Conversion Tolerance** option, to control the tessellations on the model.
- **Preview** command, to show you the model appearance and extents before it is sent to print production.

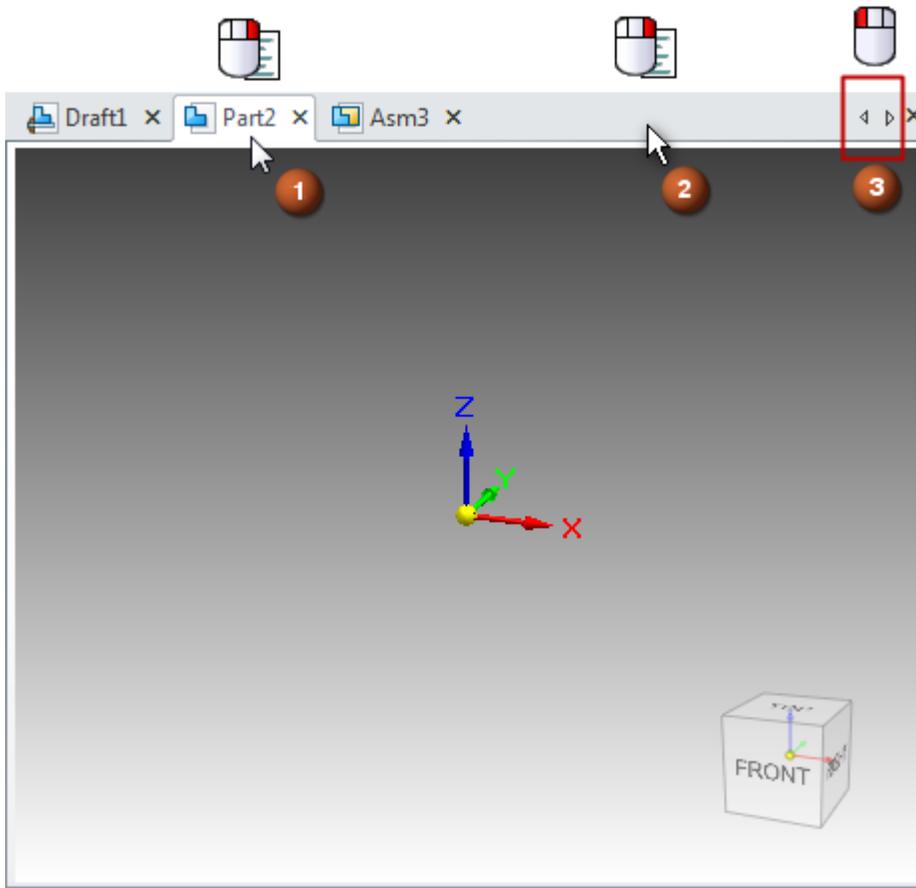


- **Print to File** command, for saving the file to **STL** format.
- **3D Builder** command, to send the .stl file to a 3D printer.

Tabbed document system

A new tabbed document interface in all environments lets you view and switch easily between all of your open document windows simply by clicking a tab. Commands that you previously had to navigate to on the **View** tab→**Window** group are now available directly in the new tabbed interface.

- Two new context menus are available when you right-click a *document tab* (1) or the *tab list bar* (2). These provide fast access to commands for manipulating document windows, including some new commands, such as **Open Document Folder** and **Copy Folder Path**.
- When you have many documents open, or when you have the same document open in multiple windows, you can scroll through the document windows using the right and left arrow controls (3). You also can use the mouse scroll wheel.
- You can drag the tabs to change the display order in the tab list bar.



Data management for unmanaged documents

A new **Data Management** tab on the ribbon provides built-in data management tools within Solid Edge. The commands on the **Data Management** tab are for those working in a 3D design environment who do not need the automated processes provided by document management in SharePoint or Teamcenter.

These basic tools:

- Provide up-to-date document information.
- Prevent file and part number duplication.
- Quickly search for related documents.
- Find everywhere a file is used.
- Share, mark up, and revise documents through View and Markup and Design Manager cloud file services.

New command for sharing Solid Edge files

A new **Pack and Go** command  copies and packages a complete set of project documents, so that you can email them or share them with a customer or vendor. In the **Pack and Go** dialog box, you can set options to:

- Maintain the existing folder structure.
- Flatten the folder structure.
- Include additional files, such as simulation results and draft files.
- Create or overwrite a .zip file.

Example

The command is available on the **Application** menu→**Share** tab, as well as in Design Manager.

New command for indexing files for search

To improve the performance of search based on Solid Edge document properties, a new **Fast Search**



command is available on the **Application** menu→**Settings** tab.

The **Fast Search** command provides Microsoft Windows index-based search within Solid Edge. Clicking the command displays the new **Fast Property Search page (Solid Edge Options dialog box)**, which you use to specify the locations of Solid Edge folders and properties that you want to index for searching. You can index and search your local computer or a network file server.

Fast Search identifies the following types of document properties:

- Standard Solid Edge properties, for example, document number, revision number, project name, and document status information.
- Custom properties defined in your templates and in the **propseed.txt** file.

Note

If you are using a file server, first start the installation wizard by selecting the **Fast Search** link on the Solid Edge installation screen. This adds two new commands on the **Start** menu:

Use this Start menu option	To do this
 Fast Search Configuration	Launch the Fast Search Configuration utility (SearchConfig.exe), which sets up the fast search locations and property locations on a network file server.
 Duplicate File Report	Runs a Solid Edge command line utility (DupFileReport.bat) that generates an enhanced duplicate file report with a weighted list of duplicate files by taking into account more Solid Edge file characteristics than the standard properties. Additional files related to this utility are delivered in the Solid Edge Preferences folder.

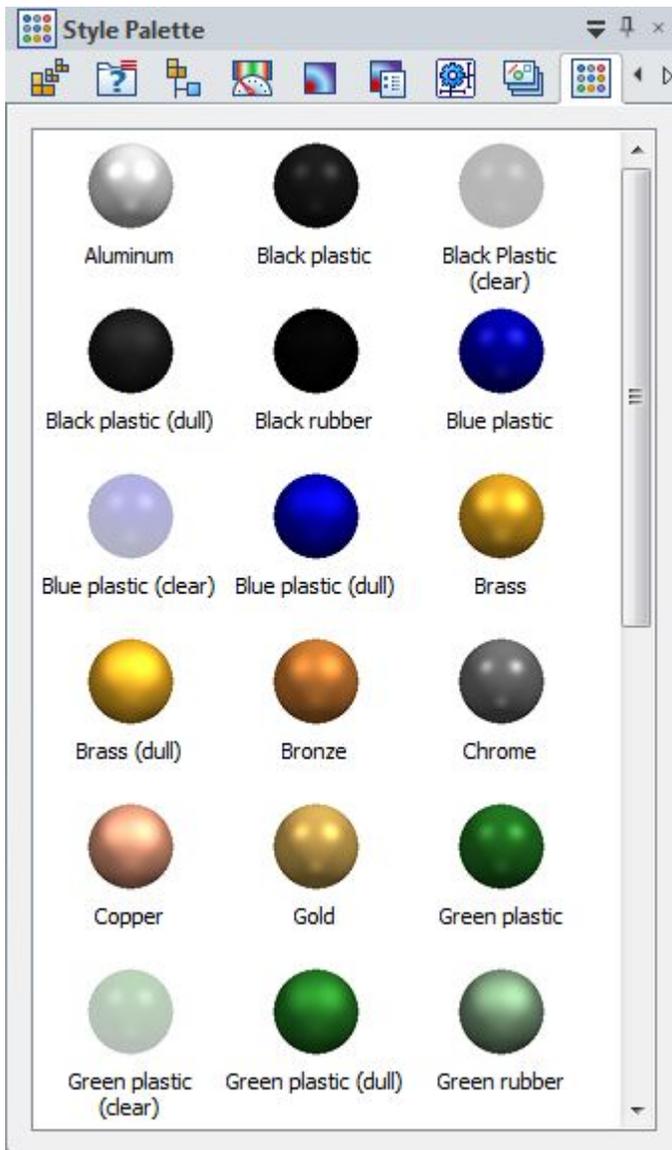
New way to manage settings and preferences

Using the Settings and Preferences Wizard or the **Solid Edge Options** dialog box→**User Profile** page, you can now easily capture information associated with settings and preferences from one machine and deploy that information to other machines.

You can capture registry settings, templates stored on a server, themes in the **AppData** folder, and customized files located in the **Preferences** folder.

New Style Palette for assembly models

A new **Style Palette** is available in the **Assembly PathFinder** tab set for applying colors to the model. You can drag materials from the style palette onto parts and features in the assembly model. The body highlights to indicate that the material color is applied.

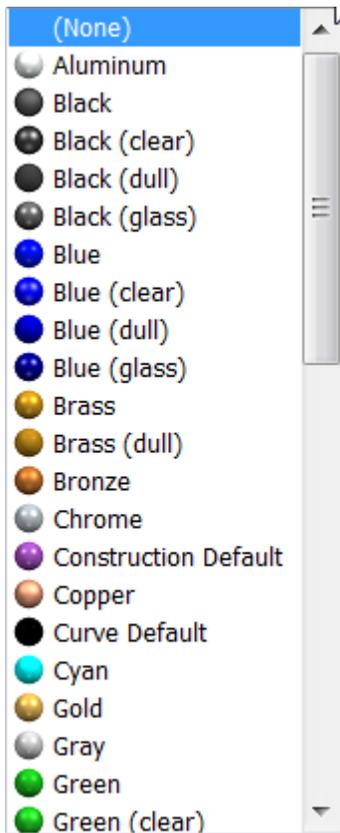


Preview swatches for styles

Preview swatches are now displayed in dialog boxes and on command bars where you select face styles, fill patterns, hatches, and materials. Previously, color options were listed by name only, without a graphical preview.

Here are some of the locations where preview swatches are available:

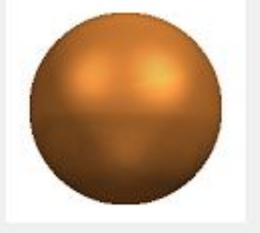
- For the **Part Painter** command, from the **Style** list on the command bar:

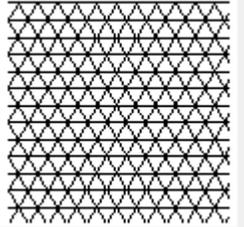
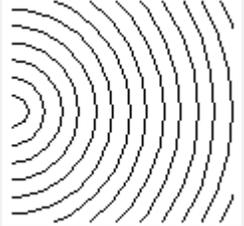


- In the **Material Table**, on the **Material Properties** tab, from the **Face Style** list.
- In the Assembly environment, alongside the **Face Overrides** command, from the **Face Style** list on the ribbon:



- In the **Color Manager** dialog box, from the **Base Styles** lists.
- For the **Styles** command, in the **Style** dialog box, for the following **Style types**:

Style type	Example
Faces Styles	

Style type	Example
Fill	
Hatch	

Save face and view style changes

Now you can save changes that you make to the appearance of model faces (such as color, shine, reflection, texture) and to the 3D view of the model (such as lighting, background, drop shadow).

- You can save the overrides to the current 3D view style or face style using the **Save As** button in the **View Overrides** dialog box and the **Face Overrides** dialog box.
- You can modify the newly created style using the **View** tab→**Style** group→**Styles** command, and then selecting the appropriate style type.

Part Painter resets Parasolid body color

A new option on the **Part Painter** command bar enables you to remove all previously applied colors from model features, faces, and bodies. The **Replace styles** option applies the default model color to the entire model, removing all other colors and color overrides.

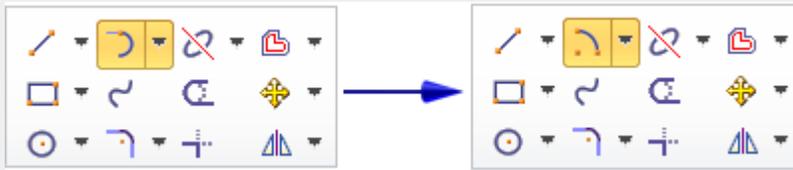
This option solves the problem of Parasolid bodies in imported or converted models not responding to color changes applied with **Part Painter**.

Last command used in lists stays on top

Now you can specify that the last command that you selected from a list on the command ribbon stays on top as the default command.

Example

When this option is selected, the default command, **Tangent Arc**, is replaced by the selected command, **Arc By Center Point**, on the top of the stack.



The new option, **Last command used (in droplists) stays on top**, is located on the **Helpers** tab in the **Solid Edge Options** dialog box. This option affects all environments.

New Academic Themes

This release has new Academic Themes that can be selected to help simplify the process of learning Solid Edge for our youngest learners. The themes provide a single-tab interface that contains a reduced number of tools and commands for each environment that should be sufficient to initially learn Solid Edge using a simplified interface.

The full-compliment of Solid Edge commands remain available by simply choosing a professional theme.

These Academic Themes can only be selected by educators and students that are operating on a Solid Edge Academic License. At this time two new sets of themes have been introduced:

First Steps

This theme is designed to provide a consolidated set of tools and commands onto a single tab to be used by our youngest modelers as they initially learn Solid Edge. It provides a simplified interface for learning part, assembly, and draft skills with an emphasis on synchronous modeling techniques.

Next Steps

This theme provides more tools, and commands for young modelers as they tackle more ambitious synchronous modeling skills.

For more information about themes in general and how you can select and apply themes, see Solid Edge themes.

Cloud-enabled Solid Edge

A new application, Solid Edge Cloud Gateway, makes it possible to use Solid Edge from multiple machines. Data can be local or synchronized between desktop and Internet users via Dropbox.

Solid Edge Gateway provides a dashboard, which you use to:

- Log in using your Webkey account.
- Get your license to run Solid Edge.
- Manage individual and group preferences across machines through the cloud.

- Download and install Solid Edge product update maintenance packs.

For more information, see the Solid Edge Installation and Licensing Guide.

Support for high-resolution monitors

Solid Edge ST9 has improved support for bitmaps, controls, and other graphic elements displayed on high-resolution monitors (3K, 4K, and higher). Although high-resolution monitor support is available for both Microsoft Windows 7 and Windows 10, Windows 10 offers superior dots-per-inch (DPI) scaling capabilities.

When you use a monitor with DPI greater than 170 pixels, we recommend you adjust settings in two locations:

- Set the primary DPI scaling resolution for the entire desktop using the Microsoft Windows **Control Panel**.
- Adjust the relative sizing of Solid Edge design tools on the **Helpers** tab in the **Solid Edge Options** dialog box.

Set the Windows screen resolution

From the Microsoft Windows 7 **Start** menu, go to:

1. **Control Panel**→**Appearance and Personalization**→**Display**.
2. In the left link pane, click **Set custom text size (DPI)**.
3. In the **Custom DPI Setting** dialog box, choose this:
Scale to this percentage of normal size=200%

Note

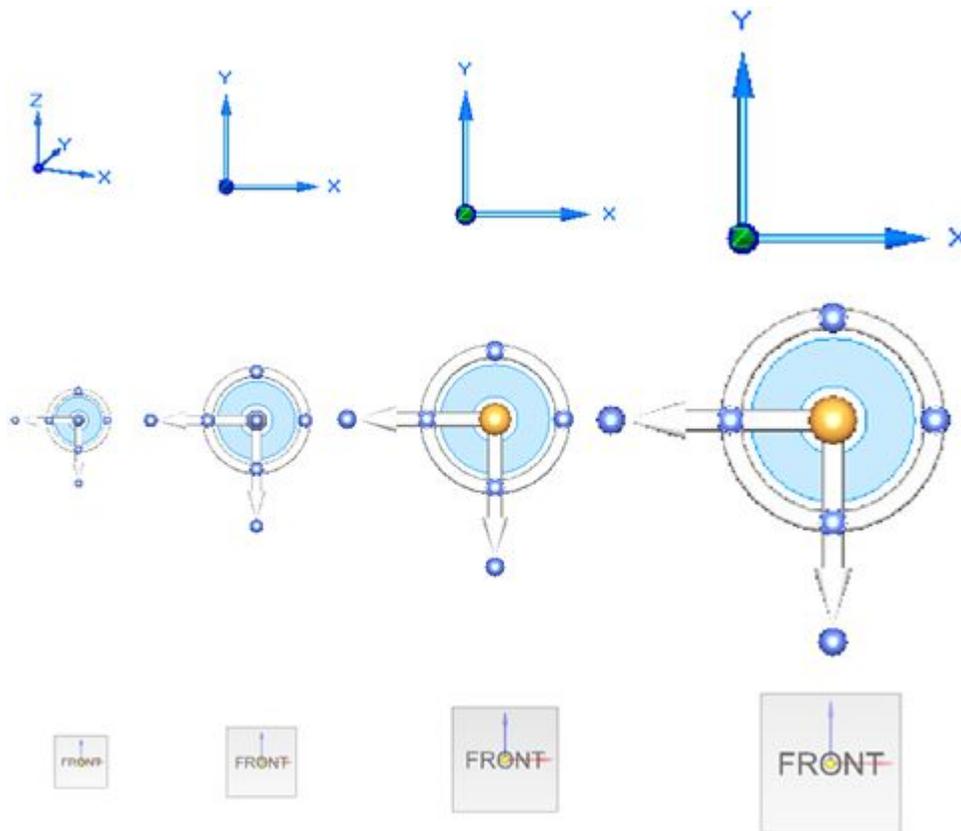
The Control Panel location for setting DPI scaling is the same in Windows 10, but additional options are available.

Adjust the relative size of Solid Edge design tools

If you work with multiple screen resolutions (including 4K), you can choose a larger display size for the steering wheel, OrientXpres tool, and Quick View Cube control, so that they are easier to work with on screens of different sizes.

1. In Solid Edge, go to the **Application** menu→**Settings** tab→**Options** command→**Helpers** tab (**Solid Edge Options** dialog box).
2. In the **General** section, from the **Steering Wheel, OrientXpres, and View Cube size** list, choose a size.

From left to right, the size options are **Small** (0.75 inches), **Medium** (1.00 inches), **Large** (1.25 inches), and **Extra Large** (1.50 inches).



Part enhancements

Part enhancements

These part modeling and surfacing enhancements were made in Solid Edge ST9.

-  Material Table enhancements
-  Multi-body cut enhancements
-  Multi-body rounding and chamfering
-  Blocks functionality now available in Ordered Part, Ordered Sheet Metal, and Assembly environments
-  Enclosure command now available in Part and Sheet Metal environments
-  Chamfers maintained during synchronous move, rotate, and dimension edit operations
-  Ordered body display while working in synchronous mode
-  New Design Intent rule (Maintain Offset Faces)
-  Duplicates command (patterning)

-  Associative Mirror command in Part
-  Dimension a hole to a virtual vertex
-  Pattern by Table command enhancements
-  Solid swept features
-  Spiral curves

Material Table enhancements

More usability enhancements were added to the **Material Table**.

In the **Material Table** dialog box, these changes include:

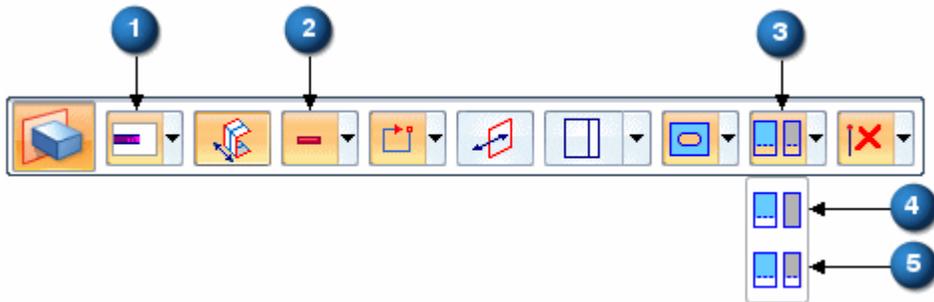
- Materials with out-of-date properties are highlighted. You can use the new command, **Update Material Properties** , to update the material definition in the current part or sheet metal document.
- Resizing handles for the dialog box.
- Sorting the material tree by ascending or descending material names.
- Listing the items in the material tree by material group, material library, or by material name.
- Automatic physical property update when you change material properties or gage properties.
- Image previews of face and fill styles. Direct access to the Styles dialog box for creating new face styles and fill styles.

