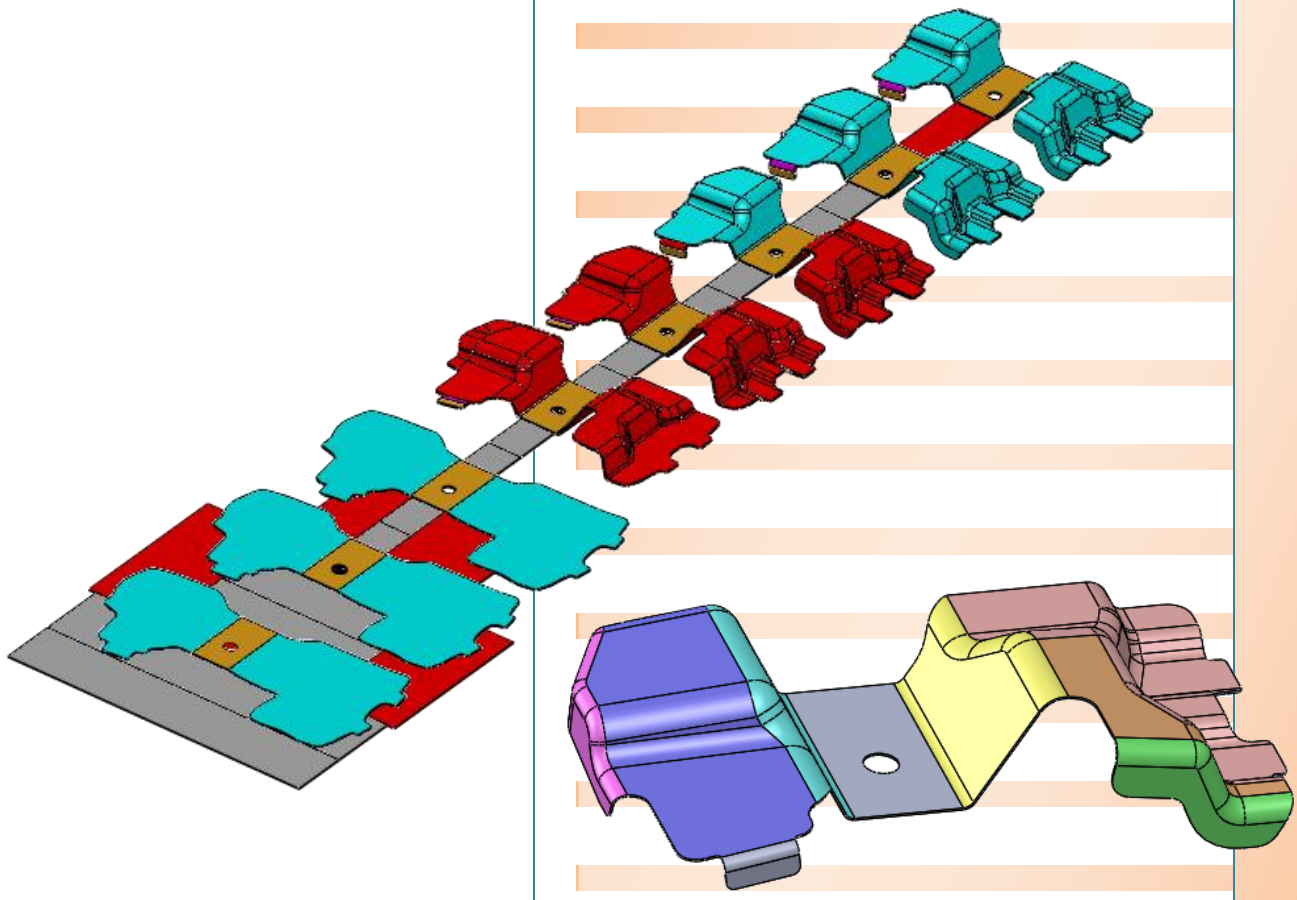
3DQuickPress

What's New in Version 6



QUICK PRESS

V6

3D QuickTools Limited

Contents

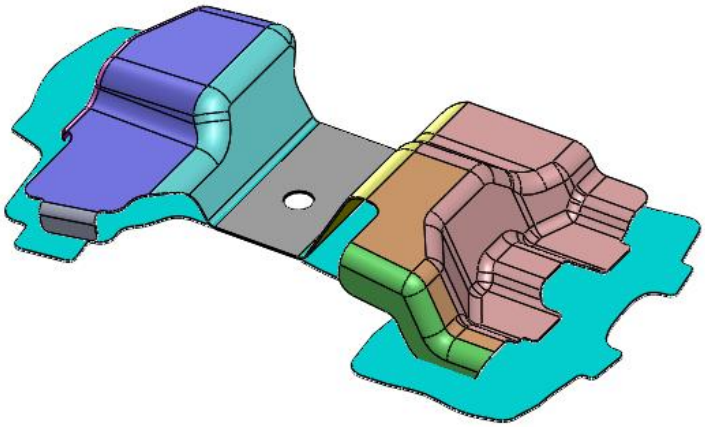
Highlights

3DQuickPress and 3DQuickForm integration

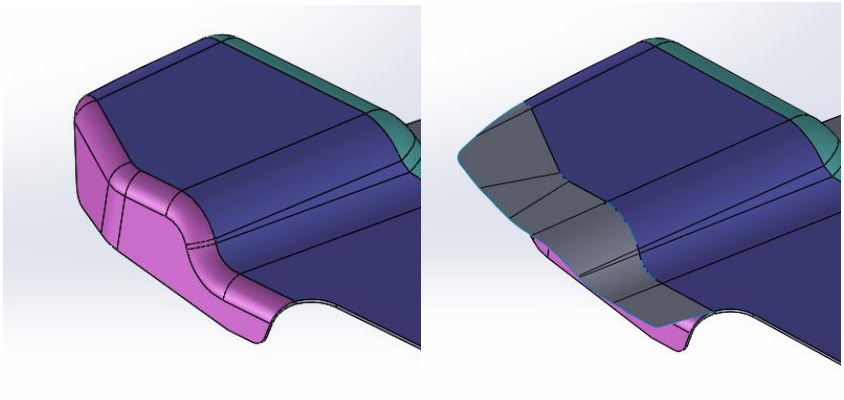
With 3DQuickPress version 6 and 3DQuickForm version 3.2, you may now apply the blank development result of 3DQuickForm directly to the unfold model of 3DQuickPress. You may unfold a complex forming parts without losing the benefits of automatic feature. And you may obtain a feature based unfold model with blank development results.

Benefits when you have both 3DQuickPress and 3DQuickForm

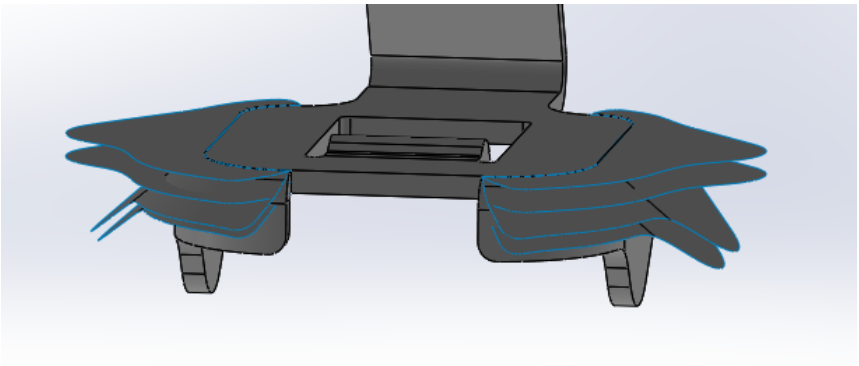
- Direct output of 3DQuickForm blank development surface for user defined features

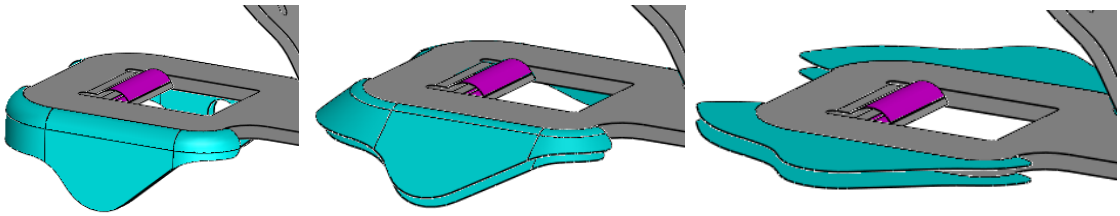


- Automatic un-flange surface creation

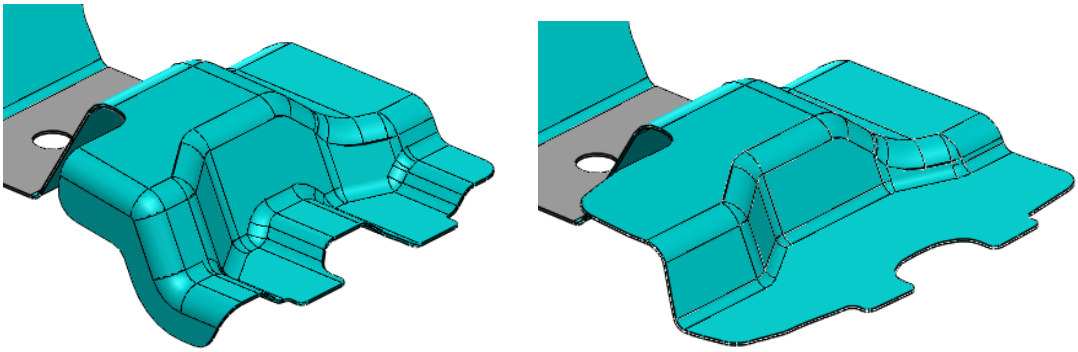


- Multiple step unfolding of flange feature





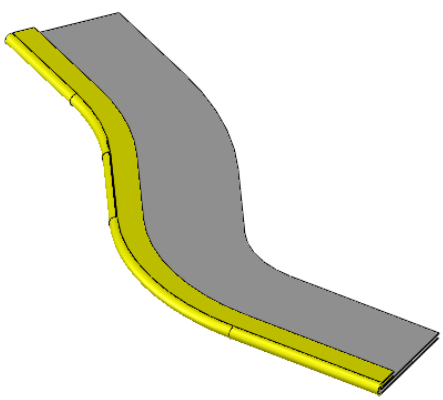
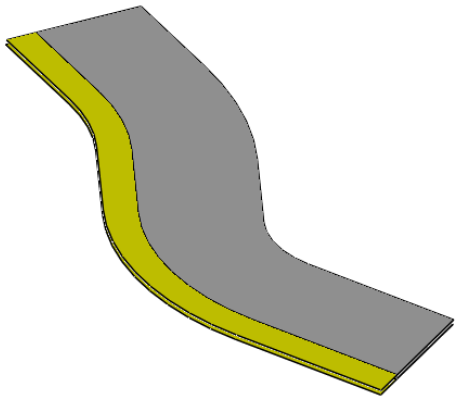
- Unfold to initial blank surface



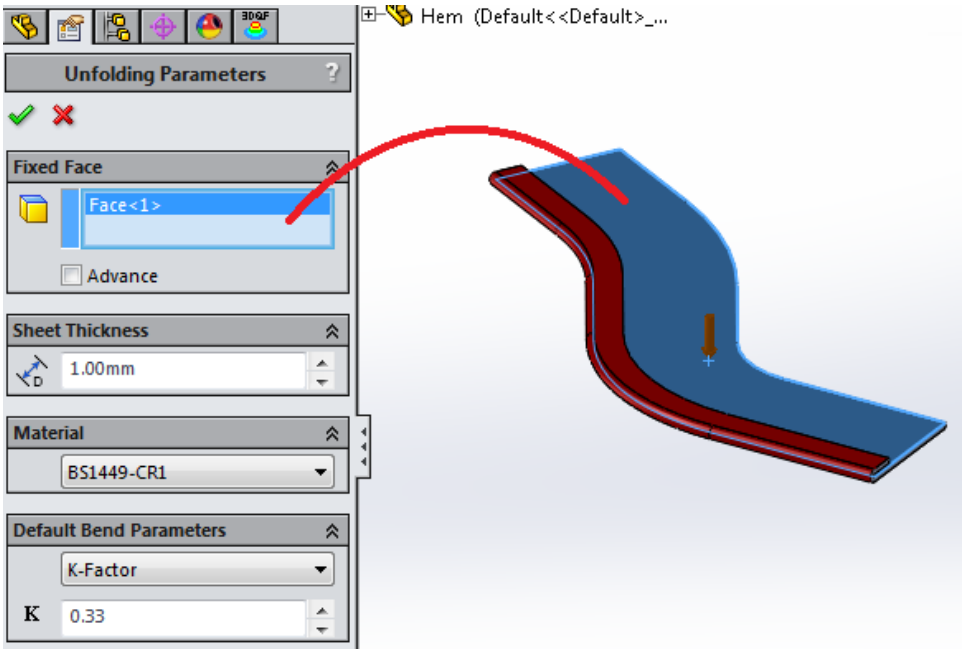
Unfold

Hem

Hem feature is a new feature induced to handle a region folded narrowly along the edge of a part. The following steps will show how to setup a hem feature in an unfold part.

