Welcome!
Solid Edge University 2016
Buckle Up!

This is going to be a fast ride.

Solid Edge Drafting
Latest Updates – ST9
Solid Edge Drafting
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• Section View Modify Enhancements
Drawing View Enhancements – Productivity Improvements

Drawing View Update (Multi-Core Support)

• Solid Edge Options – Bottom of General Tab
  - Enable multi-core drawing view processing
  - Enable and disable – Enabled by default.

• Minimum requirements for multi-core processing:
  - 10 GB memory minimum
  - Multiple physical cores

• Can provide faster update of large drawings.
Drawing View Enhancements – Productivity Improvements

Drawing View Update (Multi-Core Support)

- Example - Test Case Assembly
- Snow Blower Assembly – 6 X 6 Pattern
  - Total parts: 19836
  - Unique parts: 330
  - Total levels: 7
  - Total subassemblies: 3780
  - Unique subassemblies: 67
  - No simplification

Snow Blower Assembly

6 X 6 Pattern
Drawing View Update (Multi-Core Support)

• Example - Test Case Drawing
  • 9 Drawing Views
    • 5 Principal Views
    • 4 Isometric Views

• Workflow
  1. Modify the assembly and return to the drawing to force drawing views out-of-date.
  2. Update all drawing views.

• Update Time Results
  • No Multi-Core – 6:45 min:sec
  • Out-of-the-box Multi-Core – 4:02 min:sec
  • Fine-tuned Multi-Core – 3:05 min:sec

9 Drawing Views
Change Drawing View Orientation

- No need to delete and start again.

- New buttons on the Drawing View Selection command bar:
  - Drawing View Layout
  - Custom Orientation Window
  - View Orientation

Undesired Initial View Orientation
Change Drawing View Orientation

1. Select the drawing view.
2. Choose a different orientation from the list.
Change Drawing View Orientation

1. Select the drawing view.
2. Choose a different orientation from the list.

Warning message – Other views will change.

- Propagates changes to adjacent views based on drawing view alignment.
Change Drawing View Orientation

1. Select the drawing view.
2. Choose a different orientation from the list.

- Three aligned views change orientation
- Isometric view does not change, because it is not aligned.

![Diagram of drawing views]
Change Drawing View Orientation

- Derived views are maintained.
  - Section
  - Auxiliary
  - Detail

- Derived view annotations:
  - Remain in same relative position to source view origin.
  - Constraints are removed.
Change Drawing View Orientation

• Intended to reorient views when they are initially placed.

• Not intended to reorient a fully annotated drawing.
Change Properties of Multiple Drawing Views

• Allows multiple drawing views to be changed together.

• Click the Properties button on the Drawing View Selection command bar.
Change Properties of Multiple Drawing Views

• Allow multiple drawing views to be changed together.

• Why can’t you right-click one of the selected views?
  • RMB clears the select set!
Change Properties of Multiple Drawing Views

- Intelligence for multi-selection:
- Tabs and properties enable and disable based on selection.
- Least common denominator.
Change Properties of Multiple Drawing Views

- Intelligence for multi-selection:
  - Tabs and properties enable and disable based on selection.
  - Least common denominator.
Drawing View Enhancements – Productivity Improvements

Change Properties of Multiple Drawing Views
• Properties are indeterminate when different in selected views.
Change Properties of Multiple Drawing Views

• Use it to show and hide parts (components).
• Use it to change display configuration.
Change Properties of Multiple Drawing Views

- Use it to show and hide parts (components).
- Use it to change display configuration.
Broken Views (Constrain Break Lines)

- **Associative Break Lines**
  - You can constrain break lines to geometry.
  - Keypoints are located to connect break lines.

- **Use QuickPick when multiple keypoints are involved.**
Broken Views (Constrain Break Lines)

- Associative Break Lines
  - Example - Constrained to centers of holes
- Modify model geometry

For more information, see the help topic:
- Modify break lines in a broken view
Broken Views (Constrain Break Lines)

- Connect constraints are placed for geometry connection.
- So how do I connect at a distance?