Outside of the **Material Table** dialog box, these enhancements include:

- A new option on the **General** tab (**Solid Edge Options** dialog box), specifies whether to check for out-of-date materials and gages when a part document is opened.
- A new command, **Check Out of Date Material** , is available on the **Material Table** shortcut menu in **PathFinder**. It is also available as a button in the **Physical Properties Manager** dialog box in an assembly.
- In **PathFinder**, an out-of-date status indicator now precedes the material name.
- Materials that you apply to a model are added automatically to the Variable Table. This enables you to reference them as custom properties in the **File Properties** dialog box, and as exposed variable properties in the Draft environment.
- When copying and pasting materials, duplicate names are identified. A dialog box offers options to replace and copy, do not copy, or rename and copy the material.

Multi-body cut enhancements

In multi-body modeling, you can control how a cutout feature interacts with the design bodies it intersects.



(1, 2)	The Apply Cut Features option is only available when the Extrude options are set to: Extent=Finite (1) and Add/Cut=Cut (2).
(3)	Use the Apply Cut Features option on the Extrude command bar.
(4)	The Cut Active Body option only cuts the active design body.
(5)	The Cut Selected Bodies option cuts all bodies that are intersected by the cut. All intersected bodies highlight in green. You can deselect an intersected body by pressing the Ctrl key while selecting the design body.

When the model contains multi-bodies, the **Apply Cut Features** option is available for the following commands:

Ordered

- Cut
- Revolved Cut
- Hole

Synchronous

- Extrude
- Revolve
- Hole

Note

For linear cuts, all Extent types are valid except **Through Next**.

When patterning or mirroring a cut feature that passes through multi-bodies, the pattern maintains the multi-body selections.

Multi-body rounding and chamfering

You can now create a single round or chamfer feature by selecting edges from different multi-bodies.

Display of ordered bodies while in synchronous mode

There is an **Ordered Body** display setting so that a transparent representation of the finished model is shown while in synchronous mode.

Blocks

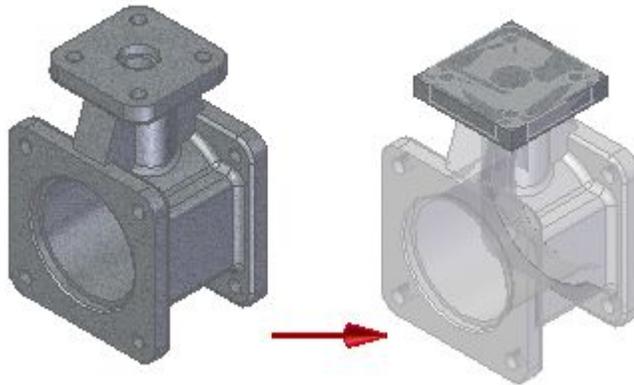
- Blocks are now available in the following environments:
 - o Ordered Part
 - o Ordered Sheet Metal
 - o Assembly layout

Common feature profiles can be added as a block instead of redrawing. Each environment has access to block libraries.

- The **Open** and **Edit Blocks** commands are not available in these environments. The workaround is to use the **Unblock** command, make edits, and then recreate the block.

Enclosure command now available in Part and Sheet Metal

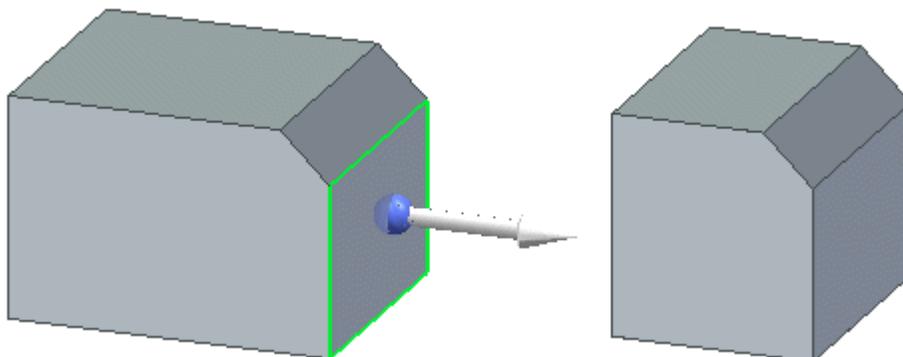
The **Enclosure** command is now available in the synchronous and ordered Part and Sheet Metal environments, so you can create a box or cylinder around the selected part geometry.



Each new enclosure adds a new body to the model.

Chamfers maintained during synchronous move, rotate, and dimension edit operations

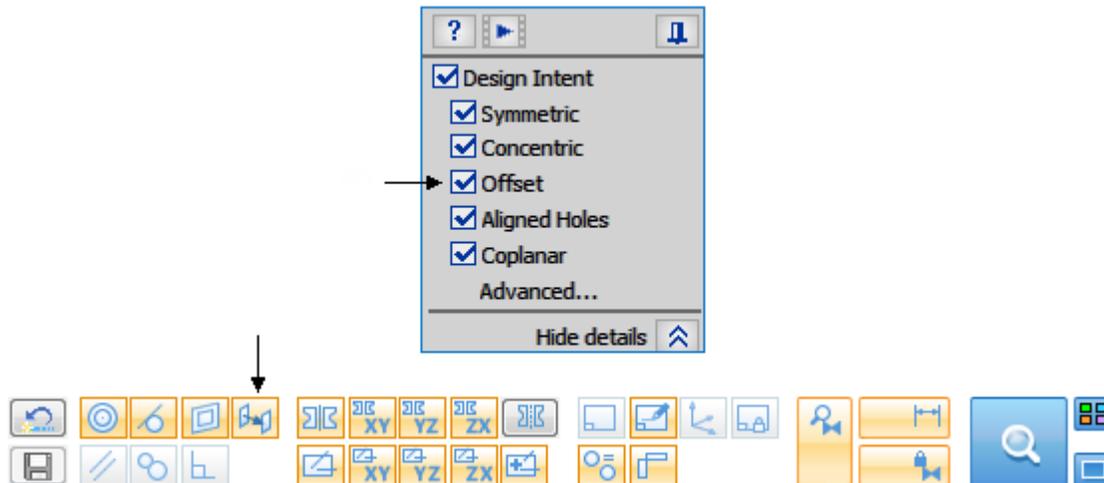
Chamfers are now recognized and maintained during synchronous move, rotate, and edit operations.



New Design Intent relationship (Maintain Offset Faces)

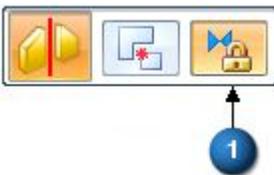
A new Design Intent relationship is available, called **Maintain Offset Faces (I)**.

Solid Edge can now detect thin features like ribs, webs, thinwalls etc. in translated models during sync edits. This new option is used to help you maintain the design intent you build into your parts.

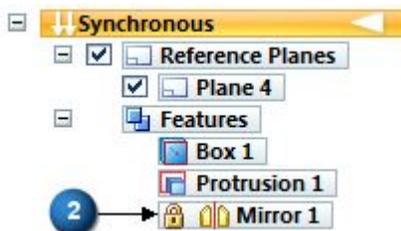


Associative Mirror in synchronous part

The synchronous **Mirror** command in part now behaves like the synchronous **Mirror** command in sheet metal. The part **Mirror** command now has a **Persist** option (1) to make the mirror associative to the parent features.



In **PathFinder**, the associative mirror feature has a lock icon (2) in front of the feature name. A non-associative mirror has no lock icon.



Duplicates command

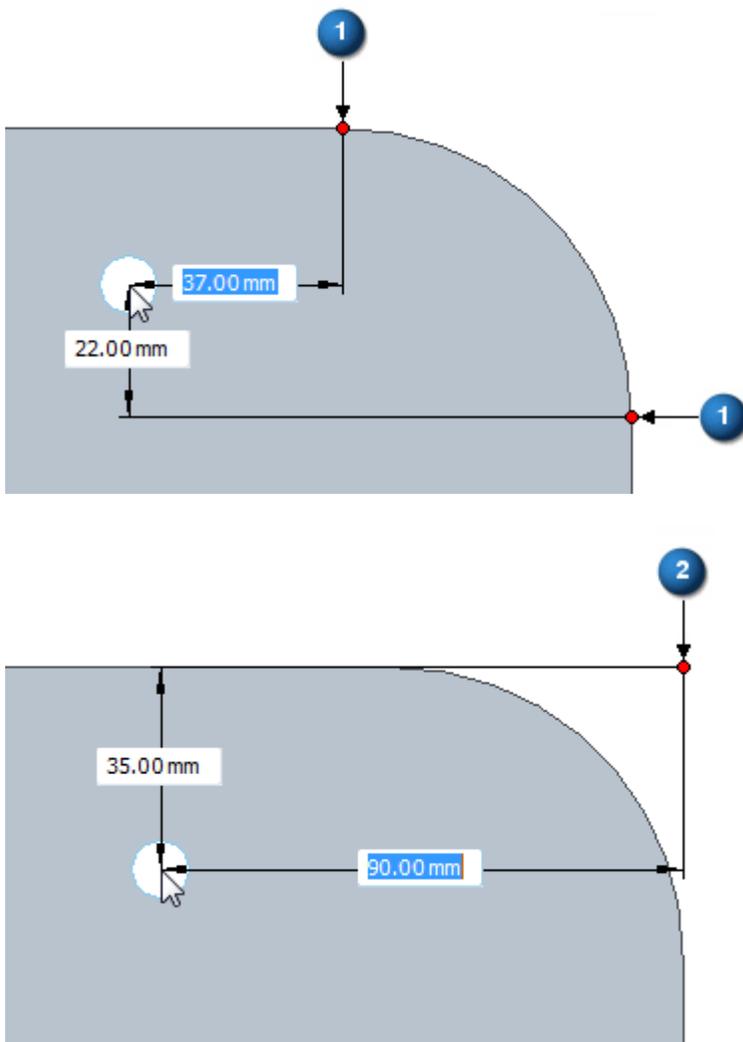
Use the new **Duplicates** command in the Part and Sheet Metal environments to create duplicate geometry using blocks and coordinate systems to define the new location. This command can be used to create a 3D pattern.



The **Duplicates** command is located on the **Home** tab→**Pattern** group→**Pattern list**. The operation creates a pattern feature which can be edited.

Dimensioning a hole to a virtual vertex

When placing a hole, you can now dimension to a virtual vertex. While in the endpoint mode **E** (1), press **J** to switch to intersection mode (2).



Pattern by Table command enhancements

- The **Pattern by Table** command now supports the use of an embedded Excel spreadsheet. A **Link** option in the **Instance Table** dialog box controls whether a *linked* or *embedded* Excel spreadsheet is used to drive the pattern.

After browsing for the Excel spreadsheet, uncheck the **Link** box to make the selected spreadsheet embedded in the file.

- You can define a **'From' point (Pattern Origin Point)** in a separate optional step, so that the pattern origin can be defined from a keypoint other than the origin of the pattern coordinate system.
- You can define a precise point using model keypoints with the **'To' point (Parent Reference Point)** in the **Instance Table** dialog box.
- The parent instance handle is marked similar to that in synchronous **Pattern** commands.

Solid swept features

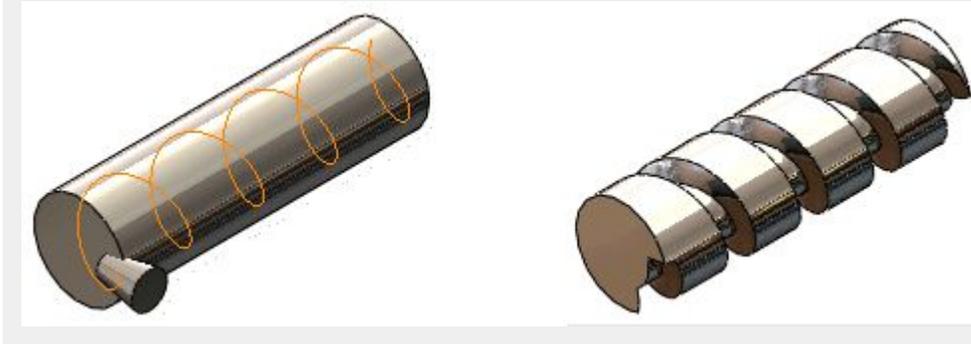
For machine tool operations such as the creation of tines on a drill bit or the well in a barrel cam, typical cutout features do not suffice to accurately represent the realistic faces of a tooled part. Now you can model these solid swept extruded features and solid swept cutout features using the following new commands:

- **Home tab**→**Solids group**→**Add list**→**Solid Sweep Protrusion**  .



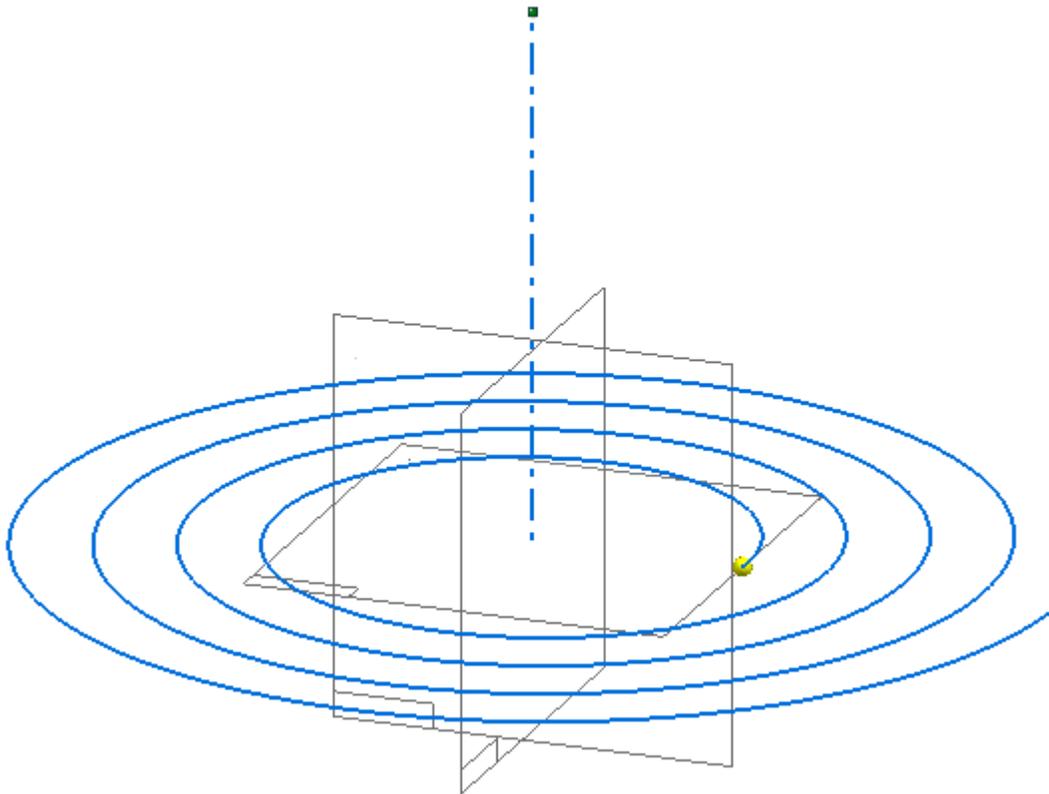
- **Home tab**→**Solids group**→ **Cut list**→**Solid Sweep Cutout**  .

Example



Spiral curves

Now you can create a 2D spiral curve using the **Helical Curve** command and the new **Spiral** option in the **Helical Curve Parameters** dialog box.



Sheet metal enhancements

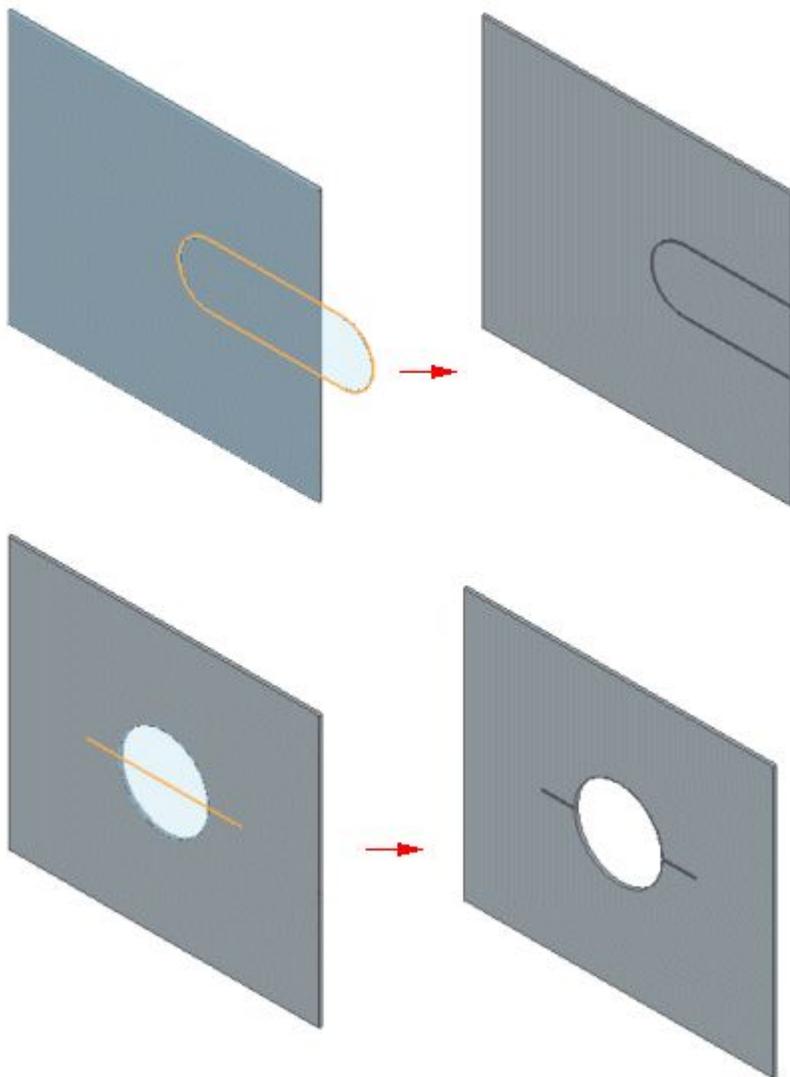
Sheet metal enhancements

These sheet metal enhancements were made in Solid Edge ST9.

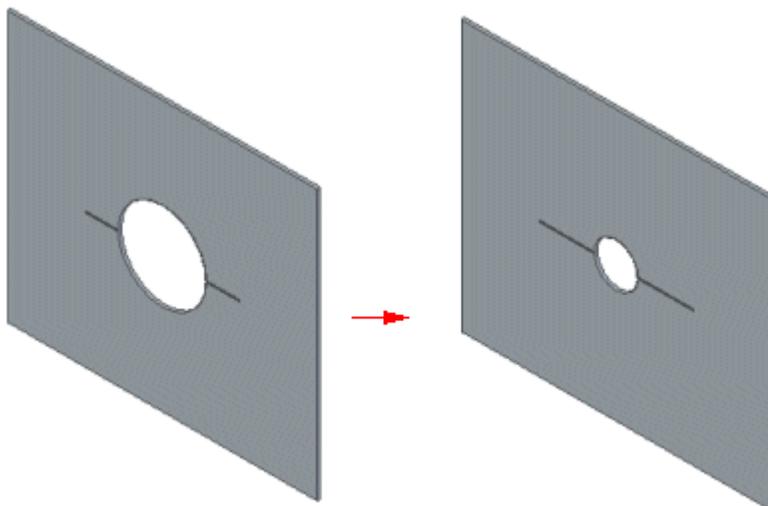
-  Etch command enhanced
-  Material Table enhancements
-  Move and Rotate commands now available in Flat Pattern
-  PathFinder now indicates clashes in flat patterns
-  Blocks functionality now available in Ordered Part, Ordered Sheet Metal, and Assembly environments
-  Enclosure command now available in Part and Sheet Metal environments
-  New Design Intent rule (Maintain Offset Faces)
-  Duplicates command (patterning)
-  Spiral curves

Etch command enhancements

- The **Etch** command now supports the selection of objects across the boundary of a tab. If you select an object that intersects the tab boundary, the etch extends only to the boundary of the tab.