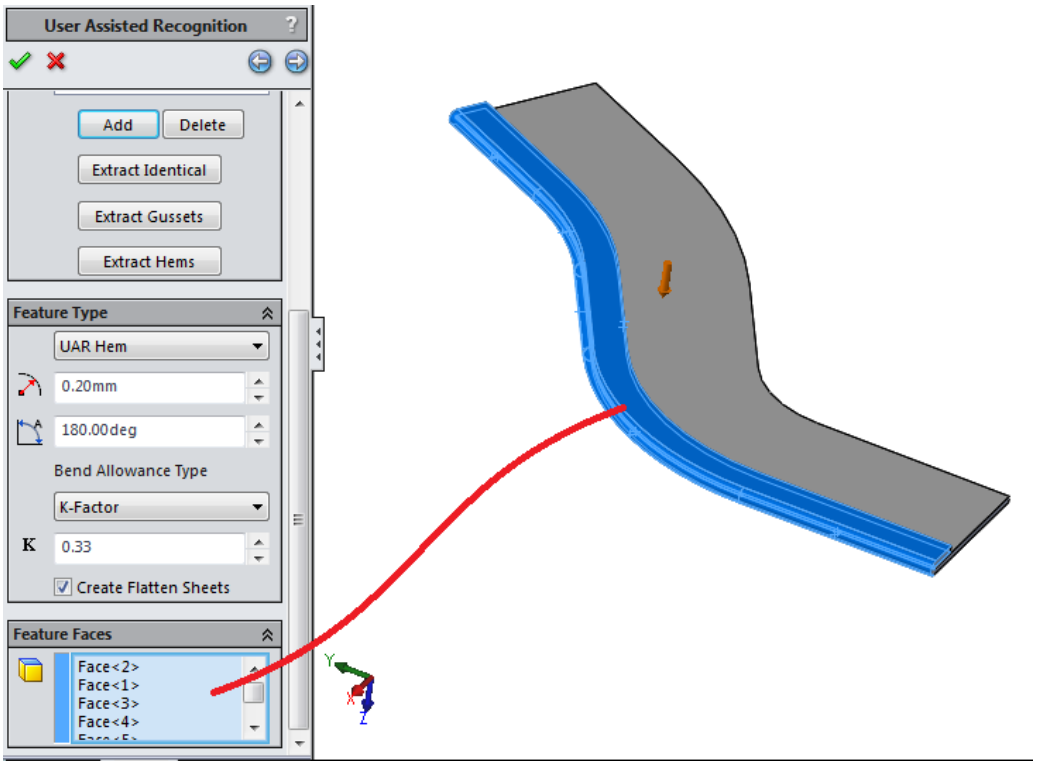



1. Click **Unfold**  on the **3DQP Toolbar**
2. Select the highlighted face as the **fixed face** and click **OK** 

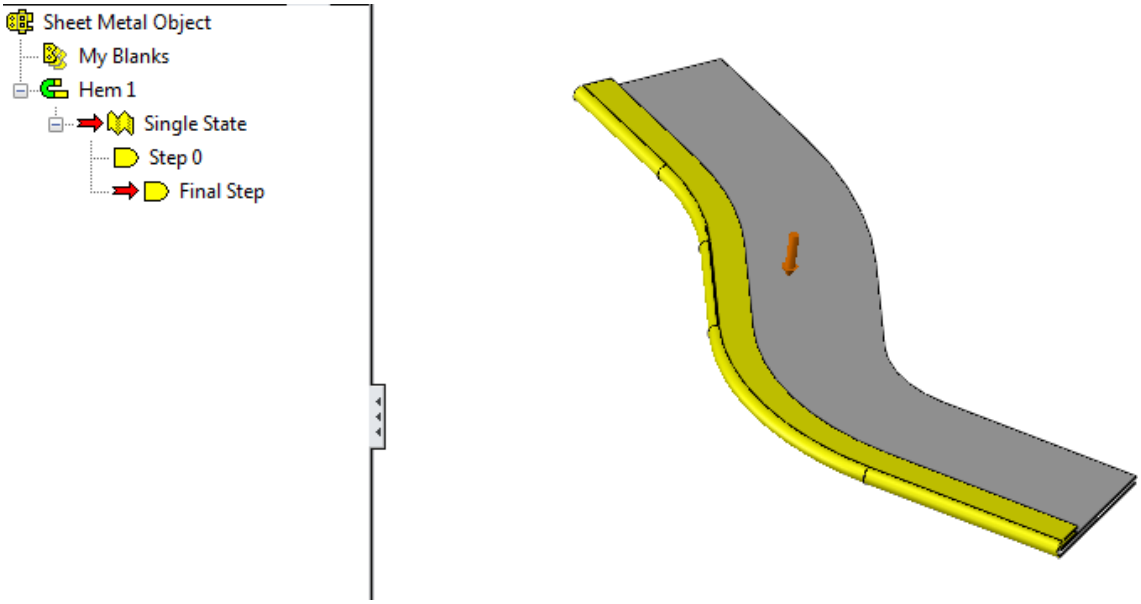


3. **RMB** on **Sheet Metal Object** and select **User Assisted Recognition…**
4. Click **Add** to add feature

5. Select the feature type to be **UAR Hem**
6. Select the highlighted faces as the feature face (For faces inside the hem, please select with **RMB** select other)

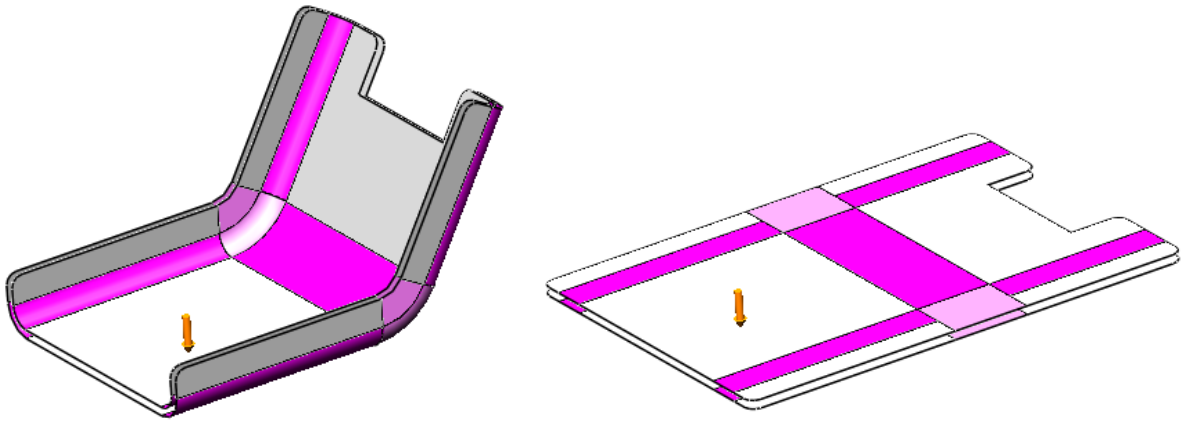


7. Enter 0.2mm as **hem radius**, and 180deg for **hem angle**.
8. Click **OK** to rebuild the unfold part
9. **Hem Feature** is created

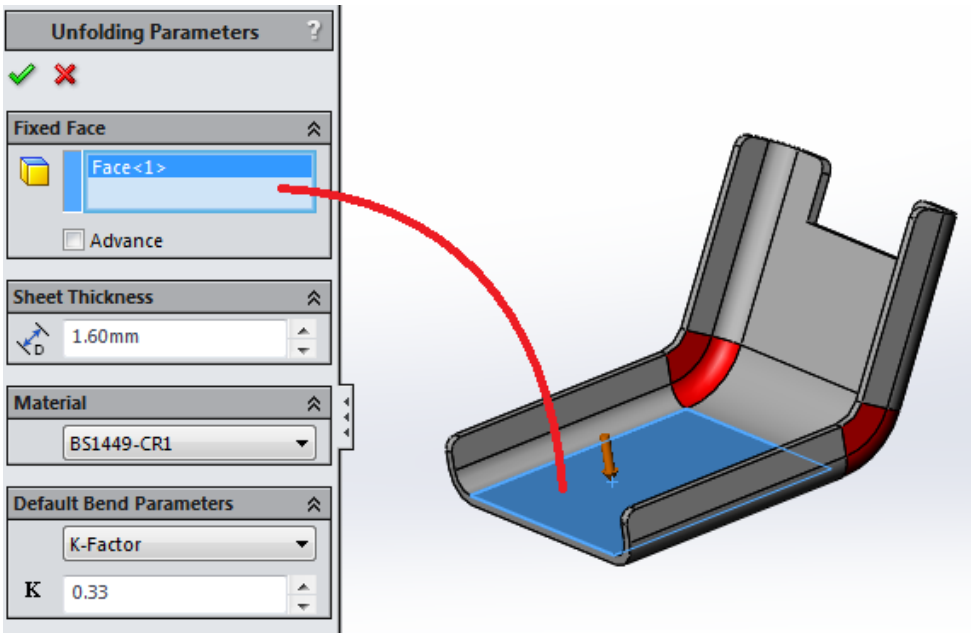


Bend Corner

Bend corner feature represent the connecting area between 2 bend features. The following steps will show how to setup a bend corner feature in an unfold part.

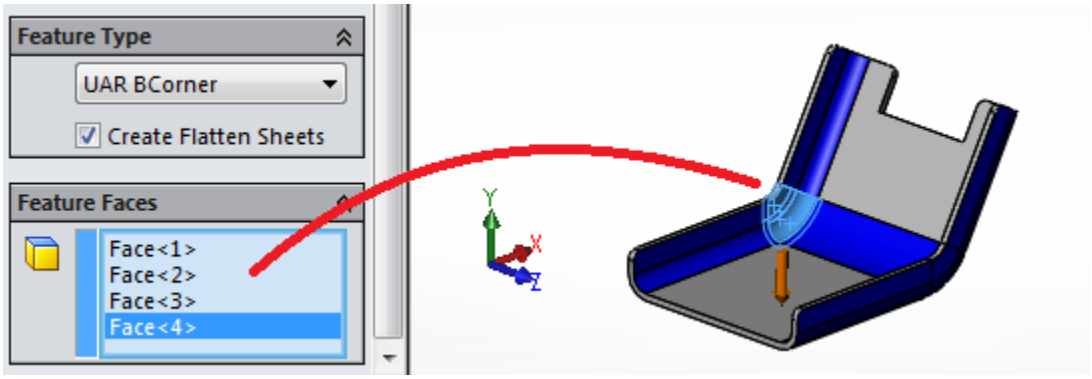


1. Click **Unfold** on the **3DQP Toolbar**
2. Select the highlighted face as the **fixed face** and click **OK**

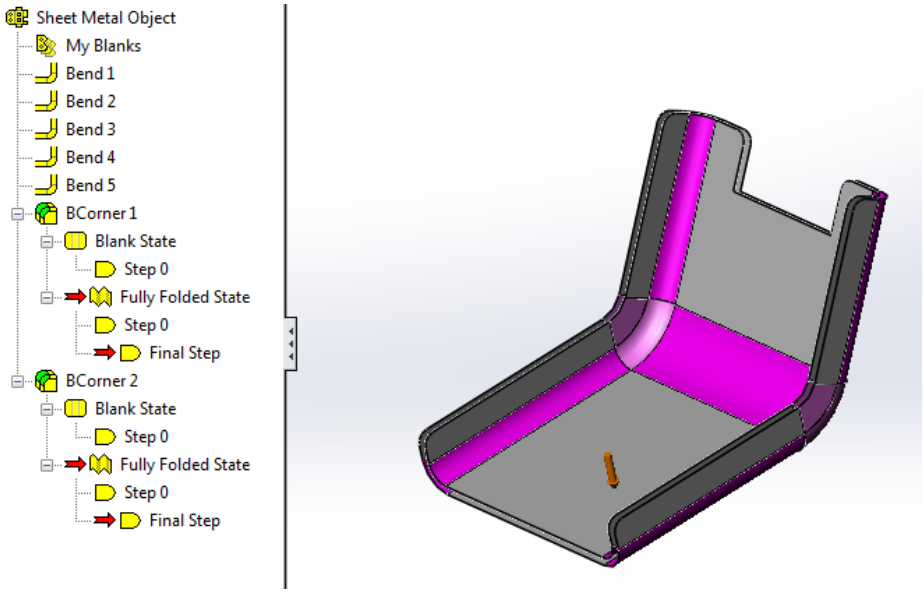


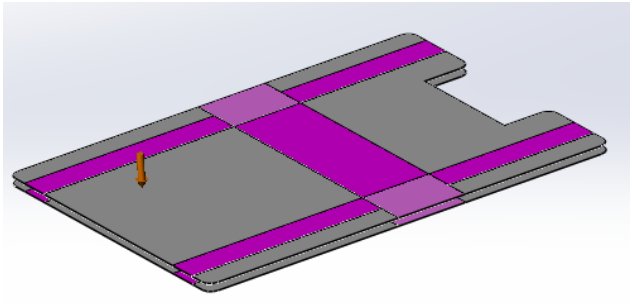
3. **RMB** on **Sheet Metal Object** and select **User Assisted Recognition…**
4. Click **Add** to add feature
5. Select the feature type to be **UAR BCorner**

6. Select the highlighted faces as the feature face



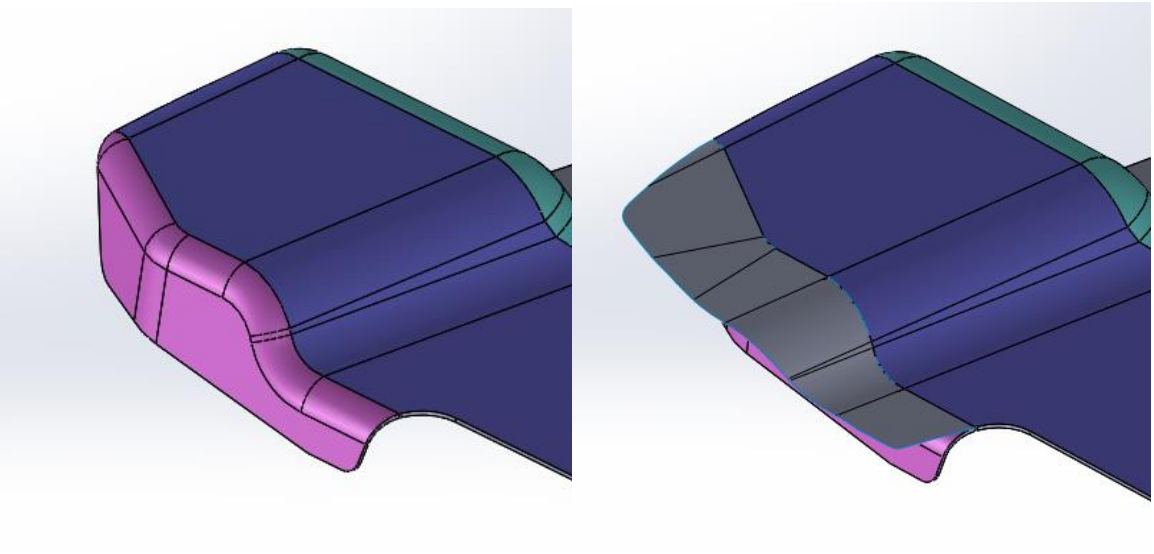
7. Repeat step 4 to 6 to create the 2nd **Bend Corner feature**
8. Click **OK** to rebuild the unfold part
9. **Bend Corner Features** are created



