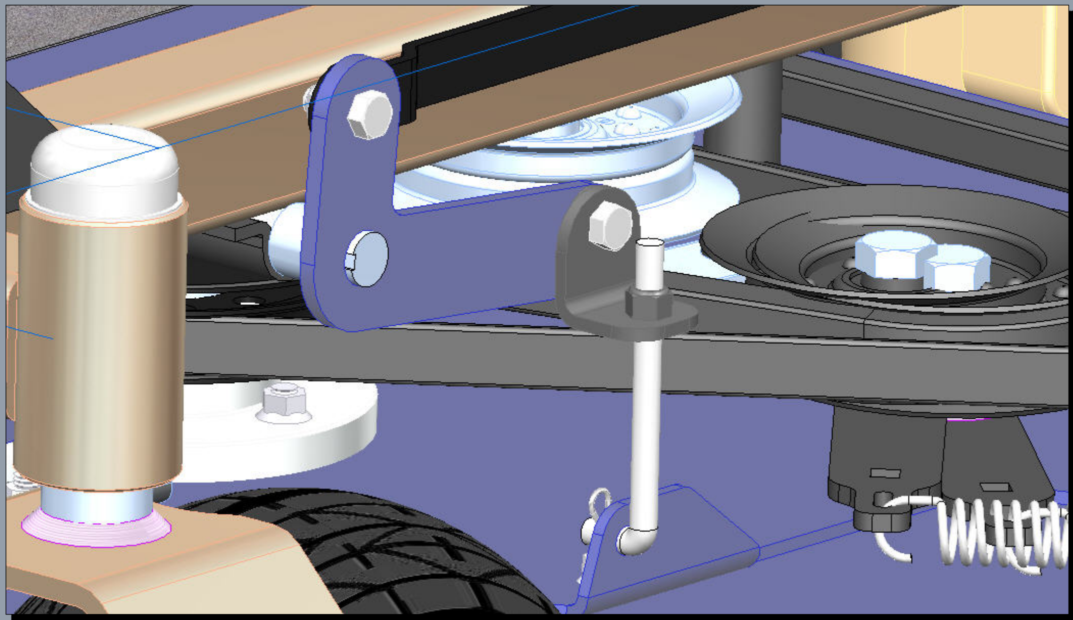
Solid Edge ST4 Update Training
Assembly



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Siemens PLM Software

Assembly Topics

- Short Cut Menu Simplification
- Assembly Relationship Enhancements
  - Grounded Component Indicator
  - Ground Relationship
  - Tangent Relationship Flip option
  - Edit Relationship Flip Option
  - Relationship Command Processing
  - Relationship Dimension Color
  - Center Plane Relationship
  - Range (Value Limits)
- Key-point Processing Improvements
- Round Feature in Assembly
- Selection Performance Improvements
- Steering Wheel in Assembly Improvements

Assembly Topics

- ERA – Flow Line Enhancements
- Replace Part uses “Select Configuration” option
- Auto scroll in PathFinder
- Replace part can now use part with same name
- Mass computation in Assembly (Qty Overrides)
- IPA Shift+Click uses Assembly Coordsys planes as input
- Simplified Assembly command processing
- Assembly Pattern support pattern adjustable sub-assemblies
- Weldment Assembly Icons
- How to Add the ST4 View Styles

Short Cut Menu Simplification

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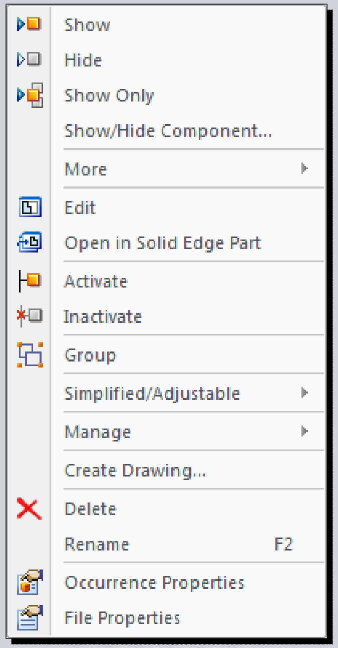
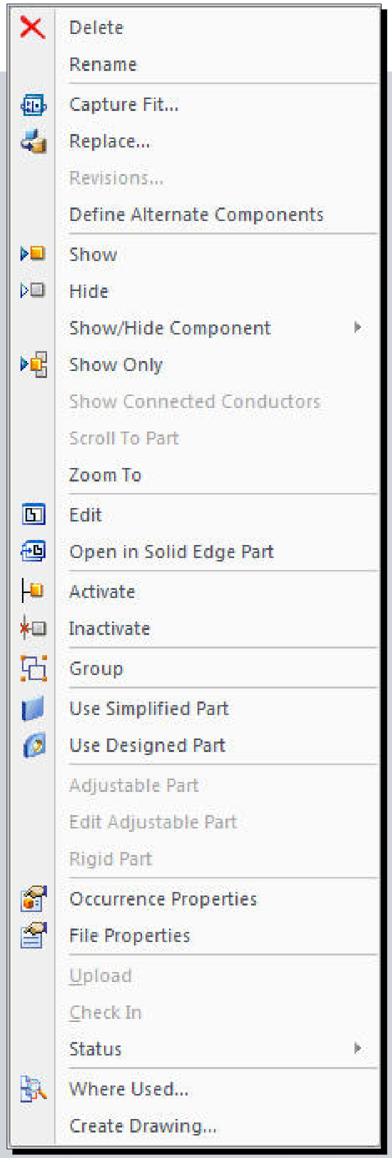
- Some changes have been made to simplify the assembly component menu
- First, we have reduced the number of commands on the assembly occurrence selection menu as shown in these examples
- Secondly, there have been a few behavioral changes to menu options which enhance the assembly user's experience. We cover this in the next few slides

ST4

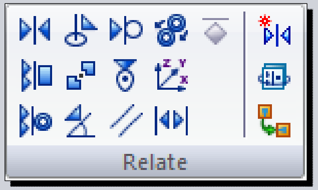


ST3

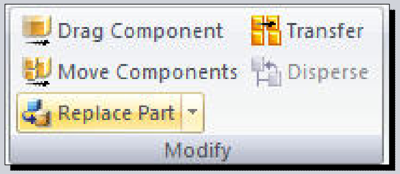


Short Cut Menu Simplification

- First, let's look at what was removed or moved
- "Capture Fit" was removed and currently resides in the "Relate" group on the command bar



- "Replace" was also removed, but also resides on the command bar in the "Modify" group



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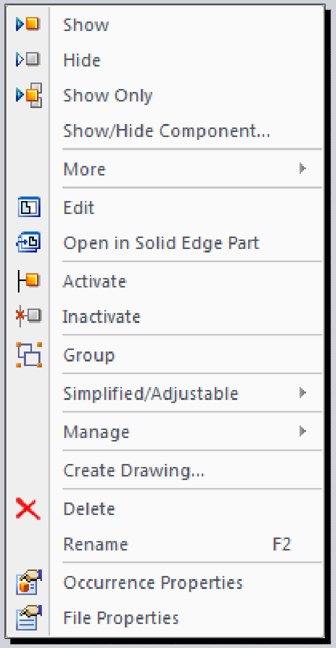
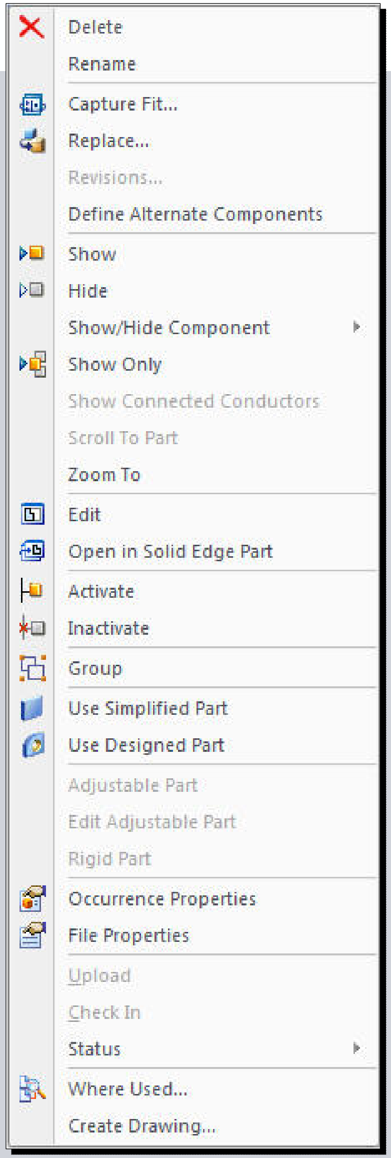


ST3



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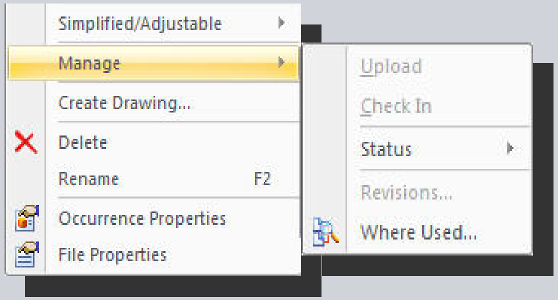
Short Cut Menu Simplification

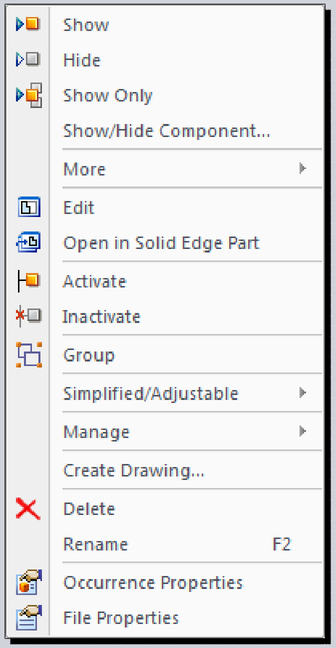
- “Revisions” was removed from the top level menu and moved into a new sub menu called “Manage”



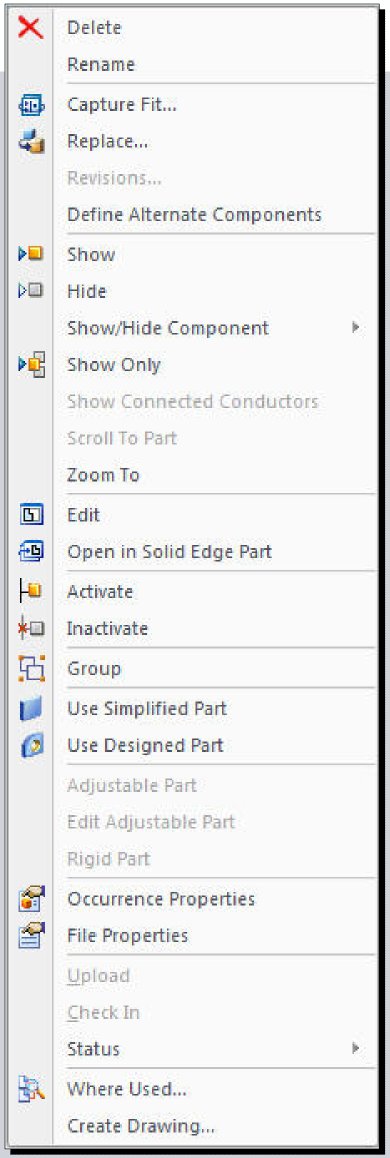
- Also notice under the “Manage” menu you find “Upload”, “Check In”, “Status”, and “Where Used”

- All four of these commands were removed from the top level

ST4

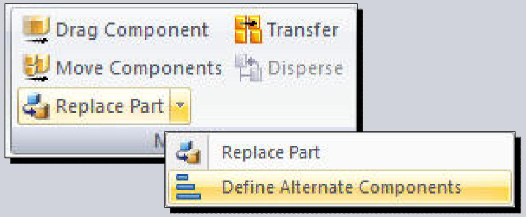


ST3

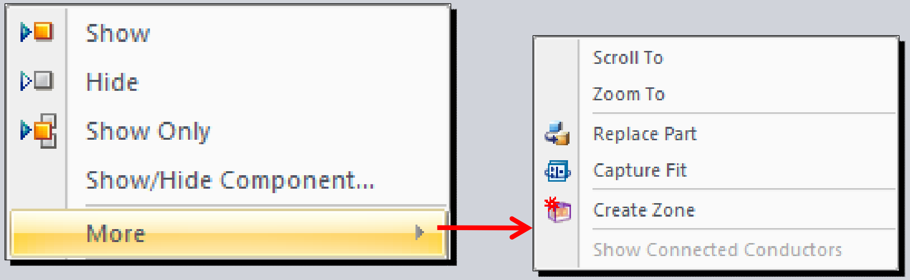


Short Cut Menu Simplification

- “Define Alternate Component” was removed, but can be found under the “Replace Part” pull down menu on the command bar under the “Modify” group



- Scroll To, Zoom To, Replace Part, Capture Fit, Create Zone, and Show Connected Conductors have been moved to the “More” menu



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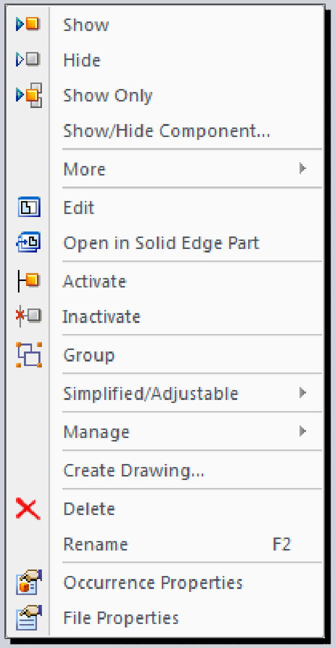

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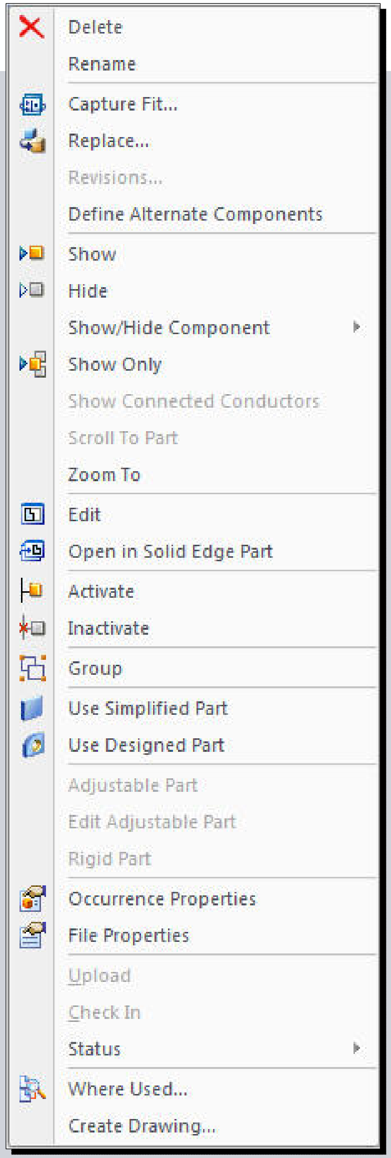

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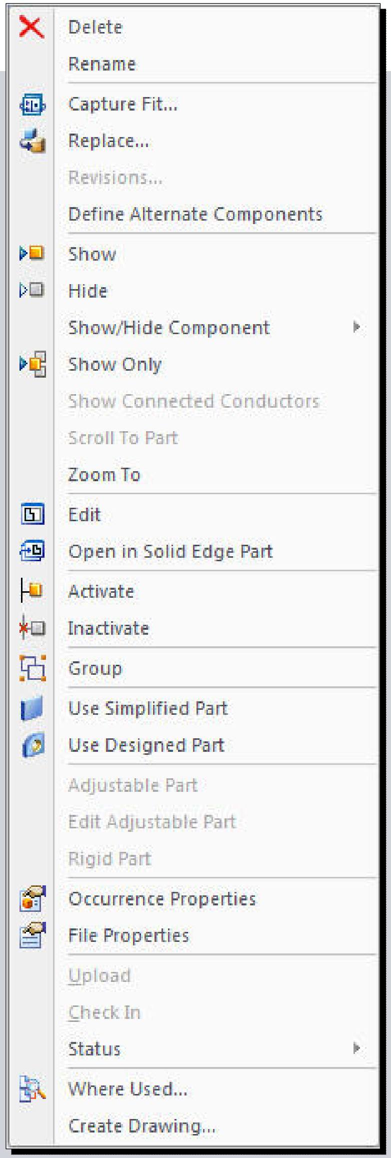
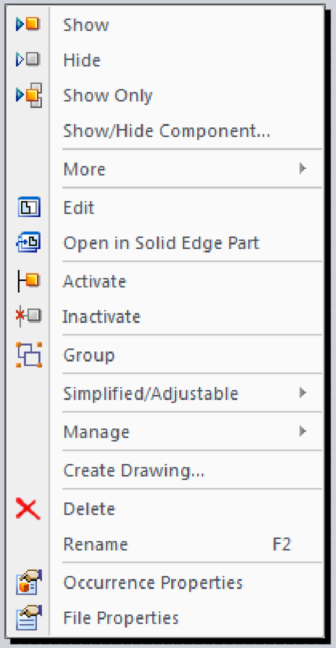
Short Cut Menu Simplification

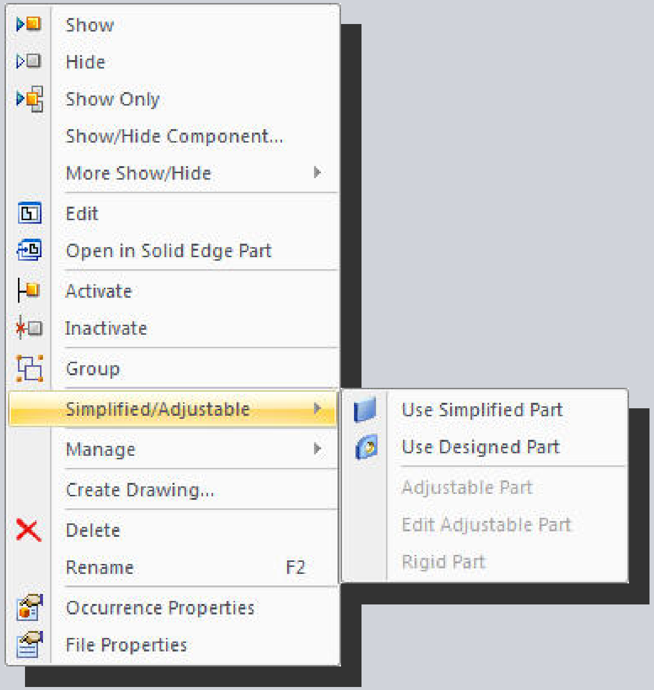
- “Use Simplified Part”, “Use Designed Part”, and adjustable commands have been moved to a “Simplified/Adjustable” pull out menu

ST3



ST4





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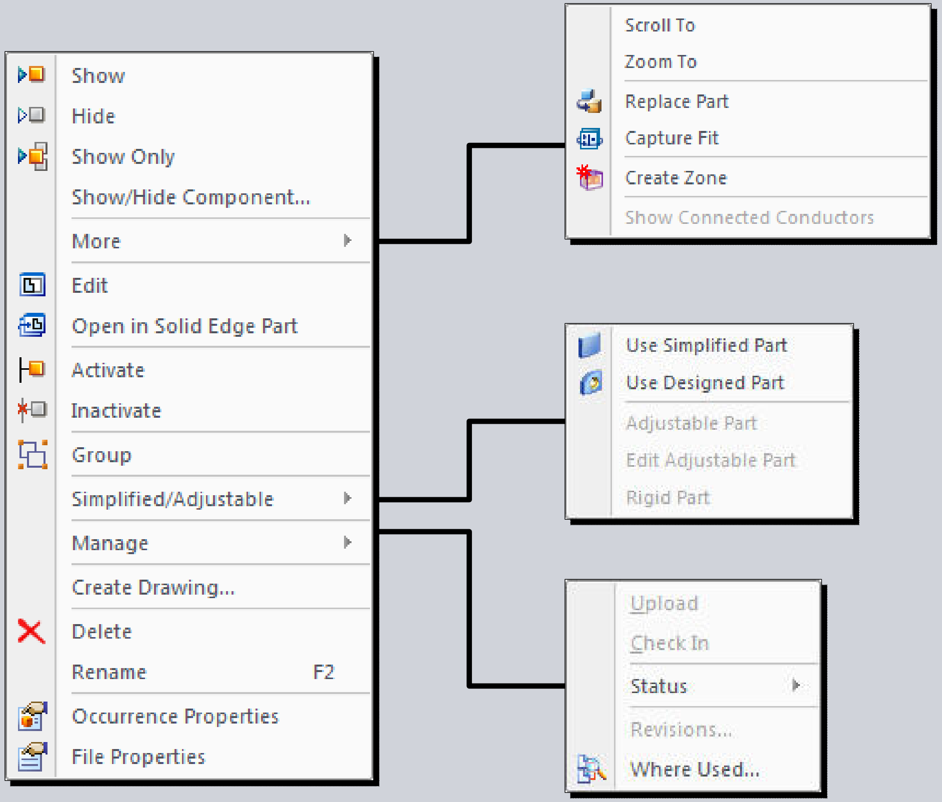
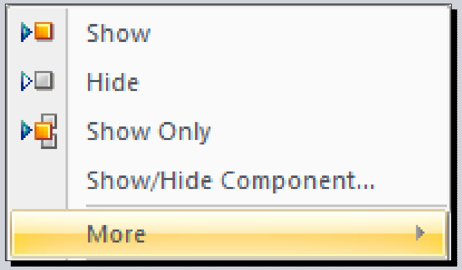
Short Cut Menu Simplification

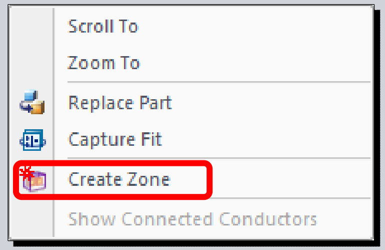
- The new menu in expanded form



Short Cut Menu Simplification

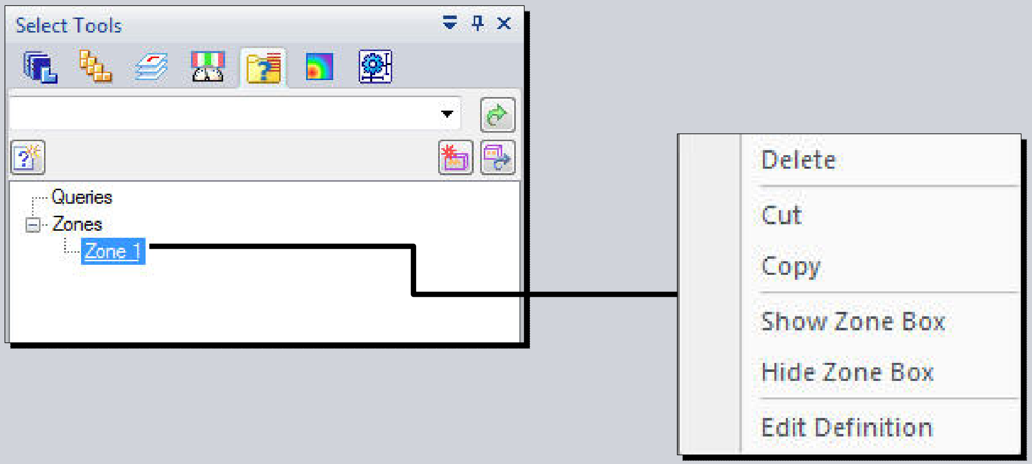
- From the “More” pull out menu , a user can now use “Create Zone”





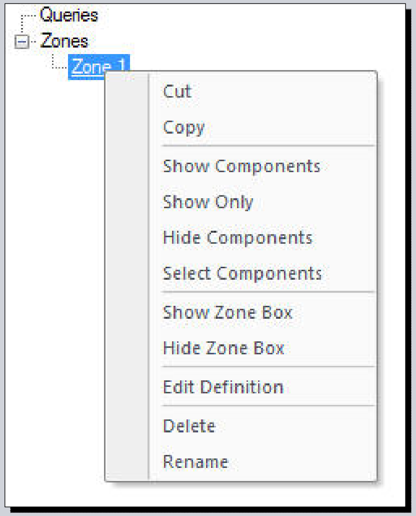
- Prior to ST4 the user would navigate to the “Select Tools” tab in PathFinder