If you edit the boundary, the etch automatically adjusts to the change.



- The following new options are available in the **Save as Flat DXF Options** dialog box when you export the sheet metal file to dxf format:

- o Etch Feature Line Type
- o Etch Color

Two parameters were added to the sesmf.ini file to define the line style and color mapping when exporting an etch feature as sketch geometry.

- o 100=UseEtchFeatureLineType defines the line type.
- o 8=Use Etch Feature Color defines the line color.

Note

To include this new functionality in legacy Solid Edge files, you need to add these values to the sesmf.ini file.

- Stick fonts are now delivered with the Solid Edge fonts.

Material Table enhancements

More usability enhancements were added to the **Material Table**.

In the **Material Table** dialog box, these changes include:

- Materials with out-of-date properties are highlighted. You can use the new command, **Update Material Properties** , to update the material definition in the current part or sheet metal document.
- Resizing handles for the dialog box.
- Sorting the material tree by ascending or descending material names.
- Listing the items in the material tree by material group, material library, or by material name.
- Automatic physical property update when you change material properties or gage properties.
- Image previews of face and fill styles. Direct access to the Styles dialog box for creating new face styles and fill styles.

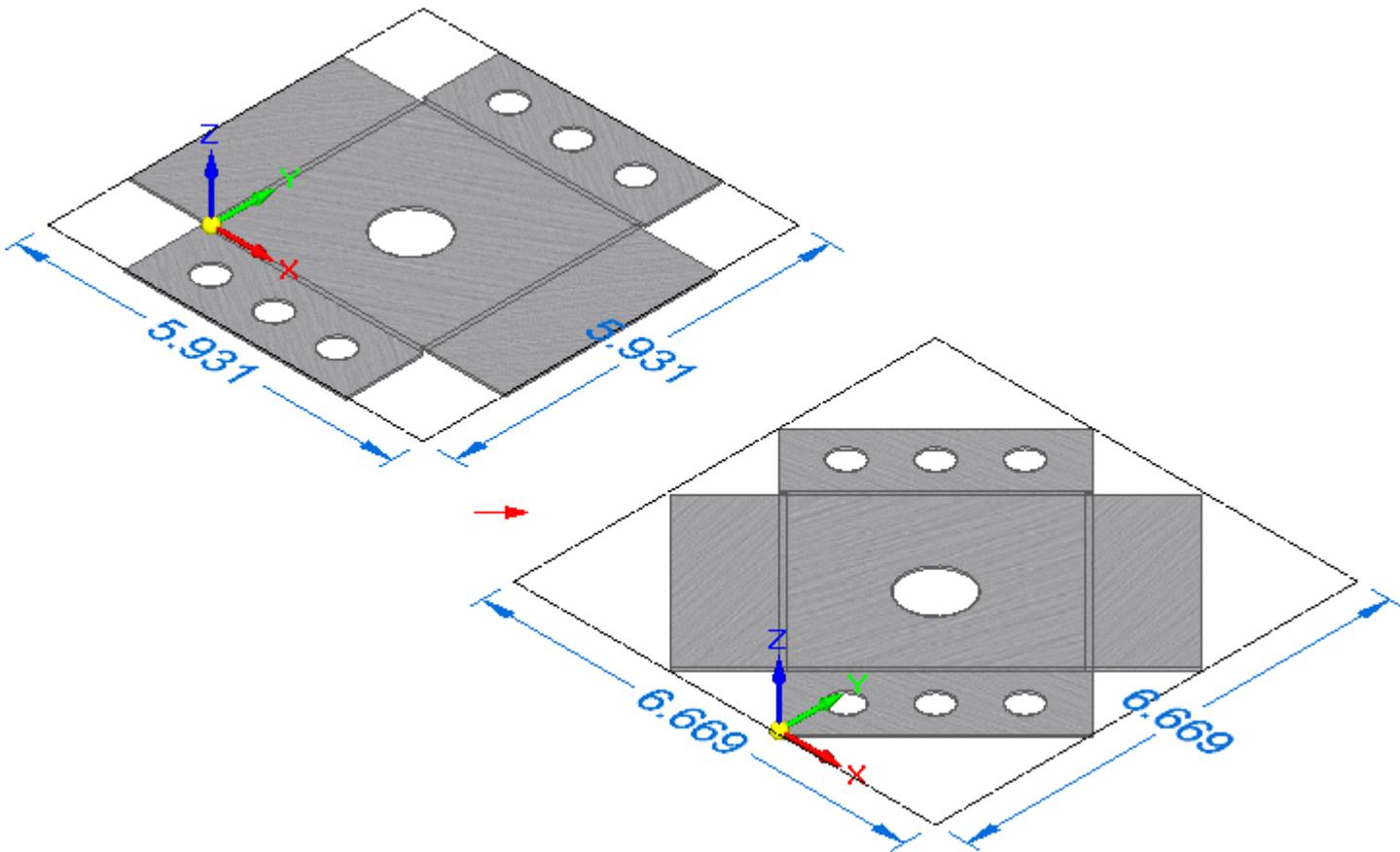
Outside of the **Material Table** dialog box, these enhancements include:

- A new option on the **General** tab (**Solid Edge Options** dialog box), specifies whether to check for out-of-date materials and gages when a part document is opened.
- A new command, **Check Out of Date Material** , is available on the **Material Table** shortcut menu in **PathFinder**. It is also available as a button in the **Physical Properties Manager** dialog box in an assembly.
- In **PathFinder**, an out-of-date status indicator now precedes the material name.

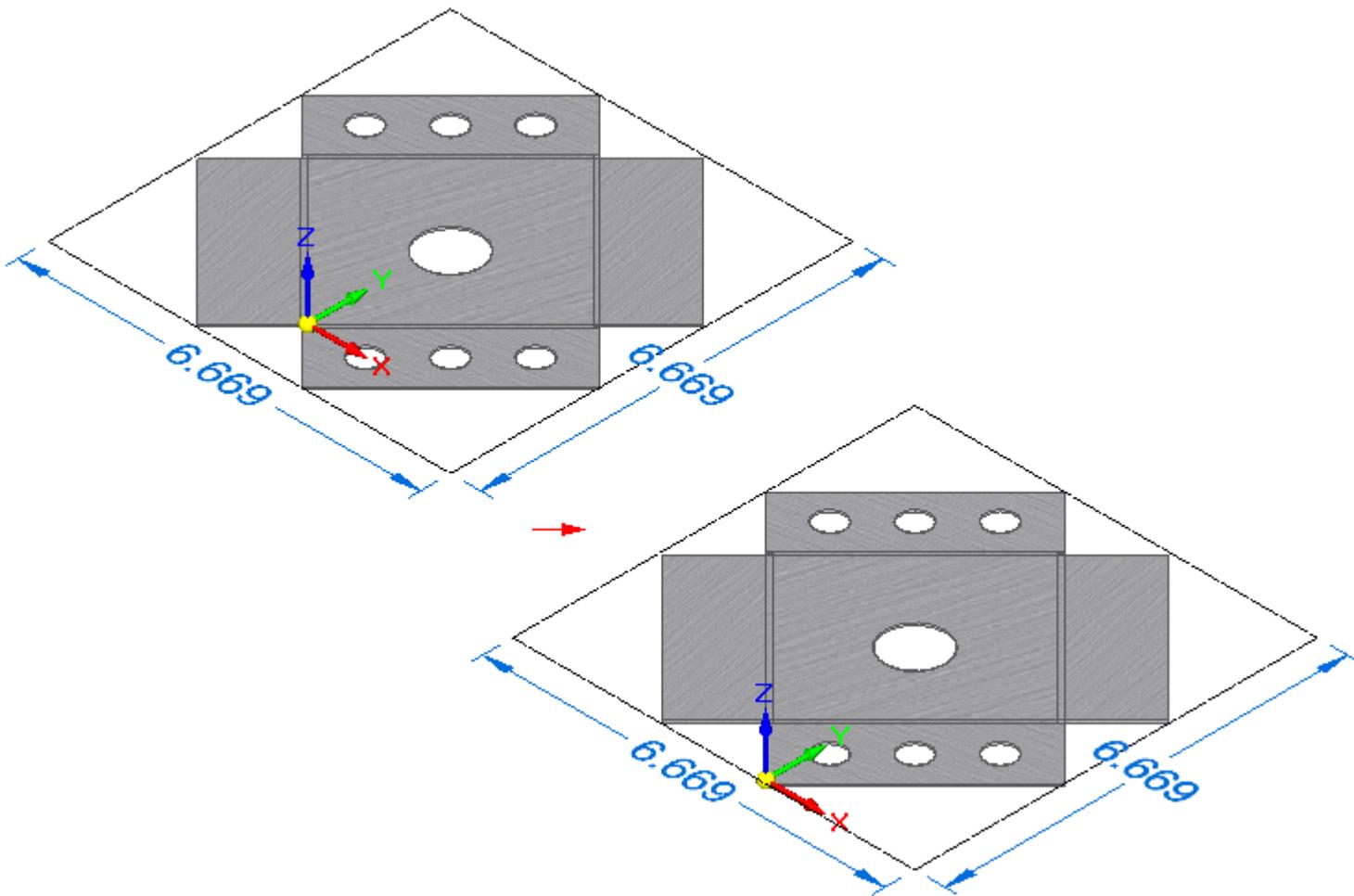
- Materials that you apply to a model are added automatically to the Variable Table. This enables you to reference them as custom properties in the **File Properties** dialog box, and as exposed variable properties in the Draft environment.
- When copying and pasting materials, duplicate names are identified. A dialog box offers options to replace and copy, do not copy, or rename and copy the material.

Move and Rotate commands now available in Flat Pattern

The **Rotate Face** and **Move Face** commands are now enabled in the **Flatten** environment. Use these commands to change the orientation



or position of a flat pattern.



Note

The commands only support the **Body** selection type.

PathFinder now indicates clashes in flat patterns

PathFinder now displays an arrow next to any flattened feature in a flat pattern that contains a clash between sheet metal tabs.



In addition to the indicator in **PathFinder**, a warning message indicates the flattened feature contains overlapping tabs.

Note

The warning is displayed for legacy files when the flatten feature is recomputed in ST9.

Blocks

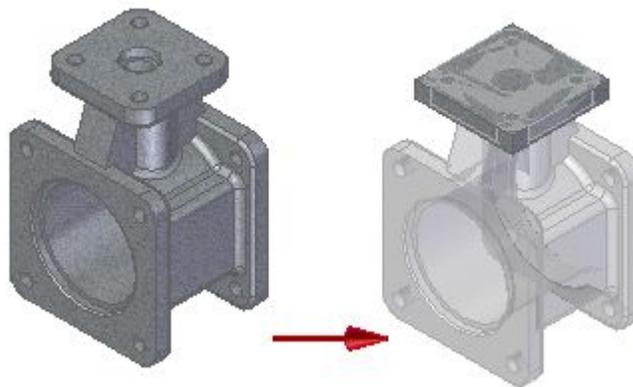
- Blocks are now available in the following environments:
 - o Ordered Part
 - o Ordered Sheet Metal
 - o Assembly layout

Common feature profiles can be added as a block instead of redrawing. Each environment has access to block libraries.

- The **Open** and **Edit Blocks** commands are not available in these environments. The workaround is to use the **Unblock** command, make edits, and then recreate the block.

Enclosure command now available in Part and Sheet Metal

The **Enclosure** command is now available in the synchronous and ordered Part and Sheet Metal environments, so you can create a box or cylinder around the selected part geometry.

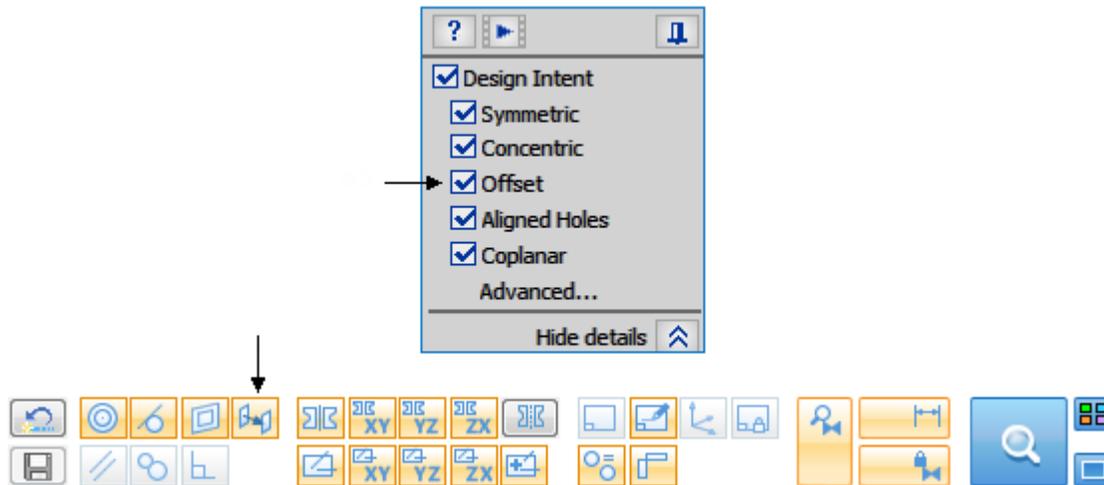


Each new enclosure adds a new body to the model.

New Design Intent relationship (Maintain Offset Faces)

A new Design Intent relationship is available, called **Maintain Offset Faces (I)**.

Solid Edge can now detect thin features like ribs, webs, thinwalls etc. in translated models during sync edits. This new option is used to help you maintain the design intent you build into your parts.



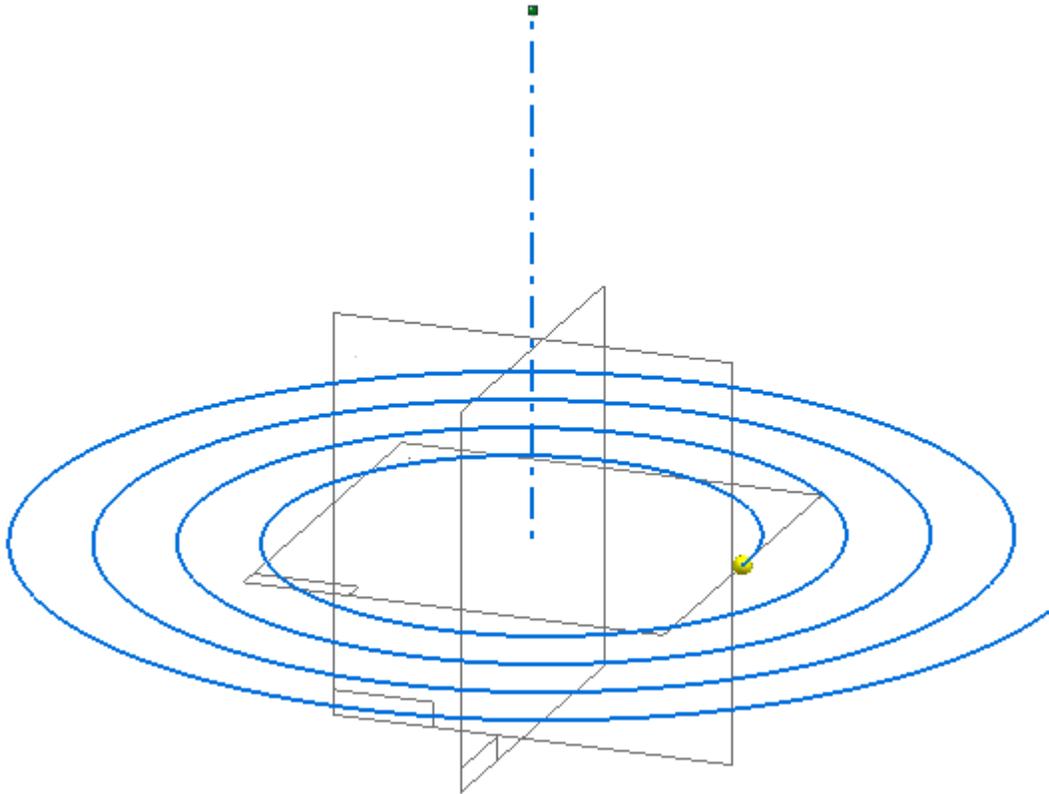
Duplicates command

Use the new **Duplicates** command in the Part and Sheet Metal environments to create duplicate geometry using blocks and coordinate systems to define the new location. This command can be used to create a 3D pattern.

The **Duplicates** command  is located on the **Home** tab→**Pattern** group→**Pattern list**. The operation creates a pattern feature which can be edited.

Spiral curves

Now you can create a 2D spiral curve using the **Helical Curve** command and the new **Spiral** option in the **Helical Curve Parameters** dialog box.



Assembly enhancements

Assembly enhancements

These enhancements were made in the Assembly environment in Solid Edge ST9.

-  Blocks
-  Assembly Relationship Manager
-  Contour Flange command available in assemblies
-  Custom occurrence properties in assembly
-  Duplicate Component command enhancements
-  Ghost Mode command
-  Isolate command
-  Movement of electrical connectors enhanced
-  Occurrence property enhancements

-  Piping enhanced
-  Replace Face command in assembly
-  Standard parts enhanced
-  Subtract command enhancement
-  Toggle Display command
-  Thicken command enhancement
-  Variable Table enhancements

Blocks

- Blocks are now available in the following environments:
 - o Ordered Part
 - o Ordered Sheet Metal
 - o Assembly layout

Common feature profiles can be added as a block instead of redrawing. Each environment has access to block libraries.

- The **Open** and **Edit Blocks** commands are not available in these environments. The workaround is to use the **Unblock** command, make edits, and then recreate the block.

Assembly Relationship Manager

The **Assembly Relationship Manager** shortcut command displays a table of assembly relationships and provides a way to modify relationships. Starting the command with occurrences selected displays the relationships associated with those occurrences. If no occurrences are selected, the command shows all of the relationships for the assembly. The relationships are analyzed and displayed. You can find and repair sick and failed relationships by either suppressing, deleting, or editing the relationships.

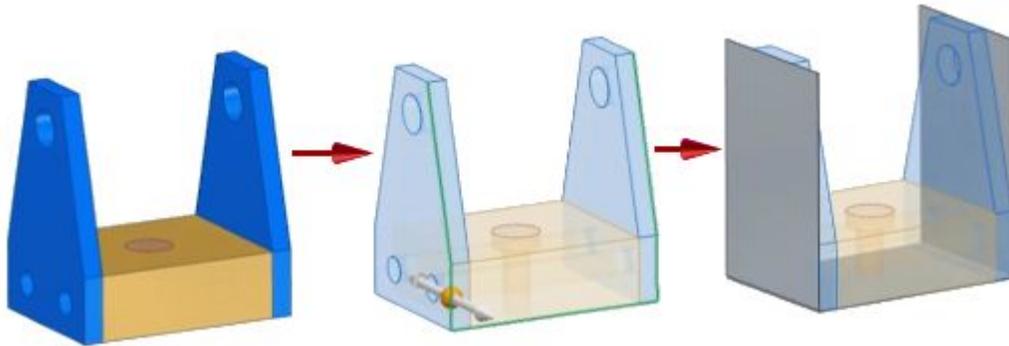
Contour Flange command available in assemblies

Now you can use assembly geometry to create a synchronous sheet metal model using the **Contour**

Flange command . This enables you to match faces at different angles without having to draw sketches. You can use this command only if it is the first feature (the base feature) in a sheet metal body.