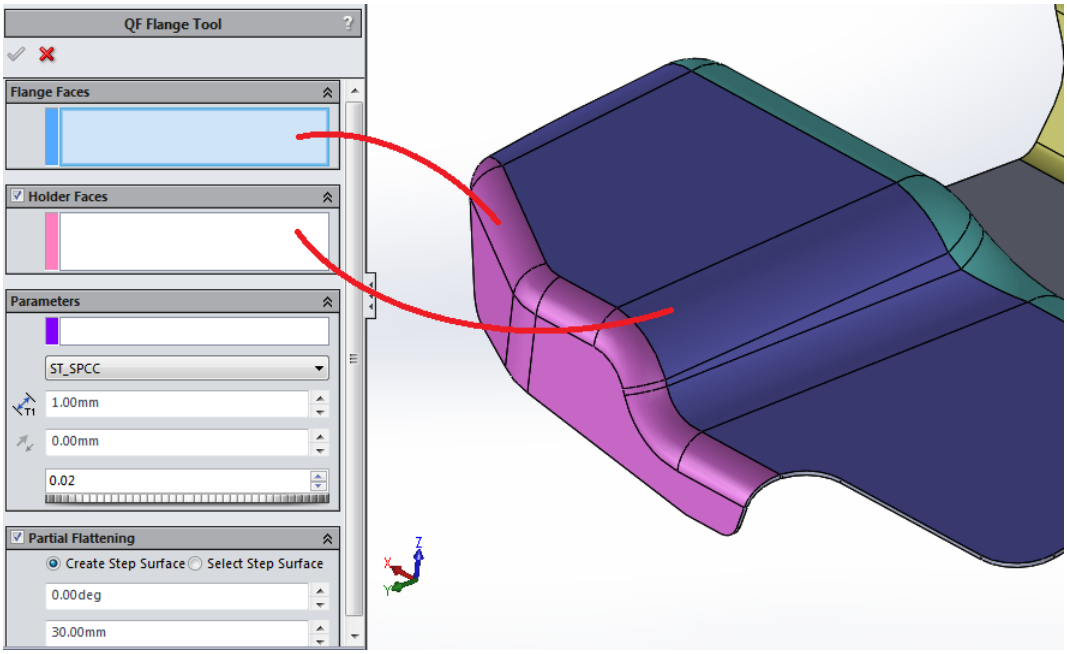


QF Flange Tool **(Required 3DQuickForm Professional version installed)**

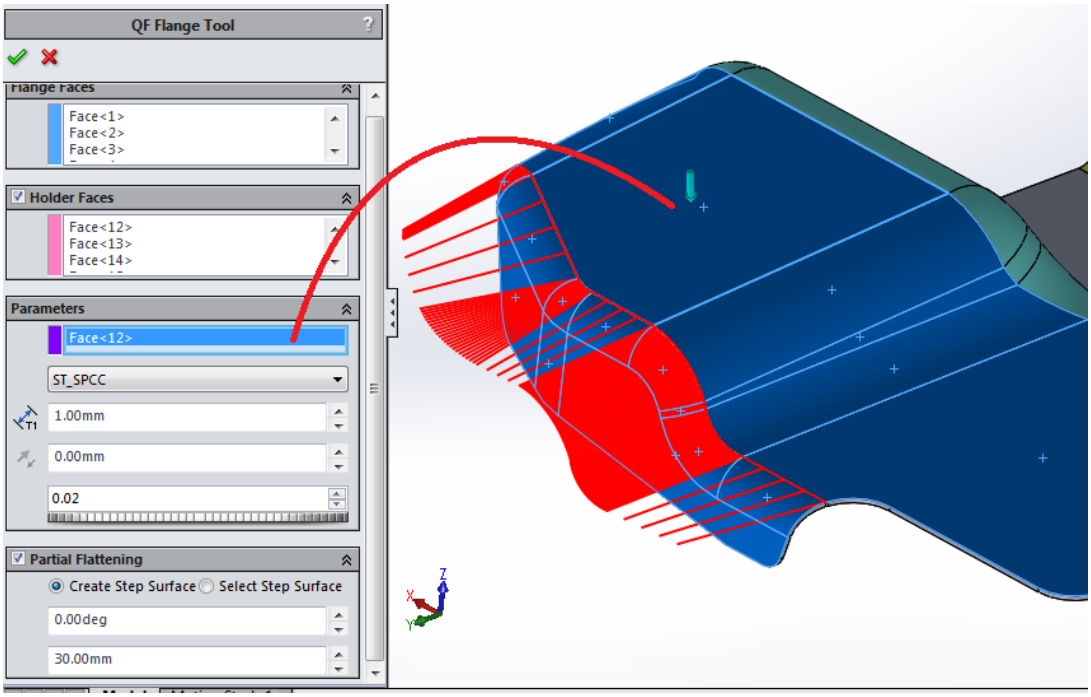
QF flange tool is a function to calculate the blank profile of forming area by using the 3DQuickForm Professional solver. Result blank profile will be created as a planar surface, which can be used to setup user defined feature. The QF Flange Tool is also capable to unfold flange area. Below procedures show the unfolding of a flange area with the QF Flange Tool.



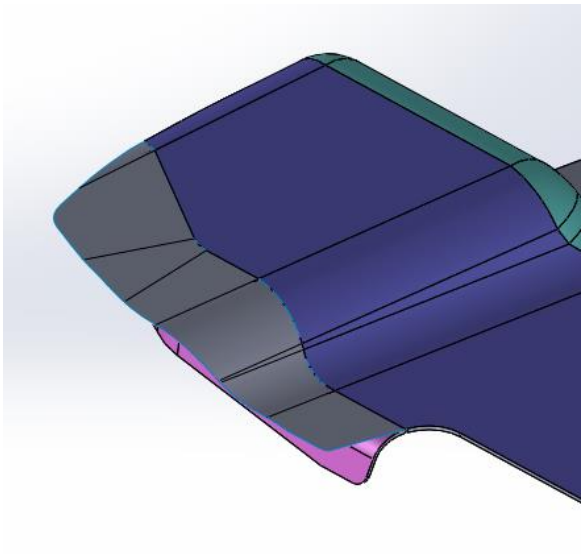
1. Click **QF Flange Tool**  on the **3DQuickPress Surface Tools Toolbar**
2. Select the faces in pink into the **Flange Faces**, and faces in blue into **Holder Faces**



3. Select the top face as the **Punching direction**



4. Enter 1mm for the **Material Thickness**
5. Enter 0.0deg as **Partial Flattening Angle** and 30mm as **Partial Flattening Extension**
6. Click **OK** to create the unfolded surface



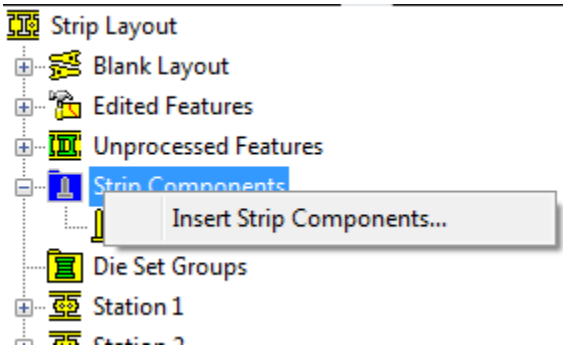
Strip Layout

Strip Components

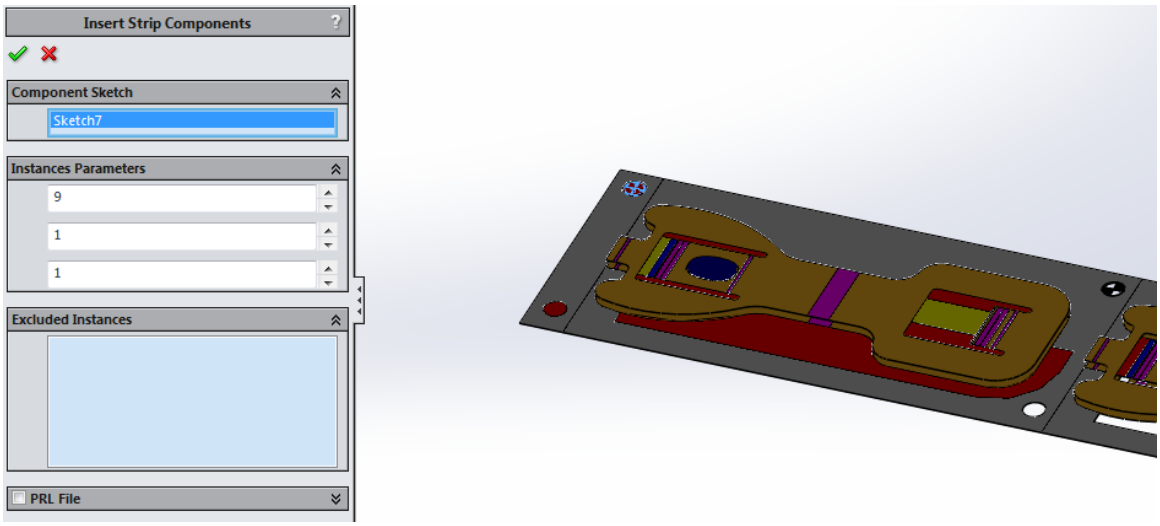
Strip components can be used to represent patterned features along the progressing direction with the spacing equal to the progression pitch, like pilots and lifters. A strip component feature will be attached with a sketch to represent the size and shape of this strip component.

To create a strip component feature and use it to create a pattern of pilot:

1. **RMB** on the **Strip Components** on the strip layout feature tree
2. Select **Insert Strip Components…**