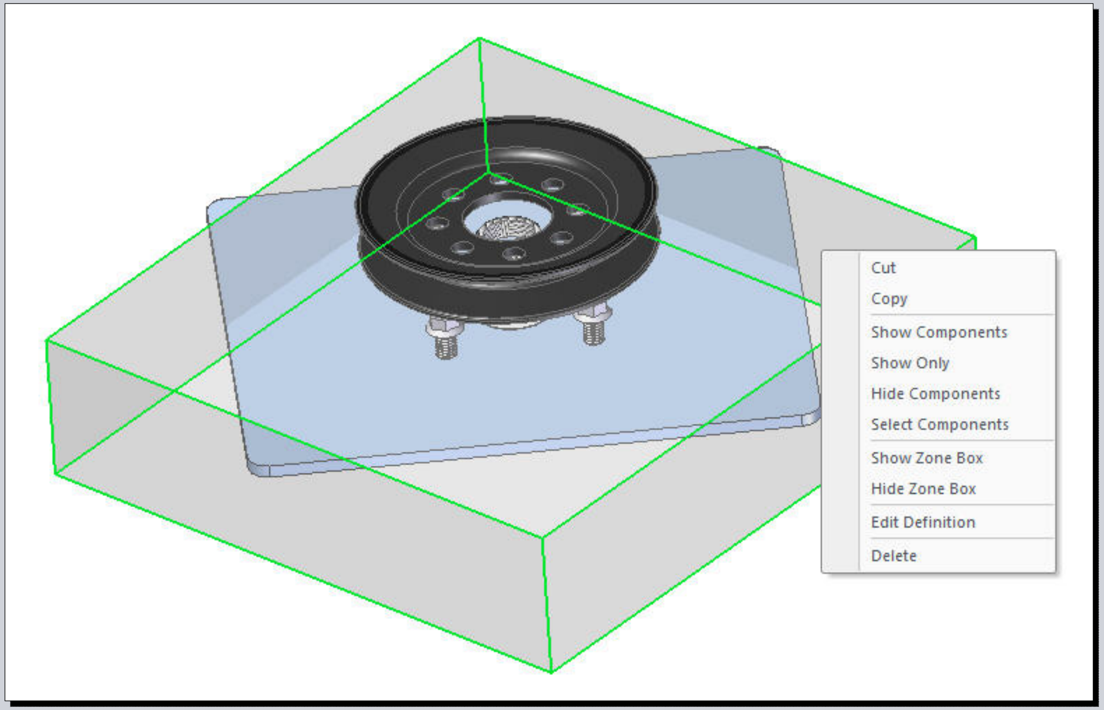
- A right mouse button click on a zone would give the user some zone options



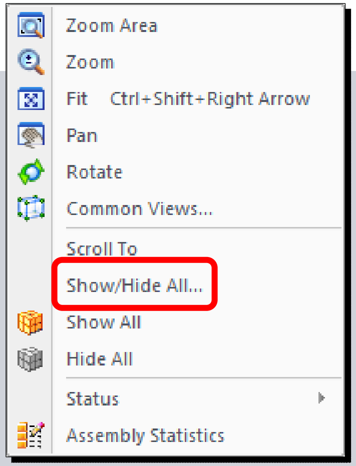
Short Cut Menu Simplification

- In ST4, the zone box short cut menu has been enhanced to include the ability to "Show/Hide Components", "Show Only", and "Select Components"

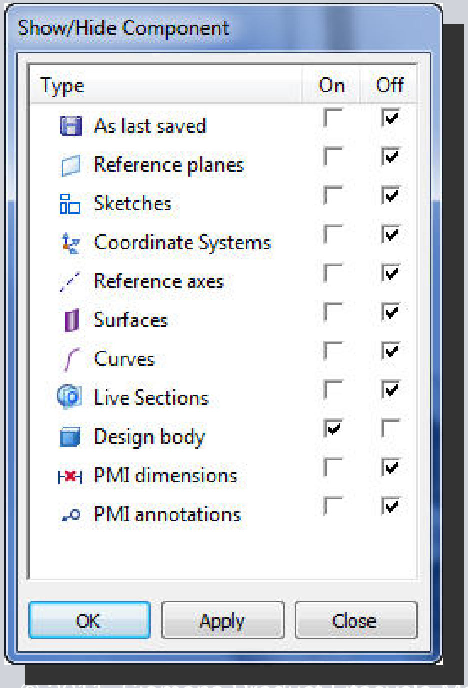


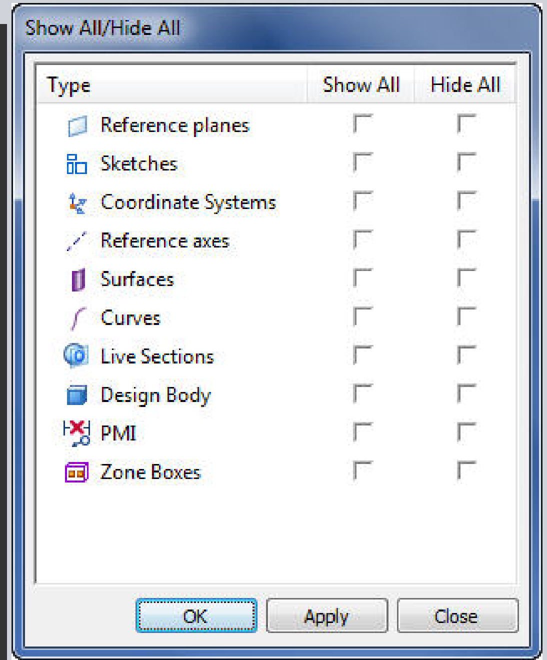


Short Cut Menu Simplification

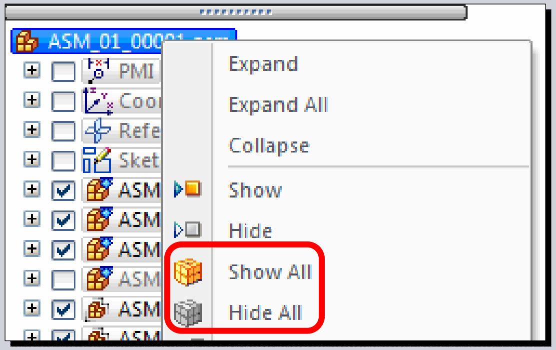


- Show All/Hide All commands have been combined into new a table found under "Show/Hide All…"
- Allows many items to be selected and changed at once



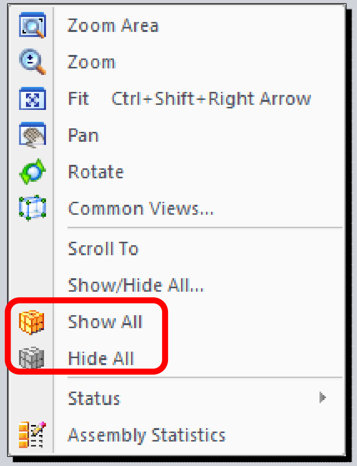


Return to Topics



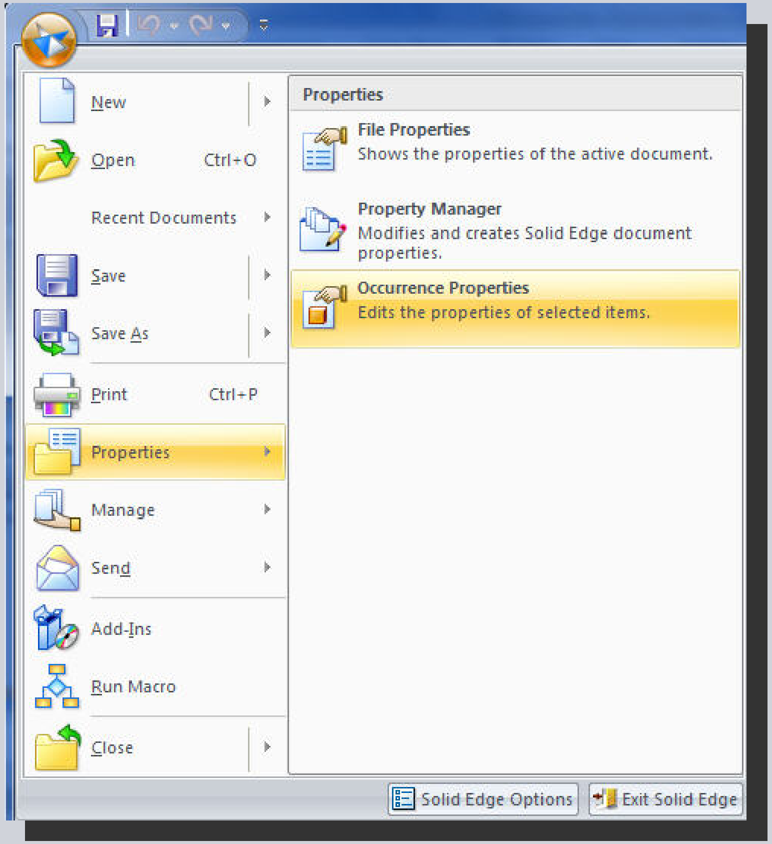
Short Cut Menu Simplification

- The “Show All” and “Hide All” options found under the top level assembly in PathFinder has been added to shortcut menu
- These options are only shown when no component is selected which implies these commands work on the entire assembly
- The user no longer has to scroll to the top of PathFinder to execute the command.



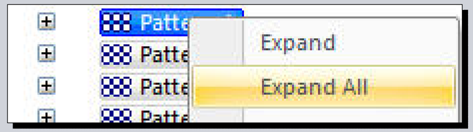
Short Cut Menu Simplification

- The "Scroll To" command now works for Sketches, Planes, and other component types
- "Occurrence Properties" has been added to the Application menu under "Properties"
- If no document is selected, SE shows the top-level assembly properties



Short Cut Menu Simplification

- Expand All now works with the “pattern” branch in PathFinder



- New stackable “Activate”  command stacks on top of currently running commands following current command stacking behavior

- If “Activate” is started with components in the select set, the command:
  - Activates the selected components
  - Terminates automatically
  - Restarts the prior-running command

Return to Topics

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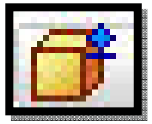
Short Cut Menu Simplification

- If "Activate" is started by clicking the button on the Command Ribbon or activating a keyboard accelerator, the command :
  - Starts
  - Changes cursor to Activate cursor
  - Remains active until the user presses the ESC key or right-clicks
  - Restarts the prior-running command upon user-termination
- The Activate action is not be added to the Undo list
- The "Activate"  command has been added to the Select group
- The "Activate" command now supports fence select

- This has also been added to profile/sketch environments while in-place activate (IPA) as well as in the Assembly sketch environment

Grounded Component Indicator

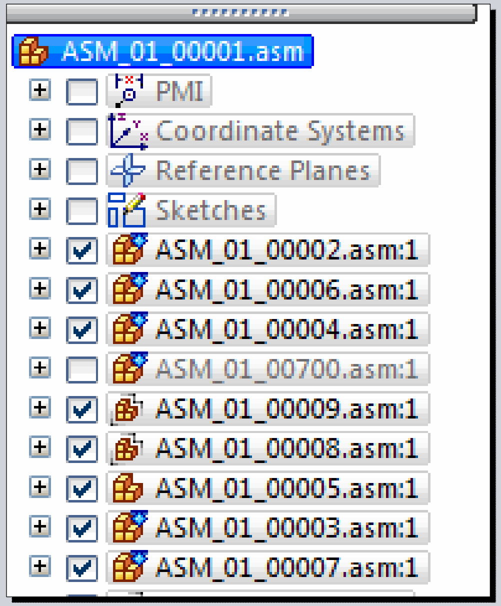


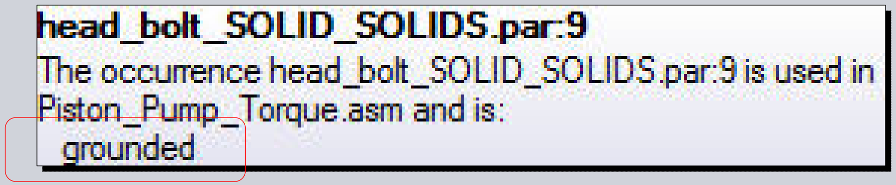
- Assembly components that have a “Ground” Relationship now have an indicator in PathFinder to display the grounded status

- This applies to Part, SM, and ASM occurrences

- The “Ground” relationship indicator only applies to unsuppressed relationships

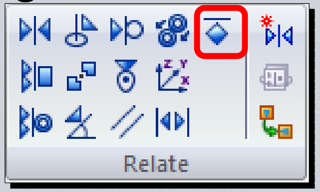
- The “ground” relationship has also been added to the tooltip for that occurrence



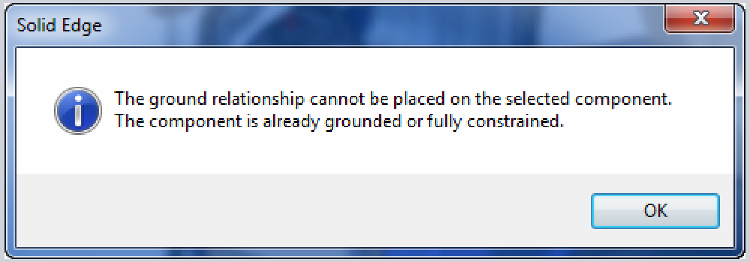


Grounded Relationship

- The “Ground” relationship is now action/object, meaning it is an active command when no components are selected
  - The ST3 object/action still exists without change

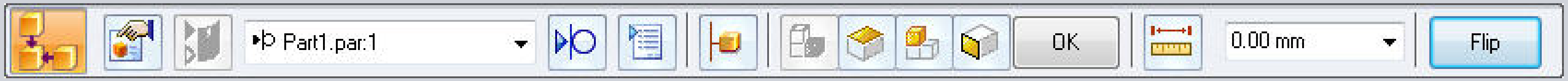


- The user can select only occurrences in the active assembly through either PathFinder or in the graphics window
  - Because it only supports single component selection, fence selection is not valid
- Once a component is selected and a ground created, the command restarts, unless the user presses the “Esc” key, a RMB, or selects another command
- If the component selected cannot be grounded, the user is presented with a message



Tangent Relationship Flip Option

- The flip direction button Flip has been added to the Tangent relationship
- This allows the user to change the mating sides used for the tangent relationship
- Before ST4, the user could not edit the tangent side of a surface or solid and in some cases had to use connect and construction geometry to achieve what they needed



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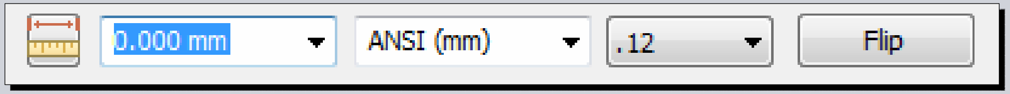
Edit Relationship Flip Option

- We added the flip option to the “edit value ribbon” so the user does not have to edit the relationship to flip it



- This applies to Mates, Aligns, Tangent or any relationship that offers a “flip” option

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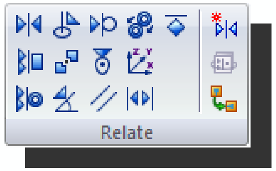
Relationship Command Processing



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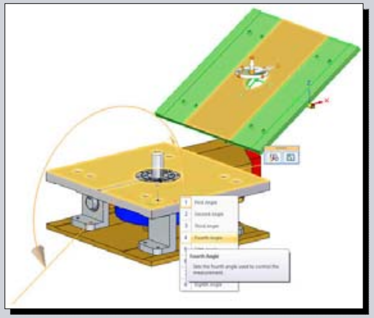
- All ASM Relationship commands now restart instead of terminating when the relationship is created
- A RMB click when in the second step of the command resets the command back to Step 1 with nothing selected
  - This makes the commands work like the “Assemble” command
- A RMB when in the last step of the command (Reduced input mode = Off) creates the relationship and restart the command
- A RMB during the Assemble Command when the command is in Create Relationship mode with no graphics selected terminates the command and runs NWA
  - In ST3 a RMB click this state clears the field but the command continues

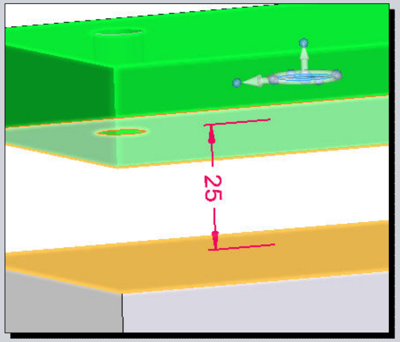
Relationship Dimension Color

- Prior to ST4, when editing a relationship the graphic input of the relationship (faces, etc.) was shown in highlight color
  - If present, the relationship dimension and other graphic indicators are also shown in the highlight color which made these items very hard to see



- We now display these using the handle color defined in Options/Colors which applies to the relationship graphics and dimensions used for all relationships

Note: The handle color is used for sketch relationships so this is consistent with how relationships appear in sketch



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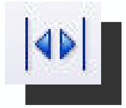
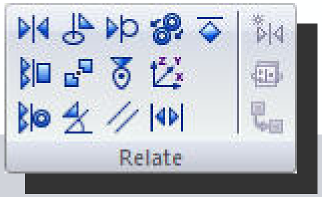
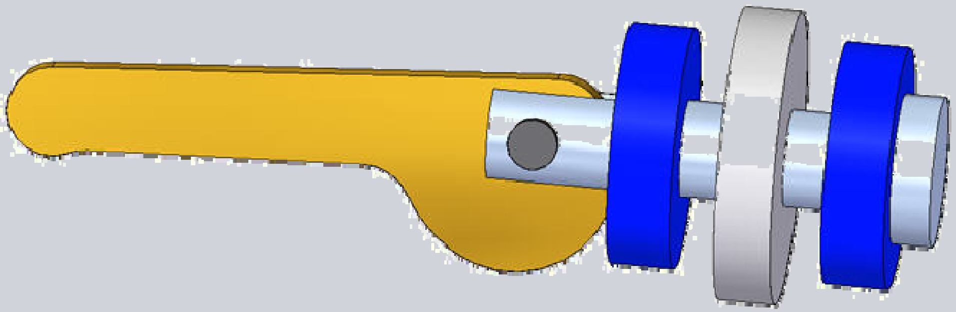
Center Plane Relationship

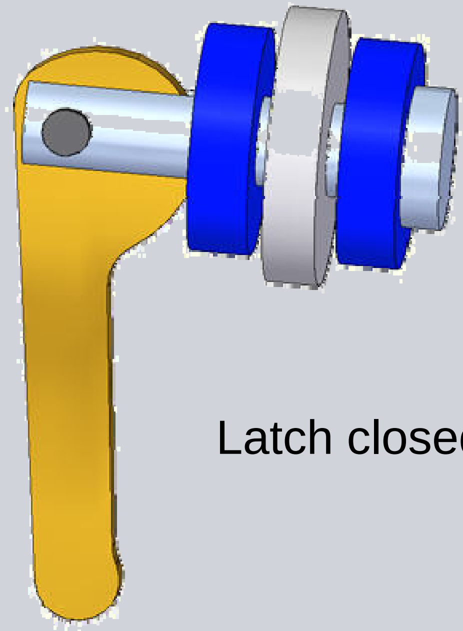




- We have a new relationship in assembly called “Center-Plane”
- The relationship “centers” a selected key-point, planar face, plane, edge, axis or plane/face pair onto the symmetry plane defined between two planar faces/planes or two key-points



Latch open



Latch closed

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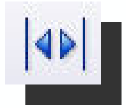
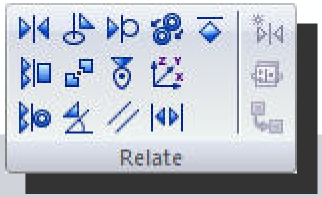
Center Plane Relationship

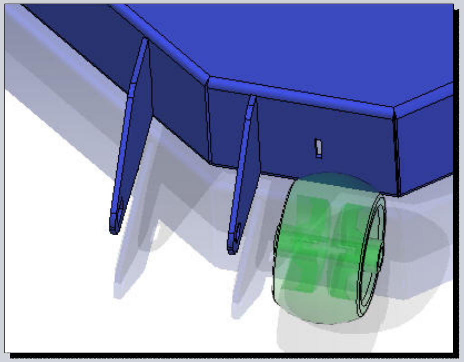




- The command centers objects between non-parallel faces/planes
- The command is not intended to be a symmetry relationship although it does exhibit symmetry behavior in some geometric/relationship cases
  - It is only intended to position a component on a plane centered between 2 objects such as a bolt in a slot
- The relationship can be applied between 2 and a maximum of 4 components (2 top level, 2 parts in a sub-assy)
- This relationship has no Offset value (No Fix/Float option)
- This relationship supports ADJ ASM and FOA like all other relationships

Center Plane Relationship

- The command is available on the Command Ribbon and in the Assemble Command Drop List
- The command must be explicitly invoked and is not part of “Flashfit”
- This relationship is not supported for “Capture fit”
- Let’s see how this relationship works
- One of the most common things is to place a part like this wheel where you want it in the center of the two side plates
- Here we have already added the axial alignment



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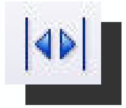
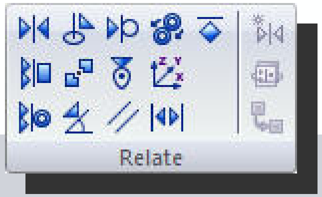
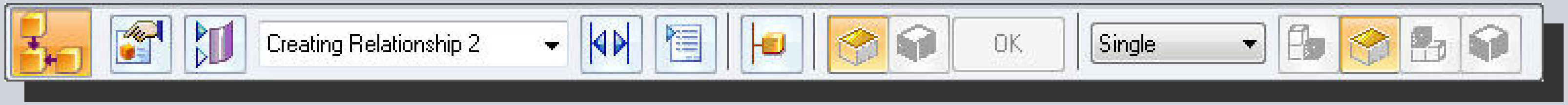
Center Plane Relationship





- The next step is to invoke the “center Plane” command



- Like other relationships, Quickbar will appear
- This command has a 2 step process
  - The Geometry to be Centered (Step 1)
  - The Geometry that defines the Center-plane (Step 2)
- Notice in Quickbar the “Single” option means single input
  - Step 1 defines the geometry to be centered, so using “single” means we can find the center of the geometry with a planar face, edge, vertex, axis or reference plane

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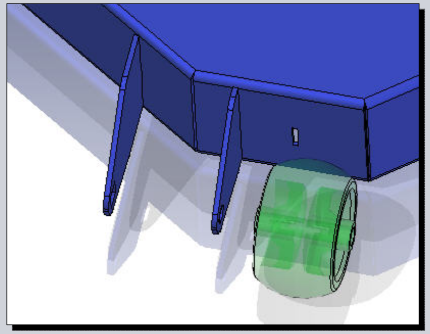
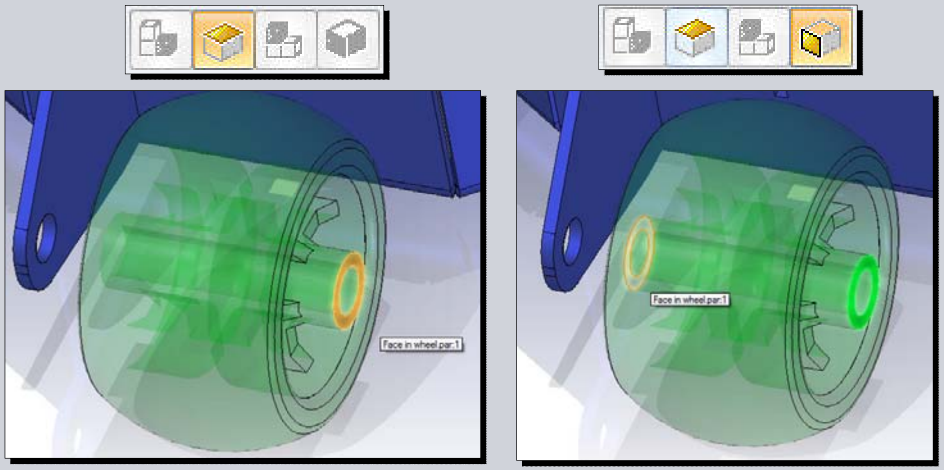
Center Plane Relationship

- For the wheel placement, we use the “Double” option

- With the Double option, we can select a planar face, reference plane, or keypoint. Since we have two sides of the wheel available, this is the easiest way to set this up



- Select one side of the wheel
- Select the opposite side
- A plane is created between the two selections for placement



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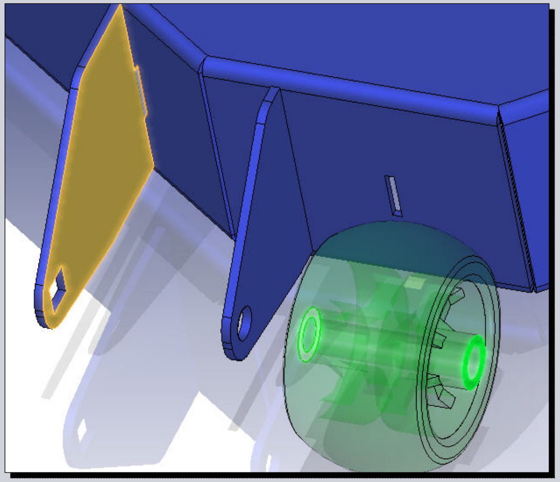
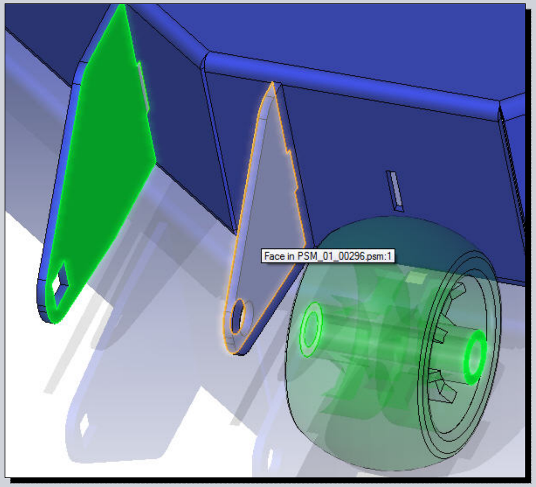
Center Plane Relationship

