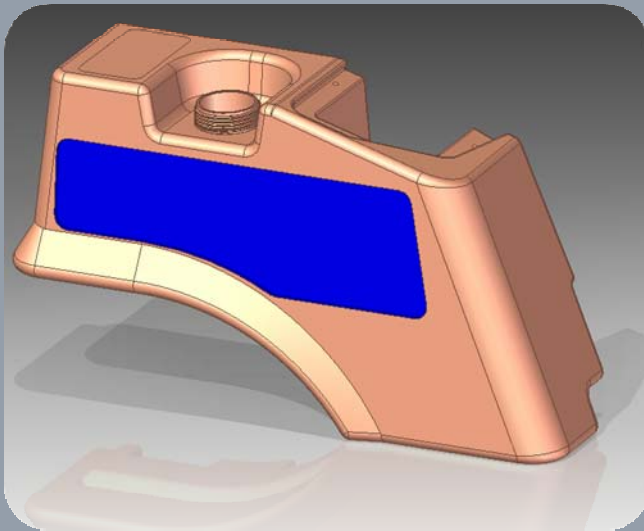


Solid Edge ST4 Update Training Part & Sheet Metal



Created By: Doug Stainbrook
Solid Edge Field Support
Application Engineer



Topic

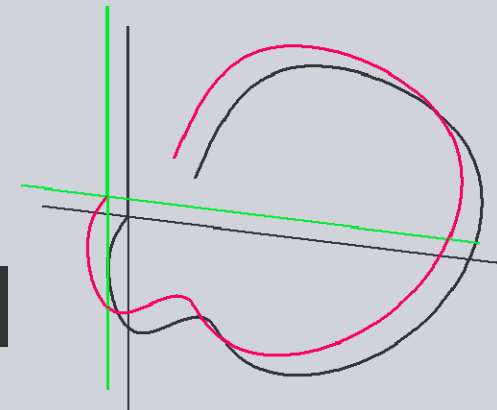
- [Bspline Curve Create Closed](#)
- [Hatch Pattern Radial Display](#)
- Variable Table
 - [Dimension Highlight With Cell Focus](#)
 - [Improved Tab Behavior](#)
- [PathFinder Honors Selection Modes](#)
- [Enable Show / Hide / Show Only in Context Menu /PathFinder](#)
- [Extrude From Assembly Faces](#)
- Relate Command
 - [UI Enhancements](#)
 - [Horizontal/Vertical Relationship](#)
 - [Offset Relationship](#)

Topic

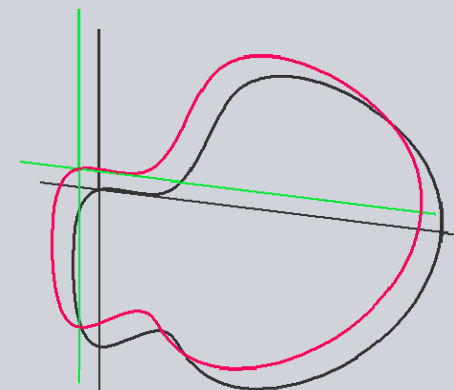
- [Coplanar Ignores Thickness Faces](#)
- [Improved Flange Mirroring](#)
- Convert to Synchronous Sheet Metal, Use Global Bend Radius
- [Hole Feature Improved Placement on Cylinder](#)
- [Selection - Feature Selection Rounds](#)
- Revolved Feature
 - [Improve Creation Workflow](#)
 - [Auto-create Live Section](#)
- [Rib/Web Network](#)
- [Material Table with Sheet Metal Gage](#)
- [Sheet Metal Move to Synchronous Improvements](#)
- [PMI Section Improvements](#)

Bspline Curve Create Closed

- An option was added to the curve Quickbar to create a closed Bspline curve
 - If the 'Close Curve' option is OFF (not pressed), the curve being created is an open curve
 - Like ST3 and before

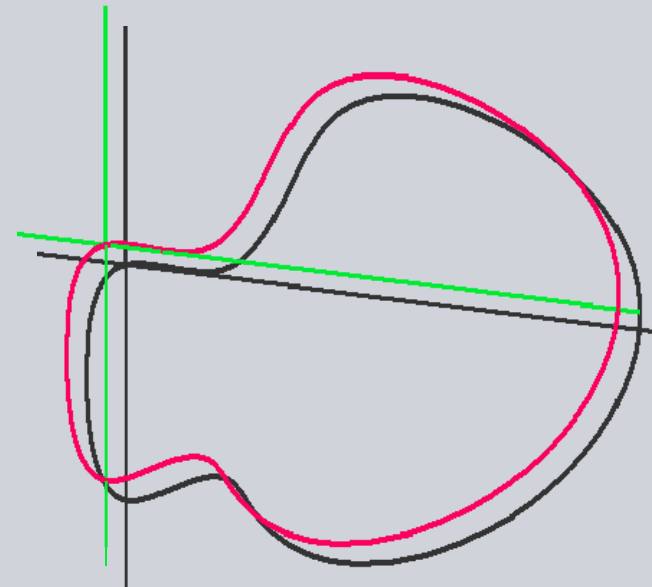


- If the 'Close Curve' option is ON (pressed), the curve being created is a closed curve



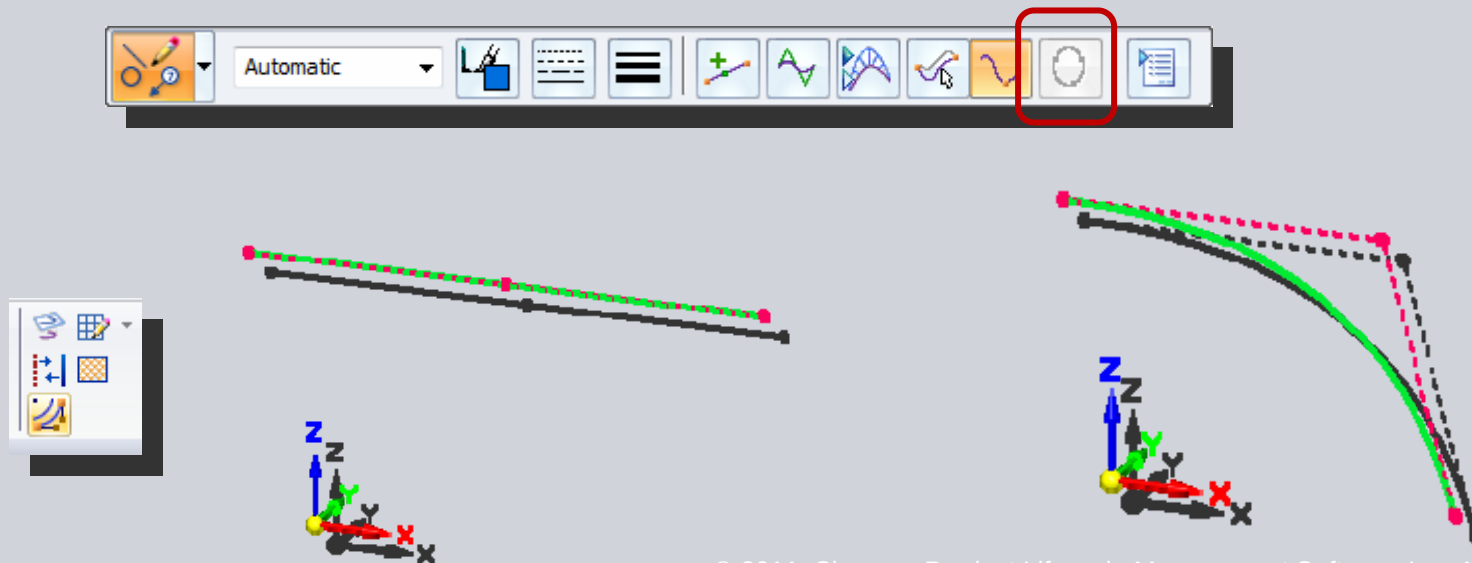
Bspline Curve Create Closed

- The new closed option is available in following environments:
 - Draft
 - Free 2D Drafting
 - Profile
 - Sync and Ordered Sketches
 - Draw in View
 - Cropping boundary profile
 - Detail profile
 - Broken out section profile
 - 2D model view
 - Feature Profile (ordered)



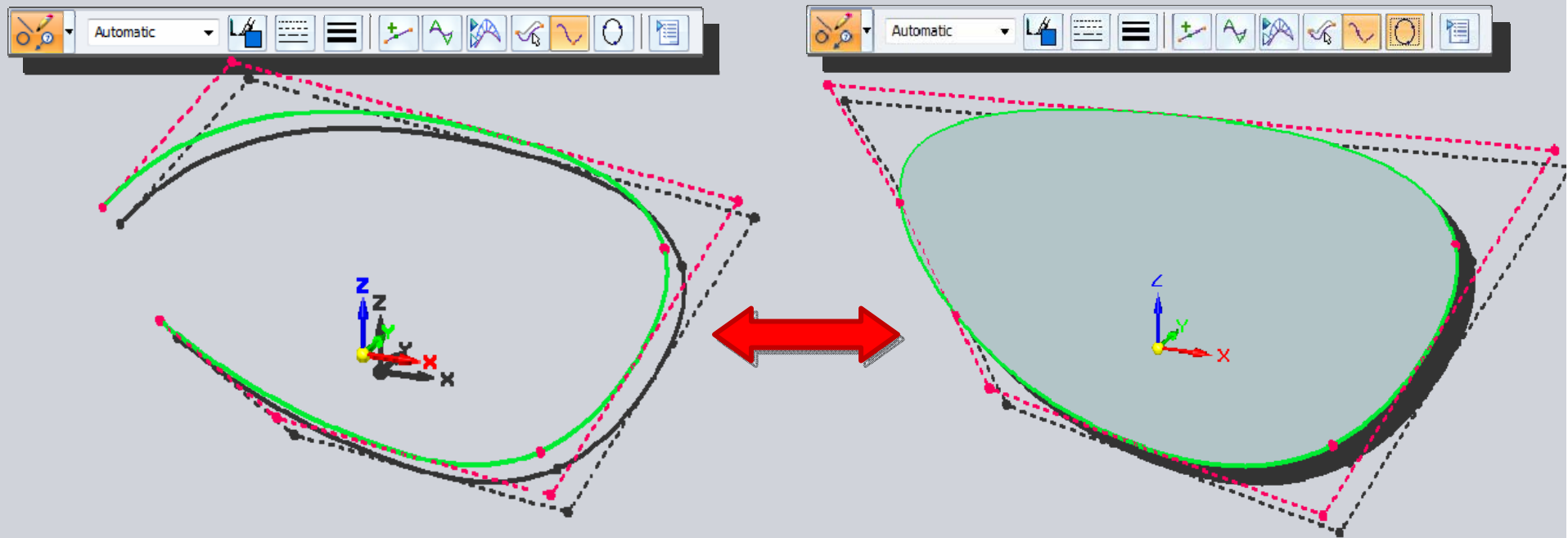
Bspline Curve Create Closed

- During edit, whenever a curve has less than 3 points, 'Close Curve' option is disabled
 - Such as the result of converting a line into a curve
 - In the edit workflow, the user can remove points along the curve and when the available points are less than three, the 'Close Curve' button is disabled



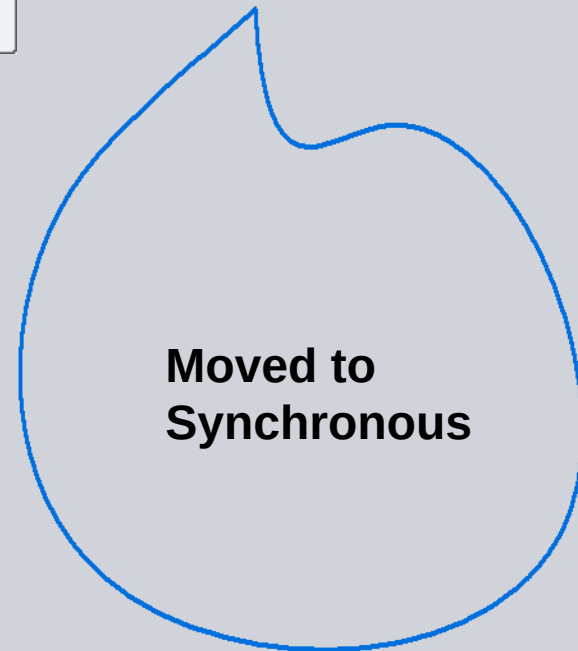
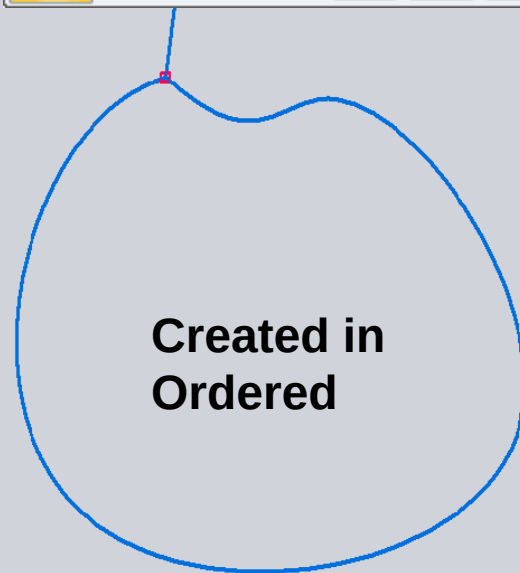
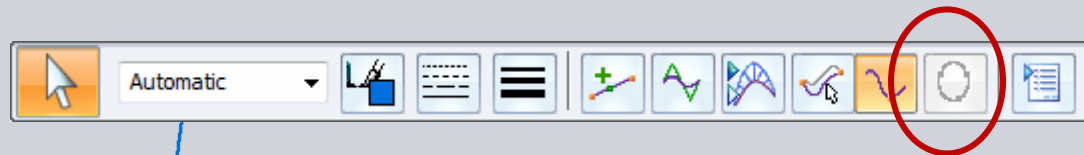
Bspline Curve Create Closed

- A curve can be changed from open to closed or vice versa by editing the curve
 - The 'Close Curve' OFF/ON state is remembered based on the state at the curve creation



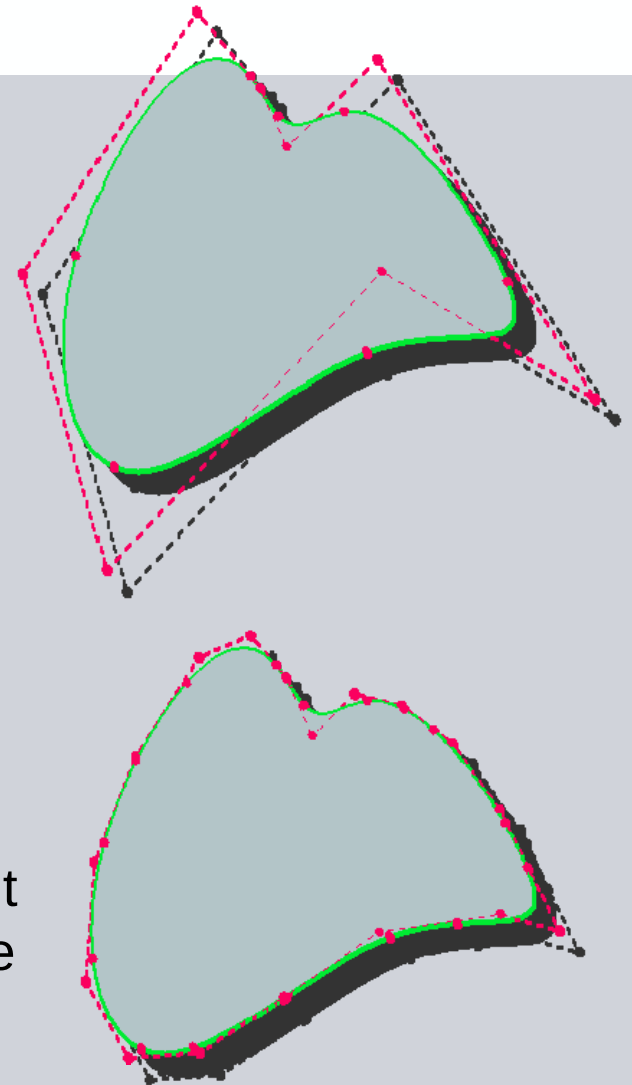
Bspline Curve Create Closed

- During creation or edit of a curve, if Solid Edge encounters any non-tangential closed curve conditions, the 'Close Curve' button is DISABLED
 - A closed curve must be a continuous, tangential curve



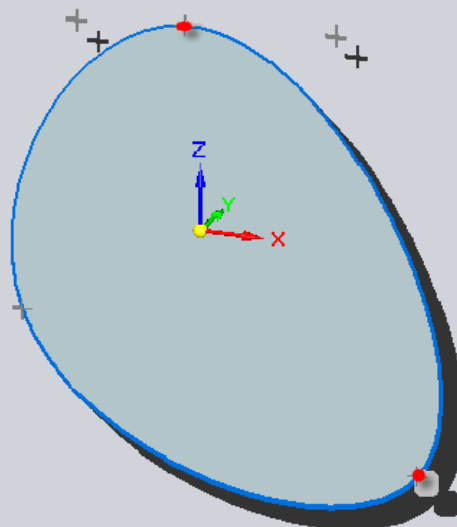
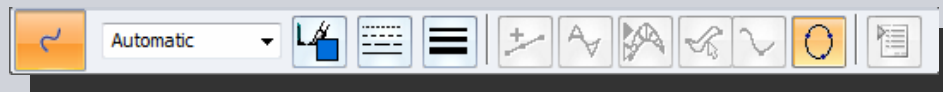
Bspline Curve Create Closed

- When a closed curve is created using the 'Closed curve' option, the curve shape is the same as that of a 3D closed periodic keypoint curve, however this curve is not be a true periodic curve
 - A true periodic curve is a closed curve with no beginning point and no end point
 - The shape of the closed curve matches a periodic curve, but in reality, is a tangentially closed curve
- When the user edits a curve (i.e. some edit point position or control point position is modified), the curve remains a tangentially closed curve



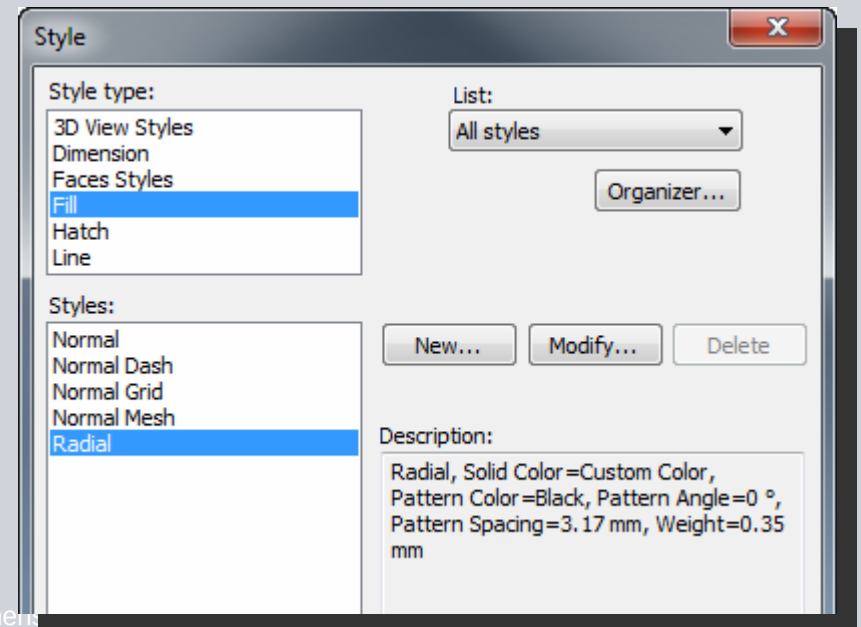
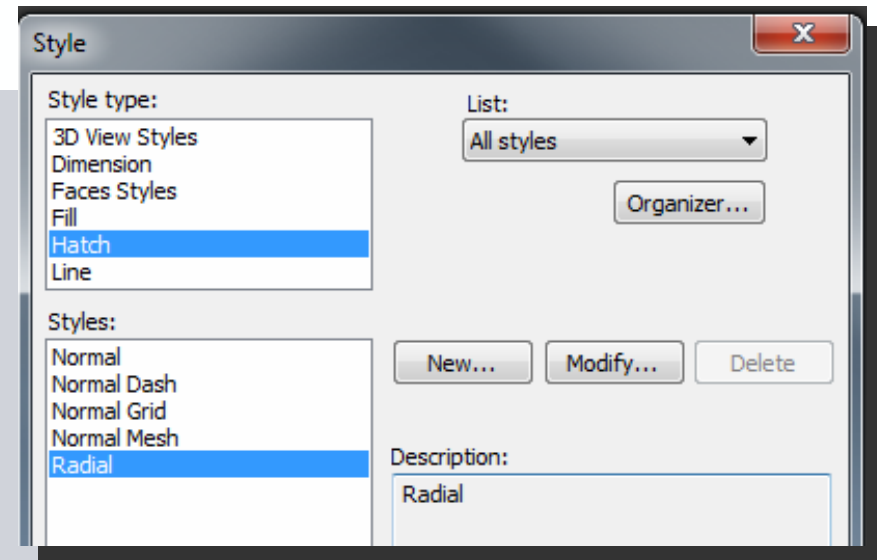
Bspline Curve Create Closed

- Note that creating a closed curve and then opening can result in a drastic change in the shape of the curve
 - An open curve is not equal to the closed curve minus the section between the first and last point



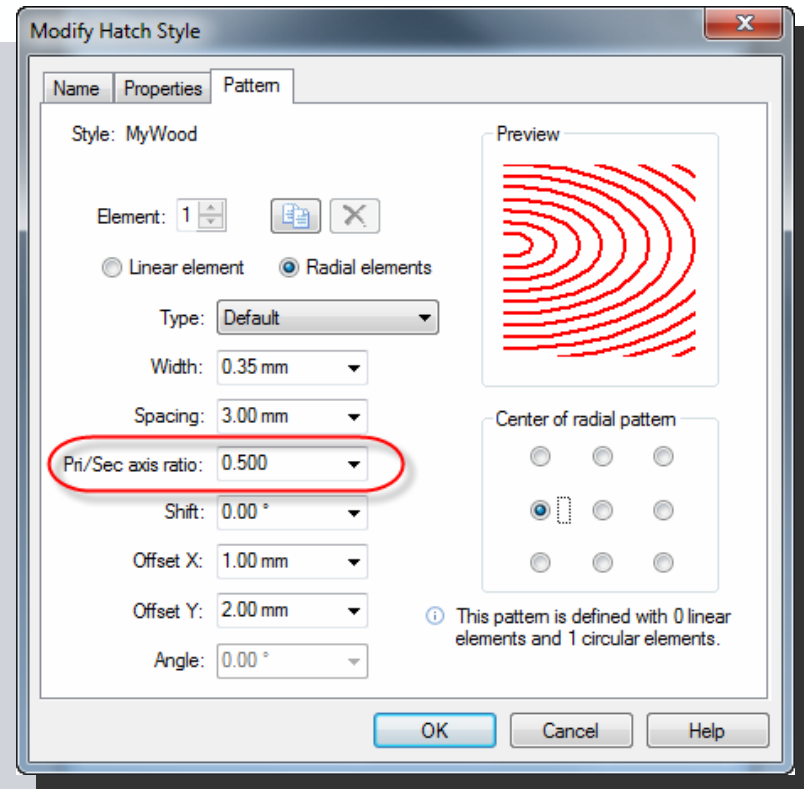
Hatch Pattern Radial Display

- A new **Hatch Style** has been added called Radial
 - This hatch pattern contains concentric circles
- A new **Fill Style** has been added called 'Radial'
 - This fill style has the new 'Radial' hatch style as the hatch pattern
- These new hatch and fill styles have been added in Part, Sheet Metal, Assembly and Draft environments



Hatch Pattern Radial Display

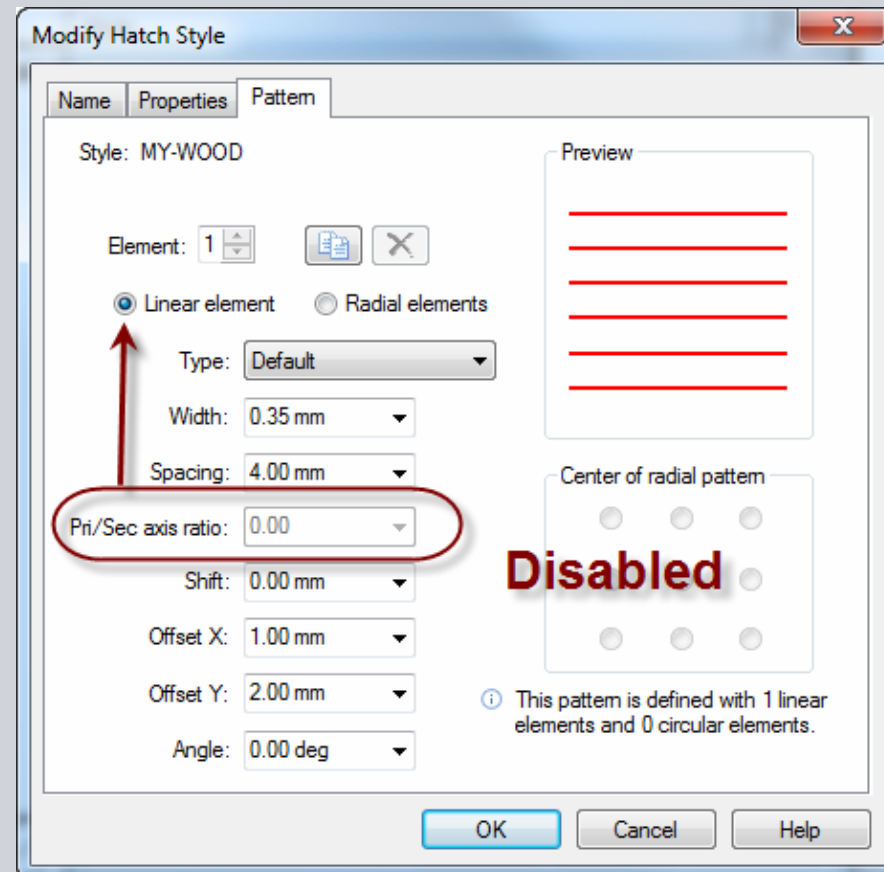
- A new parameter has been added to the Hatch Style dialog to change the 'primary axis / secondary axis' ratio of radial elements
 - User can display ellipses in the radial hatch patterns that represent wood
 - Commitment to Russian and Chinese standards



- Spacing = primary axis of first element in the radial pattern
 - Example: If Spacing = 3 and Ratio = 2
 - The primary axis of first ellipse = 3, secondary axis = 1.5
 - The remaining ellipses in the pattern are calculated as per spacing and ratio

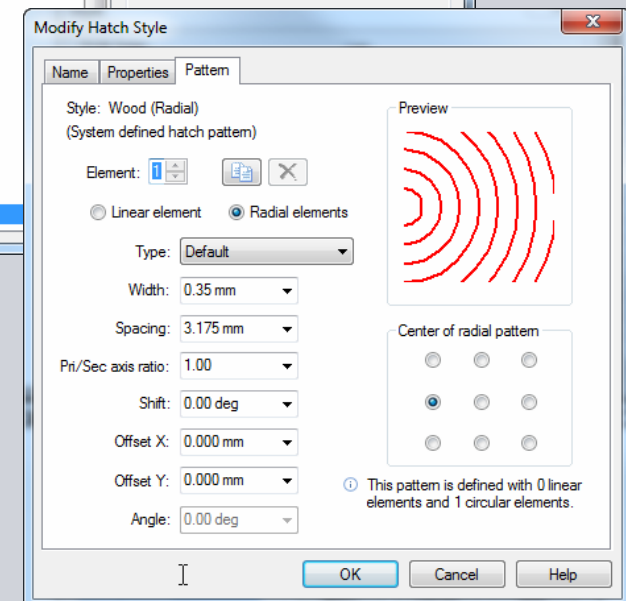
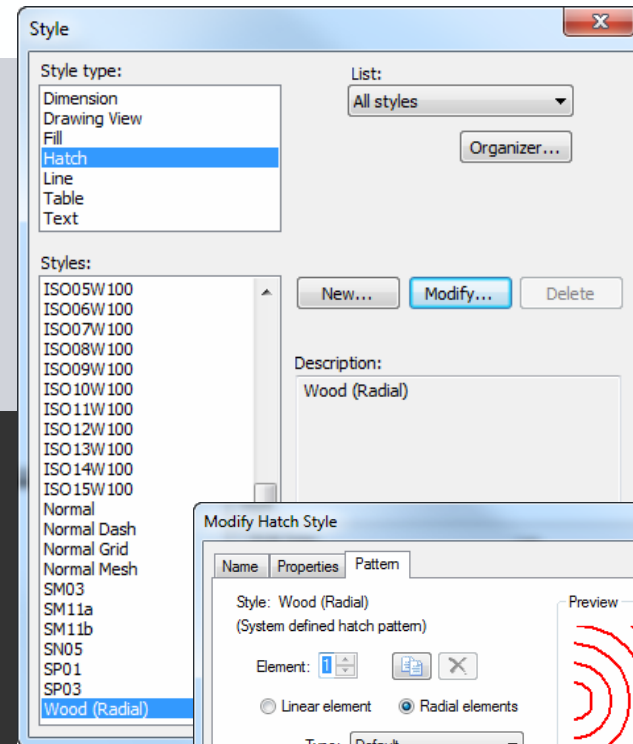
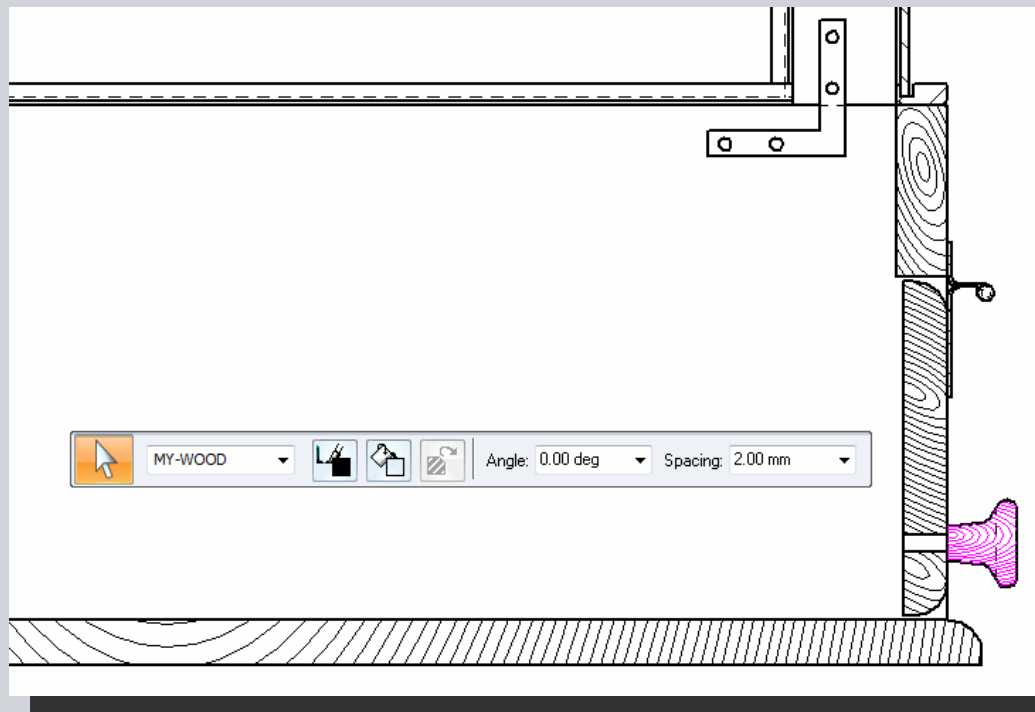
Hatch Pattern Radial Display

- This axis ratio option is only applicable to radial elements and is disabled if the 'Linear element' radio button is selected



Hatch Pattern Radial Display

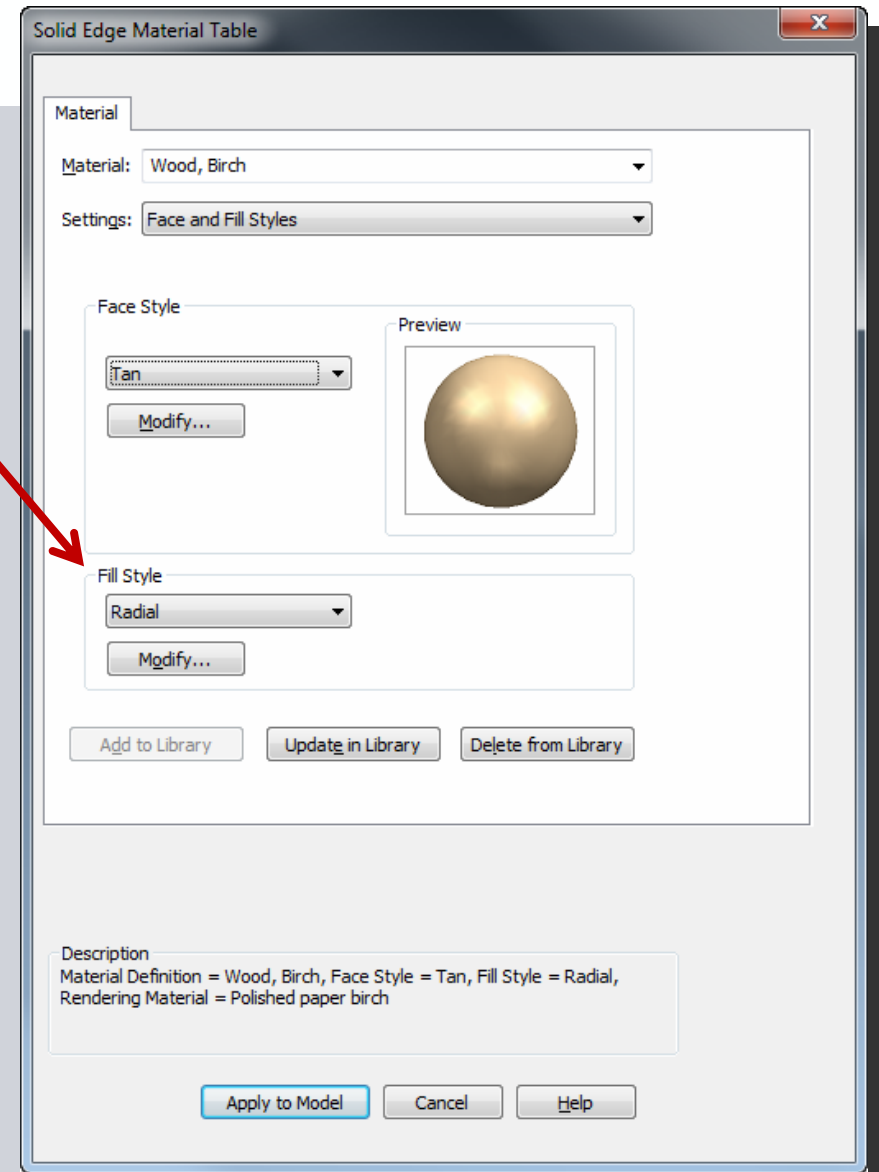
- Example: A new hatch style called "Wood (Radial)" is applied in a drawing view at different angles and spacing



Hatch Pattern Radial Display

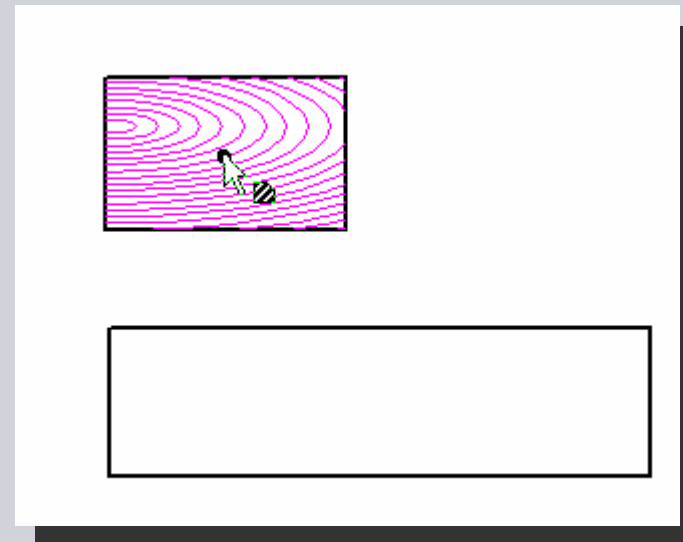
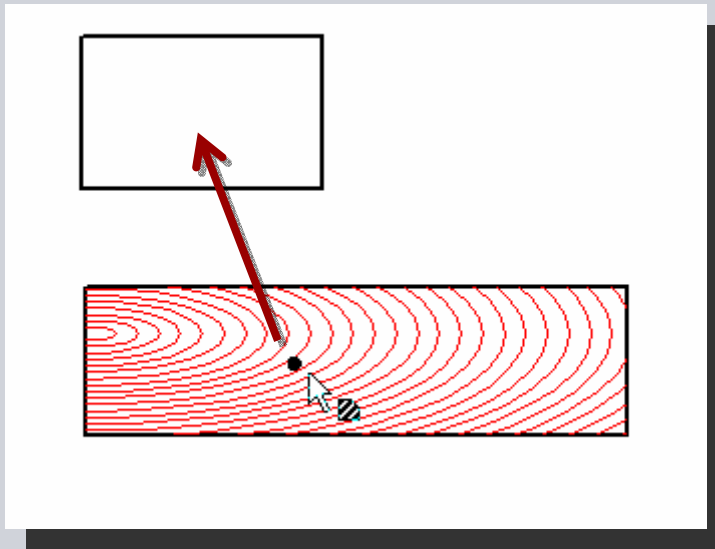
- In ST4 all wood materials have a fill style of Radial

Wood, Ash
Wood, balsa
Wood, Birch
Wood, Cherry
Wood, Mahogany
Wood, Oak
Wood, Pine
Wood, Walnut



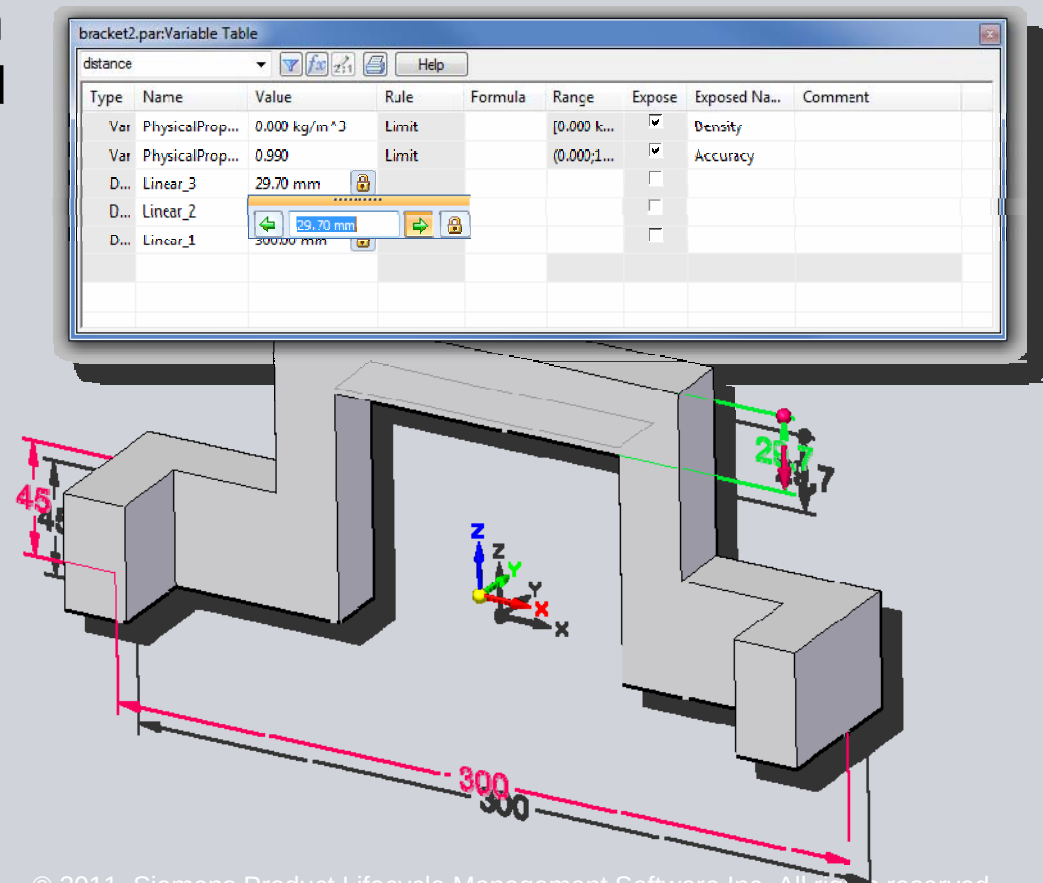
Hatch Pattern Radial Display

- Radial Pattern contains a handle that can be used to drag the fill style to a different area