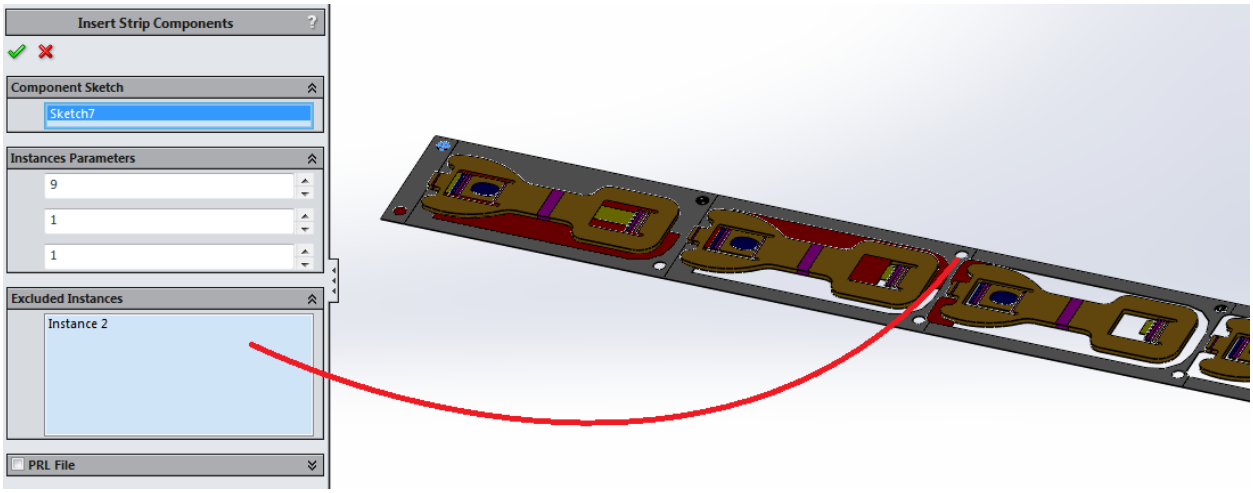


3. Select a predefined sketch as the **Component Sketch**

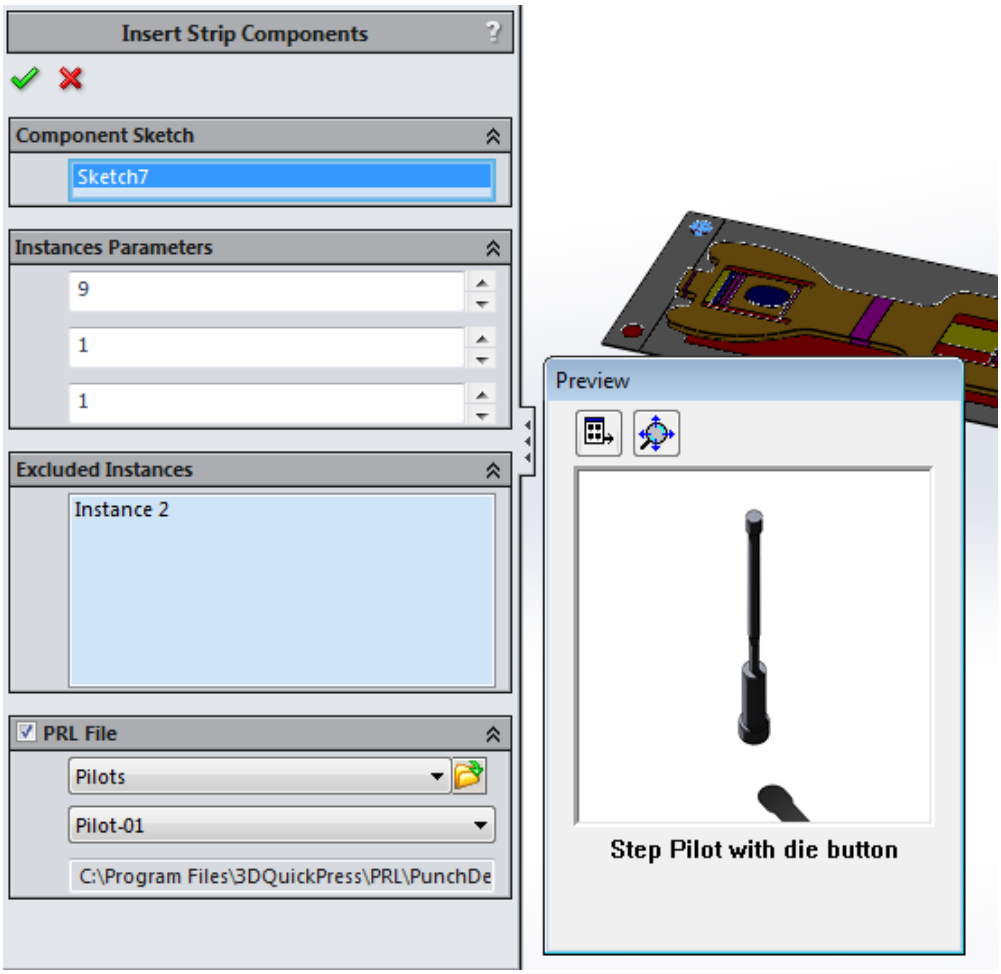


4. Enter 9 as the **Number of Instance** and 1 as the **Start Position**

5. Select the sketch display on station 3 as the **Instance to Skip**

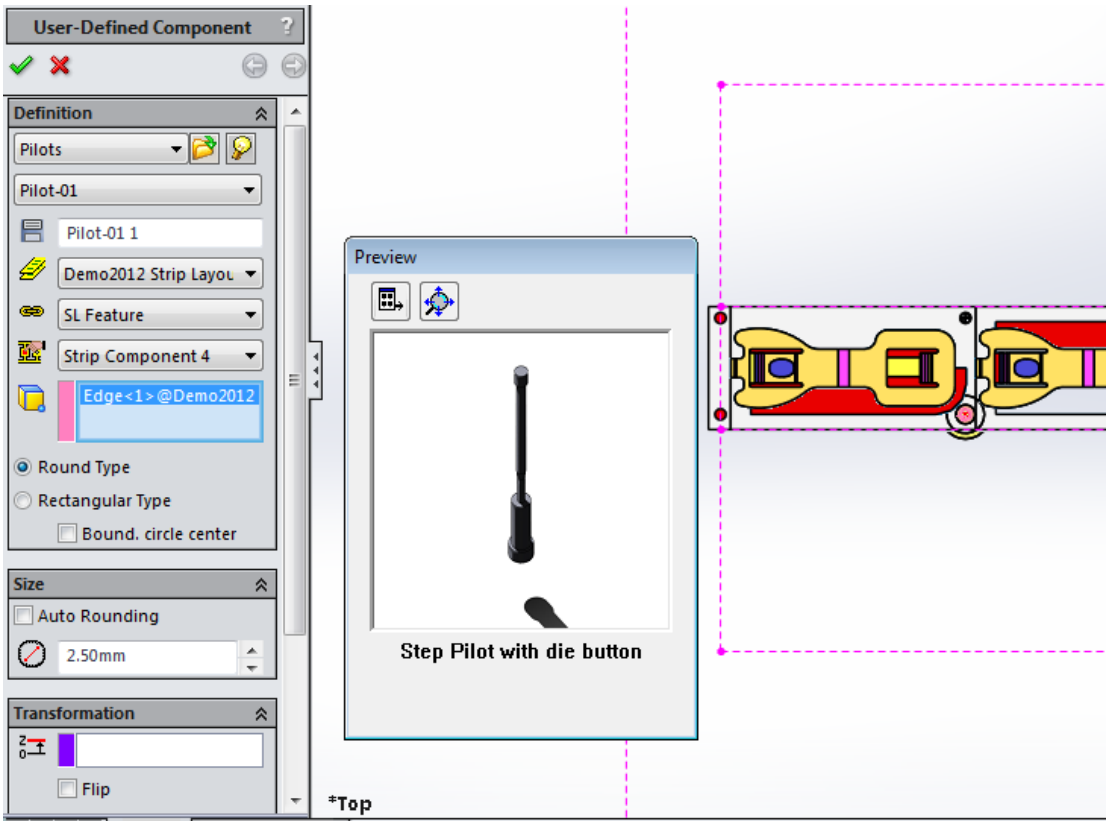


6. Check the **PRL File** option
7. Select the Pilot-01 template

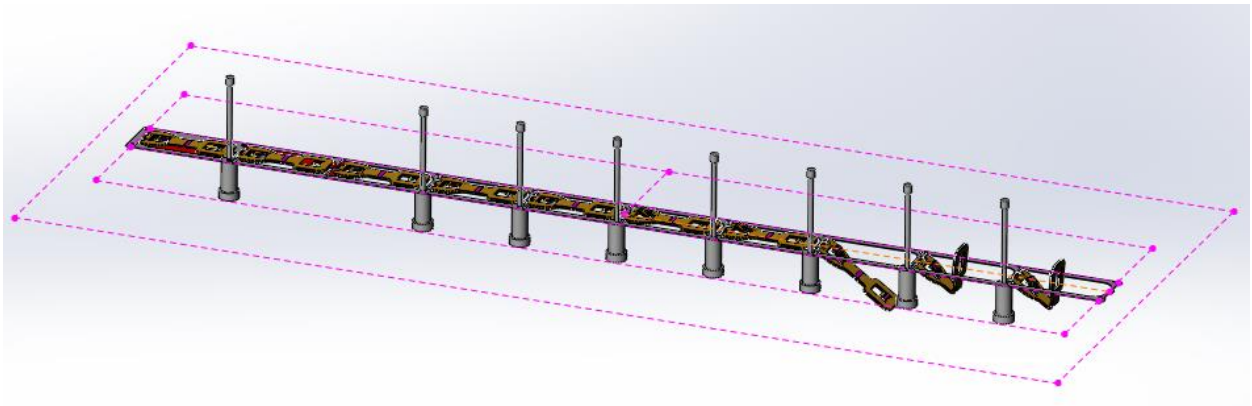


8. Click **OK** to create the strip component feature

9. Create or switch to punch design assembly
10. Click **Create UDC** on the **3DQuickPress PRL toolbar**
11. Select the sketch symbol of strip component 2



12. Click **OK** to create the pilot component pattern with the 2nd instance skipped

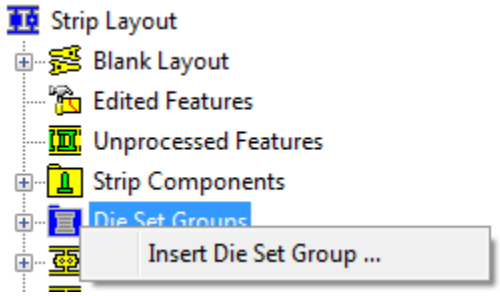


Die Set Groups

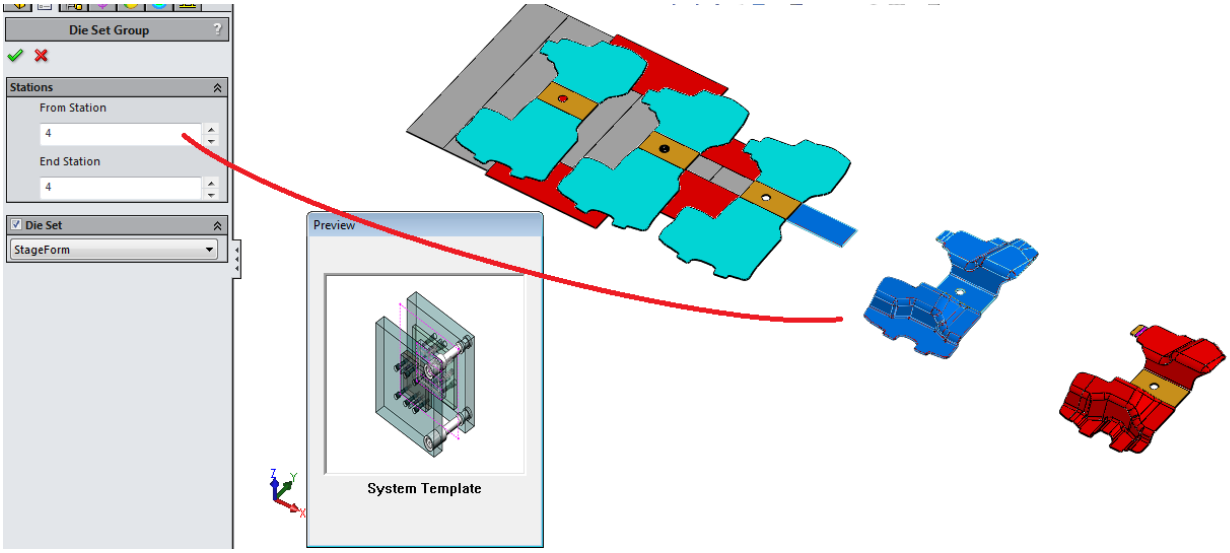
Die set group is used to define die set template for specified station/stations on a strip layout. Multiple die set template can be assigned to a single strip layout on different station/stations. This is particularly useful when defining stage tooling die sets.

To create a die set group for a stage tooling:

1. **RMB** on the **Die Set Groups** on the strip layout feature tree
2. Select **Insert Die Set Group…**

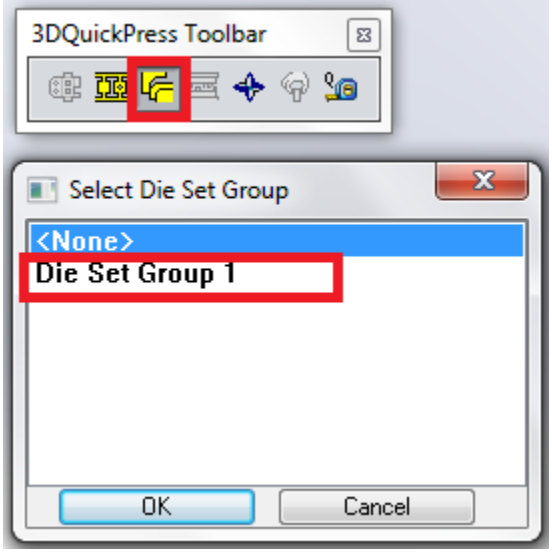


3. Select the 4th station
4. Check the **Die Set** option and select a die set template

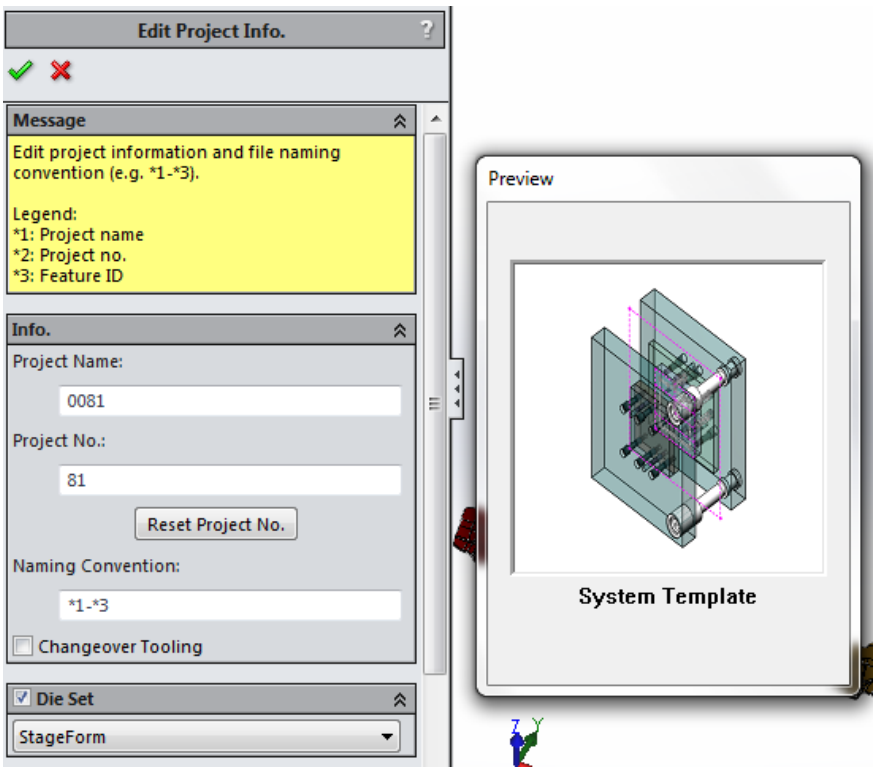


5. Click **OK** to define the **Die Set Group feature**

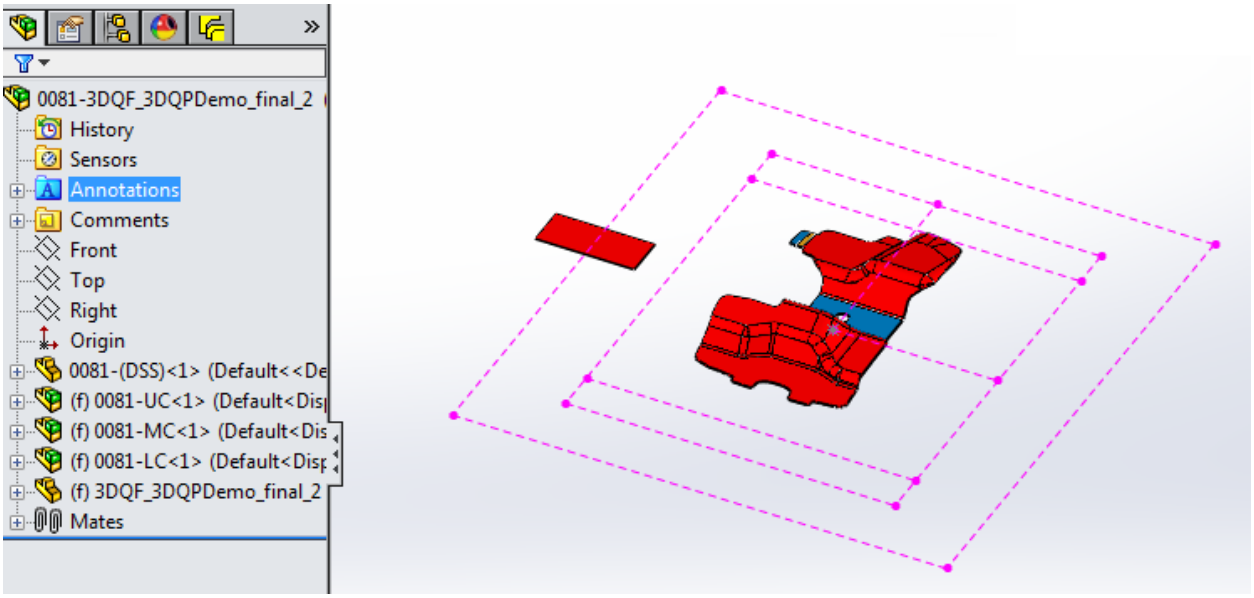
6. Click **Create Punch Design** on the **3DQuickPress toolbar**



7. Choose **Die Set Group 1** in the **Select Die Set Group** dialog
8. Click **OK**
9. Click **OK** on the **Create Punch Design** PMPage with the preset die set template