- Example – Constrain .5” from edge of hole.
Broken Views (Constrain Break Lines)

- Use Draw in View to define a line at distance.
- Line will be used for break line connection.
Drawing View Enhancements – Productivity Improvements

Broken Views (Constrain Break Lines)

• Create a layer to place a line and a dimension.
• Don’t use the Auto-Hide layer.

• Draw a line and add a dimension to the drawing view geometry
Broken Views (Constrain Break Lines)

- Close Draw in View.
- Connect the break line to the drawn line.
Broken Views (Constrain Break Lines)

- Break the view.
- For the broken view, hide the new layer.

- Break line is constrained to hidden geometry.
Broken Views (Constrain Break Lines)

• Beware of crossing break regions:

  • Cannot create break regions that cross.

  • However, constraints can cause break regions to cross.
Broken Views (Constrain Break Lines)

- Beware of crossing break regions:
  - Invalid state for break regions.
  - View will unbreak during update.
  - Break lines will display in Error Dimension color.
  - Must be resolved before breaking again.
Drawing View Enhancements – Productivity Improvements

- **Section View Modify Enhancements**

- **Change between**
  - Section Only and Full Section

![Section View Modify Enhancements](image)

Section Only

Full Section
Drawing View Enhancements – Productivity Improvements

• **Section View Modify Enhancements**

  ![Section View Modify Enhancements](image)

• Change between
  • Revolved Section and Normal Section

  ![Revolved Section](image)  ![Normal Section](image)
Dimension/Annotation Enhancements

- Hole Count in Callout
- Annotation Saved Settings in External File
- Dimension Initial Stack Distance
- Retrieve Dimensions to Multiple Drawing View Orientations
- Symmetric Diameter Dimensions
- Annotation Edit Fields (Cut, Copy, Paste)
Dimension/Annotation Enhancements

Hole Count in Callout
Use two-letter codes to get the hole count in a callout and a dimension.

- Uses hole parameters to determine same holes.
- Supports three options:
  - %QC – Coplanar Holes (Quantity Coplanar)
  - %QP – Parallel Holes (Quantity Parallel)
  - %QA – All Holes on Model (Quantity All)
- Define a smart note with Smart Depth tab:
  %QN = “%QC X”
- Automatic display when count > 1
- Highlight the hole geometry

For more information, see the help topics:
- Hole callouts now show hole count
- Define smart feature properties in the Dimension style
- Place a feature callout dimension
Dimension/Annotation Enhancements

Annotation Saved Settings in External File

- Saved Settings in external file for the following:
  - Feature Control Frame (FCF)
  - Surface Texture Symbol (STS)
  - Balloon
  - Edge Condition
  - Dimension Prefix

Note: Legacy FCF and STS saved settings were internal to the file. Names will be listed with an ‘*’ prefix. Re-save it to write to external file.

For more information, see the help topic:
- Working with *legacy annotation saved settings
Dimension/Annotation Enhancements

Dimension Initial Stack Distance

- Defined in Dimension Style and Dimension Properties
Dimension/Annotation Enhancements

Dimension Initial Stack Distance

- Used by the following commands:
  - Auto-Dimension
  - Arrange Dimensions
  - Retrieve Dimensions

- Note: The example is with Arrange Dimensions.

Before Arrange Dimensions

<table>
<thead>
<tr>
<th>Initial Stack Distance</th>
<th>Initial Stack Distance</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 X Font Size</td>
<td>3 X Font Size</td>
</tr>
</tbody>
</table>

Before Arrange Dimensions:

- 1 X Font Size
- 3 X Font Size
Dimension/Annotation Enhancements

Retrieve Dimensions to Multiple Drawing View Orientations

- New option for the Retrieve Dimensions command:
  
  - Multiple Views

  - If a dimension is visible in the view orientation, it can be retrieved.
  
