SIEMENS

What's New in Solid Edge 2022

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Contents

1. Highlights of Solid Edge 2022

Solid Edge Help includes videos and activities

To help you learn Solid Edge, the help collection includes a Solid Edge video gallery and an Activities collection. You can access these resources from the Solid Edge overview landing page, under Learn Solid Edge 

Sketching usability enhancements

Enhancements were made in 2D and 3D sketching and to the interface to make it easier for new users to start a sketch, and to identify the sketching tools that are available.

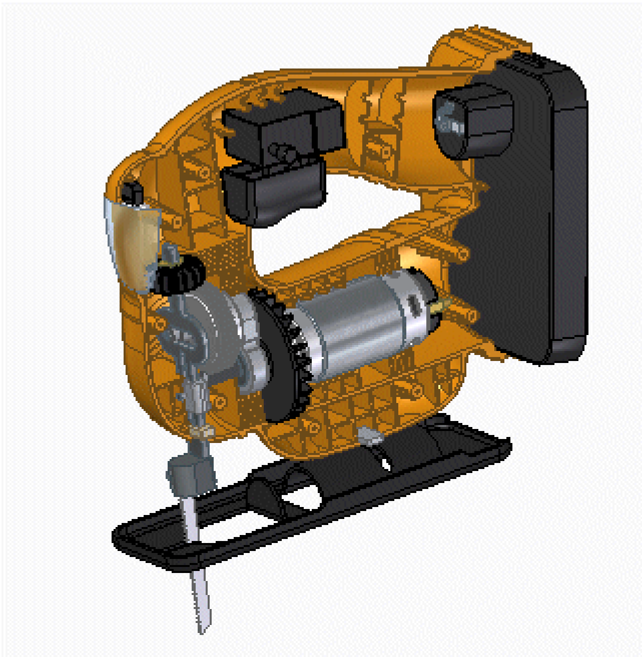
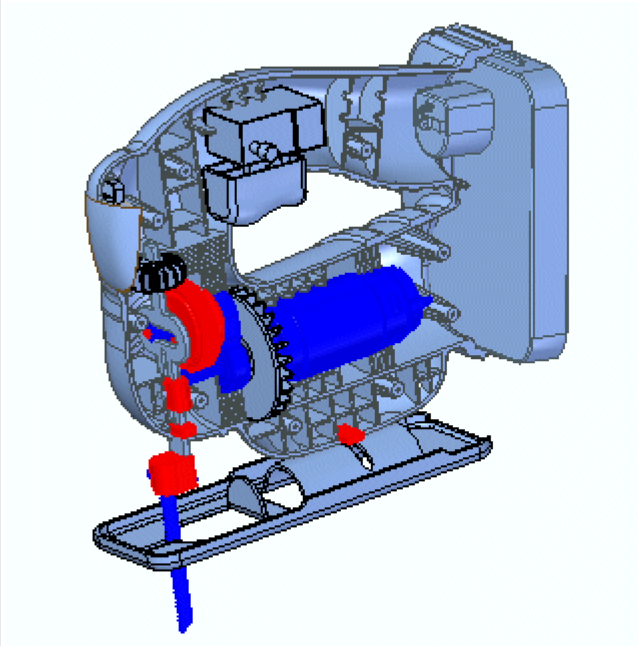
Here are a few examples:

- Improved 2D sketching workflow in assembly and in ordered mode
- Automatic base reference plane display
- Ribbon update for Draw group commands
- Vertical command bar visibility
- Auto-complete for finding commands
- Context toolbar for 2D sketching
- Dimension editing enhancements for 2D and 3D

Dynamic Visualization

Provides a single visual interactive environment that delivers document property analytics for Solid Edge assemblies. Using Dynamic Visualization, you can interactively view, filter, and generate rules for querying property information for parts in the active assembly. Temporary color overrides are associated with each query, and results are displayed real-time in the model.

For more information, see Using Dynamic Visualization to view property information.

Original assembly	Assembly with property filtering rules applied
	 Red—Purchased parts Blue—Stainless steel parts Light Blue—Unfiltered property information

Frame environment enhancements

The Frame environment provides new modeling command options to assist you when splitting, extending, and trimming frames, as well as new capabilities and commands to connect the frames in a structural simulation study.

- The new **Split intersecting frames** option on the **Frame Options** dialog box automatically splits the frames where the paths intersect. It also splits the sketch used to create the frames.
  For more information, see Splitting intersecting frames.

- With the new Trim/Extend command you can now trim frames to multiple objects, such as faces, bodies, or planes.
  For more information, see Trimming frames.

- When you create a new Solid Edge Simulation study for the frame model, you can now automatically add the rigid link connectors between beam curves, as well as edit existing connectors.
  For more information, see **Connector enhancements for frame models**.

Subdivision Modeling offers four new commands

Expanding on the capabilities Subdivision Modeling provides you, Solid Edge 2022 offers four new commands to assist you with creating free-form shapes:

- **Align to Curve** —Fit the vertices of body cage faces to one or more existing curves or to curves you interactively sketch.
- **Bridge** —Create a loft-like feature that connects edges or faces selected on a single cage or two separate cages.
- **Offset** —Move or lift multiple faces by a specified offset value.
- **Split with Offset** —Add detail to a face by offsetting the new faces inward by an amount you define.

To learn more about Subdivision Modeling, see Introduction to Subdivision Modeling.

2. Administering Solid Edge

Introducing Floating License Manager

The **Solid Edge Floating License Manager** enables you to return dynamically checked out add-on license features, such as XpresRoute and Wire Harness when they are not in use, freeing them for use by others, all without having to restart Solid Edge. Releasing floating add-on licenses back to the pool of available licenses makes the most efficient use of the total number of licenses needed at any given time. For more information, see License management for add-on licenses.

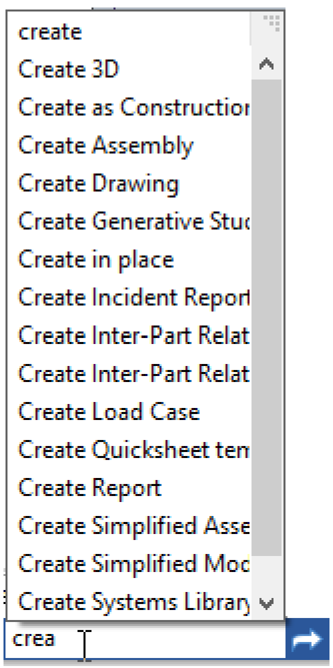
For more information about Solid Edge licenses, see the Solid Edge Installation and Licensing guide.

3. User interaction changes

Auto-complete for finding commands and components

Command Finder

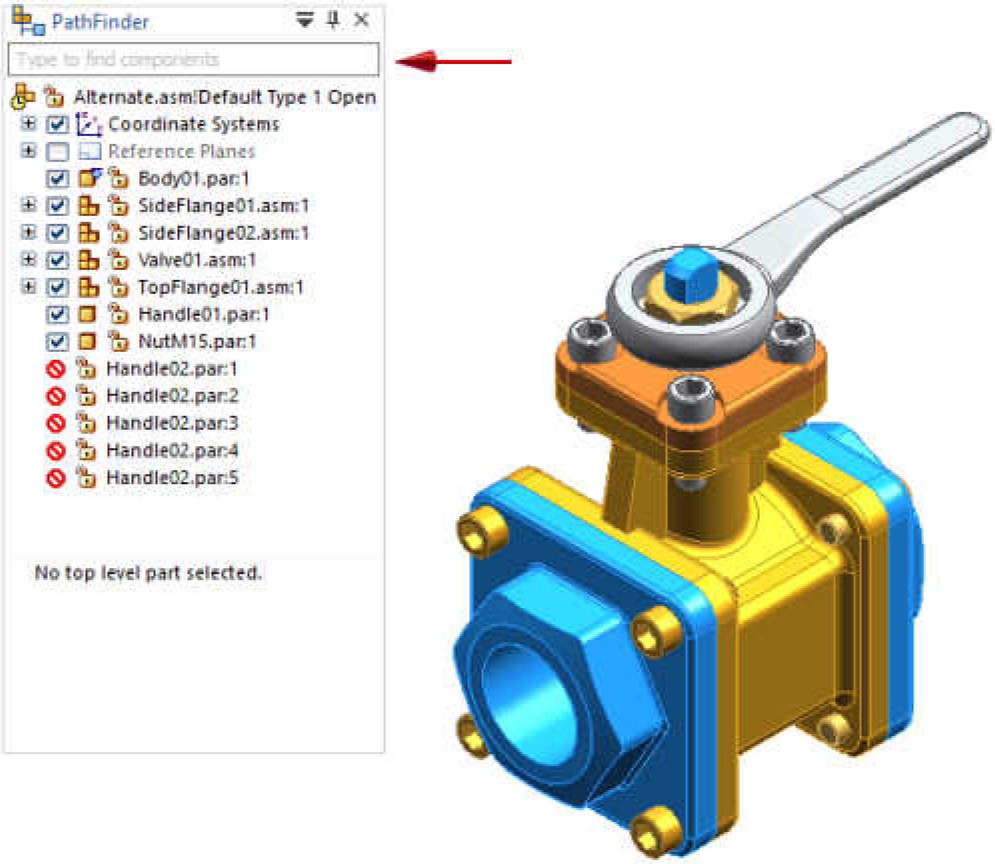
When you use **Command Finder** to find a command in the Solid Edge interface, auto-complete suggests matches for the search text as you type it, and displays the commands in a list. Select a command from the list, and click to display the **Command Finder** dialog box.



For more information, see Find a command with Command Finder.

Component Finder

A new search capability, **Component Finder**, was added to the top of **PathFinder** in an assembly. Begin typing the name of a subassembly or part or sheet metal component that you want to locate in the assembly feature tree. Automatic completion of the search text finds all available matches in the document. When you click the instance you want to find, it is located and highlighted in **PathFinder** and in the graphics window.



For more information, see Find a part in an assembly.

Floating graphic windows for multiple monitors

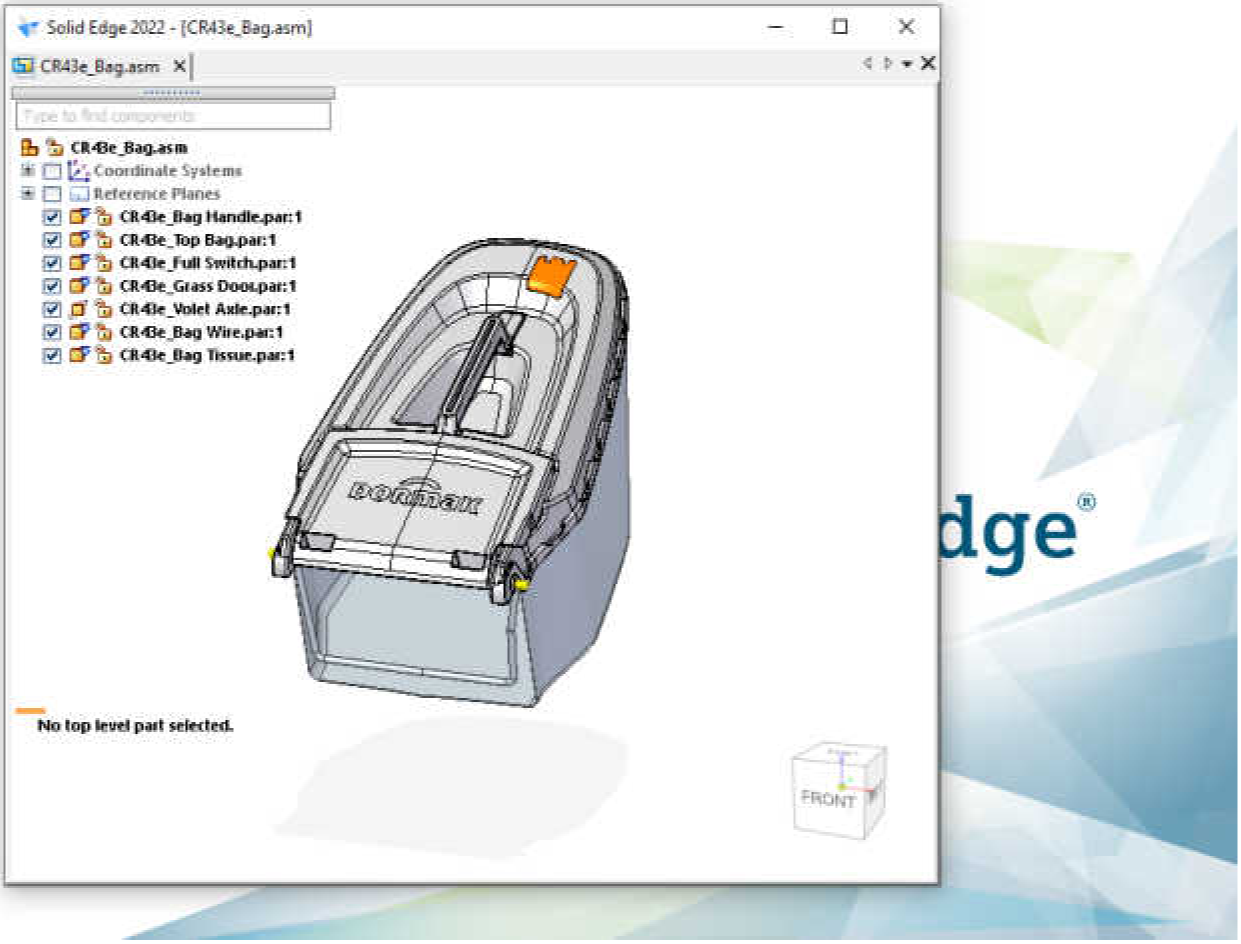
Solid Edge graphic windows can now be detached from the application frame. When detached, you can position the window where you want it on the same monitor, or you can drag it to a different monitor. **PathFinder** is displayed in the model window that is currently active.

This feature is convenient when you want to work with your model in multiple Solid Edge windows set to different model views, for example, top, right, left, and isometric.

The following new commands are available on the context menu of a document tab in all environments. Some of these commands are also available in the **View** tab→**Windows** group on the ribbon:

- **New Floating Window**—Creates a new floating window containing the active document.
- **Float Window**—Detaches a window from the main Solid Edge application frame and floats it.
- **Dock to Main**—Docks the active floating window to the main Solid Edge application frame.
- **Dock All to Main**—Docks all floating windows back to the main application window.

For more information, see Floating Solid Edge windows.



Give us your feedback

We want to hear from you. While you are in Solid Edge, you can let us know what is working well, if you experience problems with Solid Edge, or if you have ideas for new functionality.

From the top-right of the Solid Edge application window, click the smile or the frown . Using 1000 characters or less, tell us what you like or what can be improved.

For more information, see Give feedback on Solid Edge and Solid Edge Product Improvement Program.

New shortcut commands in Windows Explorer

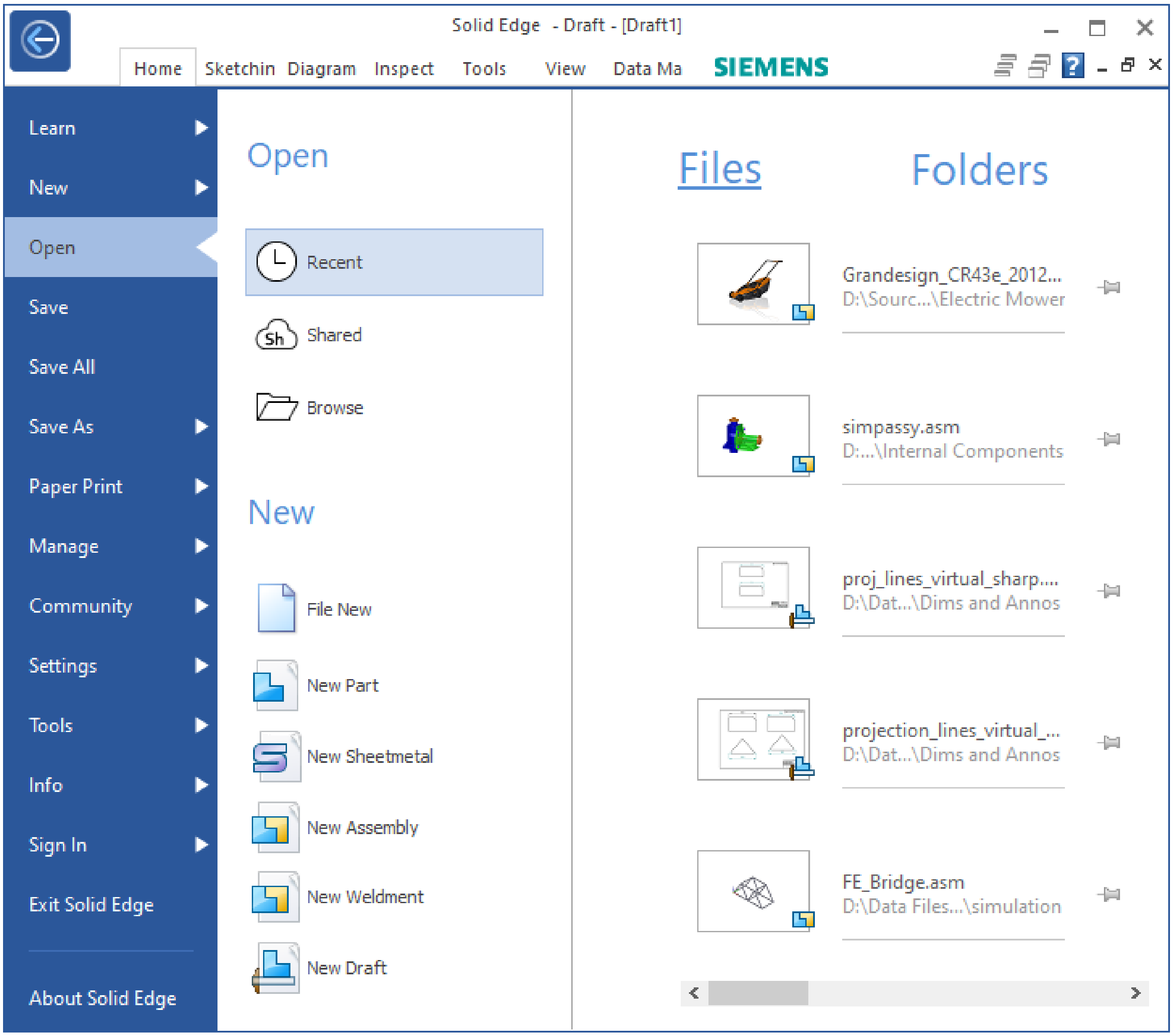
Windows File Explorer makes it easier to share and organize data by providing shortcut (right-click) access to the commands **Pack and Go** and **Broken Links**. Now you can perform these commands on the selected document without having to open the document first in Solid Edge or Design Manager.

For more information, see the help topics, Sharing documents or Search for broken links.

Open page versatility

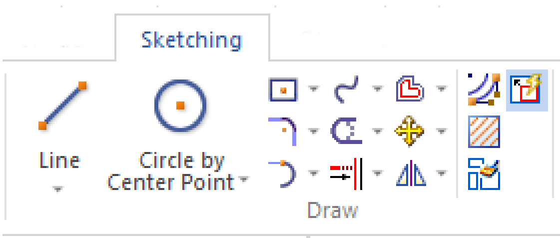
In addition to accessing recently opened files and folders, you can now create new documents starting from the Solid Edge **Application** menu→**Open** page. This enhancement improves productivity by eliminating switching between the **Open** page and the **New** page.

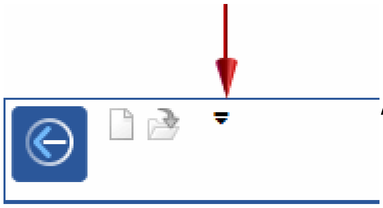
To learn how to set this page as your default, see Changing the start page in Solid Edge.



Ribbon update for Draw group commands

The default arrangement of the commands on the **Sketching** tab→**Draw** group was updated based on usability feedback from customers. The most frequently used drawing commands—**Line** and **Circle**—have large-sized icons to aid in locating them, and the less frequently used commands are available from the adjacent list buttons. To prevent the ribbon from being too long, other drawing commands were moved to adjacent list buttons.



You can return to the previous ribbon arrangement by selecting the **Customize the Ribbon** command , and then on the **Ribbon** page (Customize dialog box), in the right pane, selecting one of the following options from the shortcut menu of the commands you want to change:

- **Small Button**
- **Large Button**
- **Text**
- **No Text**
- **Button Options...**

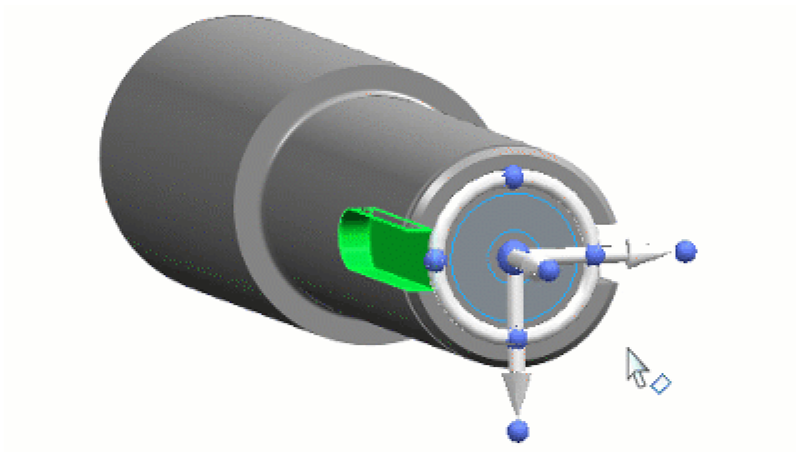
For more information, see *Control the size of ribbon bar buttons and whether or not they display text*, in the help topic, Customize the ribbon.

Steering wheel rotation enhancements

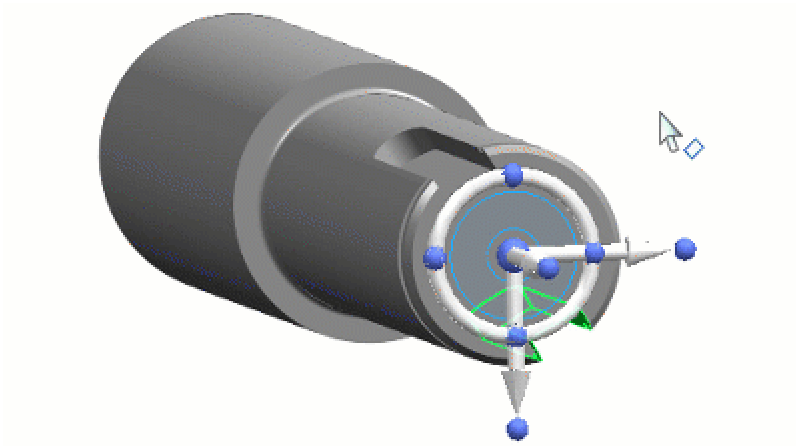
More rotational controls are available on the steering wheel used in synchronous part, sheet metal, Subdivision Modeling, and assembly.

Now when you click the 3D steering wheel torus to begin rotation:

- The steering wheel snaps to 90-degree positions.