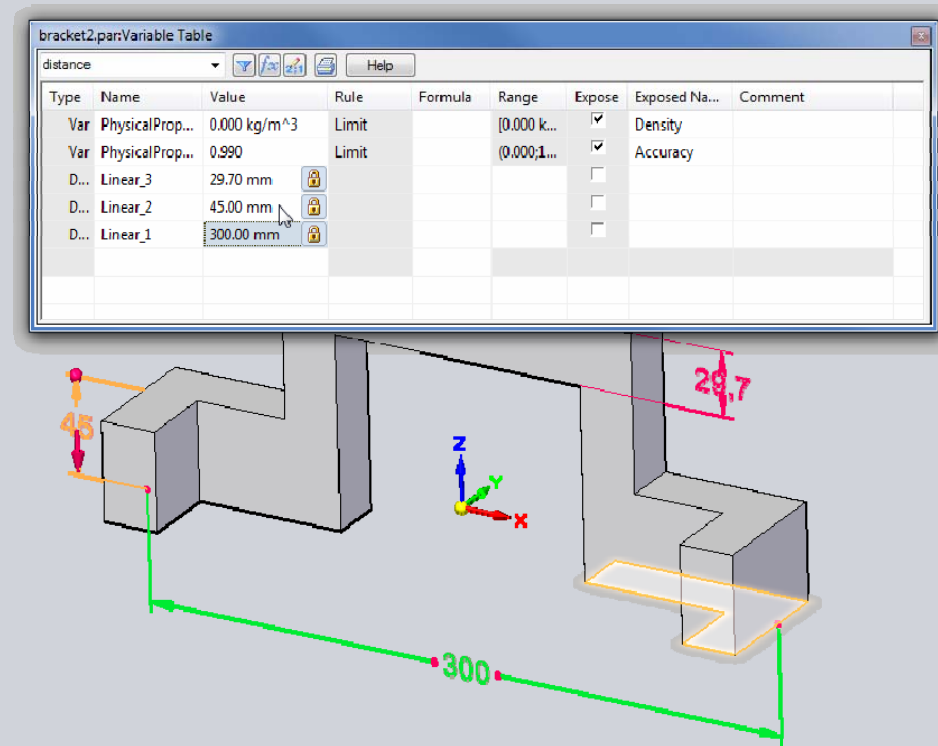
Dimension Highlight With Cell Focus

- Solid Edge now maintains highlight of a dimension in view when it is being edited in the variable table
 - Previously hovering over an entry in variable table would highlight the dimension in view, but clicking to edit the dimension would drop the highlight during edit
- In ST4 the users see the highlight even during edit



Dimension Highlight With Cell Focus

- This maintains consistency with behavior seen if the dimension is edited in the graphics window
- Applies to PMI and profile dimensions edited through the variable table or through peer variable tables



Improved Tab Behavior

- Unique behavior for TAB and ENTER within the same cell of the variable table
 - Made to work similarly to Excel
 - TAB applies the change and shifts focus one cell to the right
 - ENTER applies the change and shifts the focus one cell down
 - SHIFT+ENTER moves up one cell

ordered-vertexbind.psm:Variable Table

distance

T..	Name	Value	Range
Var	Flange_1_Wi...	1.00 mm	(0.00 mm;)
Var	MaterialThic...	3.00 mm	(0.00 mm;)
Var	BendRadius	1.00 mm	[0.00 mm;)
Var	ReliefWidth	1.00 mm	(0.00 mm;)
Var	ReliefLength	0.00 mm	(0.00 mm;)

ordered-vertexbind.psm:Variable Table

distance

T..	Name	Value	Range
Var	Flange_1_Wi...	1.00 mm	(0.00 mm;)
Var	MaterialThic...	3.00 mm	(0.00 mm;)
Var	BendRadius	1.00 mm	[0.00 mm;)
Var	ReliefWidth	1.00 mm	(0.00 mm;)
Var	ReliefLength	0.00 mm	(0.00 mm;)

TAB

ordered-vertexbind.psm:Variable Table

distance

T..	Name	Value	Range
Var	Flange_1_Wi...	1.00 mm	(0.00 mm;)
Var	MaterialThic...	3.00 mm	(0.00 mm;)
Var	BendRadius	1.00 mm	[0.00 mm;)
Var	ReliefWidth	1.00 mm	(0.00 mm;)
Var	ReliefLength	0.00 mm	(0.00 mm;)

ENTER

ordered-vertexbind.psm:Variable Table

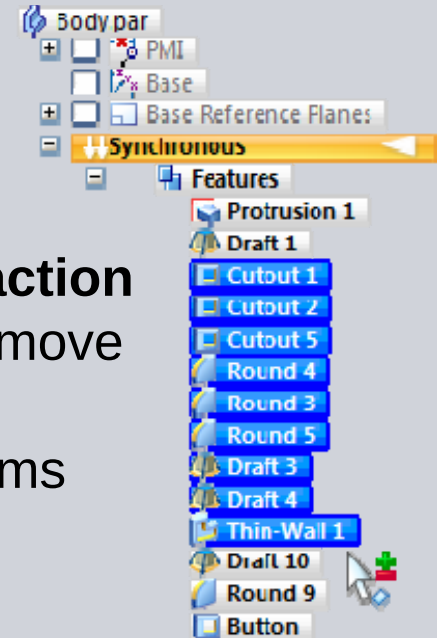
distance

T..	Name	Value	Range
Var	Flange_1_Wi...	1.00 mm	(0.00 mm;)
Var	MaterialThic...	3.00 mm	(0.00 mm;)
Var	BendRadius	1.00 mm	[0.00 mm;)
Var	ReliefWidth	1.00 mm	(0.00 mm;)
Var	ReliefLength	0.00 mm	(0.00 mm;)

SHIFT+ENTER

PathFinder Honors Selection Modes

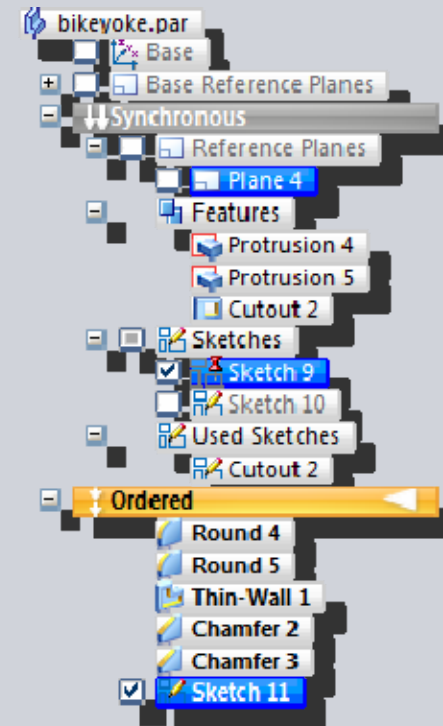
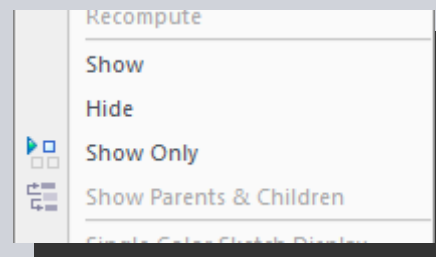
- PathFinder selection honors the +/- selection modes
 - **Shift** select **always adds** to the select set in + and +/- selection modes
 - Shift select in minus (-) selection mode performs **no action**
 - Shift select does not invert the selection mode or remove items from the set
 - **Ctrl** select in minus (-) selection mode will not add items to the select set
 - It will remove items if they are already selected
- Clicking the white space in PathFinder (when not in the document view) with no shift or ctrl key down clears the select set
- Applies to Assembly, Weldment, Part and Sheet Metal PathFinders
 - Also applies while in-place-activated



Enable Show / Hide / Show Only in Context Menu of PathFinder

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- Show Only has been added to the shortcut menu in synchronous mode
- When eligible elements from both environments are selected, the commands for show / hide / show only act on all eligible elements, regardless of native environments type
 - Ineligible items in the select set are ignored
 - The elements that are available for show / hide / show only controls are:
 - Reference planes
 - Coordinate Systems
 - Surfaces
 - Sketches
 - Curves
 - Sketches
 - Design body



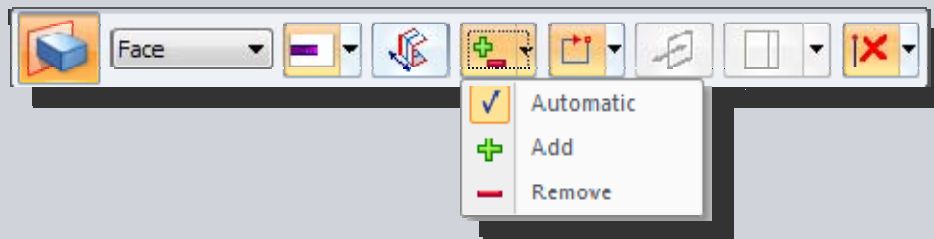
Enable Show / Hide / Show Only in Context Menu of PathFinder



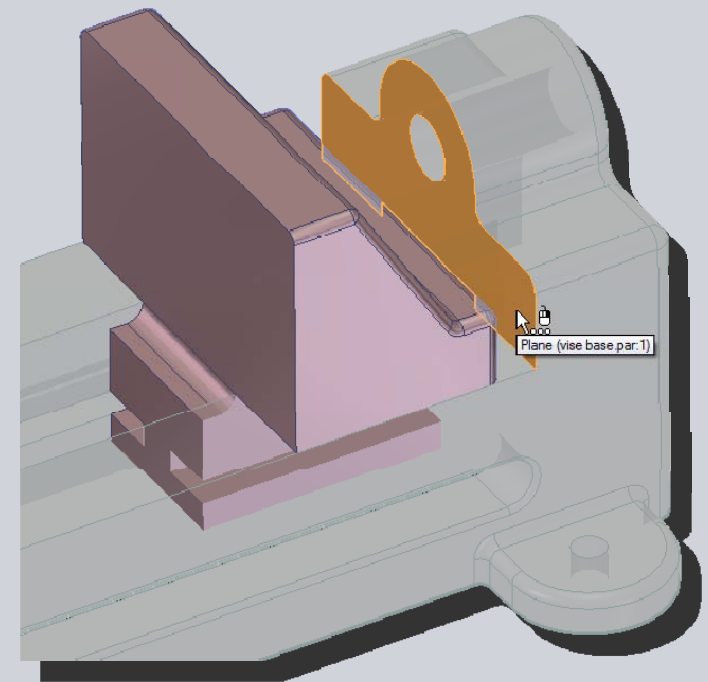
- The display state behavior is applied cumulatively
 - If the user is in the sync mode with an integrated design environment model, show / hide/ show only commands:
 - Are not available on the ordered mode elements
 - Display state changes apply to synchronous elements in the model
 - If the user is in the ordered mode with an integrated design environment model, show / hide/ show only commands:
 - Are available on eligible elements (sync and ordered)
 - Display state changes affect the entire model

Extrude From Assembly Faces

- In ST4, the synchronous part modeling Extrude and Revolve commands allow the selection of assembly faces while in-place-activated
 - To add or remove material

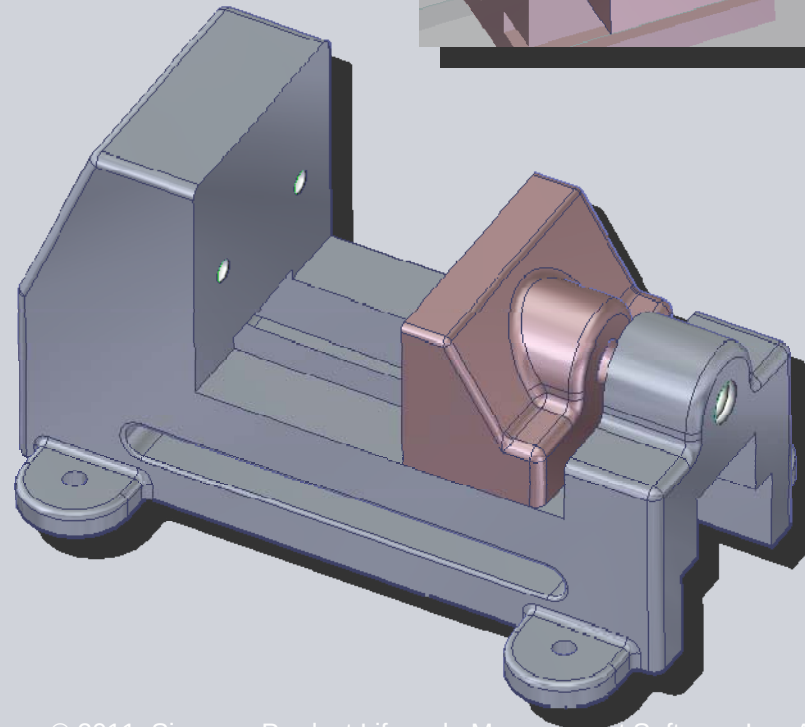
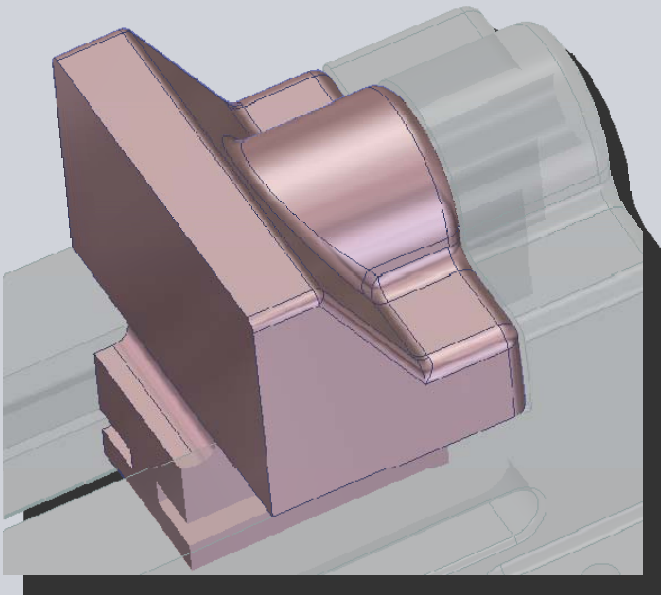
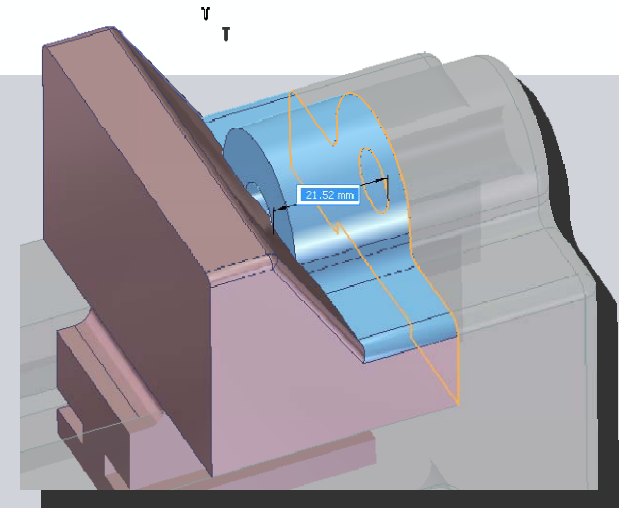


- Assembly faces are available for selection during the sync action/object workflow only
 - Select the Extrude or Revolve command, then select the face or faces
 - In order for faces to be selectable, the peer part must be displayed and active



Extrude From Assembly Faces

- Faces from multiple peer parts may be selected as input
- There is NO associativity back to the input face

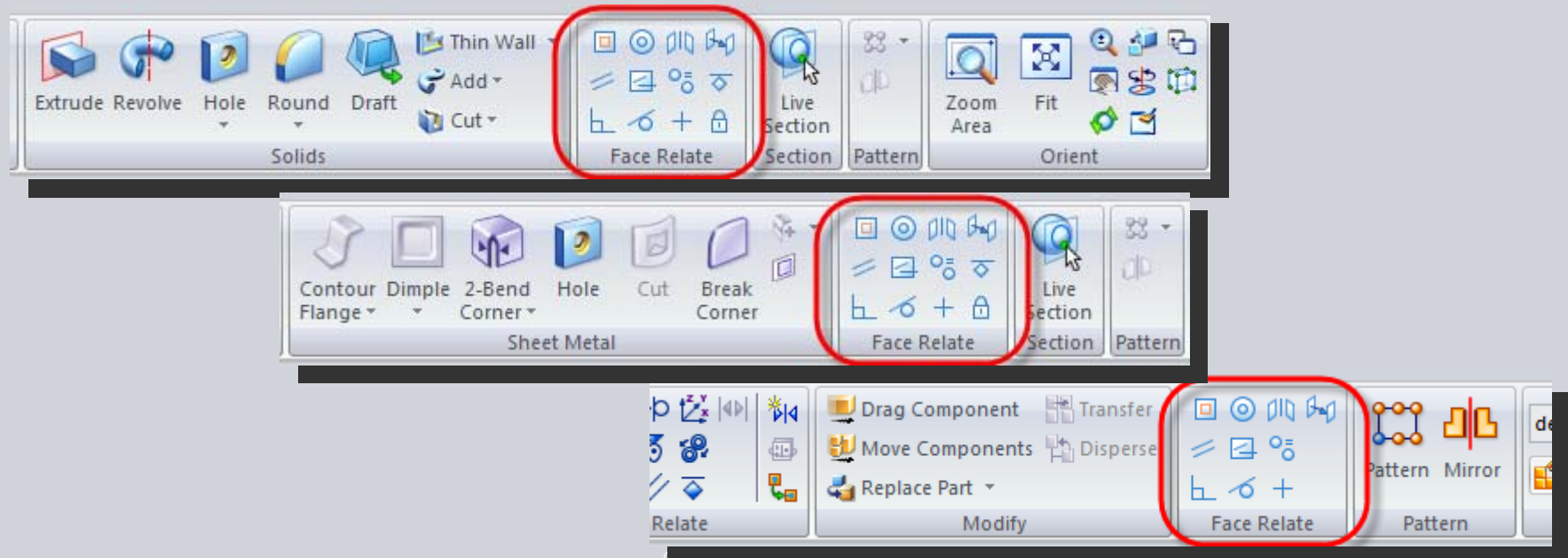


DEMO 3

Page 24

Relate UI Enhancements

- The Relate commands are now exposed on the Command Ribbon for easier access
 - In synchronous mode for Part and Sheet Metal
 - In Assembly
- Coincident is now Coplanar for consistency



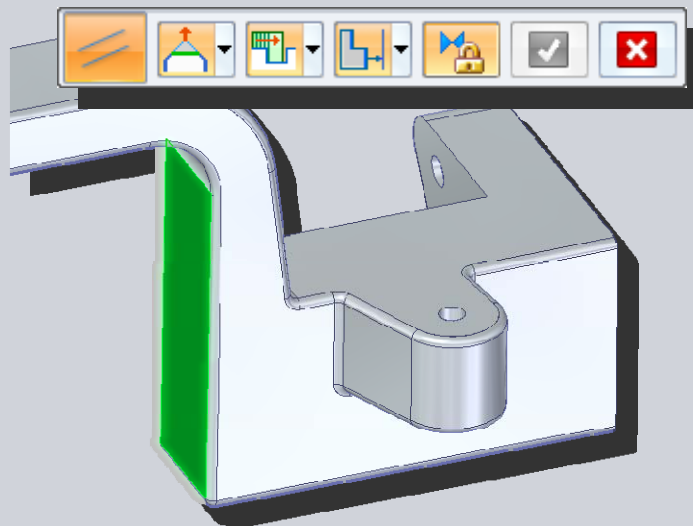
Relate UI Enhancements

- Relate commands are now action/object or object/action
 - Previously, Relate commands were always object/action

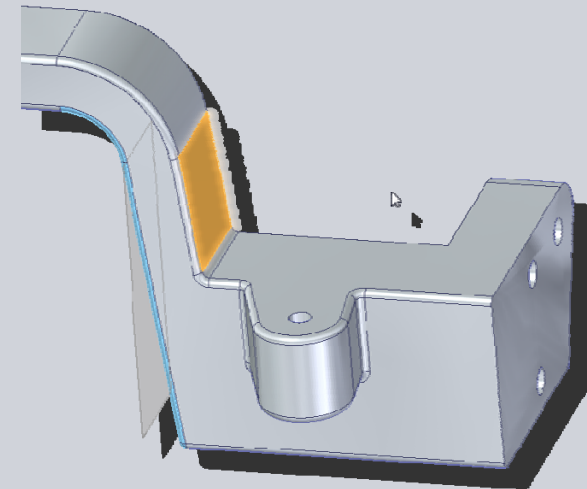
Action/Object workflow example:



Select a Relate command:
"Parallel"



Select Seed element(s)
Seed face(s) will move

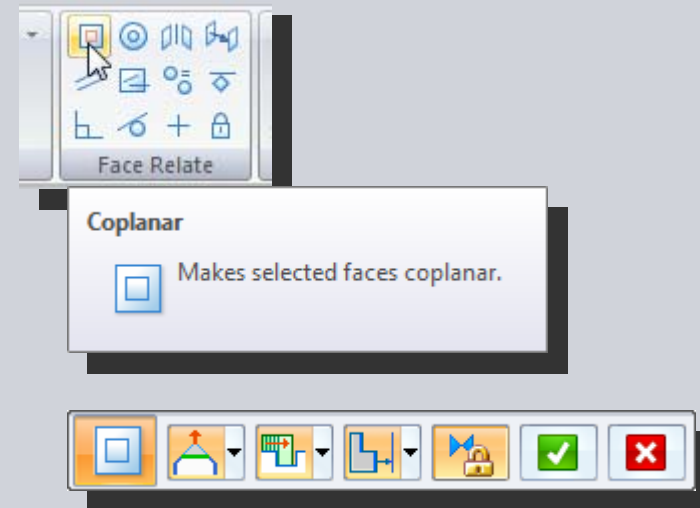
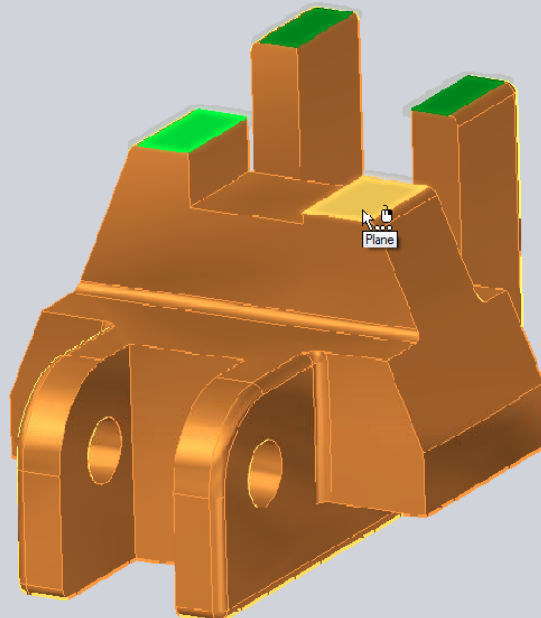


Select Target element

PromptBar
Click model elements to Relate. Deselect faces by holding Ctrl + Left Mouse click. Right click to continue. Click "X" to cancel.

Relate UI Enhancements

- Most Relate commands stay active until the select set is cleared
 - These are commands that require seed and target elements
 - Exceptions are:
 - Ground and Rigid: there is no target element selection for these relationship types
 - Symmetry: has a unique workflow from other relationships types in that it requires an additional selection of a mirror plane

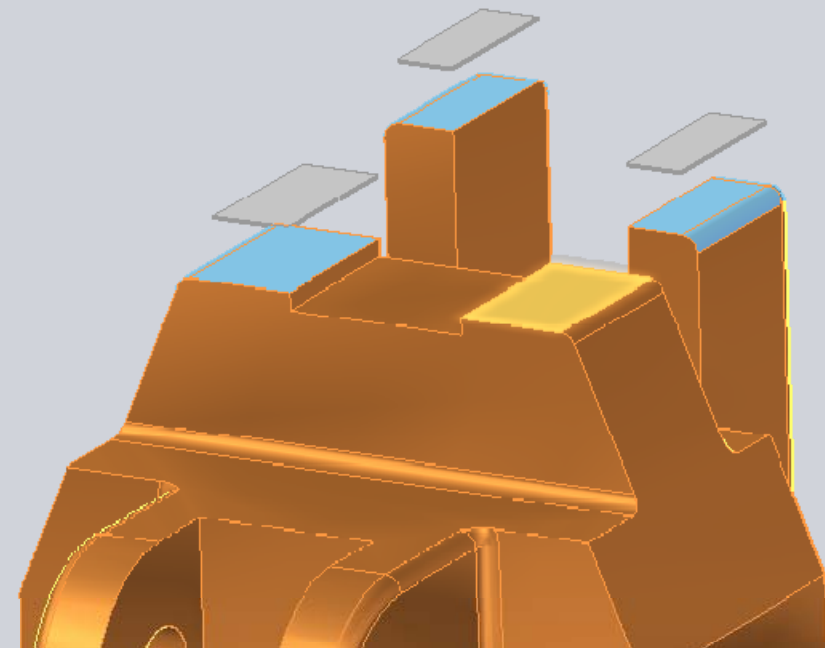
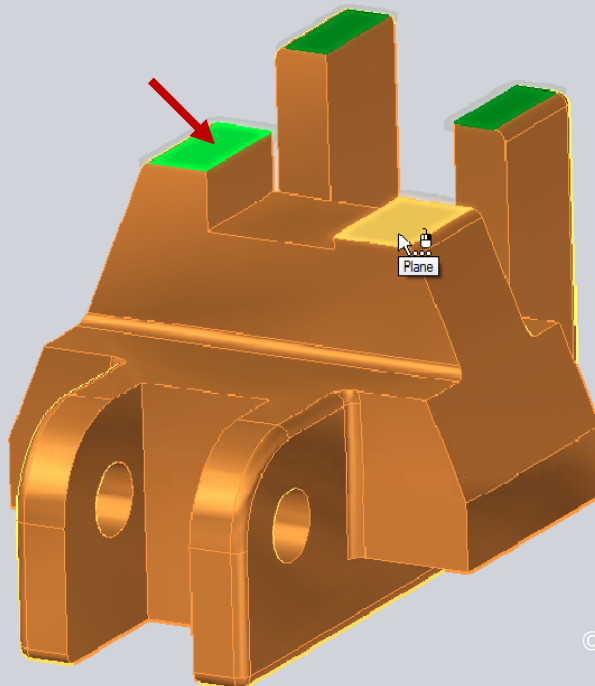
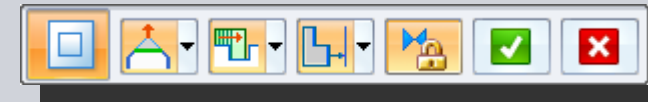
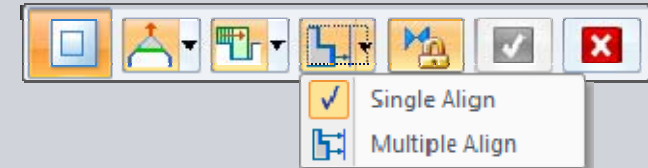


Relate UI Enhancements



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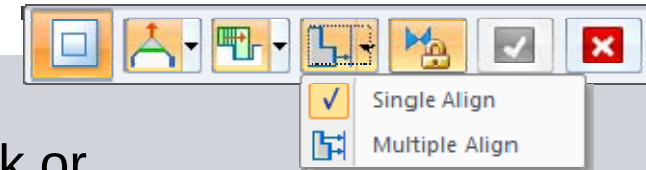
- If the "single align" selection option is used, the user has multiple seed faces
 - The first face selected is the seed face and is what aligns to the target initially
 - Other faces move as a group relative to the seed face



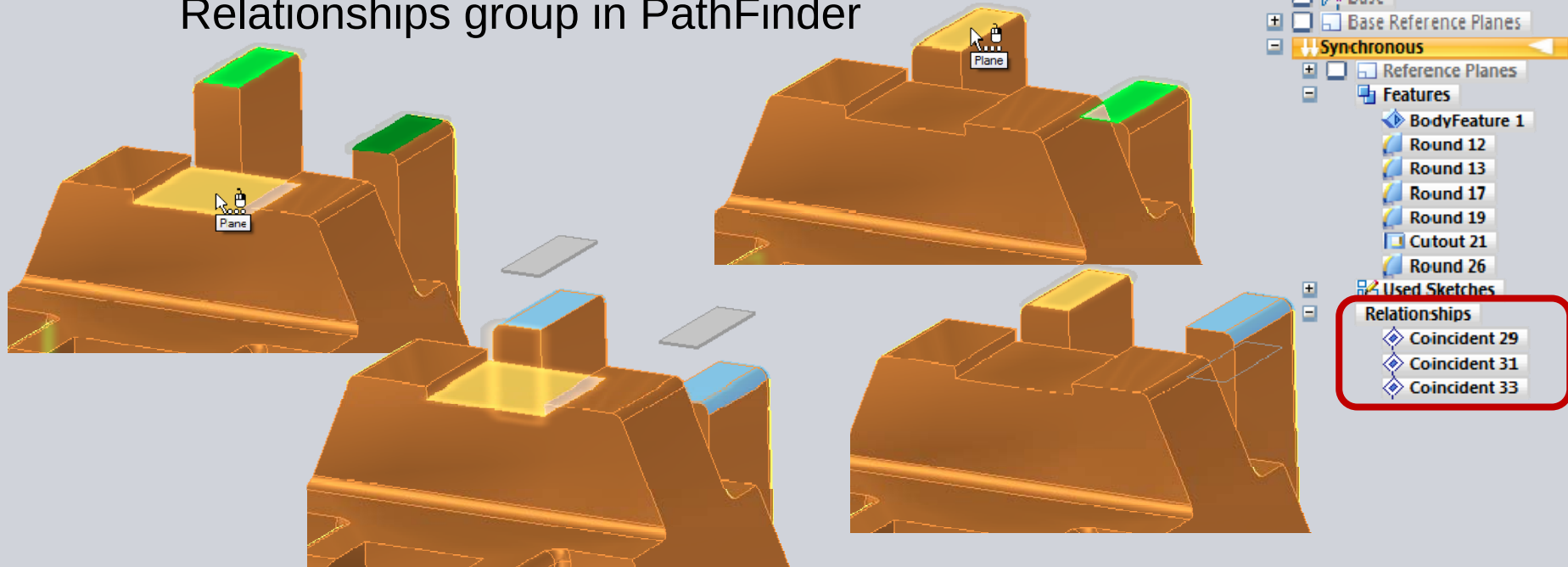
Relate UI Enhancements



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- Once the user selects the green checkmark or RMB clicks to accept the result, then the next face in the select set becomes the seed (Relate command stays active)
 - User can pick the same target face or a different one
 - If persisted, creates matched pair entries in the Relationships group in Pathfinder



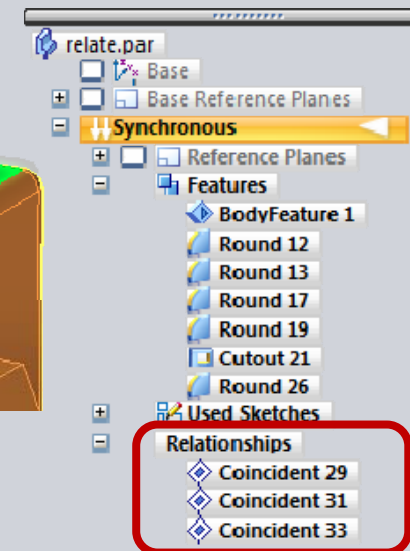
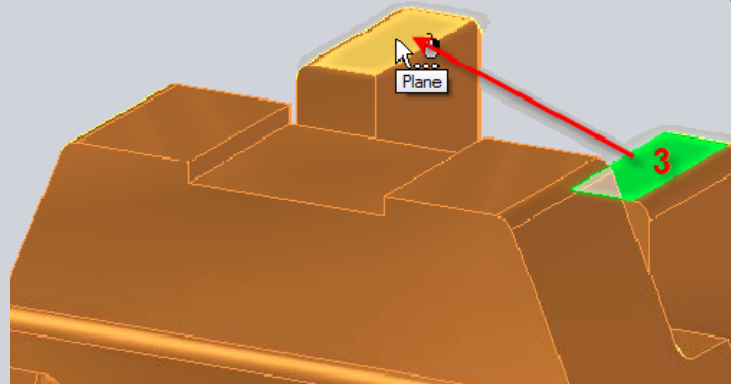
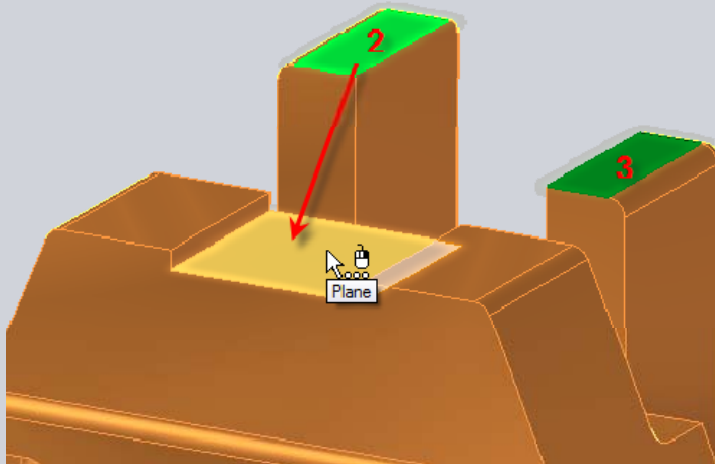
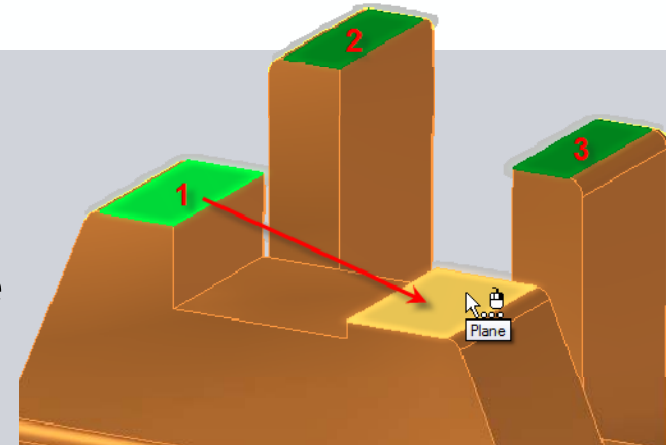
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Relate UI Enhancements



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- When these Relate command types have multiple seed faces, the user is presented with a diminishing select set as each cycle establishes a pair of relationships



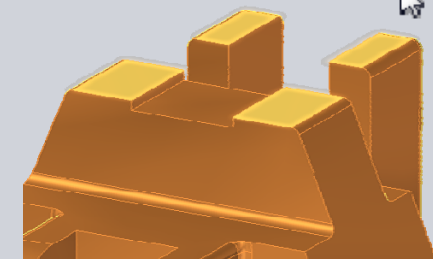
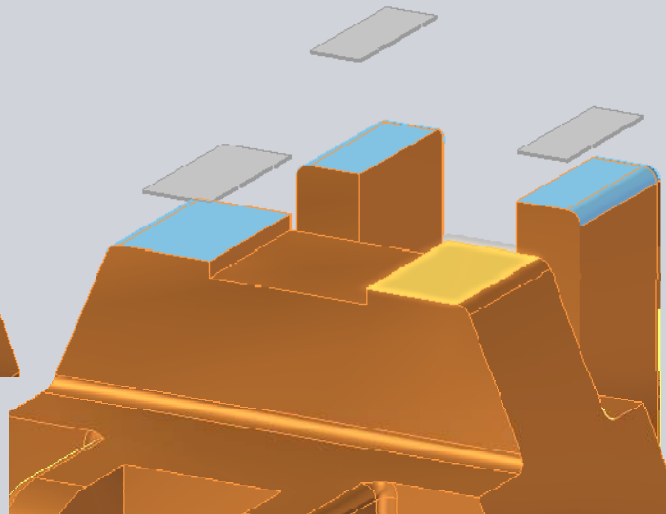
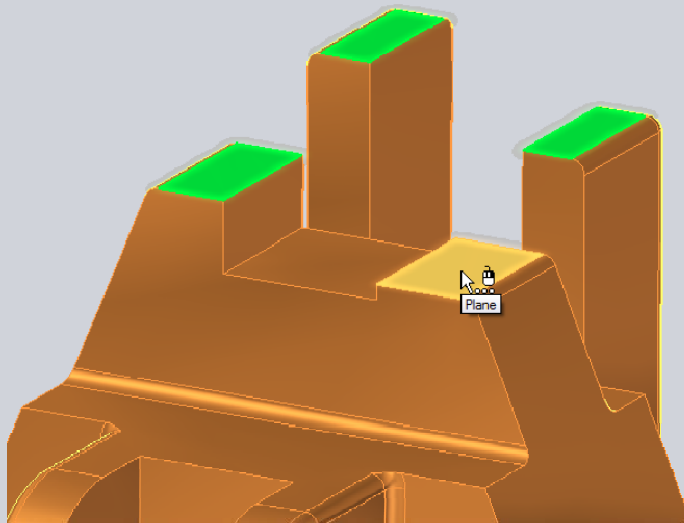
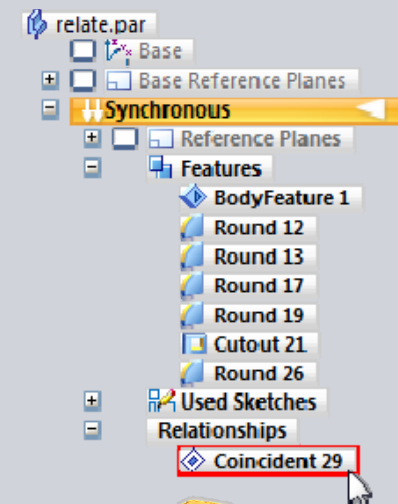
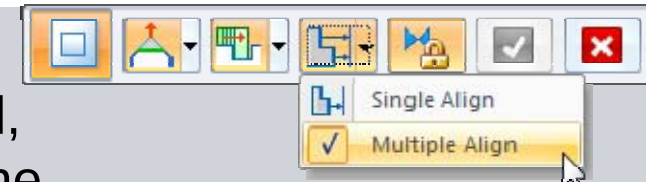
Note: Once a seed element is defined in a relationship, the element is removed from the select set and becomes available as a target element (Face 2 in this example)

Relate UI Enhancements



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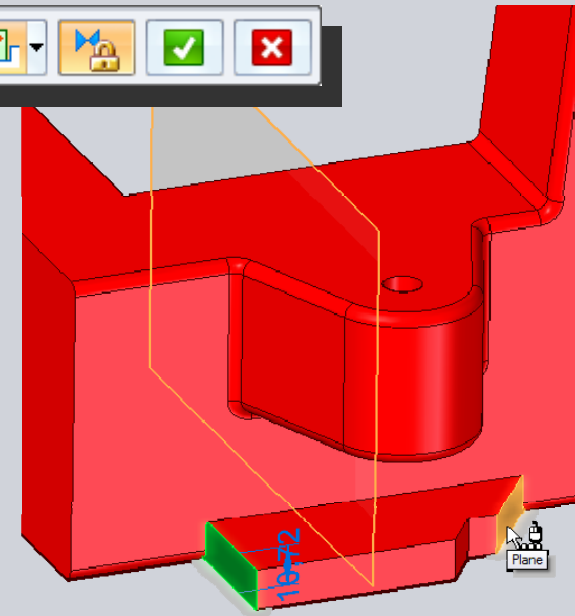
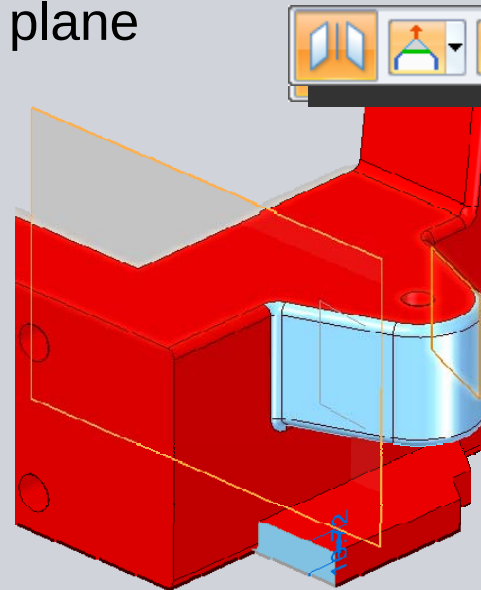
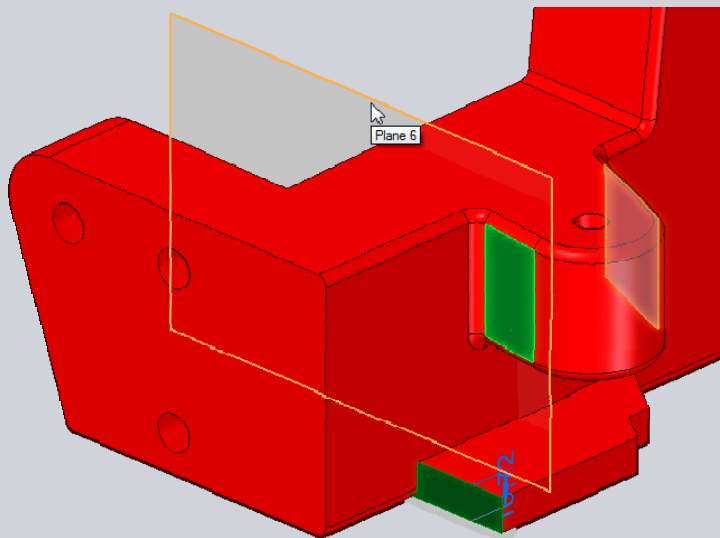
- If the "Multiple Align" selection option is used, the multiple seed faces all move to align to the target face at once
- If persisted, only one entry containing all selected elements is created in the Relationships group



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Relate UI Enhancements

- The Symmetry command requires a symmetry plane as a reference element in addition to the seed and target elements
- Multiple relationships may be created during one instance of the command
 - The relationships are defined by the same symmetry plane
 - Multiple pairs of seed and target elements are related to a single symmetry plane