To create a sheet metal body in the context of an assembly, first select the **Create Part In-Place**

command  to create an empty synchronous sheet metal file. Then you can select the **Contour Flange** command to locate and select a chain of edges from assembly occurrences to create the contour flange.



In the Assembly environment, the **Contour Flange** command is located on the **Home** tab→**Sheet Metal** group.

Occurrence property enhancements

The following enhancements were made to occurrence properties in assemblies:

- Two new shortcut commands—**Add Override** and **Remove Override**—are available for you to override the occurrence property values in the context of upper-level assemblies. This means you can configure your assembly display in reports, draft parts lists, and for placement in other subassemblies. These value overrides are stored and visible within the assembly where the change was made. They do not modify lower level documents.

Previously, occurrence properties in read-only subassemblies were not editable in upper-level assemblies.

- A new cell color-coding scheme indicates the override status of individual cells in the **Occurrence Properties** dialog box.
- A new option, **Save columns setting**, is available in the **Format Columns** dialog box for occurrence properties. You can save column order and display changes for reuse in other assemblies by typing a name for the saved settings in the box. Selecting the **Save** button writes the settings to a new text file, **OccurrencePropertyColumns.xml**, in the Solid Edge Preferences folder.
- You can define your own custom properties and assign unique values to each occurrence of user-defined custom properties in parts, subassemblies, and assemblies.

To learn more, see this companion topic in What's New, [Custom occurrence properties in assembly](#).

Custom occurrence properties in assembly

Custom occurrence properties can now be added to models used in an assembly with the new **Add Custom Occurrence Properties** button  in the **Occurrence Properties** dialog box. Custom occurrence properties enable you to assign unique metadata to a part or subassembly every time that part or subassembly is used in the assembly. (System occurrence properties, which provide higher level information, can only display the same value for each occurrence of the referenced property.)

New custom occurrence properties—BOM ID, Reference ID, Maintenance, and Notes—are defined as examples in the new **CustomOccurrenceProperty.xml** file delivered in the Solid Edge ST9

Preferences folder. You can edit the file to provide a list of selectable values for these custom occurrence properties or to add your own properties.



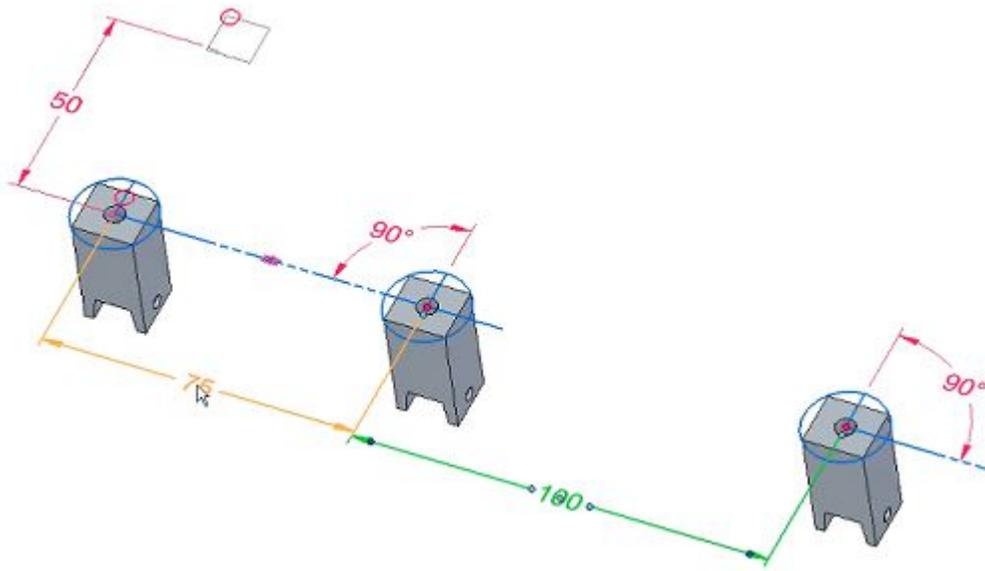
Custom occurrence properties are also available in the Draft environment for creating parts lists and annotating drawing views.

Duplicate Component command enhancements

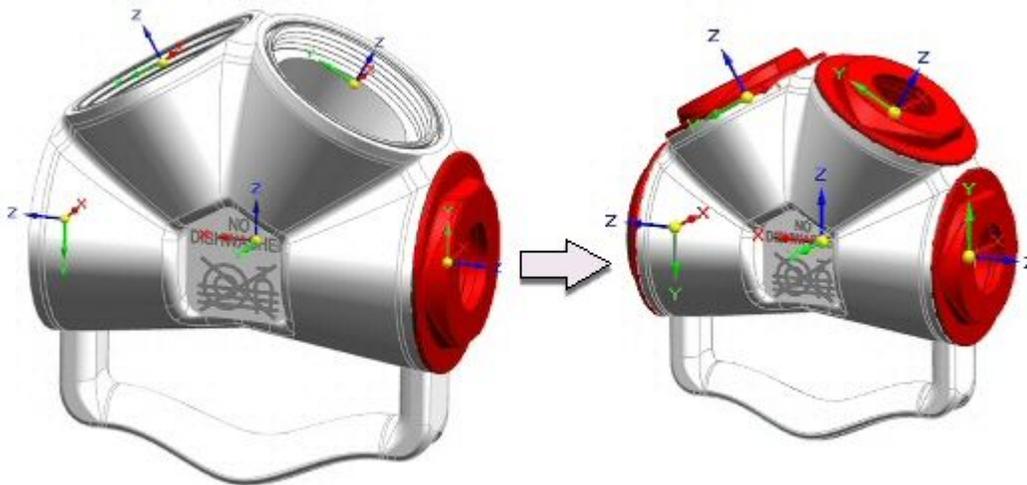
You now can use the **Duplicate Component** command  to locate an existing coordinate system or a block in an assembly sketch to position and orient the components.

Using the **Select From** and the **Select To** steps, you can:

- Select a block in an assembly or subassembly sketch, which is then matched to the orientation of the components.



- Select an existing coordinate system, which is then matched to the coordinate system of the components.

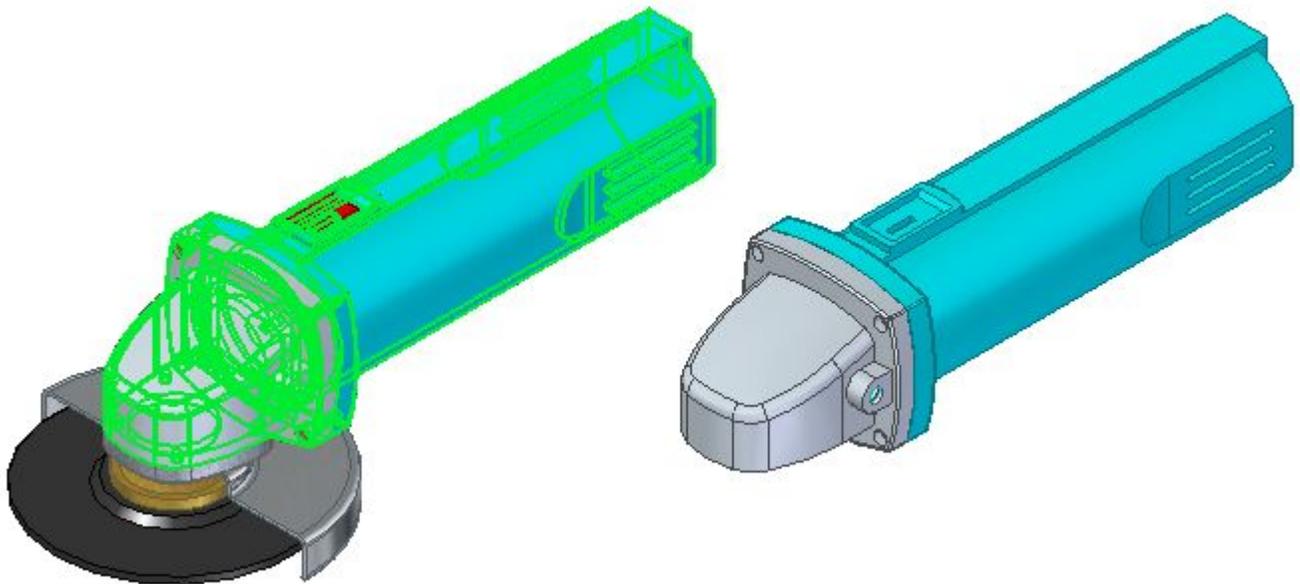


Movement of electrical connectors enhanced

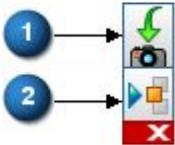
You can now move electrical connectors which contain wires inside a cable or bundle, which creates a blue dot. When you move the connector, the wires, along with the cable or bundle, maintain geometry relative to the connector.

Isolate command

Use the **Isolate** command  on the **Select** shortcut menu to display only the selected parts.

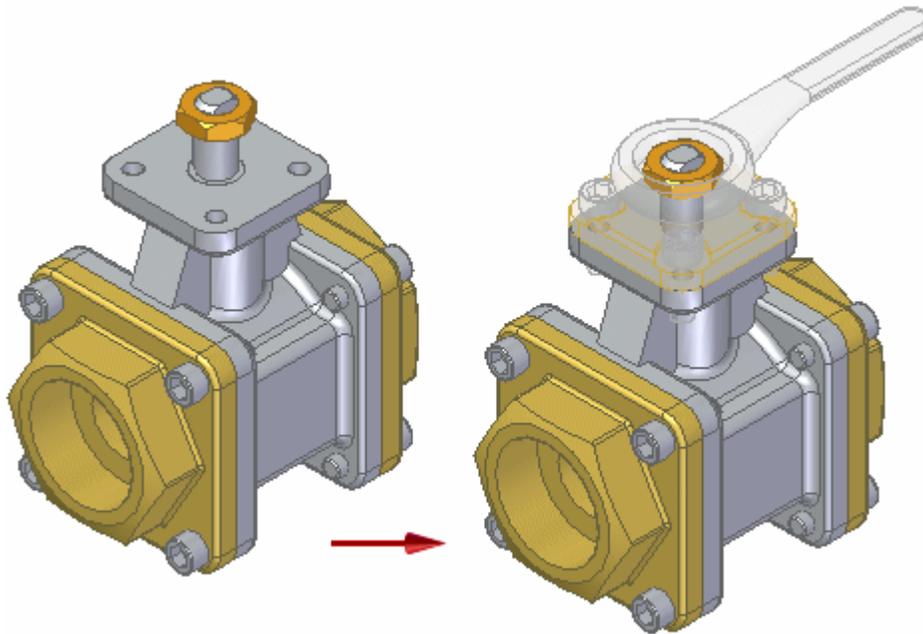


In the graphic window, click the **Restore** button (1) to return to the pre-isolate view or click the **Dismiss** button (2) to keep the isolated view.



Toggle Display command

You can use the **Toggle Display** button  on the status bar to toggle the display status of all components. Hidden components display as transparent, but visible for easy selection. This is useful for displaying hidden parts without the use of PathFinder.



For more information, see Toggle the display status of assembly components.

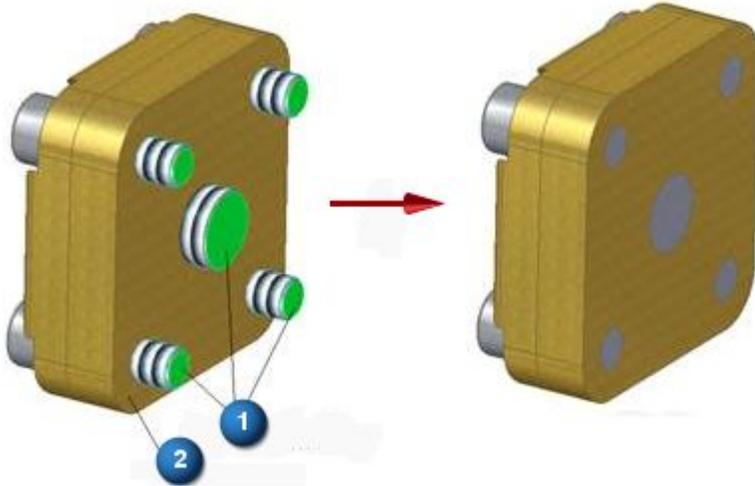
Piping enhanced

Enhancements made to the XpresRoute Piping application include:

- A new **Saved Settings** option on the **Piping Options** dialog box lists saved pipe settings, such as pipe size, standard, class, and end treatment. You can access the saved settings by selecting them from the list. The settings on the dialog box display the characteristics of the pipe you select.
- A new **Select preferred class fitting** option displays the **Standard Parts** dialog box so you can select a preferred fitting from the available fittings in the Standard Parts Library. If there are multiple fittings available, the fittings appear in a list and you can choose the fitting you want.
- Several enhancements were made to the **Companion** option.
 - When you place a flanged fitting or replace a non-flanged fitting with a flanged fitting from the Standard Parts Library, the **Companion Flanges** dialog box appears so you can select the companion flange you want to place.
 - When you replace a flanged fitting with a non-flanged fitting, any companion flanges associated with the flanged fitting are automatically removed.
 - After you place a companion flange, you can select the flange and replace it with another companion flange.
 - You can now use the **Orientation** option on the **Piping** command bar to rotate a companion flange.
- A new **WT** (wall thickness) parameter for piping parts in Standard Parts provides a more accurate list of fittings for a selected pipe size.

Replace Face command available in assemblies

The synchronous **Replace Face** command  is now available in assemblies. In the following example, you can use this command to replace the face where the bolts extend beyond the part. First, select all occurrences of the bolt faces(1), and then select the single face that is the replacement (2).



In the Assembly environment, the **Replace Face** command is located on the **Features** tab→**Modify** group. You do not need to be in-place activated to use it.

Standard Parts enhanced

Several enhancements have been made to Standard Parts:

- The *Standard Parts Installation Guide* is now available prior to installation in the **Standard Parts setup** folder on the DVD.
- While installing the Piping and Machinery library, during the **Custom Setup** step, the default installation location is the last used installation location.
- While installing the Piping and Machinery libraries, you have the option to automatically add the parts in the libraries to the database.
- During the **Custom Setup** step, there is now an option to deselect all features and subfeatures.
- During the **Custom Setup** step, the selection of default standards, such as ANSI or DIN, is based on the locale of the machine on which Standard Parts Administrator, Machinery library, and Piping library are being installed.
- While installing the new standard parts, for example ST9, and there are already existing standard parts on the machine or the server, during the **Custom Setup** step, the **This feature will not be available** option is turned on.

If there is an existing standard part configuration defined in the registry the **This feature will not be available** is turned on by default.

- During the **Custom Setup** step of Standard Parts Administrator, you can select the standards, just as you would when installing the Piping or Machinery libraries.

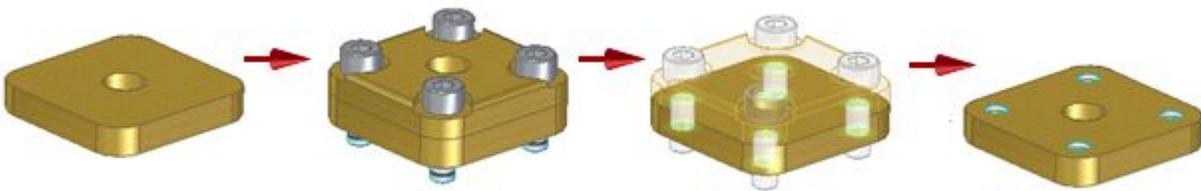
- Beginning in ST9, Solid Edge no longer supports Microsoft Access as a database for Standard Parts. Solid Edge now delivers SQL Express 2012 with Standard Parts installation.
- A series of eight new videos show you how to use Standard Parts.

Note

To find a list of videos and interactive simulations that are new in ST9 or were recently updated, see the help topic, Solid Edge videos.

Subtract command available in assemblies

In an assembly, you now can use the synchronous **Subtract** command  with the **Recognize Holes** option to recognize holes in part files. For example, cylindrical cutouts created with the **Subtract** command in the part file, and threaded and clearance holes that were created with the **Hole** command in a part file, can now be used to create corresponding holes in another part in the assembly.

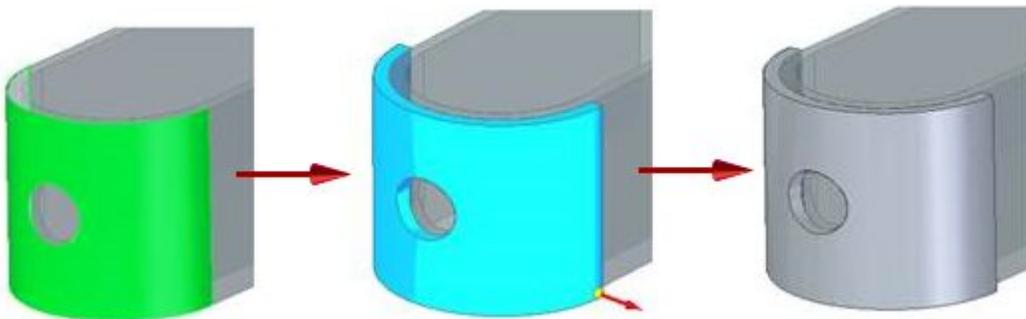


Thicken command available in assemblies

You now can use the **Thicken** command  to create a new part from faces of an adjacent part within the assembly.

In the context of an assembly, first select the **Create Part In-Place** command  to create an empty part file. Then you can select the **Home** tab→**Solids** group→**Add** list→**Thicken** command to:

- Locate and select faces from occurrences.
- Include, exclude, or thicken internal face loops selected from the face.
- Create a new part in **Assembly PathFinder** from the thickened faces.



Variable Table enhancements

More usability enhancements were added to the **Variable Table**. In the **Variable Table** dialog box, these changes include:

- New **Structured View**  option—Lists the variables under its parent feature. The tree is displayed in the Structure column.
- New **List View**  option—Shows a flat list similar to the existing view. The Structure column lists the feature to which the variable is attached.
- New sorting options include ascending, descending, custom sorting, and multi-level structured sorting by type and parents.
- New functions in the **Math&Trig** category, such as ASIN, ACOS, ATAN, SINH, COSH, TANH, PI(), E(), PHI(), RADIANT() and DEGREE().
- Variables selected for exposure using the **Expose** column and the **Exposed Variables** column are now available for downstream use. This ensures that their real values can be referenced in property text. For more information, see Using exposed variables in property text.

Sketching enhancements

Sketching enhancements

These enhancements were made to sketching in Solid Edge ST9.

-  **New command: Create As Construction**
-  **Construction command now has a command bar with a Chain select option.**
-  **New command: 3D Point**
-  **Stick fonts for etching**
-  **New sketch display options**
-  **Show Annotations command**
-  Type title text here. Use link tag.
-  Type title text here. Use link tag.

Create as Construction command

There is a new command called **Create as Construction** located in the **Draw** group. This command changes the sketch creation mode from profile to construction. When the button is on, you are in construction mode. When the button is off, you are in profile mode.

Construction command bar

The **Construction** command now has a command bar which provides the selection options, **Single** and **Chain**. Previously, only the **Single** option and fence selection were available. The new **Chain** option selects a connected chain of sketch elements to change between construction and profile.

3D Point command

A **3D Point** command was added to the 3D sketching tab.

You place 3D points by:

- Clicking in 3D space.
- Selecting a geometric keypoint.
- Entering an **XYZ** coordinate on the command bar. The coordinates are relative to the selected coordinate system.
- Locking to a coordinate system plane and entering planar coordinates on the command bar. The coordinates are relative to the selected coordinate system.