- You can press+hold the Shift key to rotate in 15-degree increments.

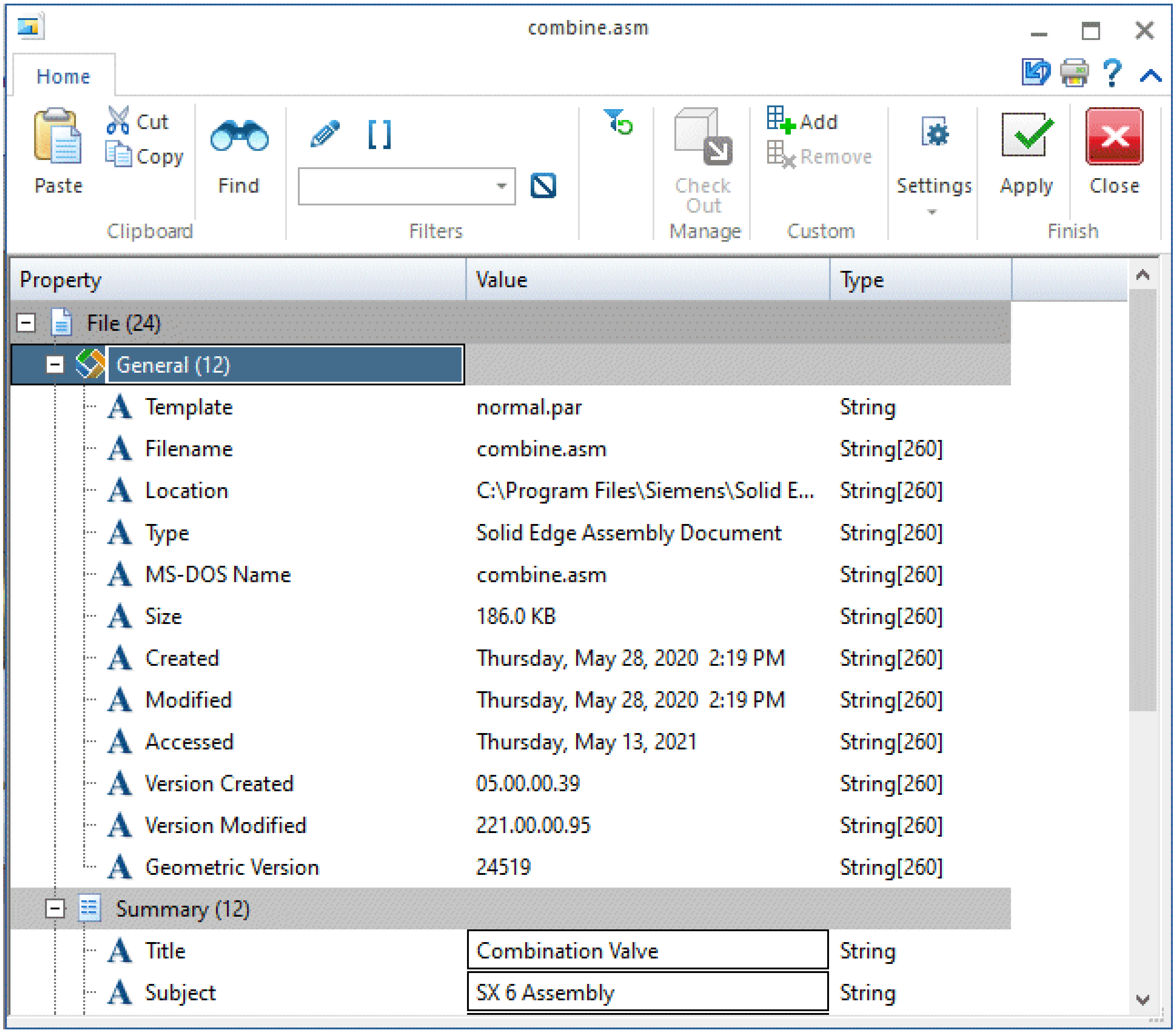


- As before, you can snap to keypoints. Keypoints have precedence over increment snapping.

For more information, see Rotate part geometry using the steering wheel.

Updated File Properties dialog box

The Solid Edge **File Properties** dialog box is greatly enhanced with new functionality. You can filter the property values of the selected document using the commands in the ribbon, change property values in the table, and add or remove custom properties.



The new interactive dialog box:

- Displays Solid Edge file properties in a vertical format.
- Organizes properties in groups.
- Enables you to print or copy the contents to your clipboard.
- Uses shortcuts commands that control the display of gridlines, identifies read-only information, and outlines cells containing writeable information.

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

Vertical command bar visibility

Improvements were made to the interaction of the vertical command bar and **PathFinder** with the docking panes. This enhances their visibility in the Solid Edge interface and provides faster access to these tools.

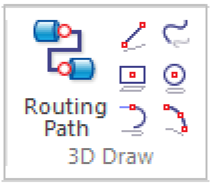
> Note:
>
> You can specify that you want to use the vertical command bar format by selecting the option, **Use vertical docking window form**, in **Application** menu→**Settings**→**Options**→**Helpers**, in the **Command User Interface** section.

For more information, see Using the command bar.

4. Design and manufacturing enhancements

3D drawing commands in Frame and XpresRoute

In the Frame environment and in the XpresRoute environment for piping and tubing, 3D drawing commands are now available on the **Home** tab→**3D Draw** group. For information about a **3D Draw** command, hover over the command and press F1.



The commands in the **3D Draw** group replace the **Segment** group on the ribbon. However, the individual commands that were in the **Segment** group are still available for you to add back to the ribbon using the **Customize** dialog box, and in the **Choose commands from** list, selecting **Commands Not in the Ribbon**.

Assembly feature and model override enhancements

The assembly feature and model override functionality has been restructured to align with the part multi-body modeling framework and commands. This ensures that most of the functionality available in part features are also available in assembly features. Creating a new assembly feature tree in Solid Edge 2022 contains the new functionality. Legacy assembly features and assembly body elements are not enhanced, but are still supported.

Enhancements to assembly feature and model override enhancements include:

- Assembly addition commands, such as **Extruded Protrusion**, **Revolved Protrusion**, and **Fillet Weld**, use the ordered part implementation.

- Assembly removal commands, such as **Revolve Cut**, **Hole**, and **Round**, use the ordered part implementation.

- The assembly commands, **Pattern Assembly Feature** (rectangular/circular), **Pattern Along Curve Assembly Feature**, and **Mirror Assembly Feature**, use the ordered part implementation.

- The **Assembly Feature Options** dialog box is now available in the context of the assembly for the following commands:

  - **Cut**

  - **Revolved Cutout**
  - **Hole**
  - **Round**
  - **Chamfer**
- New options on the **Assembly Feature Options** dialog box create synchronous cuts and treatments, when possible.
- All command options available in ordered part features are available in assembly features.
- Auto-selection of occurrence bodies in the assembly is enabled based on the interference of the tool with occurrence's design body for the following commands:
  - **Cut**
  - **Revolved Cutout**
  - **Hole**
- Assembly features can locate, select, or modify external references and internal references.
- You can modify multiple design bodies in the part with an assembly feature such as a cut. Previously, the cut would only modify the active assembly body defined in a part file.
- You can now use the **Text Profile** command to create text profiles in the parent local profile of assembly features.
- Variable table enhancements for assembly features include:
  - Assembly features are listed under the **Assembly Features** group in the variable table.
  - The naming for the assembly feature variables will be the same as the part feature naming.
  - Assembly features support suppression variables.
- Family of assembly enhancements for assembly features include:
  - Assembly features are now supported in family of assemblies.
  - Excluding and suppressing assembly features are supported in family of assemblies.
  - You can override assembly feature variables in the **Alternate Assemblies Table** dialog box.

- **PathFinder** enhancements for assembly features include:
  - Currently supported **PathFinder** functionality is supported for new assembly features.
  - The **Dynamic Edit** command, from both the shortcut menu and edit bar, is available for assembly features.
  - The display of overridden assembly instances and occurrences indicates that they are overridden within the context of the assembly.
  - You can use the **Scroll To** command to expand **PathFinder** to the selected assembly feature. If needed, the collapsed **Assembly Feature** collection is expanded.
- You can graphically locate assembly features.
  - You can locate assembly features at the current level, either the top-level or in place activated level.
  - The selected assembly feature is highlighted in the appropriate color for the feature.
  - The selected assembly feature is highlighted in **PathFinder**.

Assign Capture Fit command

Use the **Assign Capture Fit** command  to predefine relationships for parts that will be included in an assembly.

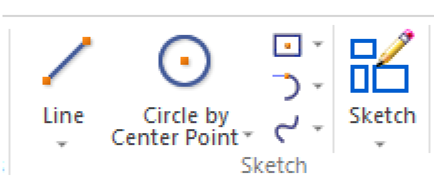
The relationships must be the same for the part you are placing and the target part in the assembly and you should add them in the same order, with the same group name.

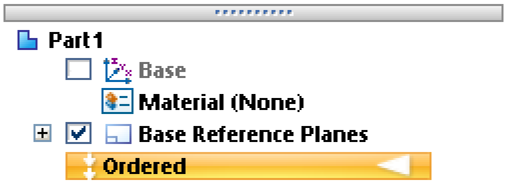
Being able to place parts with predefined relationships increases productivity by decreasing the time and effort required to place parts.

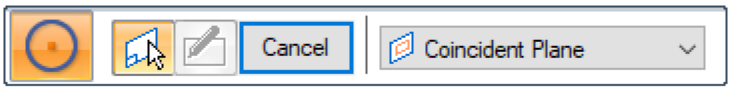
For more information, see Placing parts with predefined relationships.

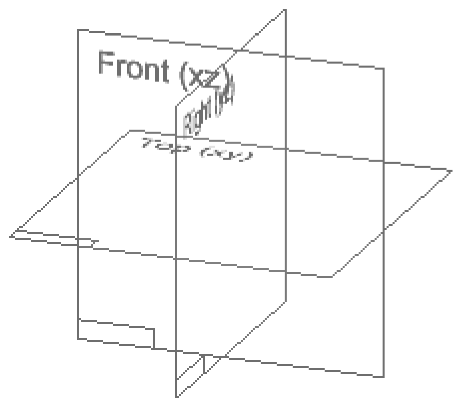
Automatic base reference plane display in a 2D sketch

When any 2D drawing command in the **Sketch** group is started in an empty document, the base reference planes are displayed automatically so that a new user can easily select a plane for sketching.









New options to control base reference plane display

You can now show or hide the base reference planes using the following options on the **Layout** tab of the **Customize** dialog box.

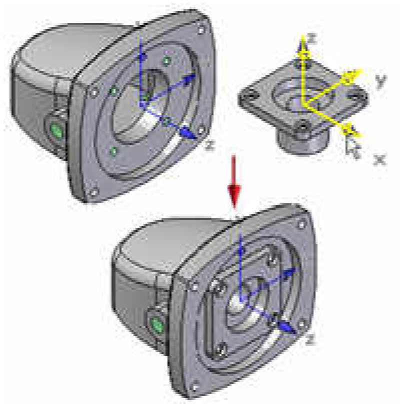
- **Show base reference planes always**
- **Show base reference planes when starting command**
- **Do not show base reference plane**

For more information, see Layout tab (Customize dialog box).

Capture Fit enhanced

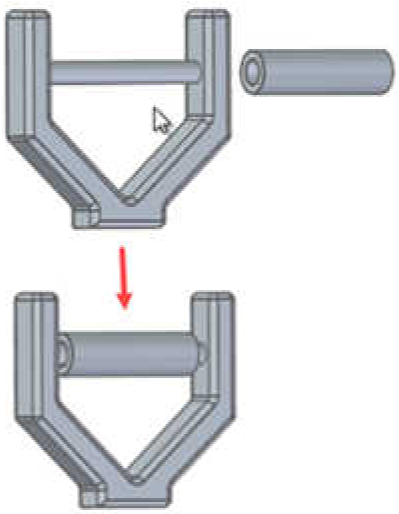
The **Capture Fit** command now supports:

- Match coordinate relationship as a single relation.



For more information, see About the Match Coordinate Systems relationship.

- Center Plane relationship



For more information, see About the Center-Plane relationship.

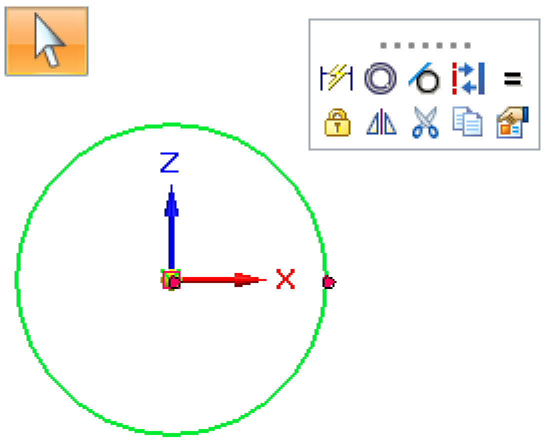
Context toolbar for 2D sketching

In Solid Edge 2022, selecting a 2D sketch element—line, arc, circle, curve, ellipse, conic, or keypoint—now displays a context toolbar at the cursor, so that you can quickly select frequently needed commands, without having to locate them on the ribbon.

The context toolbar is displayed in all environments where 2D sketching commands are available:

- Synchronous
- Ordered
- Assembly (assembly modeling and when in-place activated)
- Draft (Draw in View)

The commands on the context toolbar vary with the element you select.



For more information, see Using the 2D sketching context toolbar.

A new option on the **Helpers** tab in the **Solid Edge Options** dialog box, **Show context toolbar**, controls the display of the context toolbar. The option is selected by default.

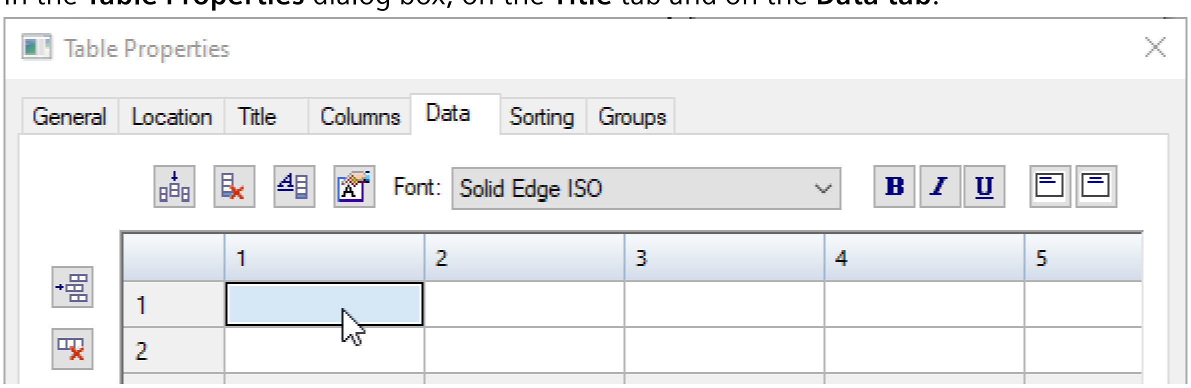
Custom properties and variables in user-defined tables

Although user-defined tables are not associative to a drawing view or to a model, you now can reference custom properties, standard properties, and variables in the table. Information and values derived from property text are updated when you select the **Update All** command.

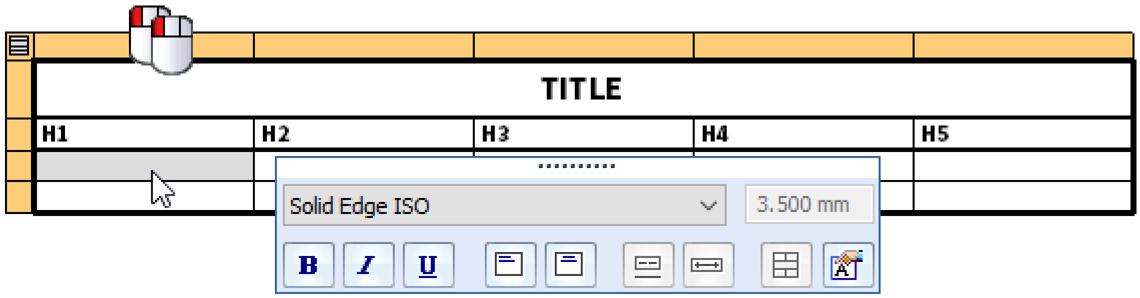
Previously, data entry was completely manual or had to be copied and pasted from other fields.

To enable you to add property text to a user-defined table, the **Insert Property Text** button is available in the following locations when you select the **Table** command:

- In the **Table Properties** dialog box, on the **Title** tab and on the **Data tab**.



- In direct edit mode, on the **Table Format** command bar.

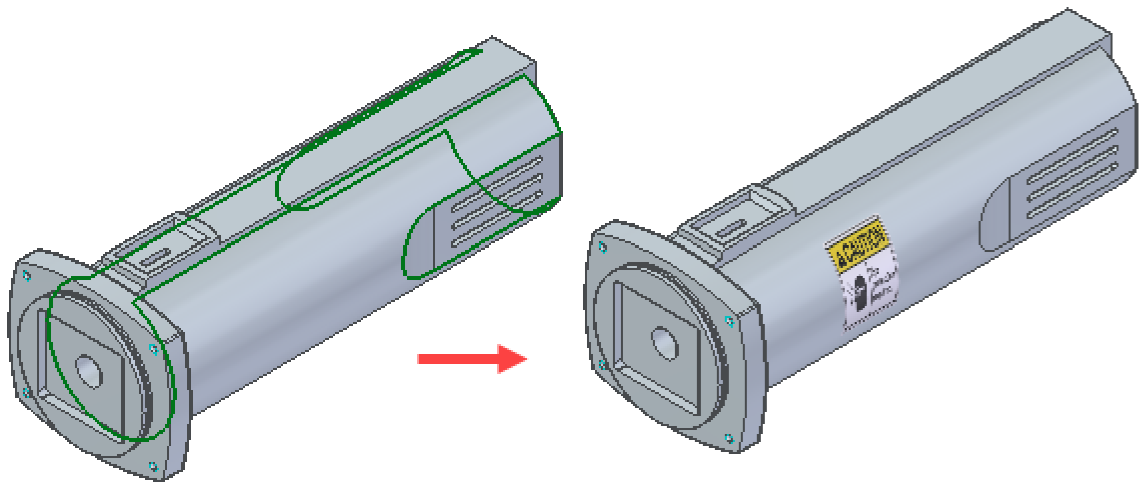


For more information, see the following help topics:

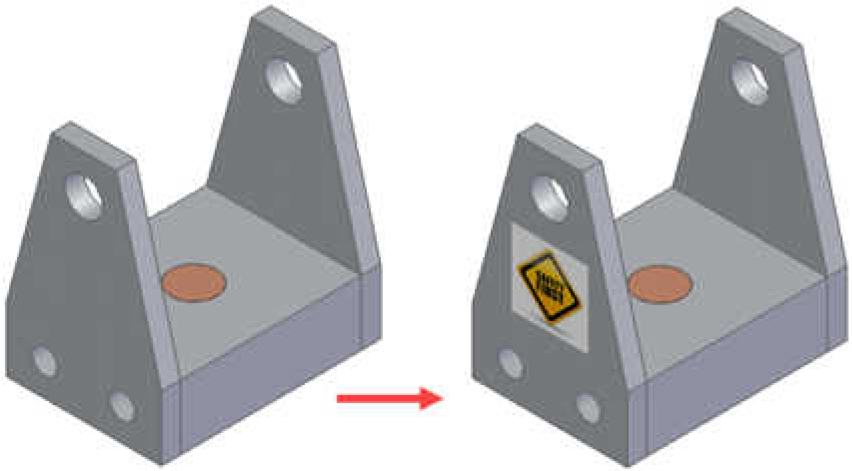
- User-defined tables
- Editing a table directly
- Edit a table cell directly

Decal command enhancements

- The **Decal** command is now available in the synchronous environment. Previously, the command was only available in the ordered enviornment.
- Use the new **Label** decal type to apply an image onto the face relative to the contour of a face.



- The **Decal** command is now available in assembly so you can place decals on the faces of parts directly in the assembly.



Placing the decal within the assembly adds a **Decals** collector within **PathFinder** that contains the decal you placed.

Previously, to add decals to an assembly, you had to place a decal on a part face, save the part, and then place the part in the assembly.

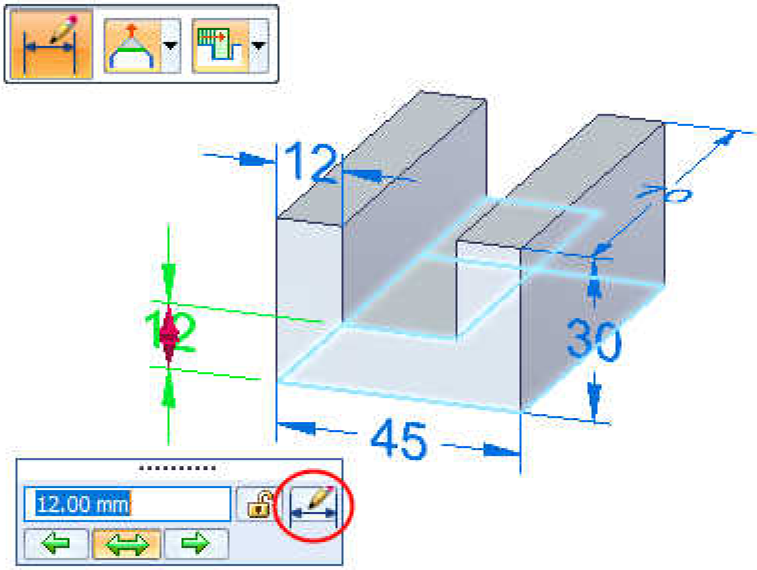
Dimension editing enhancements for 2D and 3D

To make dimension editing consistent and easier to use for new and existing users alike, the following changes were made for dimension editing in 2D and 3D environments.

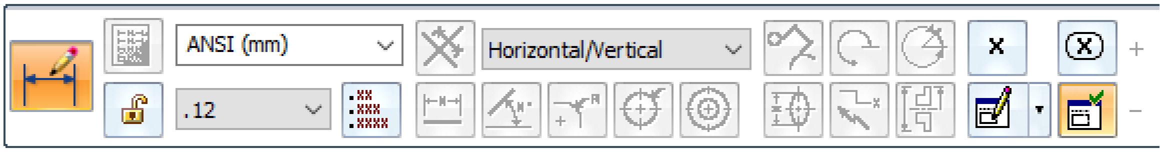
Easier access to dimension editing tools

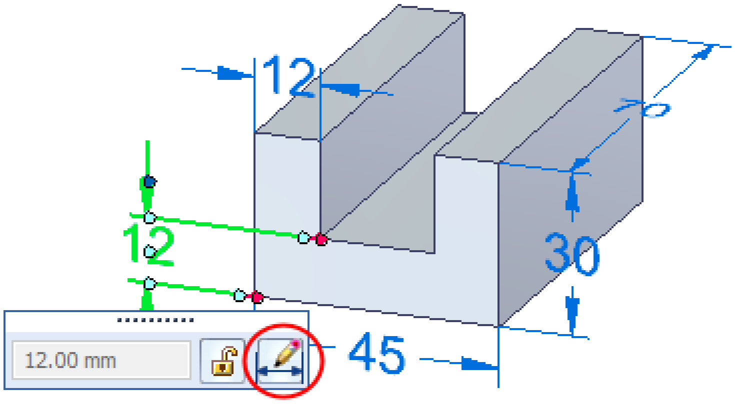
- In modeling environments and in draft, you can now click anywhere on the dimension line to access dimension value editing controls, as well as clicking the dimension text.