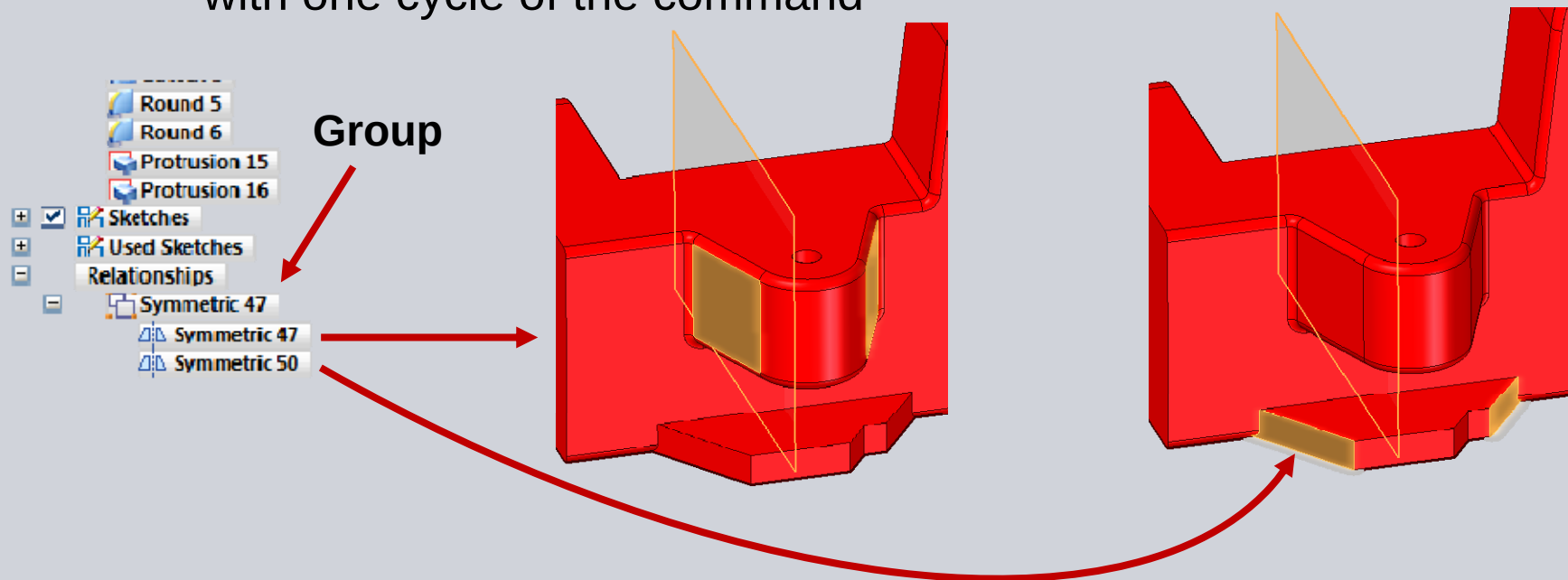


Relate UI Enhancements



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- Each cycle of the command relates the defined seed and target as a pair
 - Each instance produces a new "Symmetric xx" group
 - Each cycle produces a relationship within the group
 - Groups are not created unless multiple relationships were created with one cycle of the command



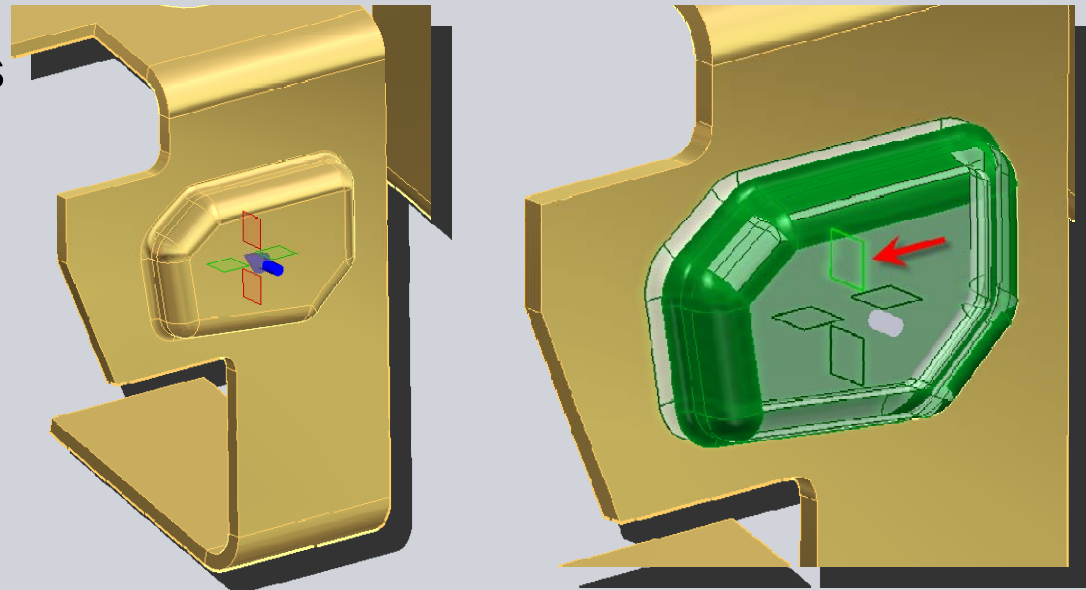
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Relate UI Enhancements



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- Faces of a procedural feature can be used as seed or target faces
 - Procedural features move as a rigid set of faces
 - Procedural features do not change shape based on relationships
- Feature Origin planes are found first, by default, when a Procedural Feature is selected (Object/Action) as the seed input to a Relate command
 - Feature Origin planes and keypoints can be used as input to relate



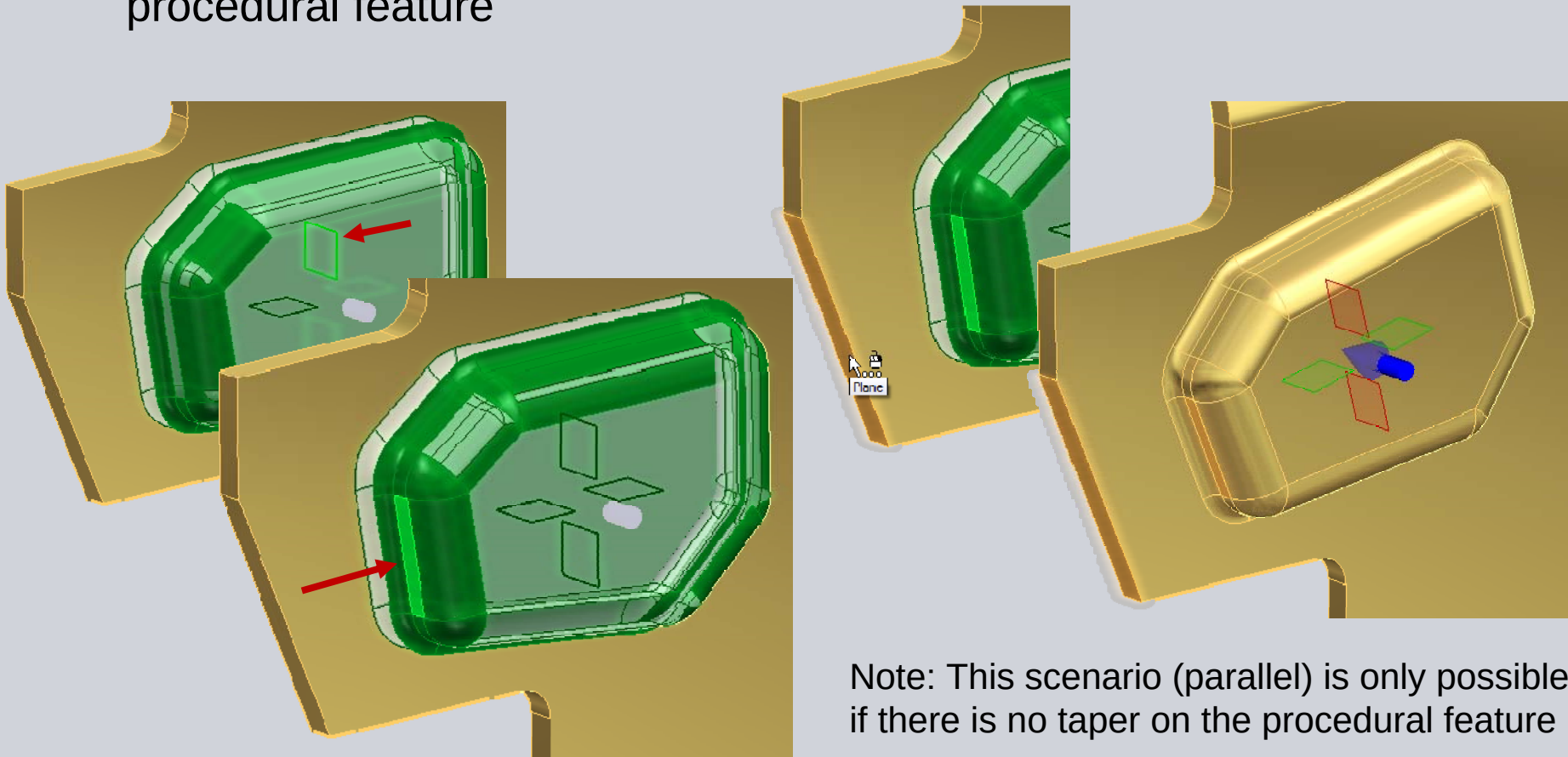
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Relate UI Enhancements



SIEMENS

- User can switch the seed element from the default to a face of the procedural feature



Note: This scenario (parallel) is only possible if there is no taper on the procedural feature

DEMO 4

Page 35

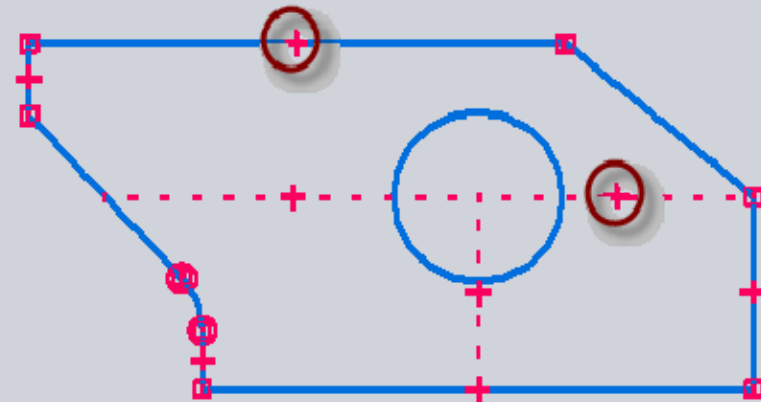
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Siemens PLM Software

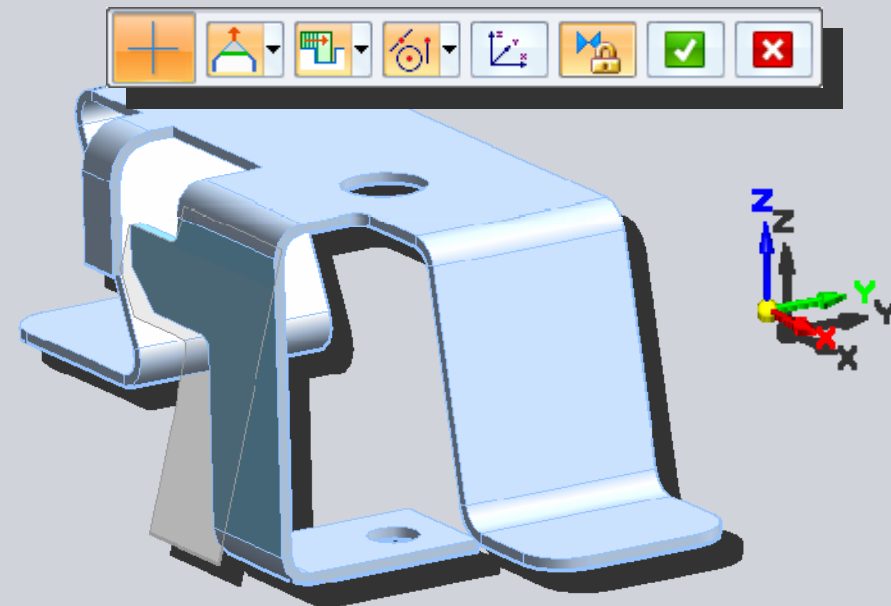
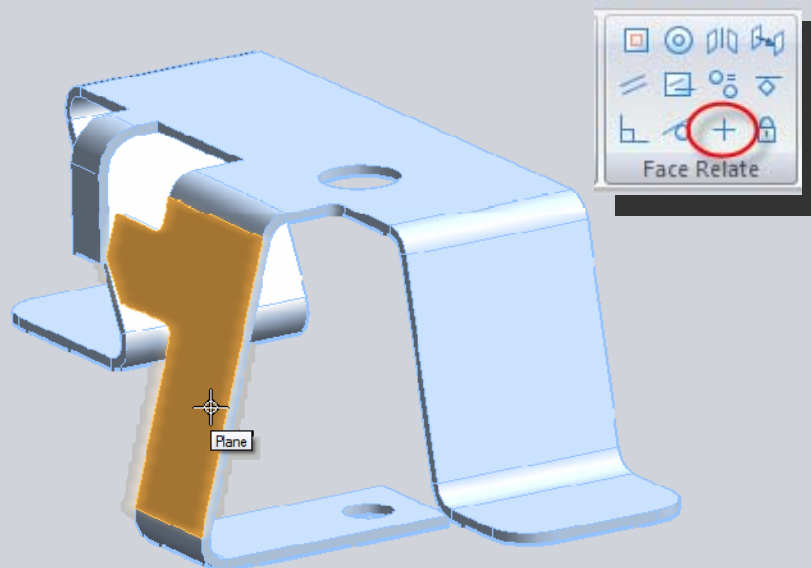
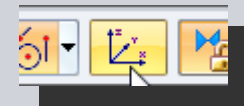
Horizontal/Vertical Relationship

- A new Horizontal/Vertical relationship is a synchronous only relationship type that applies in Part, Sheet Metal and Assembly
- The new Horizontal/Vertical relationship button has been added to the command ribbon in the Face Relate group
 
- Its function is similar to the horizontal/vertical relationship that is applied to 2D geometry
 - Used to snap skewed linear elements to a horizontal or vertical orientation
 - Used to maintain horizontal or vertical orientation
 - Used to align keypoints in a profile



Horizontal/Vertical Relationship

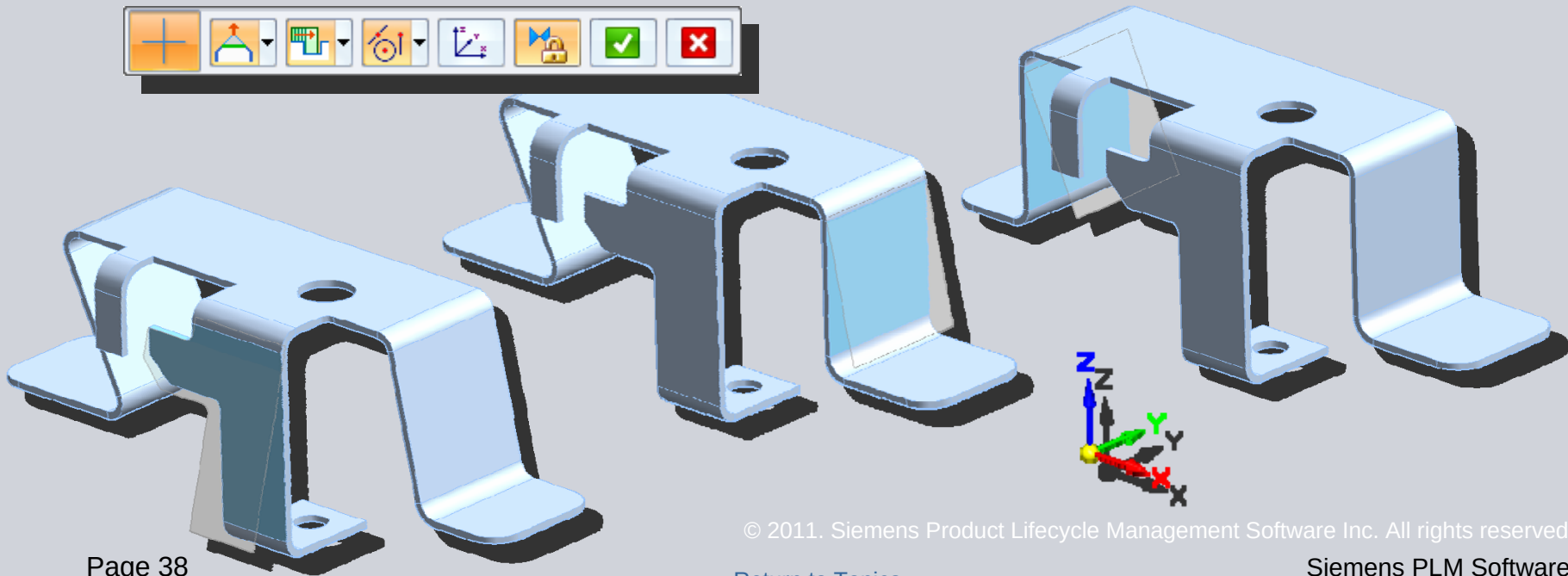
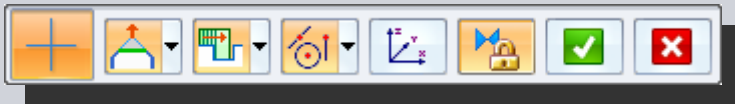
- In 3D, skewed planar faces can be snapped to horizontal or vertical
 - Relative to the Base Coordinate System
 - Relative to a selected plane or face
 - Behaves the same as a persisted parallel relationship
- By default this relationship is persisted (just like adding it in 2D in the ordered workflow)



Horizontal/Vertical Relationship

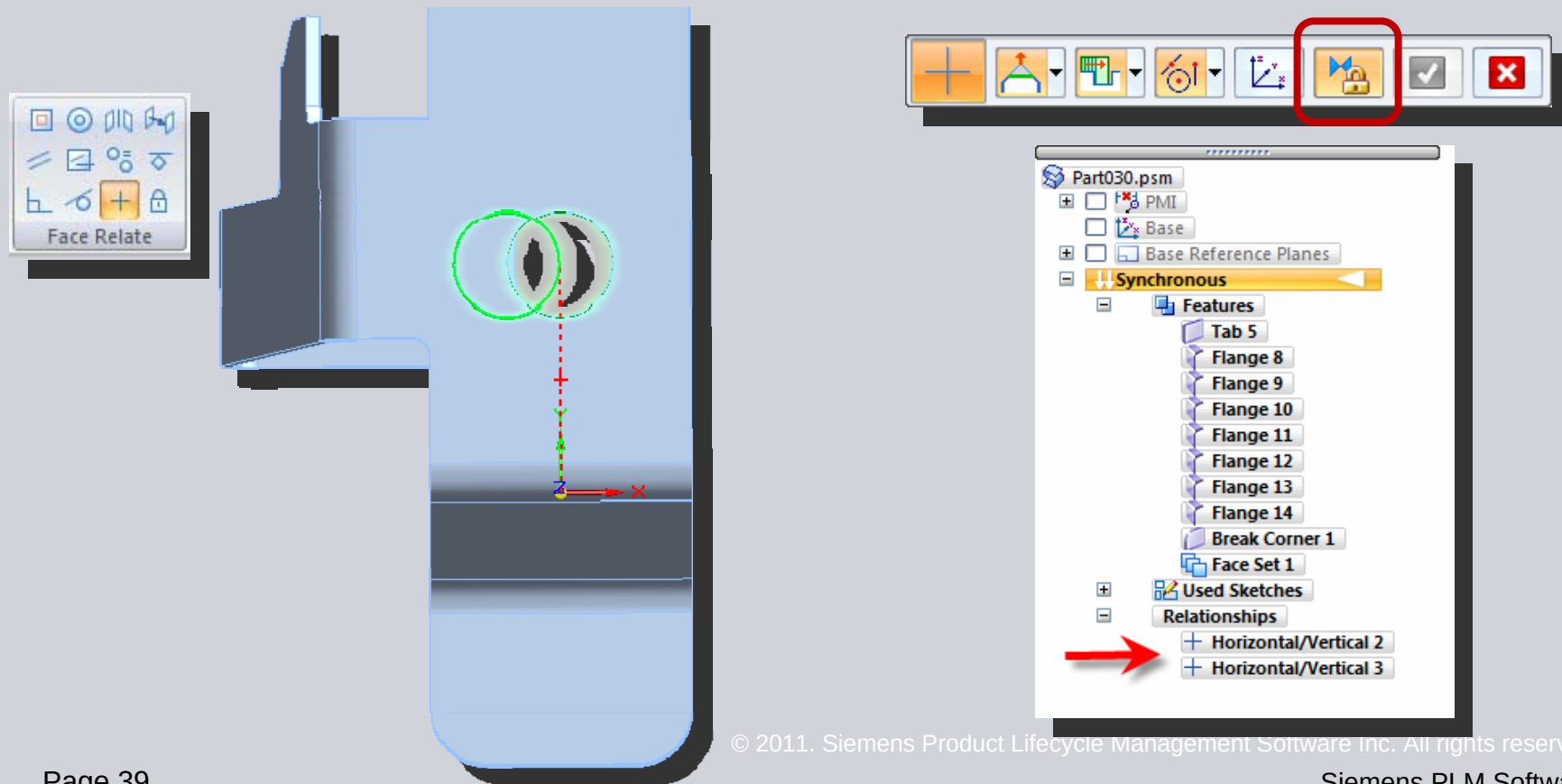


- Snapping to Horizontal/Vertical workflow:
 - Select the Horizontal/Vertical relationship from face relate group
 - Select a planar face
 - RMB click or select green checkmark to accept reorienting the selected face
 - Continue to select faces you wish to snap to Horizontal/Vertical



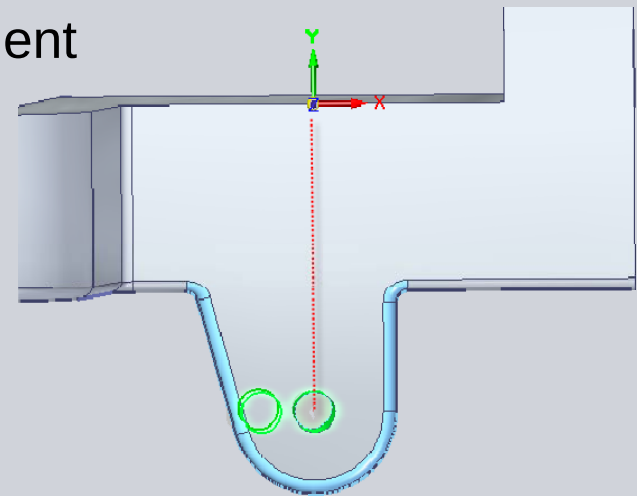
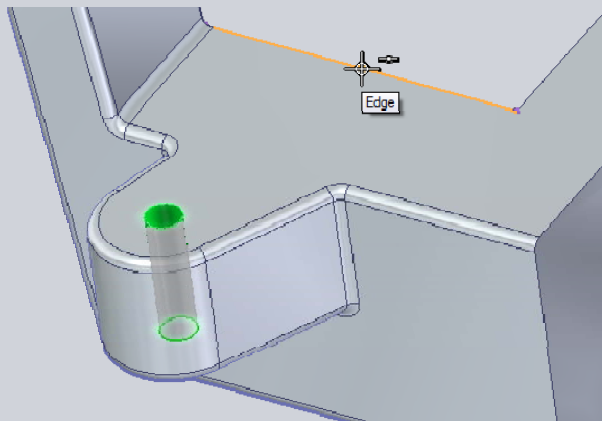
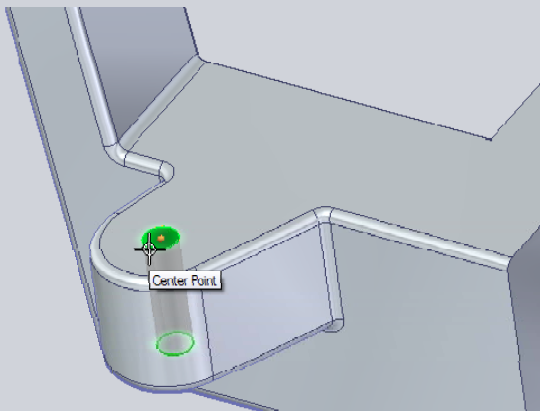
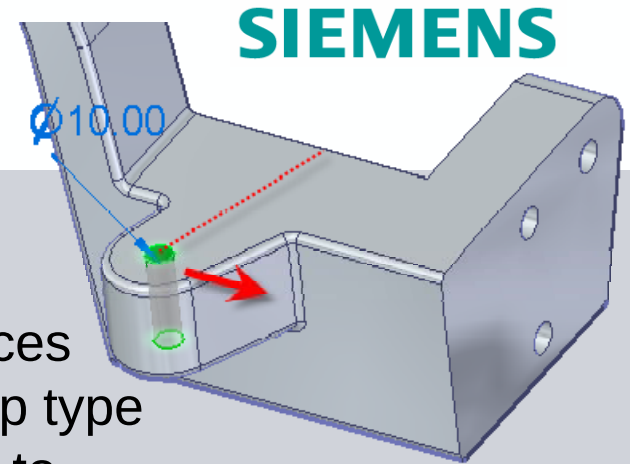
Horizontal/Vertical Relationship

- The new Horizontal/Vertical relationship can be used to align keypoints in the 3D model
 - By default, the Horizontal/Vertical relationship is persisted (saved)



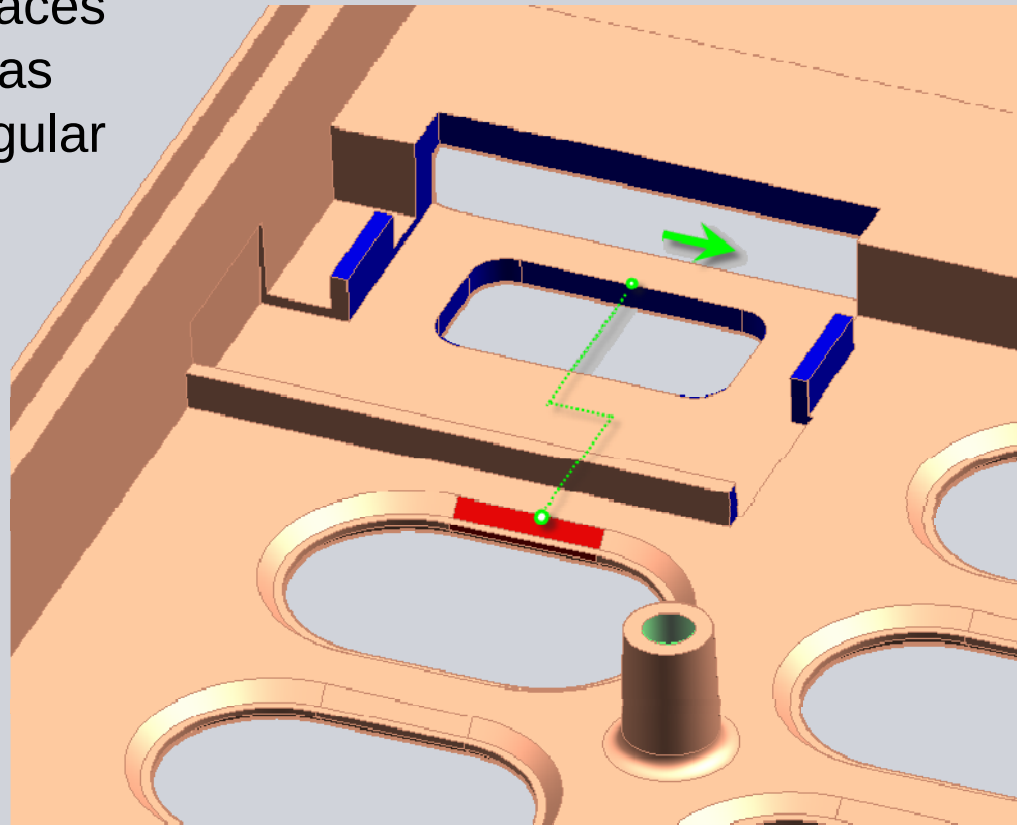
Horizontal/Vertical Relationship

- To align keypoints:
 - User creates a selection set of one or more faces
 - User selects the Horizontal/Vertical relationship type
 - Next the user selects the first keypoint relative to the "seed" faces
 - Only seed faces/keypoints are moved
 - Target keypoints do not move during any relate operation
 - Finally the user selects the keypoint to align to
 - Seed geometry moves to create the alignment



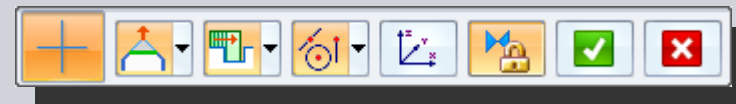
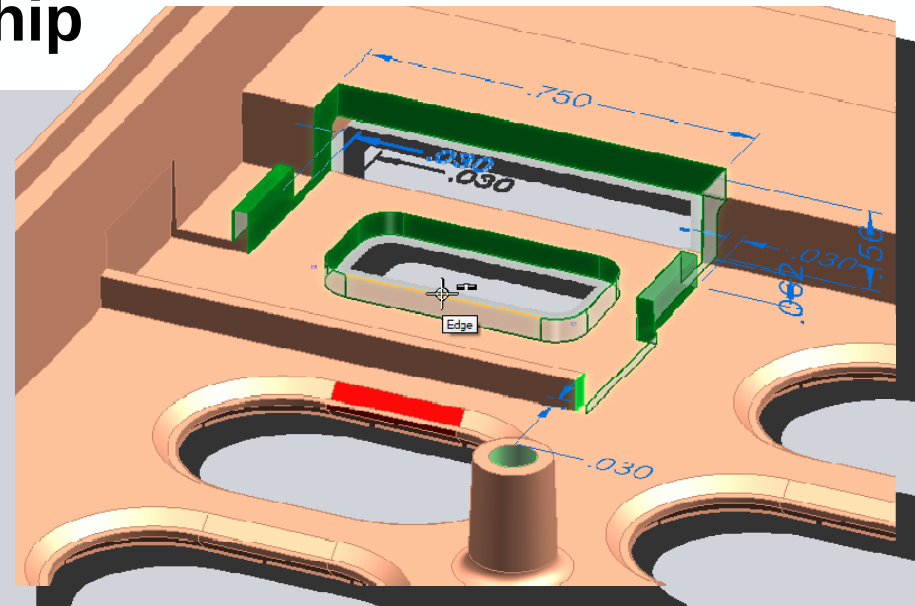
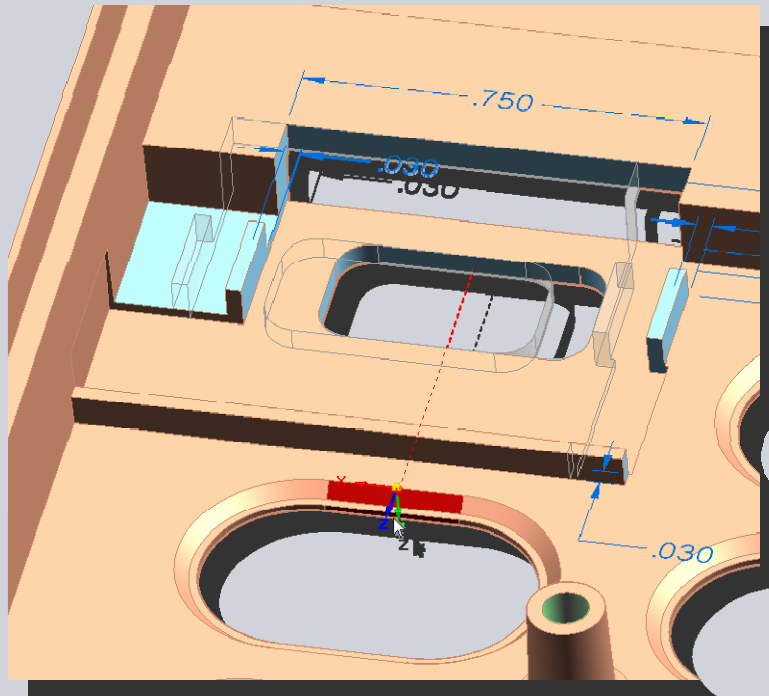
Horizontal/Vertical Relationship

- Multiple faces can be selected to be moved as a rigid set while the horizontal/vertical relationship is being created
 - In this example, the blue faces are to be moved together as the midpoint of the rectangular slot is to align horizontal/vertical with the midpoint of the red face's edge



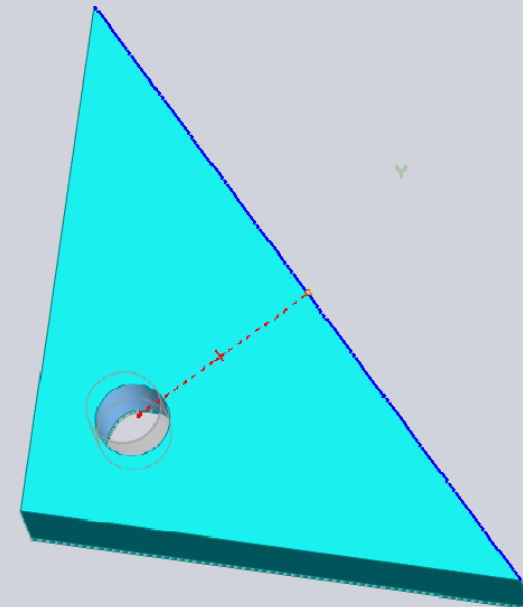
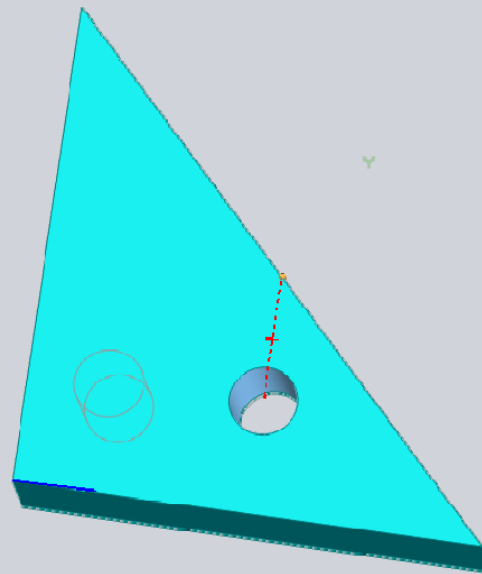
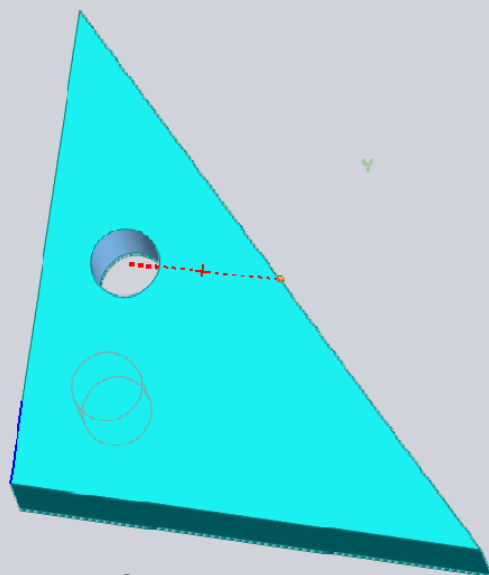
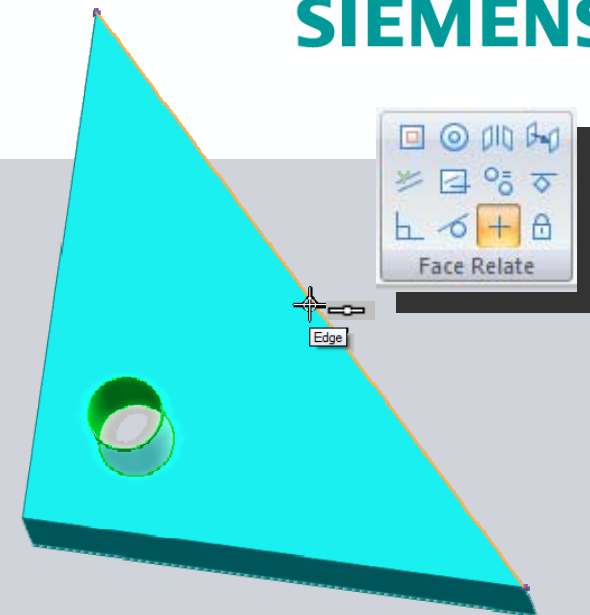
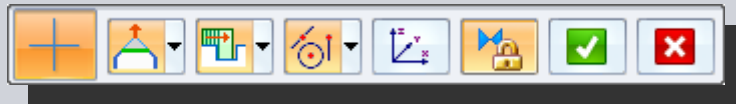
Horizontal/Vertical Relationship

- User selects the blue faces (fence)
- Select the Horiz/Vert Relationship
- Select the midpoint
- Right click to accept
- Select target midpoint



Horizontal/Vertical Relationship

- During preview, the "N" key can be used to toggle between nearest horizontal or vertical alignments or edge-based alignments

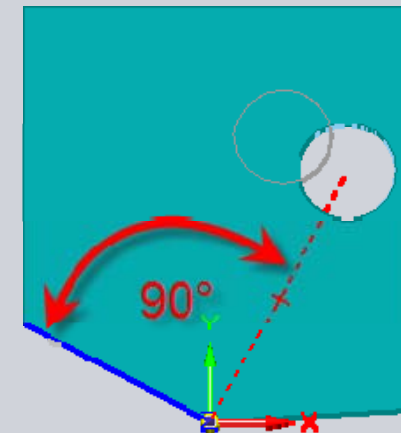
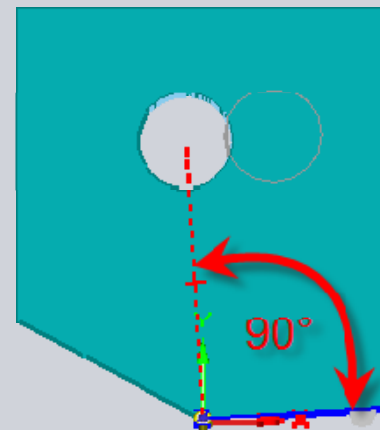
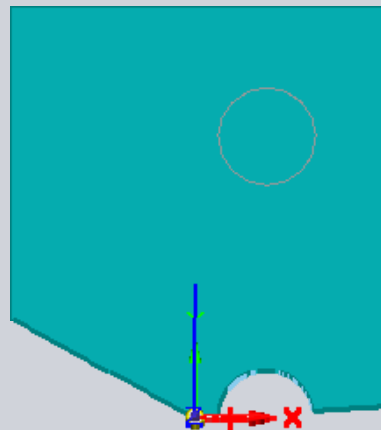
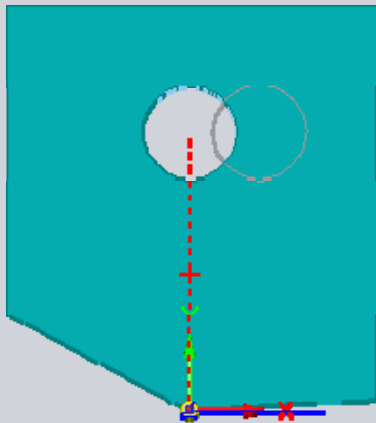
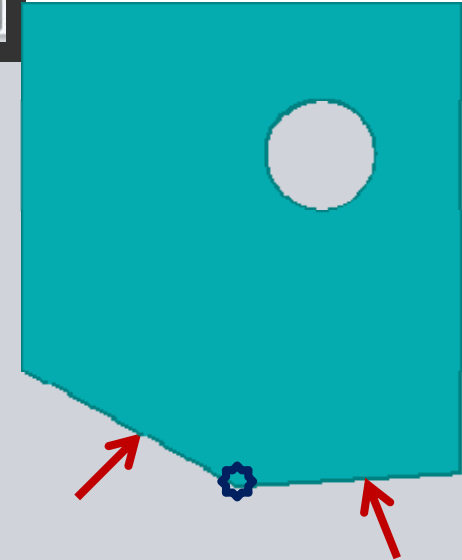


Horizontal/Vertical Relationship



SIEMENS

- If the Keypoint has more than one linear edge associated with it, each scenario is offered as a solution when toggled with "N"
 - Global coordinate system alignments
 - Both edge alignments

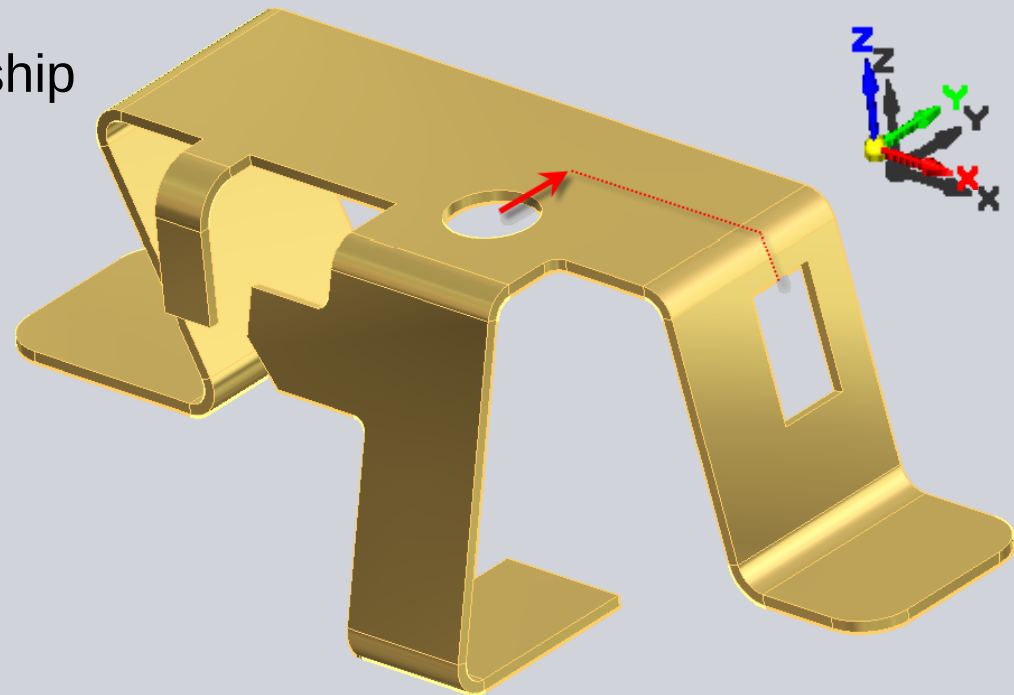


Horizontal/Vertical Relationship



SIEMENS

- In many cases, view orientation matters!
 - In 2D it is just a matter of moving seed elements along X or Y axis
 - In 3D we add moving along the Z axis
 - One of these axis gets dropped from the solve based on view orientation when using a horizontal/vertical relationship in 3D
- Example: the user wants to align the hole with the midpoint of the rectangular cutout
 - Moving the hole along Y axis is required