Stick fonts

Stick fonts are now delivered with Solid Edge fonts. These fonts provide the optimal solution for text etching in sheel metal.

Two new options support stick fonts:

- **Show only stick fonts in the list** displays the available fonts in the **Text** dialog box when you select the **Text Profile** command.
- **Export Etch text as geometry** creates an etch feature when you save the sheet metal part as a flat pattern in the **Save As Flat DXF Options** dialog box.

New ordered sketch display options

In some modeling scenarios, the number of sketch elements, colors, styles, and annotations can be rather significant. This may be the case when using blocks in the design. You may need a method of controlling the display of these sketches.

In the ordered environment, the sketch display can be controlled with the following new options found on the **Sketch** short cut menu:

- **Single color sketch display**
- **Multi-color sketch display**
- **Accelerated sketch display**
- **Show Annotations (also available in synchronous environment)**

Show Annotations command

The **Show Annotations** command is located on the shortcut menu of a sketch selected in **PathFinder**. This setting turns on or off the display of annotations, such as callouts and block labels, contained in the sketch. In the ordered environment, the annotations display while in sketch mode.

Note

If **Accelerated Sketch Display** is on, annotations do not display, even if **Show Annotations** is on.

Draft enhancements

Draft enhancements

The following enhancements were made in the Draft environment in Solid Edge ST9.

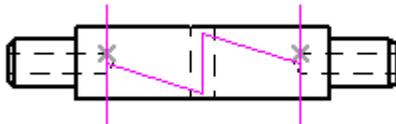
-  Associative break lines in broken views
-  Change drawing view orientation
-  Change properties on multiple drawing views simultaneously
-  Change the font size of table cells using direct edit
-  Count each subassembly occurrence in all parts lists
-  Edit a section view to change its type
-  Faster updates for large drawings
-  Hole callouts now show hole count
-  Image resolution option for Compare Drawings
-  Initial dimension stack distance
- List all image file types for insertion
-  New block connector shape type: Gap
-  New command: Delete Empty Layers
-  New command: Replace Background
-  New command: Tolerance Table
-  Retrieve Dimensions enhancement
-  Retrieve properties from the parent assembly
-  Shared saved settings for annotations

-  Short list when creating drawing views independent of assembly
-  Surface texture symbol shelf line
-  Symmetric diameter dimension enhancements
-  Text edit shortcut menu
-  Value formatting for exposed variables used in property text
-  Value formatting in table columns

Associative break lines for broken views

Now you can associate break lines with drawing view geometry. This ensures that when the model depicted in the drawing view changes, the broken portion of the drawing view changes along with it.

When creating or modifying a broken view, you can constrain the break lines by snapping to keypoints on the drawing view geometry.



Change drawing view orientation

Now you can change the orientation of a drawing view after it is created. When you change the view orientation of a principal view, the orthographic drawing views that are aligned with it react by changing their orientation.

Previously, you had to delete the drawing view and then regenerate it to change its orientation.

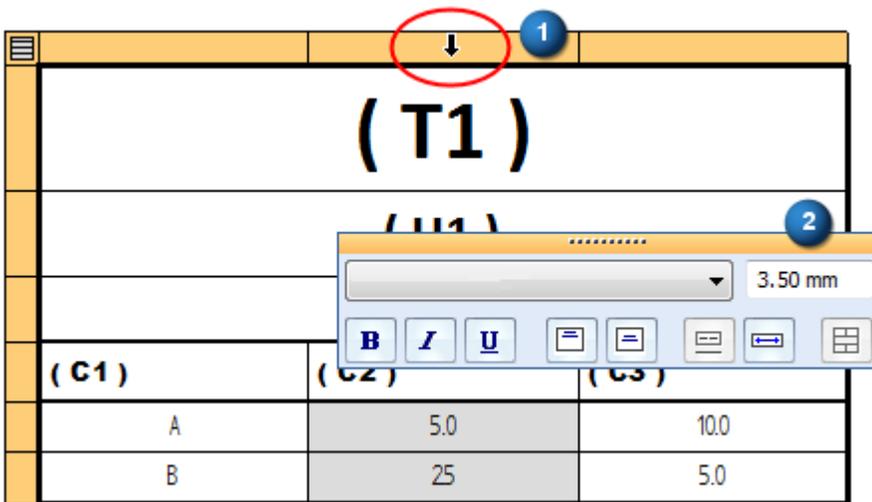
The following options are now available on the command bar when you select a drawing view:

- **View Orientation** 
- **Drawing View Layout** 

These options are not available for derived views, PMI model views, views with applied configurations, and 2D views.

Change the font size of table cells with direct edit

Now you can change the font size of table data using the orange edit frame that appears when you double-click a table. You can change the font size of data in one or more columns, or for all data cells in the table.



Previously, you could only change the font size of the text in the title rows using direct edit.

Change properties on multiple drawing views simultaneously

Now you can change the properties of multiple drawing views at the same time using the **Drawing View Properties** dialog box. When you select the **Apply** button in the dialog box, the changes are applied to all of the selected drawing views.

The drawing views that you select before you select the **Properties** command determine which tabs are displayed, as well as which options on those tabs are available for making changes.

Previously, only the **General** tab was displayed when you selected more than one drawing view.

Count all occurrences in all parts lists

When you assign a user-defined value to the Quantity occurrence property for a subassembly in the **Occurrence Properties** dialog box, the quantity override is now applied to the subassembly, as well as to all part and component occurrences, in top-level and atomic parts lists. The following information is calculated and shown in the parts list Quantity column:

- Total subassembly quantity
- Total part quantity

There is no additional interface option that you select for this calculation when you generate either a top-level or atomic parts lists.

Note

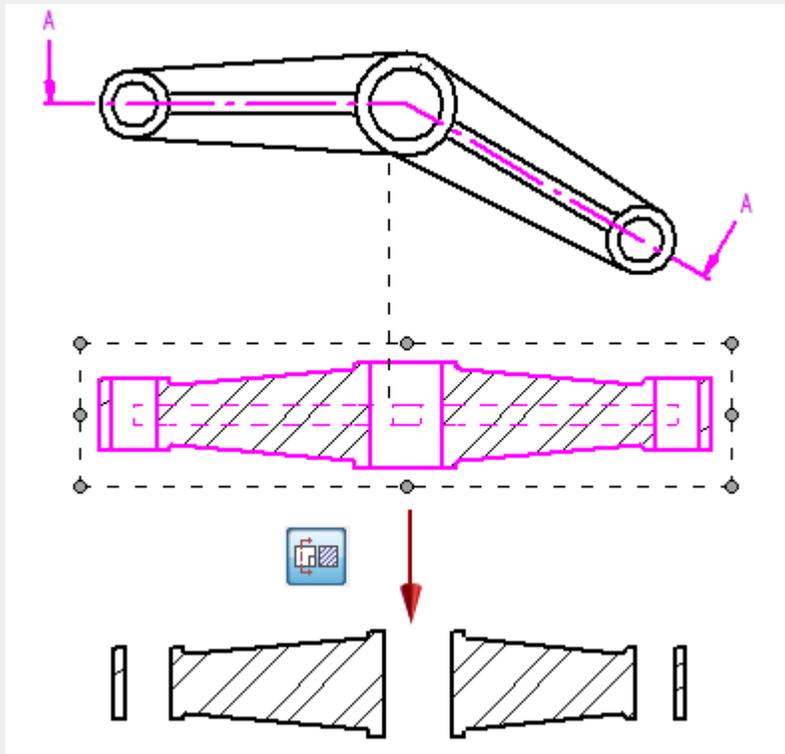
This is the same calculation method that was previously only available for exploded parts lists. For exploded parts lists, you must select the option, **Multiply by subassembly count**, on the **List Control** tab in the **Parts List Properties** dialog box. For top-level and atomic parts lists, it is the default calculation method that is used.

Edit a section view to change its type

Now you can select a section view and change it to a different type of section view without having to recreate the view.

Example

- You can select a simple section view and convert it to a thin-section (paper-thin section) view using the **Section Only** button on the command bar.



- You also can convert a simple section view to a revolved section view using the **Revolved Section View** button  on the command bar.

Previously, these buttons were only available during section view creation.

Exposed variables referenced by property text

Variables that are exposed in the **Variable Table** are now available directly as properties in parts lists, tables, and annotations, bypassing the **Custom** tab in the **File Properties** dialog box.

This means you can:

- Reference the real value of an exposed variable. Previously, exposed variable values were only available as text strings, not as real numbers, in draft, sketch, and PMI.
- Format the extracted value using the Format Values dialog box. For example, you can specify precision, no units, dual units, and tolerance.

- Choose the source of the property that you want to use on the **Columns** tab in the **Parts List Properties** dialog box and in the **Select Property Text** dialog box.

Example

Custom file properties that are listed with the same name also indicate their source.

Accuracy (Custom Property)

Source=**Custom** tab, **File Properties** dialog box

Accuracy (Exposed Variable)

Source=**Variable Table**

This enhancement introduces a new property text output modifier, /EV (exposed variable). This information was added to the table in Format codes to modify reference and property text output.

Note

These changes do not affect previous property text workflows. You still can reference exposed variables on the **Custom** tab in the **File Properties** dialog box.

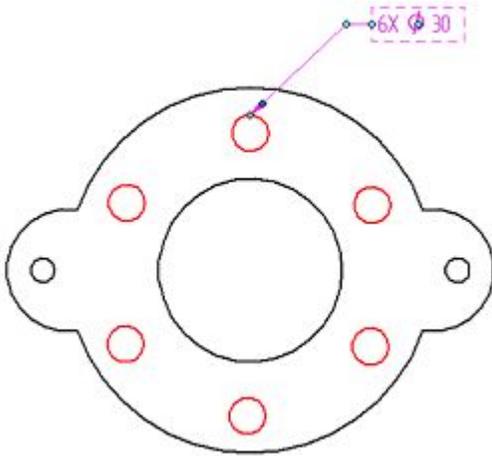
Faster update for large drawings

Multi-core processing to support faster updates of drawing views of very large assemblies is available in ST9. You can turn multi-core processing on and off using the new option, **Enable multi-core drawing view processing**, which is located on the **General** tab in the **Solid Edge Options** dialog box.

Hole callouts now show hole count

Now you can show the hole count as well as the hole type in hole callouts, feature callout dimensions, and dimension prefixes in draft, sketch, and PMI. Eligible hole features are those created with the **Hole** command in synchronous and ordered part models, and hole features created in an imported body using the **Recognize Holes** command. For assembly models, holes in parts are counted, as well as assembly hole features placed on a part. Hole count is based on hole type, location, orientation, and parameters defined in the **Hole Options** dialog box.

When you select a hole callout, the holes it references are highlighted.



The hole count is supported by four new property text codes (%QN, %QC, %QA, and %QP). For more information, see Property text codes.

- You can select and insert the property text codes using the **Select Symbols and Values** dialog box.
- You can reference the property text codes in the **Modify Dimension Style** dialog box, the **Dimension Prefix** dialog box, and the **Callout Properties** dialog box.

Note

To use the new hole count codes with legacy draft files, you must force the drawing views to update using Ctrl+Shift+**Update Views** command.

Image resolution option for comparing drawings

A new **Smoothness** slider was added to the **Compare Drawings** dialog box to control the image resolution of the output file.

Initial dimension stack distance

A new option, **Initial stack distance**, is available on the Lines and Coordinate tab (Dimension Style and Dimension Properties) to specify a unique distance between the innermost dimension in a dimension stack and the dimensioned geometry. This option applies to linear, angular, symmetric diameter, concentric diameter, and other stacked dimensions that are arranged automatically.

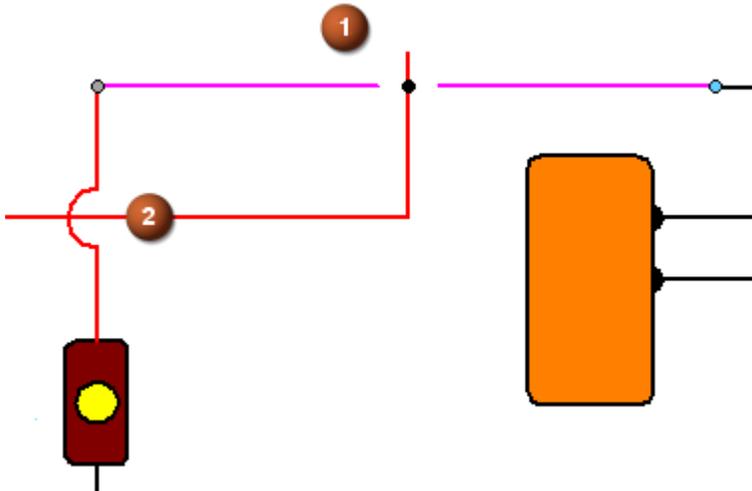
The new option works with the **Stack pitch** option, which controls the distance between *all dimensions in the dimension stack*.

The following commands use the **Stack pitch** and **Initial stack distance** values from the active Dimension style:

- **Arrange Dimensions** (in draft and sketch)
- **Retrieve Dimensions** (in draft)
- **Auto-Dimension** (in draft and sketch)

New connector shape type: Gap

When using the **Connector** command, a new **Gap** shape (1) is available to show how two connectors cross. Previously, only a **Jump** shape (2) was available.



For more information, see Connector command bar.

List all image file types for insertion

When browsing for an image file to insert into a draft or model document using the

Insert Image command , you now can set the **Files of type** list to **All Image Files** (*bmp,* .png,* .jpg,* .jpeg,* .jpe,* .tif,* .tiff).

This is useful if you do not know the file type of the image you are trying to locate.

New command: Delete Empty Layers

A new command, **Delete Empty Layers** , deletes all layers that are not associated with any geometry. This command is available on the **Layers** tab in the 2D and 3D environments.

Example

When you migrate AutoCAD files to a Solid Edge Draft document, the original files tend to have many layers in them. You can use the **Delete Empty Layers** command to remove the layers that have no contents.

For more information, see the help topic, Delete one or more layers.

New command: Replace Background

You can use the new **Replace Background** command  to copy the contents from one or more background sheets in one draft document to another draft document. The command is available on the shortcut menu of a selected background sheet tab.

component at each level. In a large assembly, this makes it faster and easier to navigate the list of parts. This option is a preference that is remembered between sessions.

Show custom occurrence properties in draft

Custom occurrence properties that are added to an assembly model in the **Occurrence Properties** dialog box are now available in the Draft environment for use in parts lists and callouts.

With the introduction of occurrence properties in the assembly, each occurrence of a part model used in an assembly can have a different value for a property. This is called a custom occurrence property. Previously, all occurrences of a part model in a parts list displayed the same value for a referenced property.

You can use custom occurrence properties:

- In a parts list, to show the different occurrence property values for a part within a single cell.
- In balloons and callouts, to show the unique occurrence property value of a part in a drawing view.

You can add custom occurrence properties by selecting them from the **Properties** list:

- On the **Columns** tab in the **Parts List Properties** dialog box.
- In the **Select Property Text** dialog box.

Look for the text (**Occurrence Property**) appended to the property name.

Example

Maintenance (Occurrence Property)

Paint (Occurrence Property)

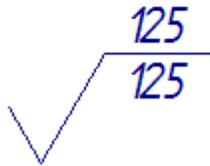
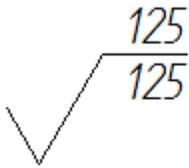
For more information, see Example: Show custom occurrence properties in a parts list.

This enhancement introduces a new property text output modifier, /OP (occurrence property). This information was added to the table in Format codes to modify reference and property text output.

Surface texture symbol shelf line

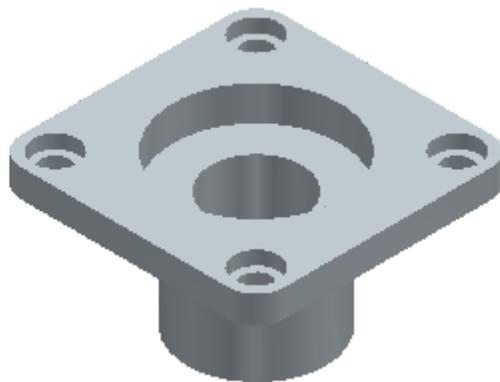
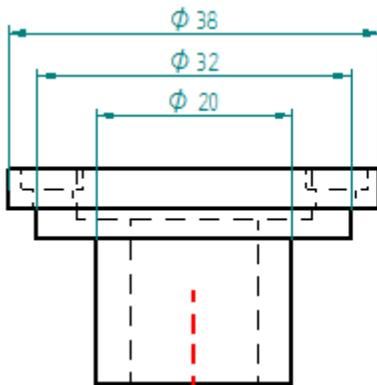
In the dimension style, you can specify the length of the shelf line that appears over or under the text in a surface texture symbol annotation. The new property, **Symbol overline extension**, is located on the **Annotation** tab in the **Dimension Style** dialog box. When the **Font** is set to **Italic**, you can use this option to ensure that the line extends the full length of the annotation text.

The value that you enter is multiplied by the font size. In the first surface texture symbol shown below, the length of the line is the default value, 0.0.

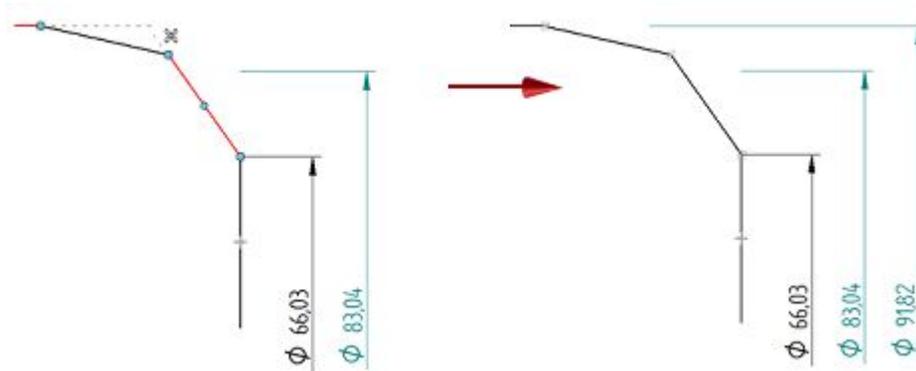


Symmetric diameter dimension enhancements

Many enhancements were made to the Symmetric Diameter command



- Improved workflow—It is faster and easier to place symmetric diameter dimensions.
 - o When you identify the center axis for originating the measurements, it is shown in the highlight color.
 - o Before placing the first dimension, you can press Alt to change the alignment by 90 degrees, or you can press Shift to locate two points.
 - o After placing the first dimension, additional dimensions are placed with a single click on the next geometric element to measure.
- Symmetric diameter dimensions are stacked, aligned, and spaced automatically during placement and when new dimensions are added to the stack.
- In draft, you can place symmetric diameter dimensions between drawing views.
- You can place 2D and 3D PMI symmetric diameter dimensions using intersection points.



- New formatting options are available for symmetric diameter dimensions:
 - **Alternate text positions**—Makes dimensions easier to read by alternating the position of dimension text in stacked and grouped symmetric diameter dimensions. This option is located on the Lines and Coordinate tab (Dimension Style and Dimension Properties).
 - **Underline symbol and prefix**—Specifies that the dimension line for the symmetric diameter dimension extends under all of the dimension text. This option is located on the **Lines and Coordinate** tab.
 - **Suppress symmetric diameter**—Prevents the diameter symbol from displaying. This option is located on the Terminator and Symbol tab (Dimension Style and Dimension Properties).
- Symmetric diameter dimensions are created in alignment groups. This means you can:
 - Drag any dimension in the alignment group, and all of the dimensions in the stack adjust along with it.
 - Alt+drag a single dimension in the alignment group to move it independently of the others in the group.
 - Use Shift+drag to adjust the spacing between two dimensions.
 - Use the following commands to manage the membership of dimensions in the dimension alignment group:
 - **Remove from Alignment Set**
 - **Break Alignment Set**
 - **Split Alignment Set**

For more information, see the help topic, Place symmetric diameter dimensions.