  - Each view must be clicked separately.
Symmetric Diameter Dimensions

- Consistency with other dimension placement workflows.
- Reduced number of clicks
- Placement by the following:
  - Horizontal/Vertical
  - By 2 points
  - By Dimension Axis

For more information, see the help topic:
- Symmetric diameter dimensions enhancements
Dimension/Annotation Enhancements

Symmetric Diameter Dimensions

- Improved Placement Workflow
  
  - Reduced number of clicks
  - After alignment set is started, single click on geometry
  - Automatic placement and stacking
  - Display center axis while placing

1. Click centerline.
2. Click diameter line.
3. Click to place first dimension.
4. Click next line.
5. Click next line.

For more information, see the help topic:
- Place symmetric diameter dimensions
Dimension/Annotation Enhancements

Symmetric Diameter Dimensions

- Dimension Style/Dimension Properties
- Alternate text positions (Same as Concentric Diameter Dimension command)
- Improve readability

Enabled

Disabled
Dimension/Annotation Enhancements

Symmetric Diameter Dimensions

- Dimension Style/Dimension Properties
- Underline symbol and prefix
- Extends the dimension line under the diameter symbol and prefix.

[Diagram showing enabled and disabled underline symbol and prefix for symmetric diameter dimensions]
Dimension/Annotation Enhancements

Symmetric Diameter Dimensions

• Improved Detail View Workflow
  • Center axis is in source view.
  • Automatic text position to Detail view.
  • Automatic trim dimension line to Detail View.

• Position of detail view does not flip dimension display.
Dimension/Annotation Enhancements

Symmetric Diameter Dimensions

• Dimensioning to Intersection Points
  • Symmetric diameter to intersection point
  • Many more enhancements.
Dimension/Annotation Enhancements

Annotation Edit Fields (Cut, Copy, Paste)

• Some edit controls were changed in ST8 to support new formatting.
• Lost shortcut menu on those edit controls:
  • Cut, Copy, and Paste Commands
  • Could still use CTRL+C, CTRL+V
• Has been added again.
• Backported to ST8 MP7.

![Callout Properties dialog box with formatted callouts]
# Table Enhancements

## Class Fit Tolerance Table

- List class fit tolerances in a table.
- Only for dimension type of ‘Class.’
- Select class dimension:
  - By Active Sheet – Associative to the sheet
  - By Drawing View – Associative to the drawing view
  - By User Selection
- Retrieve fit tolerances from related `LimitsAndFits.txt` file in Solid Edge Options dialog box.

### Example Table

<table>
<thead>
<tr>
<th>Value</th>
<th>Class Fit</th>
<th>Upper and Lower Stack Tolerance</th>
<th>Upper Limit Tolerance</th>
<th>Lower Limit Tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>h7</td>
<td>0.016</td>
<td>20.00</td>
<td>19.98</td>
</tr>
<tr>
<td>40</td>
<td>h6</td>
<td>0.002</td>
<td>40.00</td>
<td>39.98</td>
</tr>
</tbody>
</table>

For more information, see the help topic:
- [Create a tolerance table](#)
Property Text Enhancements

• Property Text Lists
• Exposed Variables with Real Values
• Custom Occurrence Properties from Assembly
• Graphic Connection to Assembly (Part Parent)
Property Text Enhancements

Property text Lists

- Properties of different types with same name.
- Hard to determine which property is of which type.

- Strings added to help identify Property Text type.
- Corresponding property text qualifier:
  - (Custom Property) – ‘/CP’
  - (Exposed Values) – ‘/EV’
  - (Custom Material) – ‘/CM’
  - (Occurrence Property) – ‘/OP’
Exposed Variables with Real Values

- ST8 and Before – Variable table variables were only exposed through custom properties as strings.
- Limited formatting of these values when in property text.
- ‘/CP’ modifier for custom property

- ST9 – a new modifier ‘/EV’ to get the real value of exposed variable table properties.
- Allows formatting of numbers in property text.
- Round-off ‘@2’, etc.

For more information, see the help topic:
- Exposed variables referenced by property text
Property Text Enhancements

Custom Occurrence Properties

- Define Custom Occurrence Properties in the Assembly
- `shaft_cap.par` – Each occurrence has a different value.