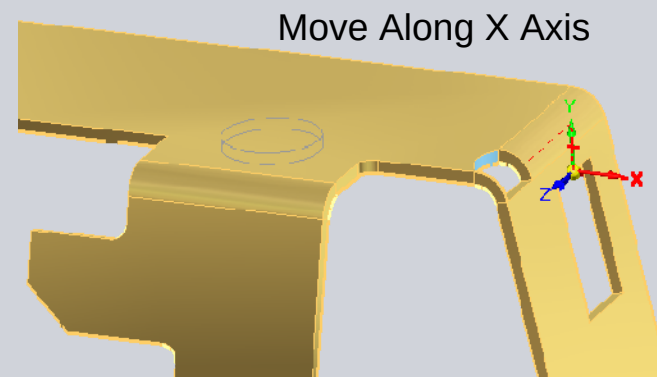
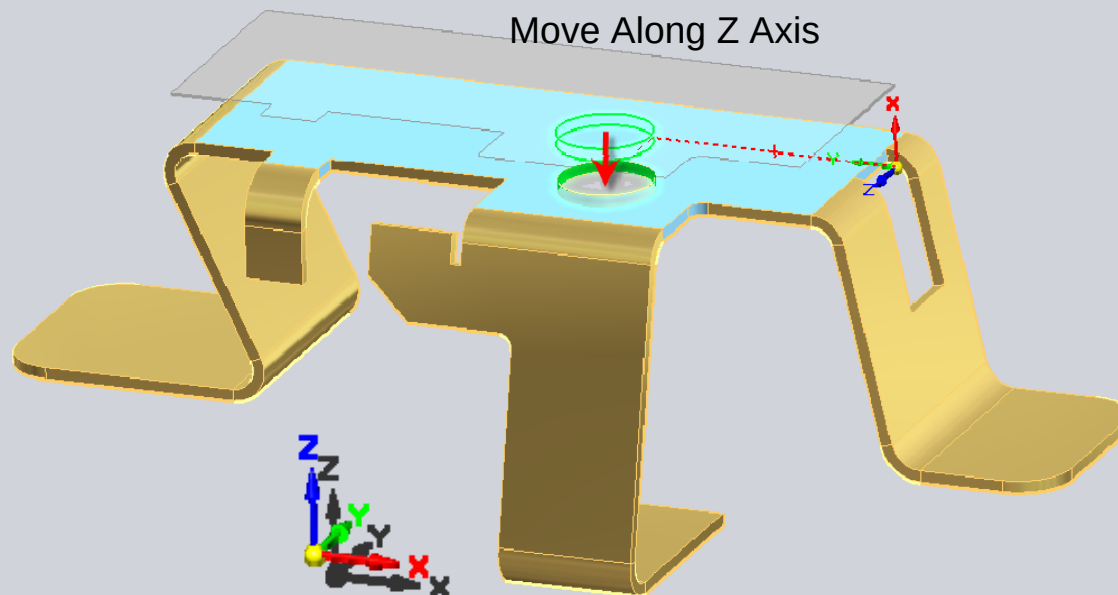
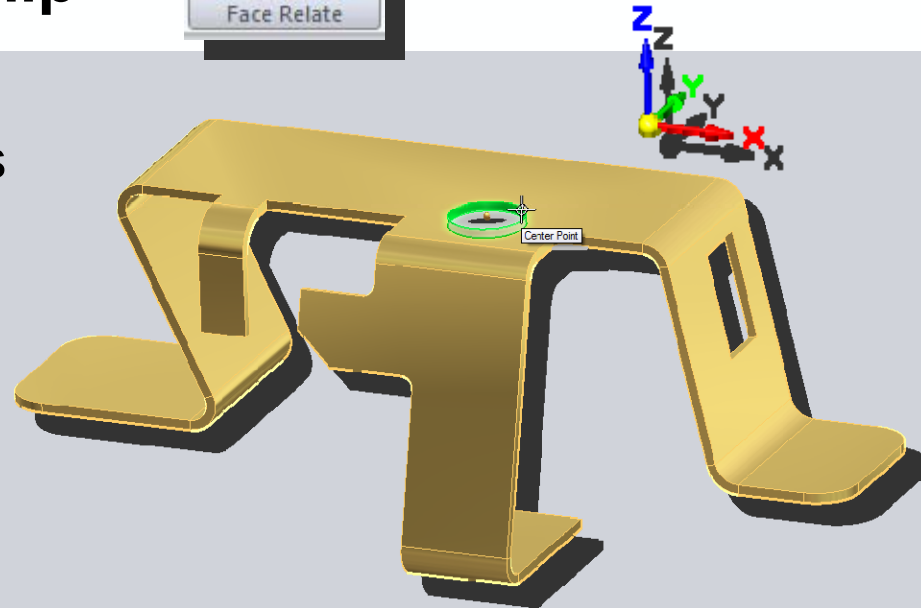


Horizontal/Vertical Relationship



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- Based on the view orientation in this image, Z and X are most normal to the view
 - Y axis is dropped out of the solve, so from this orientation we will not get our desired result



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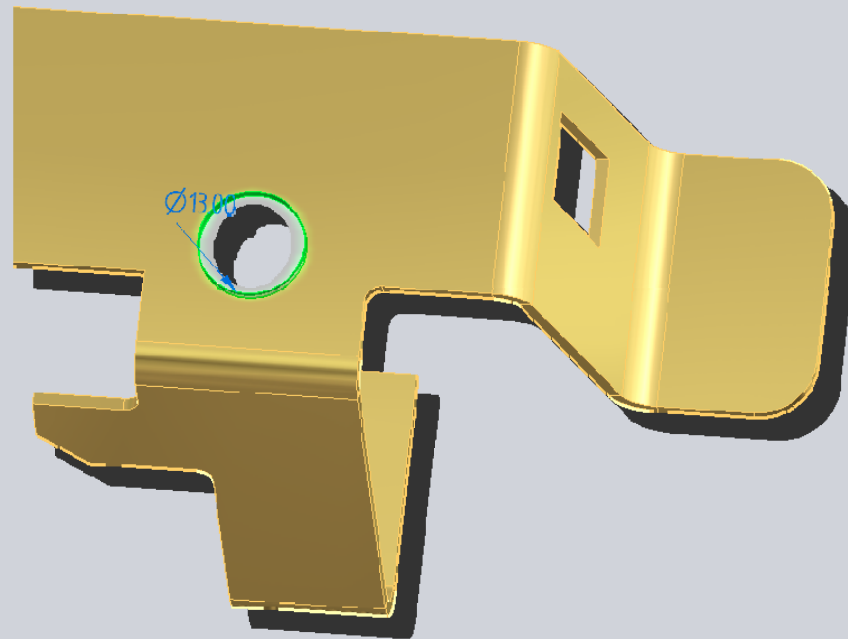
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Horizontal/Vertical Relationship

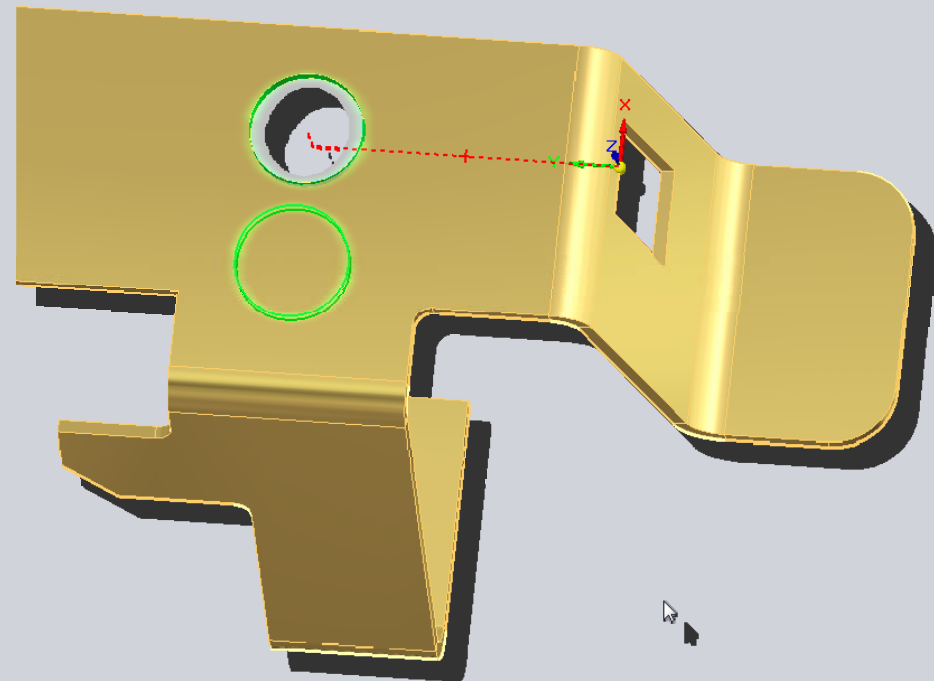


SIEMENS

- The best practice is to rotate the model to make the direction the object is to be moved most normal to the view before running the command
 - Z axis is dropped from the solve



Move Along Y Axis

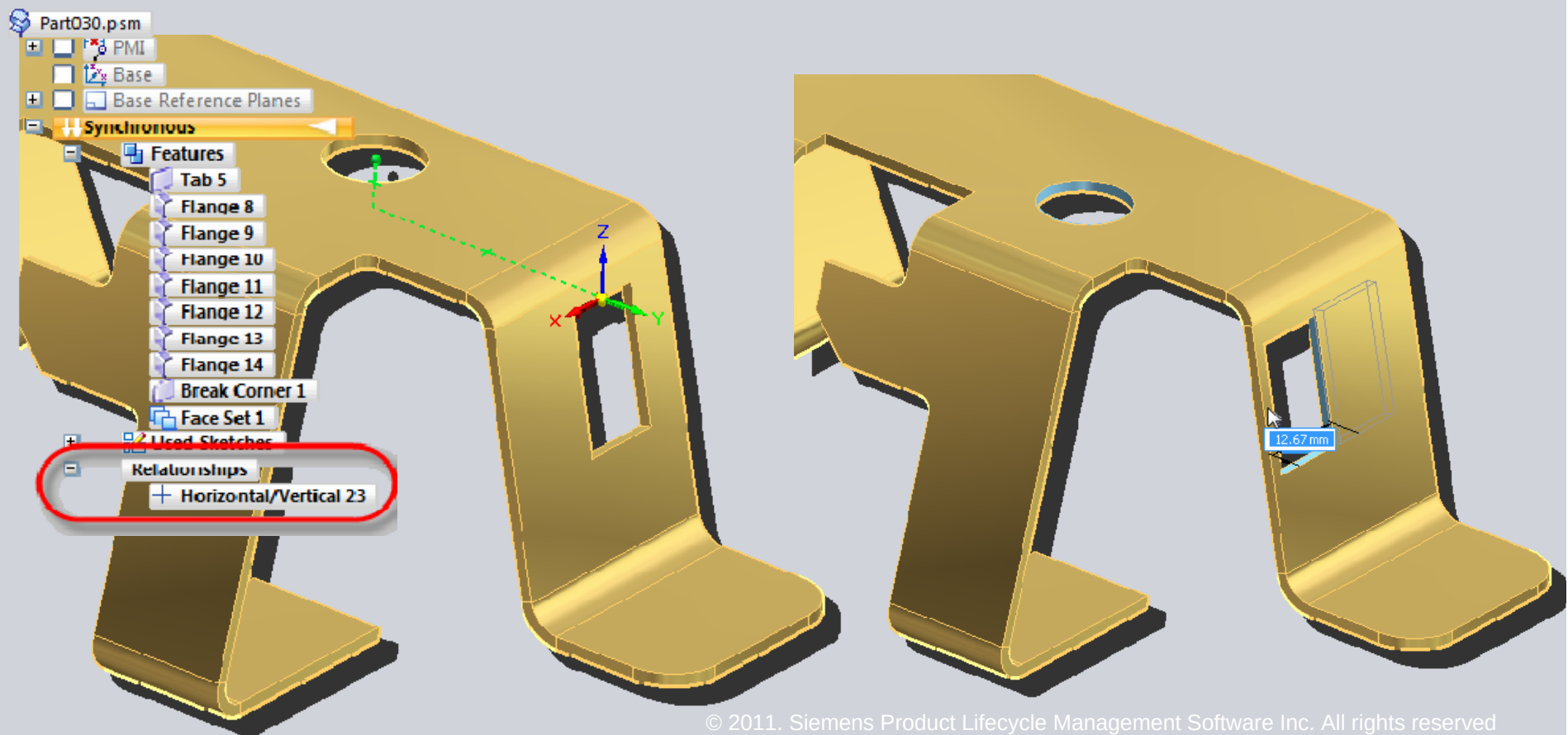


Horizontal/Vertical Relationship



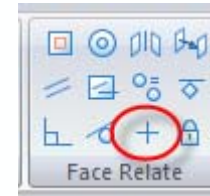
SIEMENS

- Remember this relationship is persisted (saved) by default
 - During later synchronous changes, the design intent is preserved



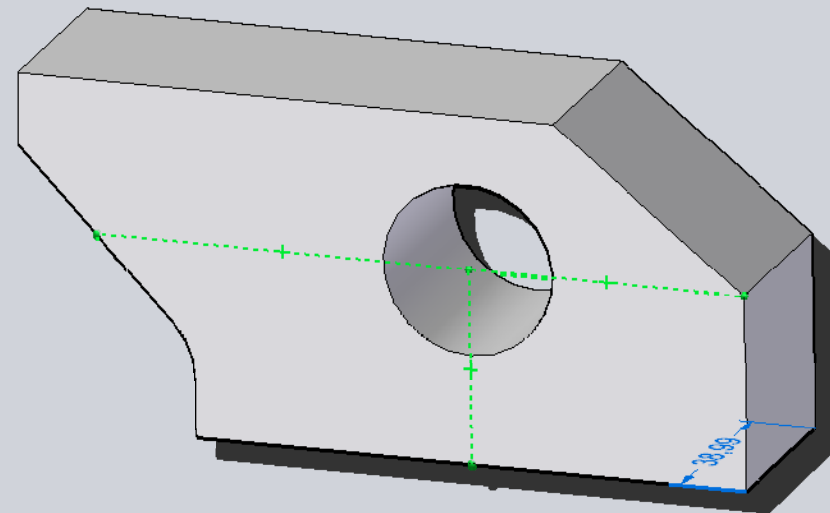
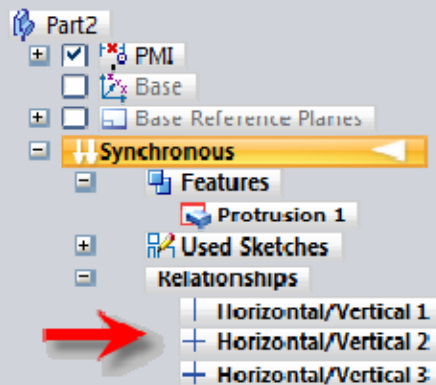
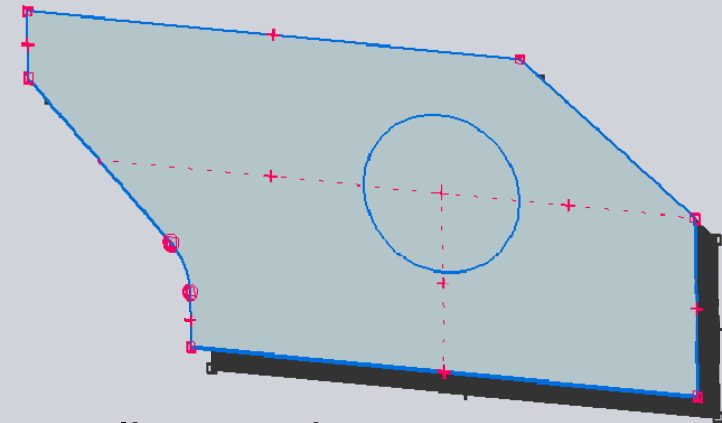
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Horizontal/Vertical Relationship



SIEMENS

- 2D horizontal and vertical constraints now transfer to 3D in a synchronous part as persisted relationships
 - Relationships are created using the coordinate system implied by the sketch plane
 - NOTE that horizontal/vertical constraints on linear edges are not transferred to 3D faces
 - Could limit sync workflows

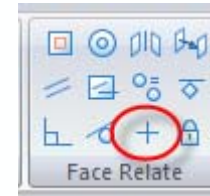


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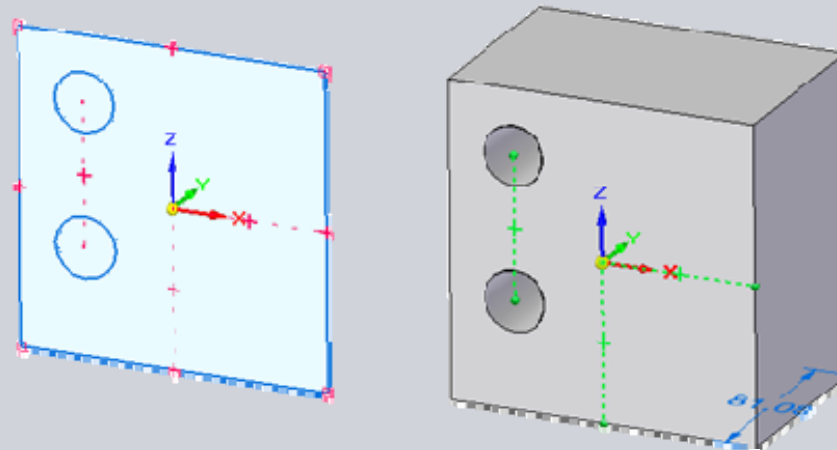
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Horizontal/Vertical Relationship



SIEMENS

- The following relationships are not created automatically from the 2D sketch moving to 3D
 - These scenarios are handled by Live Rules
 - Coplanar Axis
 - Symmetry

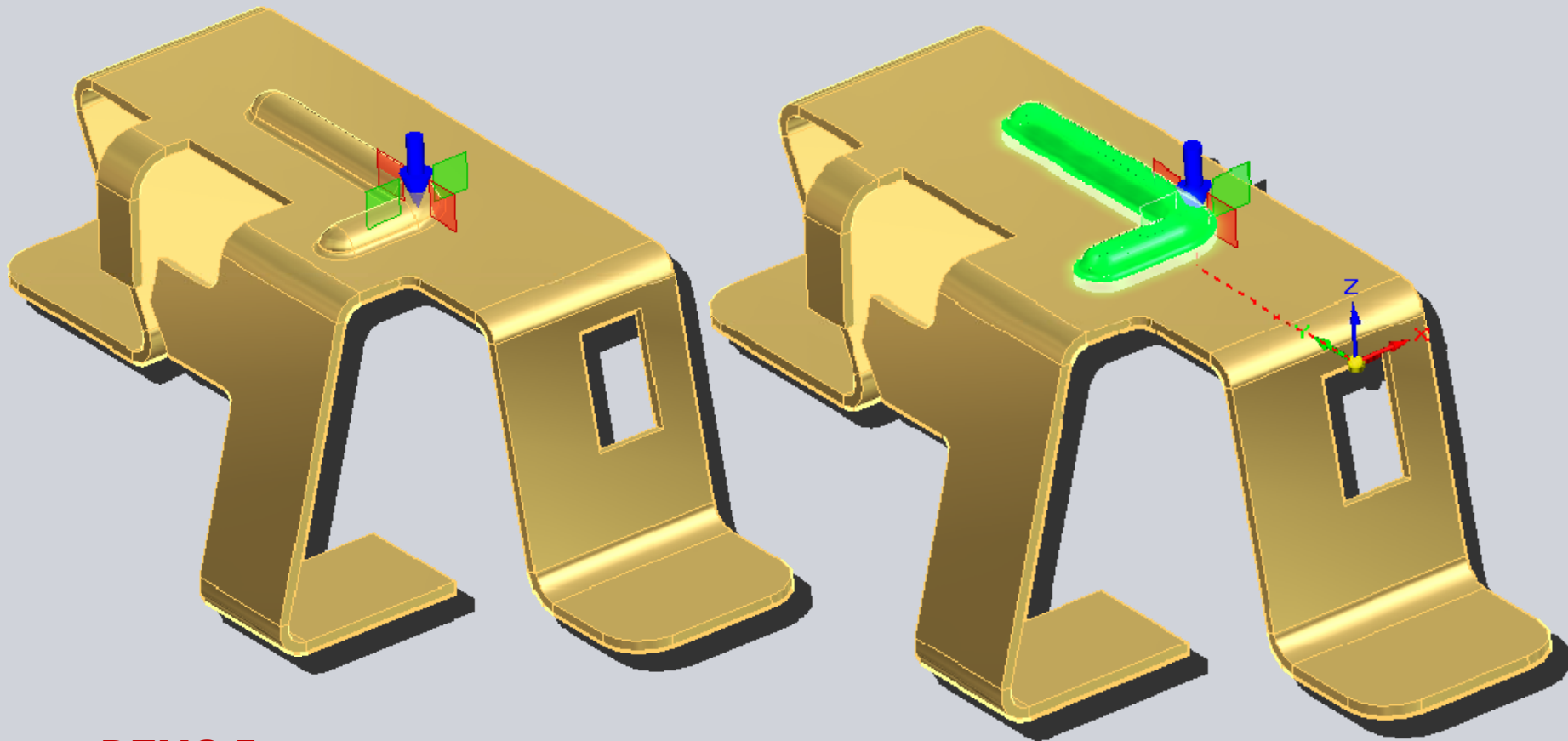


Horizontal/Vertical Relationship



SIEMENS

- Feature origins can be used for horizontal/vertical alignments



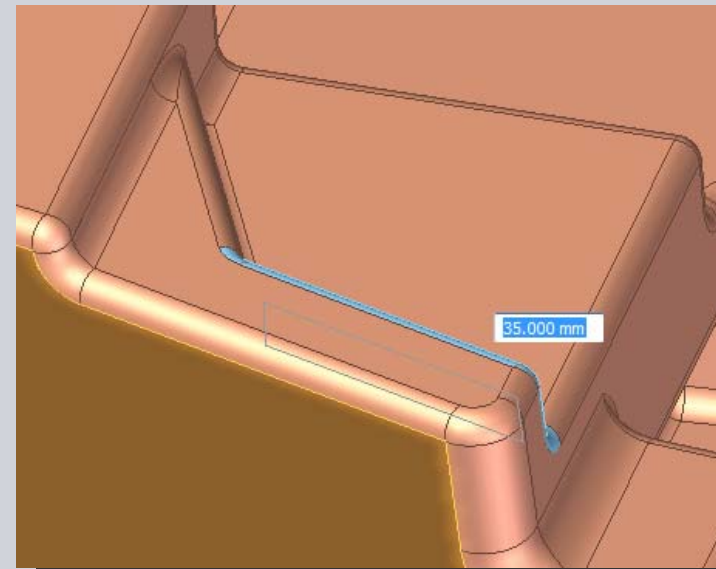
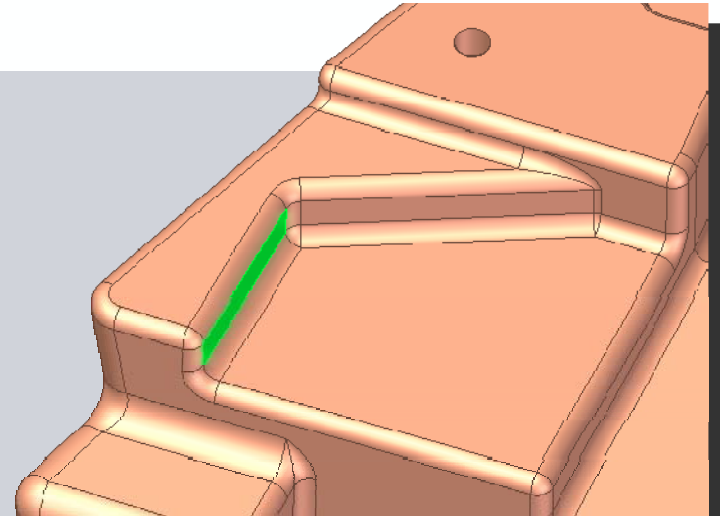
DEMO 5

Offset Relationship



SIEMENS

- A new Offset relationship is a synchronous only relationship type that applies in Part, Sheet Metal, and Assembly
- The new Offset relationship button has been added to the command ribbon in the Face Relate group
- Used to align edges to be parallel and at some offset value

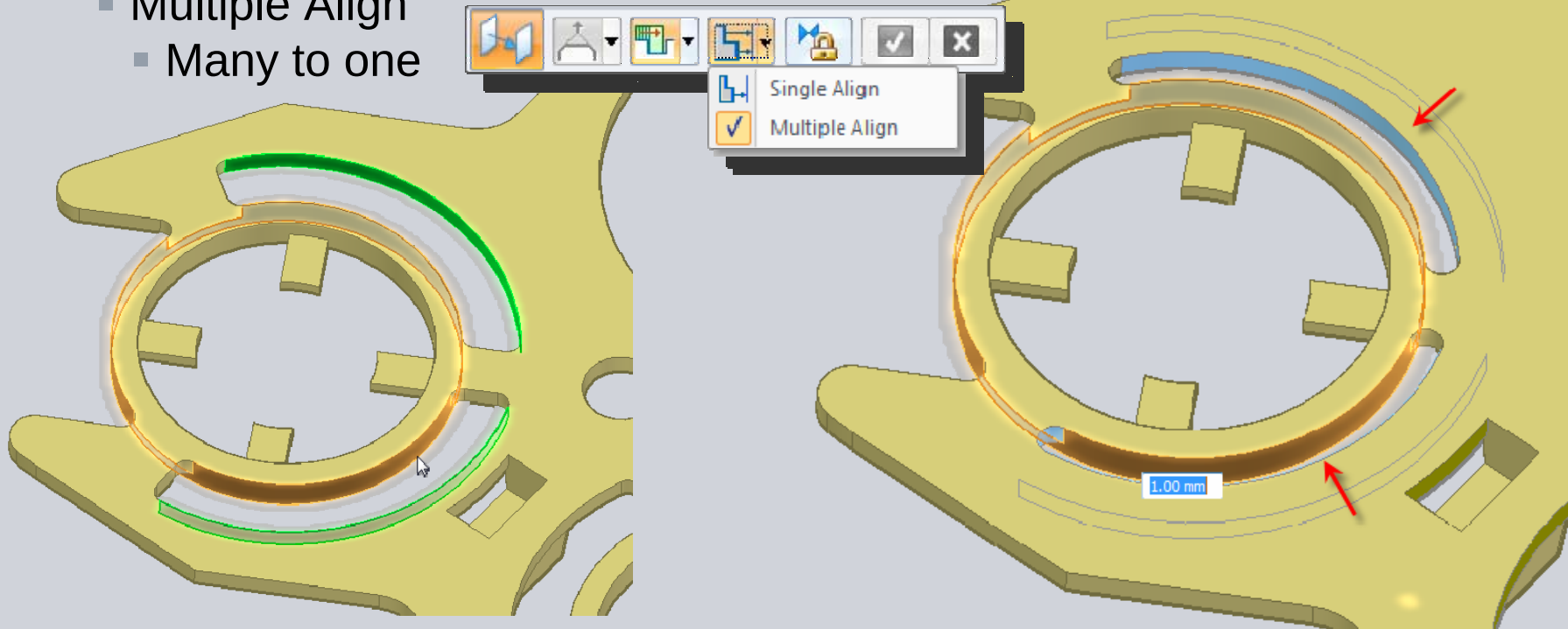


Offset Relationship



SIEMENS

- This command supports multiple offset pairing iterations within a single run of the command
 - Single Align
 - Iterative pairing if multiple faces are selected
 - Multiple Align
 - Many to one

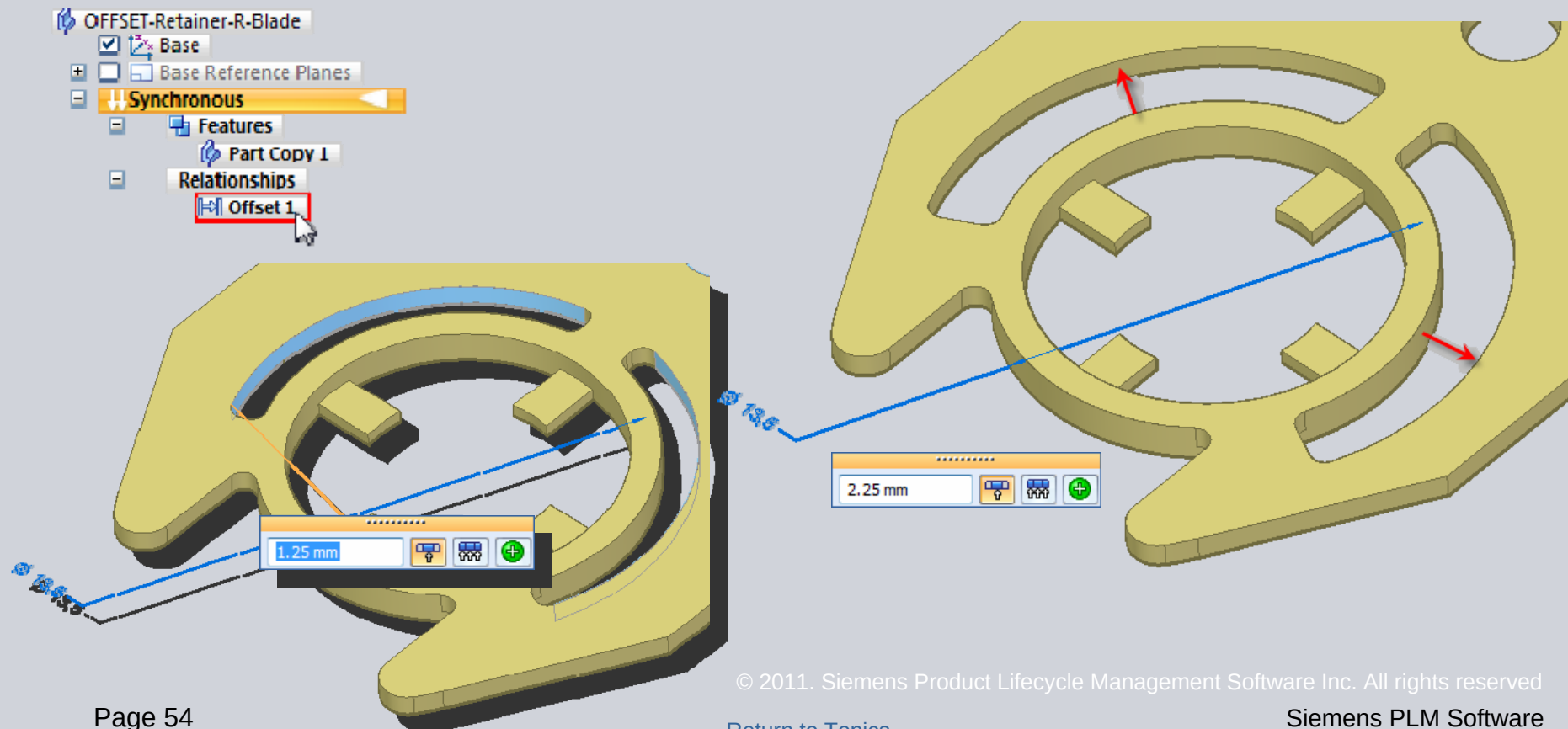


Offset Relationship



SIEMENS

- Multiple Align operation applies the offset value between the seed faces to a single target
 - Value from the first seed face is what drives the other seed faces



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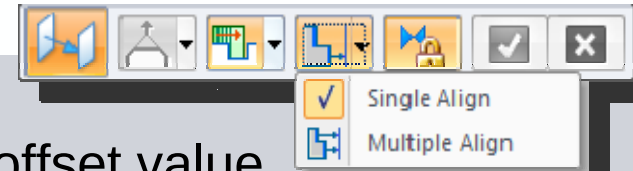
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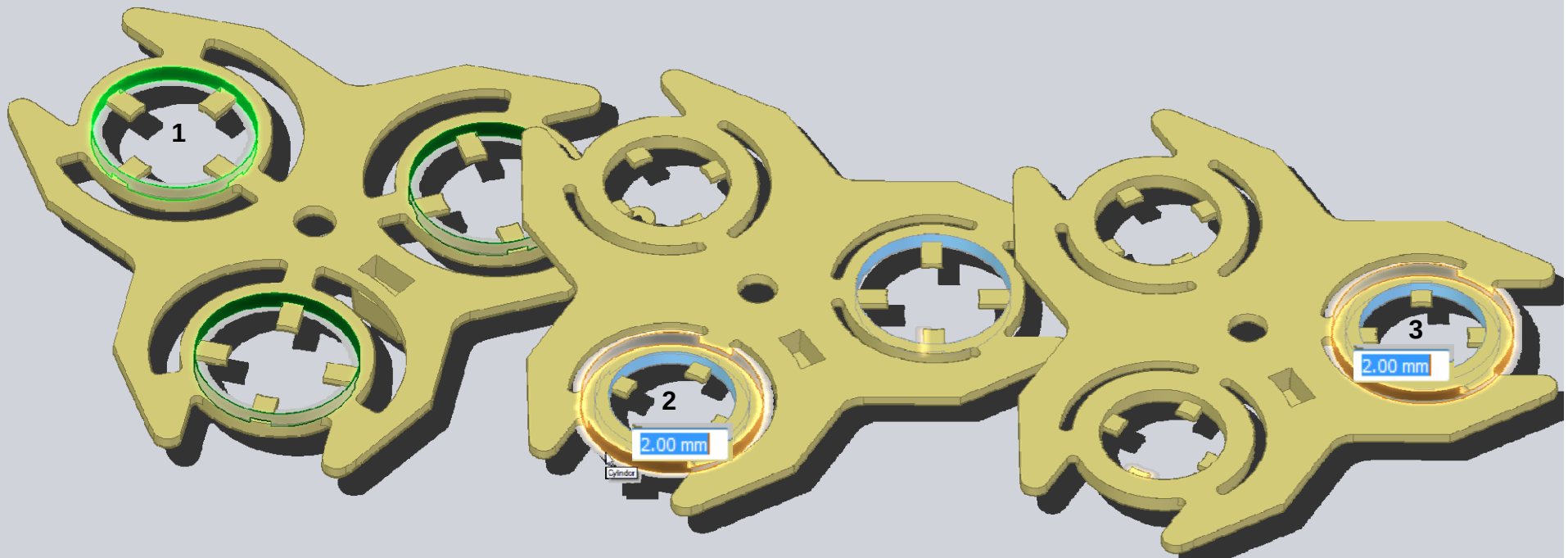
Offset Relationship



SIEMENS



- Each Seed/Target pairing may have a unique offset value
 - The **initial** default offset is the “minimum distance” between the first Seed and the Target face
 - Subsequent Seed/Target pairs **reuse the value from the last iteration** as the default offset (2.00 mm in this example)



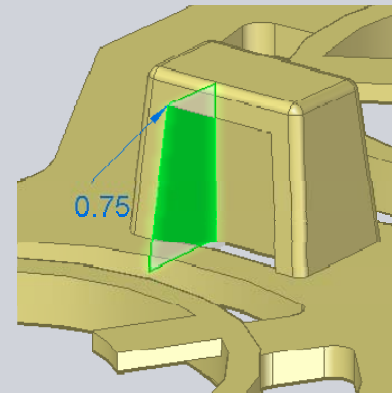
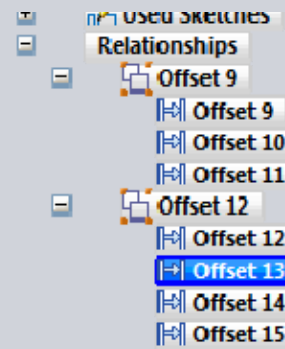
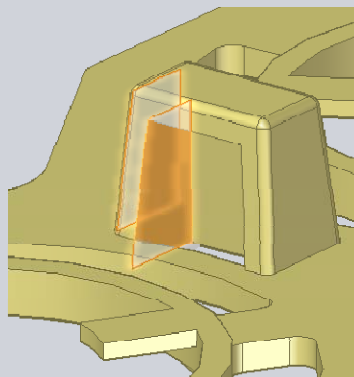
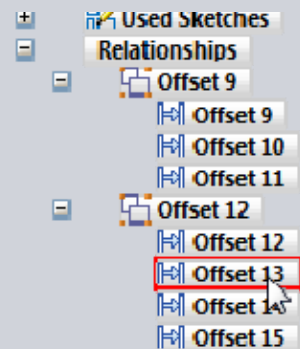
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Offset Relationship



SIEMENS

- Each persisted Offset relationship gets added to the Relationships collector in PathFinder
 - If multiple faces were offset in the same sequence of the command, they are located under a master offset collector
 - Hovering over an Offset relationship in PathFinder highlights both the seed and target paired faces
 - Selecting the offset relationship **from PathFinder** only adds the seed faces to the select set
 - Changing the offset value only moves seed faces



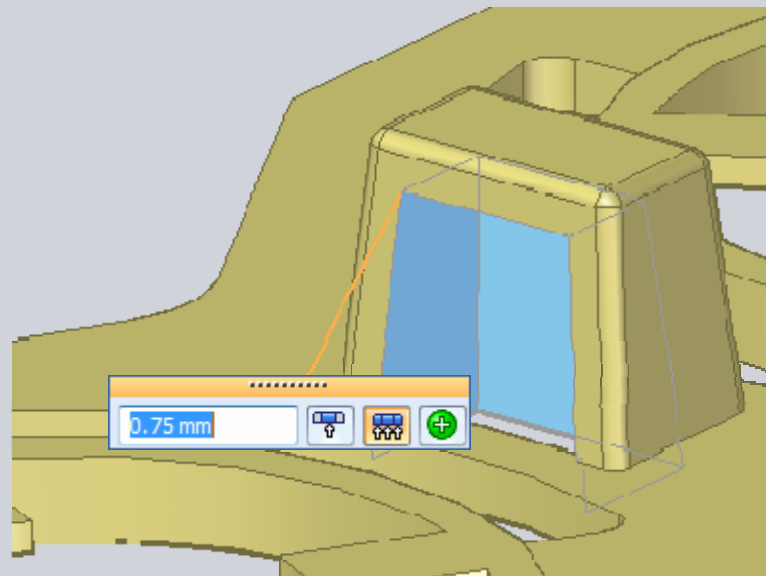
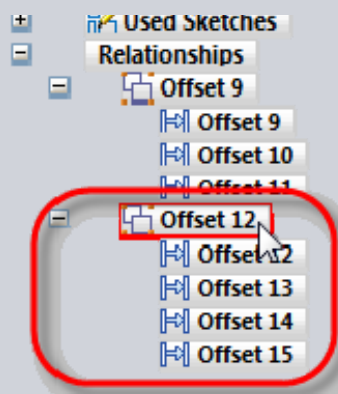
Offset Relationship



SIEMENS

- After selecting a face with a persisted offset, there are 3 options available when a user selects the value edit handle
 - Default All Feature Faces option (middle) selects all offsets that are of the same value and that were created in the same sequence of the command
 - Moves only the seed faces

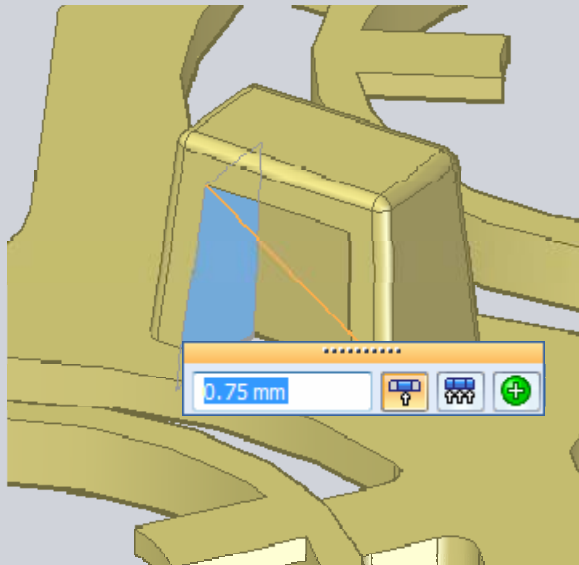
Continued on next slide



Offset Relationship



- After selecting a face with a persisted offset, there are 3 options available when a user selects the value edit handle (**cont'd**)
 - Selected Faces Only option (first) is to modify only selected face(s)
 - Moves seed or target faces, whichever is selected
 - Selection Manager option (third) locates ALL persisted offset faces in the entire model that have the same offset value
 - Only seed faces move