- Once the two faces that define the center of the placement part are complete, Quickbar moves to the “Center Plane” for the target step or 2nd step

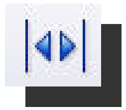
- In the 2nd step the default is “Double” and Solid Edge prompts the user to select the two planar faces, reference planes, or keypoints that define the center plane for the target step

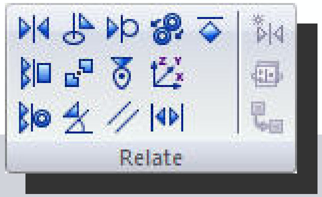
- After selecting the 1st target face, we can then select the 2nd target face which creates the center plane for our relationship

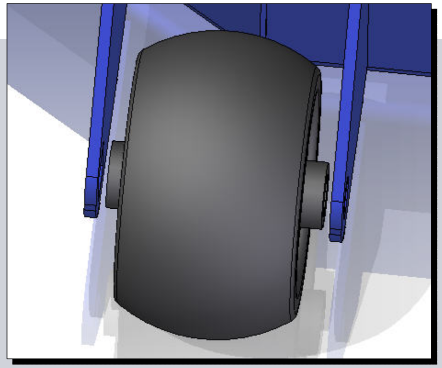




Center Plane Relationship







- This command makes centering components very quick and easy

- Some other things to understand when using this command

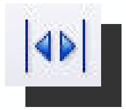
  - If the user has defined a plane using the “Double” option in the first step, the user can optionally select a “Single” plane in this step to be mated to the plane defined in the first step

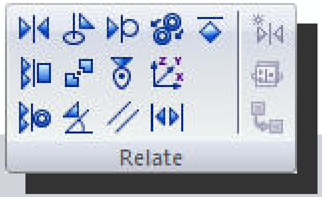
  - The default is always “Double” in the 2nd step

  - In this way, the command can work in reverse based on the inputs
    - Center a sub-assembly about a reference plane on place part

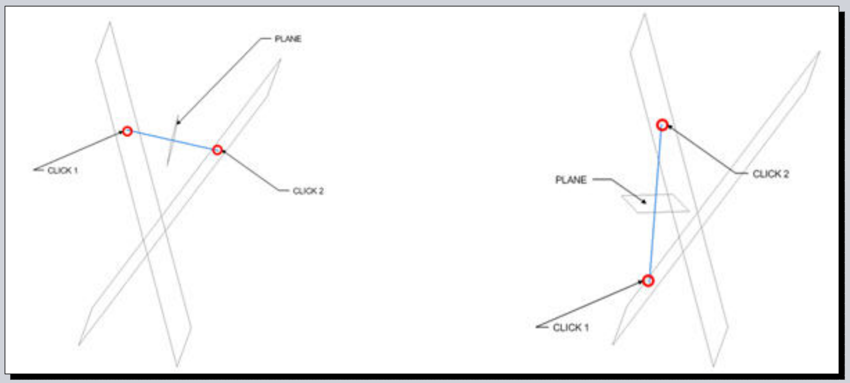
Return to Topics

Center Plane Relationship

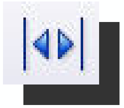


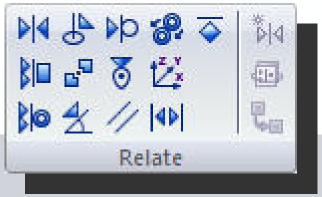


- Symmetry plane creation
  - When face/plane pairs are used to define the Center-Plane or the Center Placement geometry an internal construction plane is created and then a symmetry relationship is applied between the two input faces using the construction plane as the plane of symmetry
  - This moves the construction plane to the centered (symmetric) location between the two input faces
  - The initial position of the plane is such that it is geometrically/visually between the two input faces based on the users initial click points on the input faces
    - Using the two click points on the faces/planes create a vector between the two points
    - Create a plane normal to the above vector positioned at the mid-point of the two click points



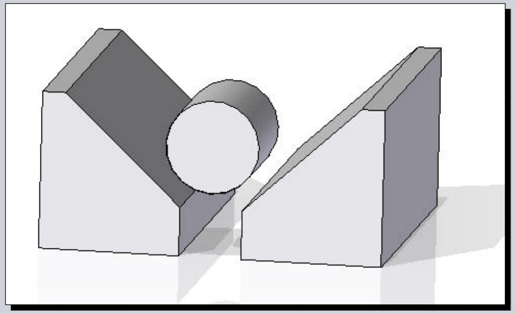
Center Plane Relationship



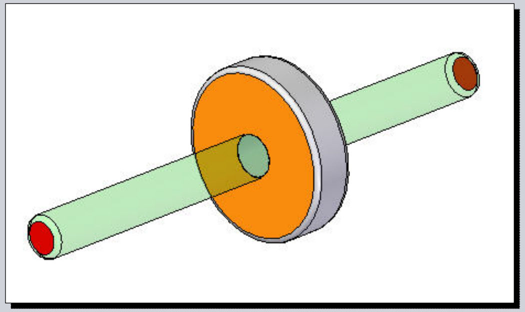


- Some examples:

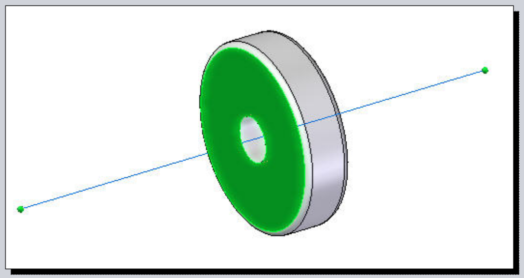
- In this example, the cylinder axis was made coincident with the centered plane defined by the two angle faces from two separate parts



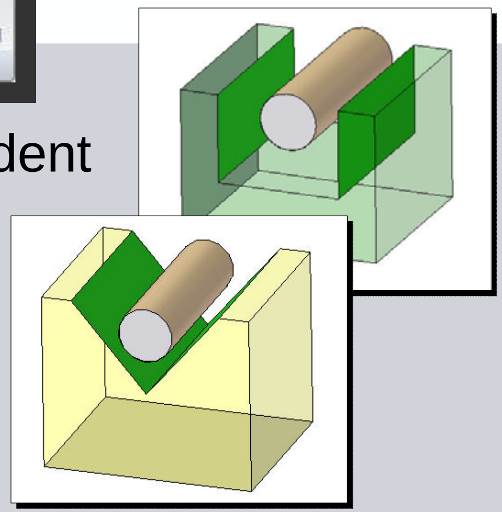
- In this example the orange face in the disc part is centered between the plane defined by the two red faces.
  - The red faces are from the same part



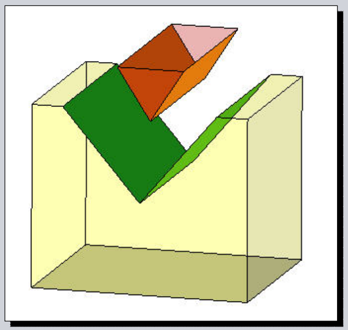
- In this example, the Center Plane is defined by two key-points from an assembly sketch

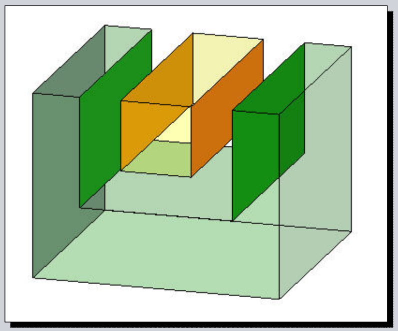


Center Plane Relationship

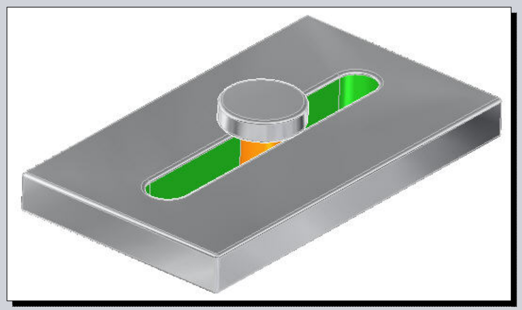


- In this example the Tan cylinder axis is made coincident with the sentered plane defined by the Green faces
  - Placement axis-2 center faces
- In the examples below the Orange faces are the placement faces
  - A symmetric plane is created between these two faces
  - The Green faces define the target Center-plane
  - A coincident relationship positions both parts

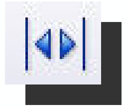


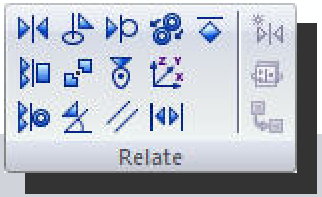


Can’t forget about the slot example!



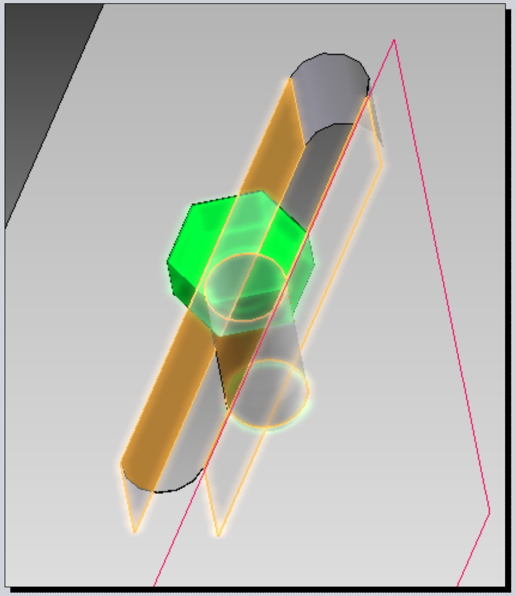
Center Plane Relationship

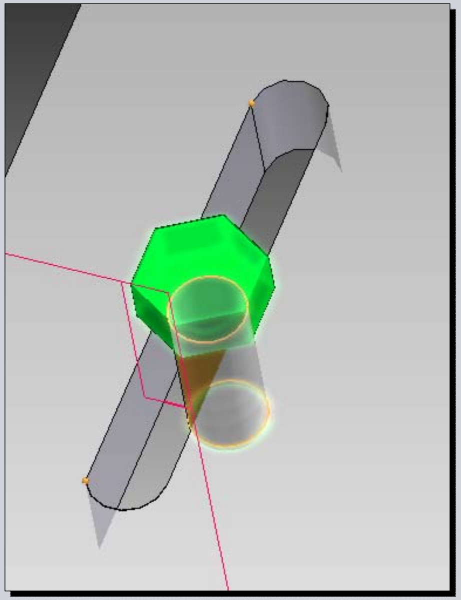




SIEMENS

- How to quickly center a bolt in a slot?
- Select the Pin Cylinder, then select the two side faces of the slot
- To center in the other direction repeat and select the 2 key-points at the end of the slot

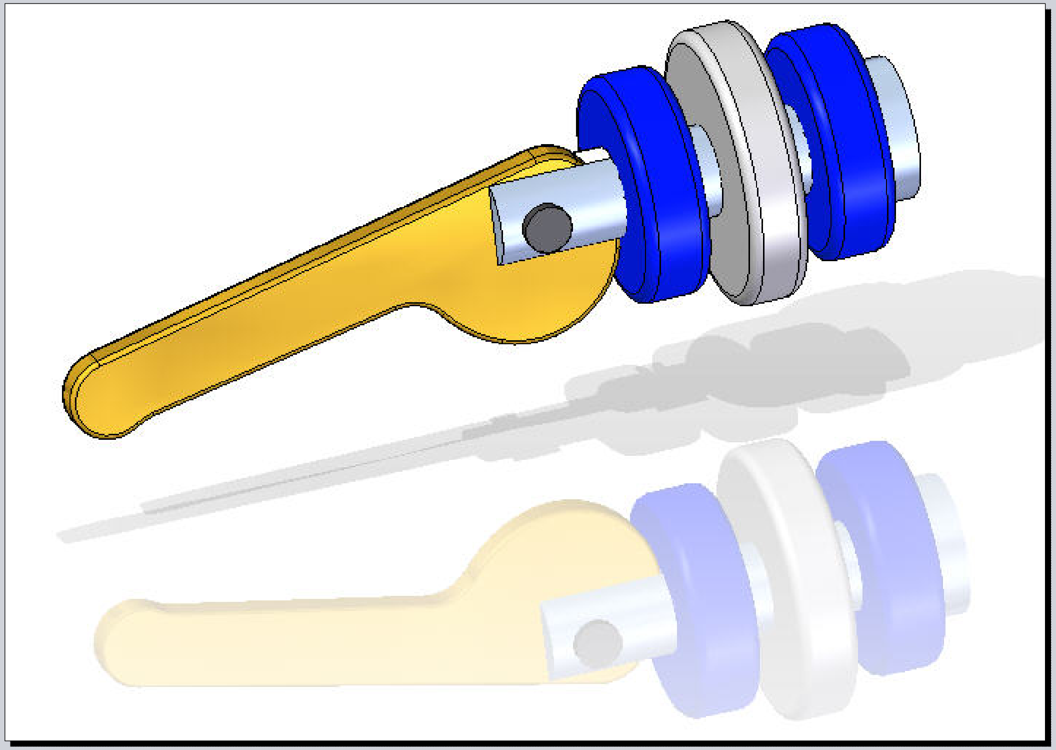




Return to Topics

Center Plane Relationship

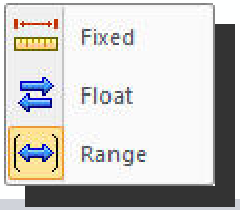
- Let's take a look at a couple examples



DEMO 1

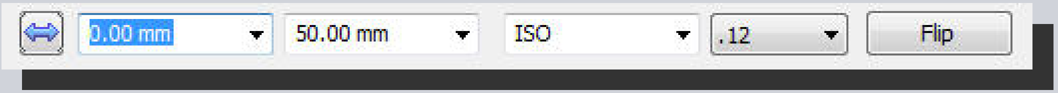
Return to Topics

Range (Value Limits)



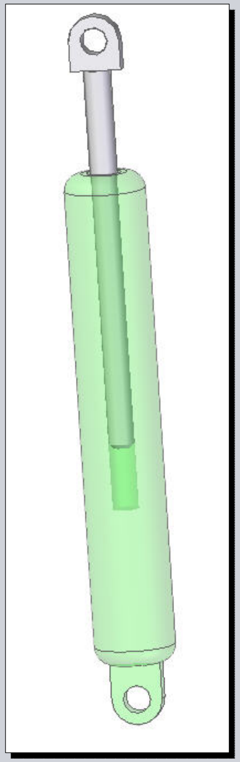
SIEMENS

- The “Range” options allows us to set a fixed relationship value to vary its value between a specified “Range” of values during assembly solve
  - Negative numbers are supported

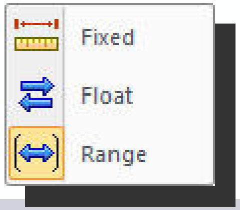


- The “Range” value is intended to support placement of parts in the assembly that can move between fixed limits as other components in the assembly solve
(Slides, hydraulic cylinders etc.)

- Range is supported in all relationships where we have fixed offset values such as Mate, Align, Connect, Parallel, Tangent, and Angle

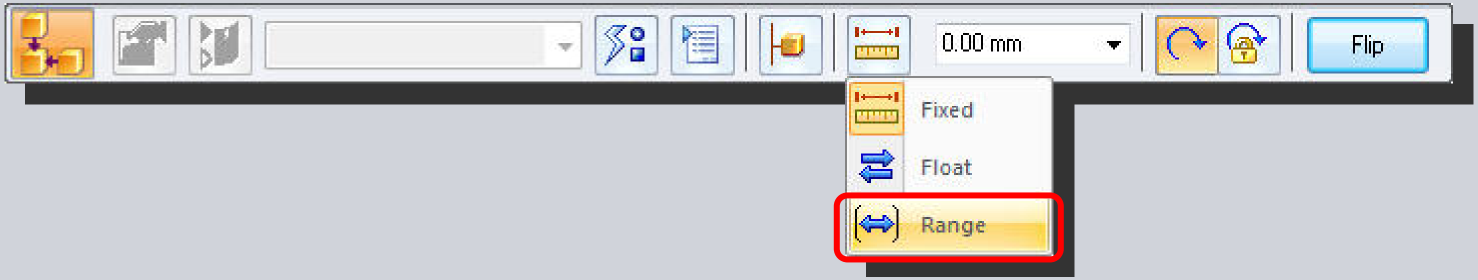


Range (Value Limits)



SIEMENS

- The “Range” option works the same as when you place a relationship with a fixed constraint, except you select the “Range” option instead

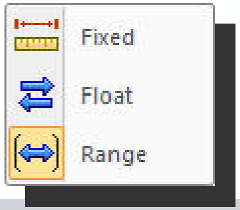


- When the “Range” option is selected the range inputs are activated

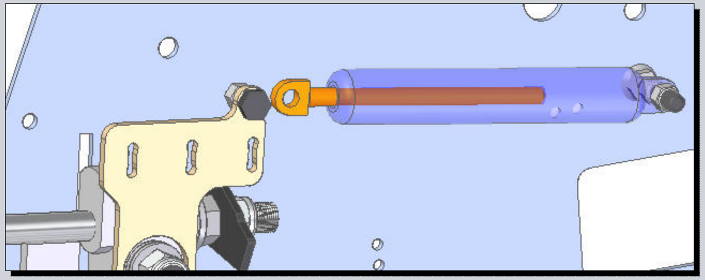
- This does not support a range of discrete values, only a specified Max and Min value

Return to Topics

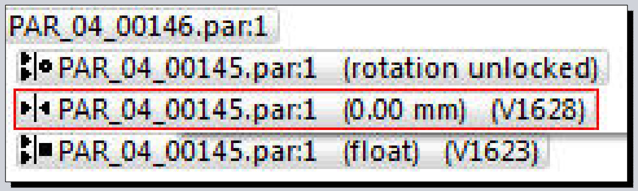
Range (Value Limits)



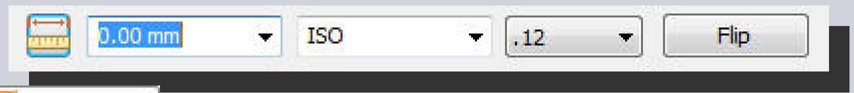
- We can use this cylinder assembly to see how this works
- When placed the pushrod in the shock, it was originally placed with a fixed relationship

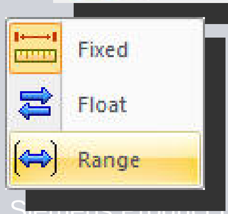


- With ST4, we can easily change this relationship to the new "Range" option
- To do so we can select the orange pushrod and then select the "mate" relationship from PathFinder

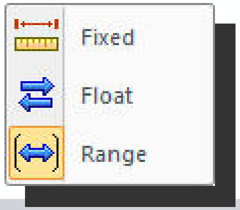


- Click on the "Fixed" constraint button and change it to "Range"



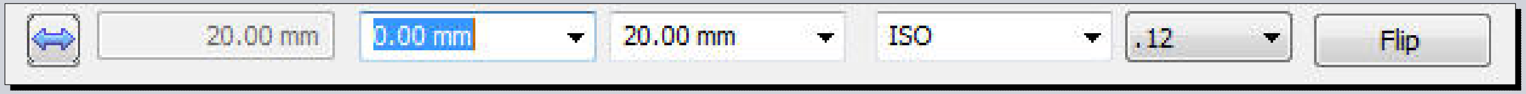


Range (Value Limits)



SIEMENS

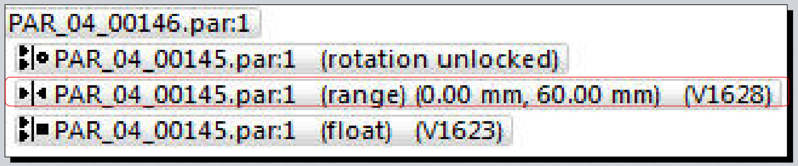
- When the “Range” option is selected the menu changes where the user can key in the minimum/maximum values



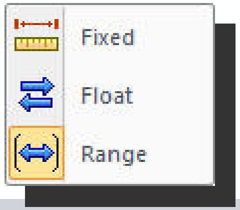
- In this dialog, the first field is always the minimum value, so it can be zero or any value to lift the pushrod off the bottom of the cylinder

- For this demo we can leave the minimum at 0mm and set the maximum to 60mm just to see how this works
  - NOTE: Use the “Enter” key to set the values

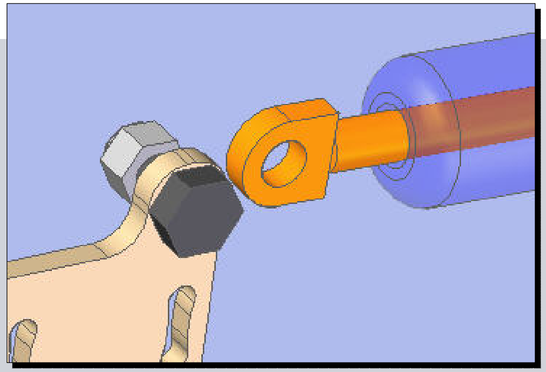
- When this is completes, you see that the relationship in PathFinder changes to the Range option



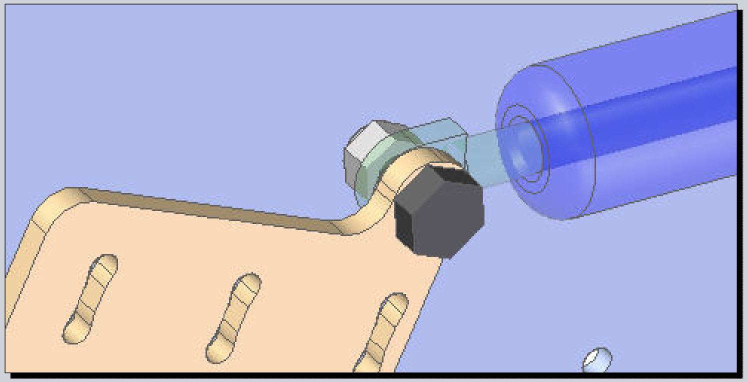
Range (Value Limits)



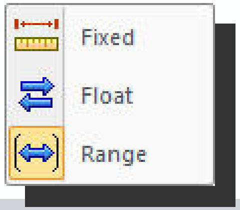
- Now that the range set, we can connect the two parts in the assembly by adding an axial align between the pushrod and the bolt



- Once they are connected, we can use the drag to see how the parts interact with one another



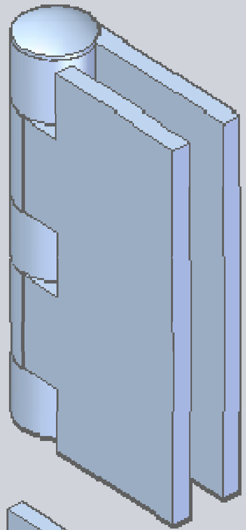
Range (Value Limits)

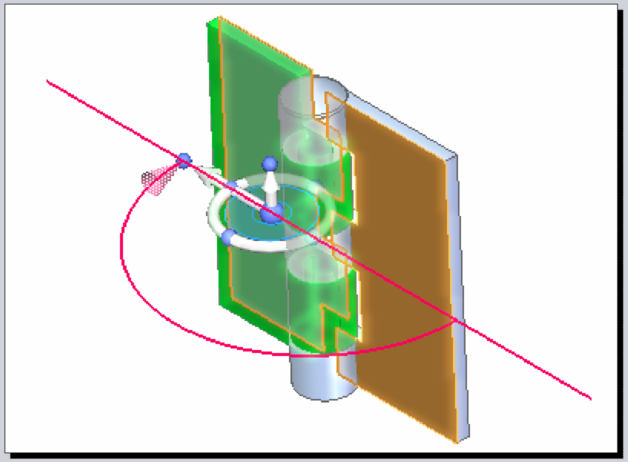


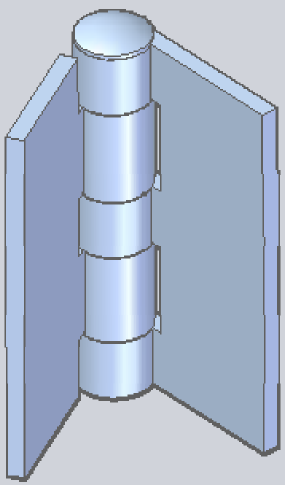
SIEMENS

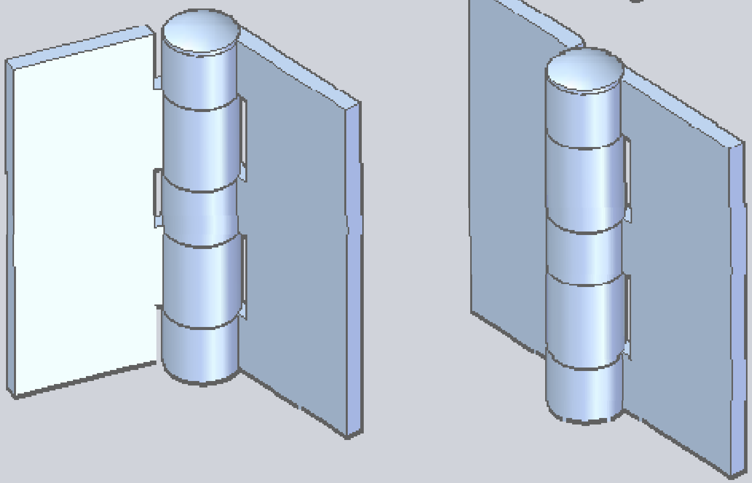
- Using "Range" with the angle relationship