10. A punch design assembly showing the specified station will be created



Nesting

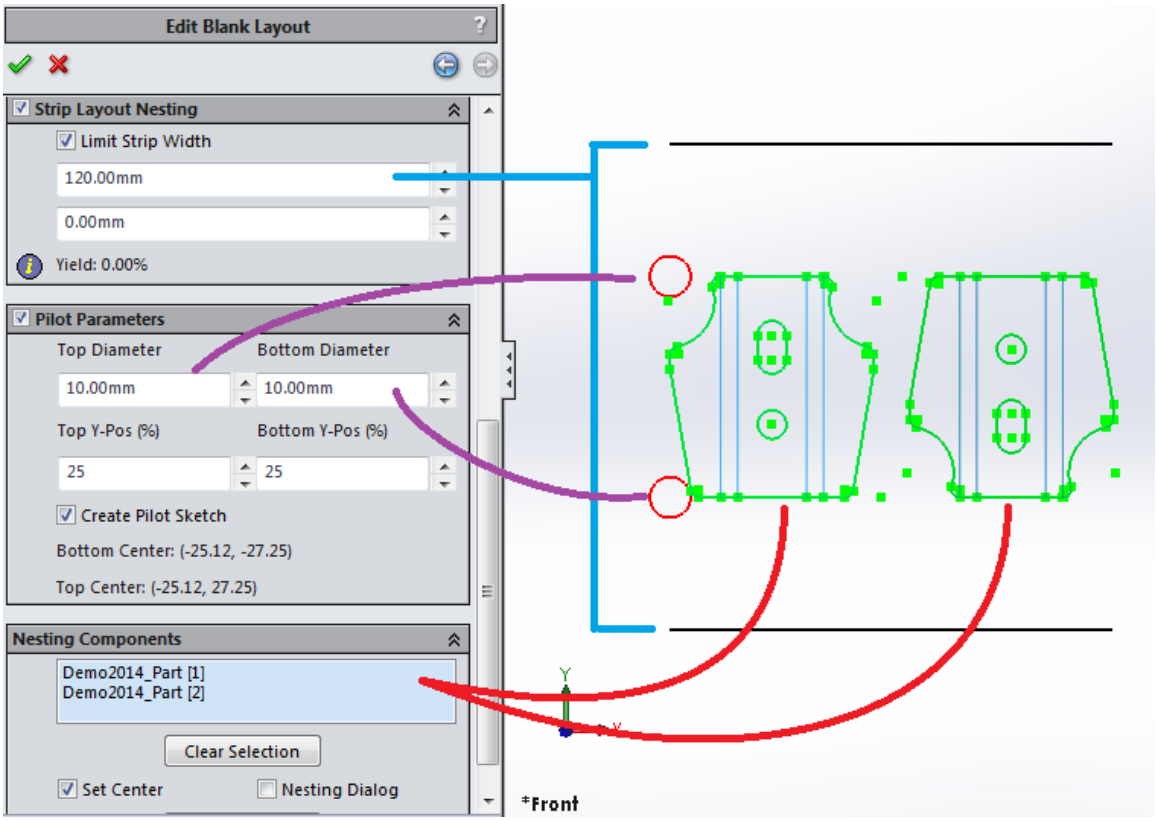
New features in nesting function:

1. Nesting of pilot location with blank parts
2. Nesting with stock width limit
3. Nesting with constraint on blank parts
4. Multiple nesting results

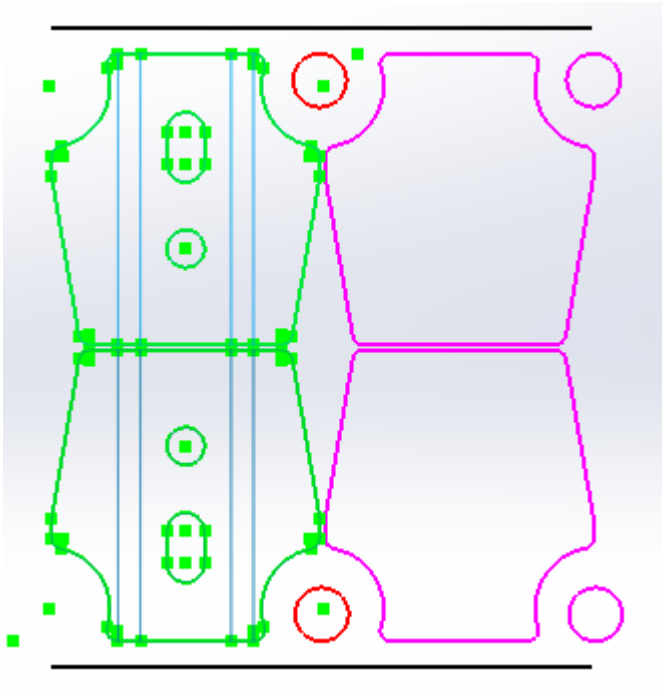
To nest 2 blanks with pilot and stock width limit:

1. **RMB** on the **Blank Layout** on the strip layout feature tree
2. Select **Edit Blank Layout…**
3. Click on the **right arrow** 
4. Check **Strip Layout Nesting**
5. Check **Limit Strip Width**
6. Enter 120mm as **Strip Width**
7. Check **Pilot Parameters**

8. Select the 2 blank part

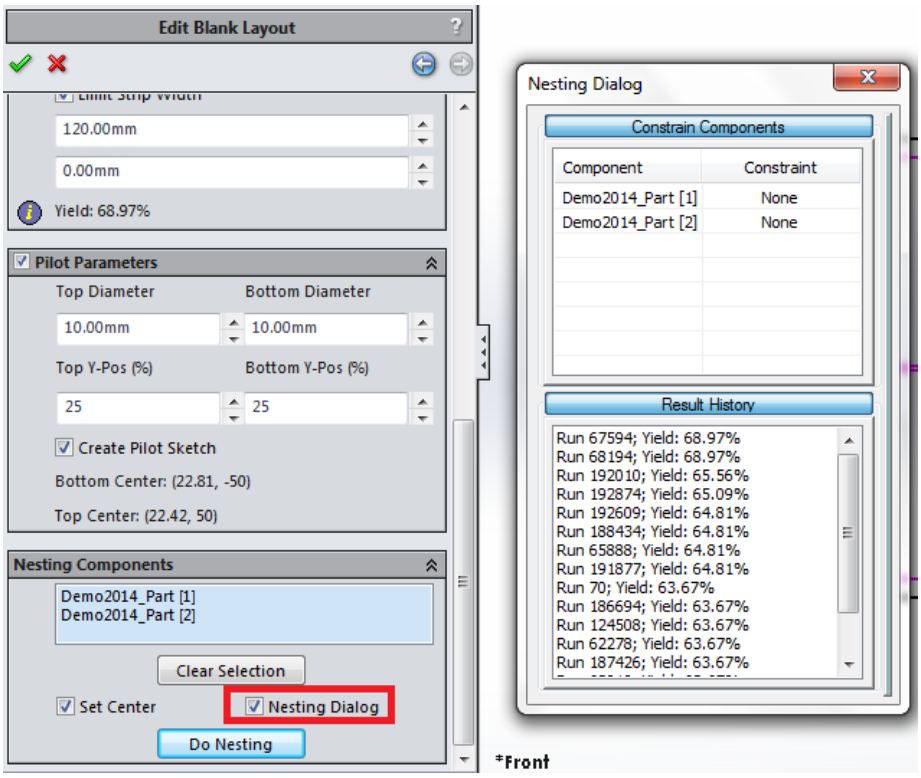


9. Click **Do Nesting** to start nesting

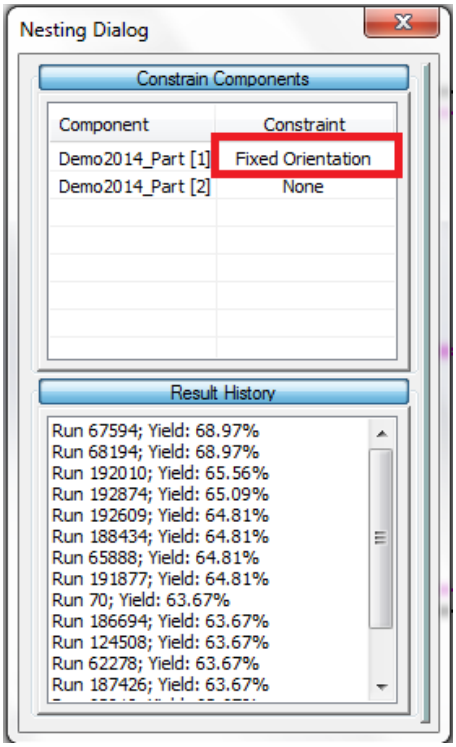


To add constraint to blank part and switch nesting result:

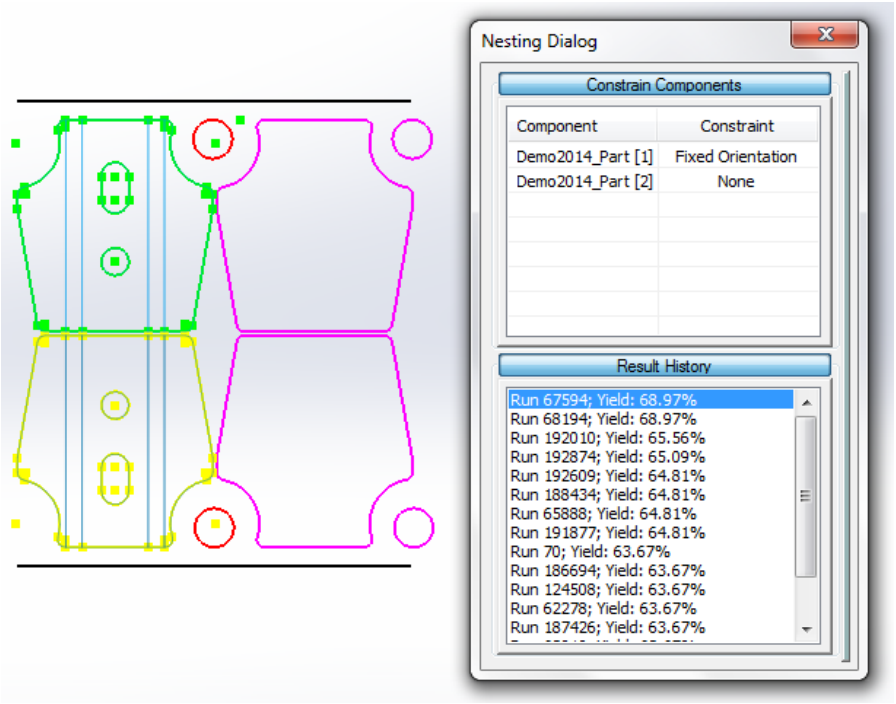
1. Check **Nesting Dialog**

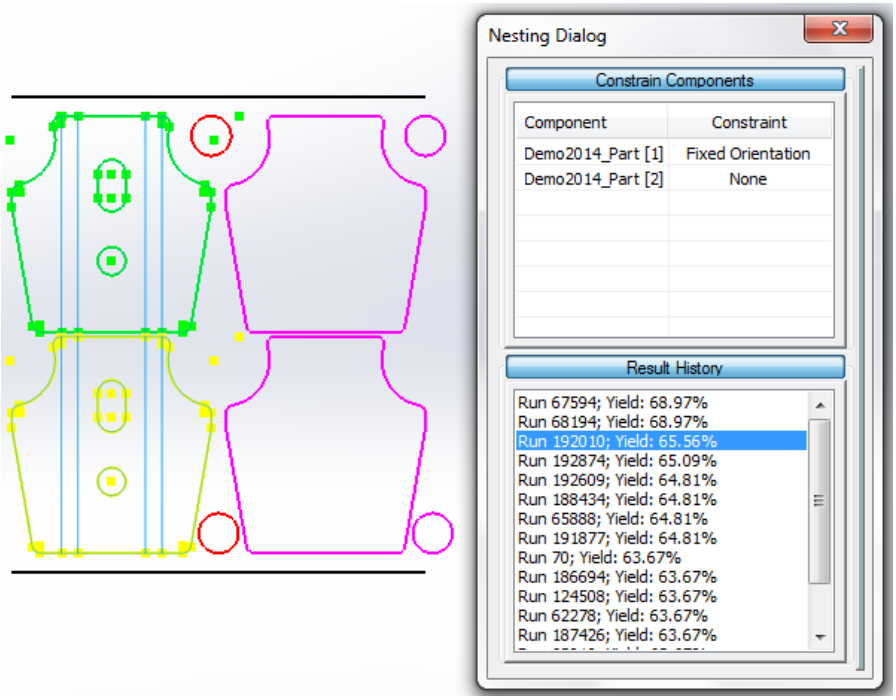


2. Click on **Constraint** of 1st blank part to fix it



3. Select the result in Result History to switch between different results





Overbend for UAR Bend

Overbend feature will support UAR bend in 3DQuickPress. The operation is the same as adding overbend to normal bend feature.

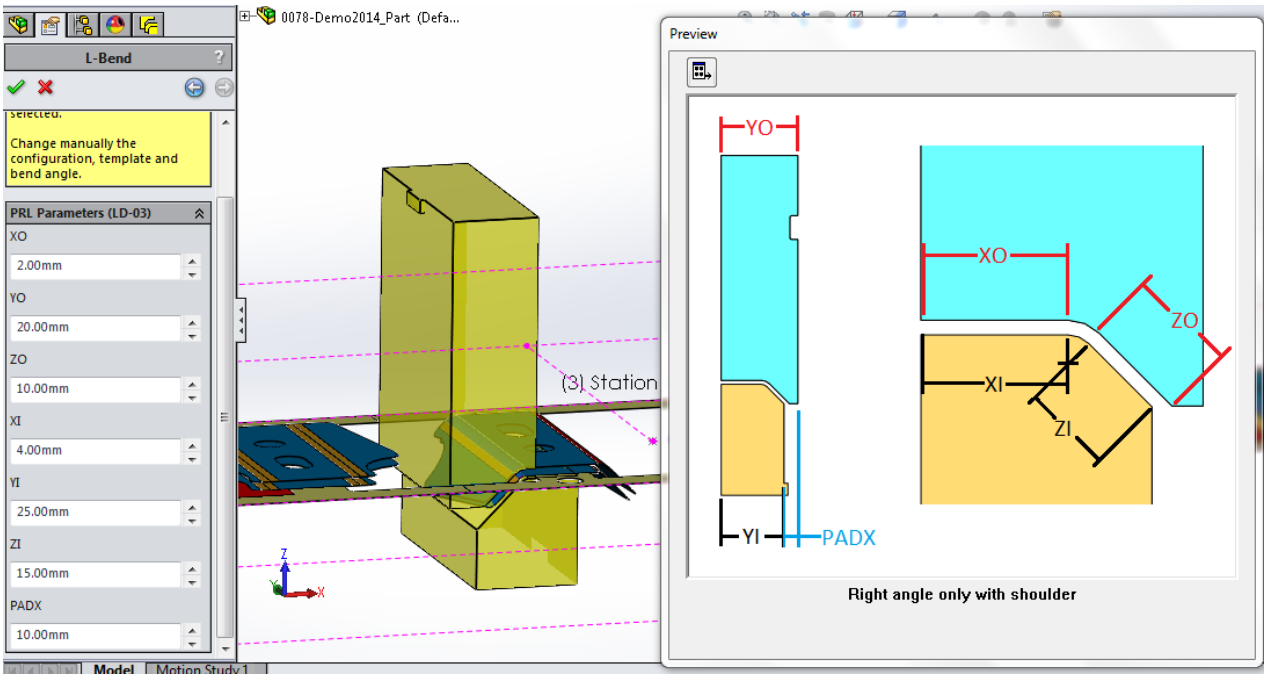
Punch Design

Bending Punch Preview

Preview of bending punch is added to 3DQuickPress. When create bending punch, a preview of the result bending punch component will be shown. Size of bending punch and die components can be adjusted by changing the parameters.