- In synchronous mode, the **Modify** button  was added to the dimension toolbars. Click this button to switch between dimension value editing mode, as shown next,



and making dimension formatting changes, as shown here:





Identifying driving and driven dimensions

It is easier to identify whether a dimension is driving or driven.

- The lock/unlock / icons on the dimension editing command bars have a new command name and tooltip to make it easier to understand how to use it:

  Driving/Driven Dimension—Lock to make it a driving dimension value. Unlock to make it a driven dimension value.

- When you click the **Driving/Driven Dimension** button on one of the dimension editing command bars, it updates the status of the lock icon on other dimension editing command bars that are displayed.

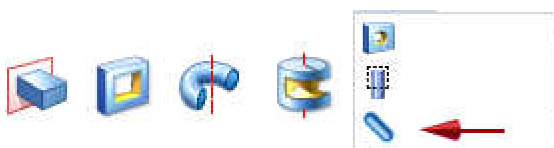
- You also can use the shortcut menu of a selected dimension to identify and modify a driving or driven dimension. The following command names and tooltips were updated:

  - **Driving Dimension—Modifies a dimension to make it driving.**
    Selecting this command locks the dimension, so that its value controls other sketch elements. It also updates the icon on command bars to show that the dimension is locked: .

  - **Driven Dimension—Modifies a dimension to make it driven.**
    Selecting this command unlocks the dimension, so that it can be controlled by other sketch elements. It also updates the icon on command bars to show that the dimension is unlocked: .

Easier to identify extent options for ordered commands

For these commands in the Ordered environment—**Extrude**, **Cut**, **Revolve**, **Revolved Cut**, and **Slot**—



the command bar now contains three option buttons to define extent direction rather than two:

The option button (1) **One-Sided Extent** is not new functionality, but its visibility on the command bar makes it easier to identify which option is active and to select the extent direction you want to apply.

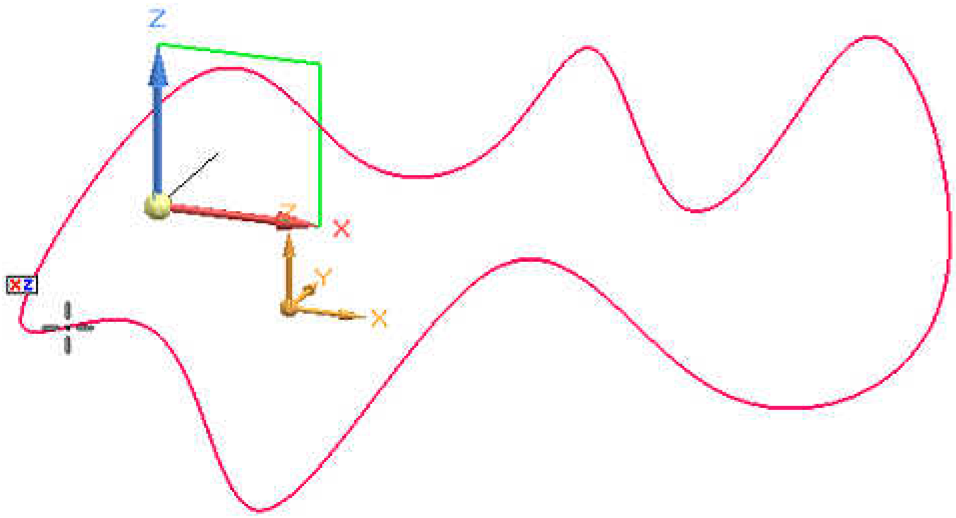
Previously, one-sided extent was not shown on the command bar, but it was the default mode when you selected the **Finite Extent** type and deselected options (2) **Non-Symmetric Extent** and (3) **Symmetric Extent**.

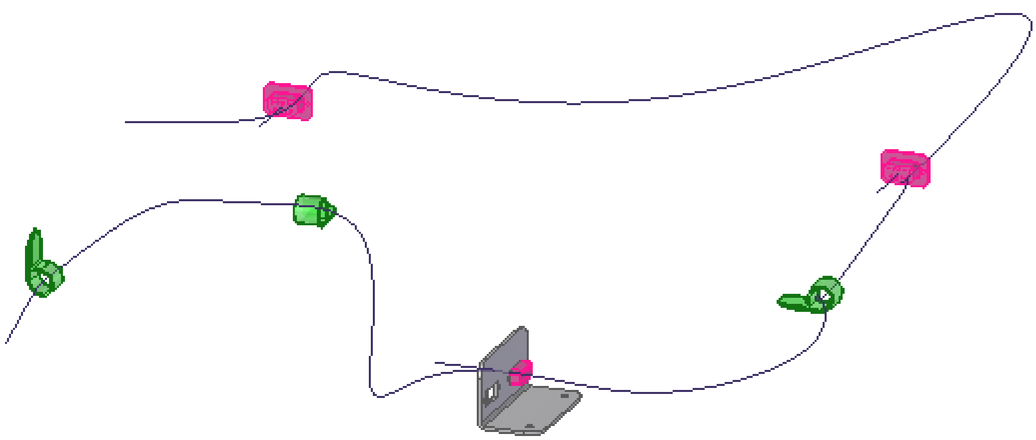
Enhancements to 3D Curves

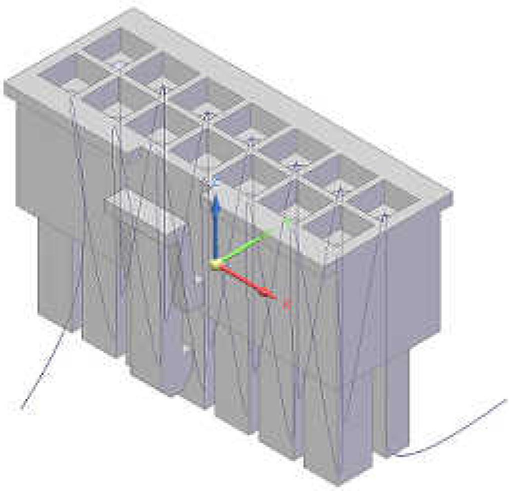
The **3D Curve** command  in the 3D Sketching environment now provides the same functionality in part, sheet metal, and assembly documents as the following commands used by XpresRoute to route paths and path segments, and by Surfacing to create surfaces:

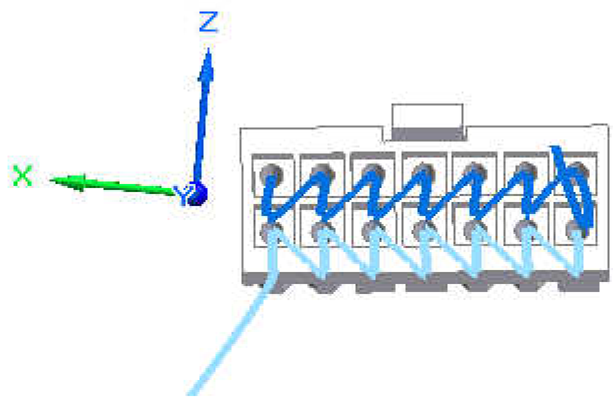
- **Keypoint Curve** command
- **Curve Segment** command

With the options added to the 3D Curve command bar, you now can use the **3D Curve** command to route many types of paths, for example:









To learn how to use the command, see Create a 3D curve.

Enhancements to 3D sketching

Some commands that were available only for 2D sketching and drawing are now also available in modeling environments for 3D sketches.

More commands available for 3D sketching

The following drawing and sketching relationship commands are now available on the **3D Sketching** tab in part and sheet metal documents:

- In the **3D Draw** group:
  - **Mirror** command
  - **Route** command
  - **Auto-Scale Sketch** command
  - **Equation Driven Curve** command
- In the **3D Relate** group:
  - **Symmetric** command
  - **Lock** command

- **Rigid Set** command

3D sketching commands available in more design environments

When creating a 3D sketch, the newly added commands listed above are also available on the **Home** tab in the following environments:

- Assembly
- Frames
- XpresRoute

Cut, copy, and paste 3D sketch elements

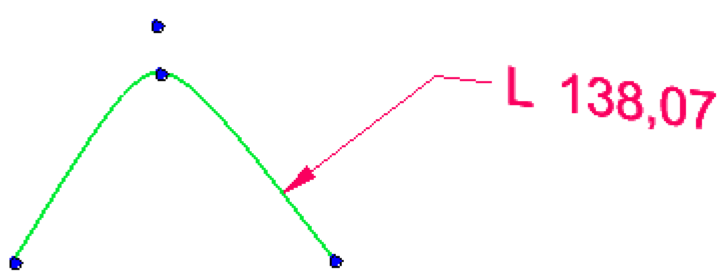
The **Cut**, **Copy**, and **Paste** commands are now supported for all 3D sketch elements in ordered and synchronous sketches. These commands are available on the shortcut menu of a selected element.

Expanded support for conic curves

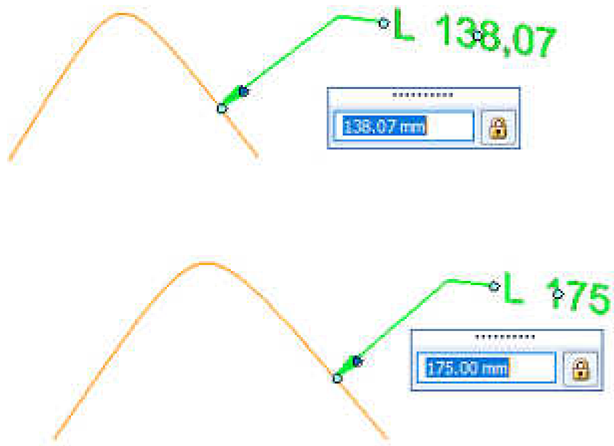
The following enhancements were made to support the **Sketching** tab→**Draw** group→**Curves** list→**Conic** command. For more information, see Draw a conic curve and Edit a conic curve.

Measure curve length with a smart dimension

You can use the **Smart Dimension** command to add a dimension that measures the length of a conic curve.

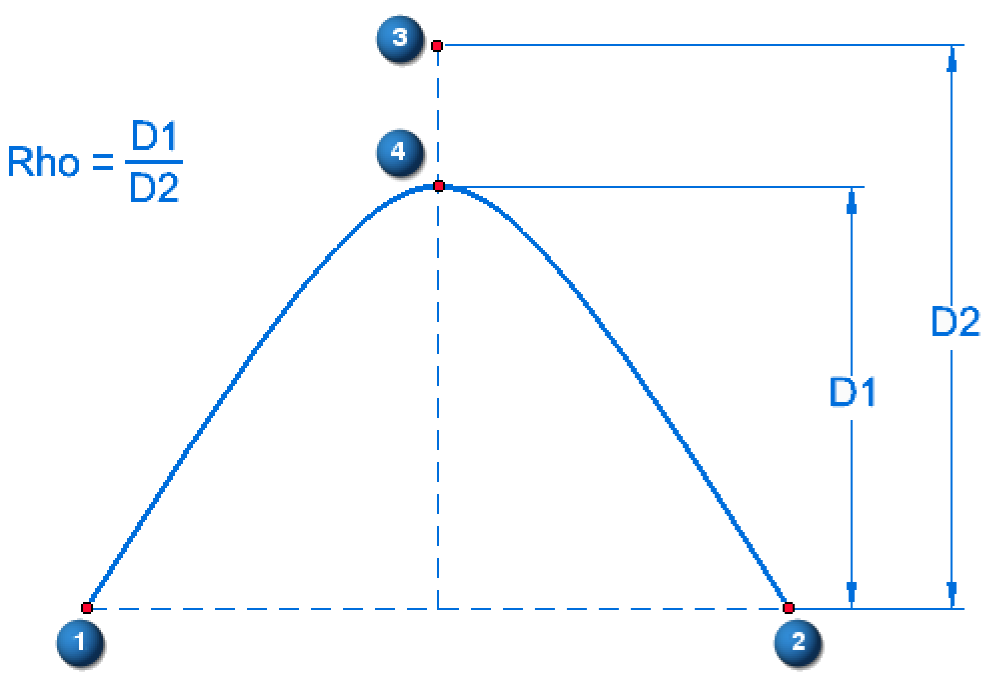


You can edit the dimension to change the curve length.



Rho value supported in the Variable Table

You can use the **Variable Table** to define a formula for the Rho value [Rho=(D1)/(D2)] in a conic curve. The closer the Rho value is to 1.00, the more elongated is the curve.



Using a conic curve

More drawing and sketching applications enable you to use a conic curve. For example, you can include a conic in a selection set when using the following commands:

- In the **Relate** group:

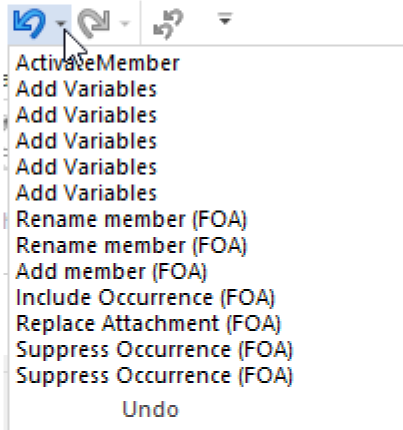
  - **Symmetric**
- In the **Draw** group:
  - **Fill**
  - **Fillet**
  - **Offset**
  - **Convert to Curve**
- In the **Pattern** group:
  - **Rectangular**
  - **Circular**
- In the **Dimension** group:
  - **Smart Dimension**
- In the **Evaluate** group:
  - **Goal Seek**
  - **Area**
- In the **2D Measure** group:
  - **Smart Measure**
  - **Distance**
  - **Total Length**

Family of assemblies enhancements

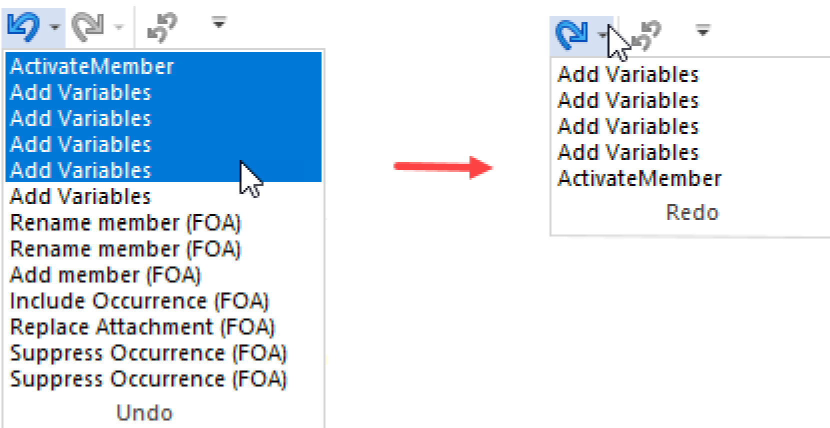
Family of assemblies enhancements include:

- There are now separate commands for Populate and **Update**, which reduces complexity and increases flexibility. Previously, Populate and **Update** were combined as a single operation.
  For more information, see Populating members.

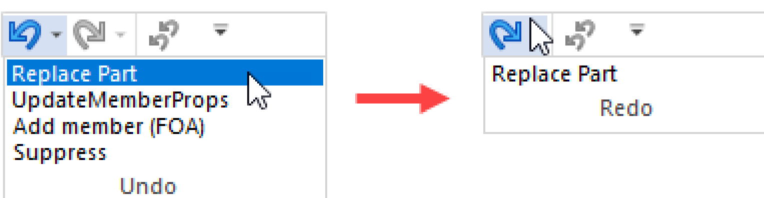
- Family of assemblies now supports Undo and Redo in the **Alternate Assemblies** table, global mode, and local mode. You can undo and redo actions, such as suppress, unsuppress, add or delete members, or modify components.
  As you perform actions on family members, entries are added to the **Undo** log.



  As you undo selected actions in the **Undo** log, entries for those entries are added to the **Redo** log.



  You can also undo or redo replace part inside or outside the **Alternate Assemblies** table.



- Use the new **Suppress in all** and **Unsuppress in all** commands to suppress and unsuppress a component in all members.

- A new **Remove Suppress Components** command removes suppressed components from **PathFinder**. To remove suppressed components, in **PathFinder**, right-click the assembly and then click **Remove Suppress Components** to remove any suppressed components from **PathFinder**.

- A new **Keep Suppress Components** command keeps suppressed components in **PathFinder**. To keep suppressed components, in **PathFinder**, right-click the assembly and then click **Keep Suppress Components** to keep any suppressed components in **PathFinder**.

- The **Add Suppression Variable** command and **Delete Suppression Variable** command are available to suppress components in the assembly environment and the **Alternate Assemblies Table**.
  For more information, see Suppressing and unsuppressing components in an assembly

- You can now selectively update and populate family of assembly members. To do this, clear the **Select member(s) to save** option and then select the members that you want to update or populate.

- There is a new **Save Option** on the include:
  If you select the **Save Only Source and Active Member (Faster)** option and then click **Ctrl+S**, only the source and active members are saved.
  If you clear the option and press **Ctrl+S**, all members are saved.
  For more information, see Saving members.

Fastener System dialog box enhancements

The **Fastener System** dialog box has been enhanced to be more user friendly.

- The **Add Fastener** button now expands the right pane of the dialog box so you can select the appropriate fastener. You can use the arrow buttons to expand and collapse the pane.

- You can use the **Preview in Assembly** button to display the fastener component highlighted in green.

- You can now view the list of custom properties and characteristics added with Standard Parts Administrator.

- You can right-click on a column in the components pane and then click **Format** to display the **Format Columns** dialog box. Use the dialog box to control the display and order of the columns.
  You can also click and drag the columns to rearrange the columns within the components pane.

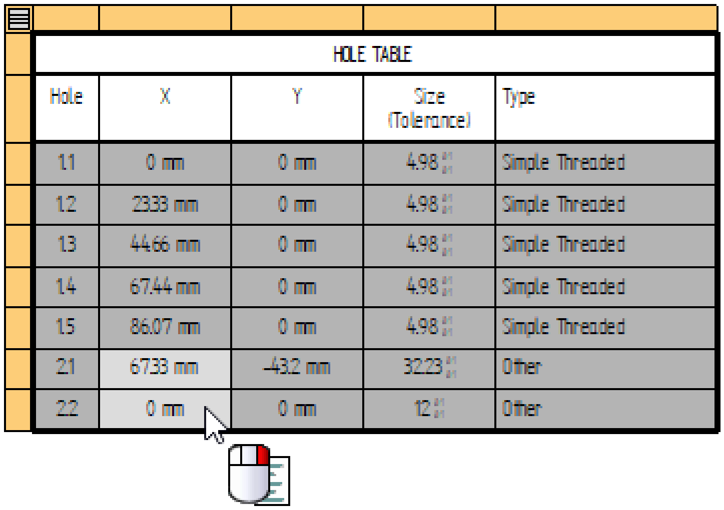
Format accuracy round-off in hole tables

When you place a hole table in draft, you now can set different accuracy round-offs on different holes based on application requirements. For example, dowel holes may require a finer round-off compared to threaded holes.

Hole	X	Y	Size
A	61.89 mm	17.36 mm	8.4 mm
B	53.29 mm	40.12 mm	8 mm
C	18.72 mm	78.56 mm	7.188 mm

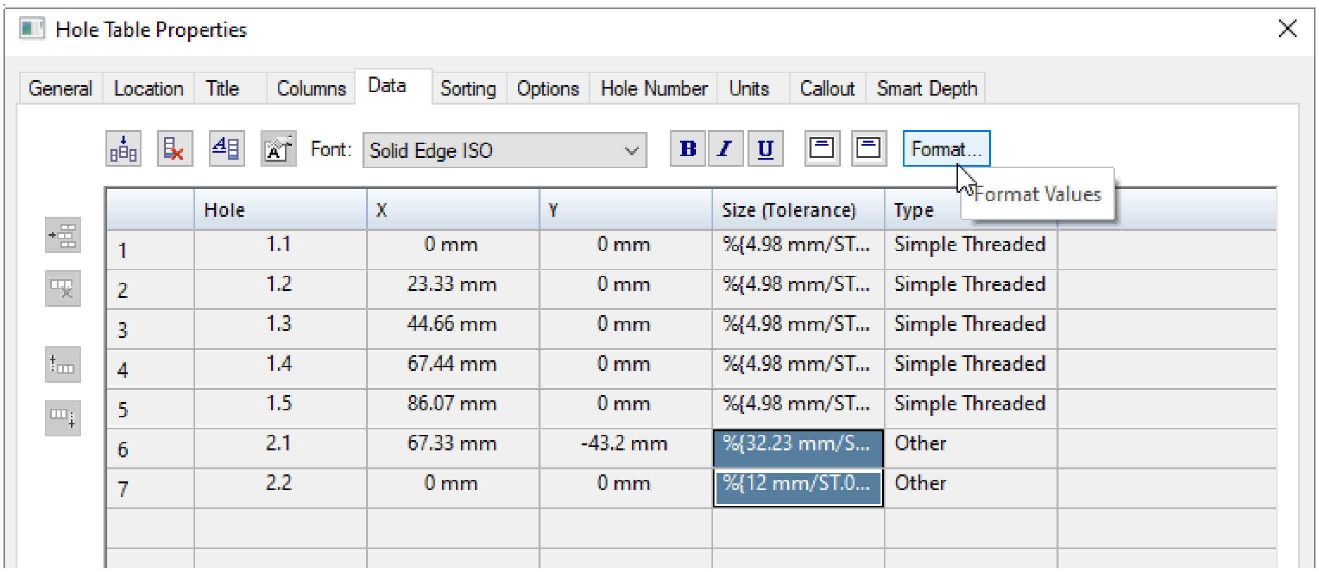
- Previously, a single round off was applied to the entire column, as it is a property of all cells in the column.

- Now you can change the formatting for hole size round-off, units, and tolerance on one or more individual cells. You can do this using the **Format Values** dialog box, which you can open from the context menu in direct edit mode,



HOLE TABLE				
Hole	X	Y	Size (Tolerance)	Type
1.1	0 mm	0 mm	4.98	Simple Threaded
1.2	23.33 mm	0 mm	4.98	Simple Threaded
1.3	44.66 mm	0 mm	4.98	Simple Threaded
1.4	67.44 mm	0 mm	4.98	Simple Threaded
1.5	86.07 mm	0 mm	4.98	Simple Threaded
2.1	67.33 mm	-43.2 mm	32.23	Other
2.2	0 mm	0 mm	12	Other

Or by selecting the **Format** button on the **Data** tab in the **Hole Table Properties** dialog box:



For more information, see Hole tables and Editing a table directly.