Shared saved settings for annotations

Enhancements were made to saved settings for annotations and dimension text in draft, sketch, and for PMI.

- Saved settings are now available for the following types of objects:

- o Balloons—Saves properties for **Balloon height**, **Balloon shape**, **Number of sides**, **Upper**, **Prefix**, **Lower**, **Suffix**, **Orientation**, and **Text alignment**.
- o Edge condition symbols—Saves properties for **Direction tolerance 1**, **Upper tolerance**, **Lower tolerance**, **Direction tolerance 2**, **No text on symbol**, and **All around symbol with leader**. Saved settings for edge condition symbols are available for both the **Current ISO/DIN** and the **Previous ISO/DIN** drawing standards.
- o Dimension prefixes—Saves properties for **Superfix**, **Prefix**, **Suffix**, **Subfix**, **Subfix 2**, and **Subfix horizontal alignment**.
- Annotation saved settings are saved to individual text files, so that they can be shared with other users. These text files are located in the **..\Program Files\Solid Edge ST9\Template\Reports** folder. They are named for the annotation type, for example, **DraftBalloon.txt**, **DimensionPrefix.txt**, and **DraftEdgeCondition.txt**.

Note

All annotation saved settings are now saved to external files, so that they can be shared with other users. Before Solid Edge ST9, the saved settings for feature control frames and surface texture symbols only existed in the document where they were saved. For more information about these previously saved settings, see Working with *legacy annotation saved settings.

Text edit shortcut menu

A shortcut menu with the following commands is available in dialog boxes where you can enter and edit text:

- **Undo**
- **Cut**
- **Copy**
- **Paste**
- **Delete**
- **Select All**

This shortcut menu is available in text fields in **Properties** dialog boxes. For example:

- On the **General** tab, when modifying callout text, technical requirements text, and edge condition text.
- On the **Title** tab, when modifying table titles.
- On the **Caption** tab, when modifying drawing view captions.

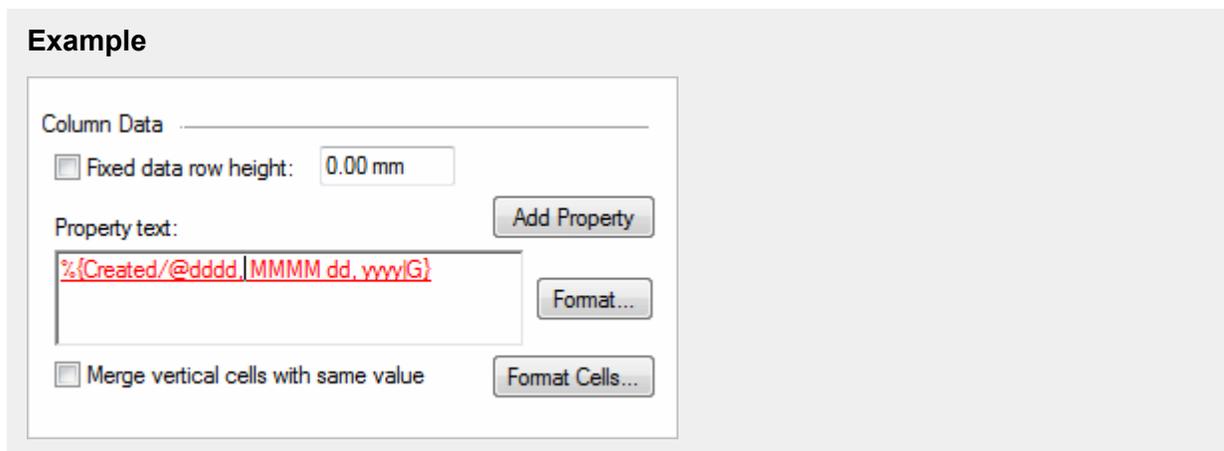
Value formatting in table data columns

Now you can format property text values displayed in table columns using the **Format Values** dialog box. Previously, only property text in table titles could be formatted.

You can specify the following types of formatting for the resolved values of property text in table data cells:

- Precision, no units, or dual units for real number values.
- Day, date, and time format for date values.
- Text case for text values.

You can open the Format Values dialog box using the **Format** button on the **Columns** tab in the **Properties** dialog box for the selected table.



You also can type the format codes to modify reference and property text output directly into the **Property text** box for the column definition.

Column data value formatting is available in all tables except hole tables and family of parts tables.

Document management enhancements

Document management enhancements

Document management enhancements and new features included in Solid Edge ST9 are:

Solid Edge Embedded Client

-  Software compatibility
-  Teamcenter Preferences
-  Updated user interface
-  New File Properties dialog box
-  Improved Open dialog box

-  Updated User Session information
-  Open multiple revisions
-  Start a workflow process
-  Lists of Values (LOV) with descriptions
-  Support for external collaboration
-  Sort in Cache Assistant
-  Enhanced support for Active Workspace
-  Sort in Cache Assistant
-  Enhancements to item type list order
-  Updated data preparation tools
-  New Where Used options

Structure Editor–Teamcenter

-  Structure Editor standalone installation discontinued
-  Sort indented lists
-  Scroll Lock command removed
-  Updated User Session information
-  Access to Managed page of Solid Edge Options

Add to Teamcenter-Interactive

-  Sort indented lists
-  Support for naming rules
-  Updated User Session information
-  Access to Managed page of Solid Edge Options

Insight

-  Insight software compatibility
-  Insight standalone installation discontinued
-  Sort in Cache Assistant
-  Updated data preparation tools

Solid Edge SP

-  Updated data preparation tools
-  Faster delete performance
-  Retaining signoff information
-  Enhanced Solid Edge SP Feature Library
-  New way to view draft and PDF information
-  Improved overall Solid Edge SP usability

View and Markup and Revision Manager

-  Where's Revision Manager?

Solid Edge Embedded Client

Software Compatibility

Solid Edge Embedded Client ST9 is compatible with the following:

Solid Edge	Teamcenter Rapid Start	Teamcenter 10.1	Teamcenter 11.2	
ST9	10.1.5.1	10.1.5.1	11.2.1	
	11.2.1	AW 3.1	AW 3.1	
<p>Teamcenter Rapid Start replaces Teamcenter Express starting with release 10.1.</p> <p>Teamcenter 11.2 adds UTF-8 support for Windows (encoding used between TcServer - Database).</p> <p>Solid Edge ST8MP2 is certified with Teamcenter 11.2 for production use.</p> <p>Active Workspace (AW) 3.1 is compatible with 10.1.5/11.2.1 with reduced functionality. Use 10.1.6/11.2.2 for full functionality.</p>				

Solid Edge Embedded Client ST9 does *not* support:

- Teamcenter 8.x, 9.x, 10.0, 11.1
Solid Edge ST8 was the last release supporting Teamcenter 9.1
- Any Teamcenter release prior to 8.x.
- Teamcenter Express

Teamcenter preferences

New Teamcenter preferences introduced with this release:

ActiveWorkspaceHosting.SEEC.URL

Specifies the URL used by Solid Edge when hosting an Active Workspace client. This preference is evaluated prior to the ActiveWorkspaceHosting.URL preference.

AWC_SEEC_OpenSupportedTypes

Enables the specified object types to be opened in Solid Edge when selected in Active Workspace. Only Item, Item Revision, subtypes of Item and Item Revisions, and Solid Edge Dataset types are supported.

SEEC_BOM_Precision_Override

Specifies the BOM precision used during the creation of a new Solid Edge Assembly. Valid values are 0, 1, or 2. Set to 0 to use the same setting as Teamcenter Rich Client. Set to 1 to override to Imprecise. Set to 2 to override to Precise. Default is 0.

SEEC_Enable_CreatePackage

When set to True (1), adds Create Package command to Solid Edge Cache Assistant toolbar. Default is 0.

SEEC_Enable_MultipleRevisions

When set to True (1), Solid Edge works with multiple revisions in a single Solid Edge session. Default is 0.

SEEC_Enable_AssemblyConstructions

Determines if an assembly occurrence marked as a construction is added to the Teamcenter BOM View Revision. When all occurrences are marked as constructions, the Item Revision is connected to the assembly using a relation.

SEEC_Item_Type_Sort

Determines the order in which item types are listed in common property dialog boxes. Set to 0 to use the order defined by each SEEC_ItemType preference. Set to 1 to build a hierarchical and sorted list. Default is 1.

SEEC_ItemTypeList_ExcludedTypes

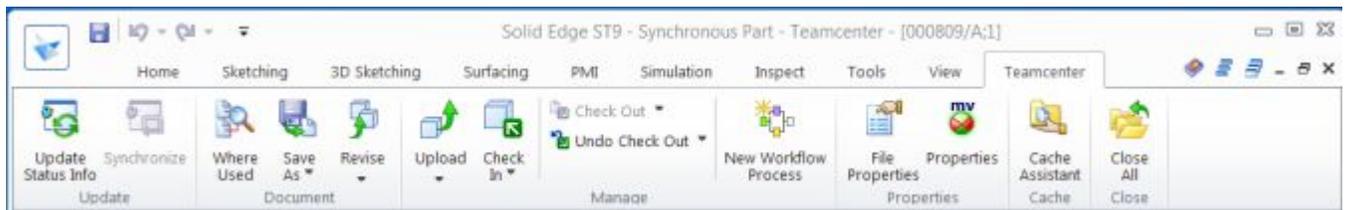
When a SEEC_ItemTypeList preference is equal to All, item types listed by this preference are removed. When a new document is saved to Teamcenter, the resulting subset of Item types is available to you.

For additional information, see the Teamcenter Preferences section of the *Teamcenter Integration for Solid Edge Guide for Users and Administrators*.

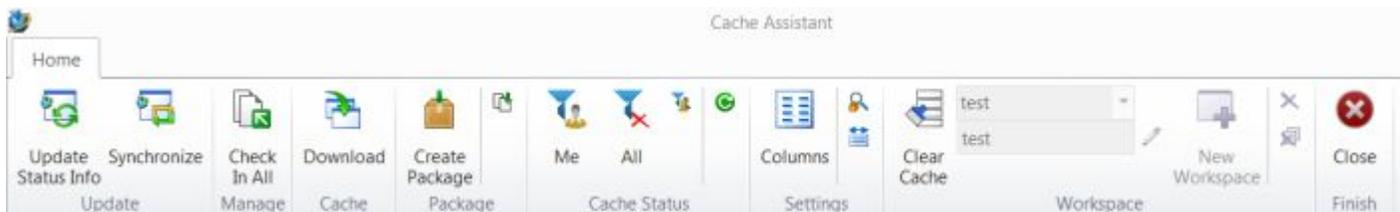
Updated user interface

The user interface for Solid Edge Embedded Client is updated to provide direct and easy access to commands and enhance your user experience. Updates include:

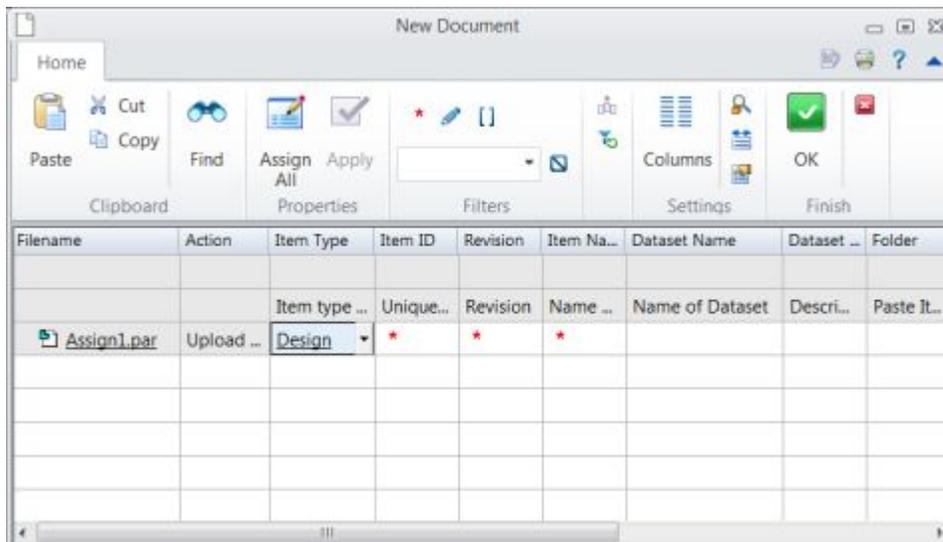
- Teamcenter tab on the Solid Edge command ribbon to collect all Teamcenter commands in a single location.



- Completely revamped and interactive File Properties (SEEC) dialog box.
- User Session Information that displays as a docking pane to provide you easy access to details regarding your connection to Teamcenter.
- Updated Cache Assistant that includes a ribbon of commands.



- Updated common property dialog boxes such as New Document and Upload Document, that include commands in a ribbon, plus the commands to hide columns and display required properties vertically.



- Improved Open dialog box.

New File Properties dialog box

The **File Properties** dialog box is updated and greatly enhanced with new functionality.

Property	Value	Type	Mapping	Owner
Teamcenter (14)				
Item (4)				
Type	000809			
ID	000809	String[128]	Document Number	
Name	* 000809	String[128]	Title	
Description		String[240]	ItemDescr	BOTH
ItemRevision (5)				
Revision	A	String[128]	Revision	
owning_gro...	Engineering	String[128]	Owning_Group	PDM
Description		String[240]	ItemRevDescr	BOTH
owning_use...	WEBB, N	String[128]	Author	PDM
Name	000809	String[260]		
Dataset (5)				
Name	000809/A	String[260]	Dataset Name	
Description		String[260]		
Date Released		Date	Dataset_Status_Date	PDM
last_release...	Working	String[32]	Dataset_Status	PDM
Version	3	String[260]		

The new dialog box:

- Is interactive and integrated with Teamcenter.
- Displays Solid Edge file properties in a vertical format.
- Organizes properties in groups.
- Respects the preference that determines the use of displayable or real names.
- Integrates Teamcenter List of Values.
- Enables you to print the contents or copy the contents to your clipboard.

Note

Accessing the enhanced **File Properties** dialog box requires that you have a document open in the Teamcenter managed environment.

For more information, see the help topic, File Properties (SEEC) dialog box.

Improved Open dialog box

The updated Open dialog box contains:

- A command ribbon.
- Improved layout, including enhanced presentation of property data.

- An Item ID text box that quickly queries for an Item ID without having to use the Advanced Search command.
- Overrides for load options.

For more information, see the help topics, Open File dialog box (SEEC) and Quickly search for an Item ID.

Updated User Session information

User session information is now easier to obtain than ever before.

- You can log in to Teamcenter and access User Session information from the Solid Edge start page.
- Once you are connected to Teamcenter, your current cache workspace is displayed.
- You can access and change user settings any time a document is checked out to you.

For more information, see the help topic, User Session Information.

Open multiple revisions

You can now open multiple revisions of a document in the same Solid Edge session. This new capability:

- Helps facilitate design reviews.
- Enables designers to view different revisions of a document in separate windows.

Additional information is available in the help topic, Using multiple revisions of documents.

The Teamcenter preference that enables multiple revision support, **SEEC_Enable_MultipleRevisions**, is described in the *Teamcenter Integration for Solid Edge Guide for Users and Administrators*.

Start a workflow process

You can initiate a Teamcenter workflow from within Solid Edge. Workflows pass documents, information and tasks between participants during the course of a particular process. The **New Workflow Process** command:

- Allows you to choose the item revisions to be targets of the workflow process.
- Submits the targets to a single workflow.
- Processes exceptions and presents corrective action.

The **New Workflow Process** command is available from:

- Application menu→Manage.
- The Teamcenter ribbon.
- A shortcut command in **PathFinder** in the assembly environment.

For more information, see the help topic, New Workflow Process command.

Lists of Values (LOV) with descriptions

You can now integrate descriptions with Lists of Values so that a row in the list includes both the value and its corresponding description. There is no need to map a separate property to contain the description. Use this capability to assist users who could benefit from a description to choose an appropriate value from a list.

This new capability is implemented for the following LOV types:

- Exhaustive
- Suggestive
- Cascading or Hierarchical
- LOV with Property Dependent Attachment
- Interdependent LOVs

This capability is also enabled for Structure Editor - Teamcenter and Add to Teamcenter - Interactive.

Additional information is available in the help topic, Lists of Values (LOV). Teamcenter preferences that influence Lists of Values are described in the *Teamcenter Integration for Solid Edge Guide for Users and Administrators*.

Support for external collaboration

The **Create Package** command in **Cache Assistant** gives you the functionality to create a package of data to send to design partners and then reintegrate the data into Teamcenter once the collaboration is complete. Using the **Create Package** command, you can:

- Create an unmanaged copy for archival.
- Create an unmanaged copy for a collaboration partner who performs work and then returns the data.
- Populate a workspace so that the data can be used offline. For example, during a design review.

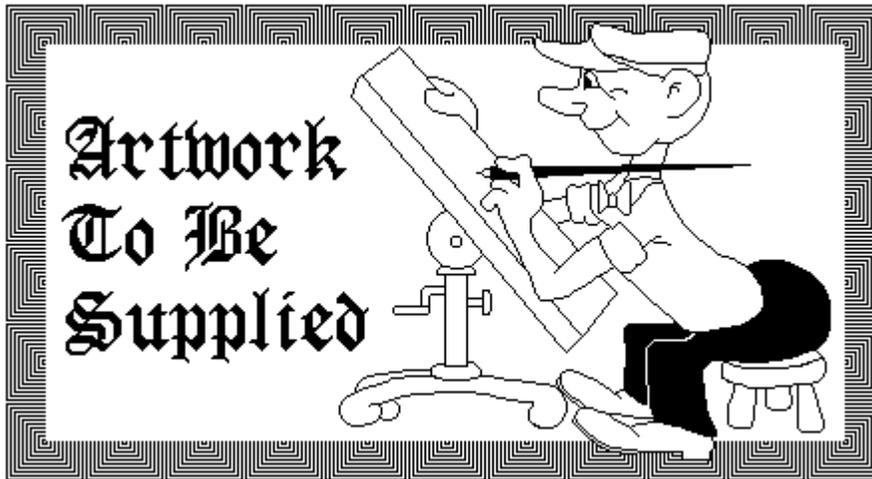
For more information, see the help topic, Create a package of the cache workspace.

Enhanced support for Active Workspace

Use Active Workspace for change management, workflow, requirements management, and visualization to help you accomplish your tasks. Solid Edge ST9 supports two integrations of Active Workspace:

- Hosted Active Workspace

After you start Solid Edge and log into Teamcenter, access the integrated Teamcenter client, Active Workspace, through a docking pane.



- Standalone Active Workspace

Accessed through a browser window where you log into Active Workspace, find Solid Edge data that is managed by Teamcenter, and send the document directly to Solid Edge.

You can select an Item, Item Revision, or Solid Edge dataset, and then click **Open in Solid Edge**, to open the associated design data in Solid Edge.

Both instances support Active Workspace 3.1 which provides the capability to:

- Select multiple documents for opening.
- Open a configured structure.
- Drag and drop to the Solid Edge graphics window.
- Use the Add Component command to add the selected model to the active assembly.

Additional information is available in Active Workspace in Solid Edge. Teamcenter preferences are described in the *Teamcenter Integration for Solid Edge Guide for Users and Administrators*.

Sort in Cache Assistant

Cache Assistant now supports multi-column sorting. Using the shortcut command, **Sort**, you can select a column and sort the contents:

- In ascending order, where the lowest number or beginning of the alphabet appears first in the sorted list.
- In descending order, where the highest number or end of the alphabet appears first in the sorted list.
- In alphabetic and numeric order, so that data containing alphabetical letters mixed with numbers is sorted logically.