To preview and adjust the size of the bending punch before it is created:

1. Click **Create L Bend** in the **3DQuickPress PRL toolbar**
2. Select L bend feature in the strip layout.
3. Click on the **right arrow**
4. Select the parameters**, Xo, Xi, Yo, Yi, Zo** and **Zi,** and then set the new values, the definition of the parameters will be shown on the preview window



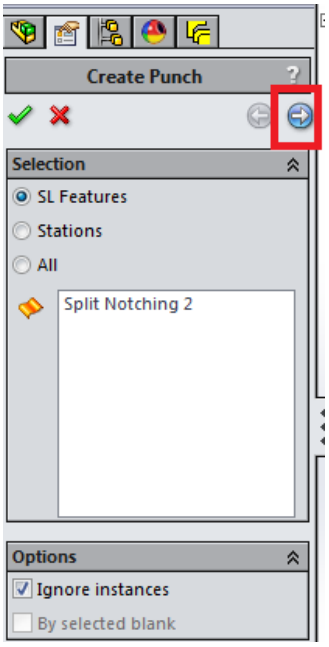
5. Click **OK** to create the bending punch as the preview displayed

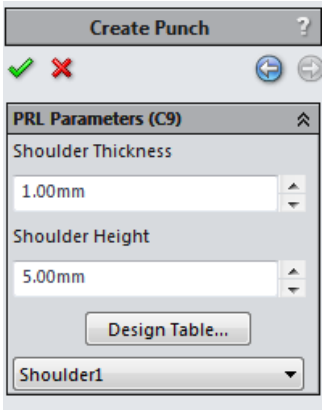
User Defined Parameters for Cutting and Bending Punches

User defined parameters are supported by cutting punches and bending punches template. User defined parameters can be introduced into the cutting punch and bending punch template with the “PRL Param.def” file or the PRL table as the standard PRL template of user defined component.

To access the user defined parameter when create cutting punch:

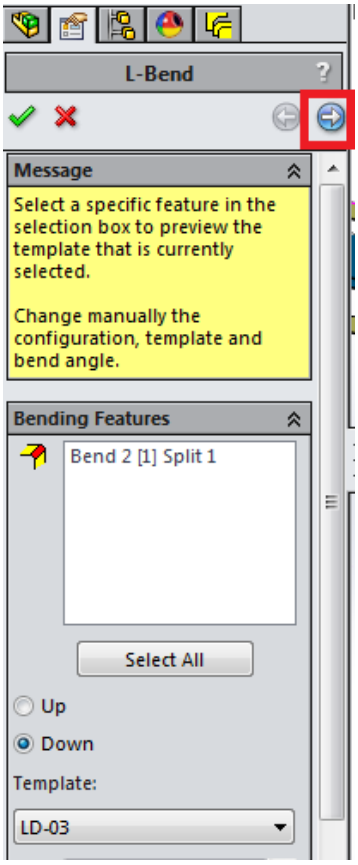
1. Click **Create Cutting Punch** on the **3DQucikPress PRL toolbar**
2. Select a cutting feature on the strip layout
3. If the cutting punch template contain the user defined parameters, the right arrow on the PMPage will be activated. Press the **right arrow** to access the user defined parameters

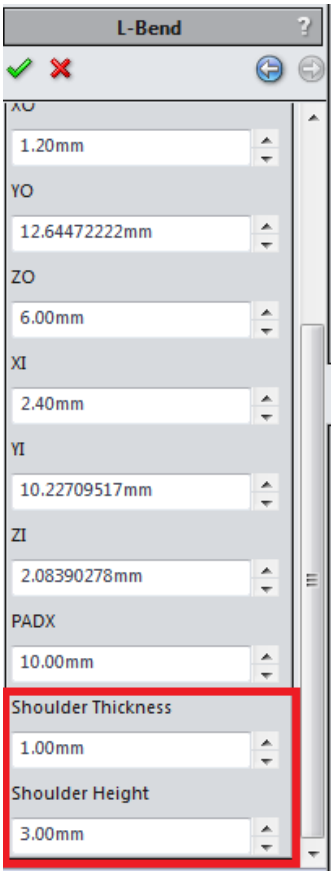




To access the user defined parameter when create cutting punch:

1. Click **Create L Bend** in the **3DQuickPress PRL toolbar**
2. Select L bend feature in the strip layout
3. Select a L bend template in Template list box
4. If the bending punch template contain the user defined parameters, the right arrow on the PMPage will be activated. Press the **right arrow** to access the user defined parameters





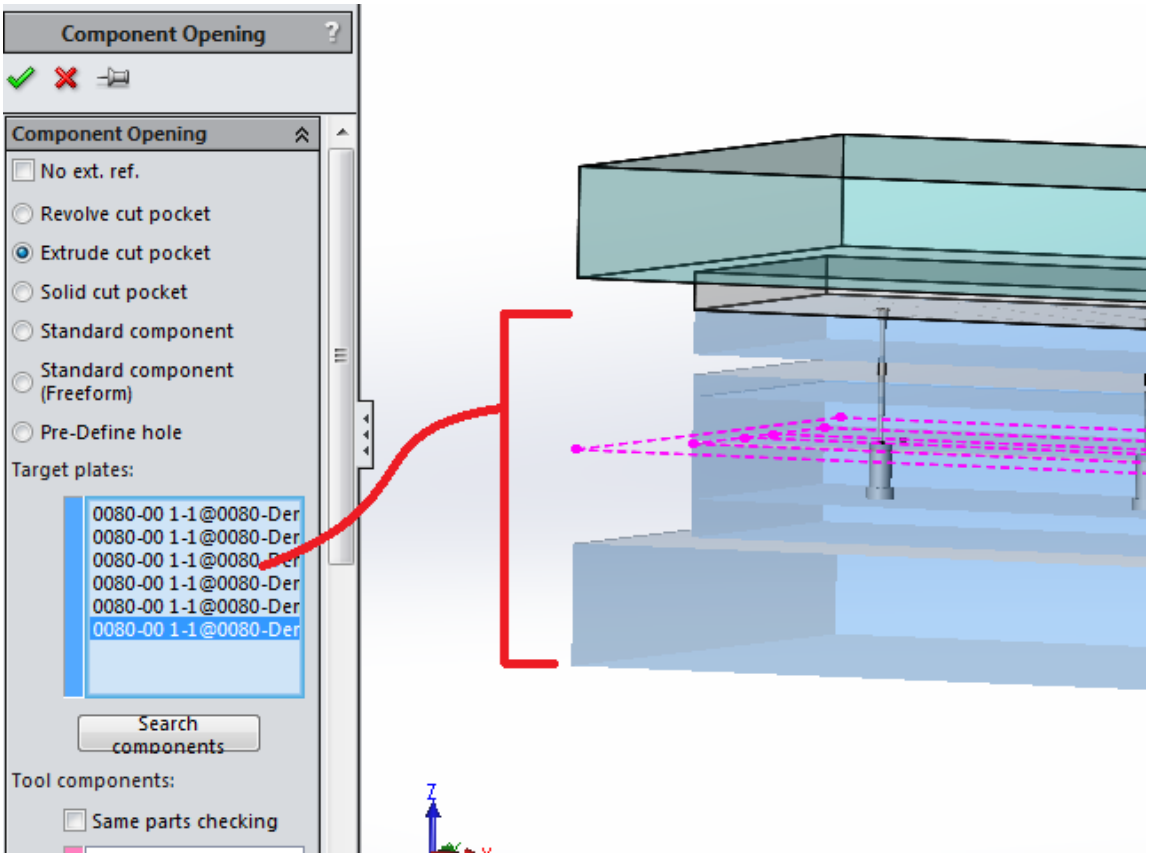
Die Set Design

Component Opening for Auto Extrude and Revolve Cuts

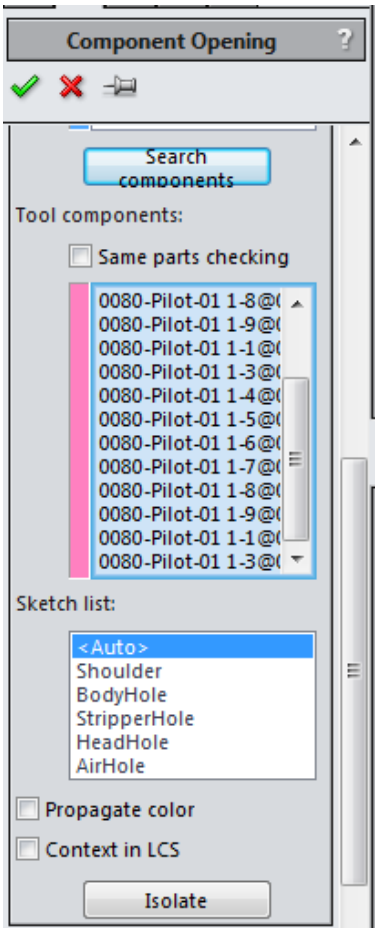
The extrude and revolve cut options in component opening function can identify the target component automatically according to the hole sketch definition. If the hole sketch defined intersects with the a target component, then the cutting hole will be created automatically without manually selecting the corresponding hole sketch.

To create cutting hole with auto extrude cuts:

1. Click **Component Opening** on the **3DQuickPress Misc. Tools Toolbar**
2. Select option **Extrude cut pocket**
3. Select all die plate components that requires cutting holes into **Target Plates**



4. Click **Search components** to search for all interfered components
5. Select **Auto** in **Sketch list**



6. Click **OK** to create all extrude cut holes

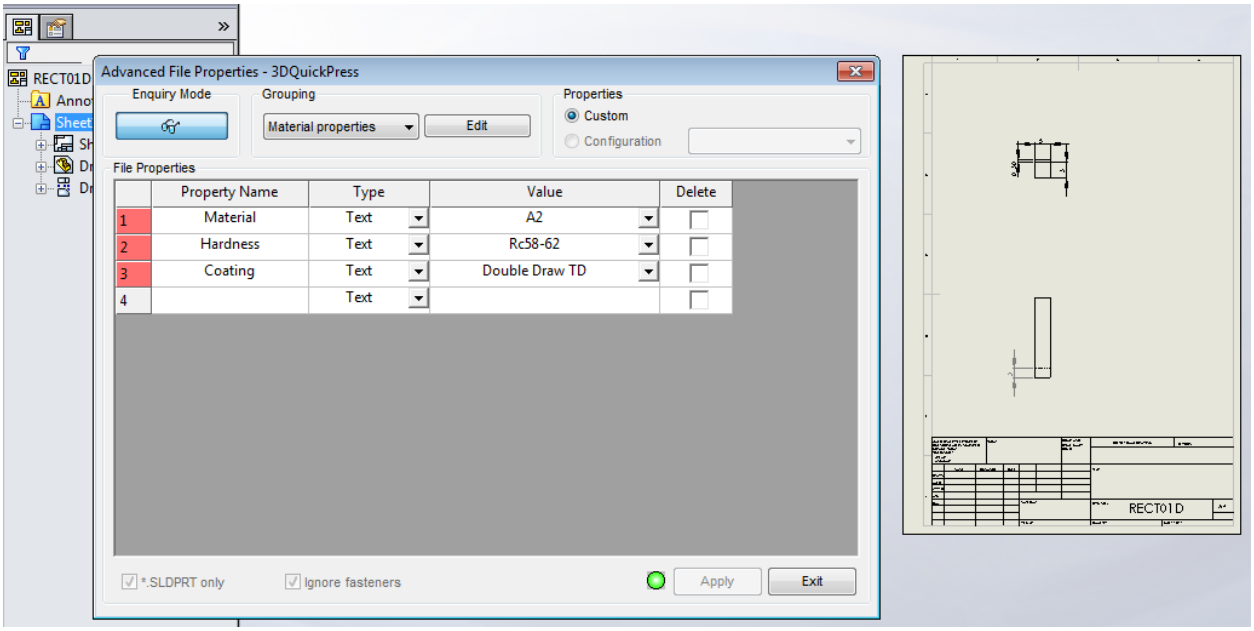
Standard Round Punch supports auto extrude cut in component opening

Standard round punch inserted from the standard catalog supports the using of auto extrude cut options in component opening function.

Detailing

Using of Advance File Property with Drawing Documents

Advance File Property function can be used to edit the file property of drawing documents.



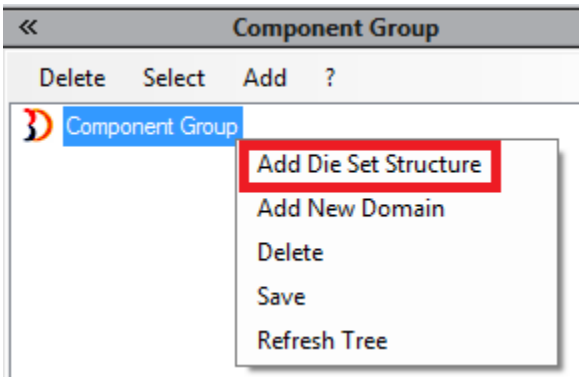
Utilities

Component Group and Animation Tool

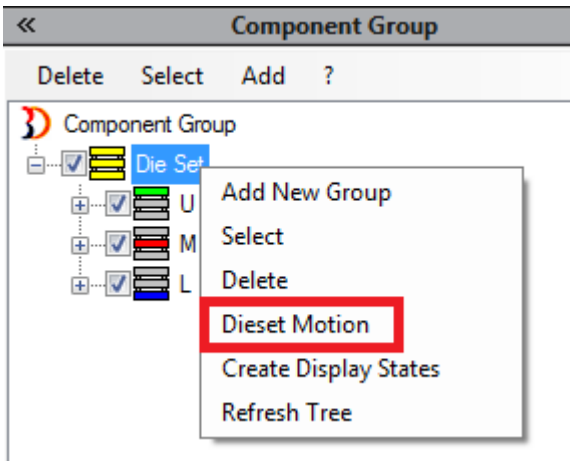
Component group is a function to group up the components in a die set assembly. It also provides tools to animate the grouped components according to die set structure to simulate the motion of the stamping process. Also, during the animation of the motion, interference of the components in the die set can be detected.