Frame splitting enhanced

Use the new **Split intersecting frames** option on the **Frame Options** dialog box to automatically split the frames where the paths intersect. It also splits the sketch used to create the frames.

Previously, you had to split all of the intersecting path sketches to split the intersecting frame. Now, you can use the new option in conjunction with the **Single frame for collinear segments** option to get the desired results.

You can also toggle the combination of which frames you want to split and the frames you want to remain unsplit.

For more information, see Splitting intersecting frames.

Frame trimming enhanced

You can now trim frames to multiple objects, such as faces, bodies, or planes. Previously, you could select only one object when trimming frames.

For more information, see Trimming frames.

Generative Design: New solver

A new solver, Topology Optimization for Design (TO4D), replaces Frustum as the underlying engine in Solid Edge Generative Design.

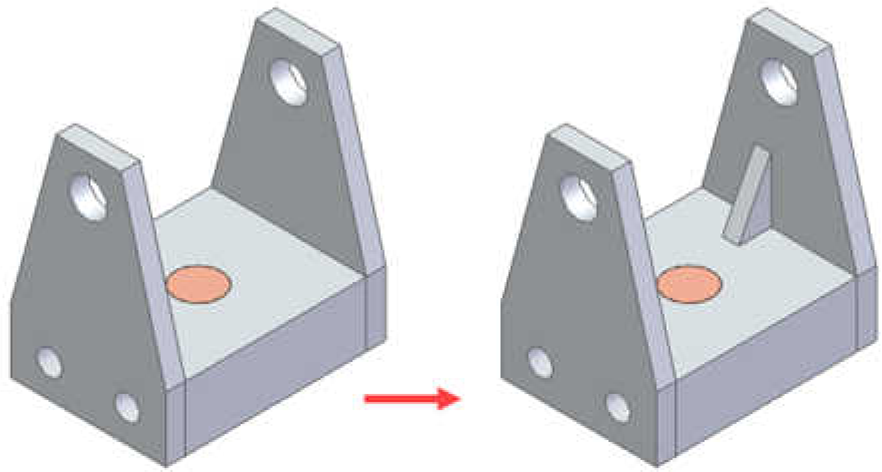
- If a NVIDIA graphics processing unit (GPU) is available on your computer, the study will use it for faster optimization when you solve it. If a NVIDIA GPU is not present on the machine, the solver will use CPU.
- There are now two factor of safety optimization options available in the **Generate Study** dialog box:
  - **Yield Stress**
  - **Ultimate Stress**
- In the **Manufacturing Settings** dialog box, the following options were removed:
  - **Prevent enclosed void creation**–This option was no longer needed, as the new solver does not produce voids.
  - **Material spreading**–Future support for this option is planned.

For more information, start with the Generative Design overview. On the **Generative Design** tab in Solid Edge, hover over a command and press F1 to learn more.

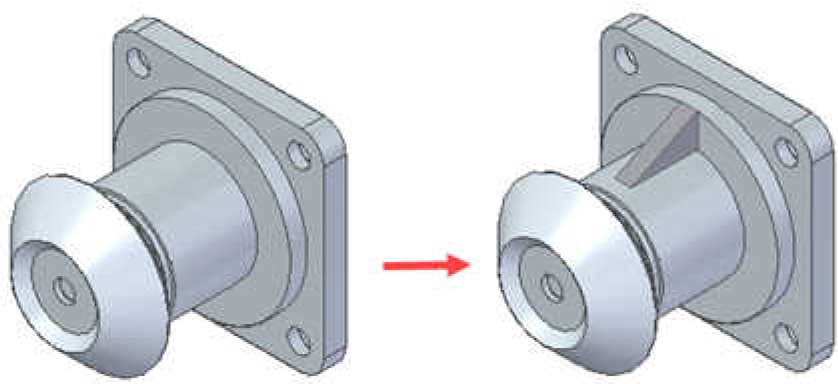
Gusset Plate command

Use the **Gusset Plate** command to create gussets on the faces of selected assembly components.

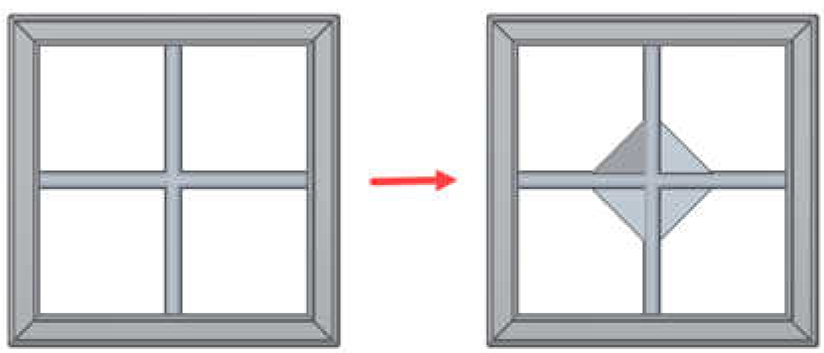
You can create the plate between planar faces



a cylindrical face and a planar face,



or between cylindrical faces.



Immediate feedback for dimension property changes

The **Apply** button was added to the **Dimension Properties** dialog box, so you can make property-related changes (such as text size, terminator style, round-off, and symbol orientation) to one or more selected dimensions and immediately see their effect.

Example:

1. Right-click a dimension and choose the **Properties** command.

2. In the **Dimension Properties** dialog box, on the **Terminator and Symbol** tab, choose a different terminator type from the **Origin type** list.

3. Click **Apply** to see your change.

   Previously, you had to save and close the dialog for the changes to be applied.

You can use the **Undo** command to undo the result of your property edit.

Improved 2D sketching workflow in assembly and in ordered mode

The following workflow enhancements were made for 2D sketching in an assembly and in an ordered document. For more information, see Draw an ordered sketch of a part.

Skip the Sketch command: Just start sketching.

The 2D drawing commands are now available in an assembly document and in an ordered document to directly begin sketching a line, circle, or arc with a single click, without first having to select the **Sketch** command. This reduces the steps required to begin drawing an ordered sketch. It also makes it more intuitive for a new user to know how to get started.

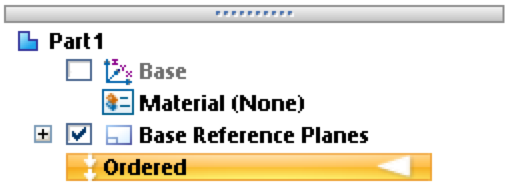
The commands were added to the **Sketch** group in the following locations:

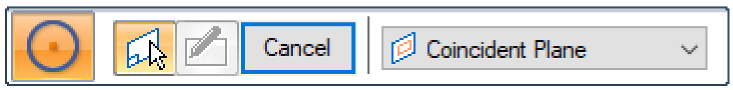
- In Ordered Part and Sheet Metal, on the **Home** tab.

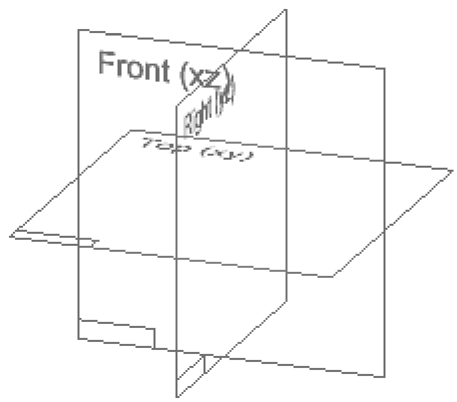
- When in-place activated from an assembly, on the **Home** tab and the **Sketching** tab.

Example:

Now when you open an ordered document, you can click the **Home** tab→**Sketch** group→**Circle by Center Point** command to directly display the drawing command bar *and* to choose the sketch profile plane.







Previously, you had to:

1. Select the **Sketch** command to open the Sketch environment.
2. Use the **Sketch** command bar to define the sketch profile plane.
3. Select the drawing command you want to use.

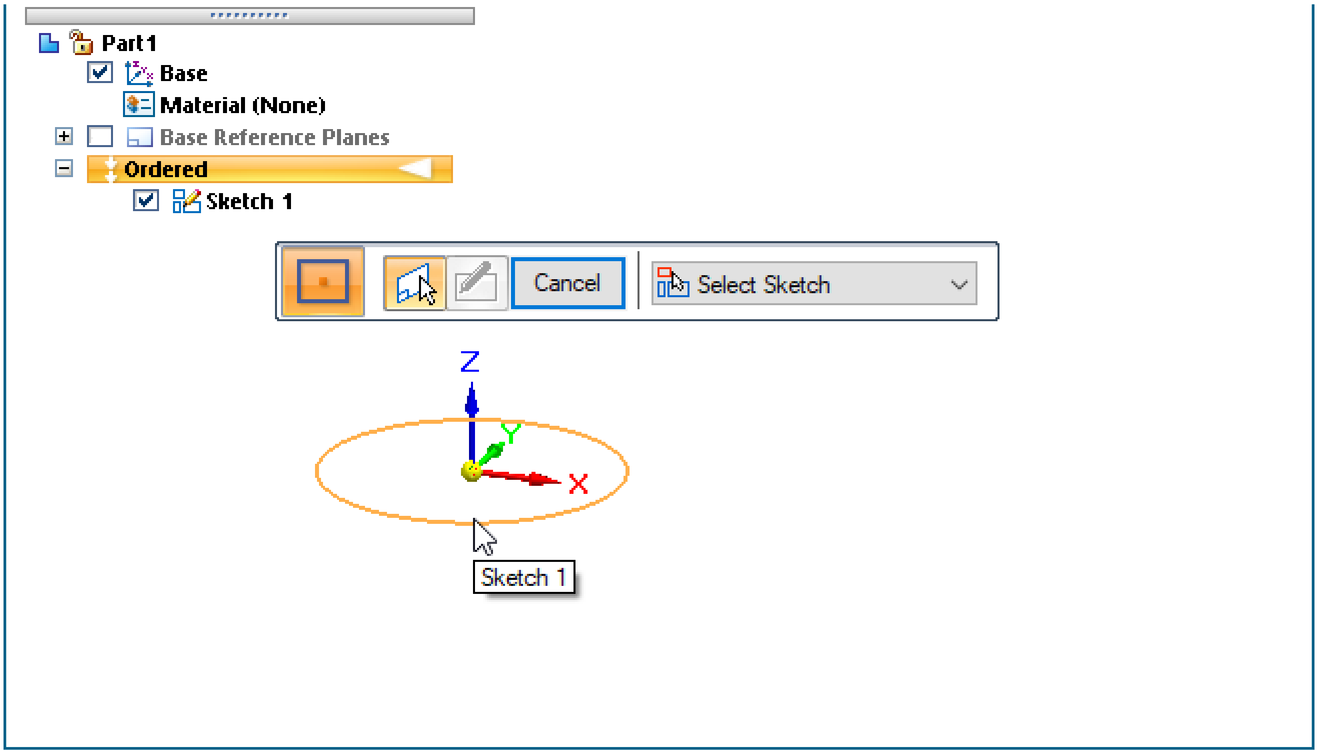
Add to an existing ordered sketch instead of starting a new one.

Instead of creating a new sketch as shown in the previous example, you also have the option to add to an existing ordered or assembly sketch. The additional text in **PromptBar** identifies the expanded workflow options:

Click on an existing sketch to edit or click on a planar face or reference plane to create a new sketch. To select a plane change the Create-From options by clicking on the list.

Example:

If you select a sketch element from an existing ordered sketch, that sketch will be edited in the sketch environment using the selected drawing command.



In an assembly document, you can only add to sketches in the top-level assembly. When in-place activated from the assembly, you cannot select sketches from other documents.

Improved performance during derived drawing view update

Support for multi-core drawing view processing, which was introduced in Solid Edge 2020 to speed the performance of large draft files, is now expanded to provide much faster update for drawing views derived from other drawing views (for example, section, broken-out section, and auxiliary views).

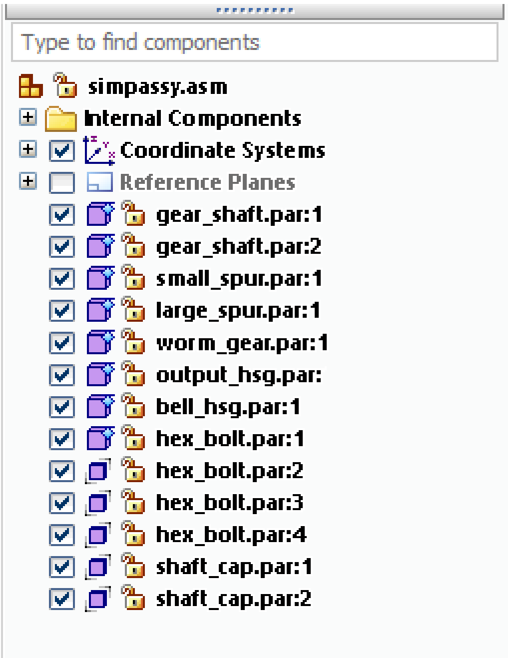
In the Draft environment, you can enable this capability in the **Solid Edge Options** dialog box. From the **Application** menu, choose **Settings→Options→General** tab, and select the check box, **Enable multi-core drawing view processing**.

For more information, see the help topic, General tab (Solid Edge Options dialog box, Draft environment).

Internal Components enhancements

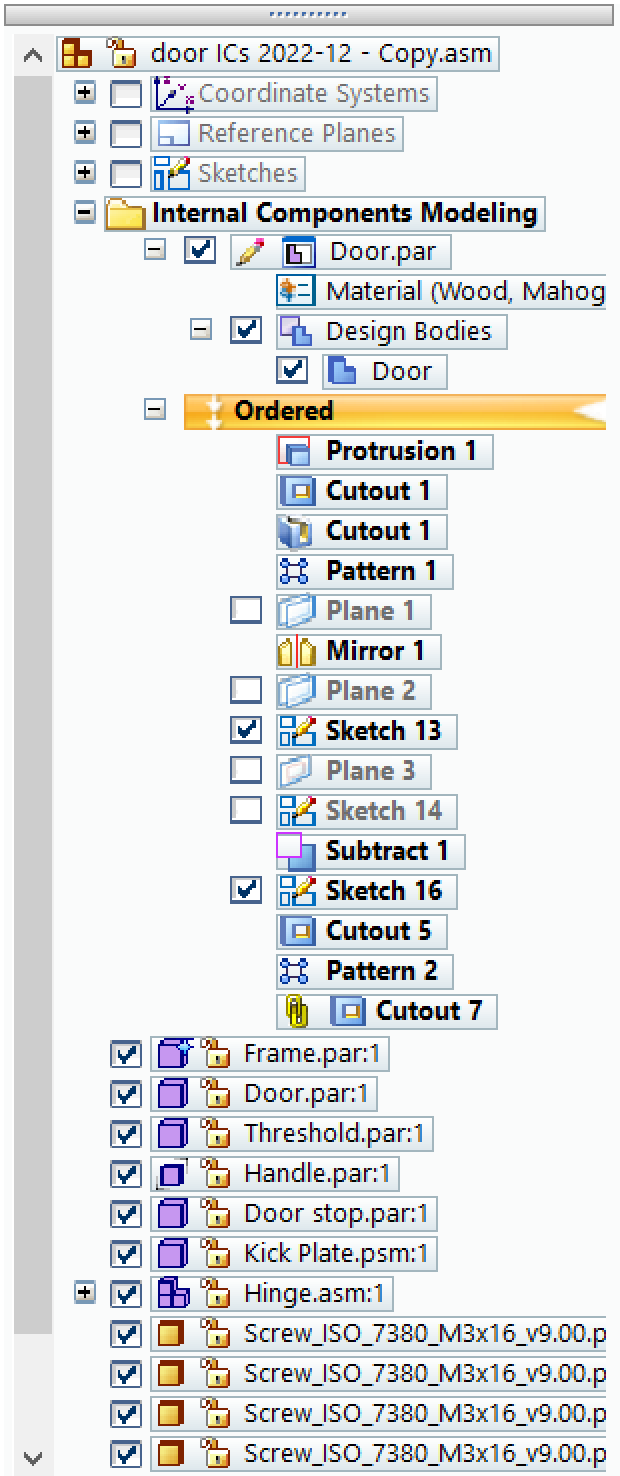
Solid Edge 2022 contains many enhancements supporting internal components:

- The **Internal Components** collector was moved before the **Coordinate Systems** collector in **PathFinder**.



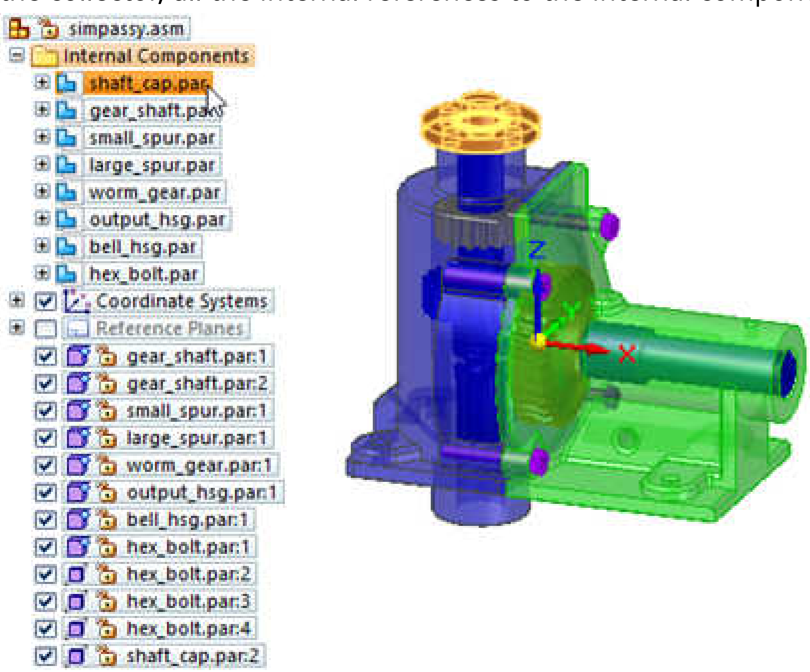
Previously, the **Internal Components** collector was located immediately before the assembly in **PathFinder**. This was confusing because when the **Internal Components** collector was expanded, it appeared that the components were actually part of the assembly.

- In the **Assembly PathFinder** feature tree, when you select an internal component for editing in-place, the **Internal Components Modeling** tree is expanded for that component.

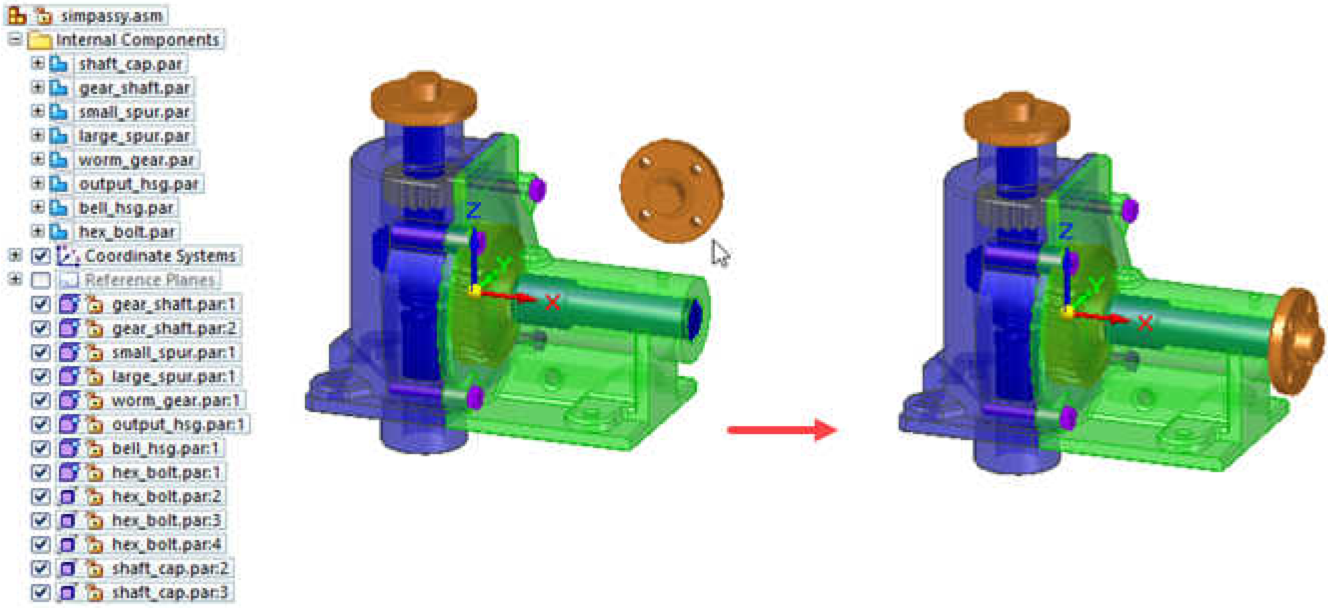


If you open another internal component for editing, then the previous internal component tree is collapsed, so you can focus on the one currently being edited.
If multiple internal components are open for editing simultaneously, then you can drag a **Design Body** from one internal component to another in **PathFinder**.

- You can assign and maintain variables in internal components using these commands on the **Tools** tab→**Variables** group:
  - **Variables** command —Opens the **Variable Table**, which shows all variables for all internal components being edited.
  - **Peer Variables** command —Displays the variables for the selected internal component.
- You can assign a material to one or more internal components in the **Material Table**. With materials assigned, you can save attributes such as density, and use the **Physical Properties** command to compute and display physical properties (center of mass, center of gravity) of an assembly that includes internal components, or for the active part or internal component. For more information, see Apply a material to internal components.
- Internal component properties are now managed through **Property Manager** . Internal components are displayed in **Property Manager** like actual parts and assemblies. For more information, see Working with internal components.
- You can now create new internal references to internal components in an assembly. These references are available for commands such as **Pattern**, **Transfer**, and **Disperse**. Previously, internal references could only be created through import translation. For more information, see Working with internal components.
- You can now drag internal components into an assembly. When you locate an internal component in the collector, all the internal references to the internal component highlight.



After you drag the internal component into the assembly, you can use the **Assemble** command to apply relationships between the internal component and one or more target parts in the assembly.



For more information, see Dragging internal components into an assembly.

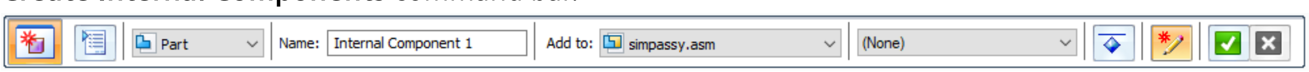
Internal Components Modeling mode

A new assembly modeling mode, **Internal Components Modeling**, is available for you to design a part, sheet metal, or subassembly component as an *internal component* in the context of an assembly.

Previously, internal components could be created only by importing a non-Solid Edge file.

- You now can create an internal component as a single multibody part using the new **Home** tab→**Components** group→**Create Part In-Place** list→**Create Internal Components** command .

- You can add bodies to the newly created internal component using the **Edit in-Place** button on the **Create Internal Components** command bar.



- You can add and edit internal component features using all the modeling commands that are available on the ribbon in ordered and synchronous part, ordered and synchronous sheet metal, and in assembly modeling.