- Here we have a door hinge setup with a range that will swing between 0 and 180 degrees

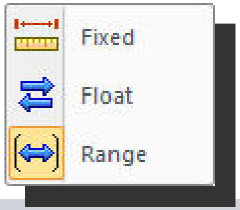








Range (Value Limits)



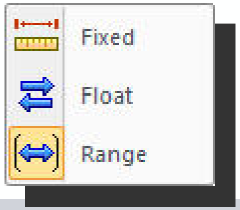
- Capture Fit supports relationship capturing and placement of range relationships
  - On placement the range definition will be placed along with the relationship

- For a given relationship the Range setting is the same across all FOA members
  - The solved value may be different between members but the Range definition is the same for all members
  - The range value controls are disabled if edits to all members is not set to “On”

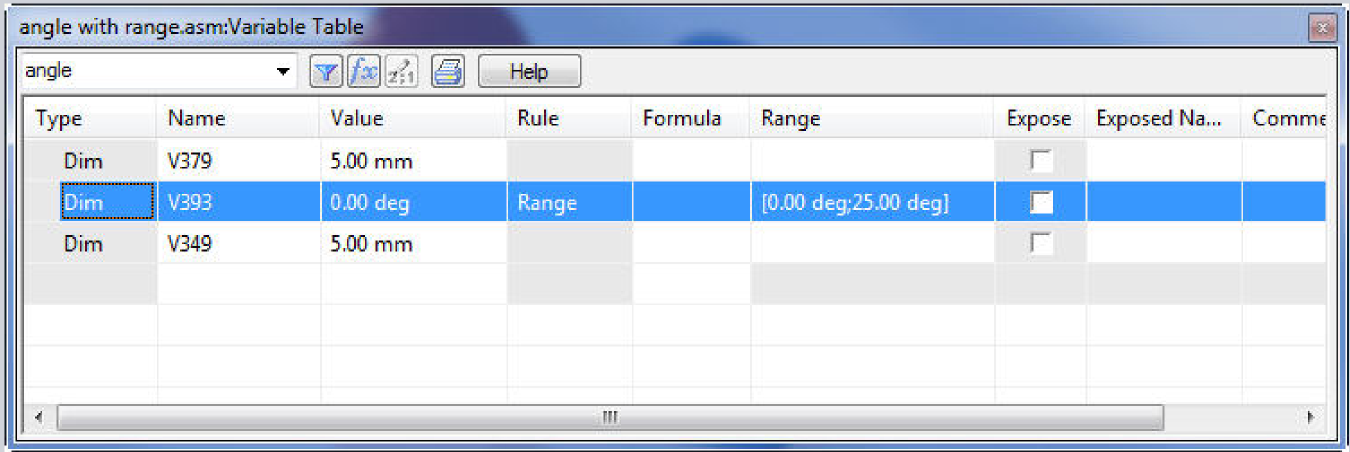
- The “Range” option works with relationships in adj sub-assemblies

Return to Topics

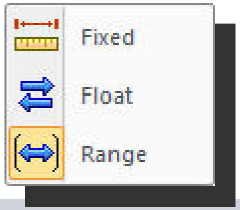
Range (Value Limits)



- Offset values defined with a Range will appear in VT as read-only values
  - The specified range will be shown in the Range Column using the Rule format.
  - The Value, Rule, Formula and Range Cells will be read-only
  - The Rule text will read “Range”

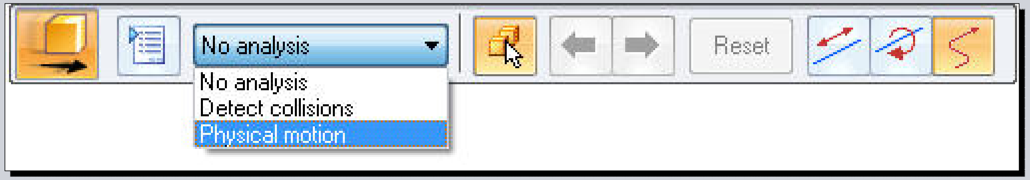


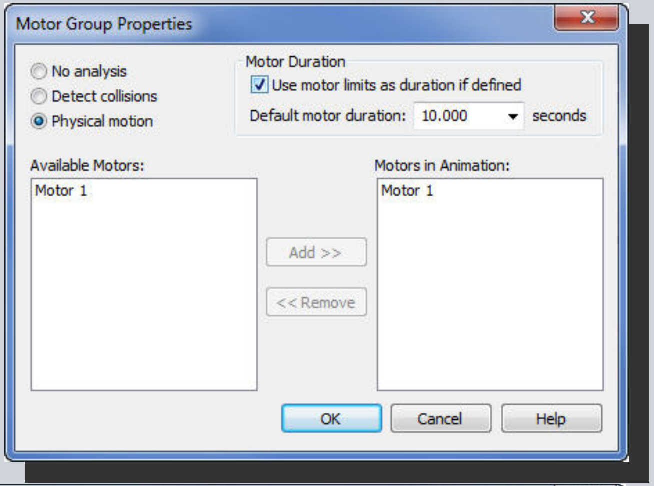
Range (Value Limits)



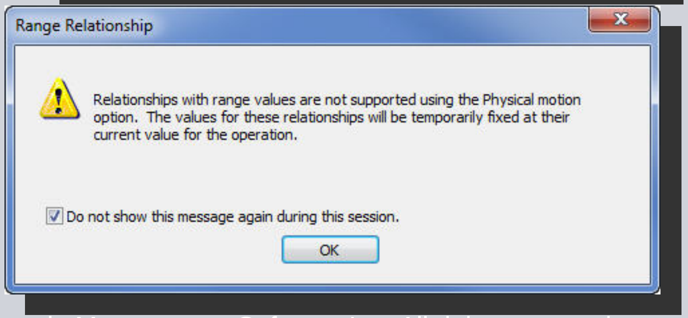
SIEMENS

- Range values only solve with “Drag Component” and “Motor Animation” using the “No Analysis” and “Detect Collision” options
  - The AEM “Physical Motion” option is NOT supported by DCM





  - When “Physical Motion” is used all variables that are set to “Range” are temporarily set to “Fixed” using the current computed values for the operation
    - If Range variables exist the following message is shown - after the operation the variable will be set back to “Range” and will solve using Range behavior

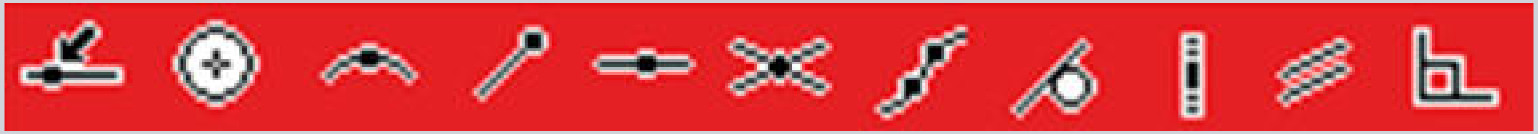


DEMO 2

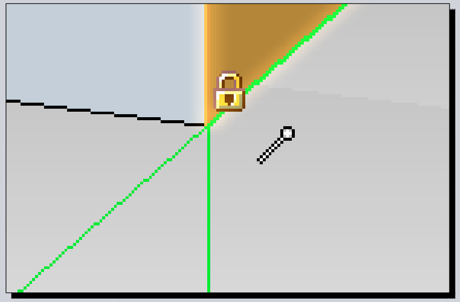
SIEMENS

Key-point Processing Improvements

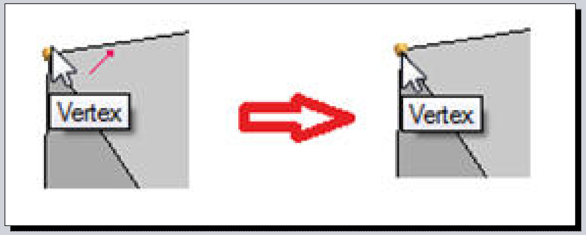
- General improvements have been made to key-points which impacts all areas of Solid Edge (Assembly, Part, Draft, etc..)



- Glyphs now have a Black with White border to improve their contrast and view-ability
  - Previous to ST4 the Glyphs were “Handle” color with no border

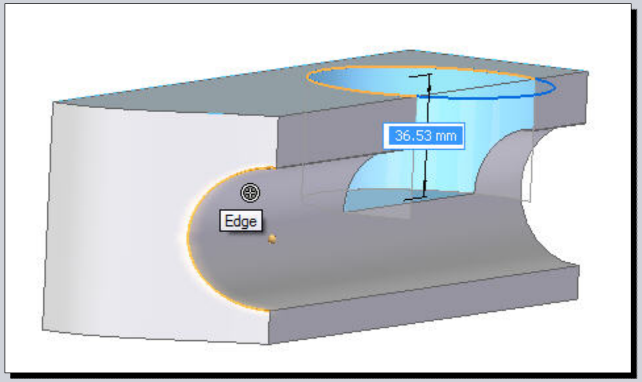


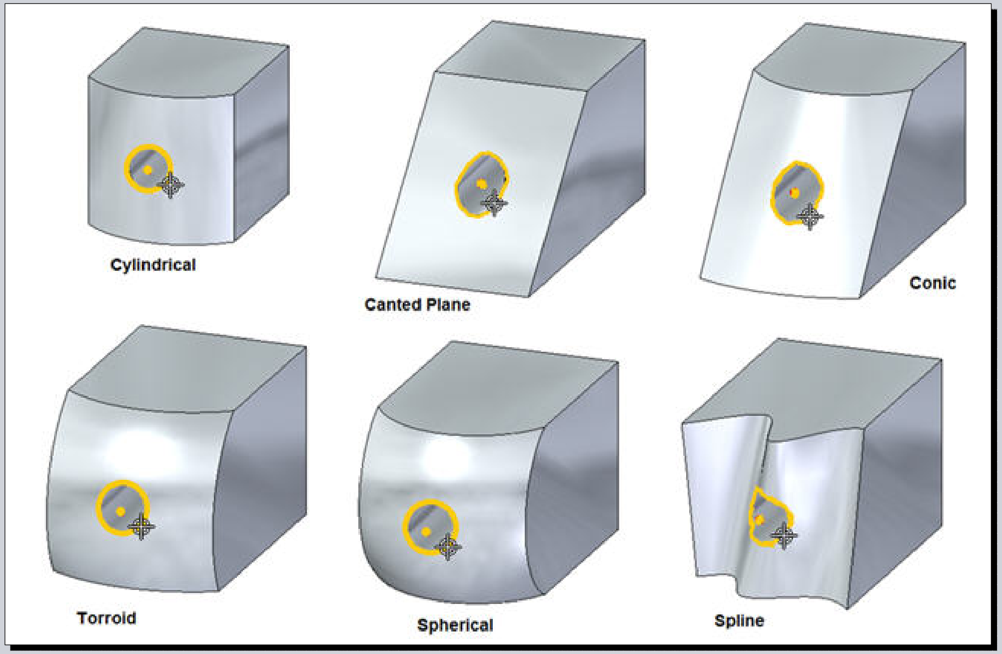
- Glyphs now persist on hover and do not “go away”
  - In many cases the glyph would present itself and then disappear



Key-point Processing Improvements

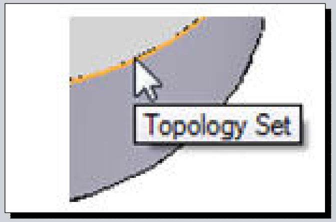
- Enhanced cylinder edge locate to find axis point for complex edges
- Previous to ST4, 3D Key Points used to define an extent did not return an axis point for the "center point" between a cylinder and a complex capping face intersection
- This works with cutouts, protrusions, and partial cylinders



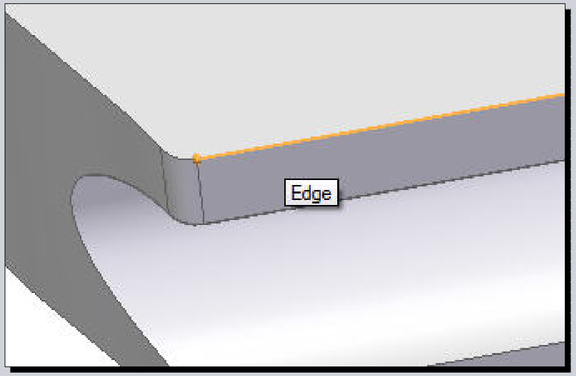


Key-point Processing Improvements

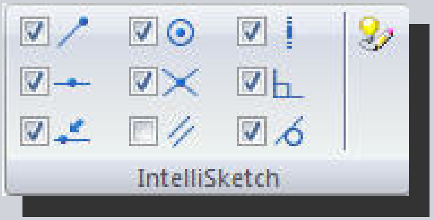
- There was a need to improve upon the “data tip” called “Topology Set” which many of us have seen when identifying an edge or key point



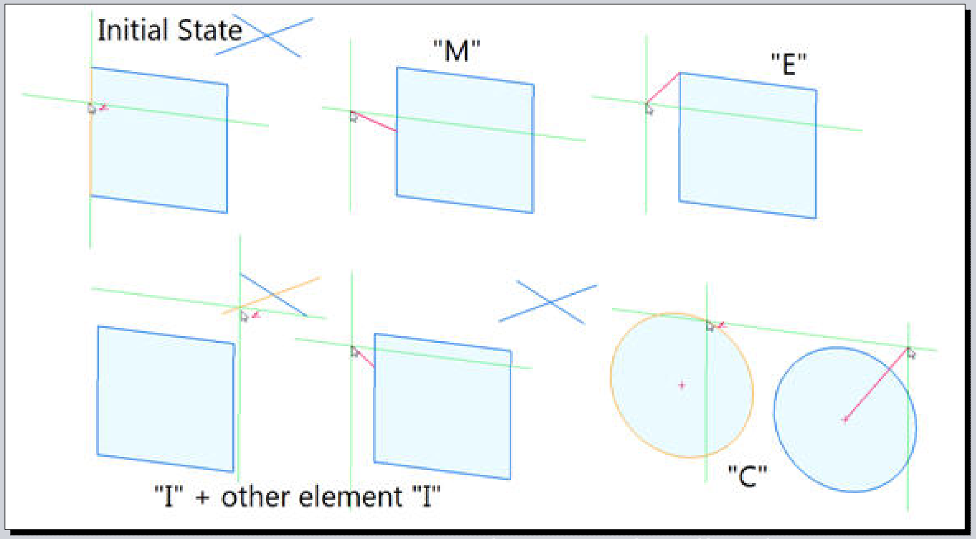
- This is typically seen in Solid Edge when using Smart Dimension, Extent/Rotate, Move, and so forth.
- In ST4, this message has been filtered and displays a more meaningful message for several elements types such as mid-point, end-point, center-point, edge, and so forth.



Key-point Processing Improvements



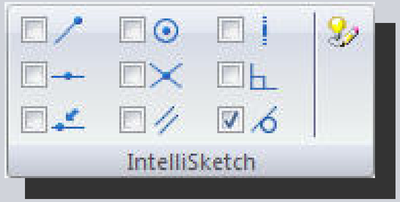
- Key Point filtering has not really changed in the Draft and Sketching environments, but wanted to make sure you were aware of the MICE hot keys
- If a user is in the line segment command and the cursor is on a current line, the “M”, “I”, “C”, “E” hot keys put the first point of the line segment on the mid, intersection, center, or end points of that line



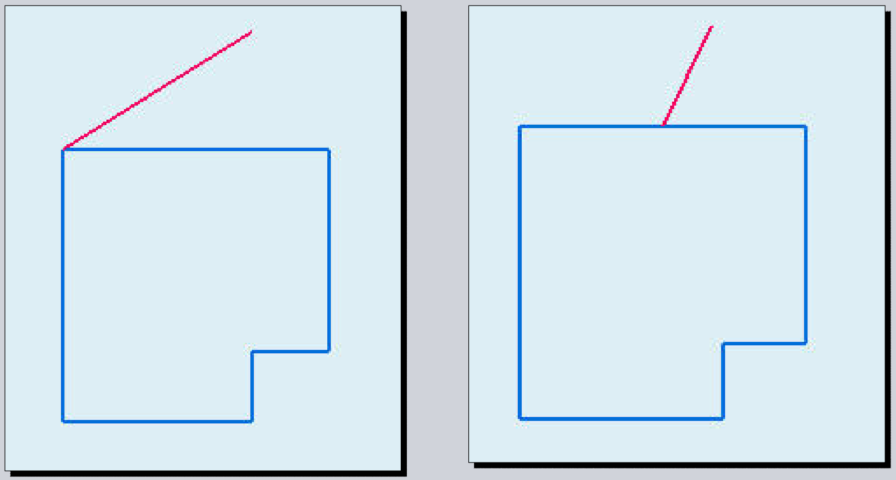
Key-point Processing Improvements

SIEMENS

- In ST4 if a user decides to turn off these options for his default (does not often occur), the hot keys still work

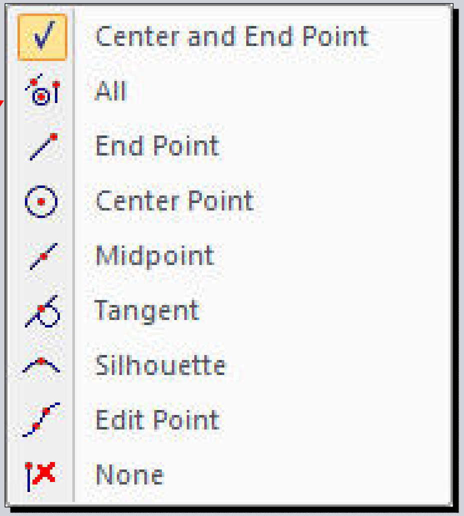


- The procedure would be to first select the line segment command, then enter the “Hot key” such as “E”, and finally to select the line segment already drawn and the first point of the new line would start at the end point



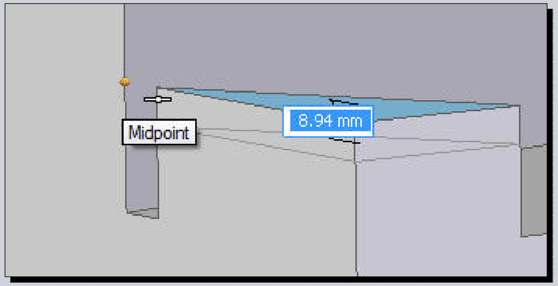
Key-point Processing Improvements

- The “Key-Point” filtering that is done through the pull down menus have been cleaned up to be more standardized throughout the product
- Added a new “Center and End Point” filter
  - It is also the new default setting
- Remove Silhouette Key-Point from "All” filter setting
- This cleanup involves removing useless key-point options where it makes sense

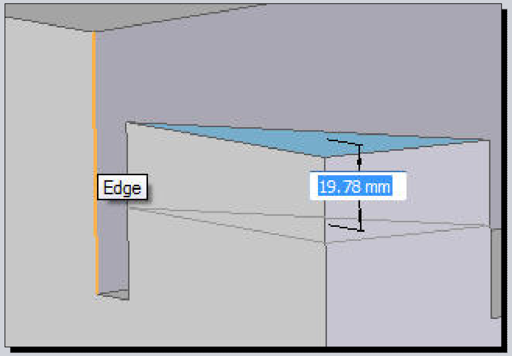


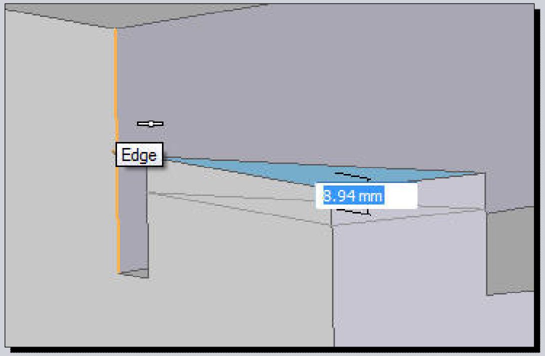
Key-point Processing Improvements

- 3D hot keys for key-point locations have been improved
- Currently, Solid Edge allows the user to select a face, use the steering wheel to lift the face, and then select a key-point on the model to define the height (still works today)



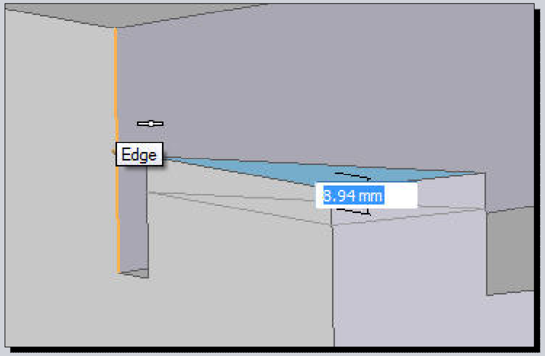
- The new 3D hot keys allow the user to use the “K” key to put Solid Edge in a state where the actual edge element can be identified and then the M.I.C.E. keys can be used to locate key points as shown





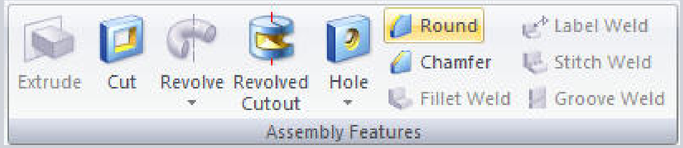
Key-point Processing Improvements

- The advantage to this technique is that you don't have to find the specific key point as the command will automatically pick it up

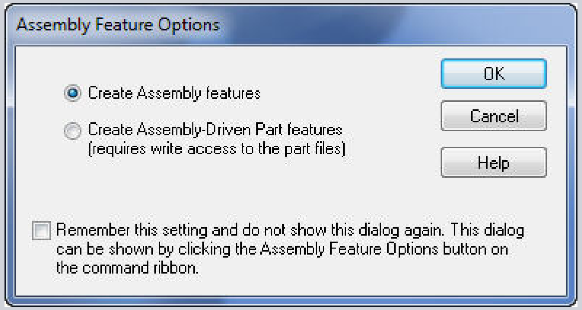


Round Feature in Assembly

- The “Round” feature has been added to the “Assembly Features” group



- When executing the “Round” feature, it gives the user an option to make it an assembly feature or a assembly driven part feature (ADPF)



- ST4 now allows these same options for Chamfer where in ST3 it only created a feature in assembly with no option

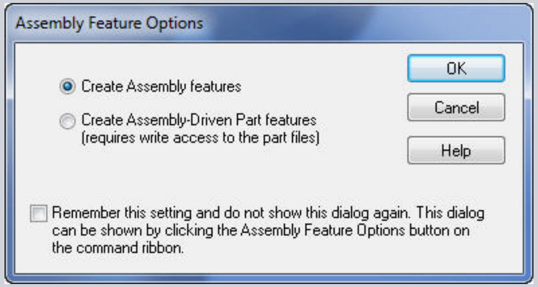
- Once an option is selected, the following ribbon bar appears

Round Feature in Assembly

SIEMENS

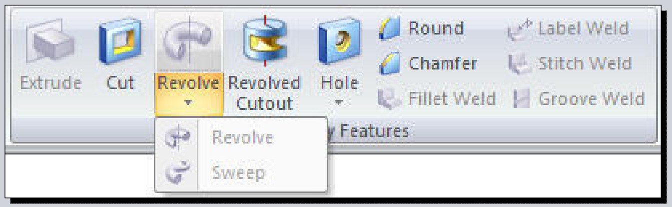
- A new option on Cutout, Hole, Revolved Cutout, Chamfer and Round assembly feature commands is the “Assembly Feature” options button  which was added so the user can easily make a change

- The main reason for the change is because there is a “Do not show this dialog” option



- If a user turns this off they can easily get it turned back on through the command bar

- The “Sweep” command was moved under the “Revolve” split button

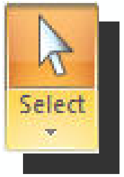


Return to Topics

Round Feature in Assembly

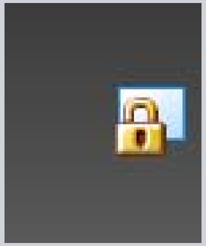
- A few things to know about the assembly “Round” command
  - The round feature only supports the constant radius option for AF and ADPF features
  - The command cannot select simplified edges/faces
  - There is no Live Preview of the command when it is created or edited
  - Edits of Round AF and ADPFs show dimension edit handles for the radius values
  - Edits of Chamfer ADPFs now show dimension edit handles like we do for the AF feature

Selection Performance Improvements



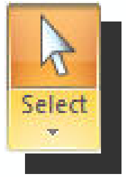
- Some work has been done to improve interactive performance with larger assembly data sets

- Solid Edge no longer uses the selection mode “on screen” glyphs
  - These glyphs are now attached to the cursor
  - Part/Face mode has been added to the cursor