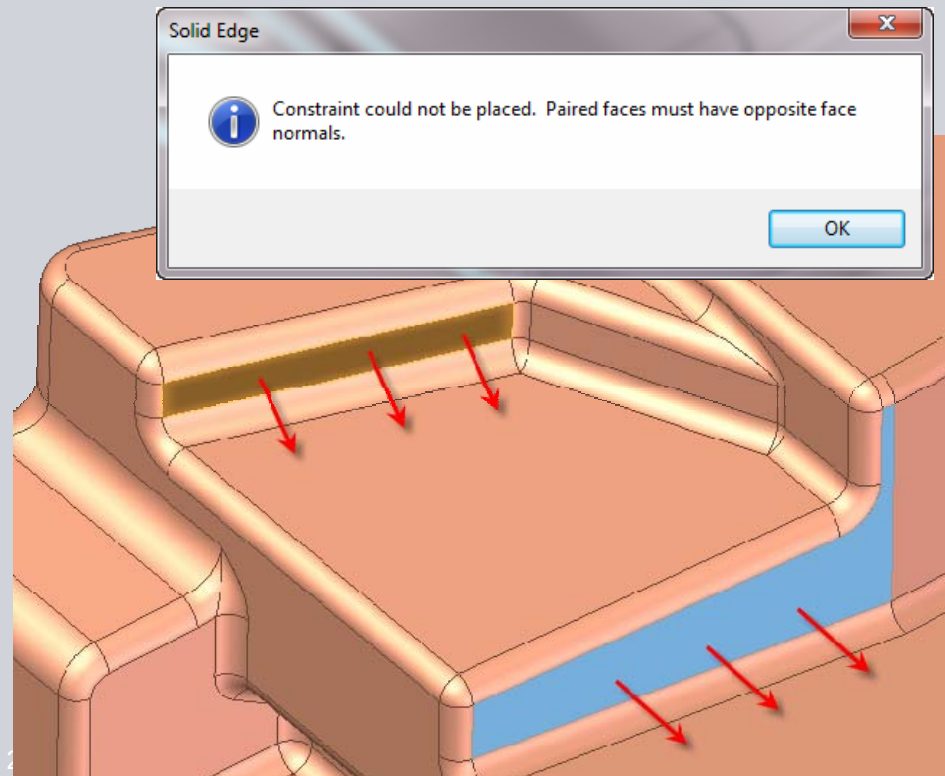
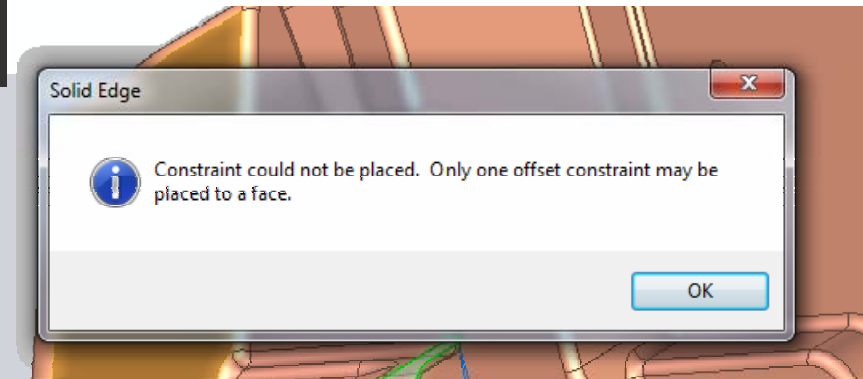
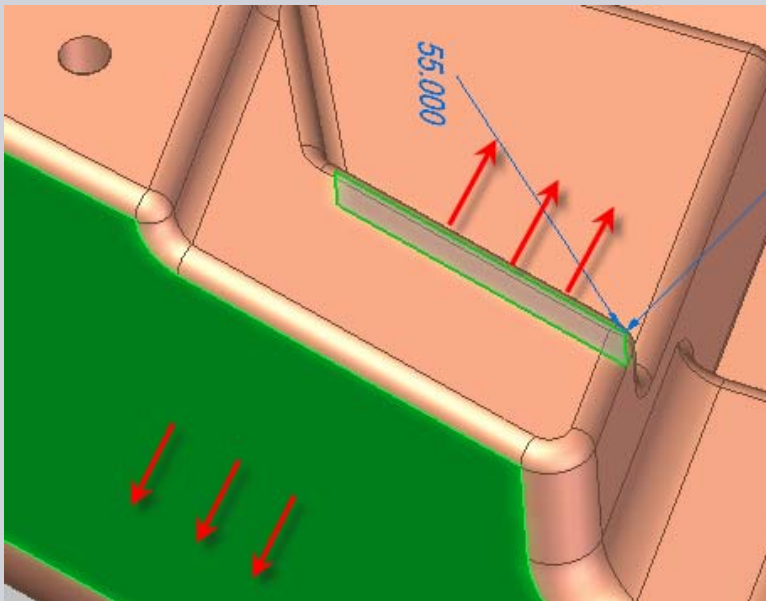


Offset Relationship



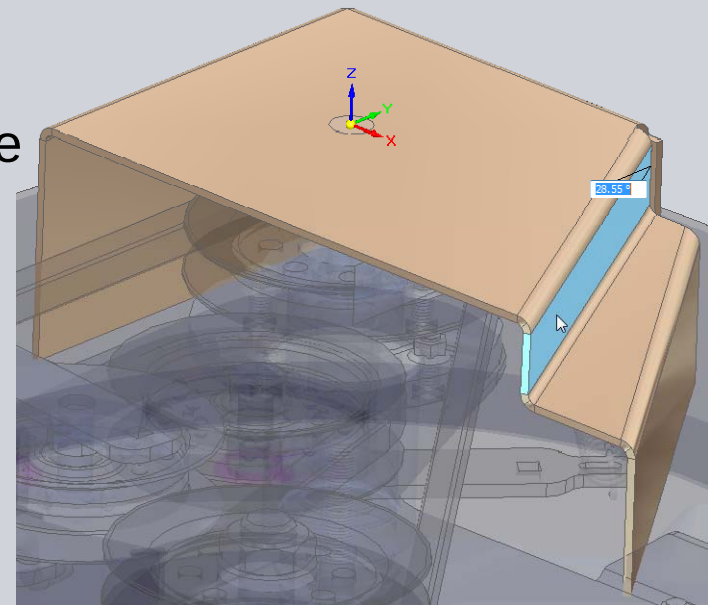
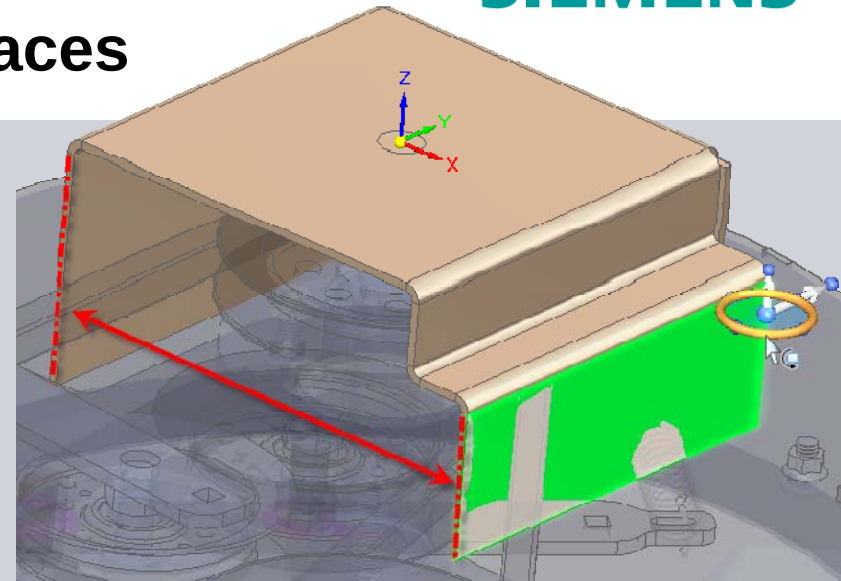
SIEMENS

- **Only one persisted offset relationship** can be placed on a single face
- Seed and Target faces must have opposite face normals



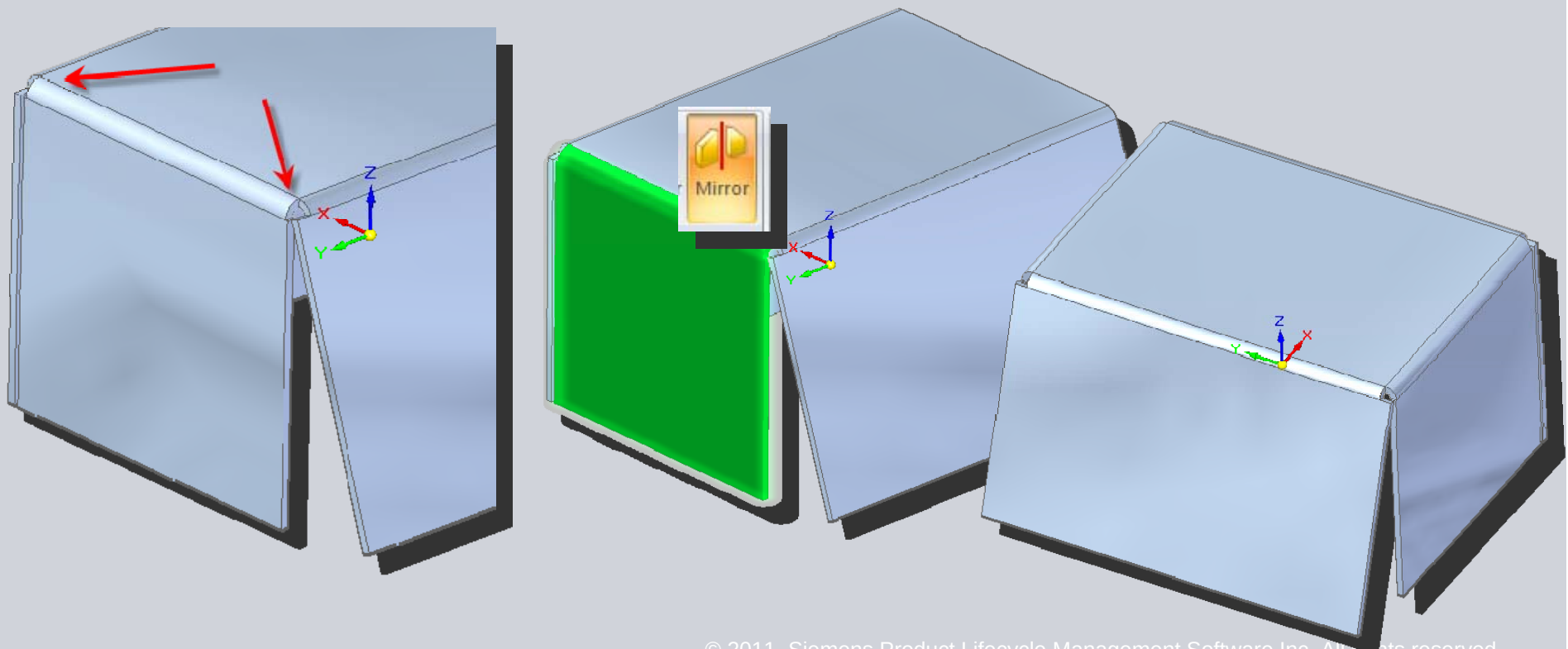
Coplanar Ignores Thickness Faces

- In ST3, the rotate operation of a plate may fail because the Coplanar live rule finds a thickness face from another plate
 - Thickness faces cannot be rotated
- ST4 enhancement is to ensure that the thickness faces, which are indirectly located, do not violate the Coplanar live rule and allow the operation to continue without causing a failure
 - No longer necessary to suspend the Coplanar live rule in the Iron Eagle sheet metal demo when rotating the side faces



Improved Flange Mirroring

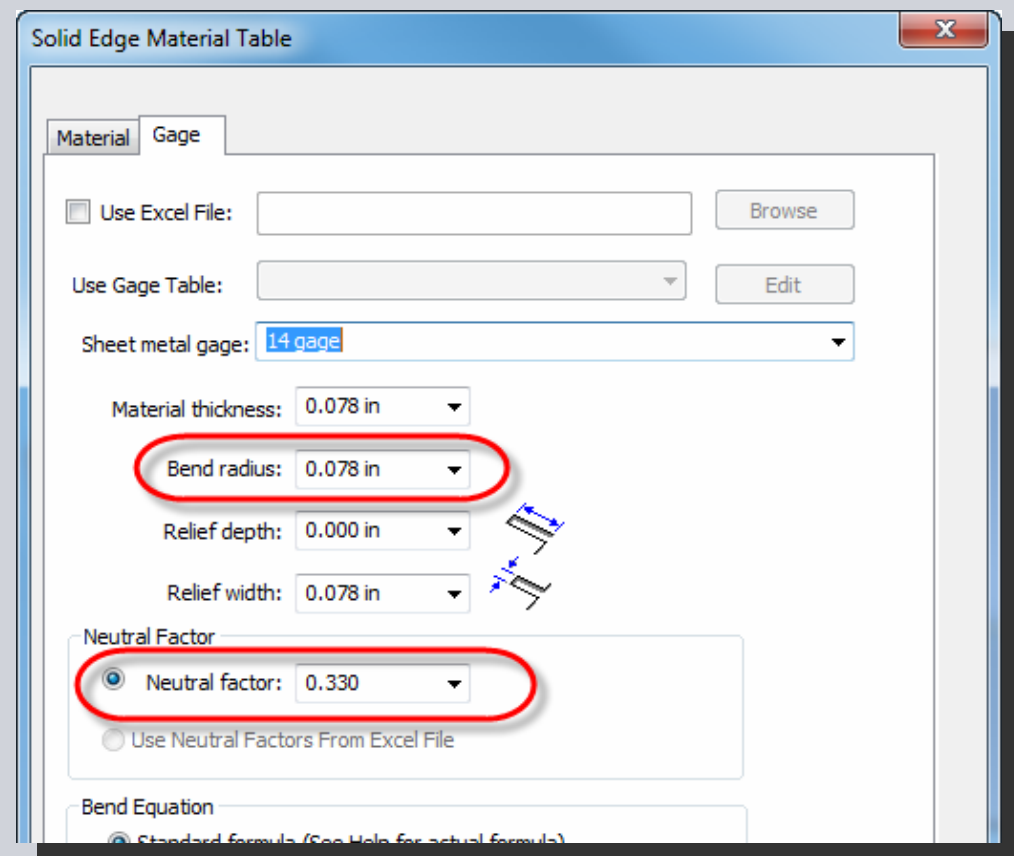
- ST4 improves the success of mirroring flanges when the resulting flange interacts with another bend at a corner
 - This failed to attach in many cases prior to ST4



Converting to Synchronous Sheet Metal, Use Global Bend Radius

SIEMENS

- ST4 provides more accurate global control for those converted bends which match the default bend property values when converting to synchronous sheet metal
(Move to Synchronous and Convert/Transform to Sheet Metal)
 - Bend properties considered are:
 - Bend Radius
 - Neutral Factor

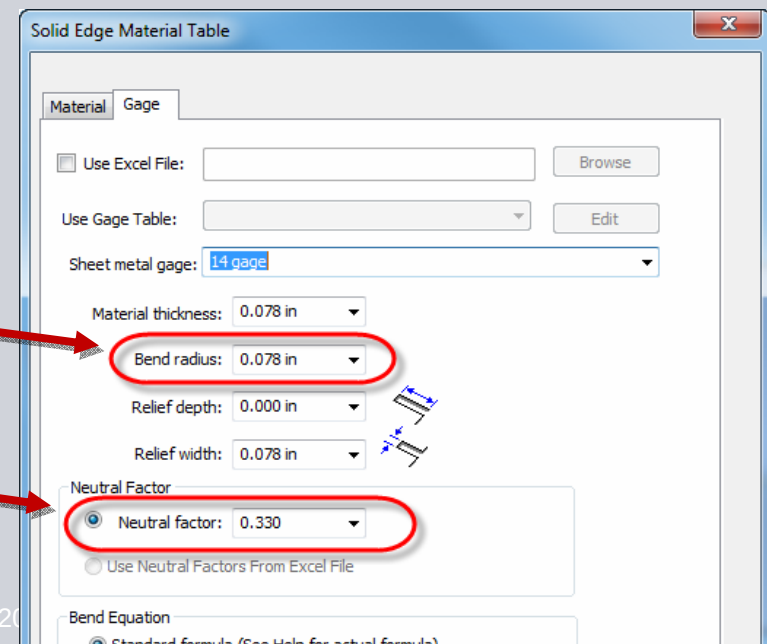
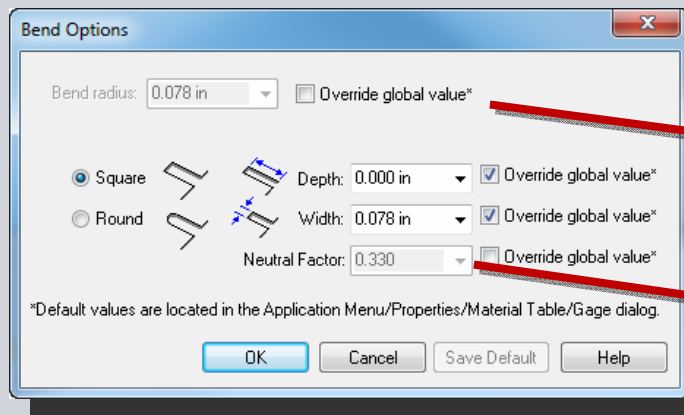


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Convert to Synchronous Sheet Metal, Use Global Bend Radius

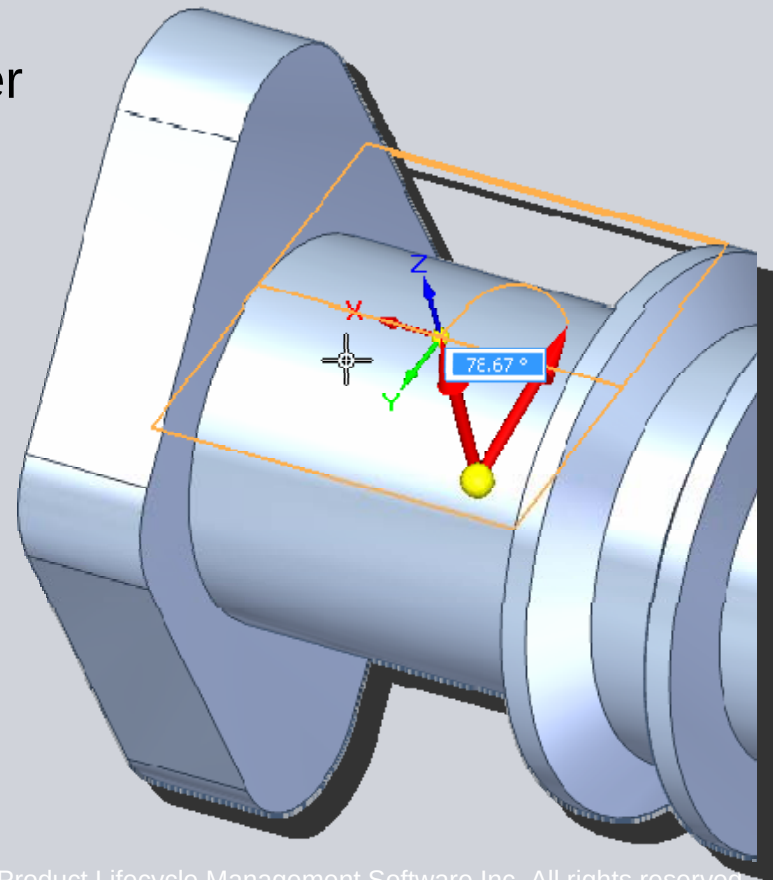
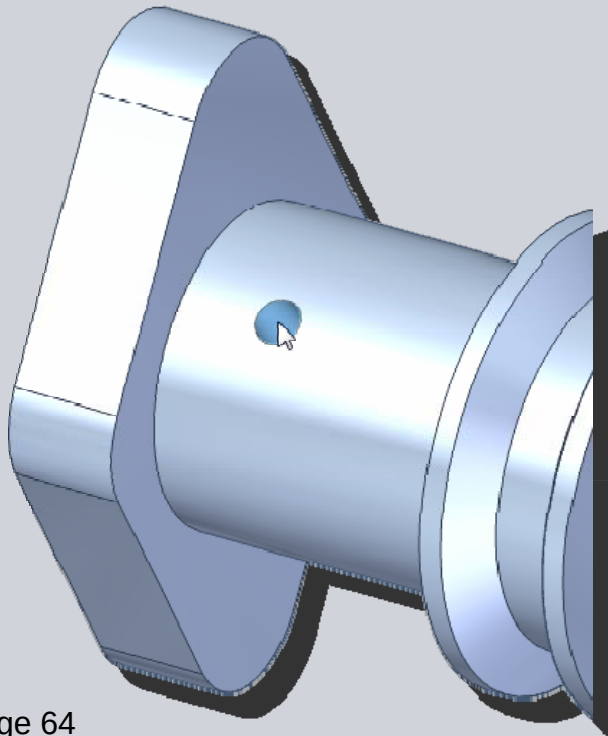
SIEMENS

- If property values of a bend match the current default values of the material gage properties, then the bend is considered as a globally defined bend
 - Can be modified by changing the values in the material table
- If value of the bend properties does not match the default values, then the bend is considered a custom bend value
 - Must be modified individually



Hole Feature Improved Placement on Cylinder

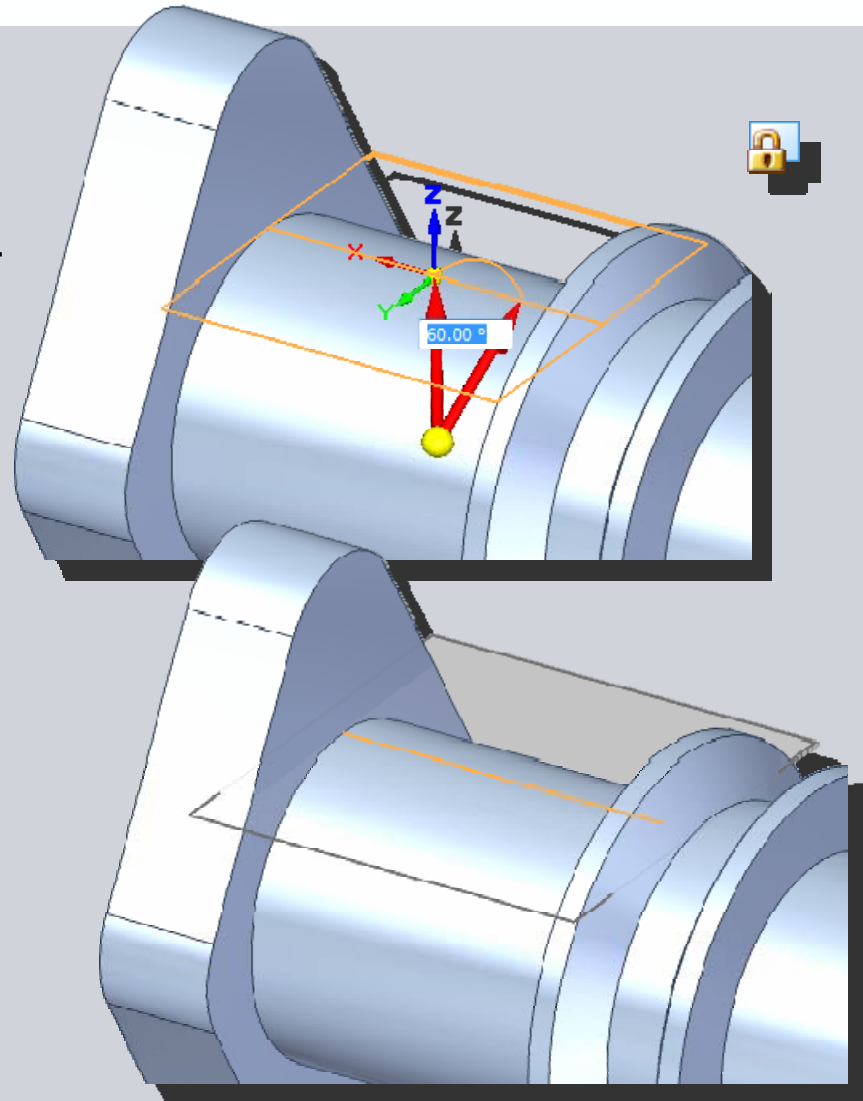
- The synchronous Hole command has been enhanced to allow the precision placement of a tangent plane within the command
 - User can key-in a precise angle
 - The tangent plane is not persisted after the command is completed



Hole Feature Improved Placement on Cylinder

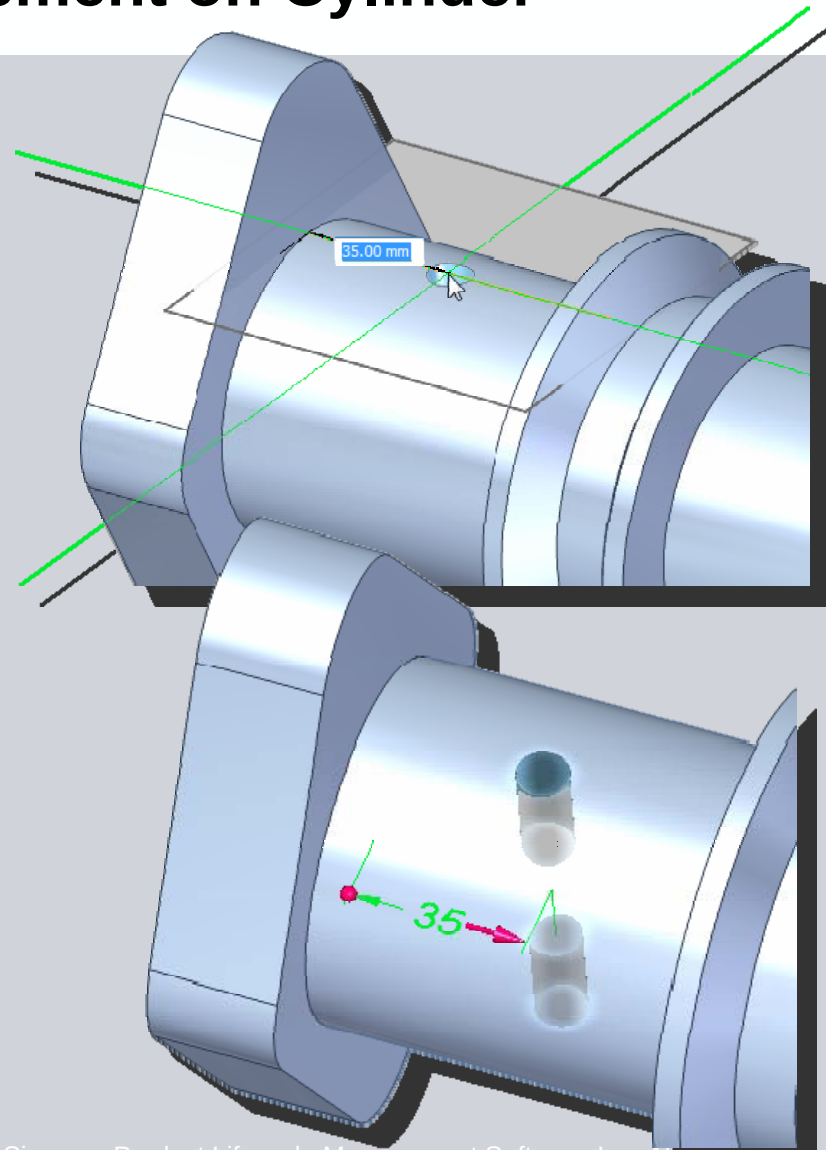
- The command workflow:
 - User starts the hole command and drags the hole over a cylinder
 - While the hole is previewed on the cylinder the user presses F3 to lock to a plane
 - Displays the tangent plane
 - User clicks in space, picks a keypoint, or sets the angle or keys in a precise value

- A tangent line is displayed on the part once the angle is set



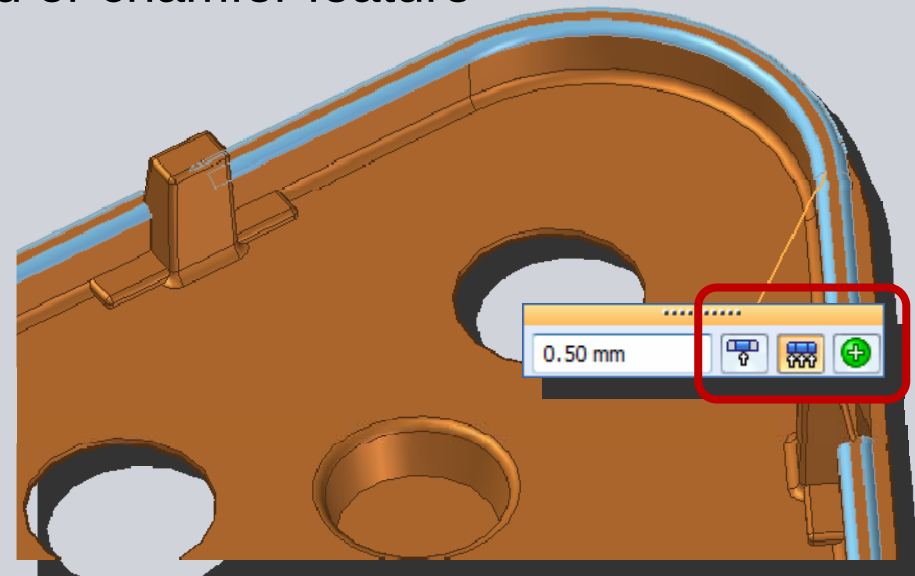
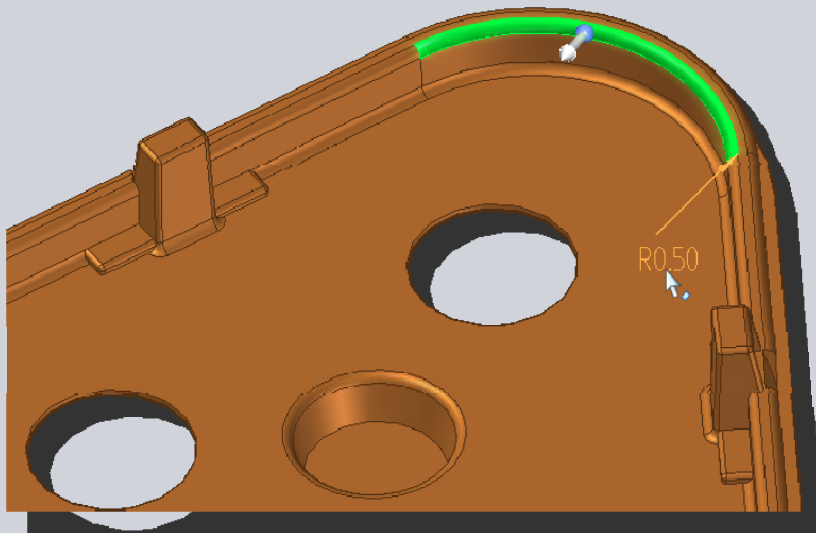
Hole Feature Improved Placement on Cylinder

- If the user drags the hole close to the tangent line, the hole snaps to the line
- Hot keys can be used for precise placement on the plane
 - Pressing "e" on the keyboard creates a driving dimension to the **cylinder axis** endpoint
 - Pressing "c" on the keyboard creates a driving dimension to the center of a circular edge



Selection - Feature Selection

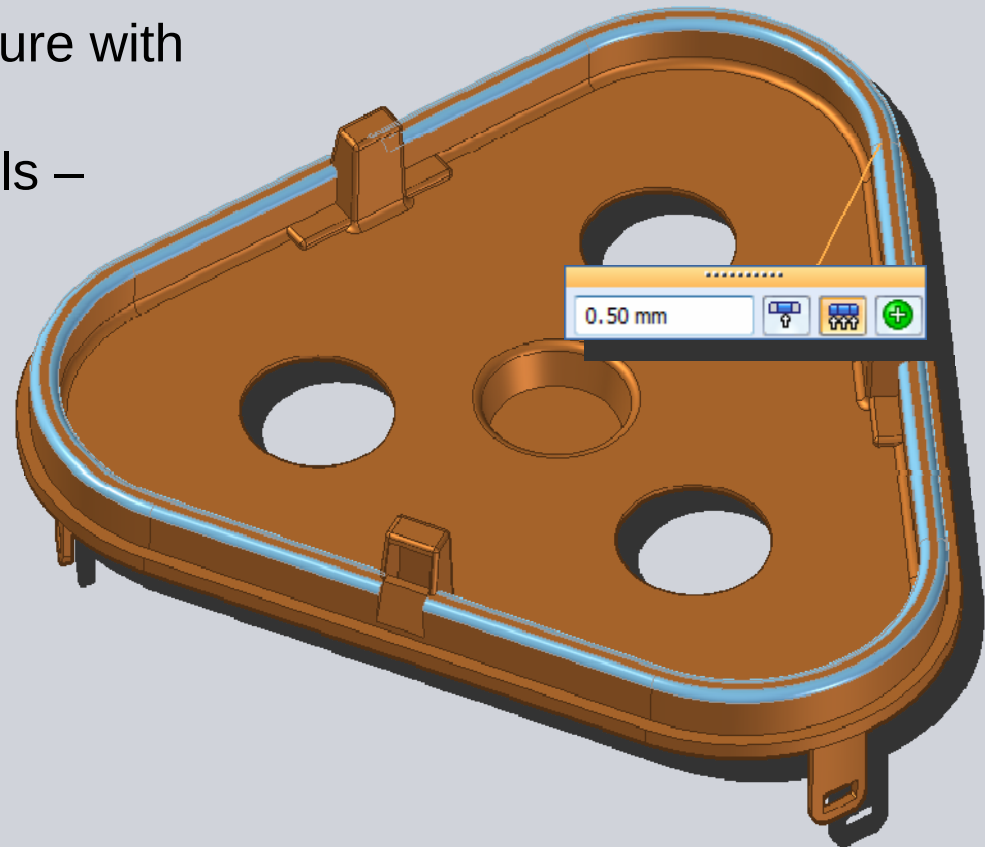
- Selection propagation controls have been added to the Quick Bar for editing synchronous rounds and chamfers
 - Eliminates the need to constantly use selection manager to find all elements added in a single feature, or all elements with the same input value
- Appears only during Edit of a round or chamfer feature



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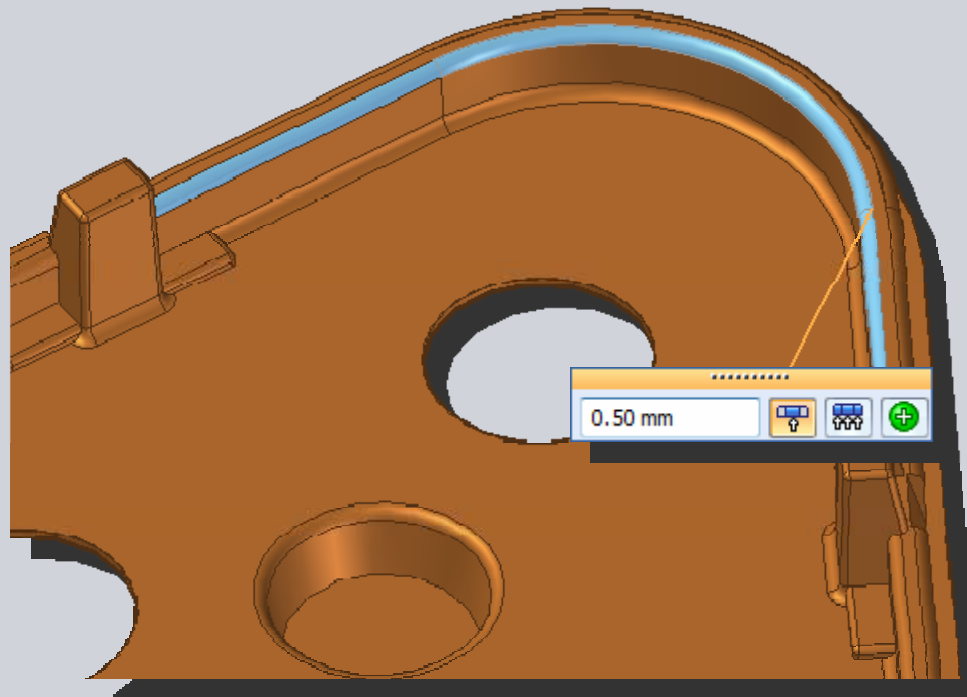
Selection - Feature Selection

- Default setting is "All Feature Faces"
 - All rounds/chamfers that were added to the model as a single feature with the same value are selected
 - Irrelevant for imported models – no feature sets



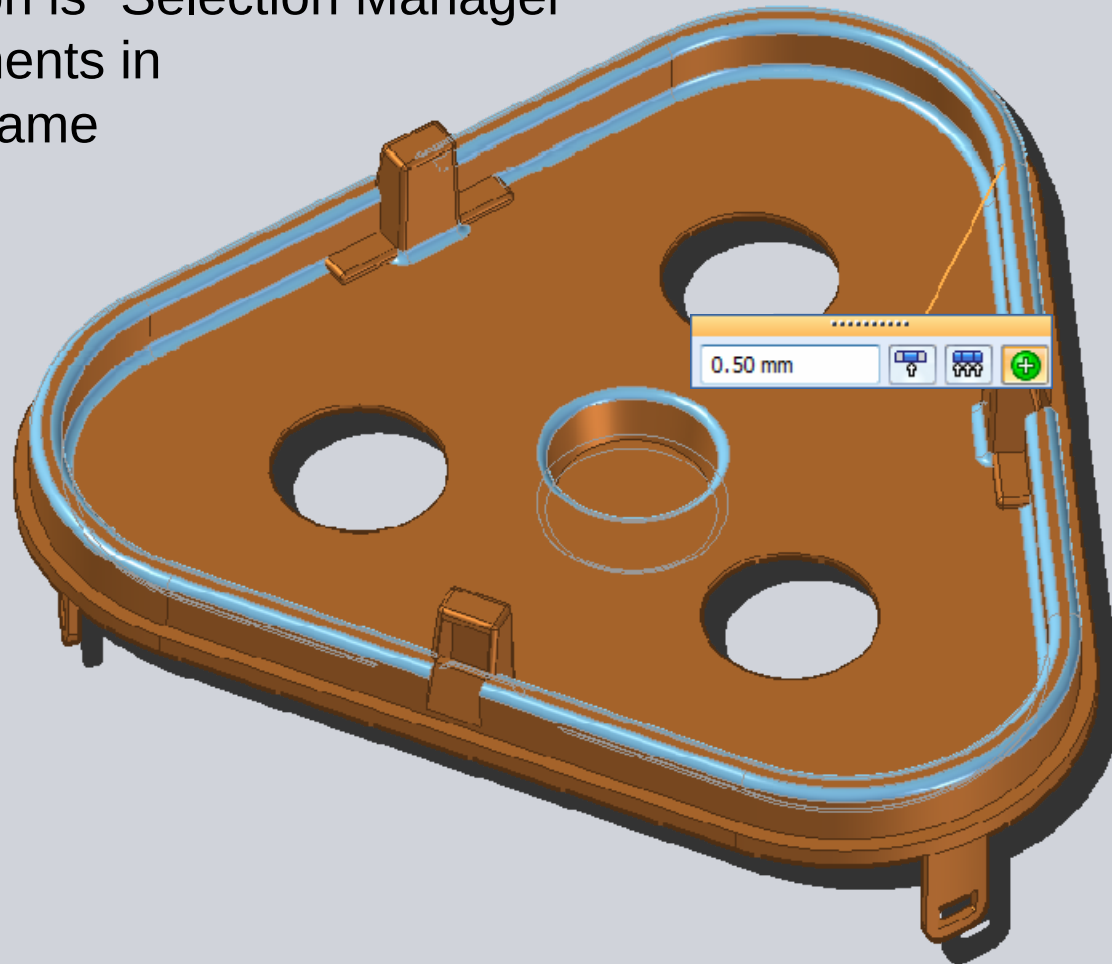
Selection - Feature Selection

- Selecting the first option - "Selected Faces Only" option results in a chain selection of rounds/chamfers as in ST3



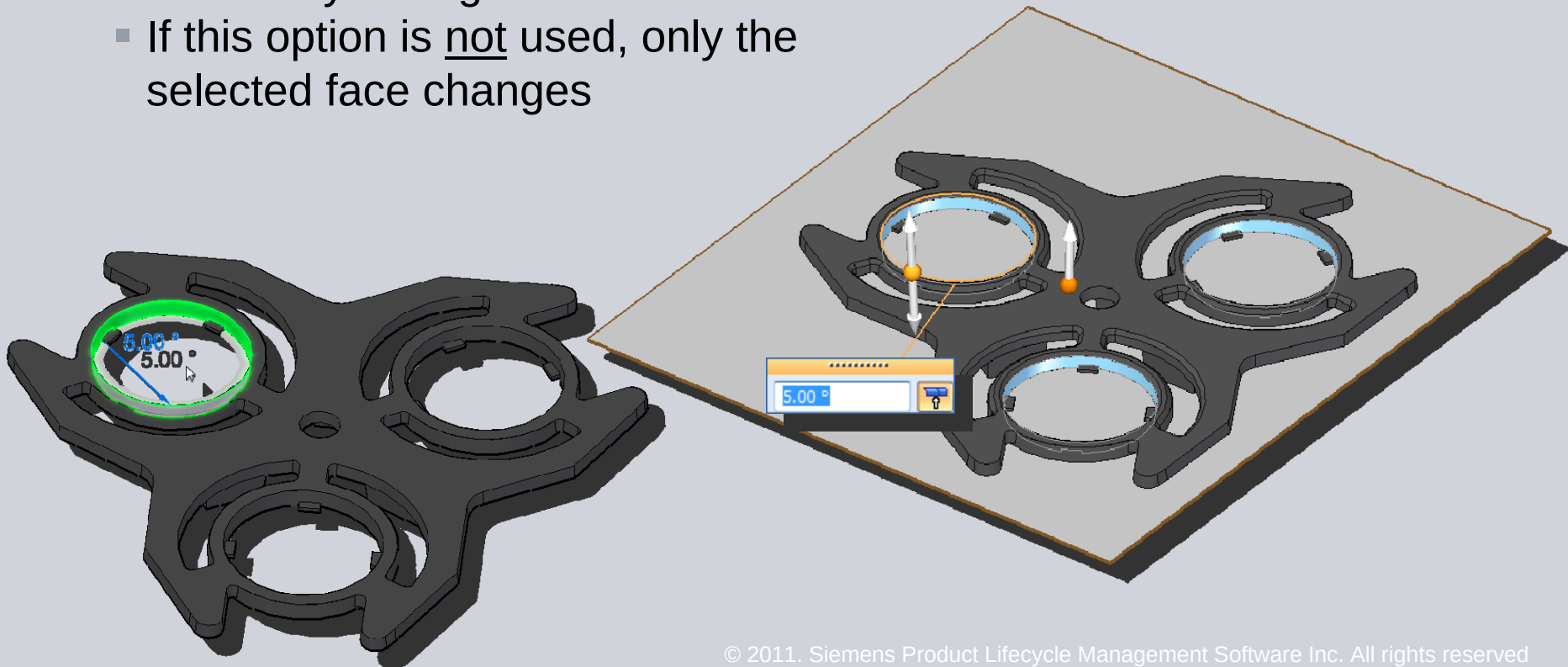
Selection - Feature Selection Rounds

- The second new option is "Selection Manager"
 - Used to find all elements in the model with the same input value