To setup a component group according to the die set structure:

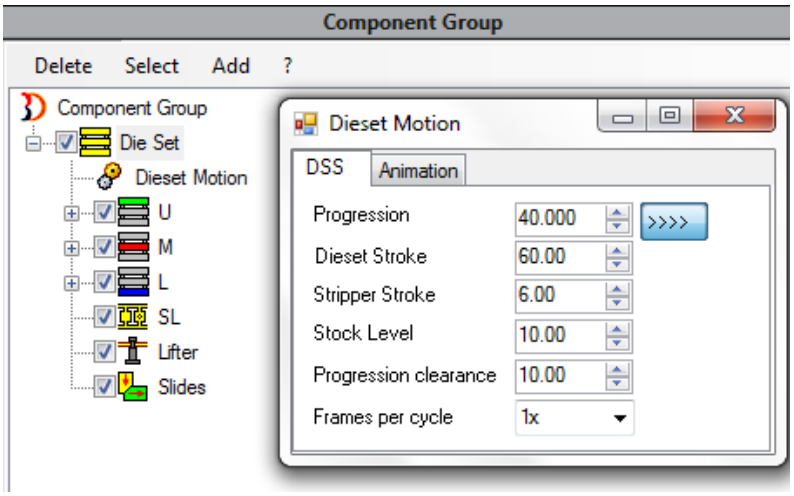
1. Click **Toggle Component Group** on the **3DQT DieSet Design Toolbar**
2. RMB on **Component Group** icon and select **Add Die Set Structure**



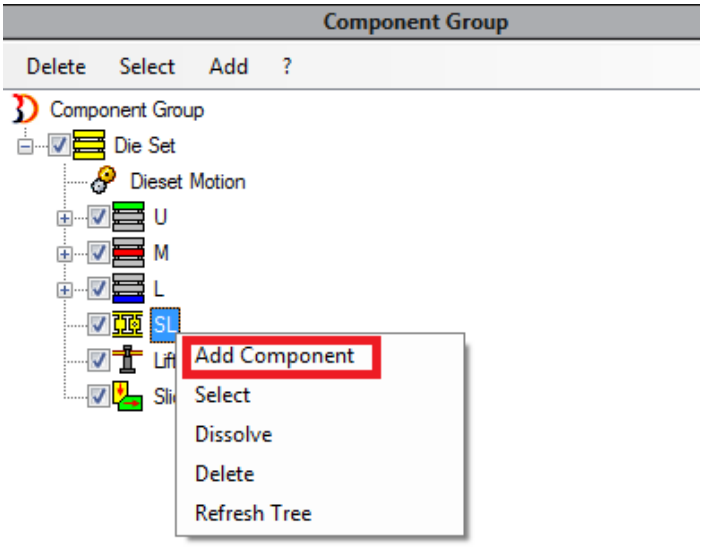
3. RMB on **Die Set** group and select **Dieset Motion**



4. Enter the values for the parameters **Progression, Dieset Stroke, Stripper Stroke, Stock Level** and **Progression clearance**



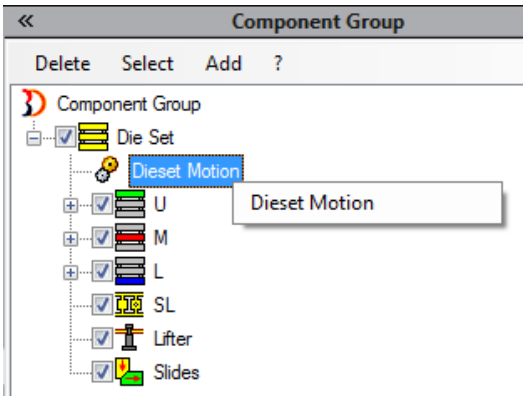
5. Click **Close**  to close the **Dieset Motion** dialog
6. Select the strip layout part
7. RMB on the **SL** group and select **Add Component** to add the strip layout to the SL group



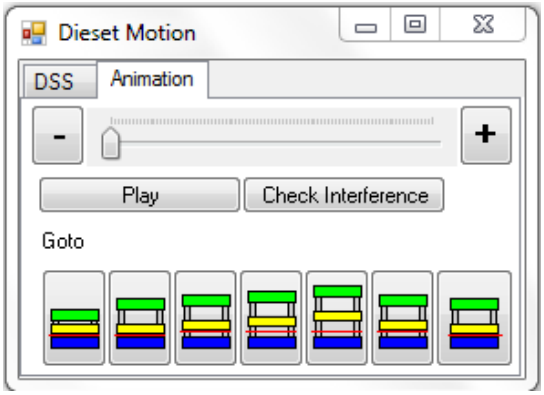
8. Select all lifter component in the die set
9. RMB on the **Lifter** group and select **Add Component**

To run the die set animation and interference detection:

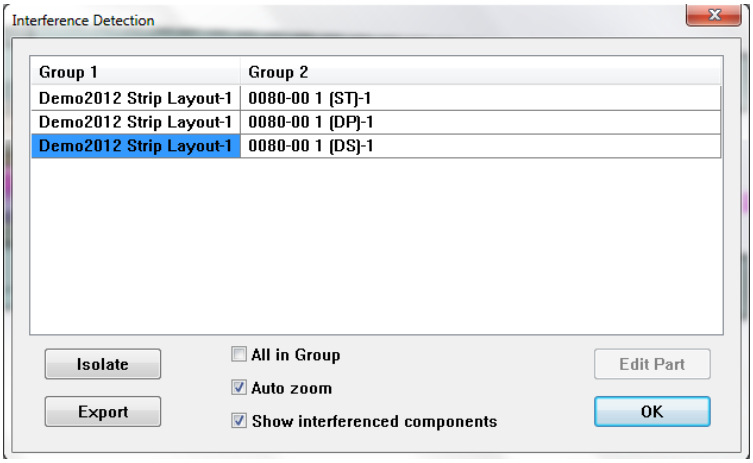
1. RMB on **Dieset Motion** and select **Dieset Motion**



2. Click **Animation** tab



3. Click **Play** to run simulation
4. Click **Stop** to pause the simulation
5. Click **Check Interference** to check interference at the paused state
6. Select the interference group to show the interference area



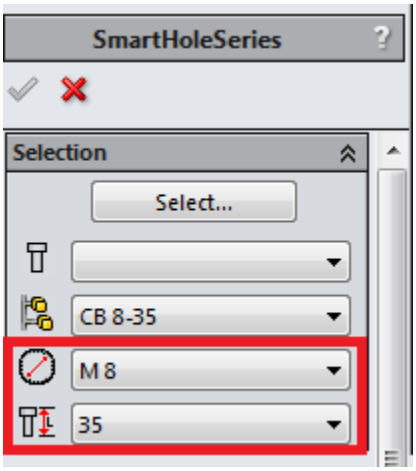
7. Click **OK** to close the dialog

New Standard Component

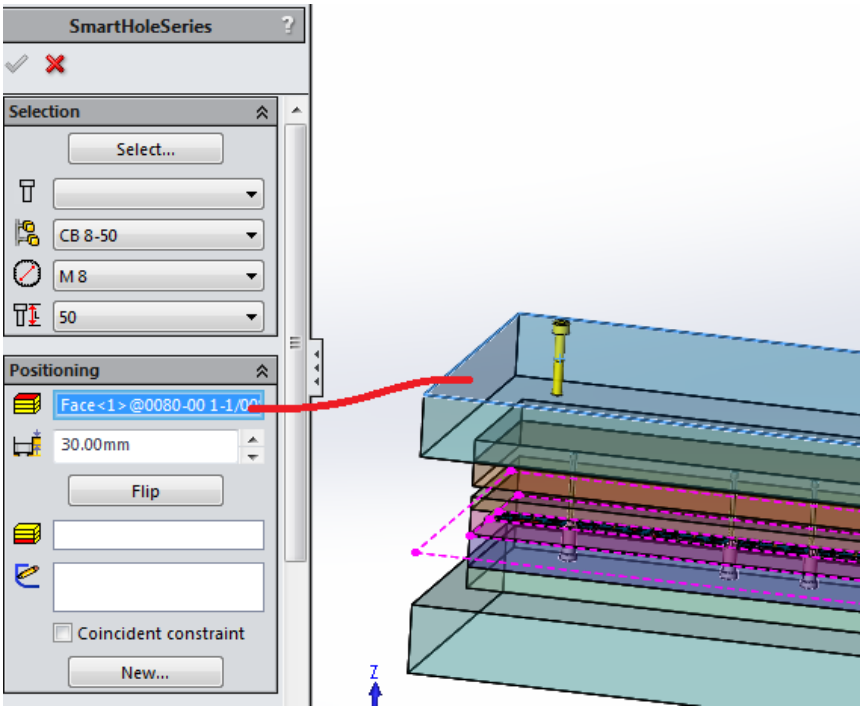
A new setup method for standard component is introduced in 3DQuickPress. To define a standard component, pre-defined hole definition is no longer required. In the new standard component definition, the hole definition is defined only by the hole sketch in the component, and the size of the component is controlled by the SolidWorks design table. A series of new functions are implemented to incorporate with the new standard component. These functions are Insert Component, Edit Component Size, Edit Layout Sketch, Edit Hole Size, Add Hole, Delete Hole and Rebuild Holes.

To insert a new standard component:

1. Click **Insert Component** on the **3DQT DieSet Design Toolbar**
2. Click **Select** and browse for the standard component to be inserted (e.g. \\3DQuickPress\Standard Components\Misumi\Socket Head Screw\CBxx-xxx.SLDPRT)
3. Click **Open** to select the component
4. Select the component size and length from the listbox



5. Select the top surface as the position to insert the component



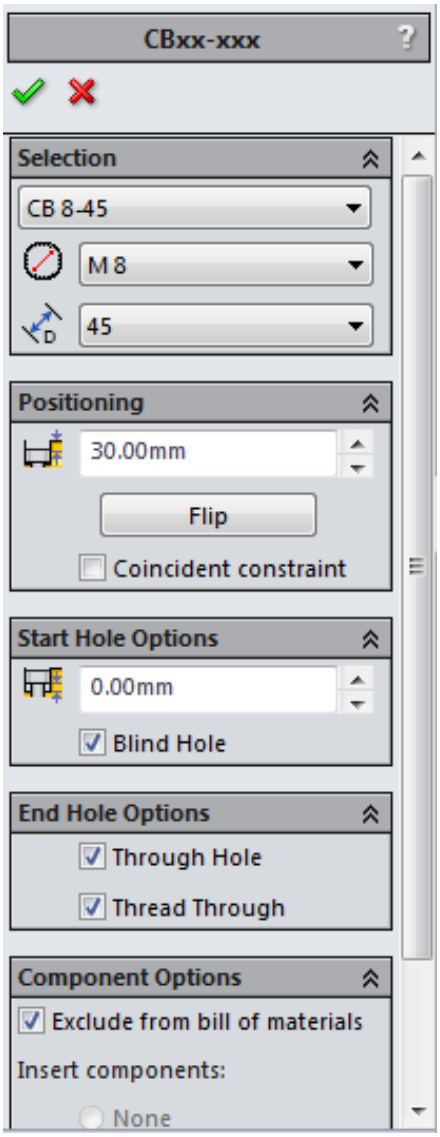
6. Click **New…** to create a new layout sketch to position the component
7. Click **Point** on the SolidWorks **Sketch Toolbar**
8. Draw points on the location where the component will be inserted
9. Click **Exit Sketch** to close the sketch
10. Click **OK** to insert the component and create the component holes on the intersected die plates

To edit the position of the component by editing layout sketch:

1. Select a standard component inserted
2. Click **Edit Layout Sketch** on the **3DQT DieSet Design Toolbar**
3. Edit the point location in the layout sketch
4. Click **Exit Sketch** to close the sketch and update the component position

To change the size of the component:

1. Select a standard component inserted
2. Click **Edit Component Size** on the **3DQT DieSet Design Toolbar**
3. Change the component size by selecting the size and length in the list box, offset position and thread options can also be changed here



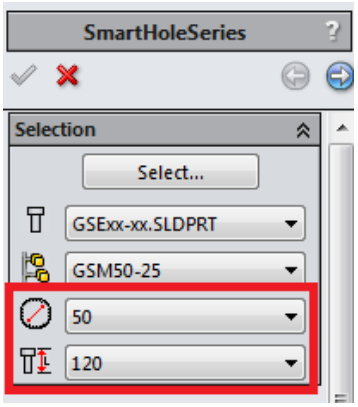
4. Click **OK** to update the component size and the component holes on the die plates

Insert standard gas spring with screws

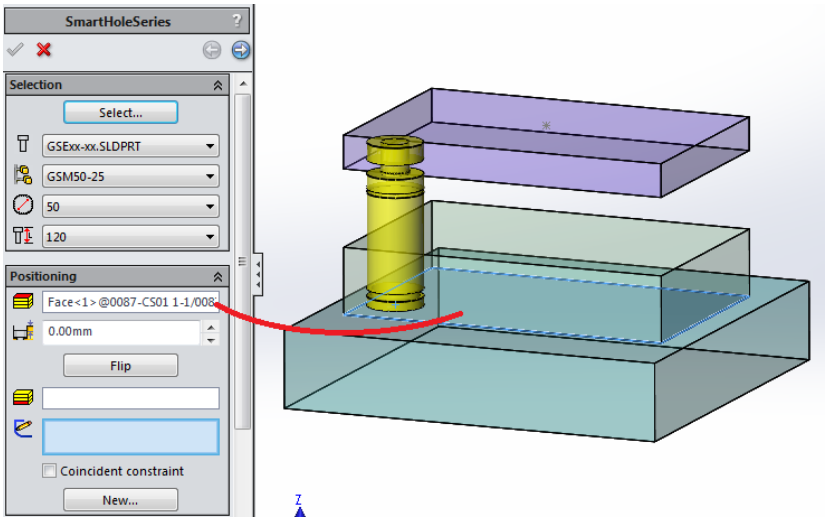
The new standard component method supports the inserting of standard gas spring with screw fasteners. The number of screws inserted will follow the definition in a gas spring. Changing the configuration of the gas spring will change the number and size of the screws inserted.