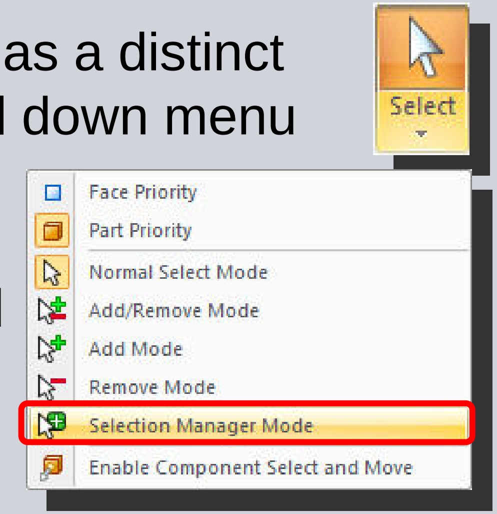
Selection Mode	Part Priority	Face Priority
Single Select		
Add/Remove		
Add Only		
Remove Only		
Selection Manager		

Selection Performance Improvements



SIEMENS

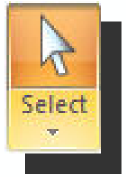
- The “Green Dot”  has been modified to work as a distinct mode the user selects from either the “Select” pull down menu or by a short cut key in (Shift + Spacebar)

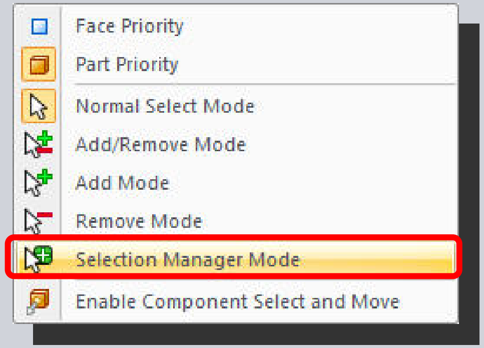


- Previous to ST4 when a face or part was selected the green dot would appear automatically when the cursor was moved over the selected part
  - One of the problems with this was locating the green dot or having to move the steering wheel if the green dot was behind the steering wheel

- The “Selection Manager Mode” was introduced in ST1, but was only available when a part or face was already selected

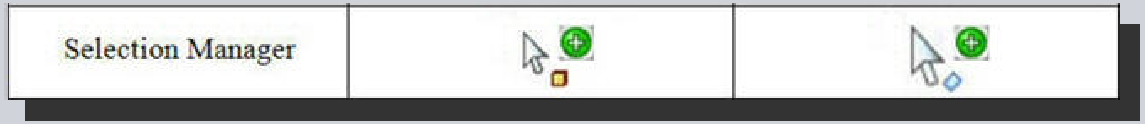
Selection Performance Improvements





- New with ST4 is the ability to enter “Selection Manager Mode” with no graphics selected

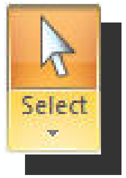
- When the “Selection Manager Mode” is entered Solid Edge will add the glyph to the cursor letting the user know the mode is active in both part and face selection mode



- Selection Manager still works the same with the exception the user no longer has to graphically click on the green dot to activate it

- It can be run with no parts, one part, or multiple parts and faces selected

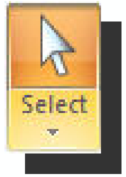
Selection Performance Improvements



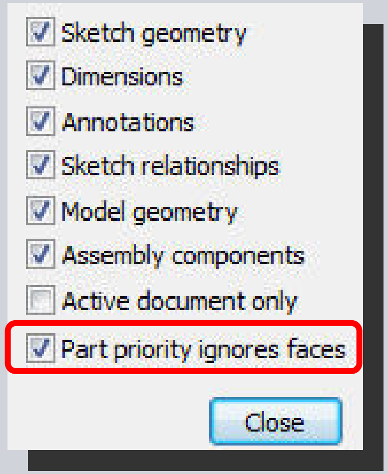
SIEMENS

- In ST4 Solid Edge selects an Active/Inactive or Simplified part while in the Green Dot mode
- You cannot select a simplified “Face” while in this mode
- Press the “Spacebar” to exit “Selection Manager Mode”
- Other keyboard option in assembly are:
- Press “Ctrl” + “Spacebar” to toggle between “Face” and “Part” mode
- NOTE: “CTRL Q” will show/hide assembly while IPA

Selection Performance Improvements

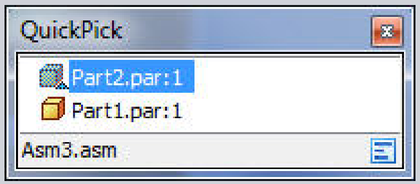


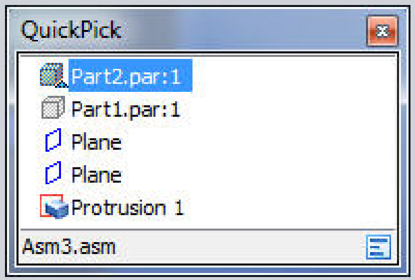
SIEMENS



- Found under the "Selection Filter"  is a new option called "Part priority ignore faces"
- This option is on by default and allows the filtering out of all faces/objects inside components
- In large assemblies this list can get very large which takes longer to process
- Notice the difference in the small two-part assembly with it on and off

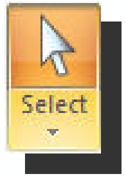
ON





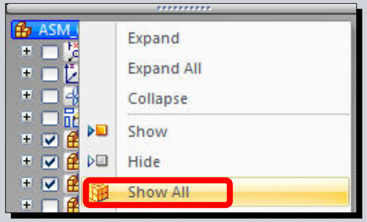
OFF

Selection Performance Improvements

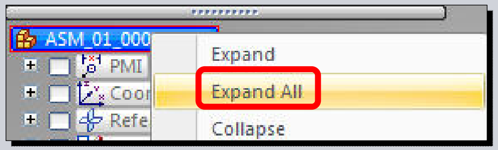


SIEMENS

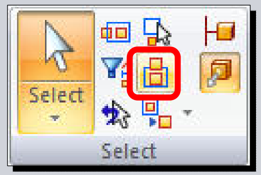
- More LARGE Assembly Performance enhancements……

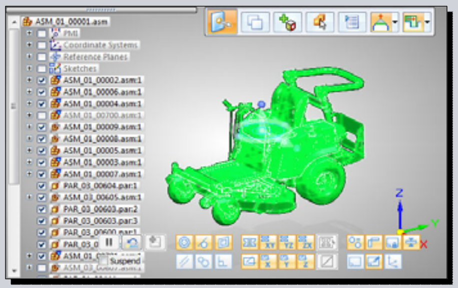


- When a user selects a “Show All” at the top level of an assembly, the PathFinder will no longer flash on and off during the process



- Things that are noticeably faster
  - “Expand All”
  - Select Visible with large selection is more than 3 times faster
  - Clearing the select set with “Esc” key
  - Fence select
  - View manipulation with select set





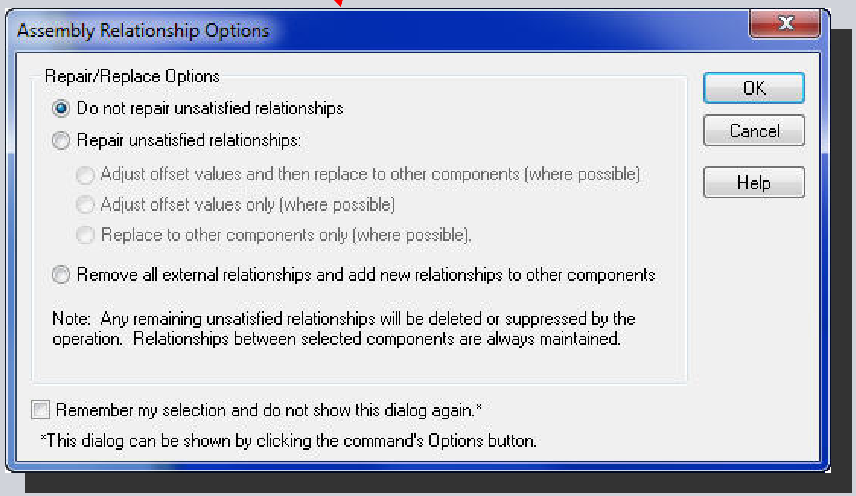
Steering Wheel in Assembly Improvements

- When selecting a part in an assembly, you will not only get the steering wheel, but also a few new options on Quickbar

Move - Copy

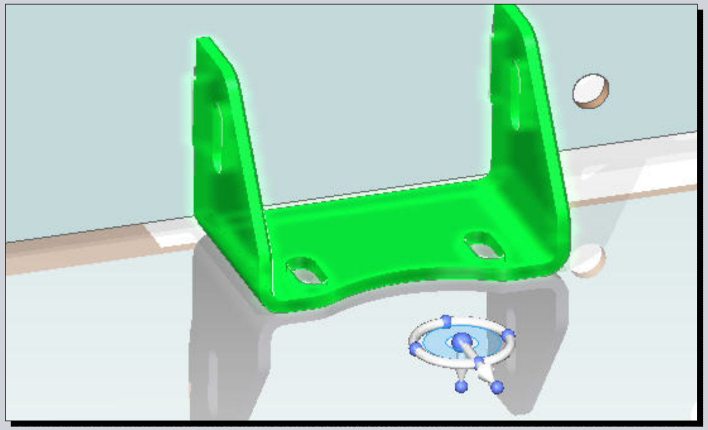
Move – Select Related Faces

Relationship Options



Steering Wheel in Assembly Improvements-Copy

- The “Copy Components” option  copies selected occurrences to a new location
- This is for occurrences ONLY - faces and other items in select set are ignored
- This operation is similar to the “Move Components” command except that it allows copies to be made of components in sub-assemblies
- When a part is selected and you want to set the steering wheel to a key point on the part you are moving for a precision move, remember the that the part has to be active in order for you to select key points



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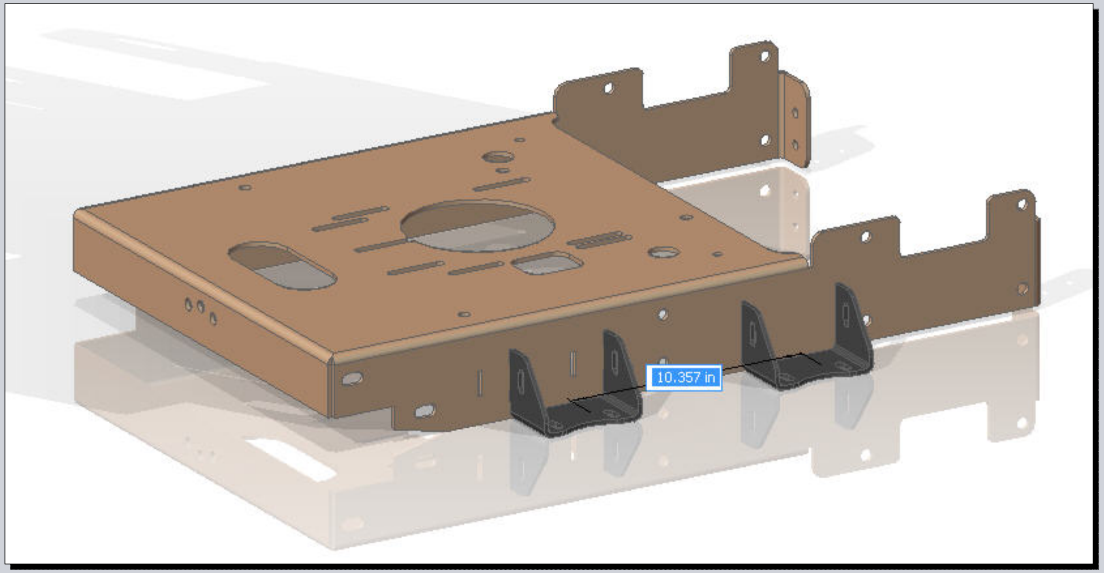
Steering Wheel in Assembly Improvements-Copy

- Here we simply want to copy the bracket along the frame using either a keyed in value, or by selecting another key point on the frame itself



- HINT – To suspend movement of the angled part so that it does not get in your way during placement, you can use CTRL-SHIFT-D to suspend movement

SIEMENS

Steering Wheel in Assembly Improvements-Copy

- Embedded occurrences (Frames, Pipes, Wire Harness) are supported
  - The operation creates a dumb body occurrence in the assembly that the copied geometry resides in
  - Note: The above component types cannot be moved, only copied
  - Selecting a frame component by itself will not show the Steering Wheel

- The operation can span multiple sub-assemblies
  - Copied occurrences reside in their source assembly

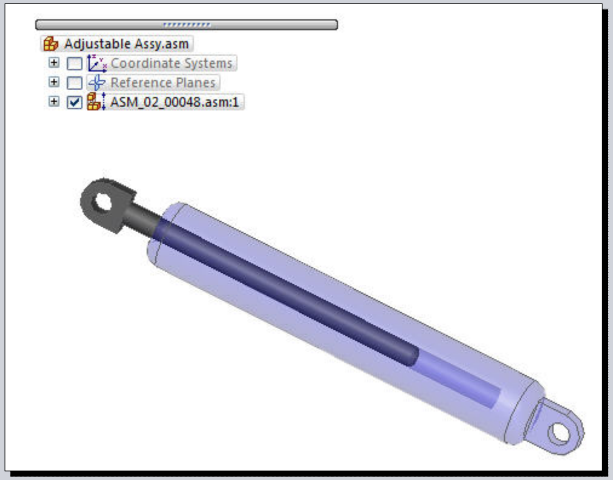
- No assembly or VTK solve occurs when “Copy” is used

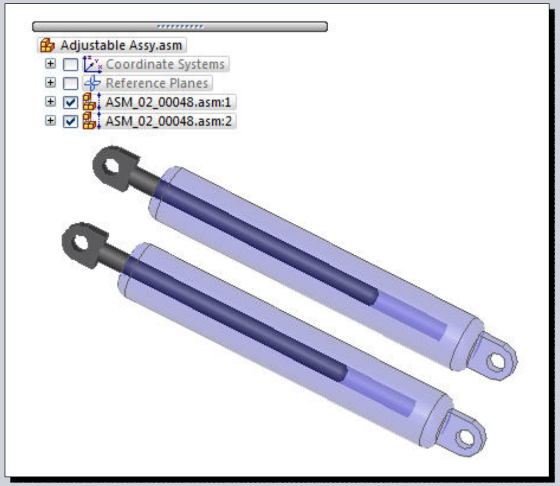
- No PathFinder group is created by this operation

Steering Wheel in Assembly Improvements-Copy

- Adjustable Parts: A copy of the adjustable part is made using the same measurement variables and/or values
  - Result is a copy of the as computed occurrence

- Adjustable Assemblies: Based on the time and technical issues the copy of these occurrences will be supported as follows:
  - They will copy in their "as adjusted" state and be marked as adjustable



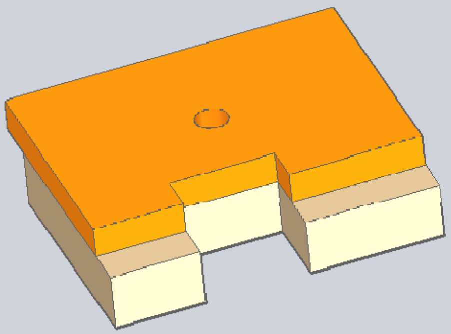


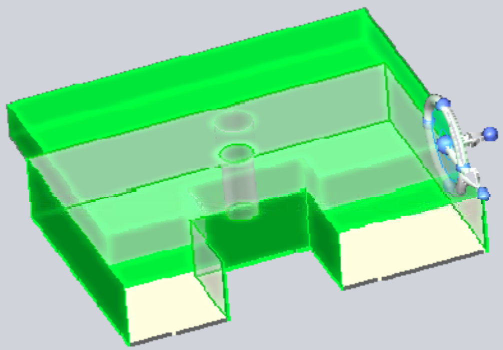
DEMO 3

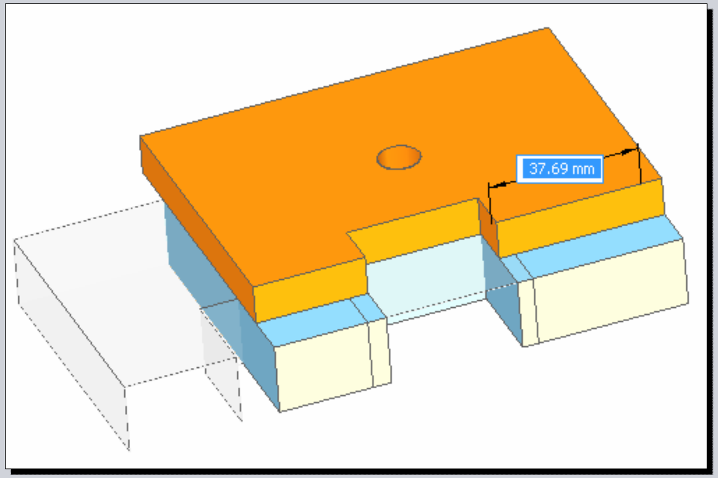
Steering Wheel in Assembly Improvements-Relate

- The “Select Related Faces”  option processes occurrences in the current select set (orange part) and selects *synchronous* faces that are related via assembly relationships

- This action is only an aid in selecting components and faces to move at the same time





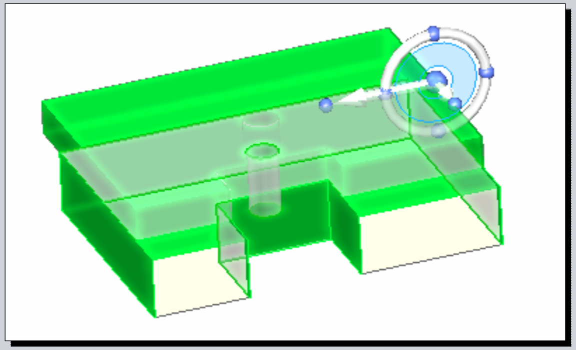


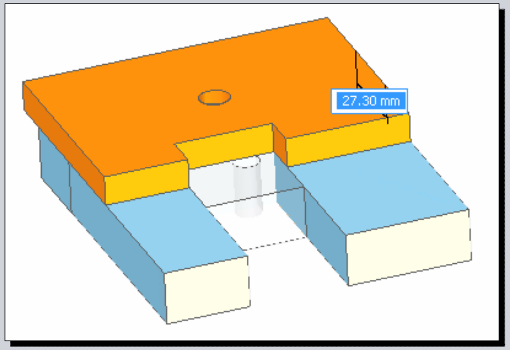
Return to Topics

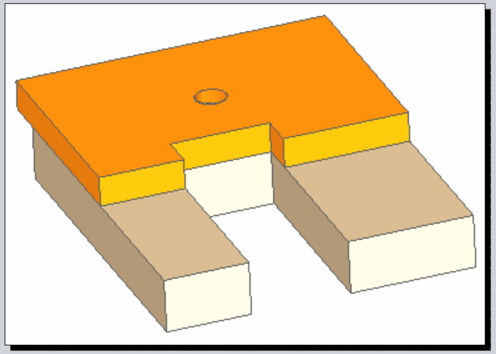
SIEMENS

Steering Wheel in Assembly Improvements

- These parts are synchronous parts that have several mates, an axial align, and planar align assembly relationships between them



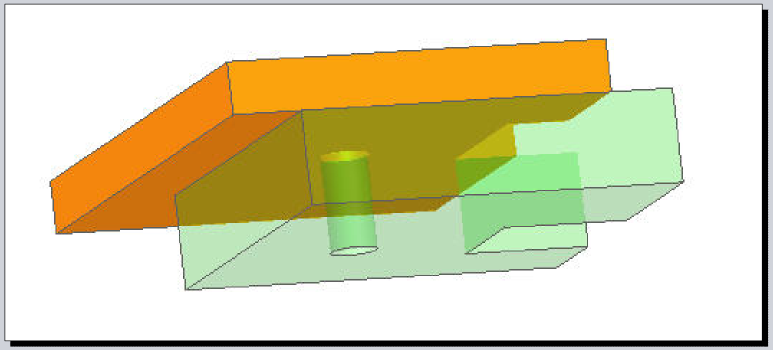


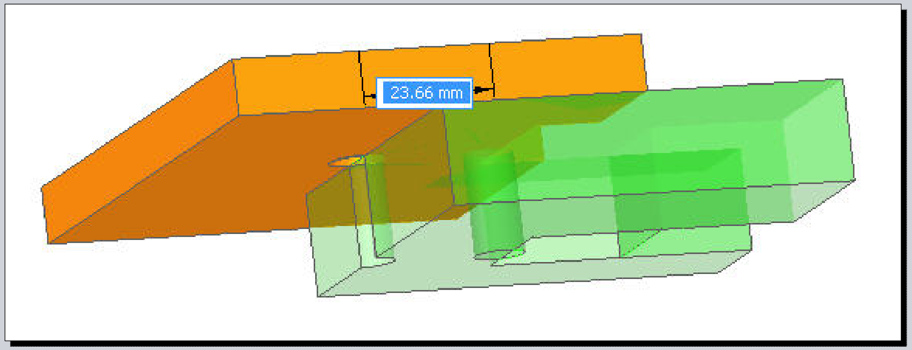


- Using the steering wheel to move the top part will allow both parts and related faces to adjust to the move

Steering Wheel in Assembly Improvements

- Notice what occurs if I move the top part to the right
- The related hole follows, but there is no side face relationship to maintain on the bottom part, so the orange part extends past the green part





SIEMENS

Steering Wheel in Assembly Improvements-Relate

- The faces are selected from within the non-selected components
- The “Select Related Faces” option can be used to build select sets
- The result is the same as if you ran Green Dot (Selection Manager) on every occurrence and selected the related faces
- Processing is the same as ST3 with the equivalent select set
- Note: This option is disabled if the “copy” option is selected

DEMO 4

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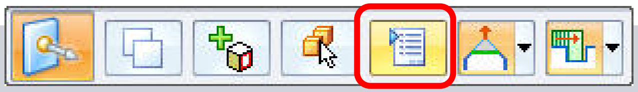
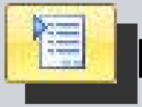
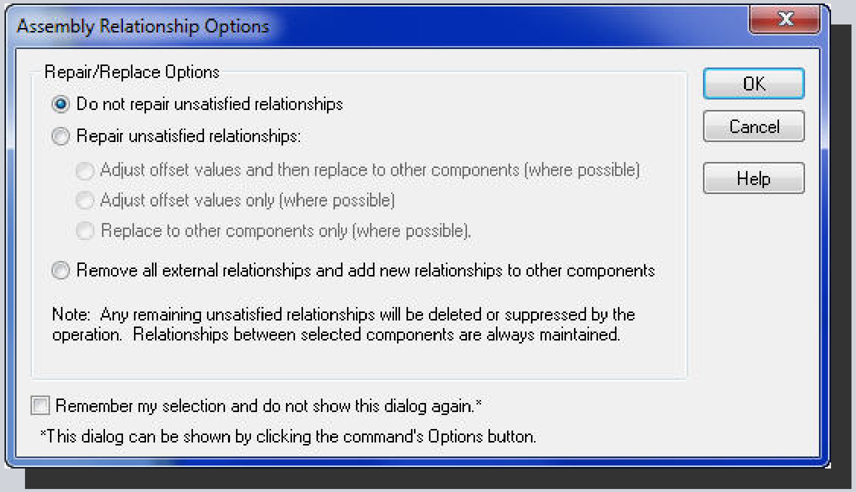
Steering Wheel in Assembly Improvements-Options



- The “Relationship Options”  have been added to the Move command to find and repair assembly relationships for affected components

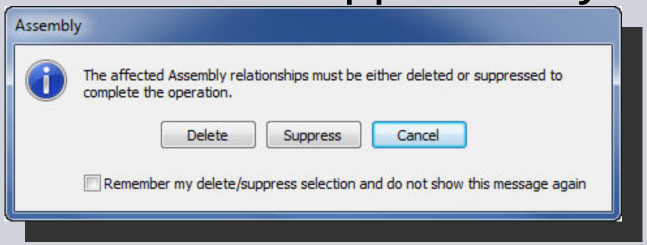
- These options control how assembly relationships are processed after the “Move” operation is complete

- The dialog shows the ST4 default settings and the “Do not repair” option is the same as what occurred in ST3



Steering Wheel in Assembly Improvements-Options

- If the “Do not repair” option is set, the following dialog will appear as it did in ST3 giving the user an option to either delete or suppress any relationships that cannot be maintained

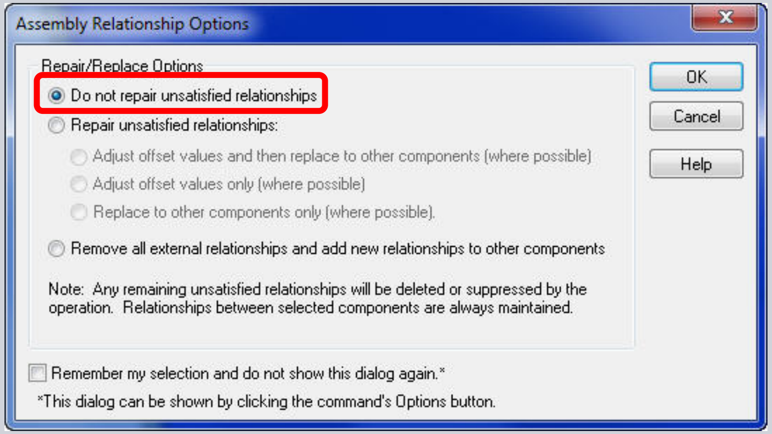


- These “Relationship Options” allow the user to process unsatisfied relationships (Float or Find) or to remove and attempt to replace all relationships using find