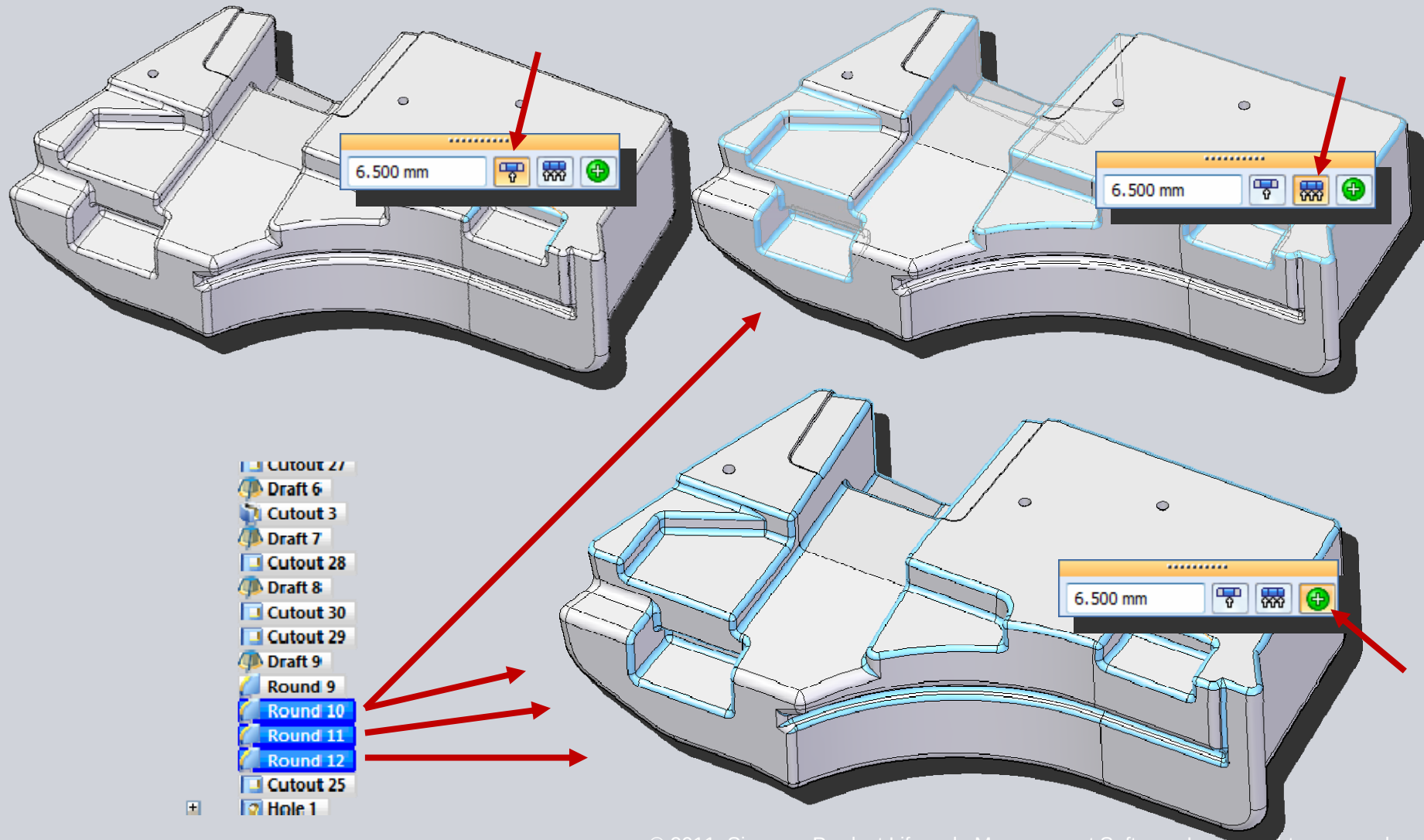


Selection - Feature Selection

- Feature Selection control also added for drafted faces and thin wall faces
 - "All Feature Faces" option only (default)
 - Used only during edit
 - If this option is not used, only the selected face changes

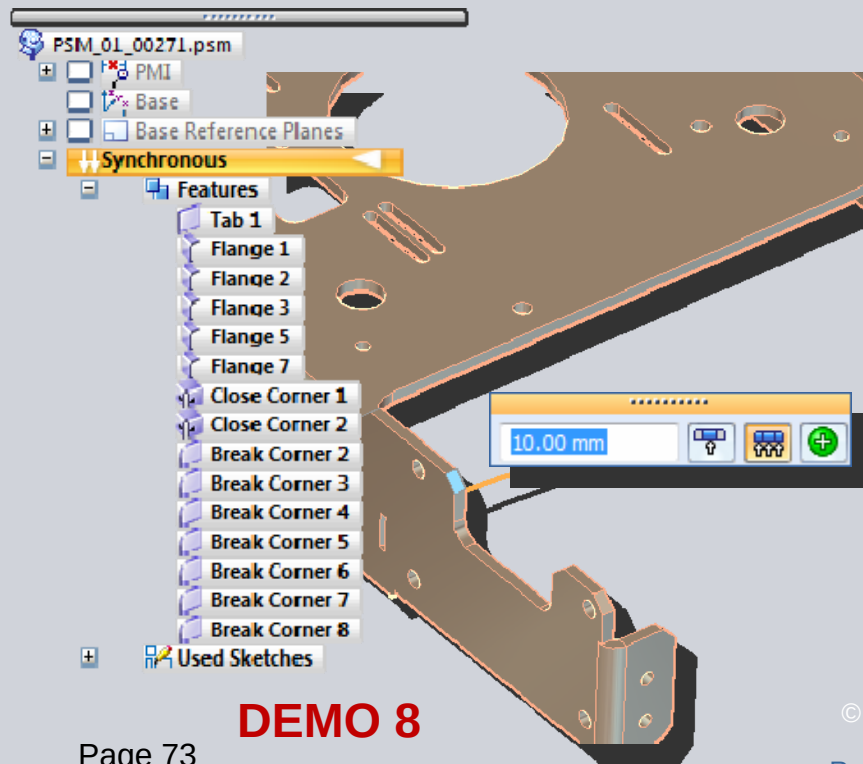


Selection - Feature Selection Rounds



Selection - Feature Selection

- New Feature Selection controls added for sheet metal break corners
 - Radius corner
 - Chamfer corner



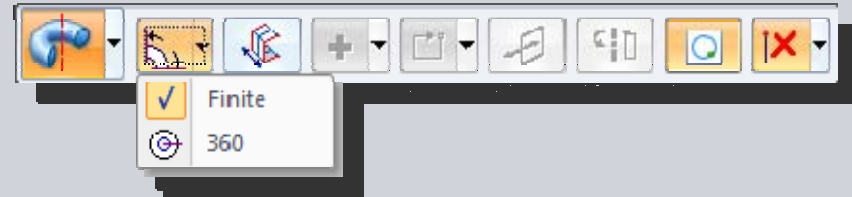
DEMO 8

Revolved Feature

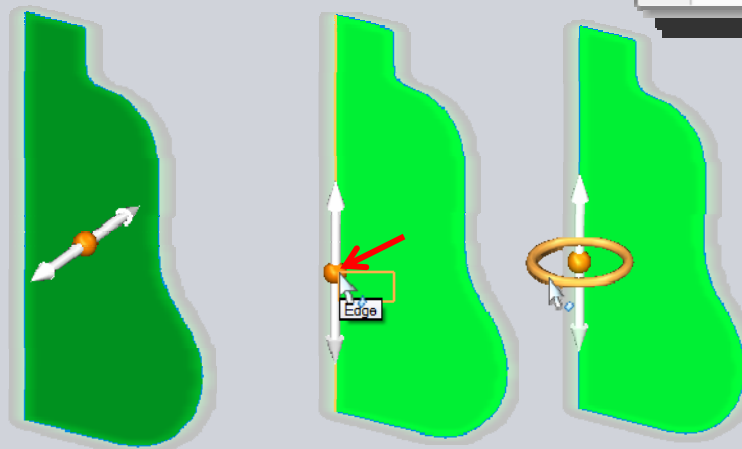
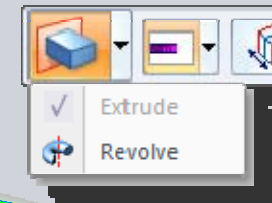
Improve Creation Workflow

SIEMENS

- Extent type is no longer set to the "most recently used"
 - New default extent type is Finite



- The user may change from a Linear extrude to revolve extrude at any time by:
 - Changing the option in the QuickBar drop list
 - Changing the placement location of the handle to a linear edge



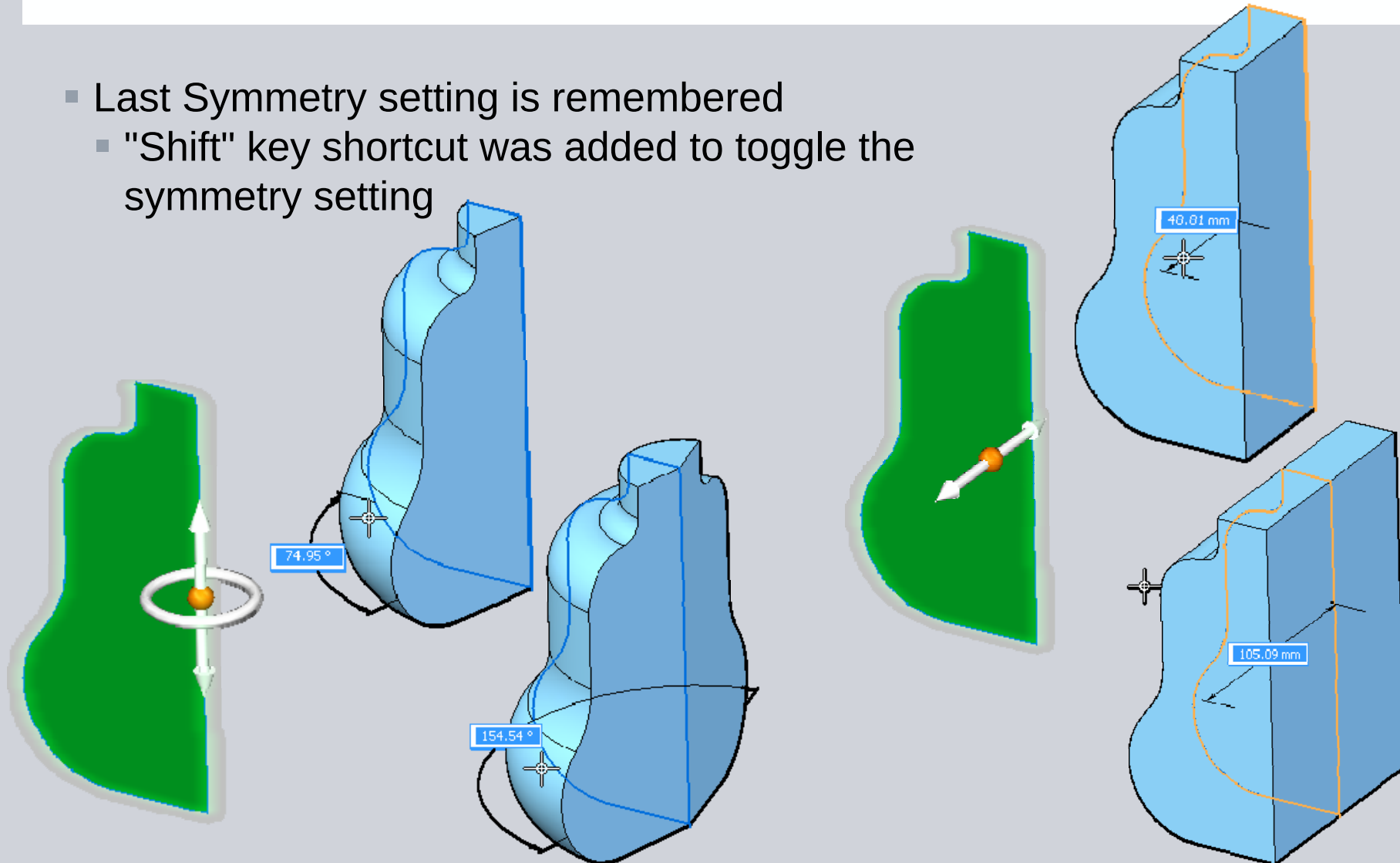
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Revolved Feature

Improve Creation Workflow

SIEMENS

- Last Symmetry setting is remembered
 - "Shift" key shortcut was added to toggle the symmetry setting

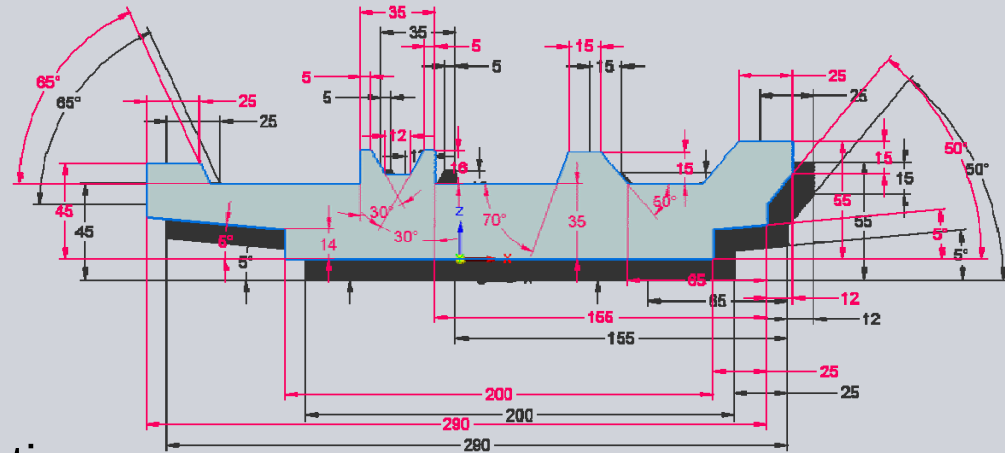


Revolved Feature

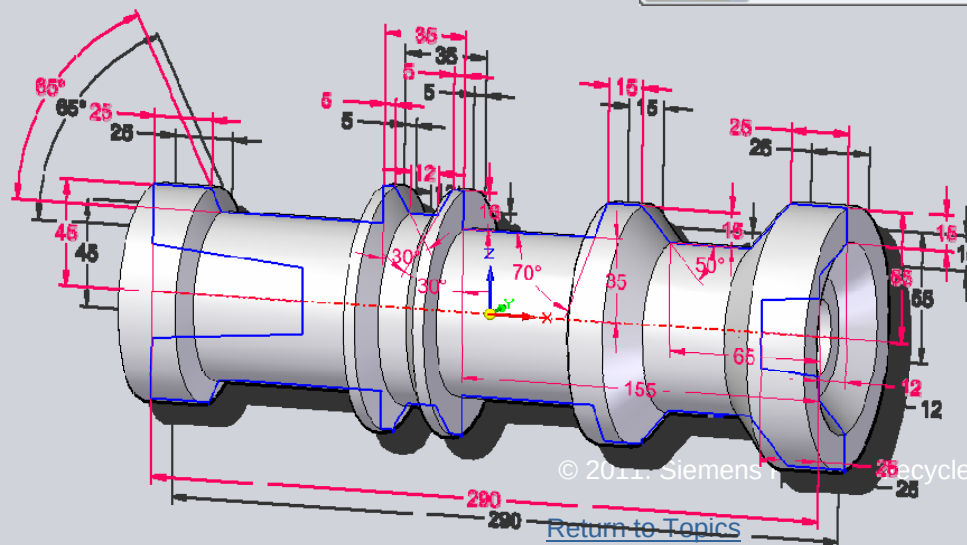
Auto-create Live Section

SIEMENS

- On creation of a revolved feature, a Live Section is automatically generated in ST4 by default – Quick Bar option
 - Profile dimensions migrate automatically to the Live Section geometry



Live Section Option

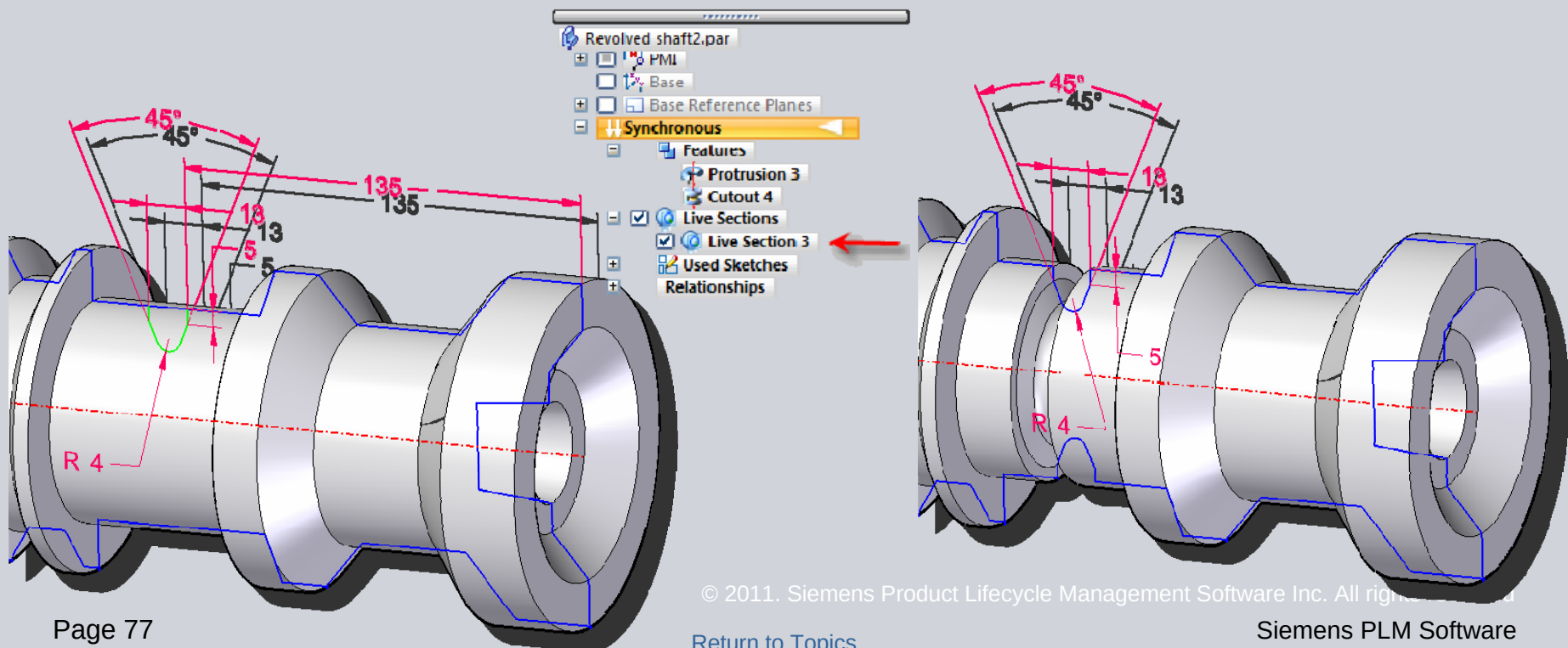


Revolved Feature

Auto-create Live Section

SIEMENS

- If the feature profile plane is not shared with any live section, then a *new* Live Section is generated
 - If the revolved protrusion profile (shown in green) is created on an existing Live Section plane (shown in blue), the sketch and dimensions are merged with the existing Live Section

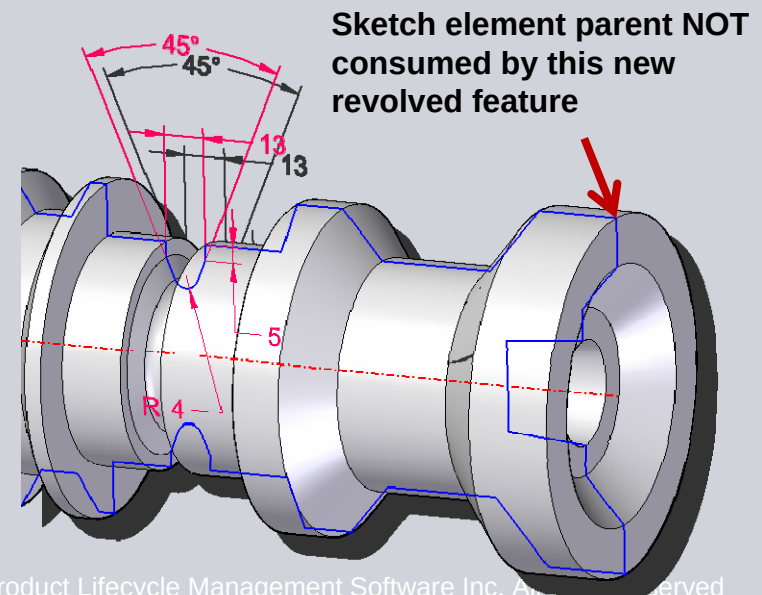
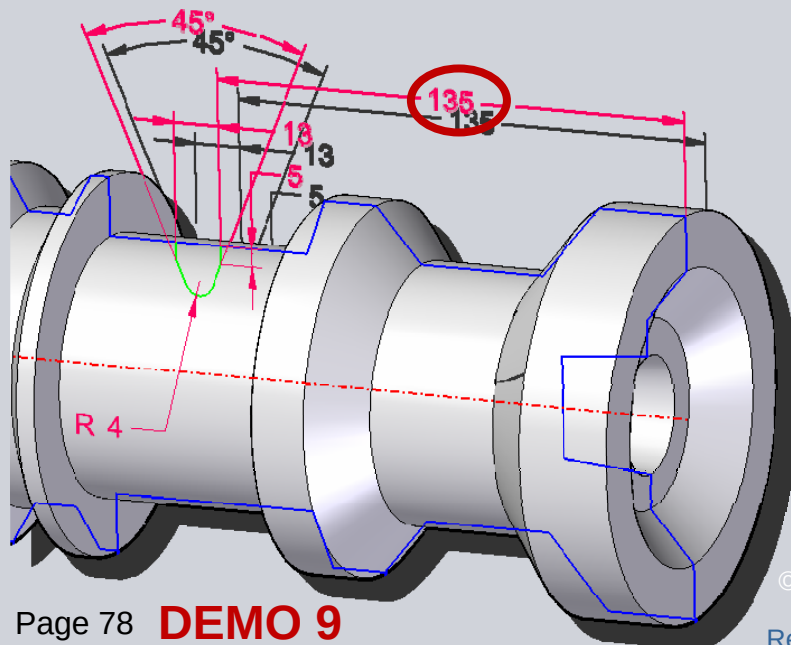


Revolved Feature

Auto-create Live Section

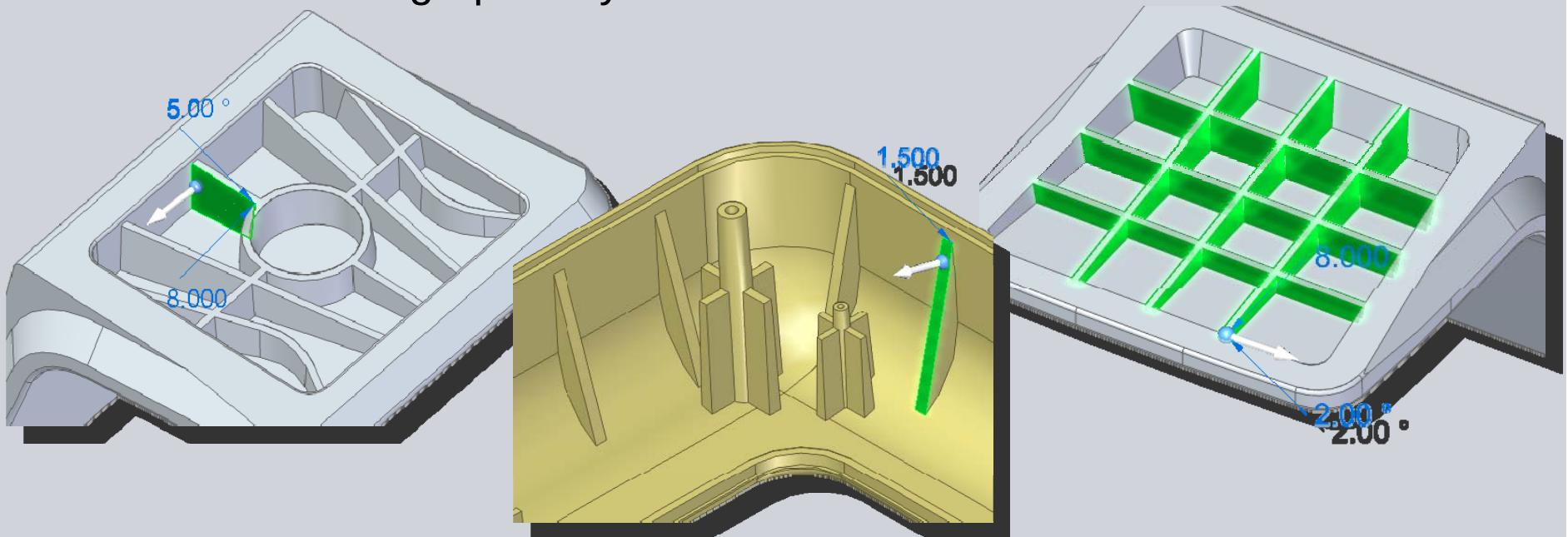
SIEMENS

- Note in this example the 135 mm dimension was removed due to the following rules:
 - If both sketch element parents to the dimension are included in the feature, then the dimension is migrated to the live section
 - If either of the sketch element parents is not consumed by the new feature, then the dimension does NOT migrate
 - The axis of revolution is considered a parent of a dimension



Rib/Web Network

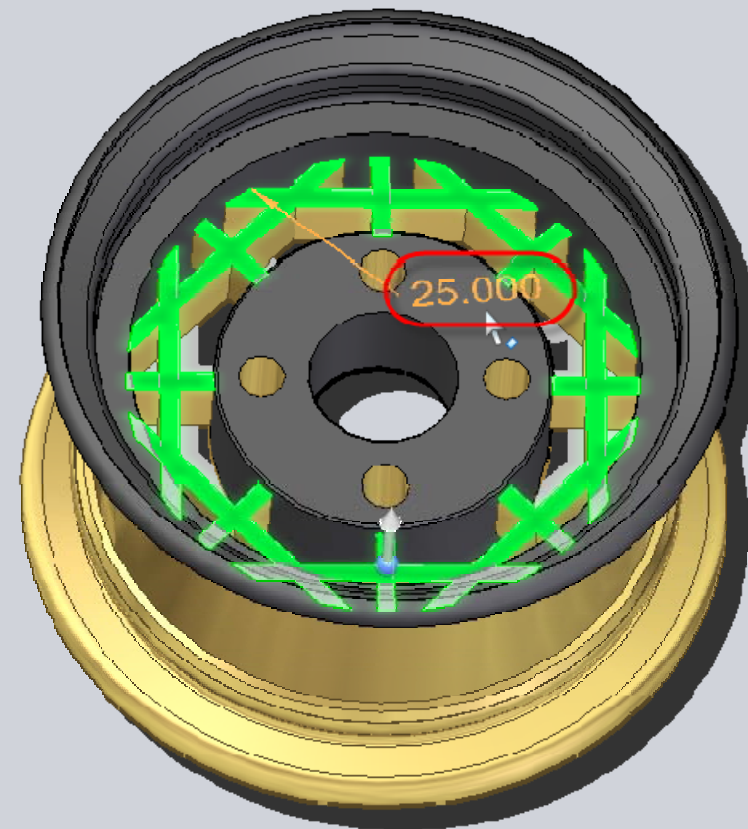
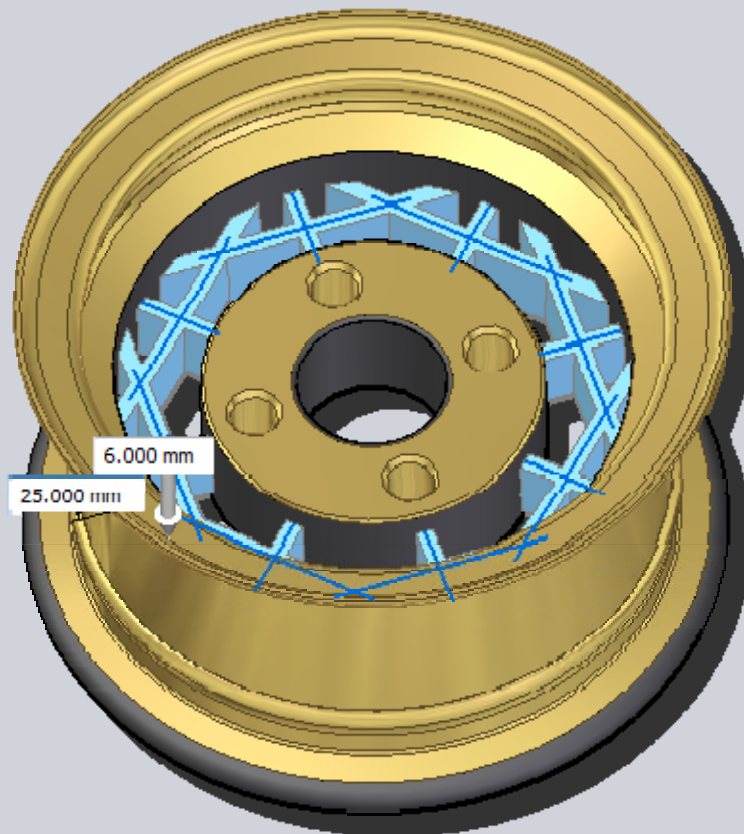
- Ribs and web networks can be created in the synchronous mode and have been enhanced to behave synchronously
 - Must be created in ST4 to see new behavior
- Edit handles are presented for thickness (and draft angle) when a rib or web is selected graphically or from PathFinder



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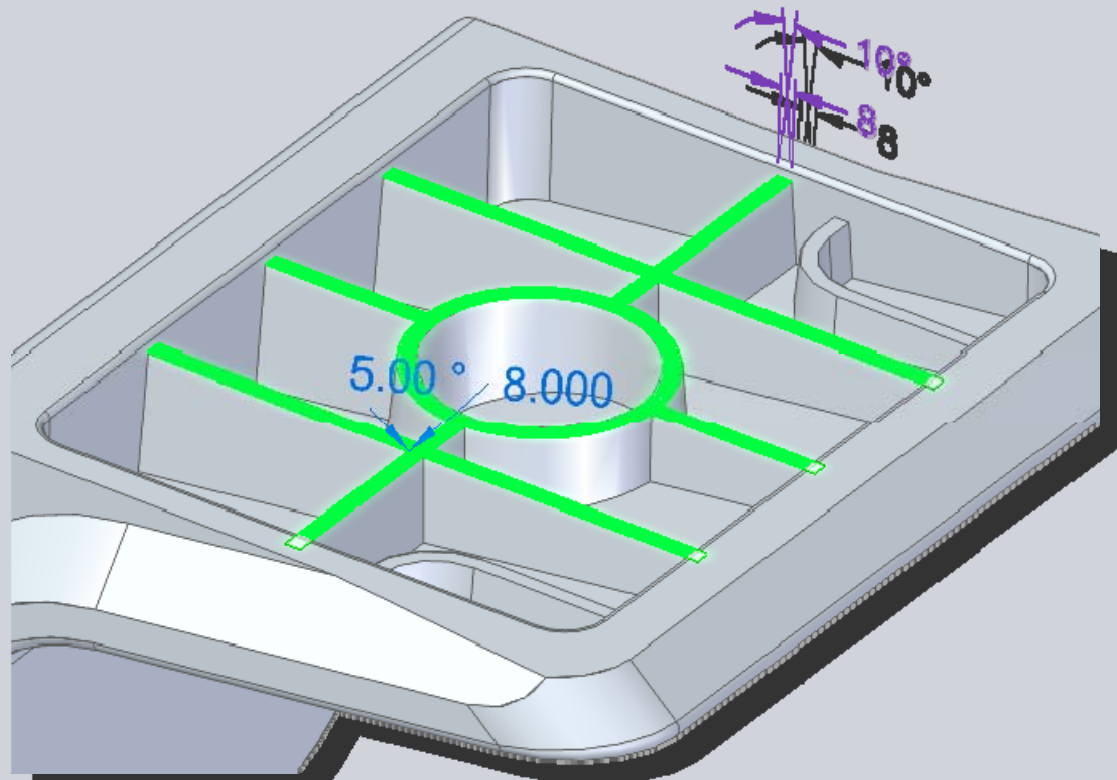
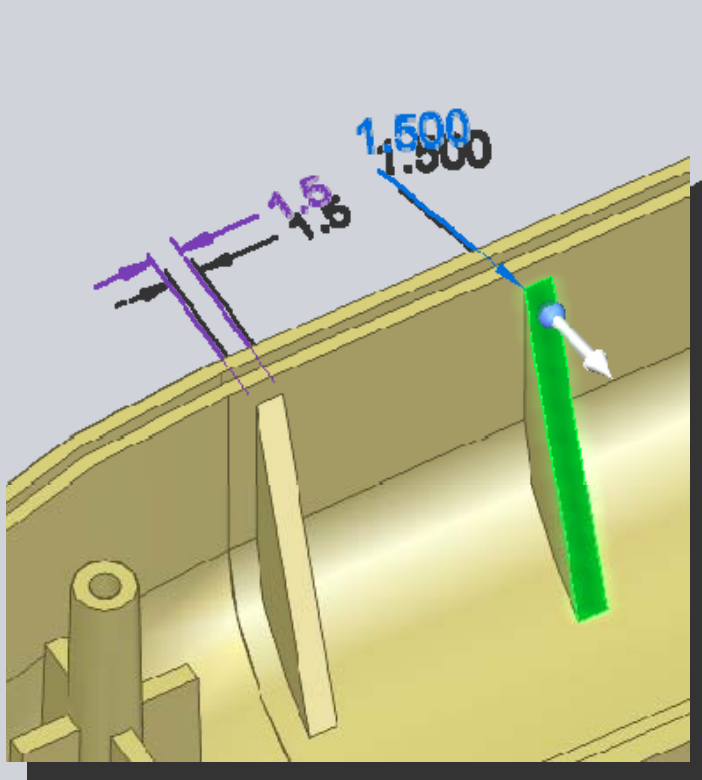
Rib/Web Network

- If a web or rib has a finite depth, the edit handle for the depth is only displayed if the bottom face is selected



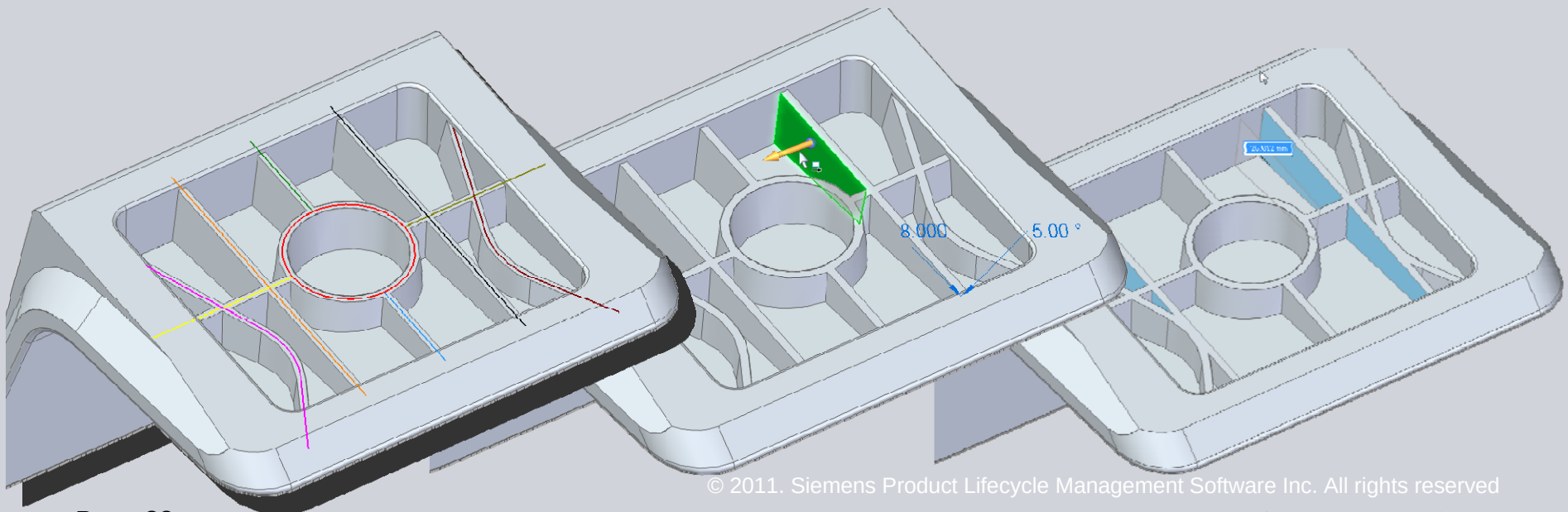
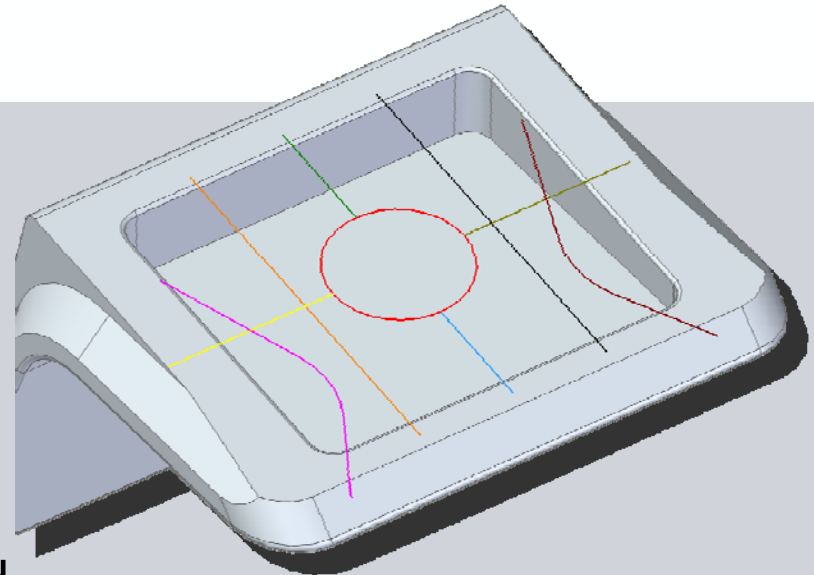
Rib/Web Network

- All edits to thickness or draft angle on web faces are only allowed through “Feature Edit”
 - Changing web parameters through PMI is disabled



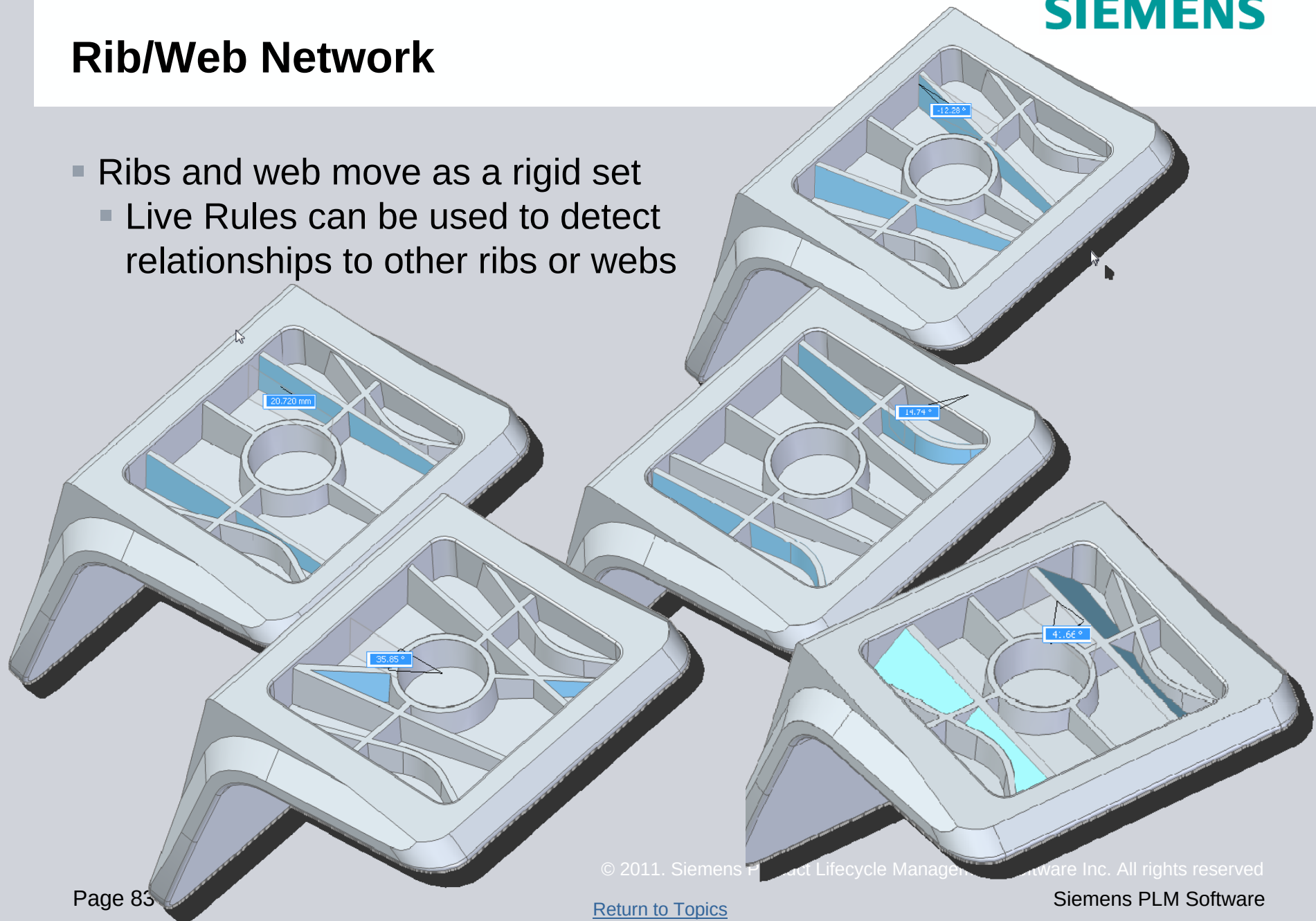
Rib/Web Network

- Individual ribs and webs that are tangentially connected can be moved via the steering wheel as a rigid set of geometry
 - Move or rotate
 - Thickness is maintained
 - If draft was added it is also maintained