Example

1
1a
2
2a
2b

- Alphabetically, where the beginning of the alphabet appears first in the sorted list.

Note

When a column has repeating values for rows, you can select additional columns for sorting values. **Shift+double-click** subsequent column headers to add columns as additional sort columns.

For more information, see the help topic, Sort multiple columns.

Enhancements to item type list order

Teamcenter administrators can take advantage of two new enhancements to item type list order. Now you can:

- Determine the order in which item types are listed in common property dialog boxes. The order is per document type and is defined by the Teamcenter preference **SEEC_Item_Type_Sort**.
- Add descriptive or *helper* text to item type lists when you define the Teamcenter preference **SEEC_Item_Type_Sort**. Descriptive information that displays in the item type lists is for display only and cannot be selected.

This new functionality applies to Solid Edge Embedded Client, Structure Editor, and Add to Teamcenter-Interactive.

Additional information is available in Set the display order in item type lists and Enter descriptive information into item type lists. Teamcenter preferences are described in the *Teamcenter Integration for Solid Edge Guide for Users and Administrators*.

Updated data preparation tools

The new Solid Edge Data Preparation Utility is delivered when you install Solid Edge and assists you in analyzing and preparing your unmanaged Solid Edge documents before importing them into a Teamcenter, Insight, or Solid Edge SP-managed environment. The new utility:

- Combines the three previously separate tools: Analyze, Link Fix-Up, and Modify, into one tool.
- Provides the functionality to generate reports for custom properties and reverse link information.
- Maintains command line functionality using the program **\Program Files\Solid Edge ST9\Program\DataPrepUtility.exe** along with options.

For more information, see the help topic, Solid Edge Data Preparation Utility.

New Where Used options

You can now choose a Revision Rule as input to the **Where Used** command. On the **Where Used Results** dialog box you can:

- Select the **Use Precise** check box to display the BOM view revisions where the current document is placed precisely.
- Clear the **Use Precise** check box and select the revision rule to display the BOM view revisions where the current document is used.

For more information, see the help topic, Where Used Results dialog box.

Structure Editor - Teamcenter

Sort indented lists

Structure Editor now supports sorting in an indented list. Using the shortcut command, **Sort**, you can select a column and sort the contents:

- In ascending order, where the lowest number or beginning of the alphabet appears first in the sorted list.
- In descending order, where the highest number or end of the alphabet appears first in the sorted list.
- In alphabetic and numeric order, so that data containing alphabetical letters mixed with numbers is sorted logically.

Example

1
1a
2
2a
2b

- Alphabetically, where the beginning of the alphabet appears first in the sorted list.

Note

When a column has repeating values for rows, you can select additional columns for sorting values. **Shift+double-click** subsequent column headers to add columns as additional sort columns.

For more information, see the help topic, Sort multiple columns.

Solid Edge Structure Editor standalone installation discontinued

The option to download Solid Edge Structure Editor as a standalone application is discontinued. You now install Solid Edge and the Teamcenter Integration for Solid Edge (SEEC), and then run Structure Editor from your Start menu.

Scroll Lock command removed

Structure Editor windows now automatically scroll in synchronization with one another. The **Scroll Lock** command has been removed from the **View** menu.

Updated User Session information

User Session information is now easier to obtain than ever before.

- You can log on to Teamcenter and access User Session information from the start page.
- Once you are connected to Teamcenter, your current cache workspace is displayed.
- You can access and change user settings any time a document is checked out to you.

For more information, see the help topic, *User Session Information* available in Structure Editor and Add to Teamcenter - Interactive help.

Access to Managed page of Solid Edge Options

The **Manage** page of the **Options** dialog box is now accessible from within Add to Teamcenter-Interactive and Structure Editor. Several new options are available to you, including options for:

- Teamcenter client communication system settings (TCCS)
- Creation of Solid Edge Assembly BOM View Revisions
- Use of Variant Rules
- Structures that contain multiple revisions of an item

For more information, see the help topic, *Manage page (Solid Edge Options dialog box)*.

Add to Teamcenter Interactive

Sort within indented lists

Add to Teamcenter Interactive now supports sorting within an indented list. Using the shortcut command, **Sort**, you can select a column and sort the contents:

- In ascending order, where the lowest number or beginning of the alphabet appears first in the sorted list.
- In descending order, where the highest number or end of the alphabet appears first in the sorted list.

- In alphabetic and numeric order, so that data containing alphabetical letters mixed with numbers is sorted logically.

Example

1
1a
2
2a
2b

- Alphabetically, where the beginning of the alphabet appears first in the sorted list.

Note

When a column has repeating values for rows, you can select additional columns for sorting values. **Shift+double-click** subsequent column headers to add columns as additional sort columns.

For more information, see the help topic, Sort multiple columns.

Support for naming rules

You can use the custom properties **Pattern_ItemID** and **Pattern_Revision** to automatically assign naming rules to files when an item type has a naming rule on either the Item ID or Revision, or both, and you are importing documents into Teamcenter. Supported file types include:

- Solid Edge document types (.par, .asm, .dft, .psm, .pwd)
- Microsoft Word (.doc, .docx)
- Microsoft Excel (.xls, .xlsx)
- Image (.bmp, .tiff, .jpg, and so on).
The properties are stored in the *.tmp* file when you use the data preparation tools to populate the values.

For additional information, see the help topic, Naming rules and revision naming rules.

Updated User Session information

User session information is now easier to obtain than ever before.

- You can log in to Teamcenter and access User Session information from the Solid Edge start page.
- Once you are connected to Teamcenter, your current cache workspace is displayed.
- You can access and change user settings any time a document is checked out to you.

For more information, see the help topic, User Session Information.

Access to Managed page of Solid Edge Options

The **Manage** page of the **Options** dialog box is now accessible from within Add to Teamcenter-Interactive and Structure Editor. Several new options are available to you, including options for:

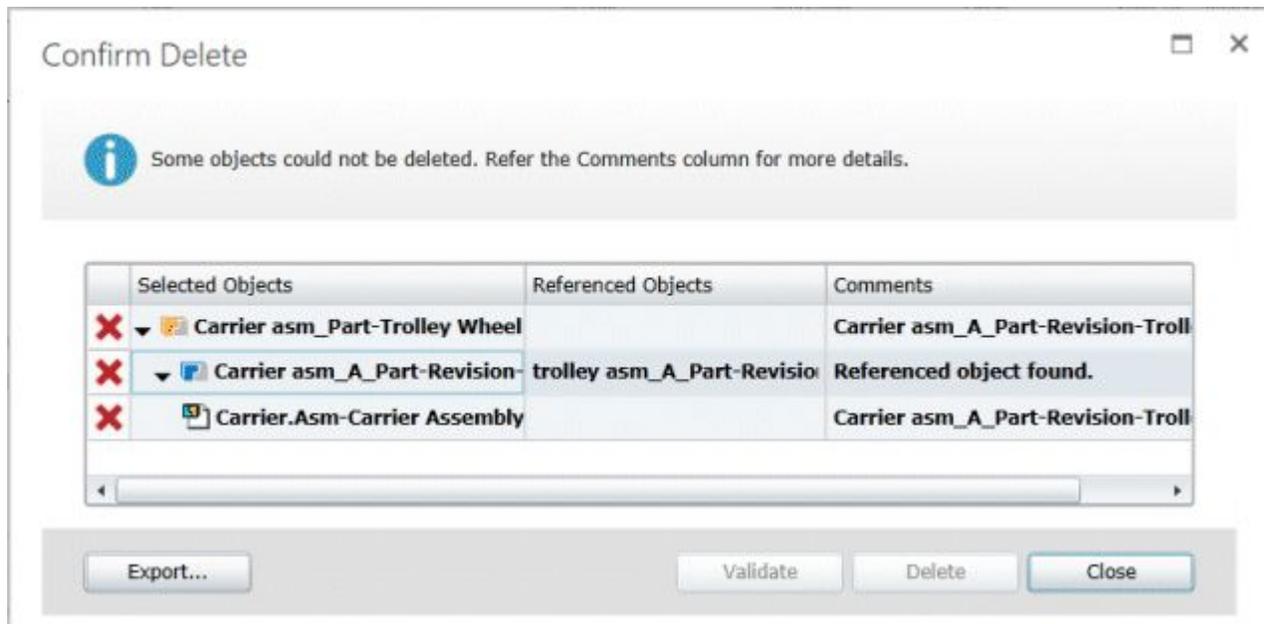
- Teamcenter client communication system settings (TCCS)
- Creation of Solid Edge Assembly BOM View Revisions
- Use of Variant Rules
- Structures that contain multiple revisions of an item

For more information, see the help topic, Manage page (Solid Edge Options dialog box).

Solid Edge SP

Faster delete performance

When you choose to delete data, Solid Edge SP now offers different levels of integrity checks, which give you the flexibility to delete objects faster and allows you to quickly return to your other tasks. In addition, several usability improvements to the **Delete** dialog box make the feedback from the data integrity checks easier to read and understand.



Retaining signoff information

Solid Edge SP now preserves your workflow signoff information in its support database instead of the workflow log list to capture this critical information more securely and reliably. The signoff information is available to all users and is read only. Utilities to migrate existing signoff information and update callouts are also provided.

Workflow ID	Workflow Title	Workflow Initiator	Object Name	Number	Revision	Task Name	Task Assigned To	Task Status	Task Completion Date	Current Status	Status Name
2c704483-633d	Prefer-OFF	SHAREPOINT\system	CSA-101_1_Part-Revision	CSA-101	1	Assign Status	NA	Completed	12/17/2015 2:48:37 PM	60	In Approval
2c704483-633d	Prefer-OFF	SHAREPOINT\system	CSA-101_1_Part-Revision	CSA-101	1	Review	Bhadekar, Kaustubh	Approved	12/17/2015 2:48:35 PM	50	In Review
2c704483-633d	Prefer-OFF	SHAREPOINT\system	CSA-101_1_Part-Revision	CSA-101	1	Review	Kulkarni, Deepak (Pune)	Approved	12/17/2015 2:48:21 PM	50	In Review

Pinned locations in Parts Library

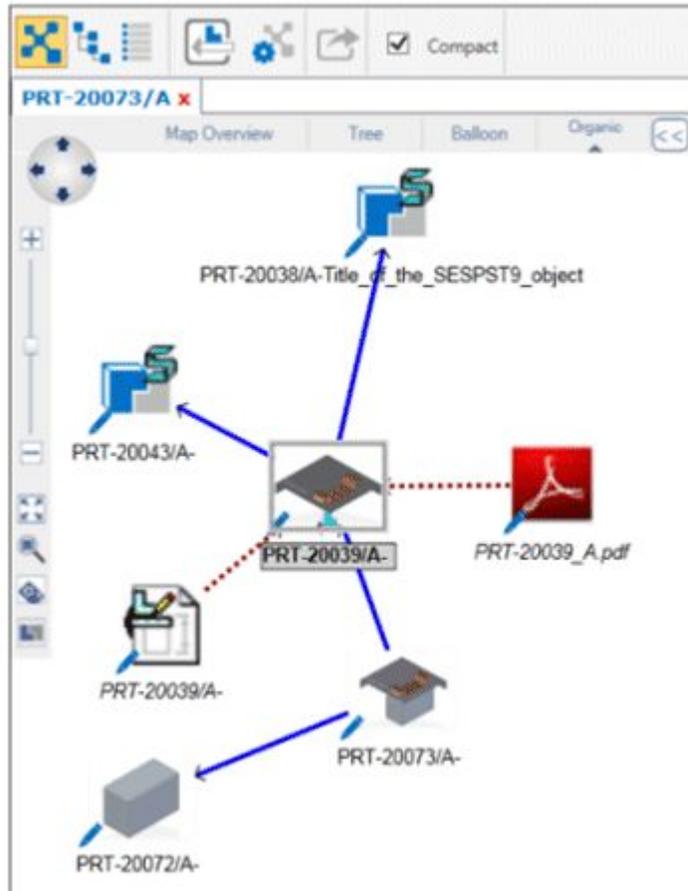
With a click of a button you can now create shortcuts to up to 10 key locations inside the **Parts Library**. Those programmable buttons (pinned locations) provide one-click access to the saved locations, instead of having to re-navigate the folder structure. This helps when creating new designs in Solid Edge SP. In addition, you can now access information about recently opened files, recent locations, and cached files.



New one-click operation to show all assembly drawings and pdfs

Solid Edge SP Web Client now has the ability to show all (missing) drafts and pdfs using the **Show Related Drafts** button  in **Relation Browser**. This replaces the laborious task of running individual where-used commands with a one-click operation, and it speeds the review and approval process.

Relation Browser



Bill of Materials	Address	Number	Revision	Part.Title	Conten...	Quantity	Filena...	Revisio...	Related Draft	Related PDF
[-] PRT-00179_B_Part...	http://...	PRT-00...	B	-	SE-Ass...	1	PRT-00...	Part-Re...		
[+] PRT-00171_B_P...	http://...	PRT-00...	B	PROJE...	SE-She...	1	PRT-00...	Part-Re...	Yes	Yes
[+] PRT-00198_B_P...	http://...	PRT-00...	B	PROJE...	SE-Part	1	PRT-00...	Part-Re...	Yes	Yes
[-] PRT-00194_B_P...	http://...	PRT-00...	B	PROJE...	SE-Part	4	PRT-00...	Part-Re...	No	No
[+] PRT-00193_B_P...	http://...	PRT-00...	B	PROJE...	SE-Part	2	PRT-00...	Part-Re...	Yes	Yes
[+] PRT-00196_B_P...	http://...	PRT-00...	B	PROJE...	SE-Part	2	PRT-00...	Part-Re...	Yes	Yes
[-] PRT-00186_B_P...	http://...	PRT-00...	B	PROJE...	SE-Ass...	2	PRT-00...	Part-Re...		
[+] PRT-00192_B_...	http://...	PRT-00...	B	PROJE...	SE-She...	1	PRT-00...	Part-Re...	Yes	Yes
[+] PRT-00174_B_...	http://...	PRT-00...	B	PROJE...	SE-She...	1	PRT-00...	Part-Re...	Yes	Yes
[+] PRT-00195_B_...	http://...	PRT-00...	B	PROJE...	SE-Part	1	PRT-00...	Part-Re...	Yes	Yes
[+] PRT-00182_B_...	http://...	PRT-00...	B	PROJE...	SE-Part	1	PRT-00...	Part-Re...	Yes	Yes
[-] PRT-00177_B_P...	http://...	PRT-00...	B	-	SE-Part	1	PRT-00...	Part-Re...	No	No
[+] PRT-00190_B_P...	http://...	PRT-00...	B	PROJE...	SE-She...	1	PRT-00...	Part-Re...	No	No

Improved overall Solid Edge SP usability

We improved the overall Solid Edge SP usability and efficiency, with the following additions:

- Create a new folder from the **Linked Objects** dialog box.
- Hide and display inactive carts from the **Content Browser**.

- The **Preview/Properties Card** now retains the last modified size.
- Send Solid Edge SP documents as email attachments to enhance document sharing.
- Use the **Favorites** folder in Solid Edge SP, Solid Edge SP Viewer, and Structure Editor.
- Display the results of your last search, when in the **Search** Web Part, by pressing the Enter key.
- Create new SESP-Documents from within Microsoft Office applications.
- Workflow improvements:
 - o Translation workflow handlers now remember the output location consistently through the newly added preference **Translated_File_Location**.
 - o Improved the display of workflows by making the **Title** box on the workflow initiation form a mandatory entry.
 - o If a workflow fails, now the workflow initiator is informed through an email notification.

Updated data preparation tools

The new Solid Edge Data Preparation Utility is delivered when you install Solid Edge and assists you in analyzing and preparing your unmanaged Solid Edge documents before importing them into a Teamcenter, Insight, or Solid Edge SP-managed environment. The new utility:

- Combines the three previously separate tools: Analyze, Link Fix-Up, and Modify, into one tool.
- Provides the functionality to generate reports for custom properties and reverse link information.
- Maintains command line functionality using the program **\\Program Files\Solid Edge ST9\Program\DataPrepUtility.exe** along with options.

For more information, see the help topic, Solid Edge Data Preparation Utility.

Insight

Insight software compatibility

Solid Edge	Operating System	SQL Server	SharePoint Foundation	SharePoint Server
ST9	Windows Server 2008 R2, 2012/2012 R2	2008 R2 or 2012	Search Server Express 2010 SP2*	2010 SP2*
	2012/2012 R2	2012 or 2014	2013 SP1	2013 SP1

This release of Insight is not supported with Microsoft Office SharePoint 2016.

SharePoint 2013 SP2 includes a known issue with Search webpart. Recommend SharePoint 2013 SP2 with hotfix.

*SharePoint 2010 is not supported on Microsoft SQL Server 2014. See Microsoft TechNet for SharePoint and SQL Server compatibility.

Windows Server 2008 and 2008 R2 life cycle ended prior to Solid Edge ST9 release. Testing with Windows Server 2008 R2 only.

SQL Server 2008 and 2008 R2 life cycle ends with Solid Edge ST9 release. Testing with SQL Server 2008 R2 only.

Microsoft Product Lifecycle (EoSM):

- 1/13/2015 Windows 2008 and 2008 R2
- 7/8/2015 SQL Server 2008 and 2008 R2
- 10/13/2015 SharePoint 2010

Insight ST9 is *not* supported with:

- Microsoft SharePoint 2016
- Windows SharePoint Services 3.0
- Microsoft Office SharePoint Server 2007
- Microsoft SQL Server Express
- Microsoft SQL 2005

For the latest information available, consult the individual **readme** files for Solid Edge and Insight Server.

Insight standalone installation discontinued

The option to download Insight as a standalone application is discontinued. You now install Solid Edge and the Solid Edge Insight Client, and then run Insight or Design Manager from your Start menu.

For additional information, see the *Insight Implementation Guide* accessible from Solid Edge installation media or from the Solid Edge help pane.

Sort indented lists

Insight supports sorting in an indented list. Using the shortcut command, **Sort**, you can select a column and sort the contents:

- In ascending order, where the lowest number or beginning of the alphabet appears first in the sorted list.
- In descending order, where the highest number or end of the alphabet appears first in the sorted list.

- In alphabetic and numeric order, so that data containing alphabetical letters mixed with numbers is sorted logically.

Example

1
1a
2
2a
2b

- Alphabetically, where the beginning of the alphabet appears first in the sorted list.

Note

When a column has repeating values for rows, you can select additional columns for sorting values. **Shift+double-click** subsequent column headers to add columns as additional sort columns.

For more information, see the help topic, Sort multiple columns.

Updated data preparation tools

The new Solid Edge Data Preparation Utility is delivered when you install Solid Edge and assists you in analyzing and preparing your unmanaged Solid Edge documents before importing them into a Teamcenter, Insight, or Solid Edge SP-managed environment. The new utility:

- Combines the three previously separate tools: Analyze, Link Fix-Up, and Modify, into one tool.
- Provides the functionality to generate reports for custom properties and reverse link information.
- Maintains command line functionality using the program **\Program Files\Solid Edge ST9\Program\DataPrepUtility.exe** along with options.

For more information, see the help topic, Solid Edge Data Preparation Utility.

View and Markup and Revision Manager

Where's Revision Manager?

Revision Manager has been renamed to Design Manager and has been overhauled to improve usability and give it a modern look and feel. We also separated View and Markup from Design Manager and then added some new functionality, such as fast search and fast where used.

For more information, see help in *Design Manager* and *View and Markup*.

Simulation enhancements

New in Solid Edge Simulation

The following enhancements were made in Solid Edge Simulation in ST9.

-  **Directional stress and strain result plots**
-  **Inertia relief for assembly studies in motion**
-  **Maximize quad elements in surface meshes**
-  **Suppress/Unsuppress commands for boundary conditions (loads and constraints)**
-  **New option processes radiation loads in small models**

Directional stress and strain result plots

The following stress and strain plots were added to the Simulation Results environment for linear static, modal, and buckling studies. The result plots are available based on the study type and the mesh type (tetrahedral, surface, or mixed body).