- ST4 attempts to find geometry on other components to repair them, but only relationships supported by “Relationship Assistant” are found
  - They are Mate, Planar Align, Axial Align, Connect and Tangent
  - Other relationships on the components are either set to float, suppressed or deleted by the operation

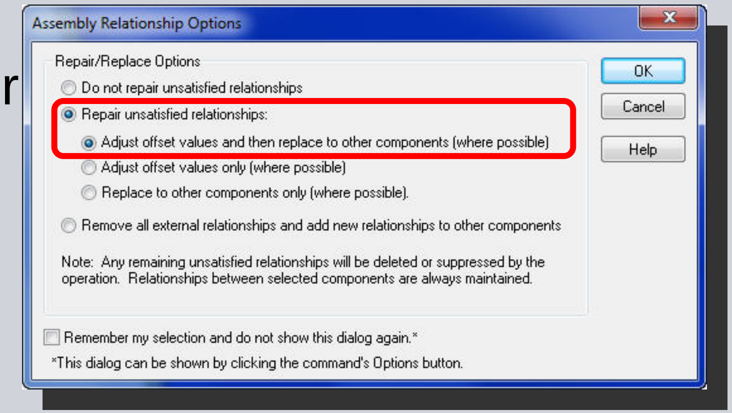
Steering Wheel in Assembly Improvements-Options

- A closer look at each of the options provided through the ARO dialog



- The “Do not repair unsatisfied relationships” option is the same as what occurred in ST3

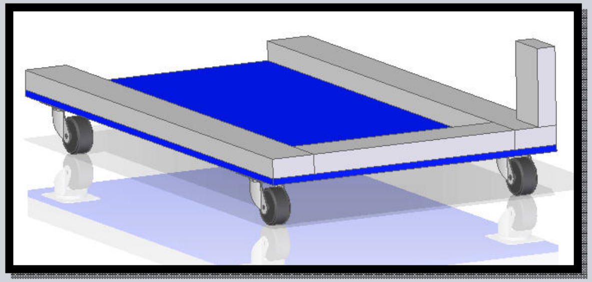
- “Repair unsatisfied relationships” offers 3 sub-options for the user to consider
  - Satisfied relationships remain the same



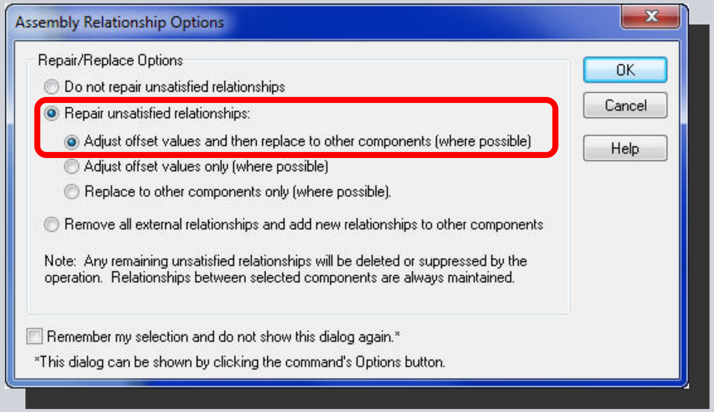
- The first relationship to discuss is the “Adjust offset values and then replace to other components (where possible)”

Return to Topics

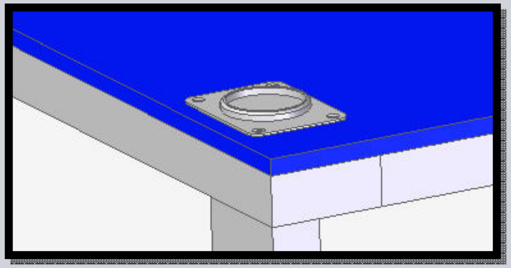
Steering Wheel in Assembly Improvements-Options



- We can use the roller table assembly to see how this first option works
- The first option basically will offset any relationships it can and if any are still unsatisfied, then it will replace to new components where possible



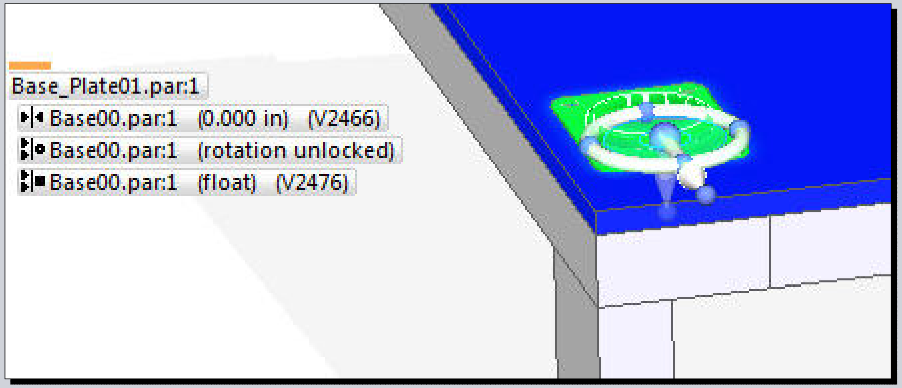
- If neither can be satisfied, then the user will get the “delete or suppress” option for those unsatisfied relationships
- In this example we want to move the swivel plate to the other side



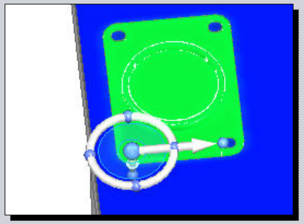
SIEMENS

Steering Wheel in Assembly Improvements-Options

- This part has three relationships
  1. Face to face mate
  2. Edge to edge floating face align
  3. Hole aligned



- We can set the steering wheel to the center of the aligned hole as shown



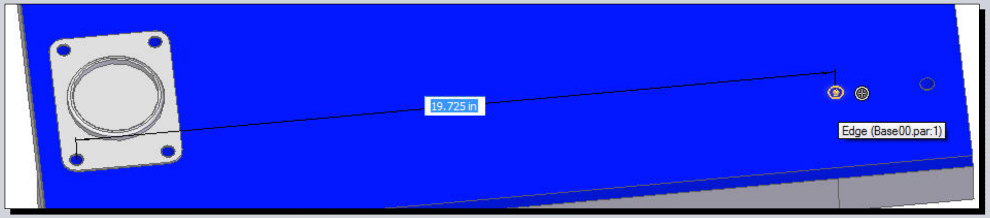
- Select the “Copy” option from Quickbar

- Select the primary axis to move/copy the part to the other side
- Also, use the CTRL-SHIFT D keys to suspend movement

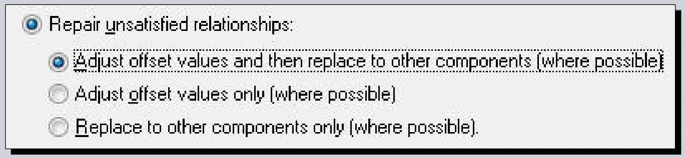
Return to Topics

Steering Wheel in Assembly Improvements-Options

- As we drag the cursor to the opposite side, we can easily locate the hole we want to align to as shown

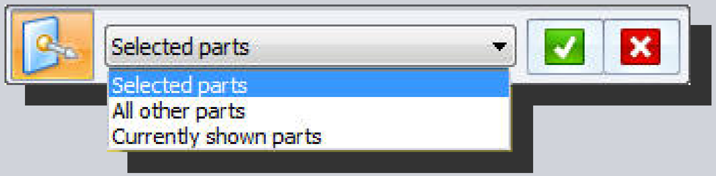


- With the option we are using, if a relationship cannot be satisfied with an offset (hole alignment in this case), then it will try to replace to new component



- When this occurs, Solid Edge will give the user the option to select the parts in which they want to have searched for a new face or in this case the hole

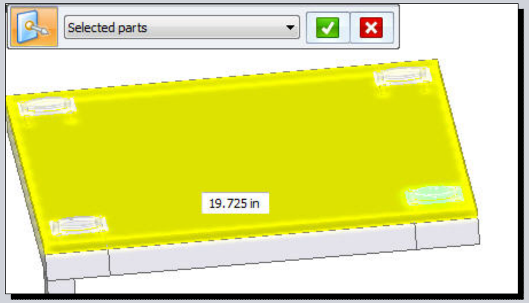
- These are the options for part selection



Return to Topics

Steering Wheel in Assembly Improvements-Options

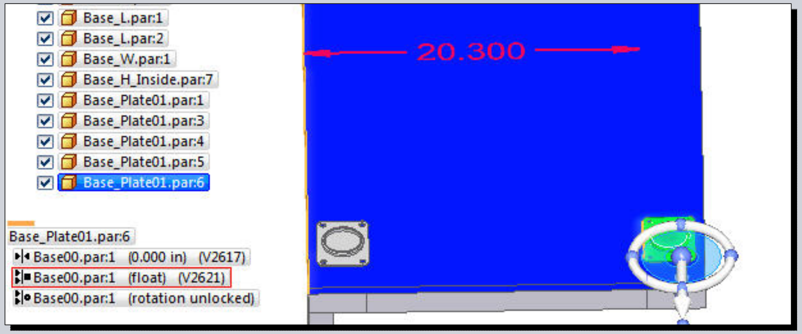
- Because we are working off the same part we can simply select the base part as shown highlighted in yellow and then select the green checkmark to accept



- When we do this, the part gets placed with the appropriate relationships

- The result from this first option is the face mate did not change as it was never unsatisfied

- The floating offset relationship was satisfy by extending the offset to 20.30



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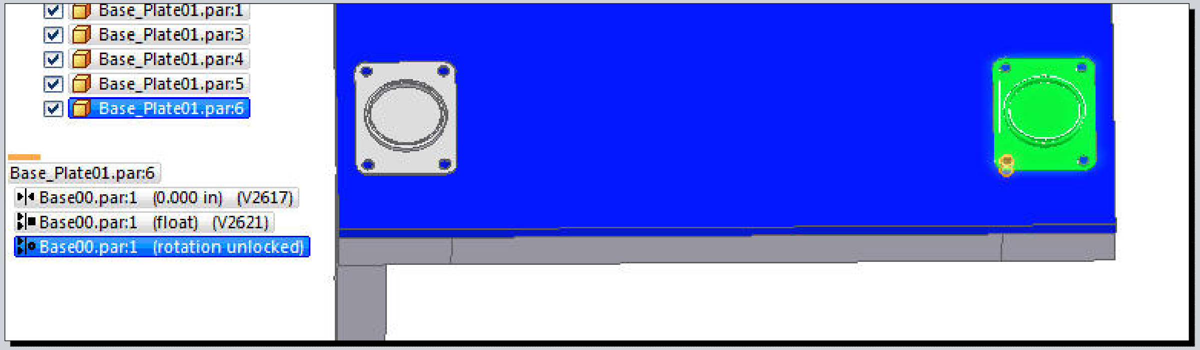
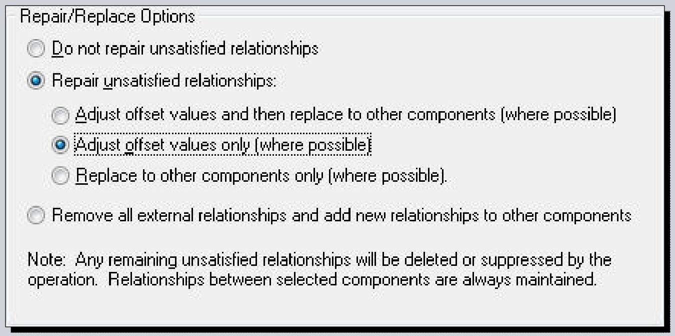
Steering Wheel in Assembly Improvements-Options

- The hole alignment could not be satisfied with an offset, so by selecting the blue plate and allowing it to search for a new face it reset the hole alignment to the new hole as shown



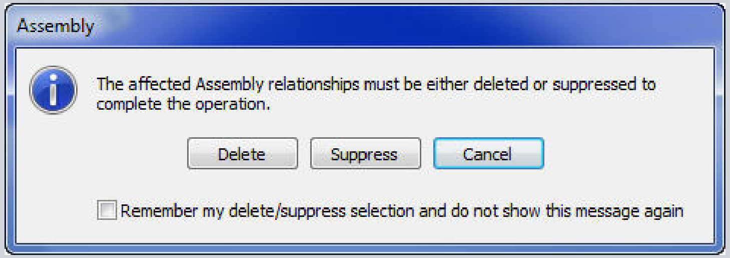
- We can now use the exact same scenario with the same parts, but using the 2nd sub option to “Adjust offset values only (where possible”



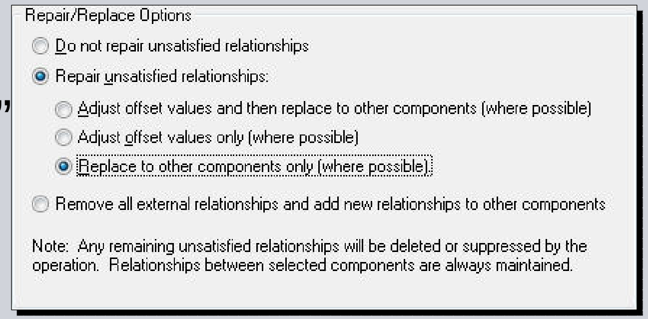
Steering Wheel in Assembly Improvements-Options

- Using this option, we know that the mate relationship is still satisfied, and the aligned edge can be satisfied as it was in our first example. But what happens with the hole alignment?

- Solid Edge cannot satisfy the hole alignment, so it gives the user the option to delete or suppress the relationship



- Of course using the 3rd option to “Replace to other components only (where possible)” does not allow the offsets, so any unsatisfied relationships will get the same delete or suppress option as above



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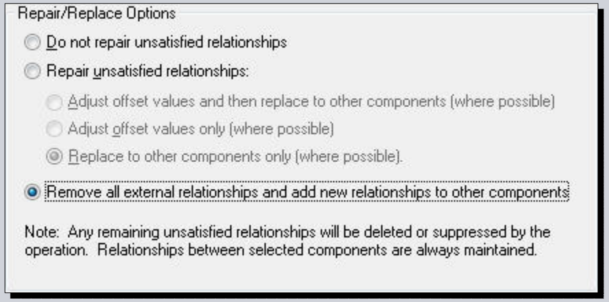
Steering Wheel in Assembly Improvements-Options

- A few things to know about the “Float” repair
  - Processing is such that affected fixed offset relationships are set to floating offset, satisfaction is tested and if it succeeds it is set back to fixed offset with the newly computed offset value
  - During this process all components are fixed in the newly translated position and will not move
  - If satisfaction fails then the relationship must be deleted or suppressed

  - Processing of Range Offset values is such that the value is set to float
  - If the relationship is satisfied the offset is set back to Range using adjusted limit values for the Range otherwise it must be deleted or suppressed
  - Only one range limit value is adjusted

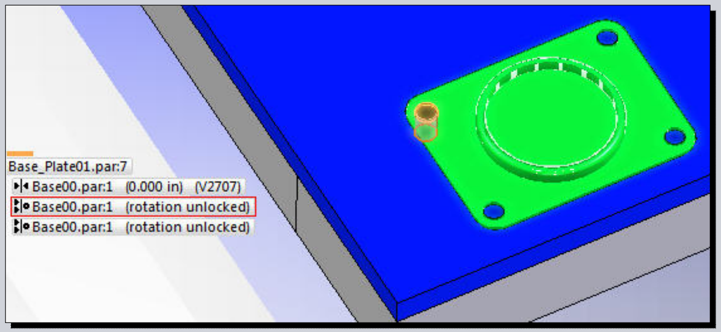
Steering Wheel in Assembly Improvements-Options

- The last repair/replace option we need to look at is the “Remove all external relationships and add new relationships to other components” options



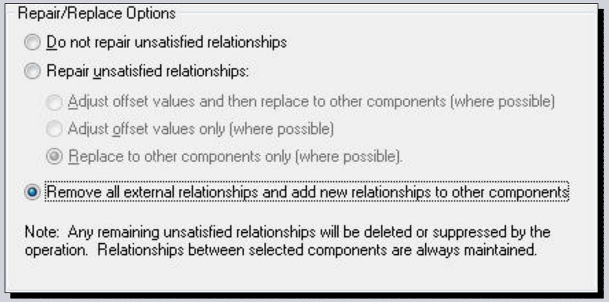
- This basically means we want to create new relationships and delete of the old ones, so how does this work with this same plate we have been working with?

- In this case after selecting the blue base part as the part to search for new relationships, Solid Edge places a new mate, and two new hole alignments



Steering Wheel in Assembly Improvements-Options

- For this option, all external relationships from the selected components are removed and new ones added using ACM



- It removes all external relationships to the select set (including grounds)
- Internal relationships are maintained
- New relationships get added if ACM finds them
- If the part had no relationships to start with, it may have some added at the end of the operation when this option is on
- The delete/suppress dialog is not used when this option is selected

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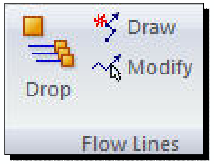
Steering Wheel in Assembly Improvements-Options

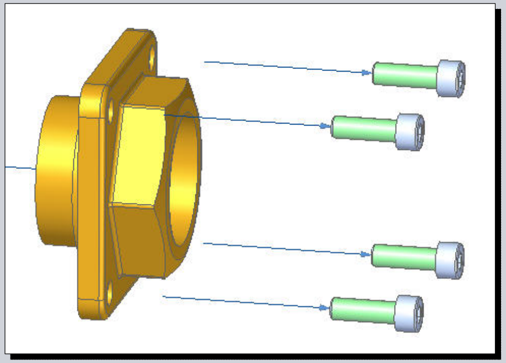
- FOA and AP Assemblies restrictions apply based on the global/local mode setting
  - Note: FOA /AP Sub-ASM cannot be modified, but can move as a unit
- FOA/AP - Move
  - Supported in Global/Local, but no repair
- FOA - Copy
  - Supported in Global/Local, but no repair
- AP – Copy
  - Supported in Global, but no repair
  - Not supported in Local
- FOA/AP – Delete/Suppress Operations
  - Supported in Global, Deletes/Suppresses the relationship(s) in all members
  - Limited in Local - Delete and Cancel are the only options – Suppress is disabled

DEMO 5

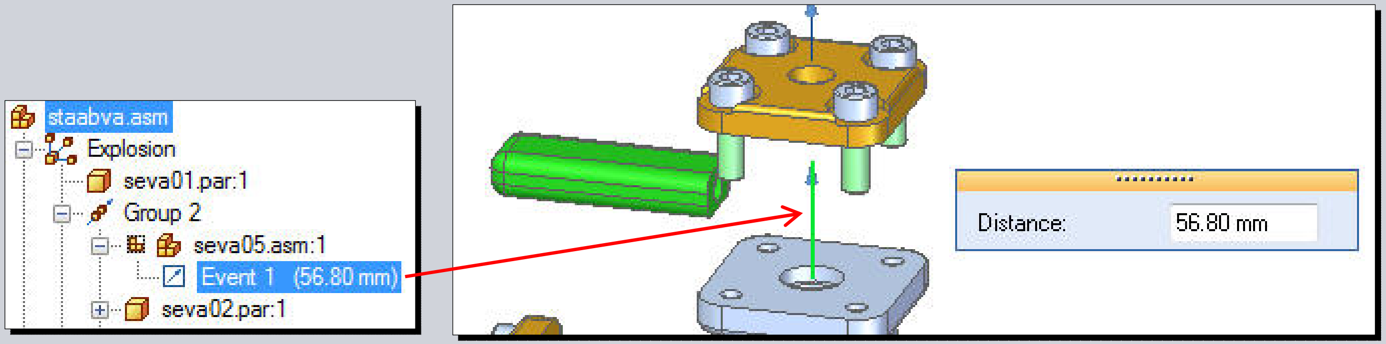
Return to Topics

Explode – Flow Line Enhancements

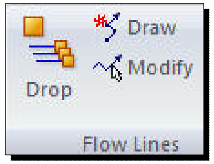




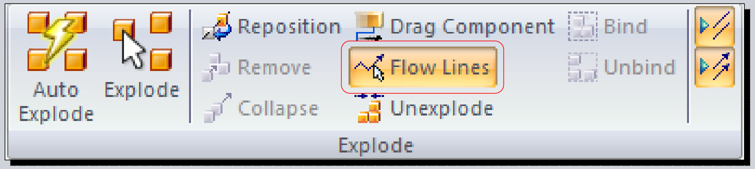
- With ST4, we have a new Annotation Flow Line command
- This new command allows the user to draw flow lines for Draft and Drawing documentation purposes
- These annotation flow lines are for display and drawing view use only and are not in any way associated with explode events
  - The auto and manual explode commands create events used in animation



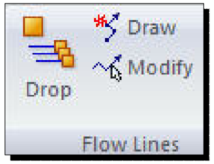
Explode – Flow Line Enhancements



- These flow lines are driven by the location and connection points to the components and update as the components are moved
- The Flow Line commands are disabled until an explosion has occurred or a component is moved
- Prior to ST4, the user could create an explosion and the flow lines were created automatically with very little user control
- We offered a Flow Lines command which basically allowed the used to extend the flow line, drag the flow line, or make adjustments to the shape of the flow line, but this was fairly limited



Explode – Flow Line Enhancements



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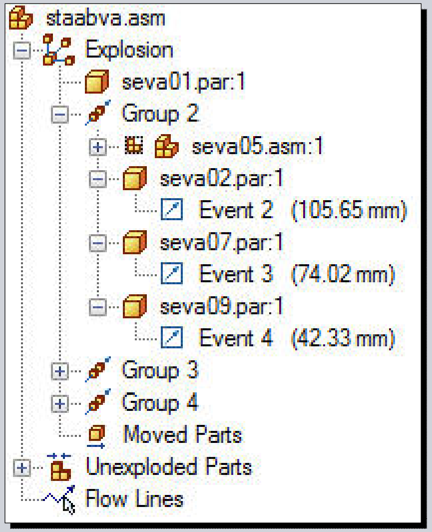
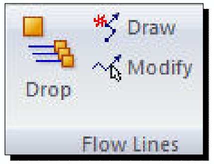
- The idea behind the new flow line command is to provide an easy to use tool to assist the user in manually creating the flow lines they want for draft and drawing documentation purposes as mentioned earlier
- A typical workflow to using this tool is to first create an exploded view using either the auto or manual method
- When this is done, Solid Edge creates “Explosion Events” in Explode PathFinder



Explode – Flow Line Enhancements



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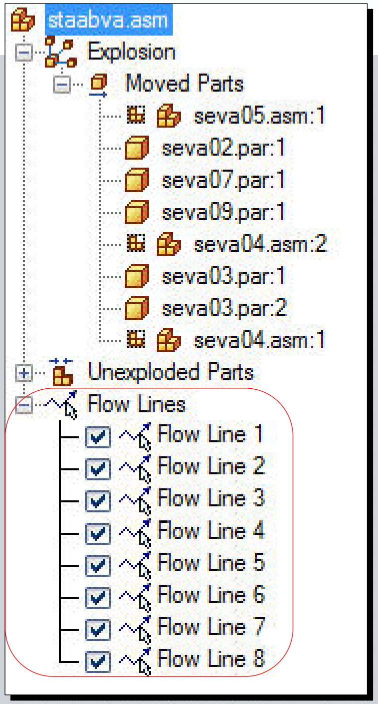
- Once the explosion is created the user can select the “Drop” option which will simply drop the flow line elements from “events” to “flow lines” and then place them in a “Flow Lines” collector in PathFinder as shown

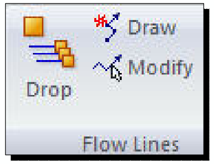
- One suggestion before dropping the flow lines would be to get the parts as close to the position as you want them
  - The manual explosion command is the easiest way to adjust things while the flow lines are still aligned



- Once this is done, then you can drop the “Event” flow lines and continue with the changes you might want to make

Return to Topics

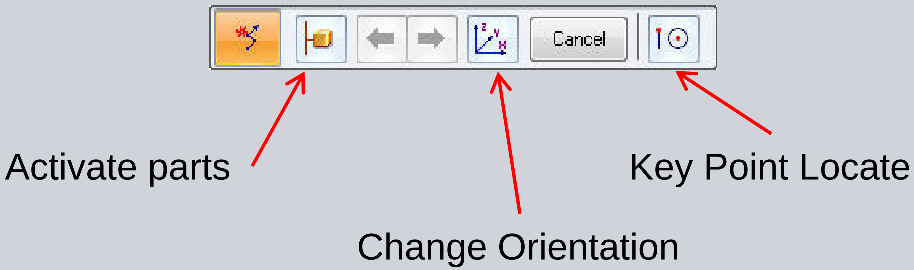
Explode – Flow Line Enhancements



- At this point the user can simply use the "Modify" command to modify the current flow lines or delete these flow lines and "Draw" new ones

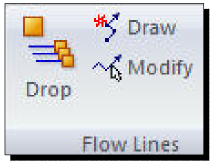
- Let's take a look at how we can use these new tools to create and modify flow lines

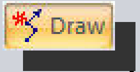
- Select the "Draw"  command and the following toolbar will appear