- You can use the new **Activate Internal Component** command, to activate a different internal component for editing, without having to close assembly modeling.

- A new shortcut command is available in assembly modeling to show only the features for the active internal component: **Show Body Features Only**.
  Use this option to focus the **PathFinder** feature tree on just the internal component you are editing.

For more information, see Create an internal component in an assembly.

Keyshot support for decals

Support for decals is extended to models rendered using **Keyshot**. Now, when you render a part or assembly containing a decal, the decal is displayed in the image.

For more information, see Decal command and Rendering parts and assemblies.

Lofted flange user interface enhanced

The **Bending Method** page on the **Lofted Flange** dialog box has been enhanced to provide access to pre-Solid Edge 2020 functionality.

There are now three options on the tab:

Advanced

This option attempts to create only planar, cylindrical, and conical geometry. This is the default option.

Bends

This option creates planar plates and cylindrical bends.

Formed

This option provides pre-Solid Edge 2020 functionality, which may create non-planar, non-cylindrical, non-conical flanges.

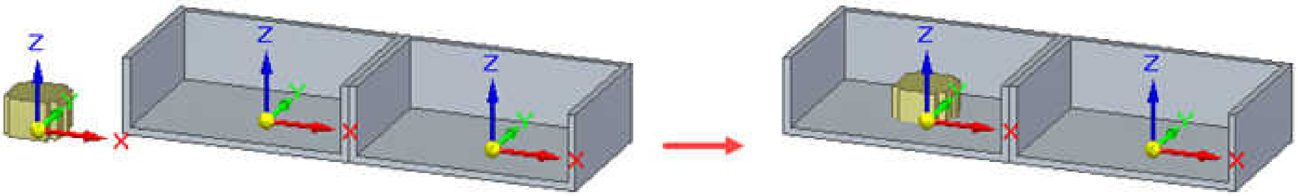
New tooltips provide information about each of the three options and bend tables are accurately calculated for each option.

For more information, see Bending Method page.

Match Coordinate Systems enhancements

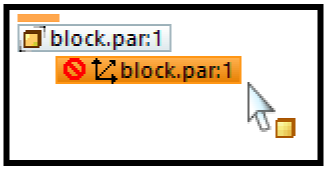
Enhancements to the **Match Coordinate Systems** command include:

- Use the new **Single Coordinate System Constraint** option  on the **Match Coordinate Systems** command bar to create a match coordinate system as a single relation.

  This provides a quick and easy edit method and is useful when editing and redefining the coordinate system. This relationship is supported in family of assemblies, **Assembly Relationship Manager**, and **Relationship Assistant**.

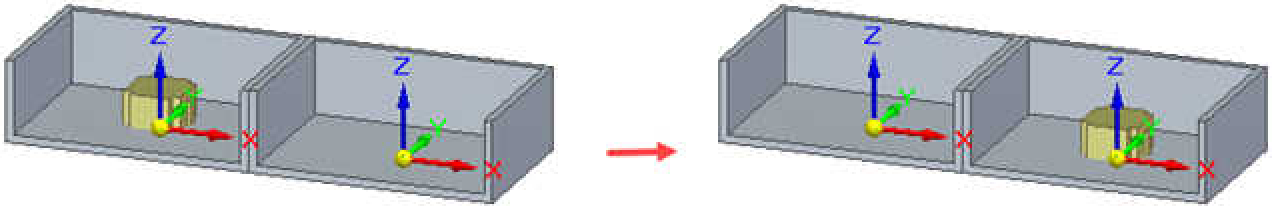
- You can suppress and unsuppress match coordinate system relationships.

- You can add a suppression variable for match coordinate system relationships. Any changes you make to the variable are reflected in **PathFinder**.

- You can see the relationship status and any failed error state in **Assembly Relationship Manager**.

- Right-click the Match coordinate system relationship and use the **Edit Definition** command to match the part to a different coordinate system.

  For more information, see Edit a Match Coordinate Systems single relationship.

More update methods for text profiles with property text

When the value of property text used in a text profile is changed, you can select either of the following commands to update the text profile and the sketches and features where it is used:

- **Update Active Level**

- **Update All Open Documents**

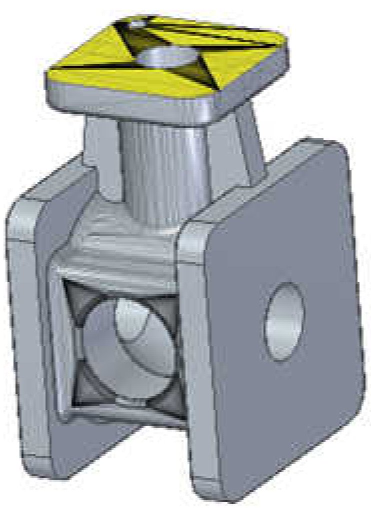
Previously, only the **Update All** command was available for explicitly updating a text profile.

For more information, see Edit a text profile.

Mixed mesh modeling supported

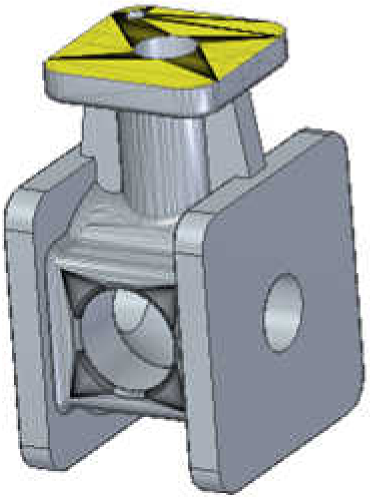
Solid Edge now supports mixed mesh modeling, where models can contain a mix of mesh and classical (analytical, spline) geometries. Such geometry can be generated in Solid Edge by mixing classical B-rep bodies with mesh bodies, or by adding features to mesh bodies output from generative design, or by importing geometry from other applications that create mixed models.

Solid Edge supports the creation and use of hybrid models in part, sheet metal, assembly, and draft. Hybrid models offer the precision and design control of classical and analytical elements, while also integrating mesh, scanned, and facet geometry in a single model.



This enhancement of Solid Edge now provides the following for mixed mesh bodies:

- Mixed bodies are shown as mesh bodies in the **PathFinder**.
- Basic ordered commands like **Extrude**, **Revolve**, and **Cut** are supported to create mixed bodies.
- Commonly used commands, such as **Physical Properties**, **Measure**, **Geometry Inspector**, and **Optimize**, as well as commands in the **Dimension** group, the **Annotation** group, and the **Analyze** group, are either partially or fully supported.
- The new **View** tab→**Style** group→**Show Facet Edge**  command controls the display of facet edges within a model. When selected, the command displays the facets; when deselected, the facet edges are not displayed.



- Reverse Engineering commands support mixed mesh bodies.
- You can import and export mixed mesh bodies from formats that support mixed bodies, such as Parasolid and PLM XML.

For more information, see Convergent Modeling.

Multi-edge supports large trim gap

The **Multi-Edge Flange** command now supports large trim gap values.

Object-action input to 2D sketching commands

For more information about these enhancements, see Using the 2D sketching context toolbar.

Preselect 2D elements

Many commands used in 2D sketching now accept one or more preselected 2D elements as input to the command you choose next. If any of the selected elements are ineligible as input to a command, they are removed from the selection set.

Previously, you had to:

1. Select the command.

2. Select one or more elements.

This enhancement applies to all the relationship commands in the **Relate** group. It also applies to sketch modification commands, such as **Construction**, **Trim Corner**, and **Extend to Next**, as well as some commands in the **Dimension** group and the **Distance** command on the **Measure** group.

> Example:
>
> Use the **Select** command to select one or more lines, and then choose the **Horizontal/Vertical** command to make the selected lines horizontal or vertical.

Preselect a keypoint

Similarly, you can preselect a 2D profile sketch keypoint as input to the following commands:

- **Horizontal/Vertical**

- **Connect**

- **Distance Between**

- **Coordinate Dimension**

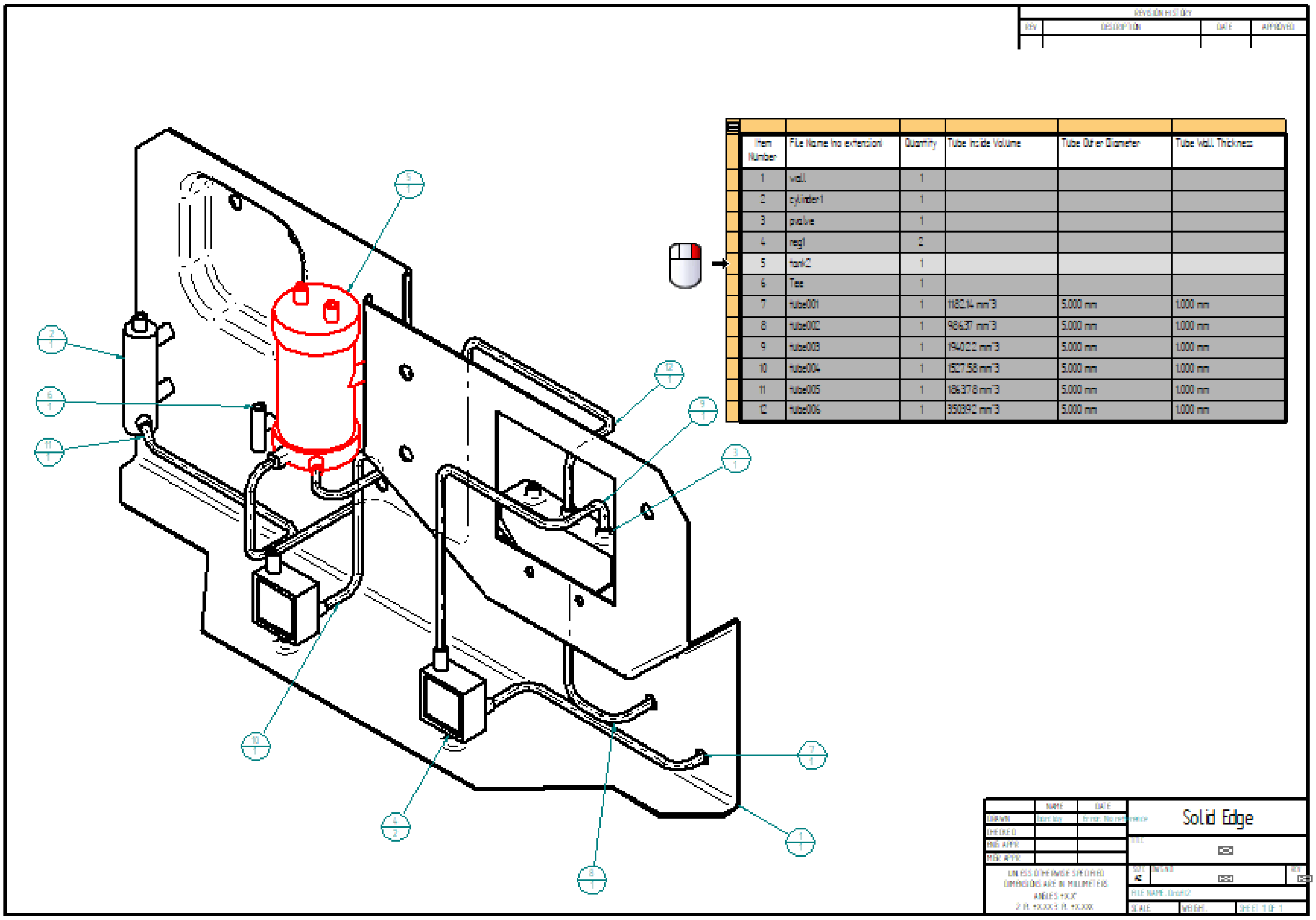
- **Measure** group→**Distance** command

> Example:
>
> When the **Select** command is active, you can select a keypoint on a 2D sketch element as input to the **Connect** or **Distance Between** command. Press F5 or Esc to clear the display of keypoints.

> Note:
>
> To turn off this workflow capability for keypoint selection, clear this new option on the **Cursor** page (**IntelliSketch** dialog box): **Do not show key points in Select command**.

Open a model from a parts list

You now can directly open the part or assembly model associated with a parts list. The **Open** command is available on the context menu of a selected item number row in a parts list, as shown in the following example.



The **Open** command also is available from a selected member quantity column in a family of assemblies parts list, or from any part name in the FOA parts list.

Previously, you were able to highlight every item in the parts list or display a preview window showing the model associated with an item in the parts list, but you had to double-click the drawing view to open the model, and then navigate to the part in the assembly.

For more information, see Open a model from a parts list.

Ordered modeling and assembly feature commands support mixed bodies

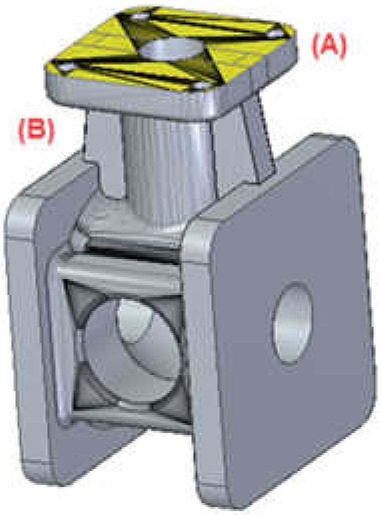
Solid Edge now provides ordered modeling support for mixed bodies, which are bodies that contain a combination of both classic and facet geometry.

In ordered mode and in an assembly, the following commands can be used to either create or modify features on mixed bodies:

- Unite
- Intersect
- Split
- Subtract
- Hole
- Slot
- Thread
- Round (constant radius)
- Chamfer

> Note:
>
> The **Round** and **Chamfer** commands support mixed mesh modeling, but they create a mesh face.

- Solid Edge also supports patterning in the mixed mesh bodies. In the following example, a hole was patterned (A) and a rib was mirrored (B).



The rib and hole were created on mesh bodies, but the faces created by the pattern and mirror are B-Rep.

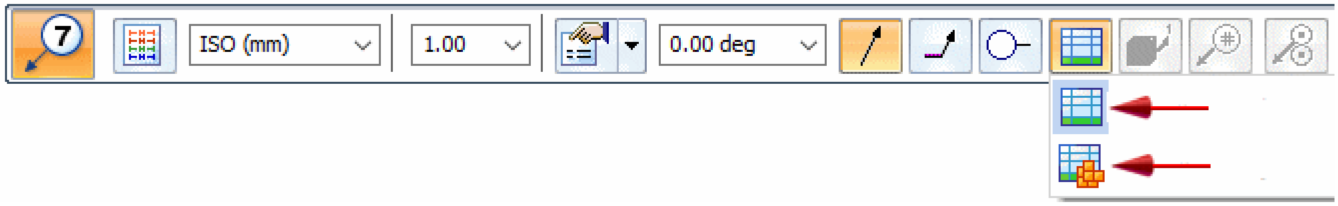
For more information, see Convergent Modeling.

Parts list for a family of assemblies (FOA)

A new command, **Family of Assemblies Parts List**, is available on the **Home** tab→**Tables** group to create a parts list for all or selected members of a family of assemblies. For more information, see Create a family of assemblies parts list.

In addition, you can:

- Add item balloons to drawing views of FOA members, and link the item numbers to an existing FOA parts list. Select the **Balloon** command and the new **Link to Family of Assemblies Parts List** option on the command bar.



- Create saved settings for your FOA parts list using the options in the **Parts List Saved Settings** section on the **Drawing View Wizard** tab (**Solid Edge Options** dialog box).
- Specify that when a drawing view of a family of assemblies member is updated, its auto-balloons are also updated based on the FOA parts list. On the **General** tab (**Drawing View Properties** dialog box), select the **Auto-Balloon on next update** check box and the new option, **Family of Assemblies Parts List**.

Peer Edge Locate command on by default

The **Peer Edge Locate** command is now on by default. The command setting (On/Off) is now a user preference and as such remains set as you specify across Solid Edge sessions.

Previously, the command setting would remain set during a Solid Edge session and would reset to off when Solid Edge was restarted.

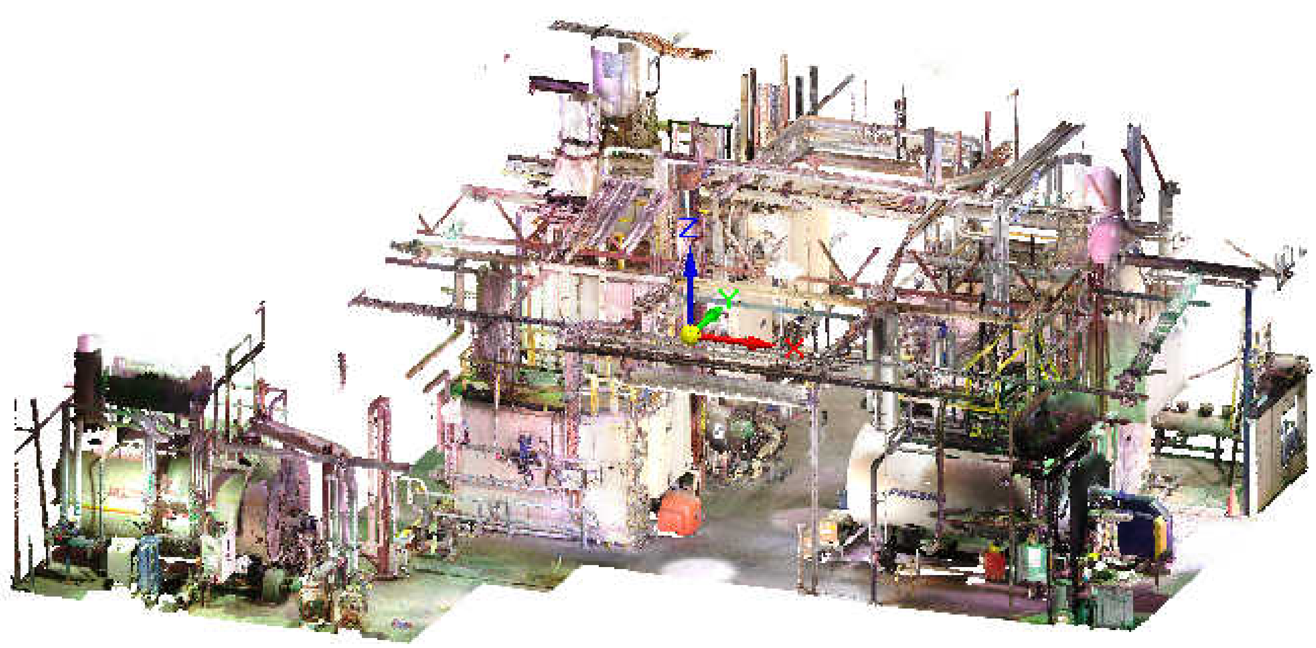
Point clouds

You can now insert scanned point cloud files into an assembly for the purposes of viewing, designing, and measuring the fit of industrial machinery within the context of a facility or plant.

Solid Edge also provides display management functions and tools to reposition and manipulate a point cloud, change the color and density of points, and take approximate measurements between points in the point cloud and model geometry.

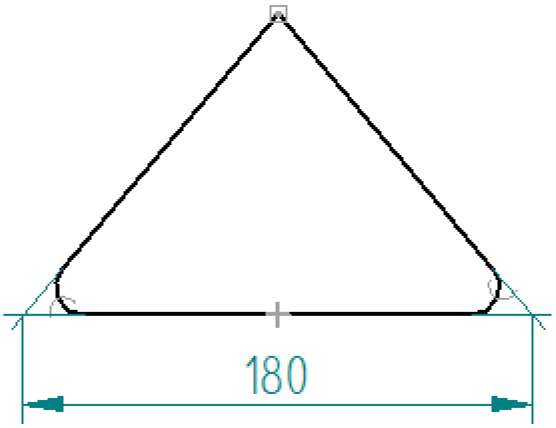
The new **Tools** tab→**Assistants** group→**Reference Point Cloud** command  and related capabilities are available in the Assembly, XpresRoute, Frame, and Electrical Routing environments, with a Solid Edge Premium license and an active maintenance or subscription agreement.

For more information, see Using point clouds.



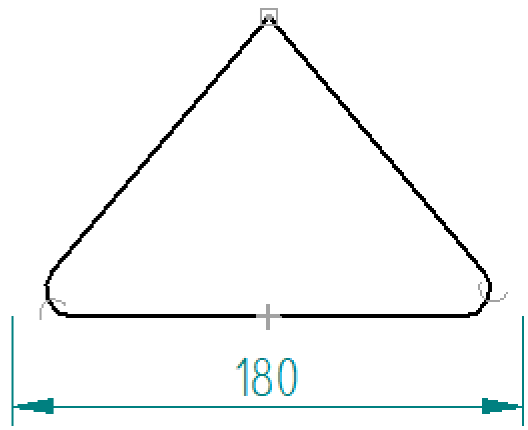
Projection lines to virtual intersection points

When you place a linear dimension to the implied intersection point of 2D geometry, you now can specify that the projection lines for the two parent edges extend to their virtual intersection point.



A new option on the **Lines and Coordinate** tab—**Projection lines to virtual intersection points**—is available in the dimension properties and dimension style for you to select the projection lines to display.

Previously, the projection lines did not extend to show the virtual intersection points. This display option is still available as the default option (**None**):

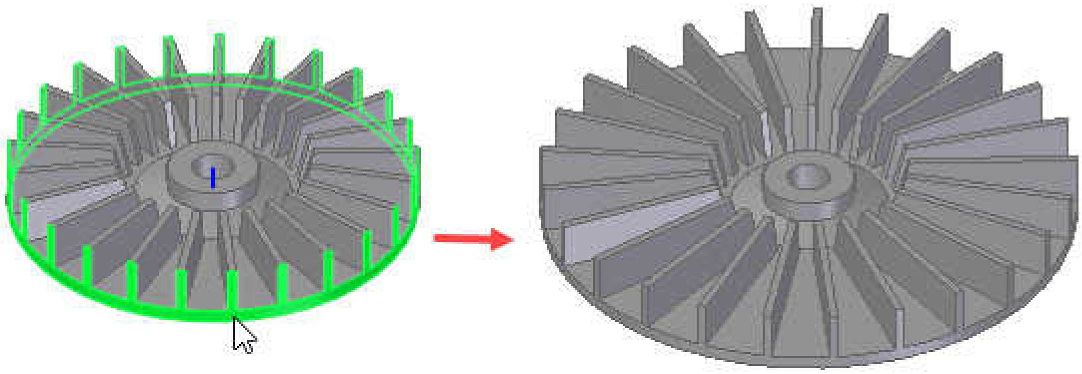


For more information, see:

- Place a dimension to a virtual intersection point
- **Lines and Coordinate** tab (**Dimension Style** and **Dimension Properties**)

Radiate command

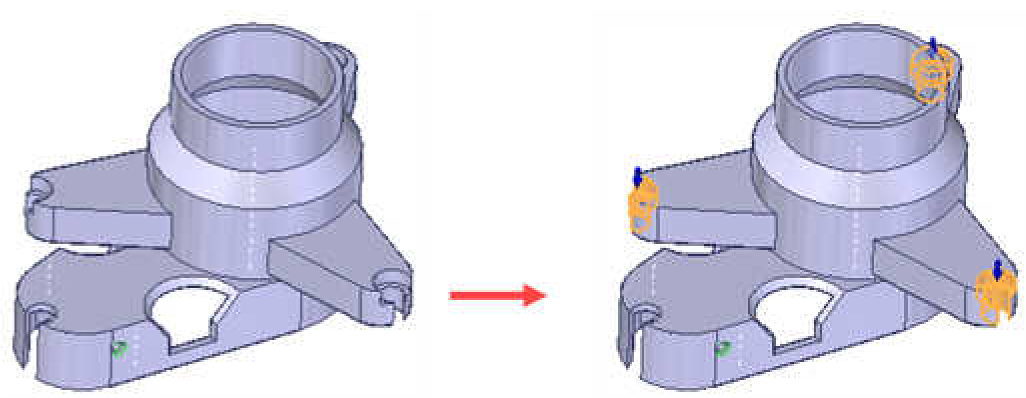
Use the **Radiate** command in the synchronous environment to edit the radius of selected coaxial, cylindrical, conical, toroidal, and spherical faces.