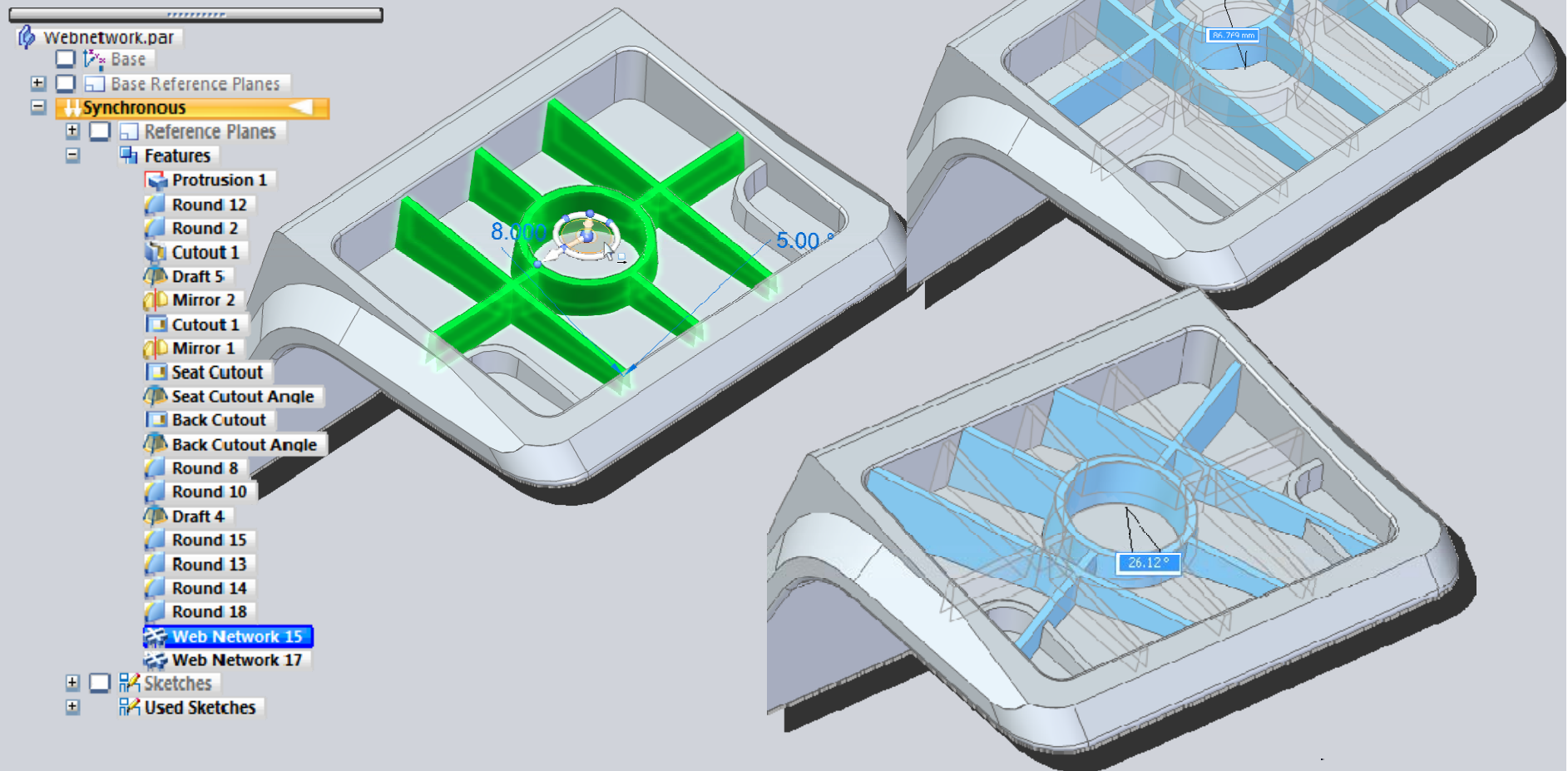
Rib/Web Network

- Ribs and web move as a rigid set
 - Live Rules can be used to detect relationships to other ribs or webs



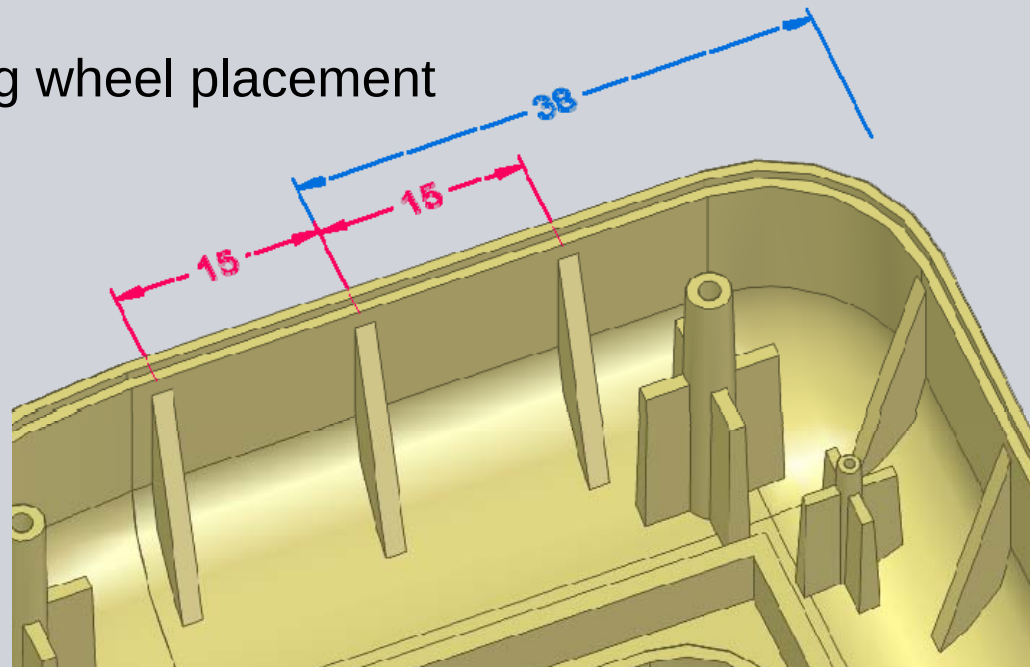
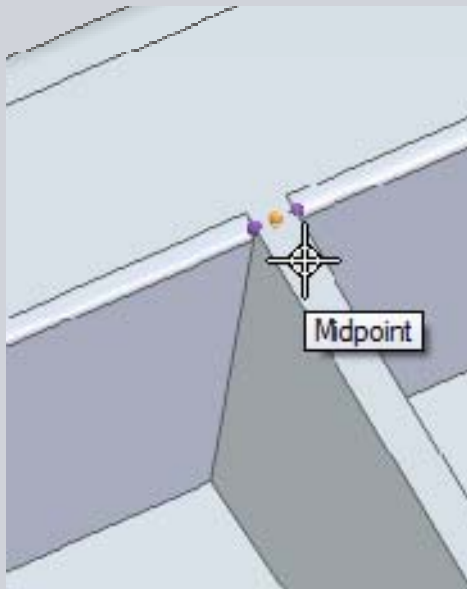
Rib/Web Network

- Entire web/rib feature may be moved or rotated



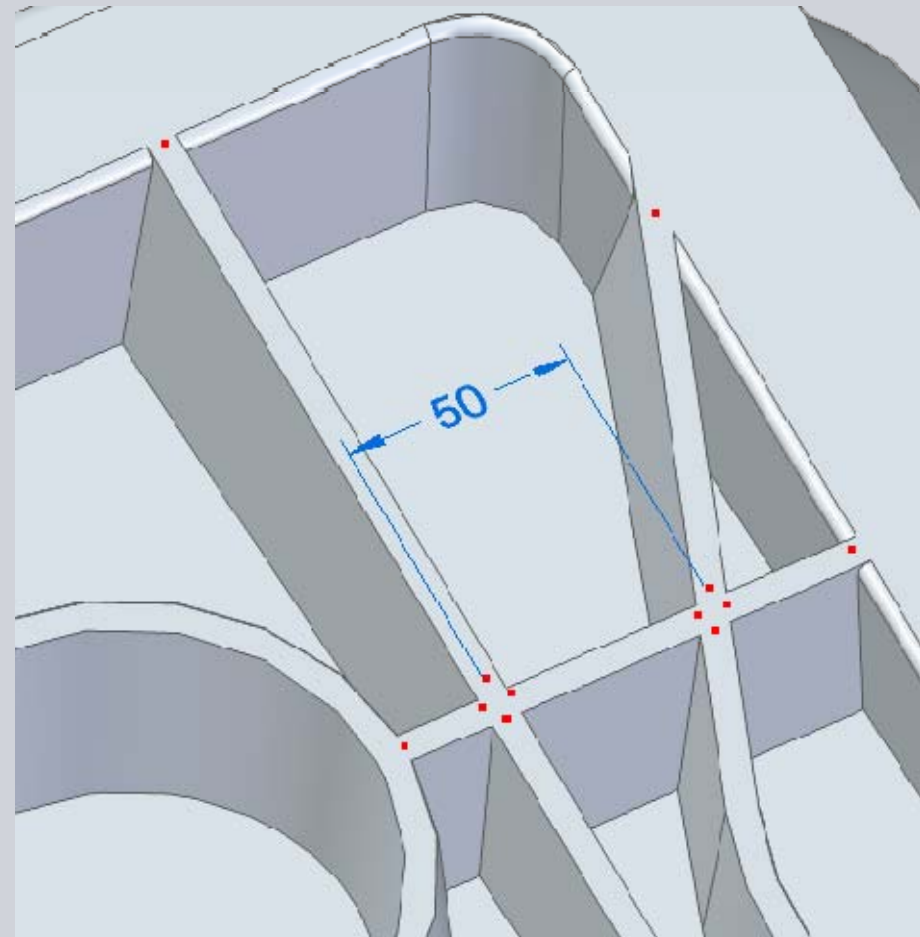
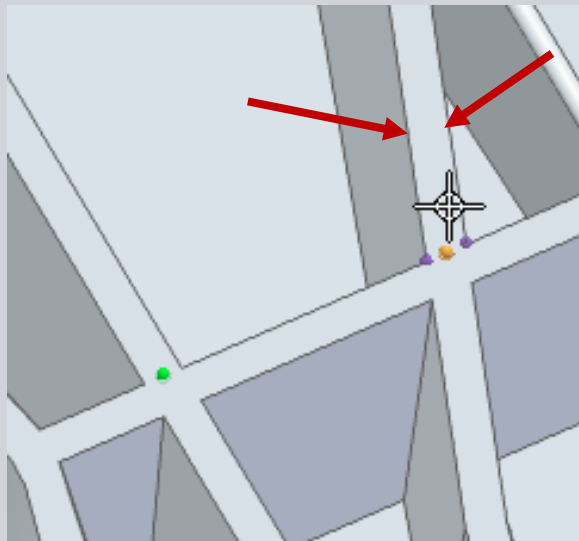
Rib/Web Network

- Ribs and webs have been enhanced to include "midpoints" and the ends (and intersections for webs) that can be used for dimensioning
 - For dimensioning to the "center line" of ribs and webs
 - Rib and web midpoints are only locatable in the context of a dimension command
 - Not locatable for steering wheel placement



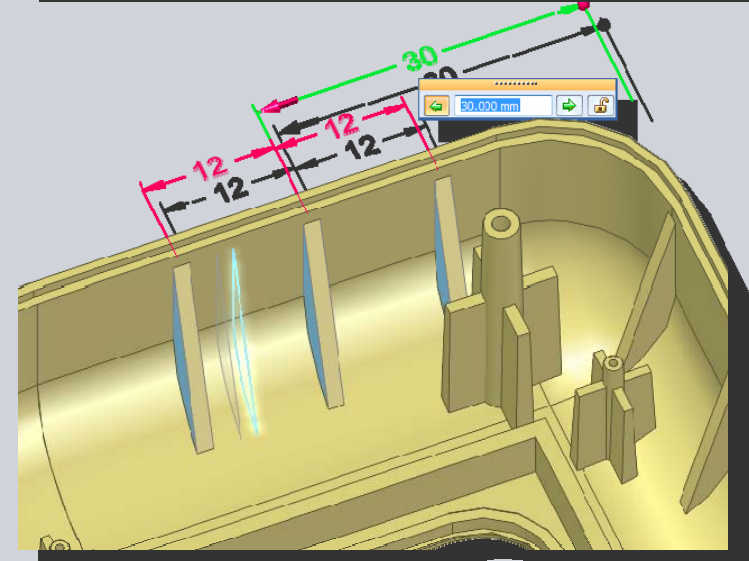
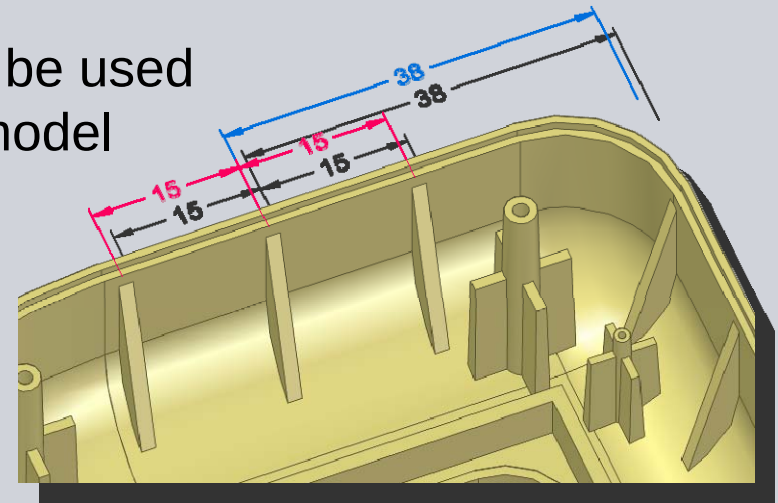
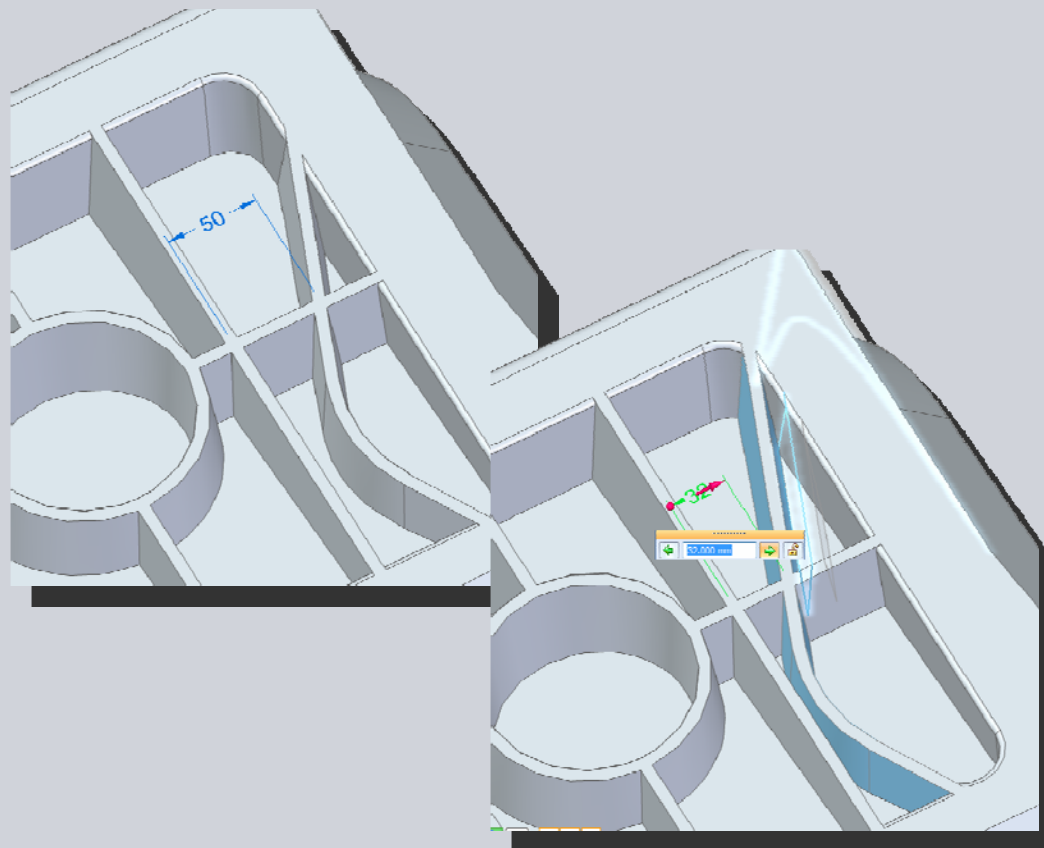
Rib/Web Network

- Each web intersection has 4 possible midpoints dependant upon which edges are highlighted during dimension placement
 - Endpoints of the edges that are related to the midpoint are also highlighted in violet



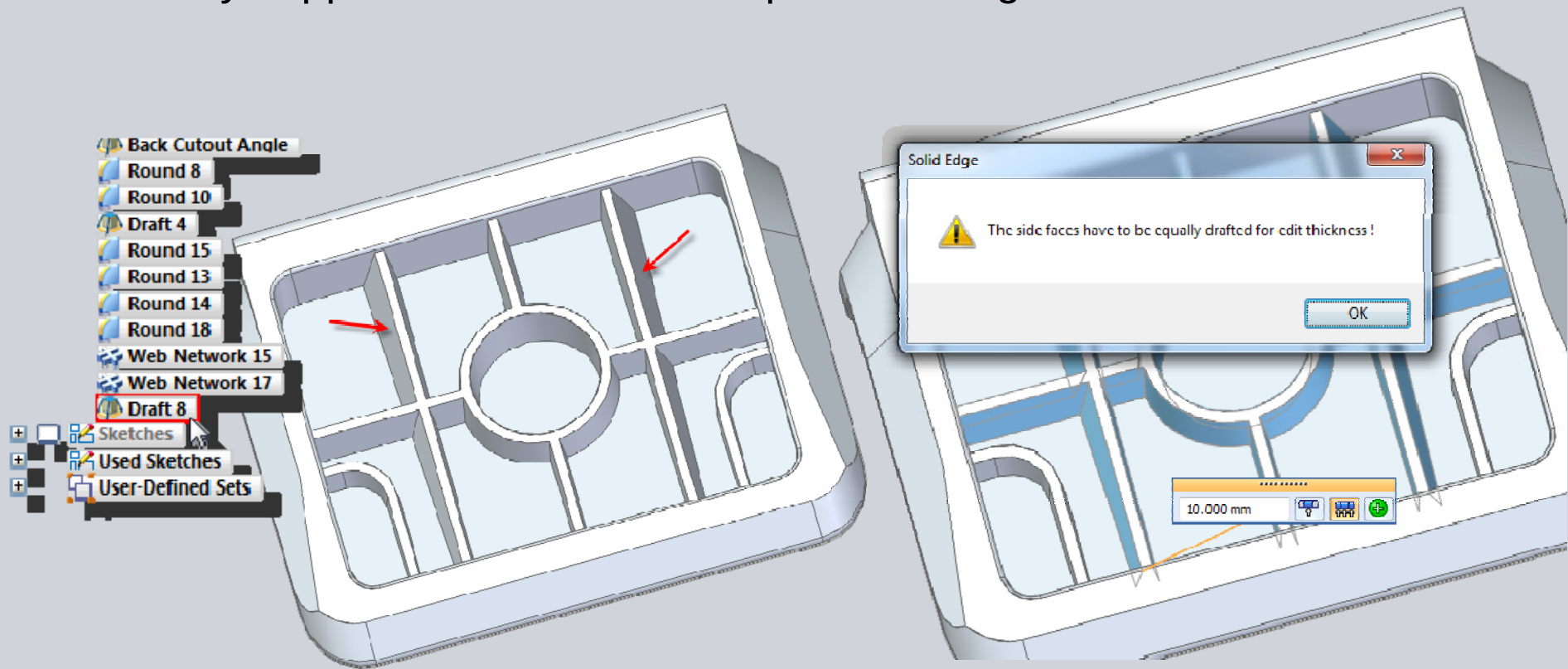
Rib/Web Network

- Dimensions to rib and web midpoints can be used to drive the location of the feature in the model



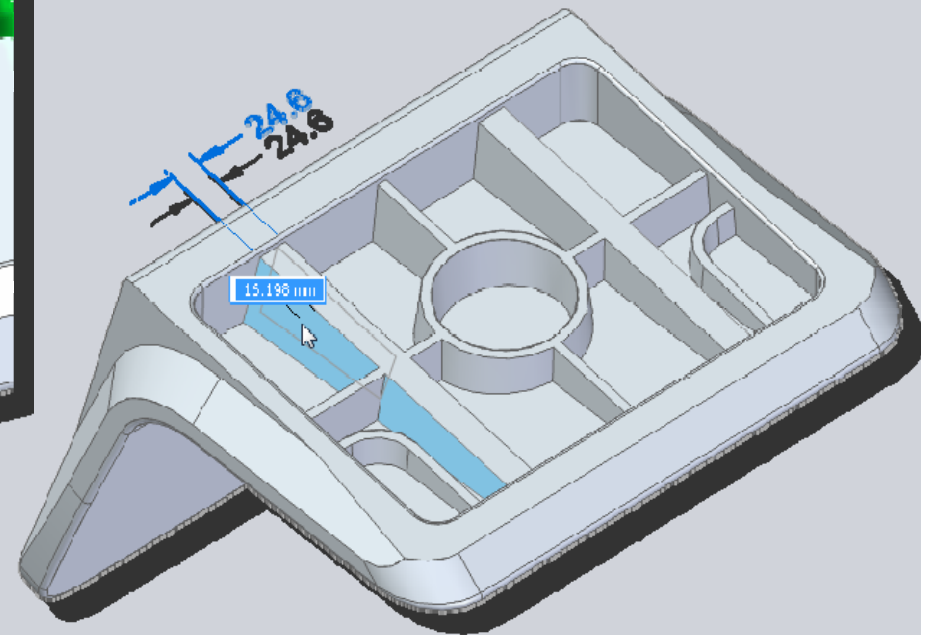
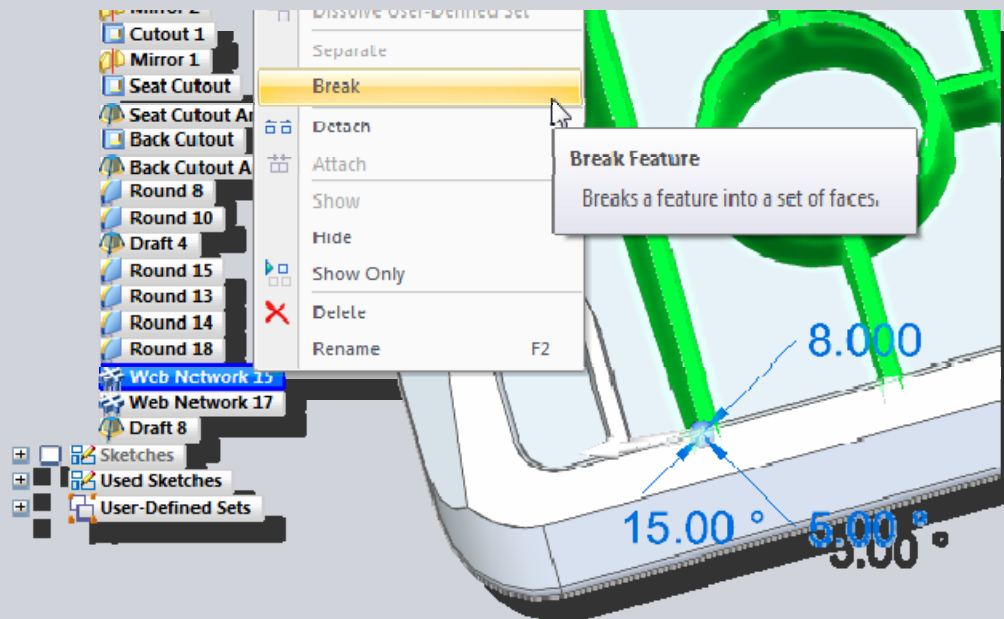
Rib/Web Network

- If the user has added draft to only one side of a web or rib, it is no longer valid for changing the thickness through the edit handle
 - Only supports webs/ribs with equal draft angles on both sides



Rib/Web Network

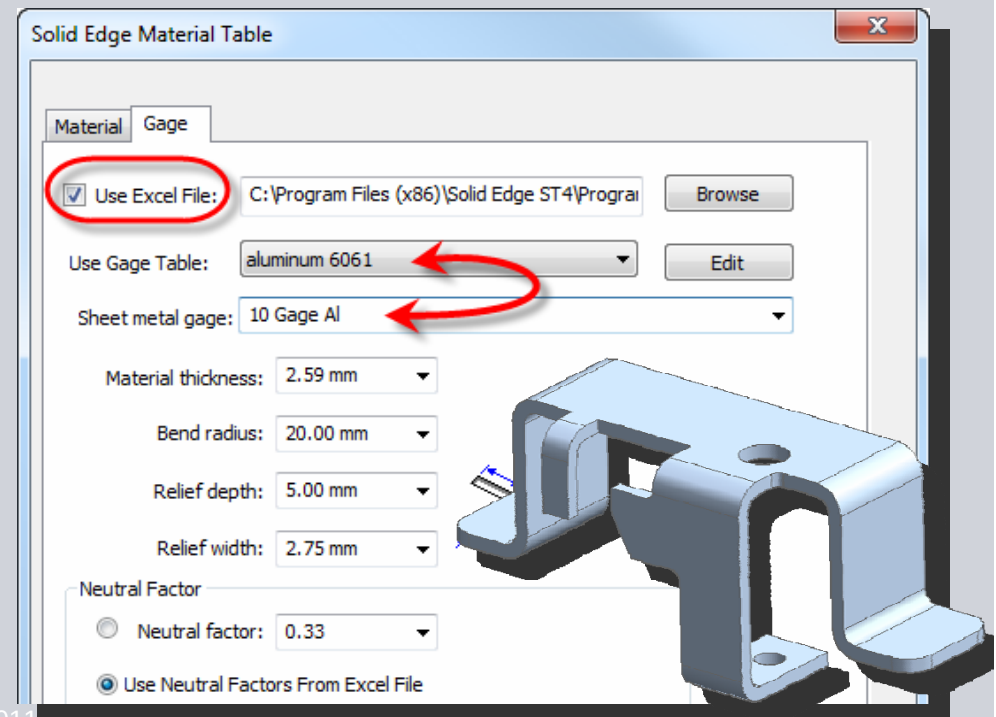
- User can "break" the web network or rib through the RMB click context menu
 - Becomes editable through PMI and synchronous drag functionality



DEMO 10

Material Table with Sheet Metal Gage

- A Gage Table option, "Use Excel File", has been added to the Material Table dialog for setting unique sets of user-defined gages for each material type (Gagetable.xls by default)
 - Example: In previous versions setting the material thickness to 10 gage aluminum changed the material thickness for all 10 gage material types
- In reality the material thickness is unique for each material type
 - Example: 10 gage steel and 10 gage aluminum are not the same material thickness and may not use the same default bend radius and relief values



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Material Table with Sheet Metal Gage

- The Gage Table contains a key which defines the start and end of the specific areas of the table for each material type

	A	B	C
1	Solid Edge	Local	
2	*100	Gage Start	
3	*200	NF Table Start	
4	*400	NF Table End	
5	*500	Gage End	
6			

- Each material type is defined as a new worksheet in the XLS file

	A	B	C	D	E	F
1	Process Type					
2	Units	inches	mm			
3	*100					
4	Gage Name	10 Gage				
5	Material Thickness	0.1345				
6	Bend Radius	1				
7	Relief Depth	0.268				
8	Relief Width	0.14				
9	Bend Parameter Type	1				
10	*200	Radius				
11	Angle	0.5	0.75	1	1.125	1.25
12		5	0.01	0.05	0.09	0.12
13		15	0.05	0.08	0.13	0.2
14		30	0.08	0.13	0.2	0.28
15		45	0.11	0.18	0.27	0.36
16		60	0.15	0.22	0.34	0.43
17		75	0.2	0.22	0.33	0.44
18		90	0.23	0.34	0.42	0.52
19		105	0.3	0.4	0.5	0.6
20		120	0.35	0.43	0.51	0.62
21		135	0.38	0.49	0.57	0.71
22		150	0.4	0.55	0.66	0.77
23		165	0.47	0.58	0.69	0.8
24		180	0.55	0.65	0.75	0.88
25	*400					
26	Notes:	Neutral Factors derived for demo				
27	*500					

Material Table with Sheet Metal Gage

- The units for each material type is specified at the top of each worksheet
- All gage information on that worksheet must use the same units
- Examples
 - All gages defined on Steel 01 use inch values in the table
 - All gages defined on Aluminum 6061 use millimeter values in the table

	A	B	C	D	E	F
1	Process Type					
2	Units	mm	inches			
3	*100					
4	Gage Name	10 Gage Al				
5	Material Thickness	2.59				
6	Bend Radius	20				
7	Relief Depth	5				
8	Relief Width	2.75				
9	Bend Parameter Type	1				
10	*200	Radius				
11	Angle	10	20	25	30	
12	5	0.1	0.1	0.15	0.2	
13	15	0.17	0.1	0.15	0.25	
14	30	0.19	0.15	0.2	0.3	
15	45	0.2	0.2	0.25	0.35	
16	60	0.21	0.2	0.25	0.4	
17	75	0.22	0.3	0.35	0.45	
18	90	0.23	0.3	0.35	0.5	
19	120	0.24	0.4	0.4	0.65	
20	135	0.25	0.4	0.5	0.68	
21	180	0.3	0.5	0.6	0.7	
22	*400					
23	Notes:	Neutral Factors derived for demo				
24	*500					
95						
96						
97						
98						
	Key	Steel 01	aluminum 6061	Stainless Steel		

Material Table with Sheet Metal Gage

- The Gage Table can also be used to define the neutral factor based on the material type, the bend radius, and the bend angle (optional)
 - Prior to ST4, the user could set one default neutral factor for the entire model
 - The user would have had to manually override any bend for which they wanted a different neutral factor based on bend radius and bend angle

Bend Equation

☒ Standard formula (See Help for actual formula)

☐ Neutral Factor:

☒ Use Neutral Factors From Excel File 

☐ Custom formula

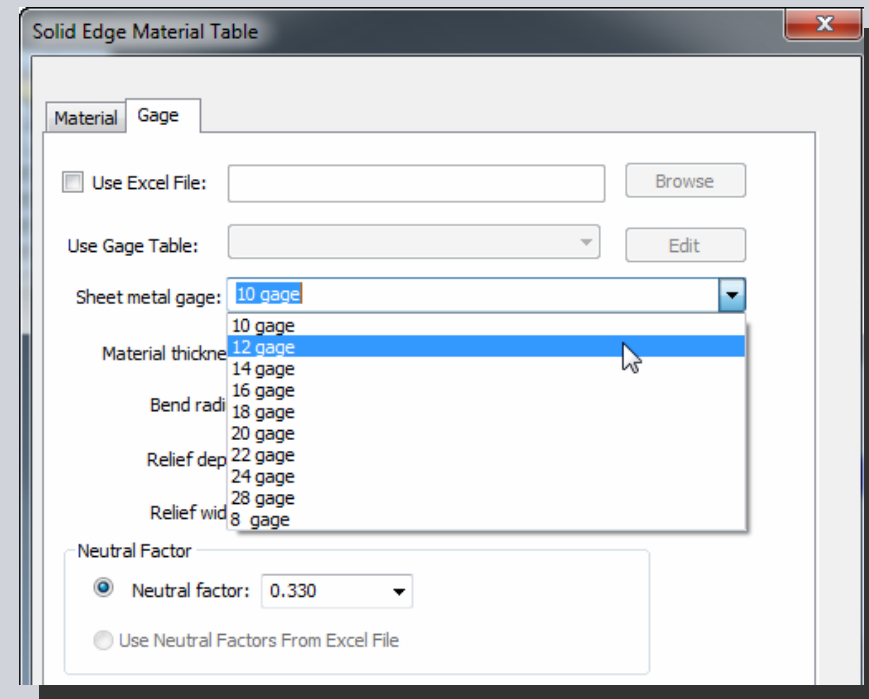
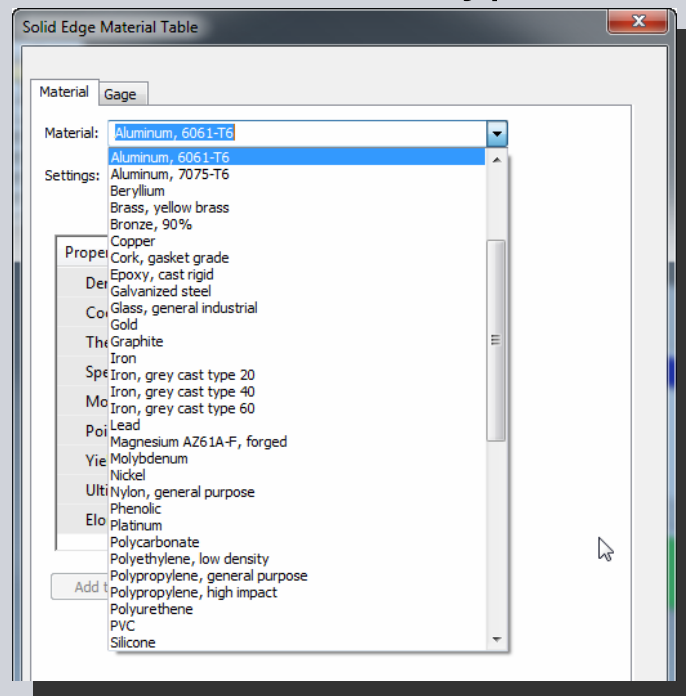
ProgramID.ClassName:

3	*100					
4	Gage Name	16 gage				
5	Material Thickness	1.59				
6	Bend Radius	3				
7	Relief Depth	3				
8	Relief Width	1.5				
9	Bend Parameter Type	1				
10	*200	Radius				
11	Angle	3	4	5	6.5	8
12		5	0.01	0.05	0.09	0.12
13		15	0.05	0.08	0.13	0.2
14		30	0.08	0.13	0.2	0.36
15		45	0.11	0.18	0.27	0.36
16		60	0.15	0.22	0.34	0.43
17		75	0.2	0.22	0.33	0.44
18		90	0.23	0.34	0.42	0.52
19		105	0.3	0.4	0.5	0.6
20		120	0.35	0.43	0.51	0.62
21		135	0.38	0.49	0.57	0.71
22		150	0.4	0.55	0.66	0.77
23		165	0.47	0.58	0.69	0.8
24		180	0.55	0.65	0.75	0.88
25	*400					

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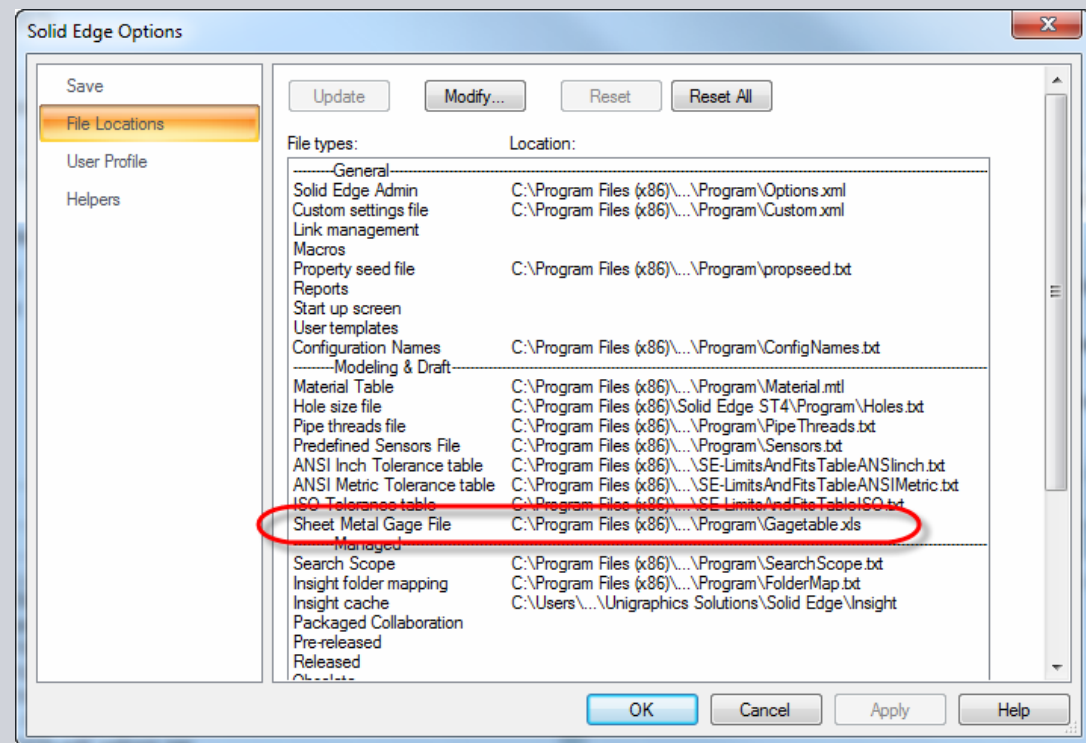
Material Table with Sheet Metal Gage

- The Gage Table is separate from the default Material Table (Material.mtl)
 - Material Table is the legacy functionality, which has not changed
 - Setting a gage thickness and neutral factor in the material table sets this same value for ALL material types



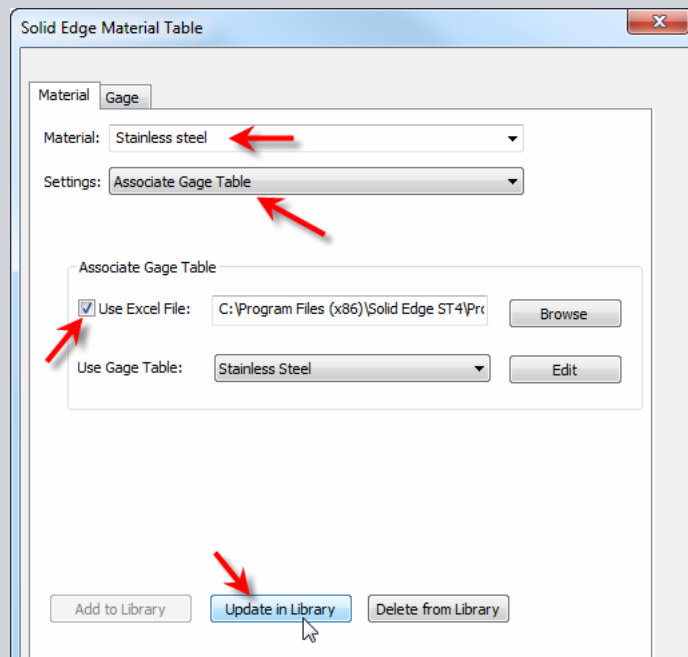
Material Table with Sheet Metal Gage

- The default Sheet Metal Gage Table file location is set in a new entry in the Solid Edge Options > File Locations
 - Enables a company to make a standard Gage Table file in a central location that all engineers in the company would point to when specifying a material for a new design
 - Gages may be changed globally to effect change in all associated parts

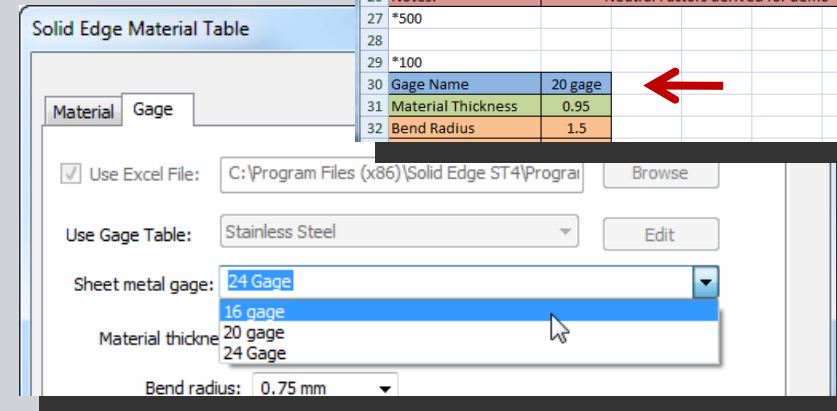


Material Table with Sheet Metal Gage

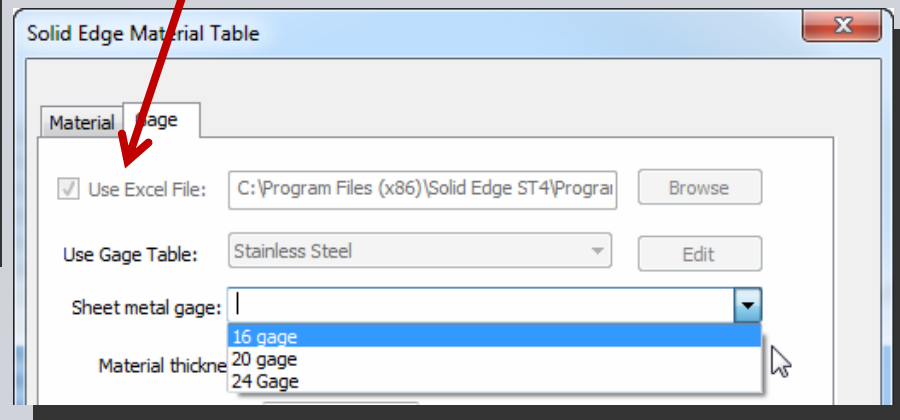
- A Material can be mapped to a Gage Table
 - "Update in Library" button forces the selected material to always use the Gage Table for that material's gage definition
 - Default Gages are defined in the Excel spreadsheet per material type



	A	B	C	D	E	F
1	Process Type	mm	inches			
2	Units					
3	*100					
4	Gage Name	16 gage				
5	Material Thickness	1.59				
6	Bend Radius	3				
7	Relief Depth	3				
8	Relief Width	1.5				
9	Bend Parameter Type	1				
10	*200	Radius				
11	Angle	3	4	5	6.5	8
12		5	0.01	0.05	0.09	0.12
13		15	0.05	0.08	0.13	0.2
14		30	0.08	0.13	0.2	0.36
15		45	0.11	0.18	0.27	0.42
16		60	0.15	0.22	0.34	0.5
17		75	0.2	0.22	0.33	0.55
18		90	0.23	0.34	0.42	0.62
19		105	0.3	0.4	0.5	0.7
20		120	0.35	0.43	0.51	0.76
21		135	0.38	0.49	0.57	0.83
22		150	0.4	0.55	0.66	0.88
23		165	0.47	0.58	0.69	0.94
24		180	0.55	0.65	0.75	0.88
25	*400					
26	Notes:	Neutral Factors derived for demo				
27	*500					
28						
29	*100					
30	Gage Name	20 gage				
31	Material Thickness	0.95				
32	Bend Radius	1.5				

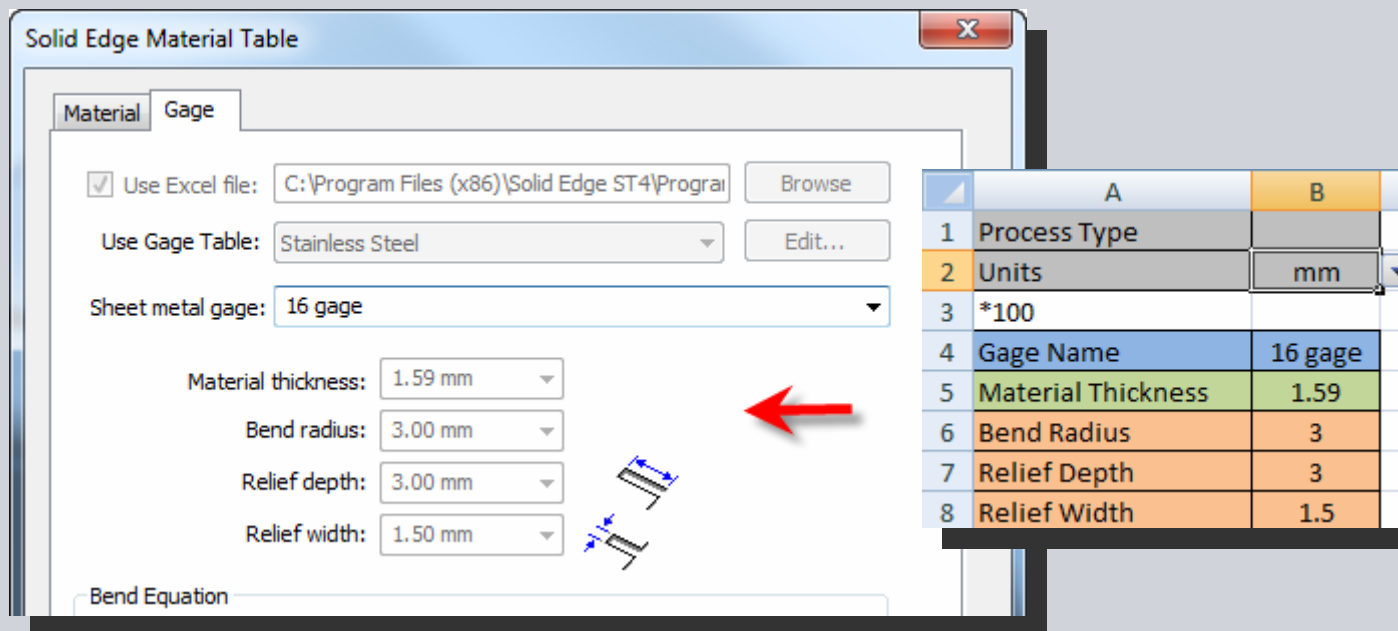


-
- A 3D CAD model of a mechanical part, likely a bracket or support, rendered in a light blue color. The model features several fillets (rounded edges) and chamfers (beveled edges). Blue arrows and text labels indicate the specific angles for these features: 65° for the top-left fillet, 65° for the bottom-left fillet, 15° for the top-right fillet, 15° for the bottom-right fillet, 102° for the top-right chamfer, and 102° for the bottom-right chamfer. The part has a central rectangular cutout and a circular hole on the right side.