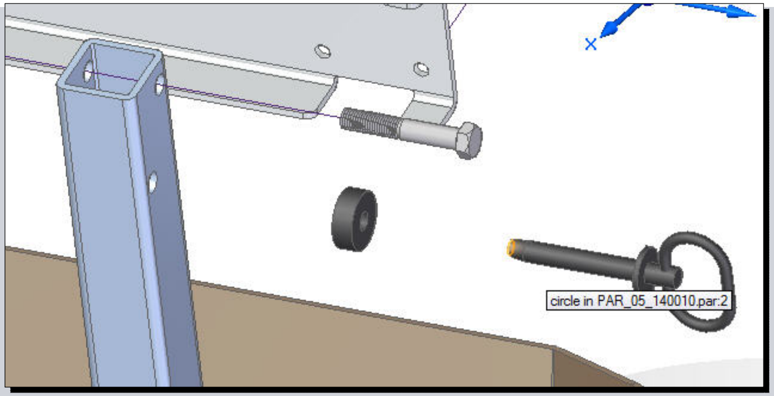
- The options on Quickbar are available to assist the user in setting up any options before starting the new flow line

Explode – Flow Line Enhancements

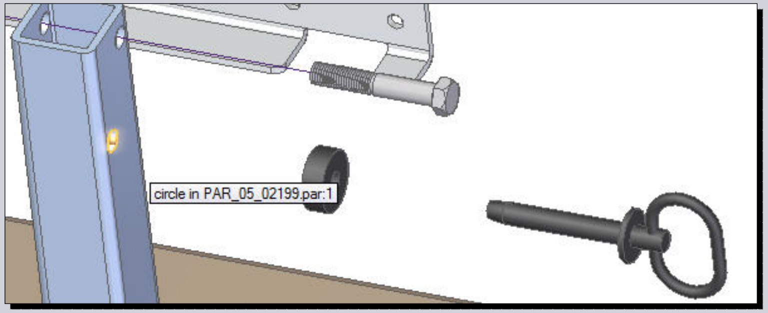




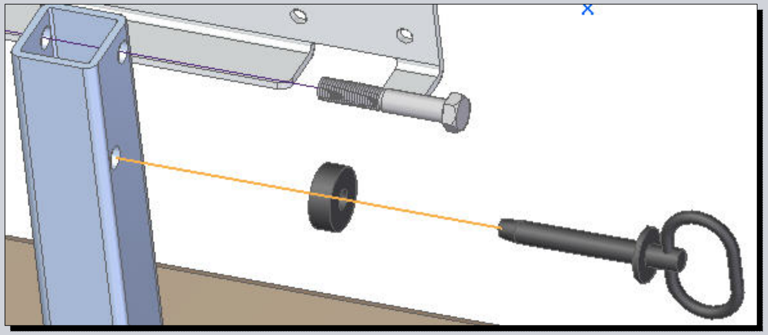
- If we want to draw a flowline between the key pin and the angled blue bar, simply click on the "Draw" button, then select the end of the key pin as shown



- The next step is to select where you want the flow line to go to, so in this case you could select the hole

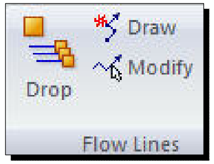


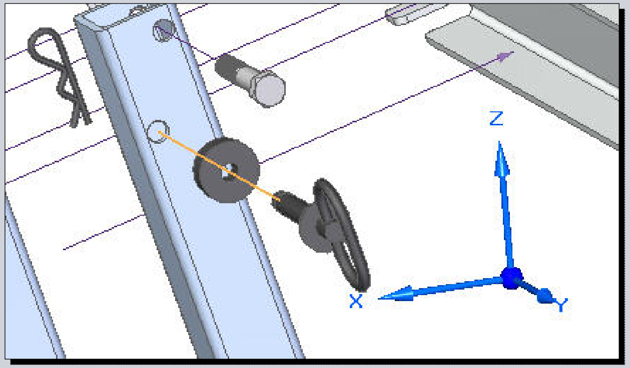
- Once selected, the flow line appears



- Notice the option for Port Segment Length?

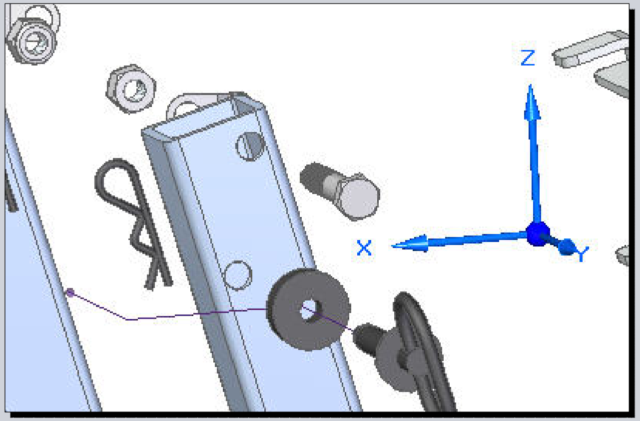
Explode – Flow Line Enhancements





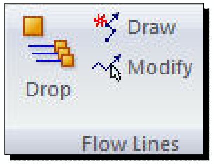
- A key thing to point out here – notice the triad and its orientation?

- If we create the flow line at this orientation and then decide to put a jog in this flow line, it will jog out at this same orientation instead of at the angle of the blue square pipe



- To create a flow line at a specific angle, we could use the “Change orientation” option on Quickbar  before creating the flow line or before clicking on the “Finish” button

Explode – Flow Line Enhancements



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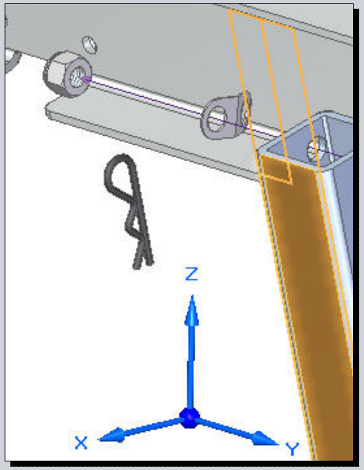
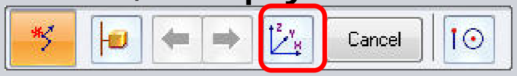
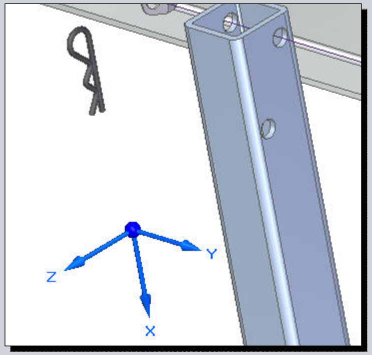


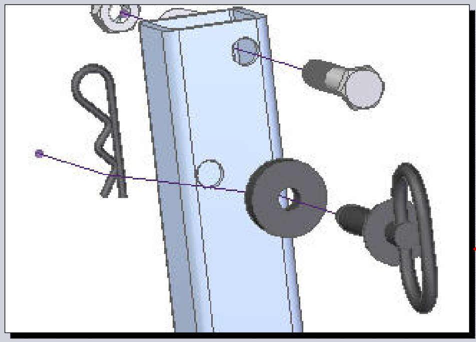
- To set the orientation, simply select the option from Quickbar 

 and when the triad appears, select a face to set the orientation you desire such as the side of the blue square tube



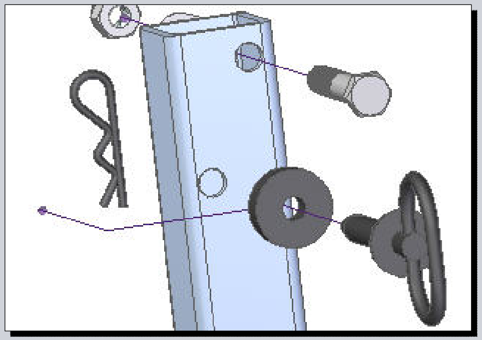
- After selecting the face, the triad re-orients itself and you are ready to create your flow line

- Now as we jog the flow line, you will notice that it now follows the correct orientation of the blue square tube

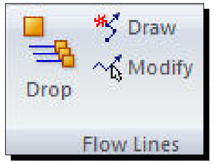


Correct →

← Incorrect



Explode – Flow Line Enhancements



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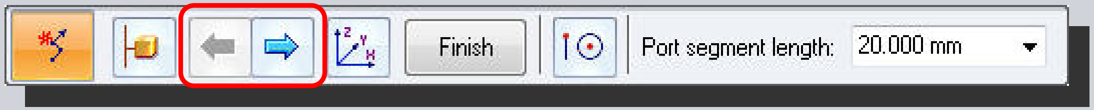
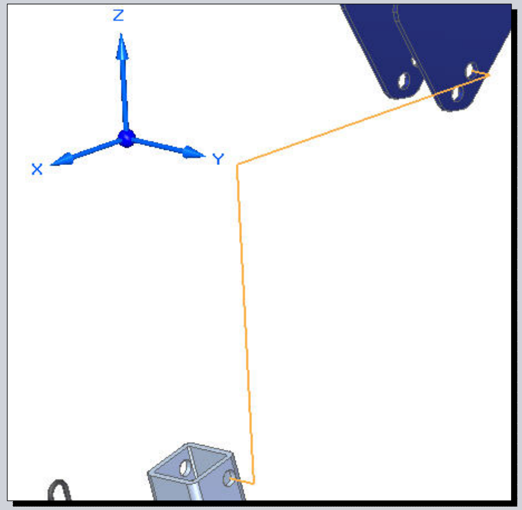
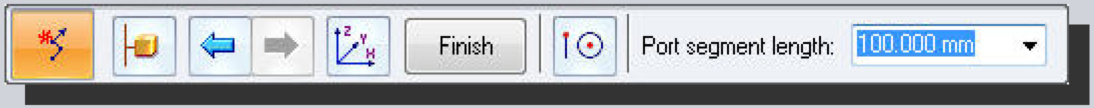
- Something added during to flow line creation is also the option to select “Previous/Next” path option that best fits your needs

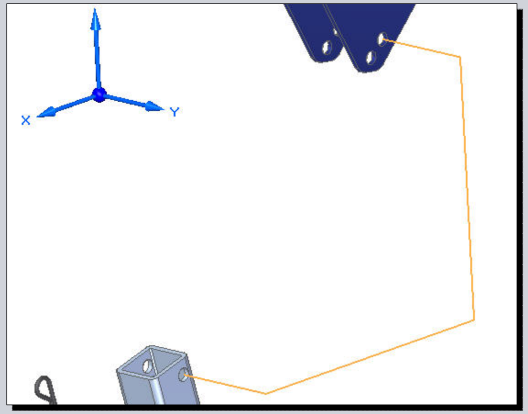




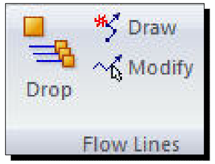
- Notice the flow line between the two parts

- By clicking on the “Next” blue arrow, it gives you a choice of other path options available for this flow line – we also changed the port length to 100 mm

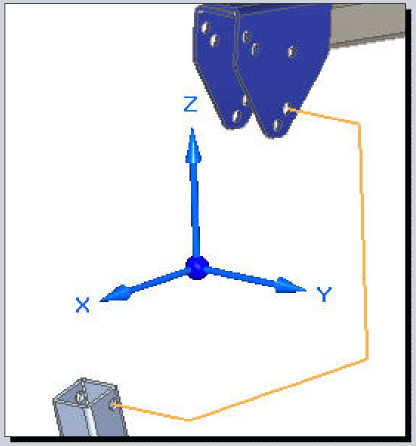


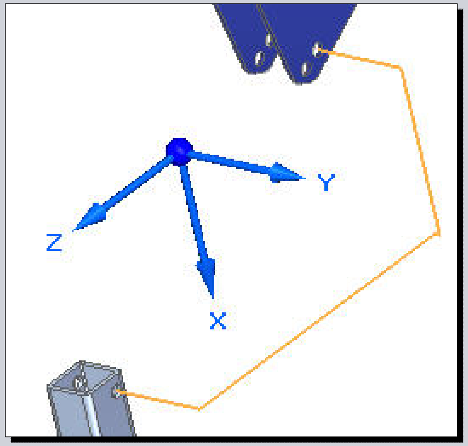


Explode – Flow Line Enhancements

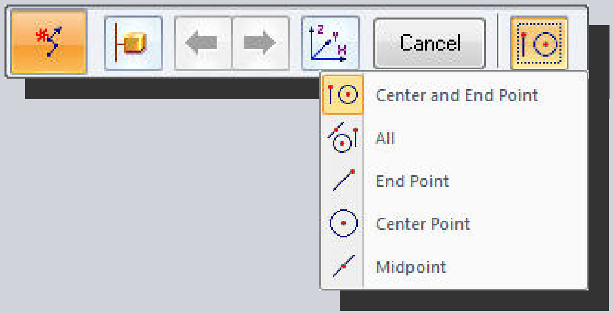


- Notice also we could change the orientation of the flow lines to follow either the dark blue part (default because it is at a normal orientation) or the light blue square tube which is at an angle

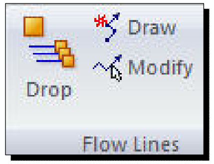




- Because flow line creation a point to point operation, Quickbar also allows the user to change the key point locate options like many of our commands in Solid Edge



Explode – Flow Line Enhancements



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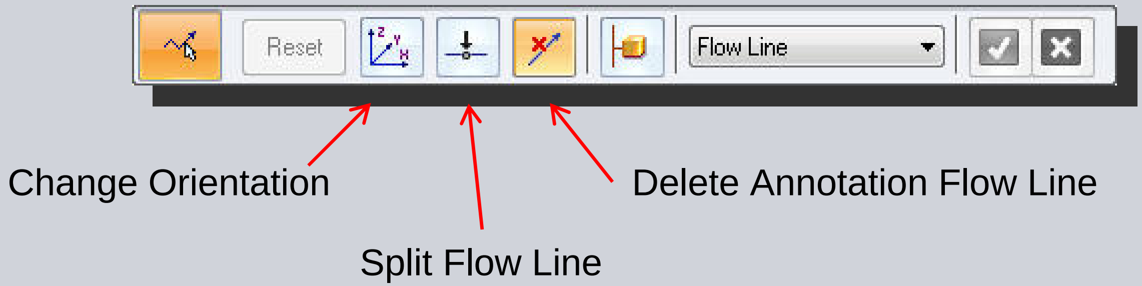
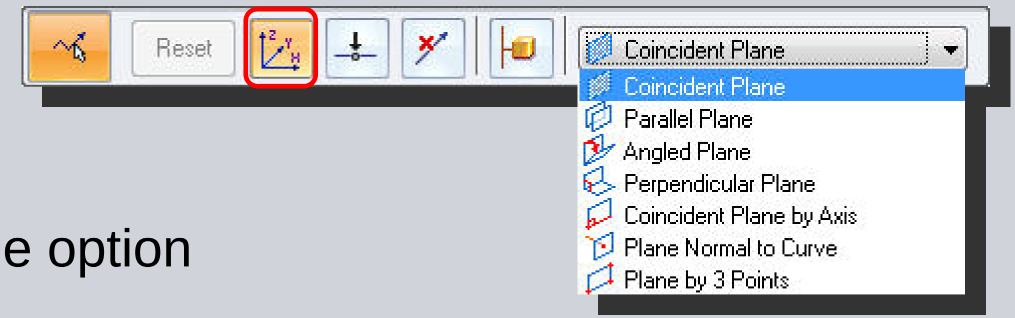
- We also have added the “Modify” Flow Line command

- Let’s take a look at how we can use this tool to create and modify flow lines to achieve the desired flow lines you are looking for

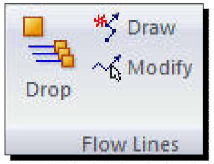


- The “Change Orientation” of a current flow line, you would first select the “Modify” command

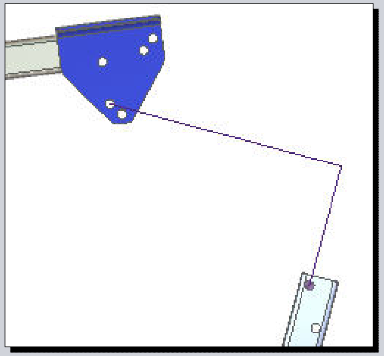


- Select desired reference plane option

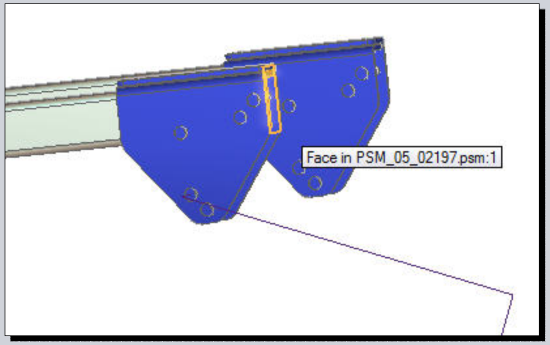
Explode – Flow Line Enhancements



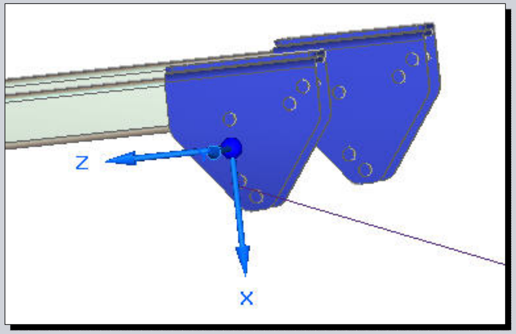
- Using the Iron Eagle example, the user may want to change the orientation of this flow line to be in line with the dark blue part instead of its current angle



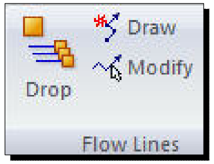
- Using the “Coincident” reference plane option, we select the front face as shown



- After selection, the Blue triad will re-orient to that selected face



Explode – Flow Line Enhancements



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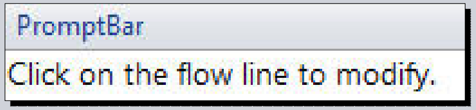
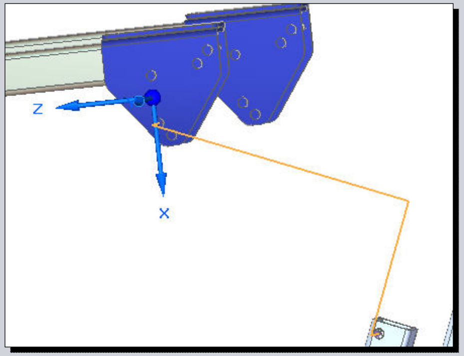
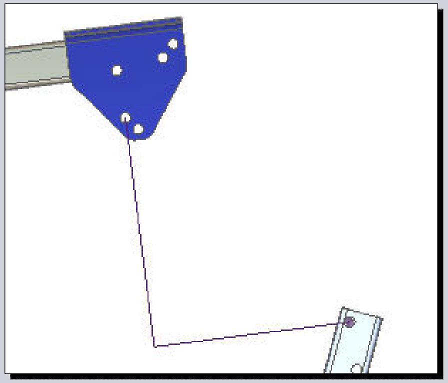
- Then the user will be prompted to select which flow line they want to re-orient



- In this example we can select the angled flow line

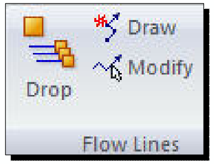




- The flow line will respond to the new orientation as shown

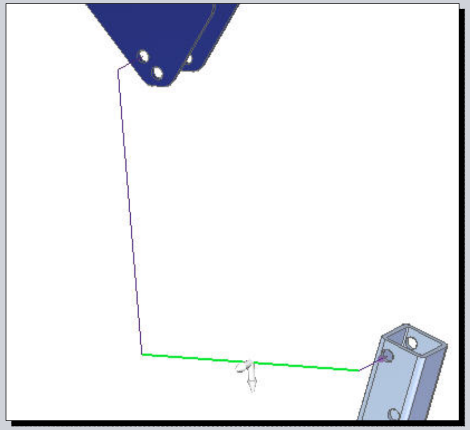
- NOTE: One thing to point out is that when an option is selected on Quickbar for the "Modify" command, it stays selected until you physically unselect it or RMB to unselect

Explode – Flow Line Enhancements



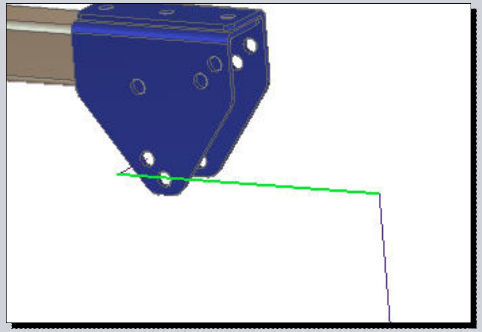
SIEMENS

- At any time if you want to adjust the flow lines, click on “Modify” and select one of the flow line segments

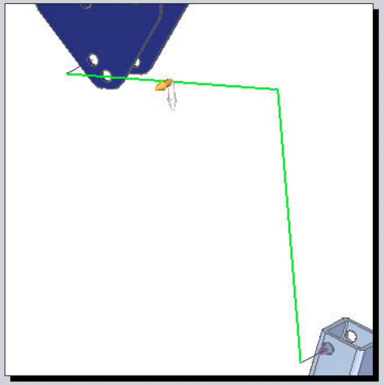


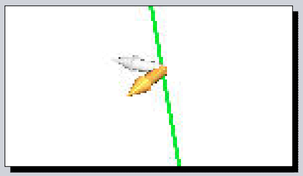
- Notice that you get arrows that can be selected to adjust the flow line position in/out or up/down

- In this case we can move the selected line upward and then using the CTRL key select both segments



- By reselecting the arrow, we can move these segments outward at the same time





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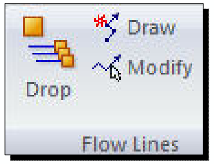
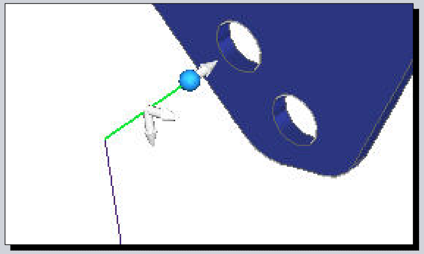
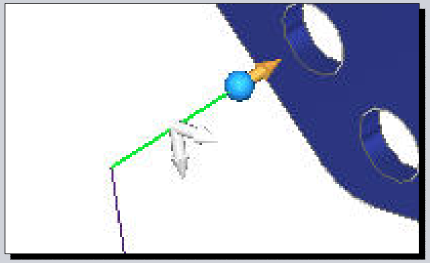
Explode – Flow Line Enhancements



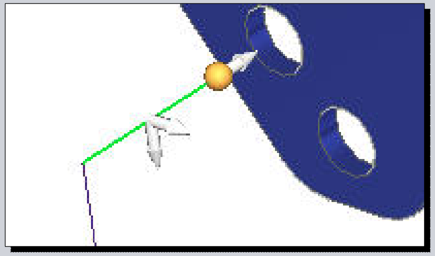
- Another cool feature while modifying the flow lines is to select an end segment which gives the user a couple of options



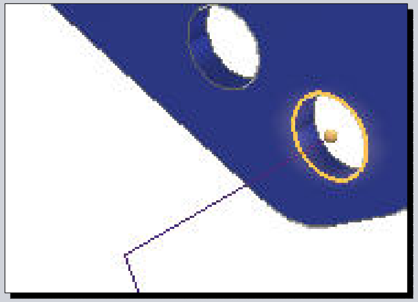
- The first is to extend or shorten the length by selecting the arrow on the end



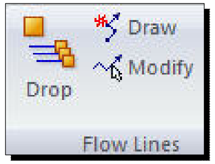
- The 2nd option is to select the blue sphere which allows the user to re locate the starting point of the flow line

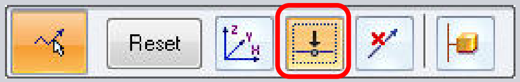


- Here we moved the flow line to the bottom hole

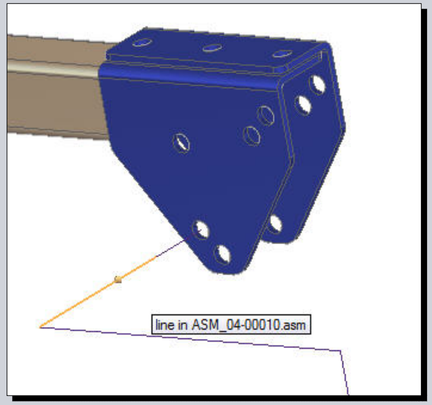


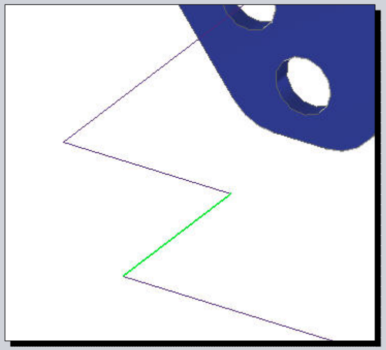
Explode – Flow Line Enhancements

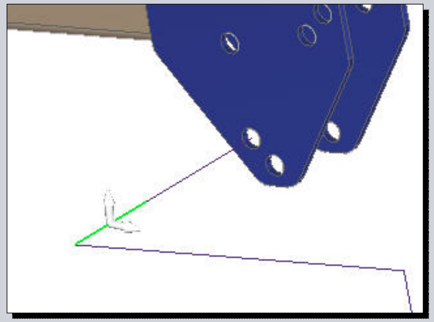


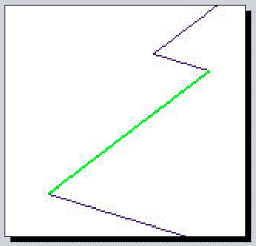


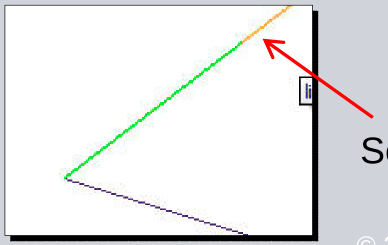
- The next “Modify” option is to “Split” a segment
- It will give the user feedback as to where the split will occur with a dot on the highlighted segment
- A simple click will cause the segment to be split
- Once the segment is split, then the user can adjust the segment to a different location
- NOTE: a segment can be adjusted to another segment as shown below