For more information, see Radiate command.

Recognize Holes command supports partial circular cutouts

The **Recognize Holes** command now identifies partial circular cutouts as hole features.



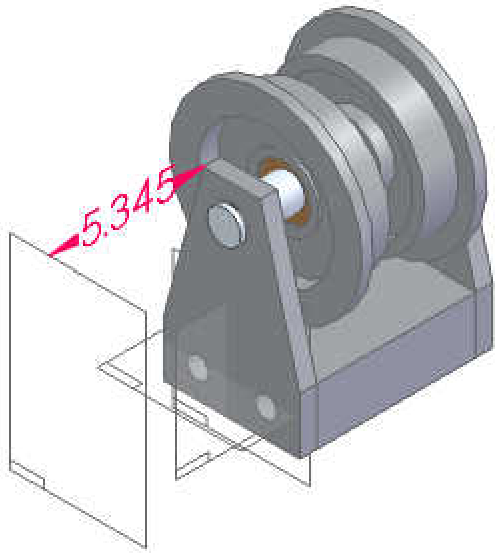
Previously, only complete circular cutouts were identified as hole features.

Reference plane enhancements

Reference plane unification enhancements between part and assembly include:

- The **Tangent Plane** command is now available in the assembly environment. Now the list of commands used to create reference planes is the same for both part and assembly.

- In assembly, placing a reference plane now places a dimension so you can use the **Edit Definition** command to edit the dimension without going into **Variable Table**.



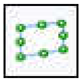
- The plane names in assembly **PathFinder** are now the same as they are in part. The name is a generic name, rather than a name based on the plane type. Previously, in assembly **PathFinder** the plane name reflected the plane type. For example, suppose you create a parallel reference plane in assembly. In **PathFinder**, the name for that reference plane would be **Parallel x**.

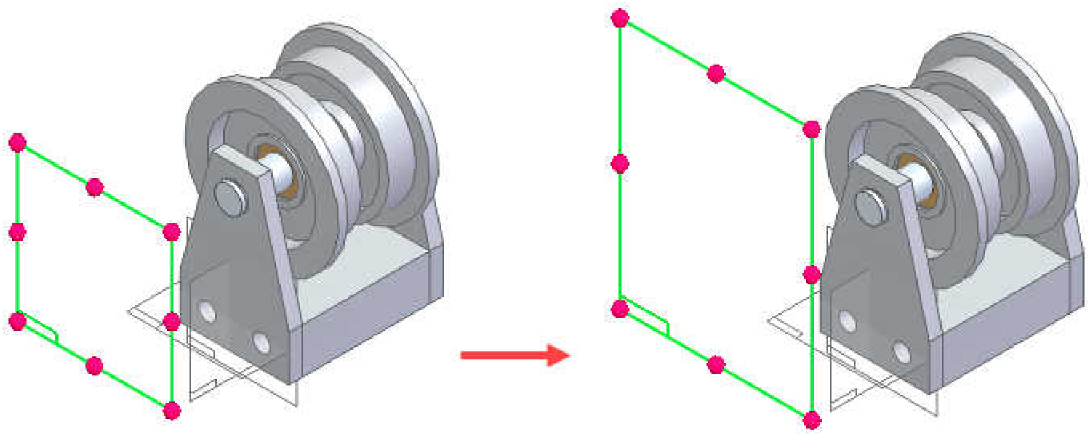
> Note:
>
> The names of existing assembly reference planes do not change.

- The reference plane icons for assembly are now used in **PathFinder** in both part and assembly to indicate the reference plane type.

> Note:
>
> Synchronous reference planes will use the basic reference plane icon.

- The P hot key is now available when creating a parallel or coincident plane in assembly. Use the P hot key to select a base reference plane to define the x-axis.

- A new **Resize Plane** command  on the part reference plane shortcut menu in part **PathFinder** changes the size of the reference planes.



- Editing reference planes is now consistent between part and assembly. The **Edit Definition** command and **Dynamic Edit** command are available when editing reference planes in part or assembly.

- The reference plane edit handle now displays relative to the selected plane for both part and assembly.

- Updates have been made to the commands found in the reference plane menus in ordered part, synchronous part, and assembly.

- The reference plane collector from the part reference plane **Edit** command has been added to the assembly reference plane **Edit** command so you can now change the assembly reference plane type.

- The reference plane icons in drawing views have been updated to be consistent with the icons for part and assembly.

- Reference planes placed in assembly now use driving dimensions to position the reference plane, as is done in part. Previously, a variable value drove the position of the reference plane in assembly.

- The icon used in drawing view properties now use the base reference plane icon used in part and assembly.

- Changes have been made to the display of offset values for the **Angled Plane**, **Parallel Plane**, and **Tangent Plane** commands.

  - The entry in the relationship tree now indicates that a keypoint is used to define the position of the reference plane. Previously, the entry displayed the value for the reference plane position.

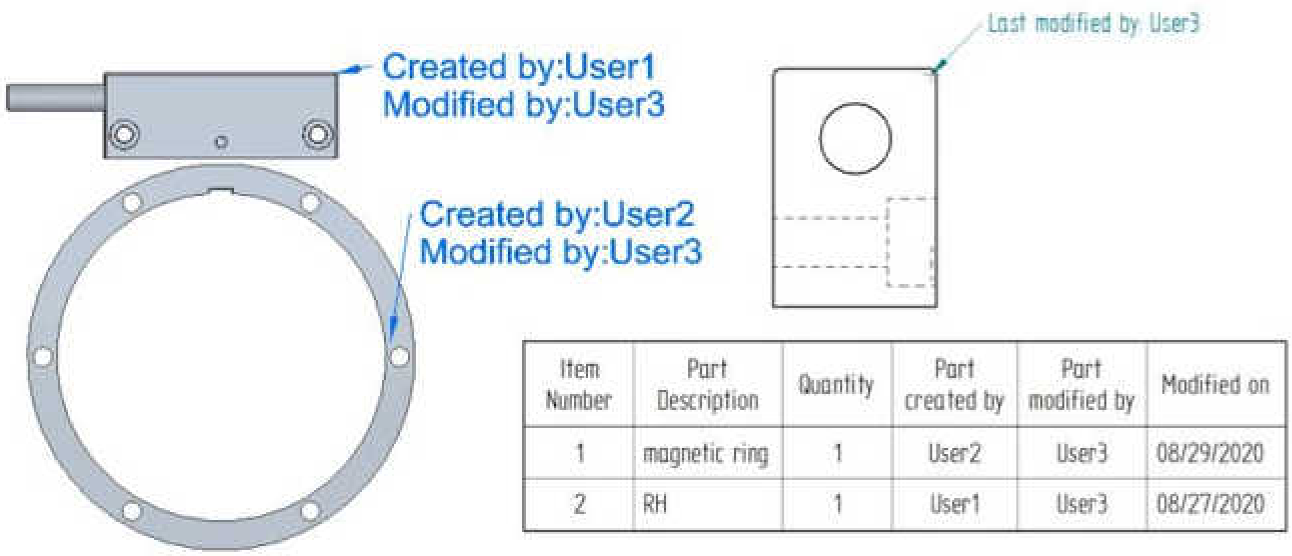
  - Clicking the entry in the relationship tree highlights the keypoint that was used to define the position.

  - The value of this entry is no longer displayed.

  - The box used to change the value is no longer displayed.

Reference user profile information in a callout

Property text can now reference three fields on the **User Profile** page of the **Solid Edge Options** dialog box: **Name**, **Initials**, and **Mailing Address**. You can extract this information into a callout or other annotation to identify which user created and which user modified the file it references. You can use this feature in draft or model files to show user profile information from the active document and from assembly components (parts or subassemblies).



Item Number	Part Description	Quantity	Part created by	Part modified by	Modified on
1	magnetic ring	1	User2	User3	08/29/2020
2	RH	1	User1	User3	08/27/2020

Previously, only the **Author** and **Last Author** property were available, which display the computer login name.

You can access the six new properties in the **Select Property Text** dialog box, in the alphabetized **Properties** list. Look for the entries that begin with the label, **User Profile**.

For descriptions of these properties, see Property Text source list (Source: From Active Document).

Retain frame orientation and origin

Frames being replaced with a new member or frames being added to an existing group retain the orientation and handle points of the existing group members.

If you add a frame between two frames that are not oriented in the same angle and direction, the frame is added with zero orientation.

A new **Orientation** option on the **Frames Options** dialog box specifies the default orientation for the frames.

Retrieve dimensions in a detail view

The **Home** tab→**Dimension** group→**Retrieve Dimensions** command is now available to use in a detail view in draft. Previously, this command could only be used in views other than derived and isometric views.

For more information, see Retrieve dimensions and annotations from the model.

Search 3Dfind.it using a 2D sketch

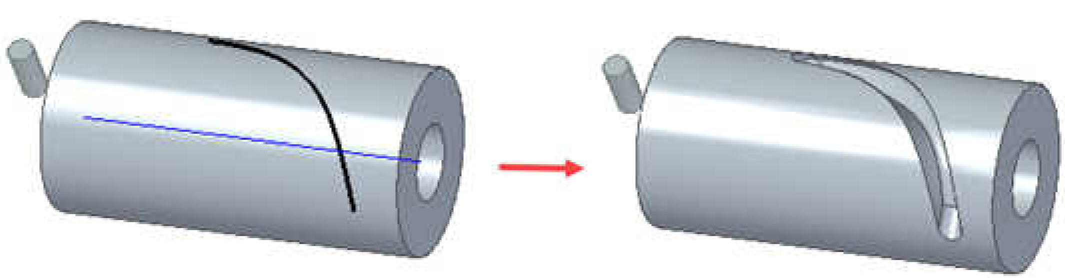
To save you time in searching for, reconfiguring, or recreating 3D models, you can now search the 3Dfind.it supplier part library based on a 2D sketch.

For more information, see Search by sketch with 3Dfind.it.

Solid Sweep Cutout command enhanced

Use the **Place Tool Normal to Surface** option on the **Solid Sweep Cutout** command bar to place the sweep tool normal to a surface on the circular or round body while creating a swept cutout.

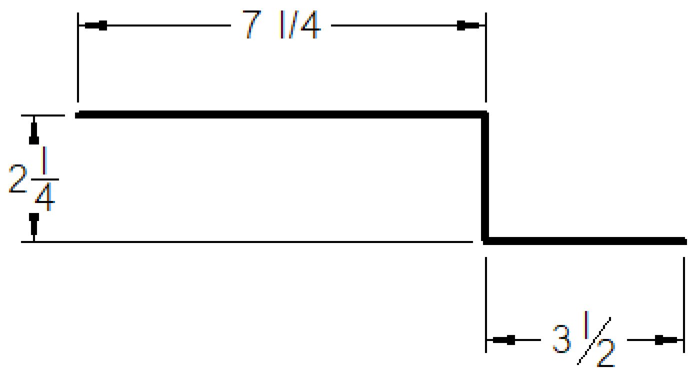
With this option selected, the sweep tool follows the path correctly on the circular body, which results in a precise swept cutout.



> Note:
>
> You should enable this option while working with circular or cylindrical models.

Stacked and skewed fractional dimensions

In a document using the **ANSI (inch)** template, you now can display fractional dimensions in stacked format and in skewed format. Previously, only the linear format was available.



The new formats make it easier to read a nominal value and fraction than when written in linear form. For example, 1 1/16 can be misread as 11/16.

The following fraction formatting options are available on the **Units** tab **Dimension Properties** dialog box and on the **Secondary Units** tab, under **Fractional Display Style**:

- **Stacked**
- **Skewed**
- **Linear**
- **Fraction Size**

For more information, see Format a fractional dimension.

Standard Parts enhanced

Many enhancements were made to Standard Parts.

- Name changes to the **Standard Parts** applications include:
  - The **Standard Parts** application that is launched from within Solid Edge is now named **Part Finder**.
  - The **Configuration Wizard** is now named **Parts Management Administrator**. Use the **Parts Management Administrator** to manually specify the location of the standard parts configuration folder before using the standard parts.
  - The **Database Administrator** application name is unchanged.
- All **Standard Parts** application dialog boxes, such as **Database Administrator**, **Parts Management Administrator**, and dialogs launched from Solid Edge frame, piping, and fastener systems now have command ribbons. Previously these dialog boxes had menus and command toolbars.

- The layout of the **Parts Management Administrator** has changed, but the options are the same.

- The .SAC file has been replace with a .PMSX file. The **Parts Management Administrator** is still able to read the older .SAC file, but will migrate the information to the .PMSX file.

- The **Add Parts** command workflow has been enhanced:

  - Previously, there was one **Add Parts** command that opened the **Add Standard Parts** dialog box, which contained options to add parts from the installation database or from a family of parts. Those options are now separate commands.

  - The **Standard Parts Assistant** combines the methods for adding user parts, such as adding parts from an Excel worksheet, from manual data entry, and so on.

  - In the new **Add Parts** command, you can now select multiple parts to add. Previously, you could only select folders.

  - The **Upgrade** command and **Add Parts from Libraries** command replace the option to add parts from the installation database using the Smart Installer.

  - The **Upgrade** command upgrades the existing database or libraries to a newer version.

    > Note:
    >
    > This command is only available when a machinery or piping library is installed.

    The command is also available when you right-click a category, so you can upgrade only the selected category.

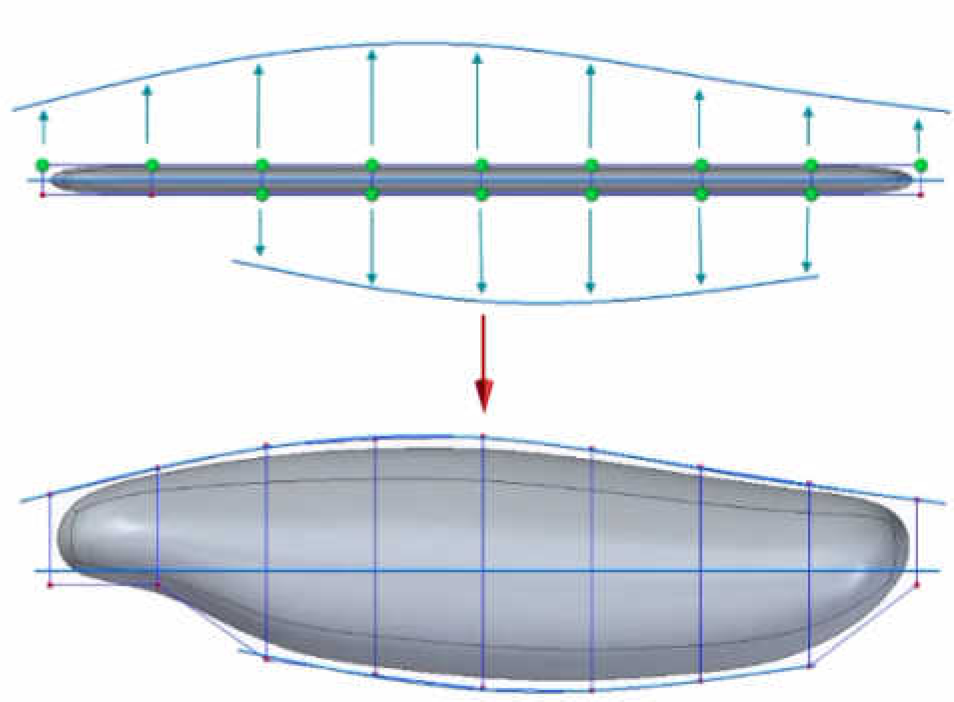
  - The **Add Parts from Libraries** command can add all or only selected categories from the installation database.

- The **Copy to working folder** option in Part Finder now remains selected or cleared, based on the last state of the dialog box. Previously, after exiting Solid Edge or Part Finder, you had to reselect the option the next time you opened the dialog box.
  The **Copy to working folder** option and the **Use assembly folder as working folder** option are now available on the **Parts Management Administrator** dialog box, so an administrator can control the options.

- The Standard Parts library setups have been combined with the Quick setup. Previously, there were separate setups for machinery and piping, which updated the existing libraries, and one Quick setup, which installed a new database. Now there is a single setup that adds parts to the existing database or to an empty database.

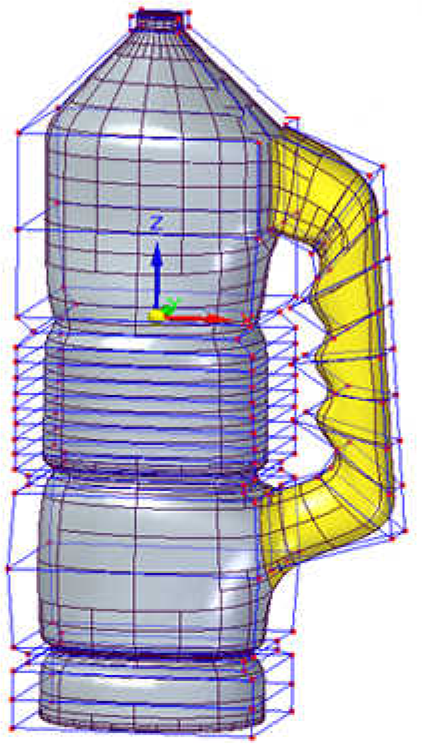
Subdivision Modeling: Align to Curve command

In the Subdivision Modeling environment, you can use the new **Align to Curve** command to fit the vertices of body cage faces to one or more existing curves or to curves you interactively sketch. You can undo and redo each curve edit until you achieve the desired shape.



Subdivision Modeling: Bridge command

In the Subdivision Modeling environment, you can use the new **Bridge** command to create a loft-like feature that connects edges or faces selected on a single cage or two separate cages.



Subdivision Modeling: Laminar edge support

Subdivision Modeling now supports the selection of faces that contain laminar (open) edges and laminar edges with the **Move** or **Rotate** command and the **Lift** option. This capability improves the general functionality of open cages.

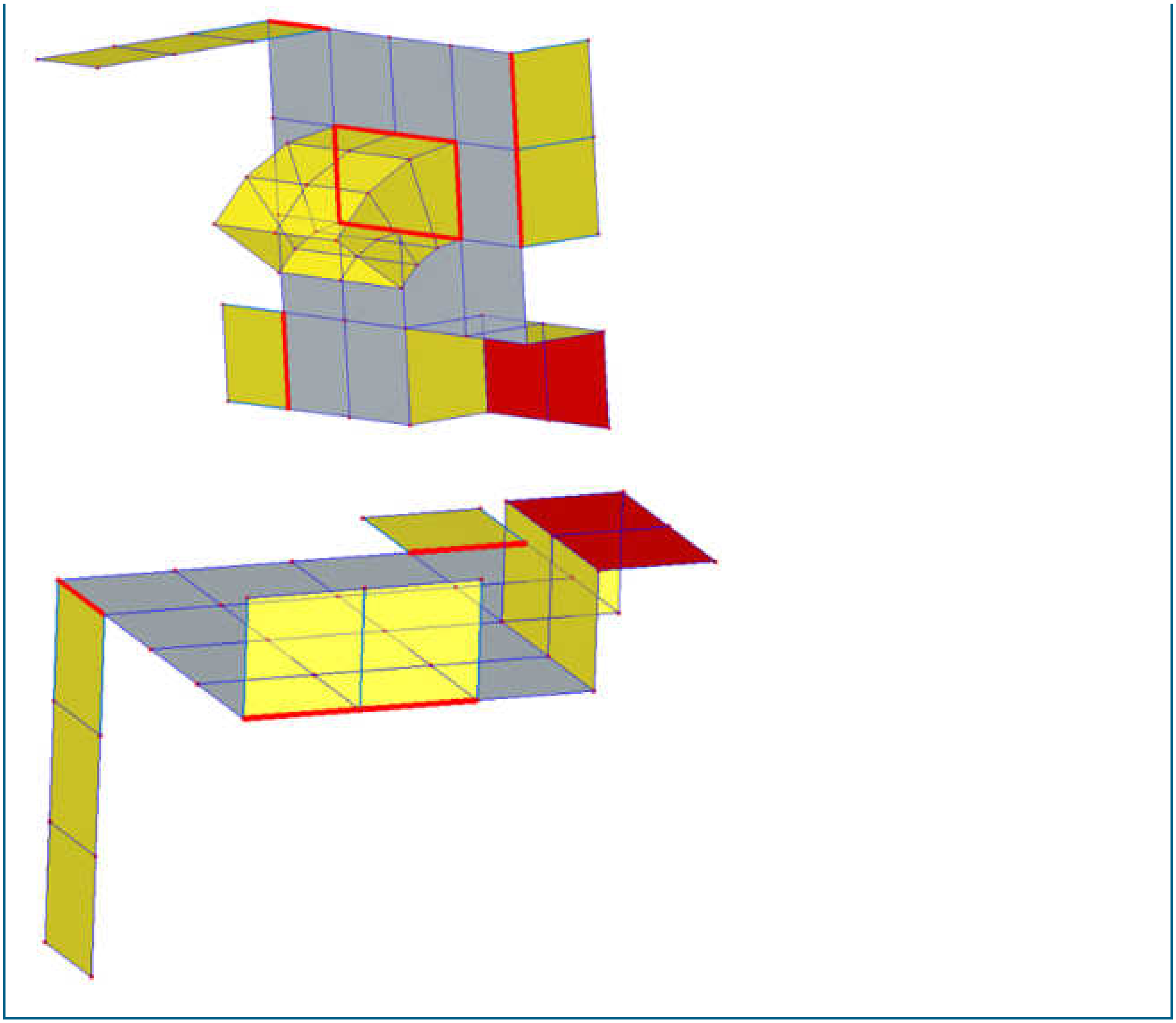
Example:

Red=Laminar faces or edges selected

Yellow=New faces created from laminar edges

When the RED FACES are lifted, no new faces are created where laminar edges exist.

When the RED LAMINAR EDGES are lifted, new faces are created.

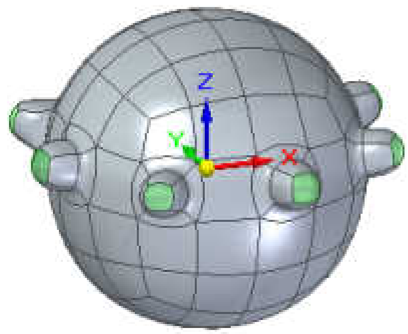


For more information, see Moving and rotating cage elements.