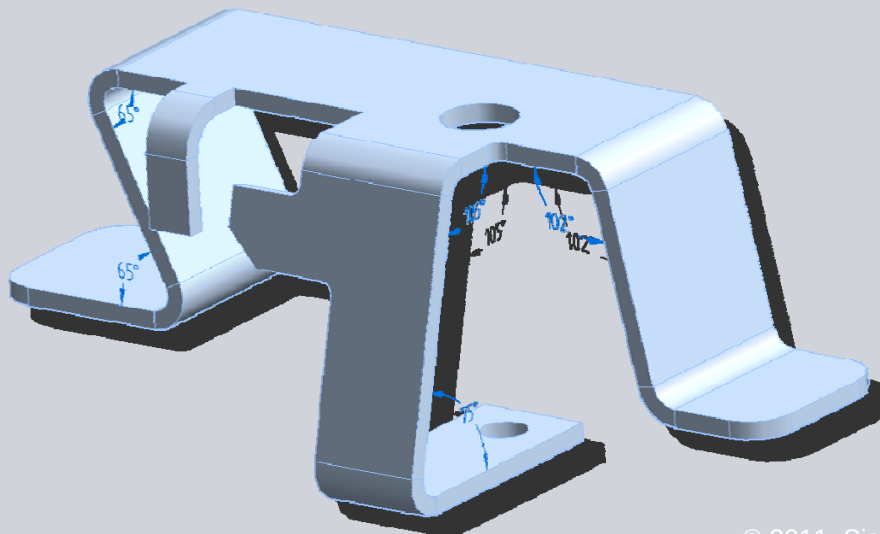
Material Table with Sheet Metal Gage

- Material Thickness, Bend radius, Relief Depth and Relief Width cannot be overridden if a gage table is being used as this is defined in the table
- Table values from the table are displayed in the dialog



Material Table with Sheet Metal Gage

- Note that the neutral factor extracted from the gage table for the individual bends is not actually applied to the model until it is flattened, so you will not see any change to the bent state of the model when changing these values in the table



	A	B	C	D	E	F
1	Process Type					
2	Units	mm	inches			
3	*100					
4	Gage Name	16 gage				
5	Material Thickness	1.59				
6	Bend Radius	3				
7	Relief Depth	3				
8	Relief Width	1.5				
9	Bend Parameter Type	1				
10	*200	Radius				
11	Angle	3	4	5	6.5	8
12	5	0.01	0.05	0.09	0.12	0.2
13	15	0.05	0.08	0.13	0.2	0.32
14	30	0.08	0.13	0.2	0.28	0.36
15	45	0.11	0.18	0.27	0.36	0.42
16	60	0.15	0.22	0.34	0.43	0.5
17	75	0.2	0.22	0.33	0.44	0.55
18	90	0.23	0.34	0.42	0.52	0.62
19	105	0.3	0.4	0.5	0.6	0.7
20	120	0.35	0.43	0.51	0.62	0.76
21	135	0.38	0.49	0.57	0.71	0.83
22	150	0.4	0.55	0.66	0.77	0.88
23	165	0.47	0.58	0.69	0.8	0.94
24	180	0.55	0.65	0.75	0.88	0.99
25	*400					
26	Notes:	Neutral Factors derived for demo				
27	*500					
28						
29	*100					
30	Gage Name	20 gage				
31	Material Thickness	0.95				
32	Bend Radius	1.5				

Material Table with Sheet Metal Gage

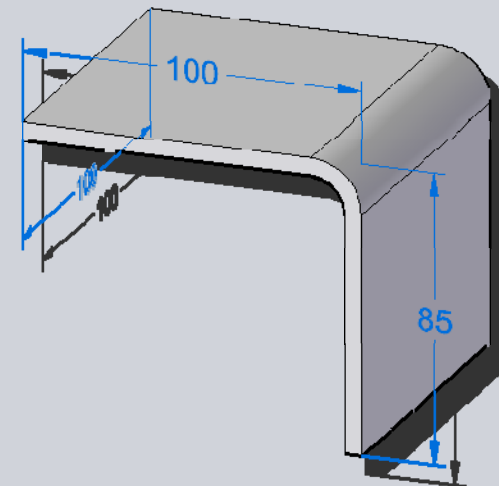
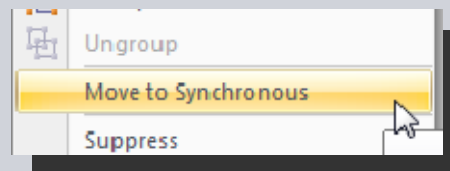
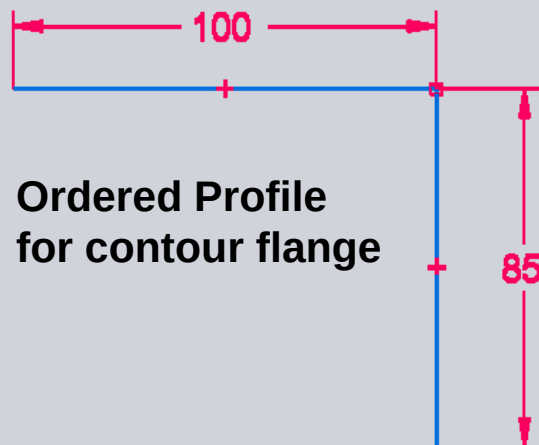
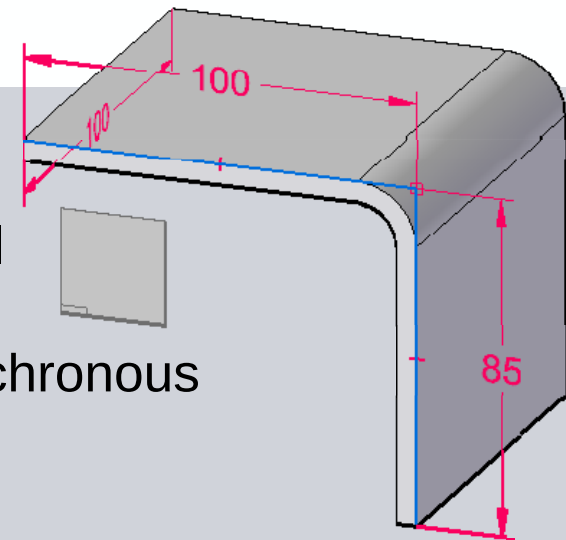
- When a material is mapped to a gage table, the associative link may be broken thorough typical file management mistakes
 - ST4 detects the missing link
 - When the link is broken:
 - The user is notified with error message:
 - on file open
 - on change of material to one with a missing link
 - on flatten
- The required gage properties values then default to their last known values

DEMO 11

Sheet Metal Move to Synchronous Improvements

SIEMENS

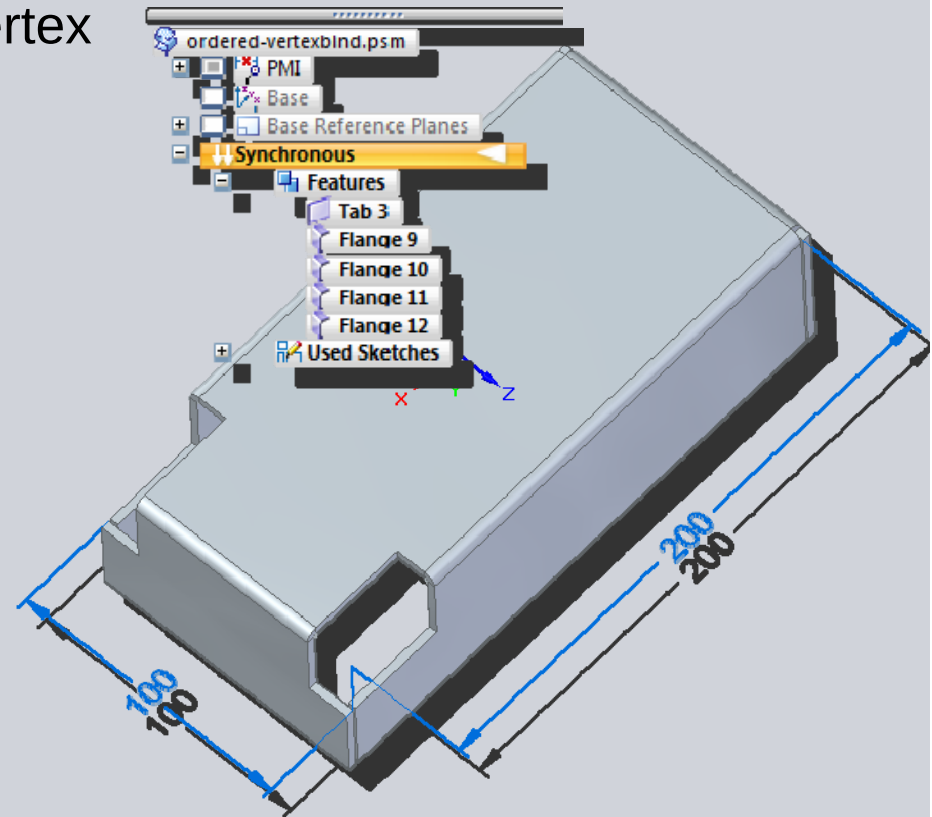
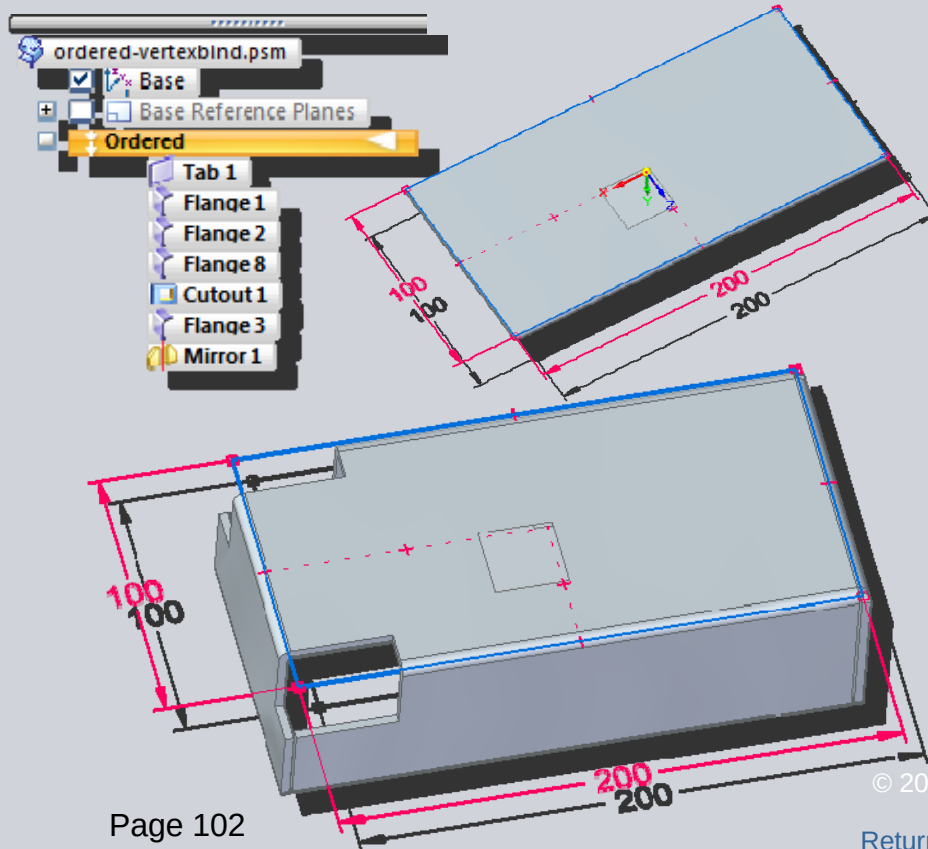
- In **ST3** Sheet Metal, the Move to Synchronous command did NOT support the migration of local profile dimensions to body edges
 - The dimensions were deleted on Move to Synchronous in sheet metal files
- In **ST4**, logic was added to all dimensions that are migrated from sheet metal profiles (ordered) to bind to virtual vertices when the features are moved to synchronous



Sheet Metal Move to Synchronous Improvements

SIEMENS

- When moving ordered sheet metal features to synchronous, Solid Edge looks for Virtual Vertices as first priority for all dimensions
 - If it is not possible to bind to a virtual vertices, Solid Edge binds dimensions to the nearest hard vertex



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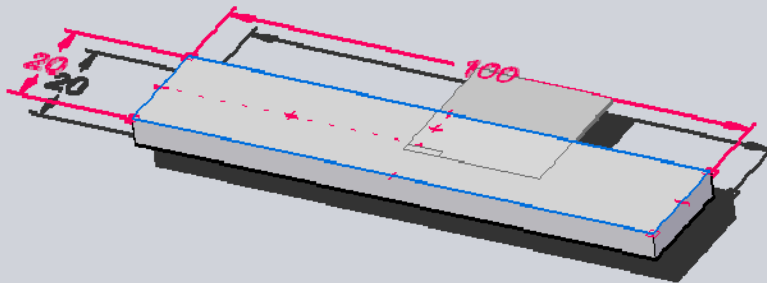
[Return to Topics](#)

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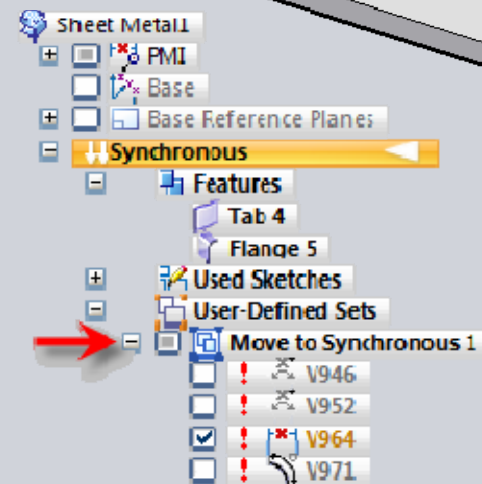
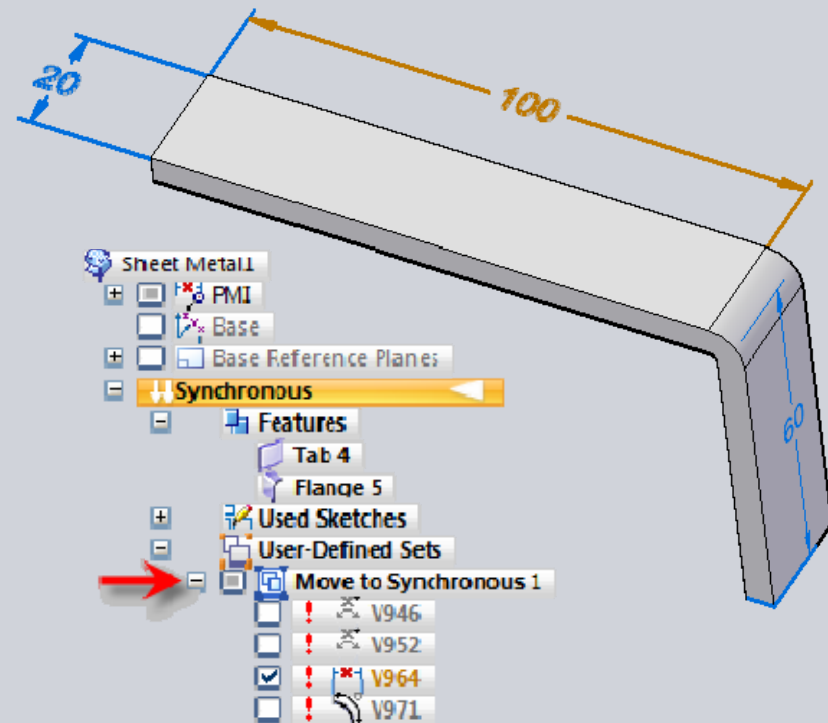
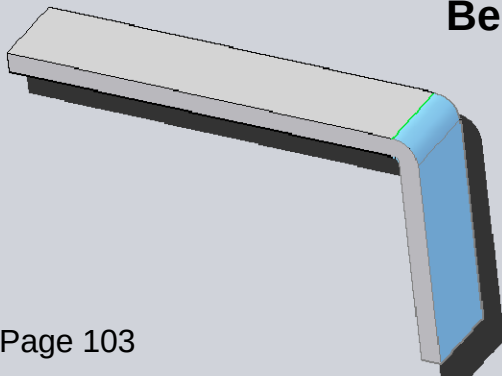
Sheet Metal Move to Synchronous Improvements

SIEMENS

- In some cases a hard edge or virtual vertex cannot be found and in this case Solid Edge leaves the dimension dangling
 - A user defined set is created for all dangling dimensions for each Move to Synchronous operation



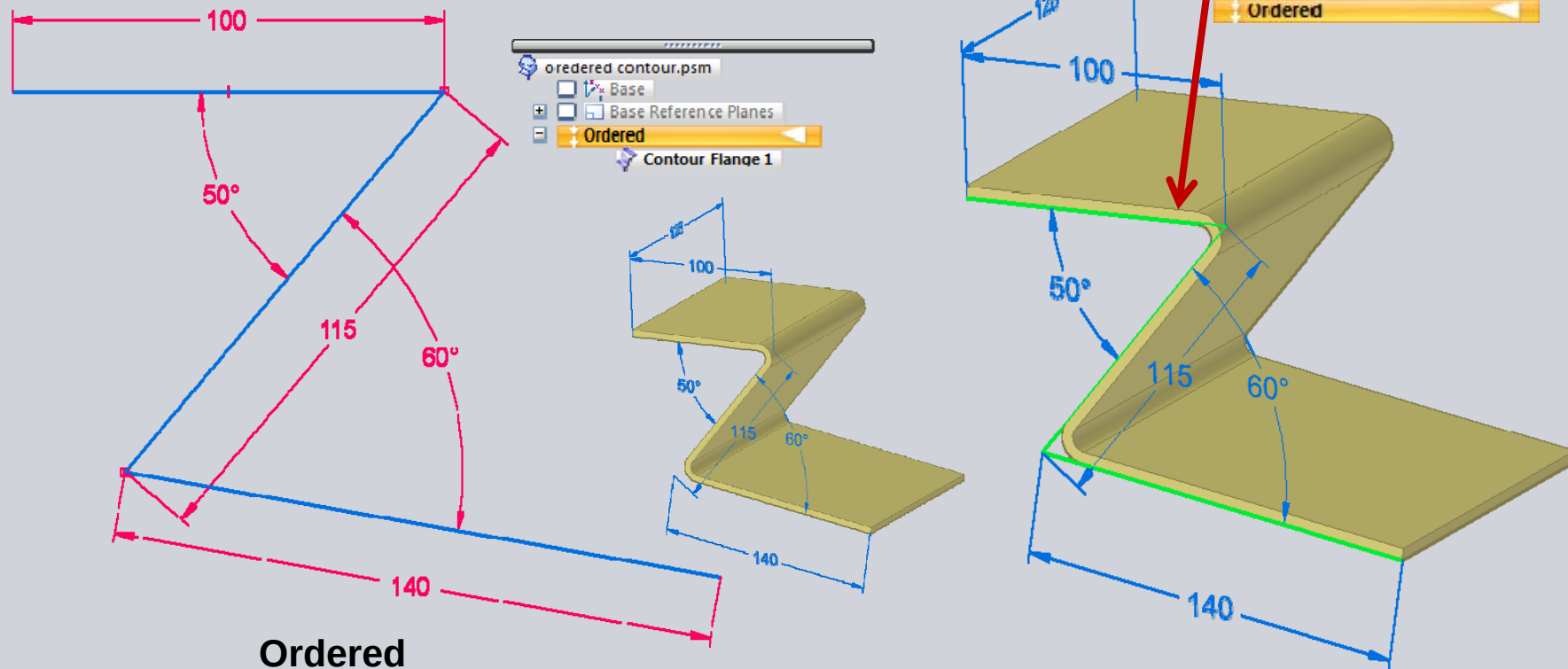
Bend Outside



Sheet Metal Move to Synchronous Improvements

SIEMENS

- Local Profiles sketches are converted to Used Sketches in synchronous
 - Profile sketch names are the same as the Ordered feature name



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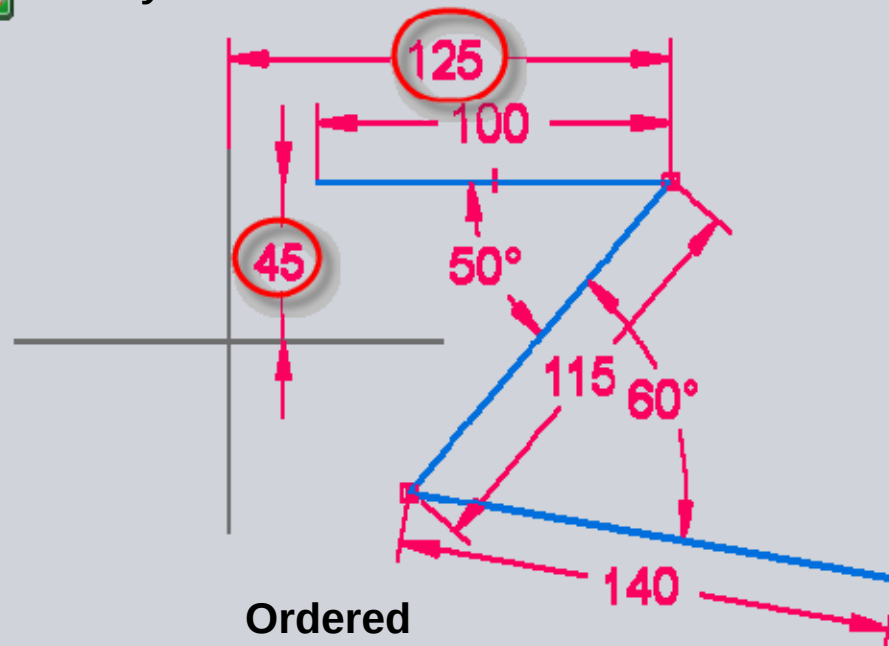
[Return to Topics](#)

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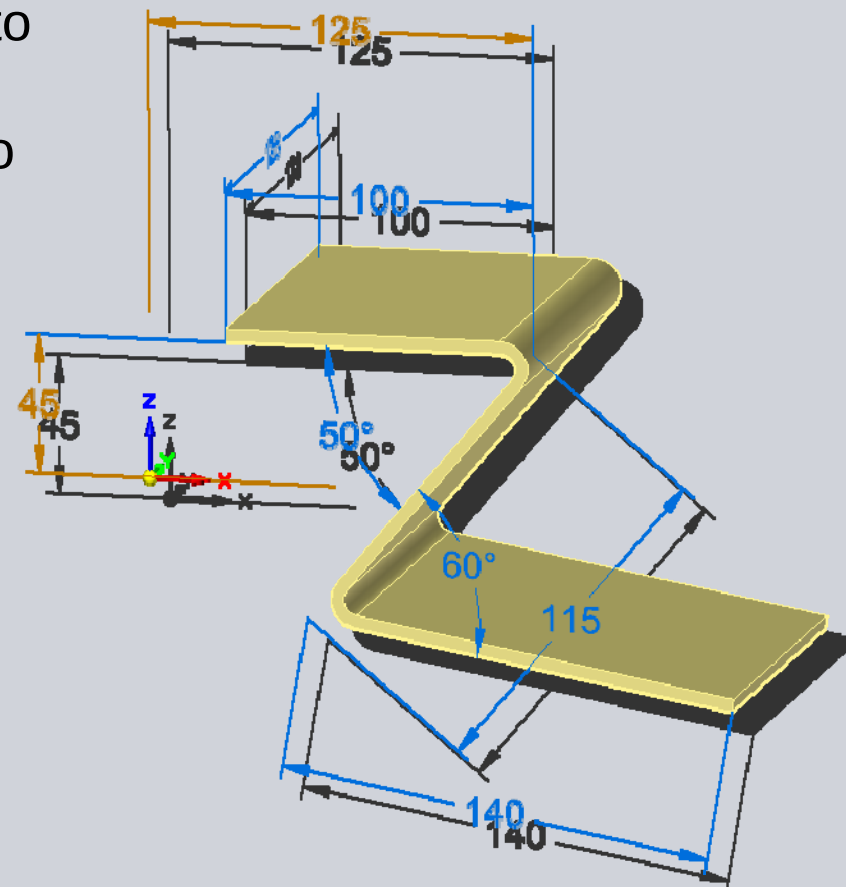
Sheet Metal Move to Synchronous Improvements

SIEMENS

- In Synchronous it is not possible to place a dimension between a reference plane and a body edge
 - If an Order sketch is dimensioned to a reference plane, it becomes a dangling dimension when moved to synchronous



Ordered



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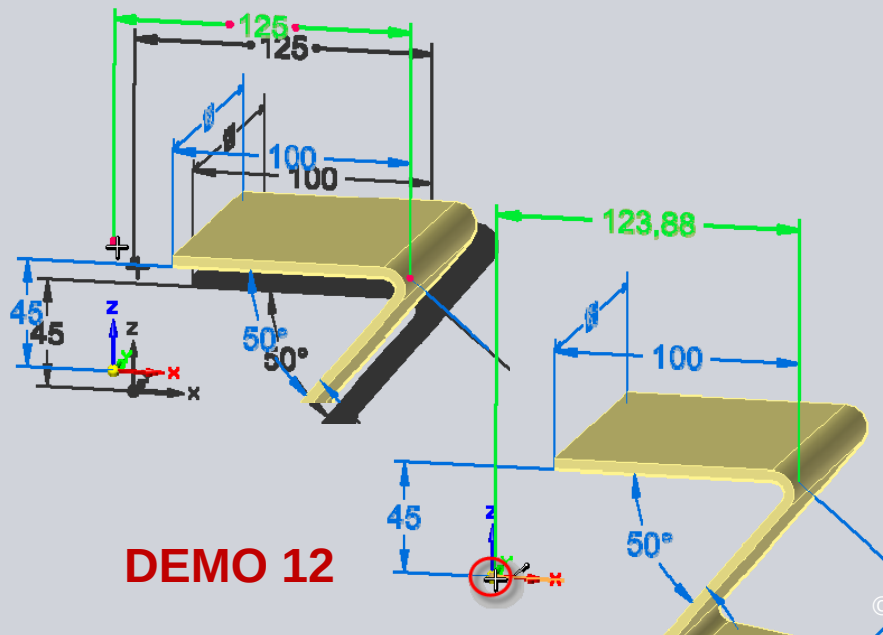
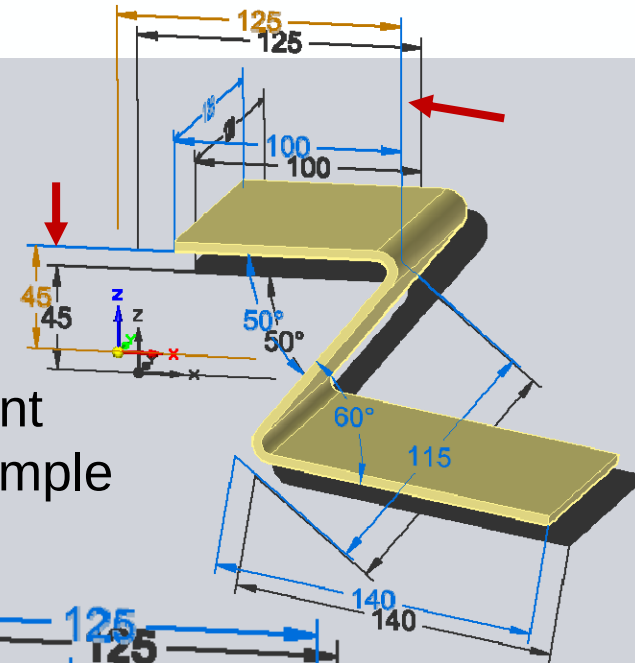
[Return to Topics](#)

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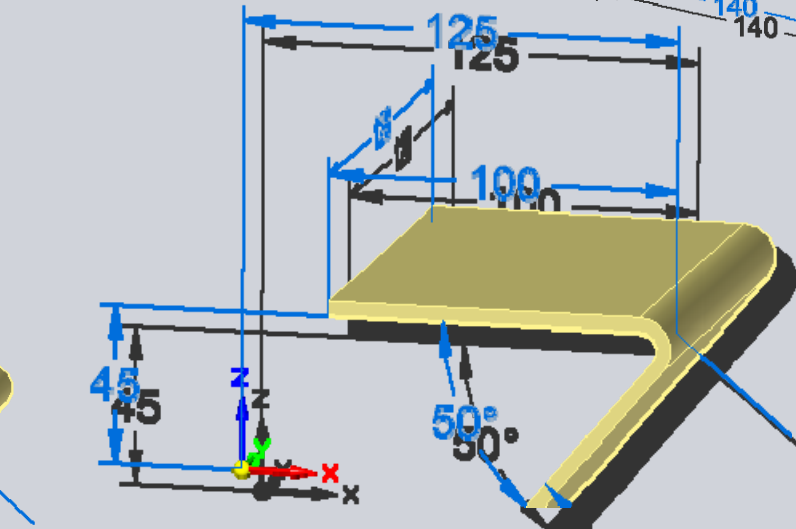
Sheet Metal Move to Synchronous Improvements

SIEMENS

- If one end of the dangling dimension attaches to the model, it is shown in the driven dimension color
- User can use the ALT+Click method to drag the dangling endpoint to a new keypoint
 - Base Coordinate System origin in this example



DEMO 12



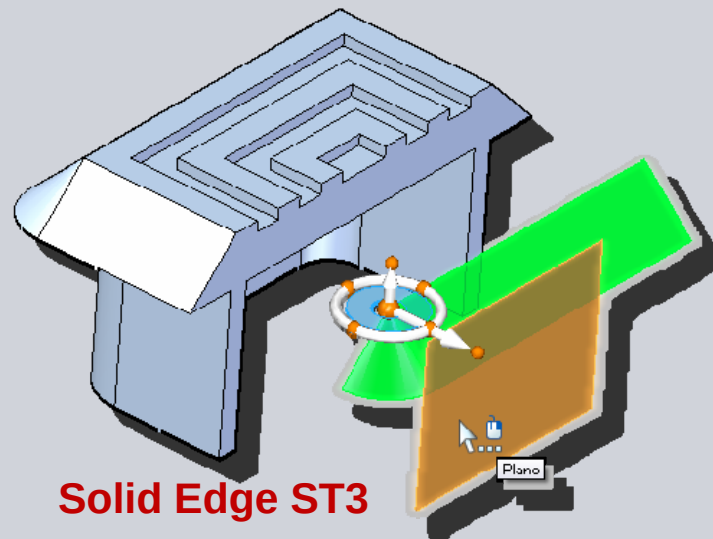
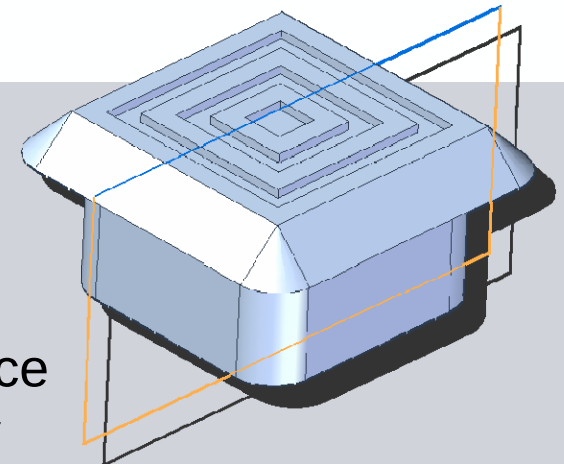
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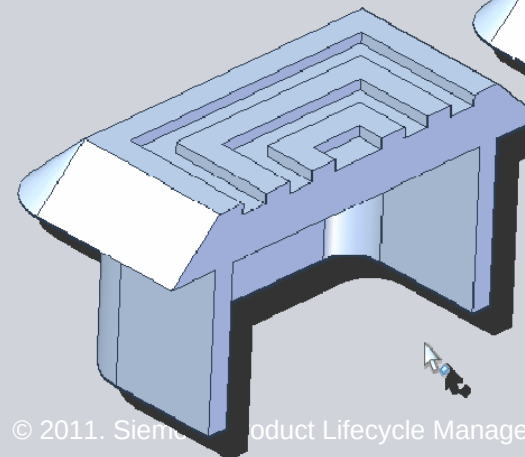
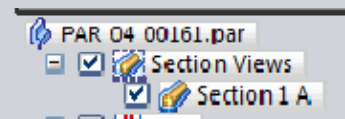
Siemens PLM Software

PMI Section Improvements

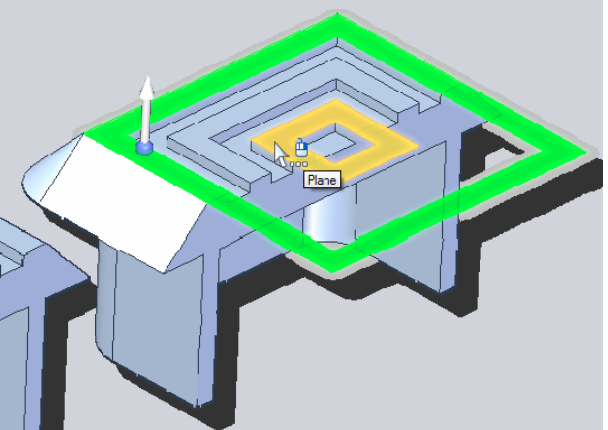
- In ST4 we no longer locate cut-away faces of PMI Section View parts
 - Locating a surviving portion of a face which has been partially cut away displays the entire face
 - Locating a surviving portion of a part in assembly which has been partially cut away displays the entire part
 - If faces are cut away they are no longer available via QuickPick



Solid Edge ST3

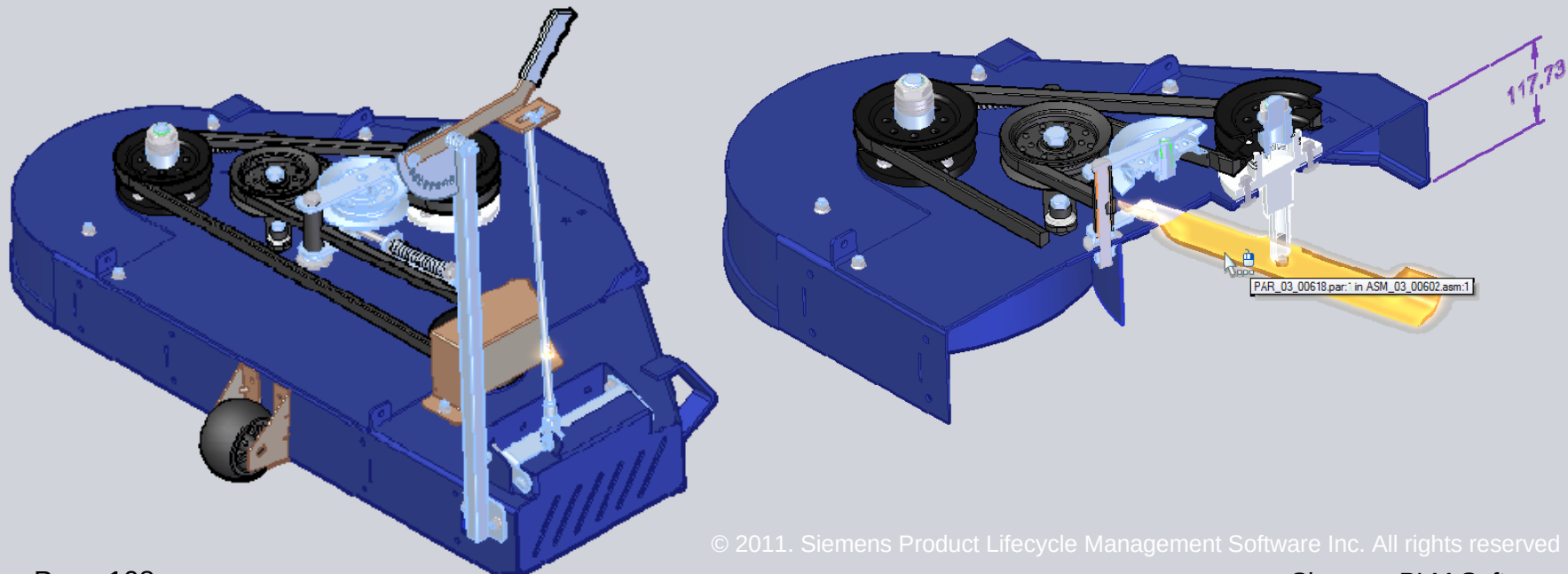
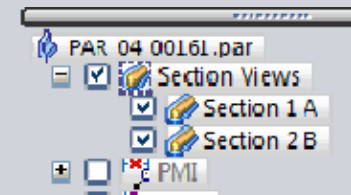


Solid Edge ST4



PMI Section Improvements

- Check boxes were added to PathFinder for enabling and disabling the PMI sections
- In Assembly only, the Fit command only considers currently displayed faces when PMI section is enabled
 - Fits to displayed geometry and PMI



THANK – YOU!