Stress plots

These directional stress plots show the results with respect to the global coordinate system.

- X Normal Stress
- Y Normal Stress
- Z Normal Stress
- XY Shear Stress
- YZ Shear Stress
- XZ Shear Stress
- Plate PrnStress Angle

For more information, see [Stress component plots](#) .

Strain plots

These directional strain plots show the results with respect to the global coordinate system.

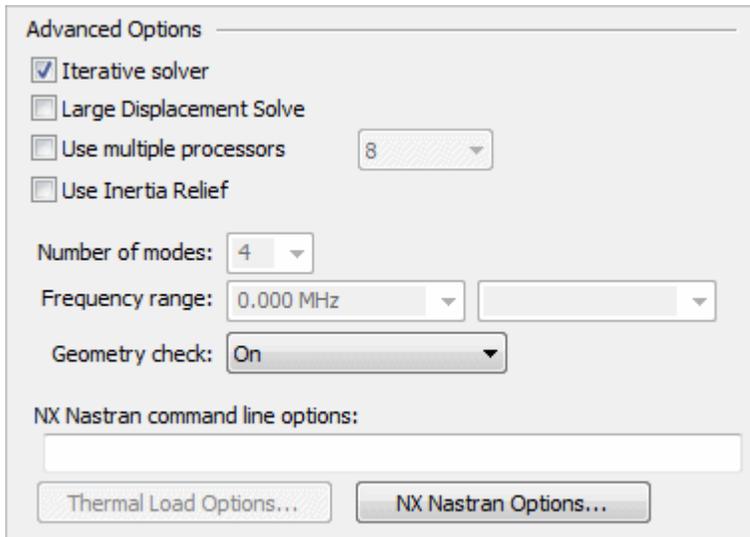
- X Normal Strain
- Y Normal Strain
- Z Normal Strain
- XY Shear Strain
- YZ Shear Strain
- XZ Shear Strain
- Plate PrnStrain Angle

For more information, see [Strain component plots](#) .

Inertia relief for assembly studies in motion

Now you can solve a linear static study of an assembly in motion without adding constraints to the model. Previously, to solve a study you had to apply at least one constraint.

The new option, **Use Inertia Relief**, is located on the **Create Study** dialog box, under **Advanced Options**.



For more information, see Advanced Options for studies.

Maximize quad elements in surface meshes

Quadrilateral surface mesh elements produce more accurate analysis solutions than triangular mesh elements. An option in Femap 11.22 is now available in Solid Edge Simulation to specify that you want to create as many quadrilateral elements as possible in a surface mesh, and minimize the number of triangular elements.

The new option, **Maximize Quad Elements**, is located on the **Surface Mesh** tab in the Mesh Options dialog box. It is selected by default.

Suppress/Unsuppress commands for boundary conditions

Now you can evaluate the effect of individual loads and constraints on study geometry by suppressing them. When you solve the study, the boundary conditions that are suppressed are not processed.

The **Suppress** command and the **Unsuppress** command are available on the shortcut menu for all of the following types of boundary conditions:

- Structural loads
- Body loads
- Thermal loads
- Constraints

For more information, see Suppress individual loads to compare results.

New option processes radiation loads in small models

An NX Nastran option is now available to solve steady state heat transfer studies with radiation enclosure loads on models with relatively small geometry and small mesh elements. The new option, **ZTOL**, is located on the Radiation Load Options dialog box.

Translator and converter enhancements

Translator and converter enhancements

These enhancements were made to translators and converters in Solid Edge ST9.

-  Aspect ratio now available for block labels
-  Custom line types supported in AutoCAD translation
-  Exporting JT using adaptor
-  Exporting to IFC (.ifc) enhanced
-  Import and export of simplified bodies enhanced
-  Translation of dimensions supported in import of AutoCAD files
-  Assembly overrides during export enhanced
-  Import and export of 3D data points now supported
-  Text automatically rotated in PDF
-  Tolerance text height override
-  Writing formula on export during JT translation
-  Parent properties written to embedded bodies during JT translation
-  Solid Works data migration enhanced

Aspect ratio now available for block labels

A new **Aspect ratio** option on the **Block Label Properties** dialog box specifies the aspect ratio for the block label.

Custom line types supported in AutoCAD translation

Solid Edge custom lines are now supported in AutoCAD translation.

- When exporting Solid Edge draft files to AutoCAD, you can use the **Solid Edge to AutoCAD Translation Wizard (Line Type Mapping)** to assign a Solid Edge custom line type to a specific AutoCAD line type. Previously, custom line types were exported to AutoCAD as continuous lines. For more information see, Solid Edge to AutoCAD Translation Wizard (Line Type Mapping).

- When importing AutoCAD files to Solid Edge, you can use **AutoCAD to Solid Edge Translation Wizard - Step 4 of 8** to assign a AutoCAD custom line type to a specific Solid Edge line type.

For more information, see AutoCAD to Solid Edge Translation Wizard - Step 4 of 8.

Export to AutoCAD now supports property text in blocks

Solid Edge now supports property text in blocks when exporting to AutoCAD.

Many customers use the property text `%{Sheet name}` or `%{Sheet number}` within blocks to place the blocks on the background sheet as part of the title block area. This provides an automatic numbering scheme as they add sheets to the draft document.

Previously, when the draft document was exported to AutoCAD as paper space, the property text `%{Sheet name}` or `%{Sheet number}` within blocks was not supported, and the sheet name and sheet number were the same for each sheet.

Beginning in ST9 property text `%{Sheet name}` or `%{Sheet number}` used in a callout within a block for each sheet, or in a block label for each sheet, creates a unique occurrence of the block for each sheet that is exported.

Exporting to Industry Foundation Class (.ifc) enhanced

- A new **IFC property set** option on the **Export Options for IFC (.ifc)** dialog box adds the IFC property sets found in an Excel spreadsheet to the IFC file during export.
- Two new parameters were added to the **seifc.ini** file:
 - o Export IFC Property Set= Specifies if the IFC property sets found in the Excel file are added to the IFC file created during export.
 - o IFC Property set path= Defines the full path and name of the Excel file that contains the property set to be included in the IFC file created during export.

For more information, see the help topic Saving to IFC.

Exporting JT using adaptor

A new translator for exporting JT files will replace the existing JT translator in ST 10. In ST9, you can use the **Use Legacy Translator** and **Use Beta Translator** options on the **Solid Edge to JT Translation Options** dialog box to specify the translator you want to use.

Note

The beta translator supports PMI except for section views, and it does not support textures.

Import and export of simplified bodies enhanced

New **Part simplification** and **Assembly simplification** options on the **Solid Edge to JT Translation Options** dialog box enhance the export of Solid Edge files to JT.

For **Part simplification**, you can specify:

- **Use Current Configuration** — Uses the current configuration to create the JT document.

- **All Simplified** — Checks each part and uses the simplified body, if available, to create the JT document.
- **All Designed** — Checks each part and uses the design body, if available, to create the JT file.

For **Assembly simplification**, along with the same values available with **Part simplification**, you can specify:

- **Top Level Simplification** — Checks the top level of the assembly and uses the simplified body, if available, to create the JT document.

Note

These options are also available on the **View and Markup tab, Options** dialog box, if you open an assembly while in View and Markup.

To support this new functionality, changes were made to parameters in the files, **SePvTrn.ini**, **SEECtoJT.ini**, and **InsightXTtoJt.ini** files.

- The ApplySimplifiedTopAsmOverride, ApplySimplifiedSubAsmOverride, and ApplySimplifiedPartOverride parameters were removed from the .ini files.
- Two new parameters were added to the .ini files:
 - Export Simplified parts — Specifies when to use simplified parts when exporting an assembly. Possible values are **All Designed**, **All Simplified**, and **Use Current Configuration**. The default value is **Use Current Configuration**.
 - Export Assembly Simplification — Specifies when to use simplified sub-assemblies. Possible values are **All Designed**, **All Simplified**, **Top Level Configuration**, and **Use Current Configuration**. The default value is **Use Current Configuration**.

Translation of dimensions supported in import of AutoCAD files

The translation of linear, angular, diametric, and radial dimensions is now supported when importing AutoCAD dxf and dwg files to part or assembly sketch. The translation of these dimensions into assembly sketch are identical to those dimensions translated into draft.

When dimensions are imported to sketch they are not attached to the line and arc elements. The lines and arcs of a translated dimension are construction elements that are placed on the **_Translation** layer. This layer is Off by default. You must reattach or construct new dimensions to modify the geometry. In other words these dimensions are simply reference dimensions.

Assembly overrides during export enhanced

Enhancements have been made to assembly overrides during export to STEP and IFC.

- Assembly features are checked for uniqueness during export. An additional subassembly is not created when you add an assembly feature to an identical subassembly.
- Show and hide assembly overrides are supported when exporting the assembly in its current display configuration.

- Assembly color overrides are now applied to all instances of the same occurrence within the assembly. If multiple occurrences of the same part have different assembly colors, only one color is applied to all occurrences.
- New **Part simplification** and **Assembly simplification** options on the **Export Options for STEP (.stp)** dialog box and the **Export Options for IFC (.ifc)** dialog box enhance the export of Solid Edge files to STEP or IFC.

For **Part simplification**, you can specify:

- o **Use Current Configuration** — Uses the current configuration to create the STEP or IFC document.
- o **All Simplified** — Checks each part and uses the simplified body, if available, to create the STEP or IFC document.
- o **All Designed** — Checks each part and uses the design body, if available, to create the STEP or IFC file.

For **Assembly simplification**, along with the same values available with **Part simplification**, you can specify:

- o **Top Level Simplification** — Checks the top level of the assembly and uses the simplified body, if available, to create the STEP or IFC document.
- o To support this new functionality, two new parameters were added to the **STEP.ini** and **SEIFC.ini** files:
 - **Export Simplified parts** — Specifies when to use simplified parts when exporting an assembly. Possible values are **All Designed**, **All Simplified**, and **Use Current Configuration**. The default value is **All Designed**.
 - **Export Assembly Simplification** — Specifies when to use simplified sub-assemblies. Possible values are **All Designed**, **All Simplified**, **Top Level Configuration**, and **Use Current Configuration**. The default value is **All Designed**.

Export mechanical symbols using AutoCAD fonts

A new Export Mechanical Symbols using AutoCAD font parameter in the Seacad.ini file specifies when to export Solid Edge mechanical symbols using an AutoCAD font rather than the Solid Edge symbol font. The default value is 0. Setting the value to 1, specifies to use the GDT.tff font where possible or draw the mechanical symbol for only callouts and balloons.

Note

This parameter is under construction for ST9, but you may use it if you find it helpful.

For more information, see Saving to AutoCAD.

Import and export of 3D points now supported

Solid Edge now supports import and export of 3D points when working with STEP and IGES documents.

When importing STEP documents to Solid Edge, use the **Import Curves and Points** option on the **Import Options for STEP (.stp)** dialog box to import 3D points. When importing IGES documents to Solid Edge, use the **Wire frame and points** option or the **Surfaces, solids, wire frame, and points** option on the **IGES Import Wizard - Geometry Types** dialog box to import 3D points.

The imported 3D points become points in a single 3D sketch in Solid Edge.

When exporting Solid Edge documents that contain 3D points to STEP or IGES, use the **Export wires and points** option on the **Export Options for STEP (.stp)** dialog box or the **IGES Export Wizard - Types** dialog box.

Text automatically rotated in PDF

A new **Auto rotate sheets for text readability** option on the the **PDF Export Options** dialog box automatically rotates the drawing sheet so the text orientations are horizontally aligned.

Tolerance text height override

A new parameter has been added to the SEACAD.ini file:

- Import Tolerance Text height — Specifies a static tolerance text height override for text being imported from DXF documents. The default value is 0 and any real number 0 or above is valid.

Writing formula on export during JT translation

A new *Export using DocnameFormula* parameter specifies whether to use the file name or the Document Name Formula when exporting to JT. Setting **Export using DocnameFormula=0** specifies that you want to use the file name. Setting **Export using DocnameFormula=1** specifies that you want to use the Document Name Formula.

The parameter is available in the following files:

- SePvTrn.ini
- SE2VM.ini

Note

The parameter is automatically enabled when you use the **Open in View and Markup** command in Solid Edge. You need to enable the parameter through the ini file if you want to use the functionality when exporting to JT.

Parent properties written to embedded bodies during JT translation

In a Teamcenter-managed environment, the property values for the parent object in an assembly is now written to the embedded bodies within the assembly when translated to JT.

SolidWorks data migration enhanced

The SolidWorks data migration command has been enhanced to include the migration of associative drawings views. Updating the migrated Solid Edge part and assembly files now updates the Solid Edge draft file. During the migration process a link is established between the resulting Solid Edge parts and assembly files and the migrated Solid Edge drawing file. The Solid Edge drawing views are created based on the SolidWorks drawing view location, type, and scale.

The following applies when migrating SolidWorks files:

- All drawing view types created in SolidWorks are supported.
- Cutting plane lines, auxiliary view plane lines, and detail envelopes are supported.
- Dimensions migrated from SolidWorks are bound to Solid Edge drawing view geometry.
- Center marks and center lines are associative to the Solid Edge drawing views.
- Hatch and solid fills are migrated maintaining their associativity to the Solid Edge drawing view.
- View shading is maintained as part of this drawing view creation process.
- Multi-sheet drawings are supported through this migration process.

Note

Annotations, such as balloons and callouts, do not migrate as Solid Edge annotations, but migrate as blocks.

Operating system requirements and information

Solid Edge ST9 has been certified to run on the following:

- Windows 7 Enterprise, Ultimate or Professional (64-bit only) with Service Pack 1
- Windows 8 or 8.1 Enterprise or Professional (64-bit only)
- Windows 10 Enterprise or Professional version 1511 or later (64-bit)
- Internet Explorer 11

Internet Explorer is not required to be the default browser.

Note

- Solid Edge ST9 is 64-bit only. Solid Edge ST6 was the last release of 32-bit Solid Edge.
- Windows 8 Home, Windows 10 Home, and Windows RT are not supported.
- Solid Edge stops certifying new releases against an operating system shortly after Microsoft drops mainstream support for it. Solid Edge ST9 will not install on Windows Vista or Windows XP. Microsoft dropped mainstream support for Windows 7 in January 2015. Solid Edge ST9 continues to install on Windows 7, however, if there is a Windows 7 specific problem, we cannot guarantee that a fix can be provided if support is required from Microsoft.
- Solid Edge ST10 will be the last release to support Windows 7.
- Solid Edge does not support the new Windows 10 browser, Microsoft Edge.

Hardware system requirements

Insight hardware and software requirements follow the recommendations for Solid Edge. Be sure to consult the Solid Edge **Readme.htm** for additional information.

Recommended system configuration

- 64-bit (x64) processor
- Windows 7 operating systems
- At least 8 GB RAM
- True Color (32-bit) or 16 million colors (24-bit)
- Screen Resolution: 1280 x 1024 (or higher, following the manufacturer's recommendations)

Minimum system configuration

- 64-bit (x64) processor
- Any of the supported operating systems
- At least 2GB RAM
- 65K colors
- Screen Resolution: 1280 x 1024 (or higher, following the manufacturer's recommendations)

Display system requirements and information

Solid Edge runs on graphics drivers that support Windows 7, Windows 8.1, or Windows 10. Contact your graphics driver manufacturer to determine whether their graphics adapter/driver support these operating systems.

For optimal performance, use a professional graphics card designed for CAD applications. For information about cards used in testing Solid Edge and results, see the Hardware and Software Certifications web site at

http://www.plm.automation.siemens.com/en_us/support/gtac/certifications.shtml.

Note

At least a 256MB graphic card is recommended when working with large assemblies or complex parts.

Running with extremely high screen resolution and color depth increases the memory requirements on the system and may result in apparent performance degradation. If experienced, reconfigure the display system to the recommended resolution and color depth for improved performance.

When running Solid Edge, if you experience an abnormally high abort rate, parts disappearing, or other graphic anomalies you may not be using the appropriate graphics driver. For details, see **Certification page of the Support web site**: <http://support.industrysoftware.automation.siemens.com/gtac.shtml>.

Also, setting Display Fonts to Large Fonts or Extra Large Fonts (larger than 96 DPI) may cause some Solid Edge user interface items to not display as intended. To resolve these issues, use Normal Fonts (96 DPI).

A wide-screen ribbon layout is used by Solid Edge. This ribbon layout is optimal for horizontal screen resolutions set to 1920 or above. Solid Edge automatically detects your resolution and sets the ribbon layout to wide-screen starting at horizontal resolutions 1600 and above. Some groups collapse on the right side of the ribbon with resolutions between 1600 and 1920.

System Resource Requirements and Information

Earlier versions of Solid Edge were enhanced to access the extended address space that is available on Windows. All running processes, including the Operating System process, share 4 GB of addressable memory available, regardless of the amount of physical RAM. Normally, the operating system reserves 2 GB of space and leaves 2 GB for applications. Running 32-bit operating systems with the /3GB switch added, reserves only 1 GB for the operating system, and leaves 3 GB for applications. This allows you to work with larger datasets without running out of addressable space.

On Windows 7 with administrator privileges, you can run the following command from the command line to enable /3GB switch:

```
BCDEDIT /Set IncreaseUserVa 3072
```

This command tells you what options are part of the OSLOADER family:

```
bcdedit /? types osloader
```

To reset the value, use:

```
bcdedit /deletevalue IncreaseUserVa
```

The 64-bit Solid Edge supports up to 128 GB of physical RAM and 16 terabytes of virtual memory, enabling applications to work with larger data sets. The 64-bit version of Solid Edge requires 64-bit Windows operating systems loaded on Intel EM64T or AMD64 processors.

Page file size should be the maximum size possible. In general, the page file size should be at least twice the amount of memory in the machine, plus the size of files you will use.

Processor Support

Component software delivered with Solid Edge, such as Parasolid and D-Cubed, started phasing out processors not supporting Intel's SSE2 (Streaming SIMD Extensions 2) instruction set. In 2009, these components will only support processors with SSE2.

SSE2 is one of the Intel SIMD (Single Instruction, Multiple Data) processor supplementary instruction sets first introduced by Intel with the initial version of the Pentium 4 in 2001. AMD added support for SSE2 with the introduction of their Opteron and Athlon 64 ranges of AMD64 64-bit CPUs in 2003.

Here is a list of common processors that do not support the SSE2 instruction set:

- AMD CPUs prior to Athlon 64
- Intel CPUs prior to Pentium 4

Solid Edge only supports processors that include the SSE2 instruction set. The Solid Edge installation checks for the presence of such a processor. If this processor does not exist, installation stops and displays a message explaining this dependency.

Temporary File Space

Solid Edge uses temporary file space to save files and to store memory mapped display files. Using temporary file space when saving files helps significantly reduce the size of the resulting file on the destination file system.

You should ensure that you have twice as much free temporary file space as the size of the largest file being saved. The file size includes the size of the assembly, the size of the subassemblies that are being used, and the size of the part files.

Running Solid Edge creates temporary memory mapped files that are used to support the display system. When you open assembly and part files, memory mapped files are created in the *temp* directory on the hard drive only while the files are displayed. The memory mapped files are deleted after exiting the Solid Edge file. If the system runs short on temporary space on the primary drive, you can set an environment variable called `JRENDER_TEMP` and define another folder on a drive with sufficient space and then Solid Edge creates the memory mapped files in the new location.

Running Solid Edge creates a file named `DCCACHE.tmp` in the system *temp* folder. This file is a cache of the file icons displayed on the dialogs for browsing, opening, and saving files.

If Solid Edge files are not properly associated with the product, try deleting this file and restarting the product.

It is good practice to periodically delete files in temporary folders so your system is not using unnecessary disk space.

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