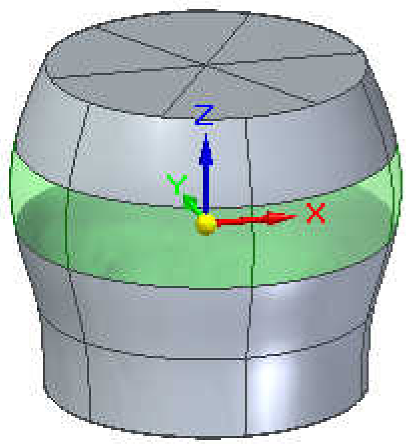
Subdivision Modeling: Offset command

In the Subdivision Modeling environment, you can use the new **Home** tab→**Modify** group→**Offset** command to move or to lift multiple faces by a specified offset value.

To add faces that are not coplanar to the selected faces on the cage body, use the **Lift** option.

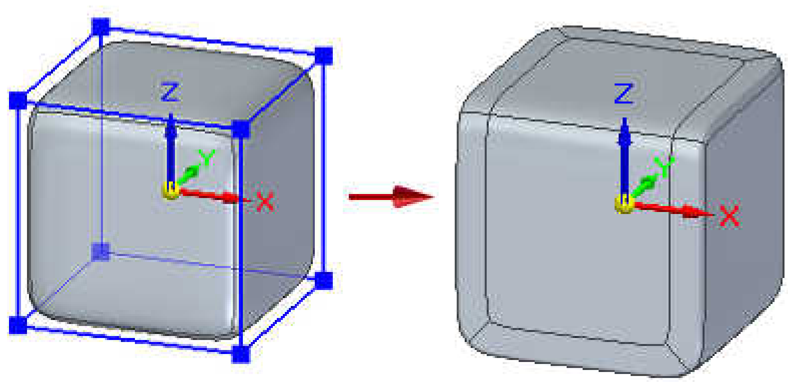


To extend the faces without creating new ones, use the **Tip** option.



Subdivision Modeling: Split with Offset command

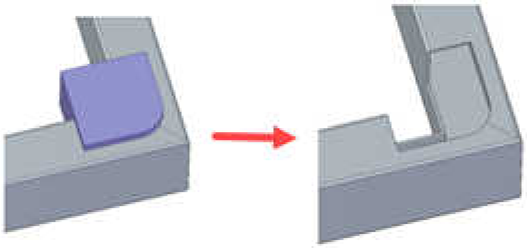
In the Subdivision Modeling environment, you can use the new **Split with Offset** command to add detail to a face by offsetting the new faces inward by a user-defined amount.



This command differs from the **Split** command, which splits the selected cage face into equal parts.

Subtract command in assembly enhanced

The **Subtract** command in the assembly environment has been enhanced to subtract assembly objects from frames, pipes, gusset plates, and internal components created in Solid Edge 2022 and subsequent versions.



Note:

The **Create Assembly Feature** option for the command is disabled for files containing weldment features created in pre-Solid Edge 2022 versions (legacy files).

To enable the option for pre-Solid Edge 2022 files:

1. Delete all weldment parts and features, such as frames, pipes, and assembly features from the active assembly.

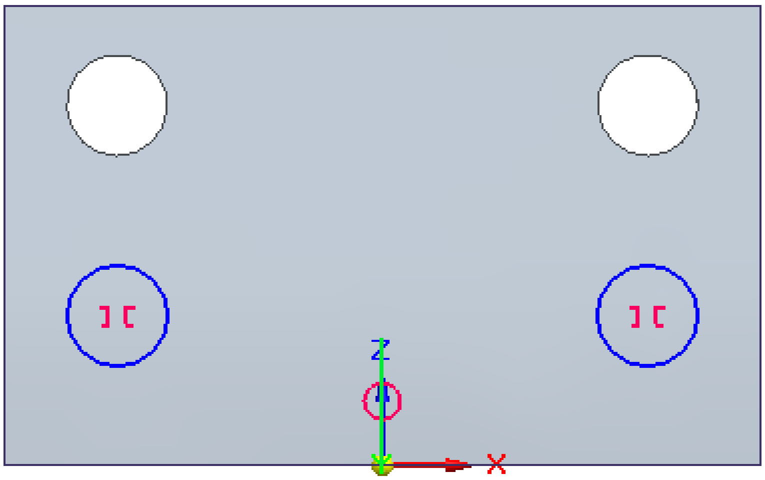
2. Save the file.

3. Recreate the deleted features.

Symmetric relationship for hole sketch profiles

The **Home** tab→**Relate** group→**Symmetric** command  is available in ordered profile sketches.

Now when you select the **Hole** command, you can add a symmetric relationship between two hole circles in the ordered sketch. This enables you to create symmetrically aligned holes within the same Hole feature.

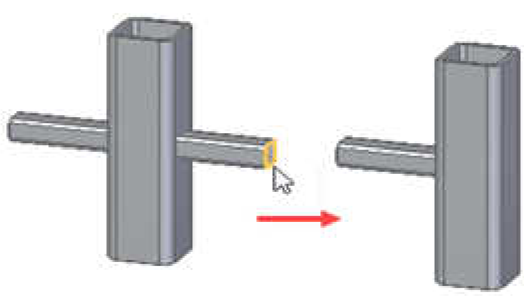


For more information, see:

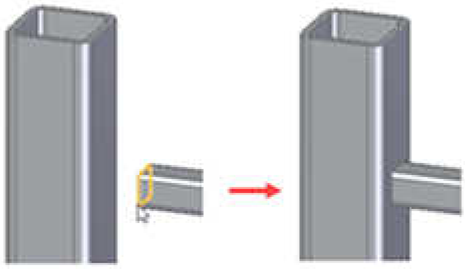
- **Hole Circle** command
- Construct a hole (ordered)

Trim/Extend command in Frames

Use the **Trim/Extend** command  to trim



or extend a frame



to a selected face, plane, or body, regardless of the order in which the frames were created.

> Note:
>
> This command is disabled for files containing weldment features created in pre-Solid Edge 2022 versions (legacy files).
>
> To enable the command for pre-Solid Edge 2022 files:
>
> 1. Delete all weldment parts and features, such as frames, pipes, and assembly features from the active assembly.
> 2. Save the file.
> 3. Recreate the deleted features.

YouTube command enhanced

The **View** tab→**Panes** group→**YouTube** command has been enhanced to launch a new web page where you can enter your Google credentials. Google no longer supports the launching of embedded framework from any applications.

The **YouTube** command is also available on the status bar at bottom-right in the application window:

See YouTube videos in Solid Edge for information about recording, uploading, and searching for YouTube videos.

Watermark command in Draft

The **Home** tab→**Annotation** group→**Text**→**Watermark**  command is now available in the Draft environment.

You can change the availability of the **Watermark** command by selecting the **Show watermarks in Draft environment** check box on the **Solid Edge Options** dialog box→**View** tab. This setting also changes the display of watermarks in the document.

While printing draft documents, watermarks can be hidden by deselecting the **Include watermarks on print** check box on the print option dialog.

Previously, you had to edit the background and add a text box to create a watermark. These text boxes could not be hidden when editing or printing.

> Note:
>
> You can convert a text box created in a previous release to the new watermark behavior using the **Set text box as watermark** option in the **Text Box Properties** dialog box.

For more information, see Using watermarks

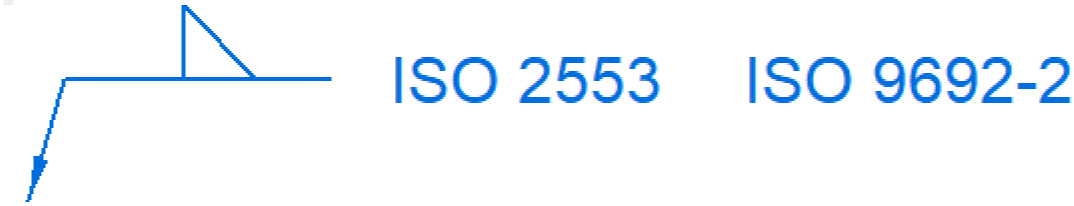
Weld symbol enhancements

The following enhancements were made for weld symbols in draft, part, sheet metal, and assembly.

- Ten weld symbols with symbol treatment options were added for the ISO, DIN, and AWS standards. The newly added symbols are:
  - Double bevel butt (with broad root face) and fillet welds (ISO 2553) and Alternate Representation
  - Double bevel butt (with broad root face) and fillet welds (DIN EN15085-3)
  - Single-U butt weld with V root (ISO 9692-2)
  - Permanent/removable Backing symbol (AWS 2.4)
  - Groove Symbol (DIN standard)
  - Consumable Insert (ISO 2553)
  - Resistance Seam (ISO 2553)
  - Resistance Spot (ISO 2553)
  - Compound Weld Symbol (AWS 2.4)
- To make them easier to identify at a glance, weld symbols are now arranged in a more horizontal than vertical layout on the **Weld Symbol** palette. Tooltips for each symbol identify the name, standard, and any distinguishing characteristics.

- You now can place a tail note on a weld symbol even if no tail symbol is placed. Use the **Tail Note1** or **Tail Note 2** boxes in the **Weld Symbol Properties** dialog box to add a tail note text. To omit the symbol, click the **None** option on the **Tail Symbol** list.

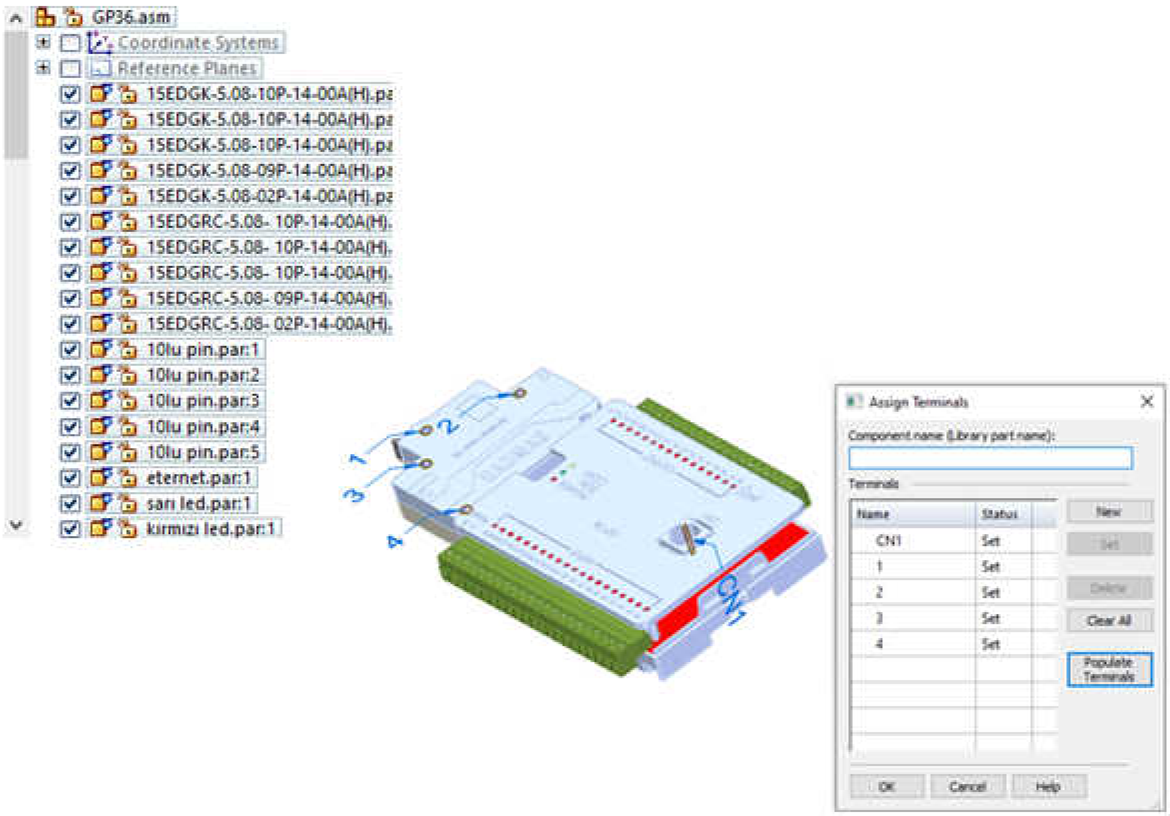
- The **General** tab (**Weld Symbol Properties** dialog box) now has a **Preview** pane to show the weld symbol as you create it.



5. Solid Edge Electrical Routing enhancements

Assign Terminals command now available in Electrical Routing

The **Assign Terminals** command is now available in Solid Edge Electrical Routing.



Previously, you could only assign terminals in part files. Now you can assign terminals to assembly components that may contain a number of parts or subassemblies, such as relays and circuit breakers. In this way any assembled component can be converted into electro-mechanical component by assigning terminal information.

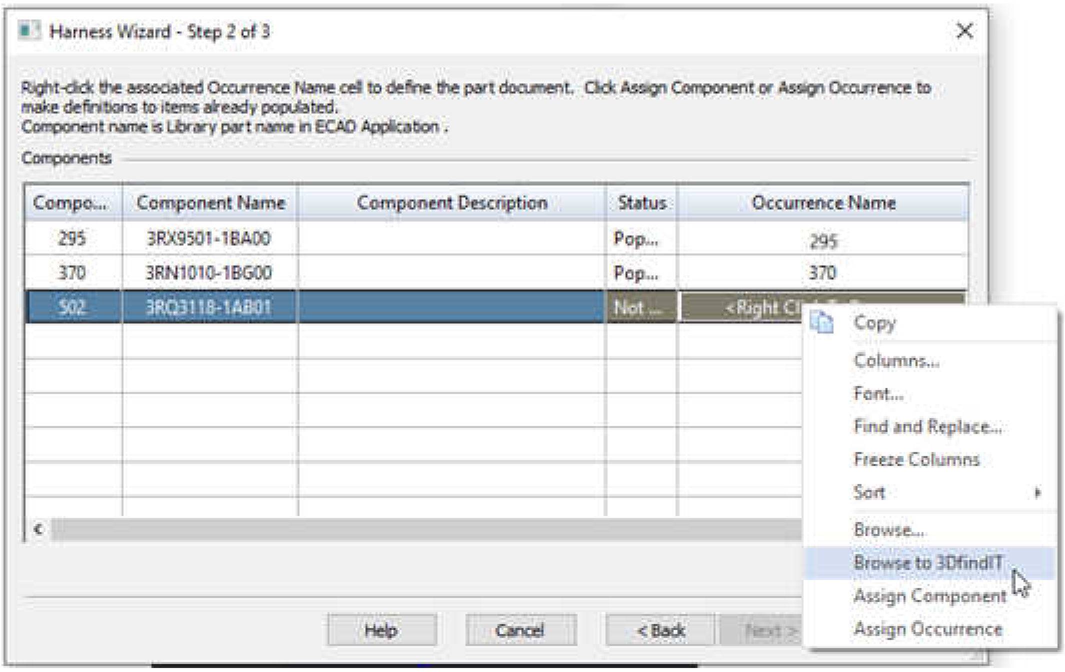
Browse to 3DfindIT supported in Harness Wizard

The **Browse to 3DfindIT** command is now available in the **Harness Wizard** so you can search for a component in the 3Dfindit portal, retrieve the component, and populate it in the assembly file in a single operation.

Currently in the harness wizard, if the component name in the .cmp files matches, the component name assigned in the part file, the part is populated in assembly.

If the 3D part is not available, you can download the part from a manufacturer's website, and then import the .cmp and .con files.

To use the **Browse to 3DfindIT** command, on the **Harness Wizard - Step 2 of 3** dialog box, right click the component that is not present in the assembly and click **Browse to 3DfindIT** on the shortcut menu.



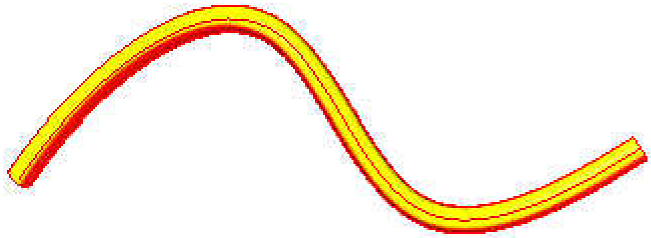
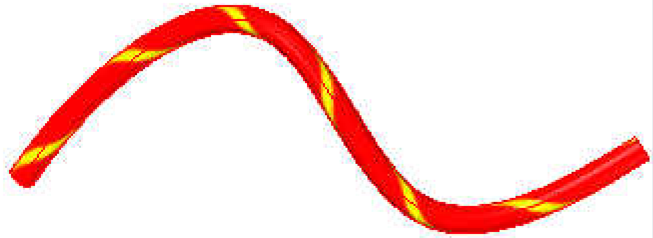
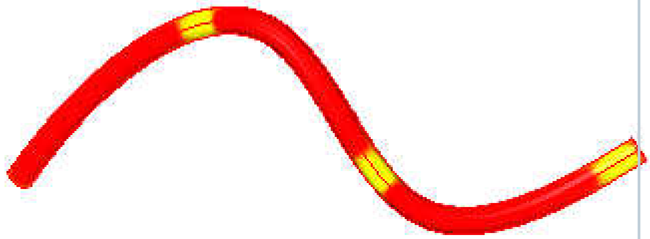
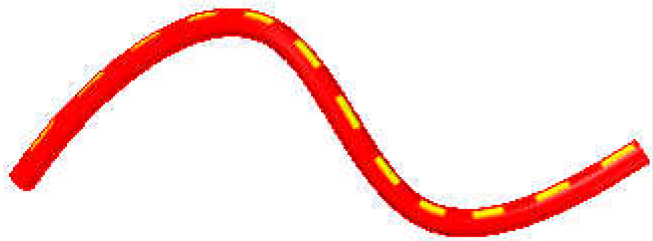
After you click the command, the search for the component name begins in the 3Dfind.it portal. Any matching components displayed. You can then click **Send to Solid Edge 2022** to directly populate the component into the assembly. This is useful if you want to download and save 3D CAD files directly from the portal, while importing ECAD file formats.

Display of bi-color wires

Solid Edge now supports the display of bi-color wires in Solid Edge Electrical Routing. Previously, wires were represented by only one color with no way to apply a texture to them. Bi-color wires are very prevalent in industrial machinery. For example, grounded wires are represented by a yellow wire with a strip of green.

Several standard bi-color wires have been added to the list of materials in the **Wire Properties** dialog box. When you select a bi-color wire, the colors are assigned to the **Primary Color** and **Secondary Color** fields on the **Wire Properties** dialog box.

Use the **Pattern Type** field on the dialog box to specify the pattern for the colors on the wire. Solid Edge supports the following pattern types:

Pattern type	
Single Strip	
Helical	
Zebra	
Dash	

For more information, see Displaying bi-color wires.

Harness BOM reports

You can now use the Harness Report dialog box to export a BOM report for a harness assembly in the Electrical Routing environment. You can save the report to .csv format.

The Harness BOM is a consolidated report consisting of all electrical components and connections in the harness assembly.

For more information, see Harness Reports dialog box.

Import and export of X2ML files supported

Solid Edge Electrical Routing now supports the import and export of X2ML files.

To import X2ML files, on the **Harness Wizard- Step 1 of 3** dialog box, set the **ECAD Application** option to **Siemens-Solid Edge Wiring Design** to display the **Wiring X2ML Document** option where you can browse the X2ML document exported from **Siemens-Solid Edge Wiring Design** in disconnected mode.

After importing the X2ML file, the components and connections are populated in the assembly and the harness assembly is created.

To export X2ML files, on the **Application** menu, point to **Save As**, then choose **Save As ECAD**. On the **Save As ECAD** dialog box, set the **ECAD Application** option to **Siemens-Solid Edge Wiring Design** to display the **Wiring X2ML Document** option where you can export the X2ML document for back annotation of wire lengths to be updated into schematics of **Siemens-Solid Edge Wiring Design**.

A new **Supplier Part Number** attribute is added from X2ML. Supplier part number details are available in the **Assign Terminals** command for connectors and devices. For wires, cables, bundles it is available in their respective properties window.

In the **Harness Wizard** dialog box, you can use supplier part number as search criteria to directly browse to 3DfindIT portal and download a 3D component on the go.

Mapping component ids with PathFinder

Component IDs or instance names are unique identifiers in 2D applications, such as Solid Edge Wiring Design. Previously, when this wiring data was imported into Solid Edge, there was no way to display these component ids in Solid Edge **PathFinder**.

You can now use the **Update with Electrical Component IDs** option to map the electrical component id to the name in **PathFinder**. When you select **Update with Electrical Component IDs**, you have the option to replace or append the component id in Solid Edge **PathFinder**.

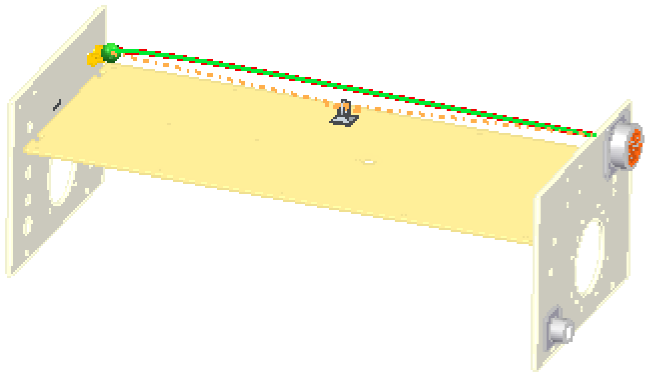
Also, with the addition of the **Update with Electrical Component IDs** option, you no longer have to manually assign the occurrence for duplicate component ids.

For more information, see Mapping electrical id components in Solid Edge.

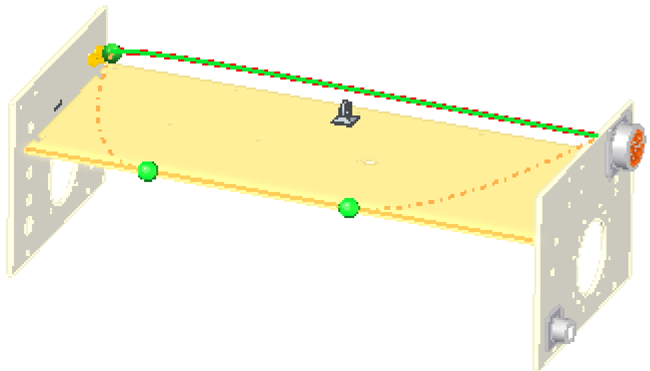
Route along Surface command enhanced

Enhancements to the **Route along Surface** command include:

- When placing points for routing in the command, you can now select cylindrical faces to route an harness entity through any cylindrical opening in an assembly.



You can also route along the edges of the selected surface.



Previously, you could only select faces.

- You can now route physical conductors along a surface.

For more information, see Route along Surface command.

Support favorites list for CMP/CON import

Solid Edge Electrical Routing now supports a favorites list of components when importing CMP/CON files in unmanaged and Teamcenter-managed mode.