For information about custom occurrence property enhancements in assemblies, start with the help topic:
- Custom occurrence properties in assembly

Assembly Occurrence Properties
Property Text Enhancements

Custom Occurrence Properties
- Use Assembly Custom Occurrence Properties in Draft
- Shown are Parts List and Callouts.
- ‘/OP|G’ qualifier for occurrence property.
- Property Text - %{MyProperty/OP|G}
- Each Callout has different display value.
Property Text Enhancements

Graphic Connection to Assembly (Part Parent)

- From graphic connection to assembly
- Callout is connected to a part edge
- Use ‘/GA’ modifier
  
  `%{File Name (no extension)|GA}
  `
- Returns the assembly/sub-assembly owner of the part.
- Will return part file name if no assembly found.
Graphic Connection to Assembly (Part Parent)

- From graphic connection to assembly
- Part is – ‘bell_hsg.par’
- Sub owner assemblies are:
  - simpassy.asm
  - simpassytest.asm

For more information, see the assembly-to-draft workflow and watch the video:
- **Example: Show custom occurrence properties in a parts list**

<table>
<thead>
<tr>
<th>Item Number</th>
<th>File Name (no extension)</th>
<th>Quantity</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>simpassy</td>
<td>1</td>
</tr>
<tr>
<td>11</td>
<td>bell_hsg</td>
<td>1</td>
</tr>
<tr>
<td>12</td>
<td>output_hsg</td>
<td>1</td>
</tr>
<tr>
<td>13</td>
<td>hex_bolt</td>
<td>4</td>
</tr>
<tr>
<td>14</td>
<td>small_spur</td>
<td>1</td>
</tr>
<tr>
<td>15</td>
<td>shaft_cap</td>
<td>3</td>
</tr>
<tr>
<td>16</td>
<td>gear_shaft</td>
<td>2</td>
</tr>
<tr>
<td>17</td>
<td>worm_gear</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>simpassytest</td>
<td>1</td>
</tr>
<tr>
<td>21</td>
<td>bell_hsg</td>
<td>1</td>
</tr>
<tr>
<td>22</td>
<td>output_hsg</td>
<td>1</td>
</tr>
<tr>
<td>23</td>
<td>hex_bolt</td>
<td>4</td>
</tr>
<tr>
<td>24</td>
<td>small_spur</td>
<td>1</td>
</tr>
<tr>
<td>25</td>
<td>shaft_cap</td>
<td>3</td>
</tr>
<tr>
<td>26</td>
<td>gear_shaft</td>
<td>2</td>
</tr>
<tr>
<td>27</td>
<td>worm_gear</td>
<td>1</td>
</tr>
</tbody>
</table>
Other Enhancements

• Update Background Sheet Geometry from Template
• Delete Empty Layers
• Insert Image (All Image Types)
Other Enhancements

Update Background Sheet Geometry from Template
- New Replace Background command
  - Replace current background sheet geometry.
  - With selected background sheet
    - From selected Draft file (updated template).
  - Can use multi-select.
  - Sheet name by sheet name replacement.
  - New sheets added if not found in current file.
- Exposed to API
  - Build your own batch macro.

For more information, see the help topics:
- Replace background sheets
- Replace Background command
Other Enhancements

Update Background Sheet Geometry from Template

- New Replace Background command

- Heads up! Take care when using blocks:
  - Such as for title block
  - When a block with same name exists, then
    - Places an occurrence of the existing block.
    - Does not replace the block.

- How do I get around this?
  - Version the block to copy – ‘MyTitleBlockV2’
  - Eliminate clashes in block name.
  - Will copy the block with the new name.
Other Enhancements

Delete Empty Layers

- ‘Layer2’ has no geometry and needs to be easily found and deleted.
Other Enhancements

Delete Empty Layers

• New button for ‘Delete Empty Layers.’
• Also on the Layers shortcut menu.
• After running the command, ‘Layer2’ is deleted.
Other Enhancements

Insert Image (All Image Types)

- New option to list ‘All Image Files’ (/*.bmp;/*.png;/*.jpg;/*.jpeg;/*.jpe;/*.jff;/*.tif;/*.tiff)
- All image file types are listed in the Open a File dialog box.
Thank You!

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