Select the segment

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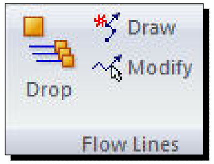
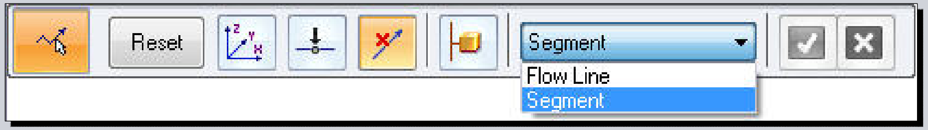
Explode – Flow Line Enhancements

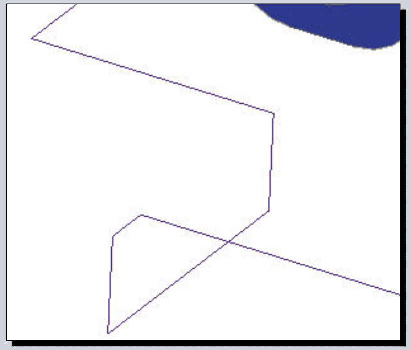


SIEMENS

- A final note on split – when creating a single straight flow line, we always split it so the user can adjust it without have to split it first

- The last option on Quickbar is the “Delete Annotation Flow Line” option

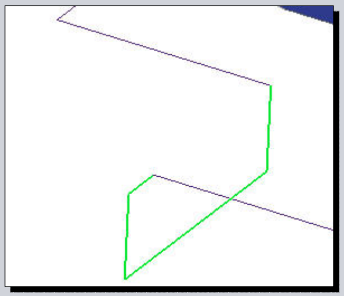


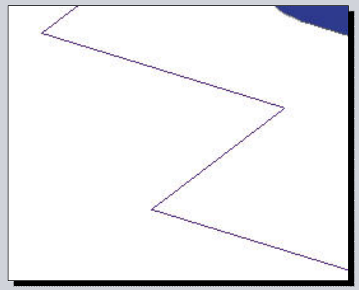


- You can delete an entire flow line or just a segment(s)

- After making a few adjustments we may need to delete a few lines as shown

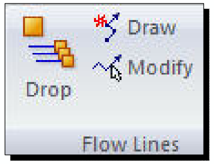
- In this case, it will create a new line connecting the segments left behind





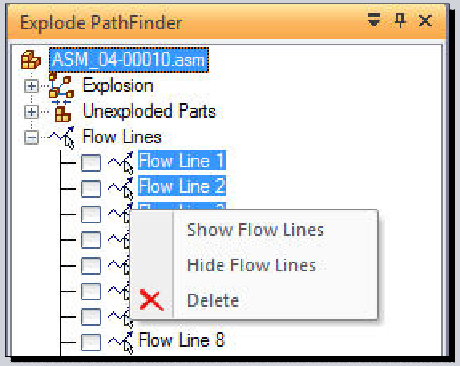
- NOTE: Cannot delete just one segment

Explode – Flow Line Enhancements



SIEMENS

- To delete all flow lines or several in the “Flow Lines” collector, you can select one, then press and hold the SHIFT key to select several in PathFinder



- There is also the option to show/hide
- The regular select tool will not locate flow lines graphically for selecting or deleting
- They will highlight graphically when the cursor passes over them in PathFinder

DEMO 6

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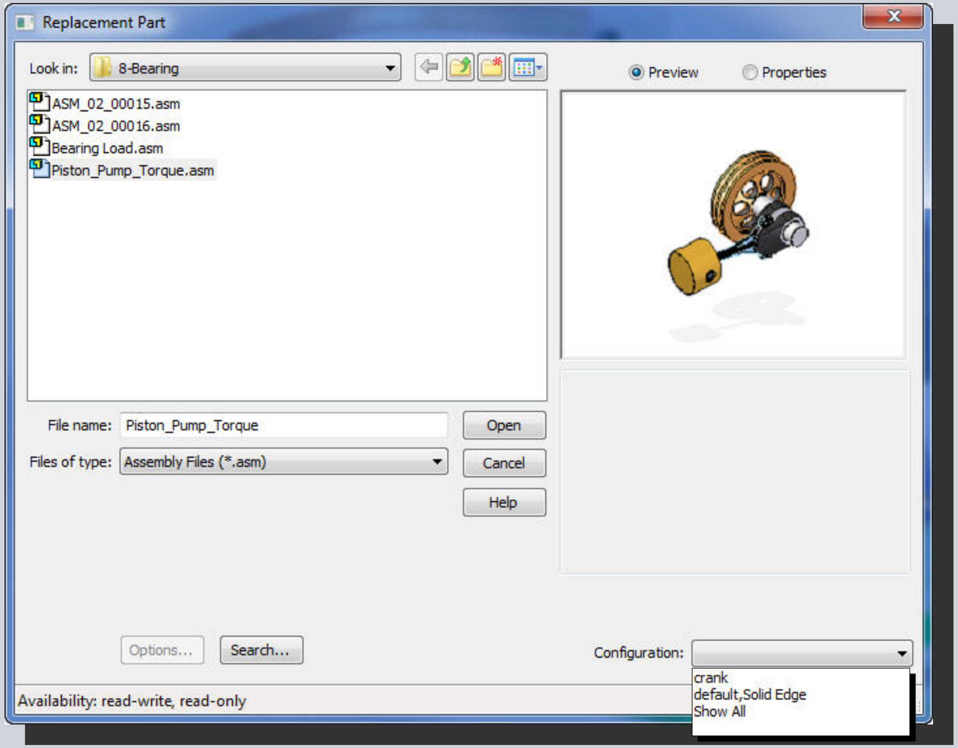
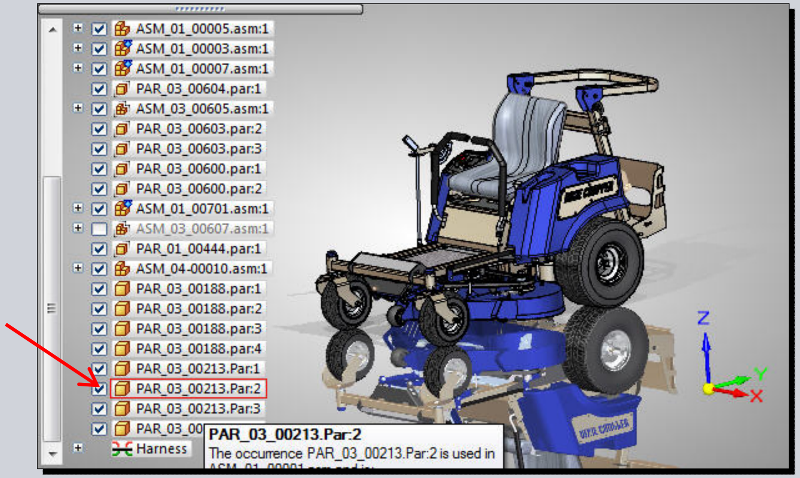
Replace Part uses "Select Configuration" option

- The "Replace Part" command now uses the file open dialog to select the target assembly and when this occurs the "Configuration" drop list will now be available

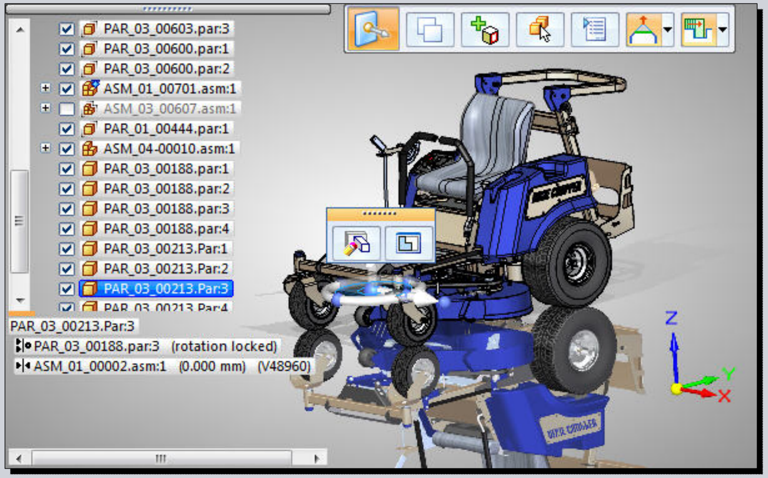


Autoscroll in PathFinder

- In a case where a user selects a component in an assembly tree (PathFinder) where the component is at the bottom of the list

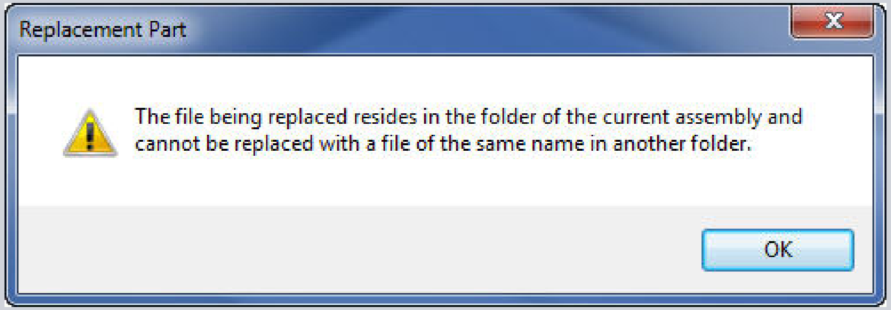


- After selecting the component, the tree will automatically scroll the part up above the relationships that appear for that part



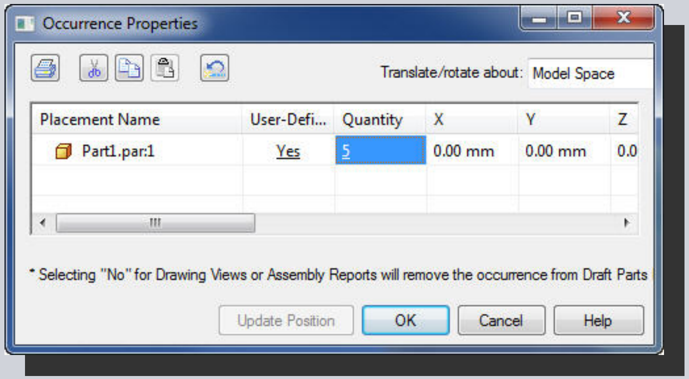
Replace part can use part with same name

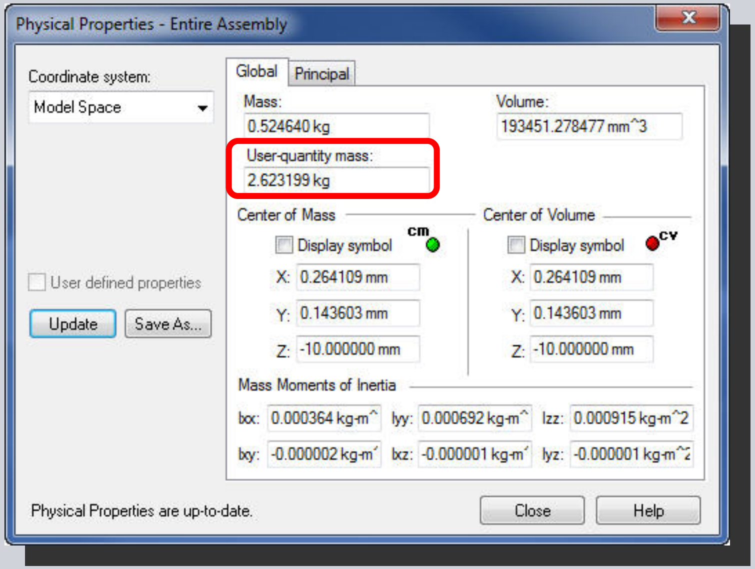
- Prior to ST4, a user could not replace a component with the same name
- In ST4, if a user opens an assembly and finds that a component is missing or has been deleted from the folder, but listed in the assembly tree, the user can now use “Replace Part” and select a part with the same name
- If a user tries to replace a part with the same name that resides in the folder of the current assembly, Solid Edge will issue the following message



Mass computation in Assembly (Qty Overrides)

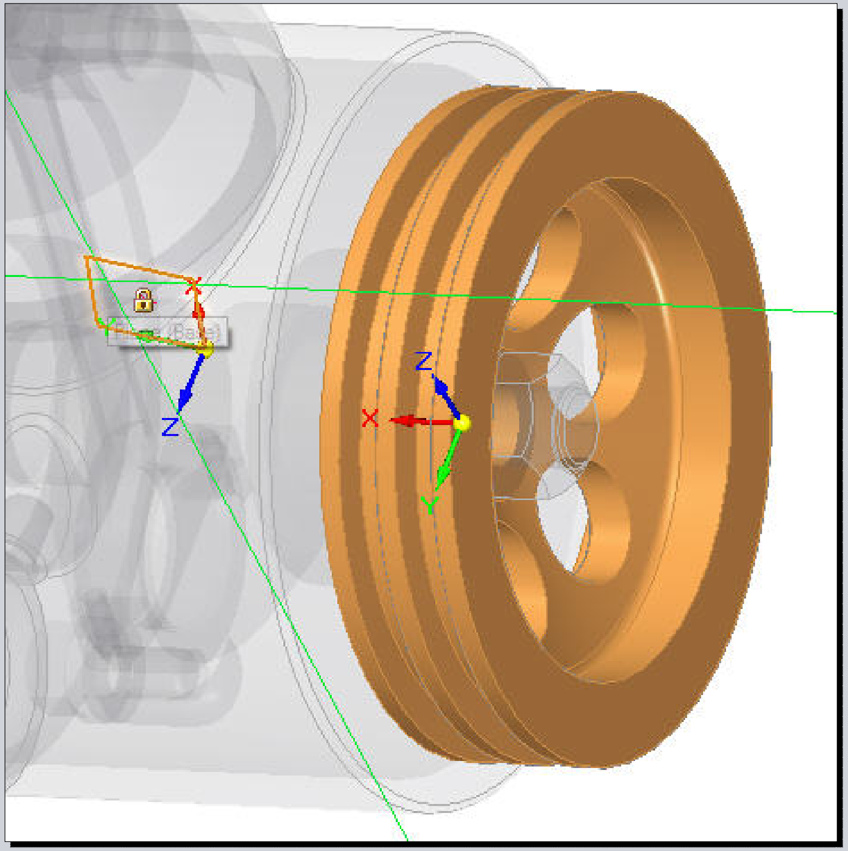
- A "user quantity mass" field has been added to the Physical Properties dialog
- When a user chooses to modify the occurrence properties of a component in an assembly and adds a specific quantity for that occurrence, the total mass is now displayed for the user defined quantity





Shift+Click to use Assembly Coordsys planes

- When IPA into an assembly component, the user can now select the “Shift” key to locate the assembly coordinate system reference planes

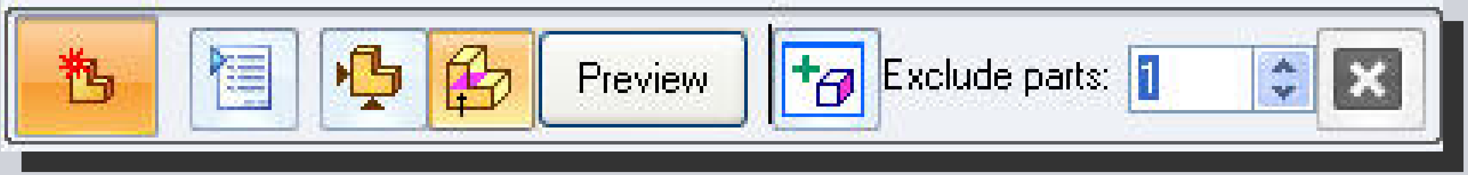


Simplified Assembly command processing

- The Simplified Assembly command bar has been updated

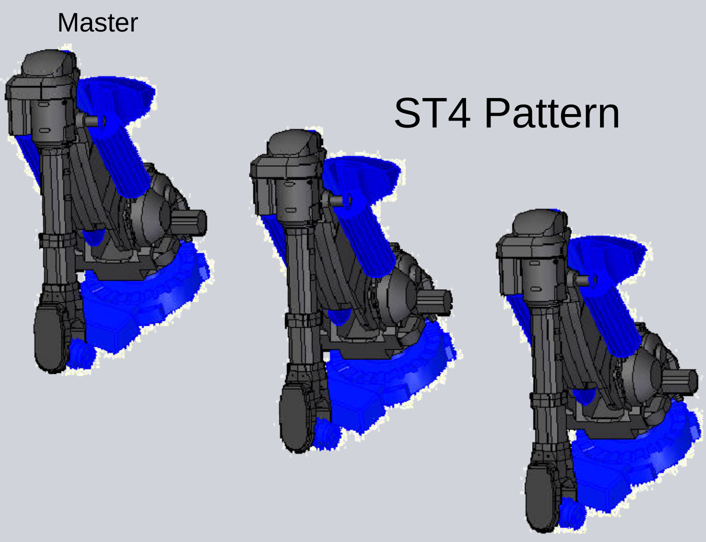
ST4

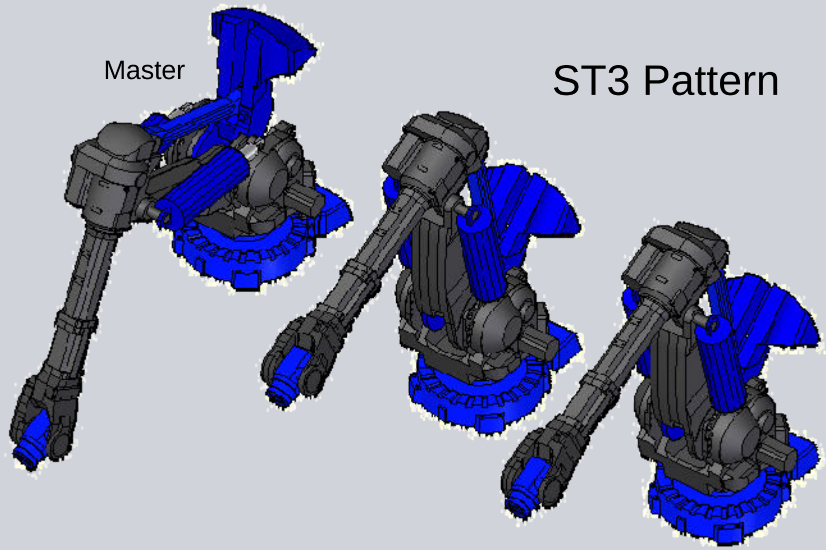
ST3



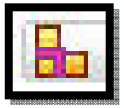
Assembly Pattern & Adjustable sub-assemblies

- In ST4 when you pattern an adjustable assembly, the pattern will take on the characteristics of the master adjustable assembly
- If the master is adjusted to a new position, the pattern is associative and will update

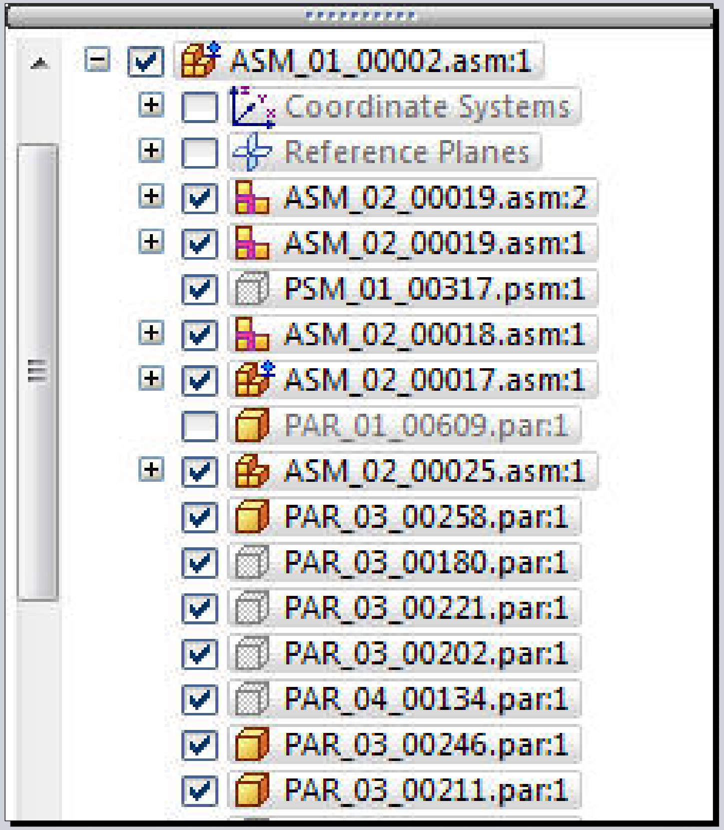




Weldment Assembly Icons

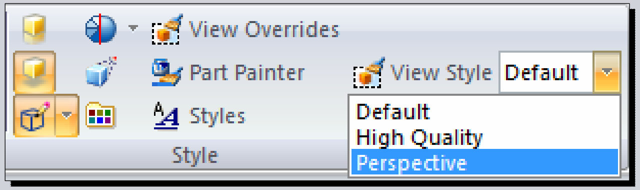


- In ST4 the weldment assembly now has its own icon
- This provides better feedback to the user regarding which assemblies are weldments

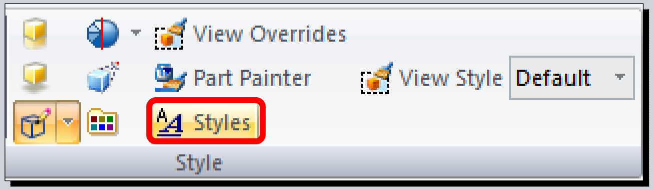


How to Add the View Styles

- In ST4 we have added new view styles, but ST3 parts and assemblies do not have these styles

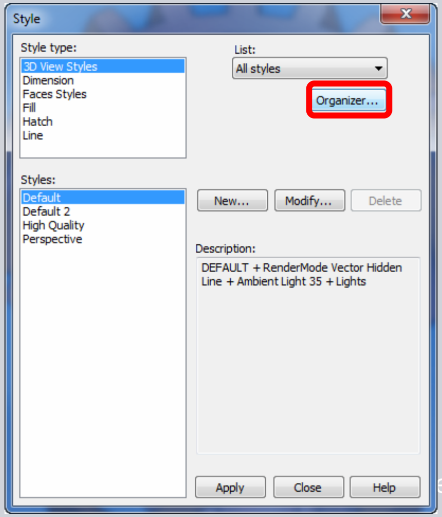


- This is just a quick overview of how to manually add them to your files



- Open your ST3 file and click on the “Styles” command from the Style group

- The Type dialog will appear



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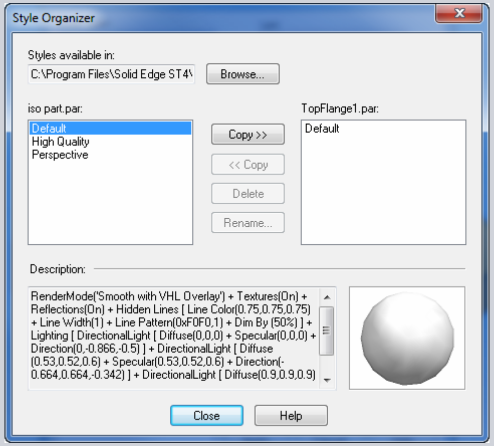
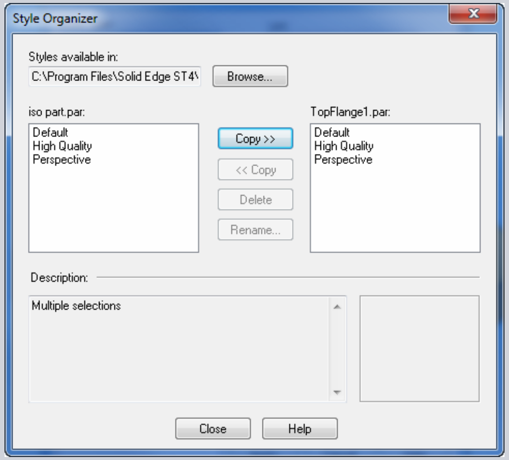
How to Add the View Styles

- Click on "Browse" and browse for a ST4 template where these are setup in the file
- When you select the template it will load the view styles into the left side of the dialog
- Highlight them and "Copy" them into your file
- Make sure you select "Apply" on the "Style" dialog and your file will now have these view styles





Thank You!