You can specify the location of *SE-PreferredElectricalComponents.csv* which contains the list of favorite component names. By default, this location is *$:\Program Files\Siemens\Solid Edge 2022 \Preferences \SE-PreferredElectricalComponents.csv*, but you can modify the path. When CMP/CON files are imported, the CMP file is directed to the preferences file first. If the component name in the CMP file matches the component name in the favorite list the relevant path is displayed in the harness wizard. To populate the part in the active assembly, click **Populate** in the harness wizard.

After the harness wizard operation completes, the component name and path for components which are manually browsed are written back into the csv file.

Note:

If the component is not found in the favorites list, Solid Edge searches the electrical parts folder. If the component is not found there, you must browse manually for the component.

6. Modular Plant Design enhancements

Modular Plant Design uses unified libraries during setup

The delivery of Solid Edge Modular Plant Design 2022 uses the new unified standard parts library in the setup.

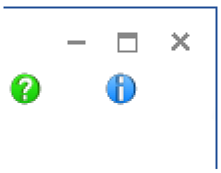
For more information, see *What's New in Solid Edge Modular Plant Design 2022*, located on the **Documents tab in Support Center for 2022**.

Smap3D Piping integration with Solid Edge

The Modular Plant Design interface is now fully integrated with the Solid Edge application window. In the Solid Edge interface:

- A new tab on the Solid Edge command ribbon--**Smap3D Piping**—contains the Solid Edge Modular Plant Design commands.
- The **Piping** window and the **P&ID Design To-Do List** window are now embedded in the Solid Edge interface as individual docking panes. You can show and hide these tool panes using the **View** tab->**Show** group->**Panes** command.

For information on how to use **Solid Edge Modular Plant Design**, click the Help button



in the upper right corner of the **Solid Edge Modular Plant Design** application window.

7. Simulation enhancements

Connector enhancements for frame models

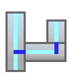
Previously when you created a simulation study of a structural frame model, rigid link connectors between frame members were only generated automatically. Automatic link generation did not consider all cases where rigid links were needed, such as for disjoint beams. Sometimes rigid links were generated where they were not needed, but the connectors were not editable.

Solid Edge Simulation now provides several enhancements for creating and modifying rigid link connectors in frame simulation models:

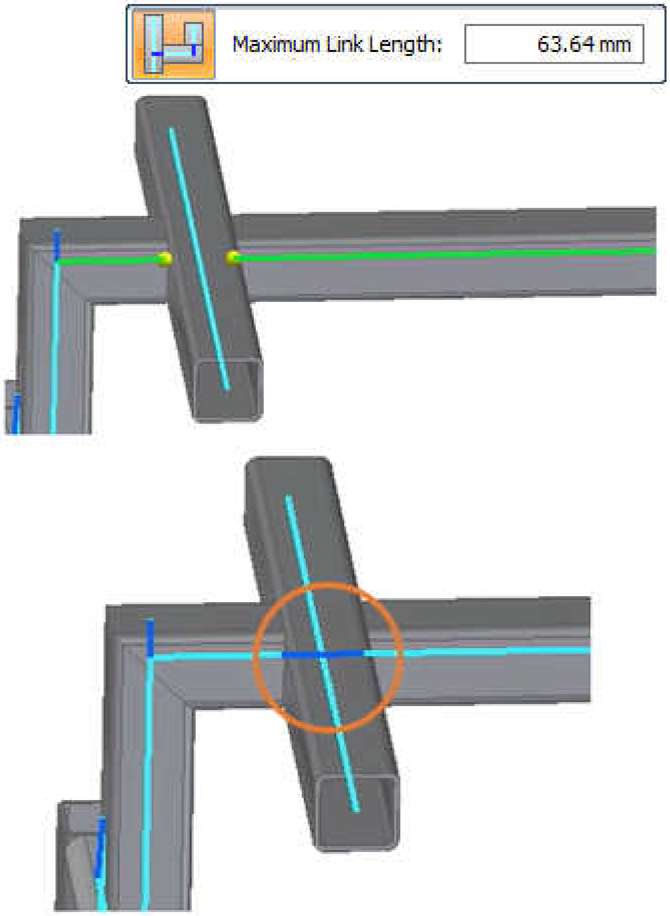
- When creating a simulation study for a frame model, the **Connector Options** section of the **Create Study** dialog box contains a new option to specify a **Maximum rigid link length**. This value is used to generate rigid links between beam curves automatically.

- You can automatically create rigid link connectors on the entire frame model using the improved **Simulation** tab→**Rigid Links** group→**Auto** command .

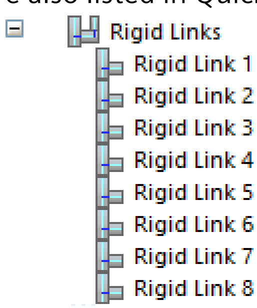
> Note:
>
> When you open a legacy beam study of a frame model, notice the rigid links that were created previously with the **Auto** command are displayed in the **Simulation** pane. To take advantage of the enhanced rigid link creation algorithm, we recommend you either create a new study using the automatic link creation method, or delete the legacy rigid links in your existing study and recreate them using the **Auto** command.

- You can manually create rigid links between two beam curves using the new **Simulation** tab→**Rigid Links** group→**Manual** command .

Example:



- A new **Rigid Links** node is now available in the **Simulation** study navigator pane. Individual rigid links are also listed in QuickPick.



- You can highlight individual rigid links on the model, and use shortcut commands to **Delete** or **Rename** rigid links.

- For more information, see:

  - Rigid link connectors

- Define rigid links manually

Hydrostatic Pressure load

In Solid Edge Simulation, you can use the new **Simulation** tab→**Structural Loads** group→**Hydrostatic Pressure** command to define nonuniform conditions for fluid pressure that varies with depth.

You can apply the **Hydrostatic Pressure** command in the following types of studies:

- **Linear Static**
- **Linear Buckling**
- **Steady State Heat Transfer + Linear Static**
- **Steady State Heat Transfer + Linear Buckling**
- **Harmonic Response**

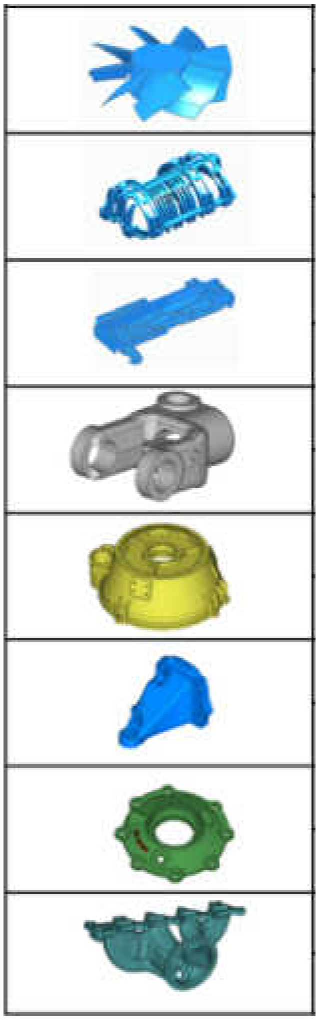
For more information, see Apply a hydrostatic pressure load.

> Note:
>
> Use the **Pressure** command to define a loading condition of constant pressure.

Simplified meshing for complex parts

Use the new option in the **Tetrahedral Mesh** dialog box—**Body Mesh**—to simplify and improve the mesh for complex parts with intricate bodies. In addition to producing a good quality mesh, it provides automatic geometry cleanup of small faces and edges.

Previously, you needed to use the **Geometry Inspector** command to improve the geometry prior to meshing. Refer to the table for examples of the types of parts that you can now mesh successfully by selecting the **Body Mesh** check box.

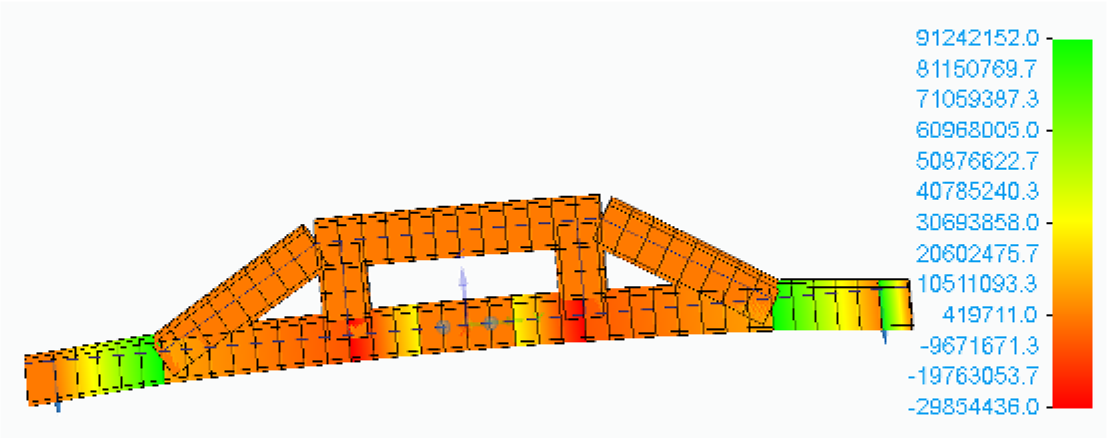


For more information, see the Mesh dialog box.

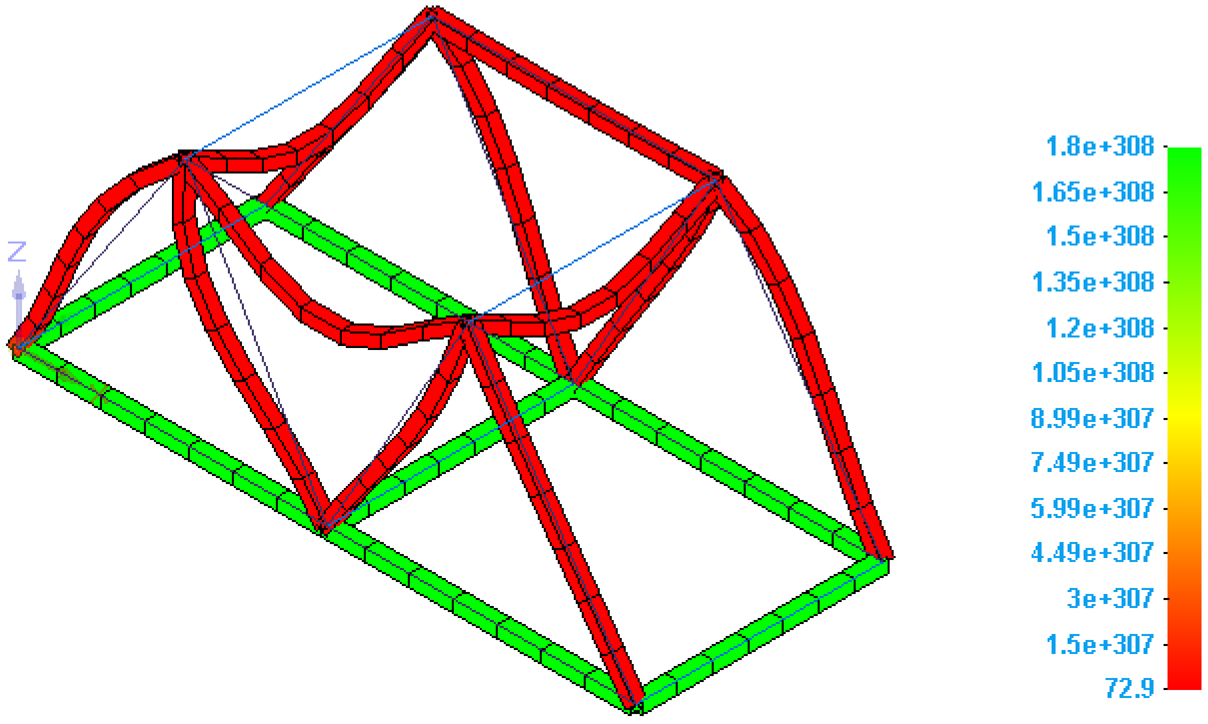
Von Mises stress plots for frames

Beam Von Mises stress plots are now available in the Simulation Results environment for a frames model.

Previously in Solid Edge Simulation, the margin of safety for a structural frame model was based on stresses calculated from **Beam Max Combined Stress** results (beam tension and beam compression), which can produce negative values.



Now when you solve a linear static study of a structural frame model, the factor of safety plots for beams are based on Von Mises stress, which provides a more accurate result.

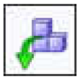


Two new Von Mises stress plots are also available.

For more information, see Beam stress component plots.

8. Data translation enhancements

CAD Direct command

Use the **CAD Direct** command  to insert non-Solid Edge files directly into Solid Edge assemblies without translating the files separately. This eliminates the task of translating non-Solid Edge files before inserting them into an assembly.

To update any changes made to the parent/linked parent file, right-click a CAD Direct component and use the **Update CAD Direct Geometry** command.

To break the link between the parent file and any CAD Direct component(s), right-click a CAD Direct component and use the **Break CAD Direct Link** command. This makes the Solid Edge geometry independent of the NX file. If you break the link, the link cannot be reestablished.

For more information, see Inserting files using CAD Direct.

Enhanced PMI support for STEP AP242 exports

You are now able to export a STEP AP242 file with more PMI support, including surface association for PMI dimensions and annotations.

For more information, see Saving to STEP.

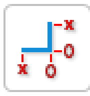
Export and import support for coordinate dimensions

Solid Edge 2D translation now exports and imports linear ordinate dimensions as true dimensions, which can be modified directly. Previously, ordinate dimensions were translated as AutoCAD blocks.

No new options were added for this enhancement. In Solid Edge, select the following translation options:

- When exporting a Solid Edge DFT file to DXF or DWG, on the **Solid Edge to AutoCAD Translation Wizard (File Mapping)** page (the first translation step), select the option **Export Solid Edge dimensions as AutoCAD dimensions**.

- When importing a DXF or DWG file to Solid Edge, on the **AutoCAD to Solid Edge Translation Wizard - Step 2 of 8**, select the option **Import dimensions as dimensions**.

> Note:
>
> In Solid Edge, linear ordinate dimensions are created using the **Coordinate Dimension** command .

Import from AutoCAD supports more symbols

Twenty-two new manufacturing symbols were added to the *Solid Edge ISO GDT Symbols* font in Solid Edge. Support is now provided for the additional symbols when they are imported from the AutoCAD AIGDT font file to the **2D Model** sheet in 2D Drafting.

The symbols are available in Solid Edge as 2-letter codes in the **Select Symbols and Values** dialog box. For a list of all available symbols, see Property text codes.

Save As JT exports surface PMI for inspection

Solid Edge 3D translation now exports to JT format the model PMI annotations and dimensions that are associated with surfaces and with edges. You can use this information to identify the model surfaces referenced by the PMI for visual inspection purposes. Surface information is required for creating inspection pathways in Coordinate Measurement Machine (CMM) software, such as Hexagon's PC-DMIS.

To support this enhancement, two new parameters were added to the JT translation file:

- **Export Surface Connections to PMI Annotations and Dimensions**
- **Export Symmetric Diameter as Standard Diameter dimension**

For more information, see:

- Save surface PMI to JT format
- Saving Solid Edge to JT.

Save the surface association of PMI in part files

A new registry switch is available to specify that surface information associated with PMI dimensions and annotations is saved with part and sheet metal files, as well as the edge information. This makes the surface data available to applications outside Solid Edge, such as NX, Femap, and Solid Edge CAM Pro.

For more information, see Saving the surface association of PMI in part files.

9. Solid Edge data management enhancements

Automatically assign document and revision numbers

You can now automatically assign document and revision numbers to files using **Property Manager** and **File Properties**, in addition to the common property dialog box (CPD). For more information, see Assign document and revision numbers automatically.

Save as Unmanaged command

When document numbering is enabled in Solid Edge data management (SEDM), the new **Save As Unmanaged** command lets you save the active document as unmanaged by not assigning a number and revision.

For information about Solid Edge data management, see Getting started with Solid Edge data management.

10. Teamcenter Integration for Solid Edge (SEEC)

Software Compatibility

The Teamcenter integration for Solid Edge (SEEC) is compatible with the following:

Solid Edge	Teamcenter 12.x	Teamcenter 13.x	Active Workspace
2022	12.1	13.1	Based on TC compatibility

To see a complete list of software compatibility for Solid Edge 2022 with the Teamcenter integration, see the **Software Field Bulletin for Solid Edge and the Teamcenter Integration**. A WebKey is required to access the site.

> Note:
>
> For additional information, see the SEEC_readme.htm.

Teamcenter preferences

The following are either new or updated Teamcenter preferences introduced with this release.

For a full description of each of the preferences, see the Teamcenter Preferences section of the Teamcenter Integration for Solid Edge (SEEC) Guide for Users and Administrators.

New preferences

SEEC_Enable_3DFindit

Enables functionality for using 3Dfind.it in the Teamcenter-managed environment, including the use of the commands: **Insert Components from 3Dfind.it**, **Replace Part with 3Dfind.it**, **Components by 3Dfind.it**, and **Search 3Dfind.it**. Requires application of the Teamcenter Feature Package for SE2021 MP1 or later.

SEEC_ProjectID_Default

Defines how the default Project ID is determined when you upload a new document to Teamcenter. You have the option to use the last Project ID specified as the default ProjectID, or you can choose to use the Project ID defined in the User Settings dialog box in Solid Edge, Hosted Active Workspace, or Teamcenter RAC.

SEEC_Dispatcher_JT

Determines when JT files are generated for Teamcenter-managed part (.par) and sheet metal (.psm) documents. You can choose from values to:

- Have Solid Edge generate the JT file on upload based on the preference value for **SEEC_Image_Generate_3D**.
- Enable Add to Teamcenter and Add to Teamcenter-Interactive to create Dispatcher requests.
- Have Add to Teamcenter and Add to Teamcenter-Interactive use the value of **SEEC_Image_Generate_3D** to determine if JTs are generated and uploaded, and enable Solid Edge to create the Dispatcher request.
- Determine that Solid Edge, Add to Teamcenter, and Add to Teamcenter-Interactive ignore the value for **SEEC_Image_Generate_3D** and create the Dispatcher request.

Updated preferences

SEEC_ExpandStructure

Default value is updated to 1. Large assemblies, as defined in Solid Edge Options, expand level-by-level, providing better performance with reduced resource consumption. This option is recommended.

Access to 3Dfind.it part library

To save you time in searching for, reconfiguring, or recreating parts, the CADENAS supplier part library is integrated into Solid Edge and made available to Teamcenter-managed Solid Edge users. You can search a collection of manufacturer product catalogs using the 3Dfind.it website, insert the part into your assembly, and then save the part to Teamcenter.

For more information, see Importing manufacturer parts.

Data Management tab

The **Data Management** tab is now available when you work with unmanaged Solid Edge files with the Teamcenter integration disabled. The commands on the tab provide easy access to common commands.

Determine the default Project ID

Traditionally, the default Project ID shown in common property dialog boxes (such as the New Document or Upload Document dialog box) is the Project ID that was last specified. With this release of Solid Edge, you can set the new preference **SEEC_ProjectID_Default** to determine if you want to use the traditional behavior or use the Project ID as specified in User Settings in Solid Edge, Active Workspace, or Teamcenter RAC.

For more information, see Change the default Teamcenter project.

Enhanced support for Teamcenter Dispatcher

There are two new enhancements for support of Teamcenter Dispatcher:

- You can defer the generation of JT files for part (.par) and sheet metal (.psm) documents from upload to Teamcenter, to a Dispatcher-managed request. Use the new Teamcenter preference, **SEEC_Dispatcher_JT** to determine when JT files are created, which enhances performance when you upload multiple files to Teamcenter.

  > Note:
  >
  > JT files for assemblies continue to be generated when the assembly is uploaded to Teamcenter.

- When you use the **Save as Translation** command and save the translated file to either the local disk or the Item Revision of the translated data set, you can offload the translation and upload process by generating a Dispatcher request. Use the new Teamcenter preference, **SEEC_Dispatcher_Translation** to manage file translation.

For more information, see the Teamcenter Integration for Solid Edge (SEEC) Guide for Users and Administrators and the *readme* in the Dispatcher Administrator folder contained in the SEEC feature package.

Improved expansion of large assemblies

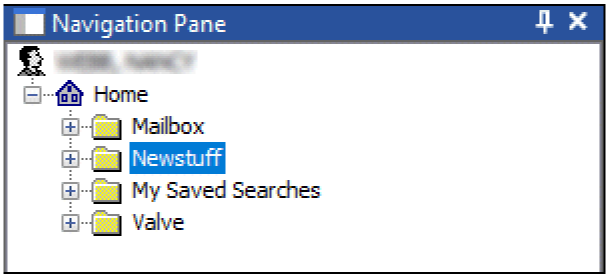
Bill of materials (BOM) expansion of large assemblies is improved for better performance through the reduction of resource consumption of Teamcenter server processes. To realize the performance enhancement, the existing Teamcenter preference **SEEC_ExpandStructure** is now delivered in the SEEC Feature Package with a default value of 1. Assemblies deemed to be large based on the values defined in the **Solid Edge Options→Assembly Open As** page, now expand level-by-level.

The value of the preference is displayed in the **BOM Expansion** section of the **Load Options** docking window, when you open an assembly in Solid Edge.

For a full description of the preference, **SEEC_ExpandStructure**, see the Teamcenter Preferences section of the Teamcenter Integration for Solid Edge (SEEC) Guide for Users and Administrators.

Navigation pane

Use the new **Navigation** pane in the **Open File** dialog box to expand the folders in Teamcenter and display the contents in the document display list. The **Navigation** pane shows all the folders in Teamcenter to which you have access.



For more information, see Open File dialog box.

New browser support for Hosted Active Workspace

Hosted Active Workspace now supports the use of Chromium-based Microsoft Edge.

For more information, see Configuring hosted Active Workspace.

Project assignment expanded

Previously you could only make project assignments to a Solid Edge document when you uploaded a document to Teamcenter. Access to project assignment is expanded to be available from the Teamcenter ribbon. Now you can make single or multiple selection project assignments using the new Assign Project ID command. When you select documents for assignment to a project, all projects of which you are a member are available for selection.

For more information, see Assign a Teamcenter project to an item.

Revert to Teamcenter command

A new command, **Revert**, is available on the **Teamcenter** tab→**Manage** group. The command cancels the check-out from Teamcenter a part or subassembly within an assembly document and then reloads the latest version from Teamcenter.

> Note:
>
> To use the **Revert** command, the assembly must be saved to cache, but not uploaded to Teamcenter.

For more information, see Revert documents to the version in Teamcenter.

Structure Editor supports cloned variants

Structure Editor supports opening an assembly with a variant rule and creating a cloned assembly. This functionality is similar to using Solution Variant from Product Configurator with the managed option or using a Variant Rule. Cloning creates a copy of all configured and impacted assemblies. Drafts are included that are in the same or separate Item Revision based on the Solid Edge Option to include drawings when revising or copying managed documents.

For more information, see General tab, Manage page (Solid Edge Options dialog box).

User help for Structure Editor is integrated with Solid Edge Help. Start with the topic, Structure Editor overview.

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