To insert gas spring components with screws to an assembly:

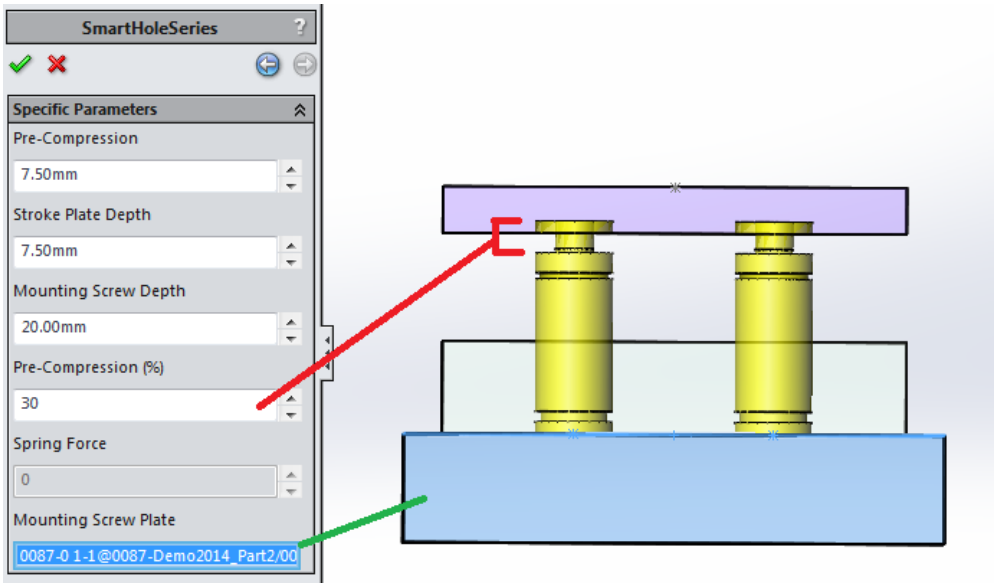
1. Click **Insert Component** on the **3DQT DieSet Design Toolbar**
2. Click **Select** and browse for the standard component to be inserted (e.g. \\3DQuickPress\Standard Components\Misumi\Gas Spring\GSExx-xx.SLDPRT)
3. Click **Open** to select the component
4. Select the component size and length from the listbox



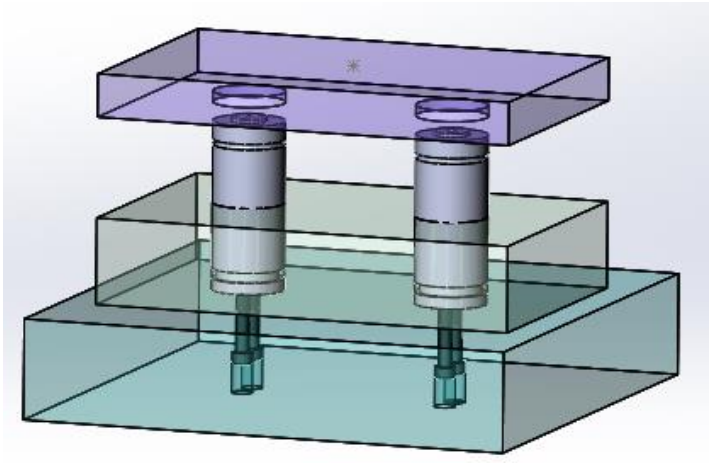
5. Select the bottom surface as the position to insert the component



6. Click **New…** to create a new layout sketch to position the component
7. Click **Point** on the SolidWorks **Sketch Toolbar**
8. Draw points on the location where the component will be inserted
9. Click **Exit Sketch** to close the sketch
10. Click **Next** to turn to the parameter page for inserting gas spring
11. Enter the pre-compression amount of the gas spring



12. Select the plate for inserting the mounting screw and the mounting screw offset
13. Click **OK** to insert the component and create the component holes on the intersected die plates

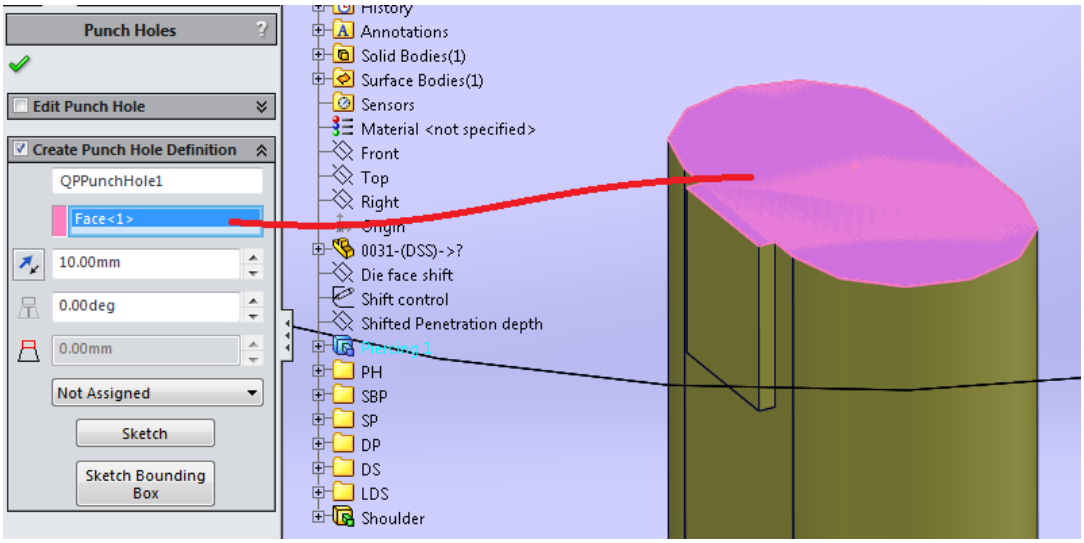


Hole Profile Manager

Hole profile manager is a tools to setup the extrude cut hole definition in a component. The extrude cut hole profile and revolve cut profile defined can be used by the component opening function to cut the extrude cut holes.

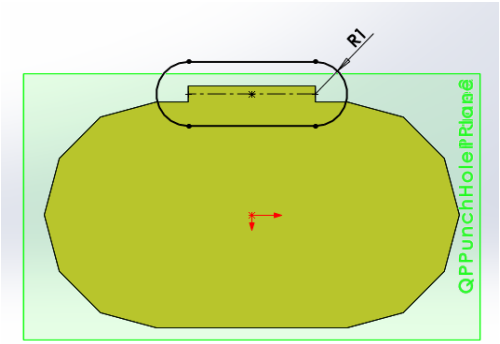
To define new extrude cut hole definition in a cutting punch:

1. Open a cutting punch component
2. Click **Hole Profile Manager** on the **3DQT DieSet Design Toolbar**
3. Check **Create Punch Hole Definition**
4. Enter the name of the cutting hole
5. Select the top face as the hole starting reference



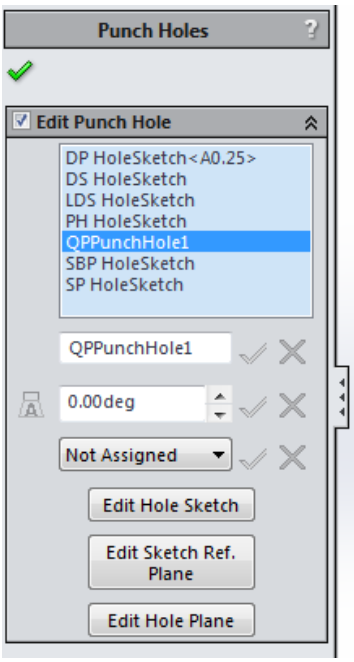
6. Enter the 5mm for the **punch hole depth**
7. Click **Sketch** to enter the hole profile sketch
8. Click **Straight Slot** on the SolidWorks **Sketch toolbar**

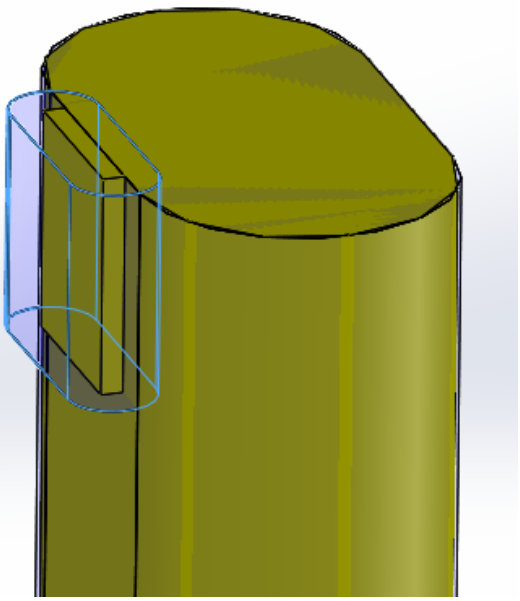
9. Create a slot as shown below



10. Click **Exit Sketch** 

11. Click **OK**  to create the hole definition



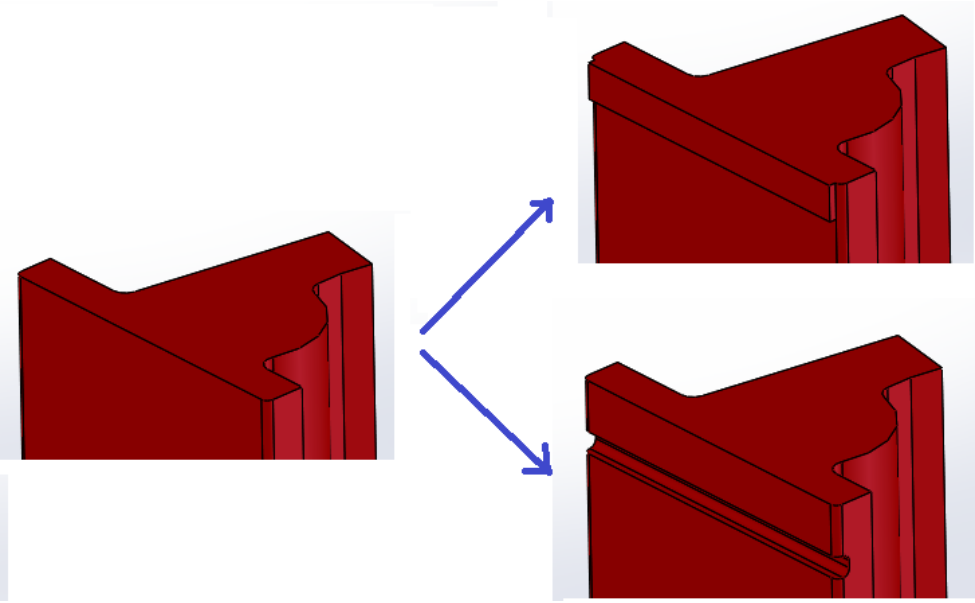


Tools for Shoulder, Circular Shoulder, Slot and Heel

New tools for punch detailing were added in 3DQuickPress. Punch features like shoulder, circular shoulder, slot and heel can be added to the punch components with these new tools.

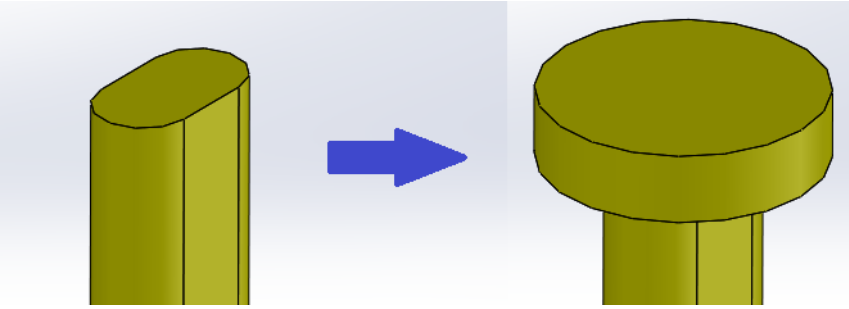
Shoulder/Slot Tool

Shoulder/Slot Tool is used to add shoulder and slot feature on the selected edge of a punch or die component. User can adjust the size of the shoulder and slot with the preview before creating the feature.



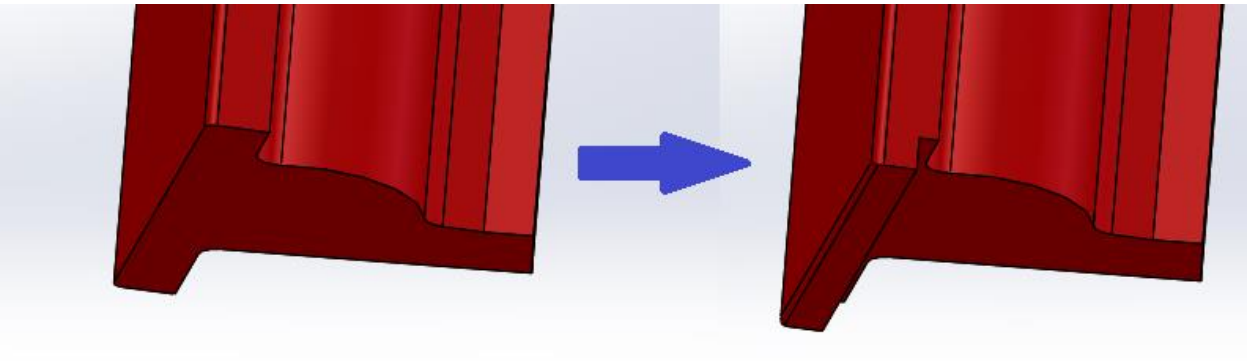
Circular Shoulder Tool

Circular Shoulder Tool is used to add a circular shoulder or a circular step on a selected circular edge on a rounded punch or die component. User can adjust the size of the shoulder with the preview before creating the feature.



Heal Tool

Heel Tool is used to add a heel feature on a selected edge of a punch or die component. User can adjust the size of the heel with the preview before creating the feature.

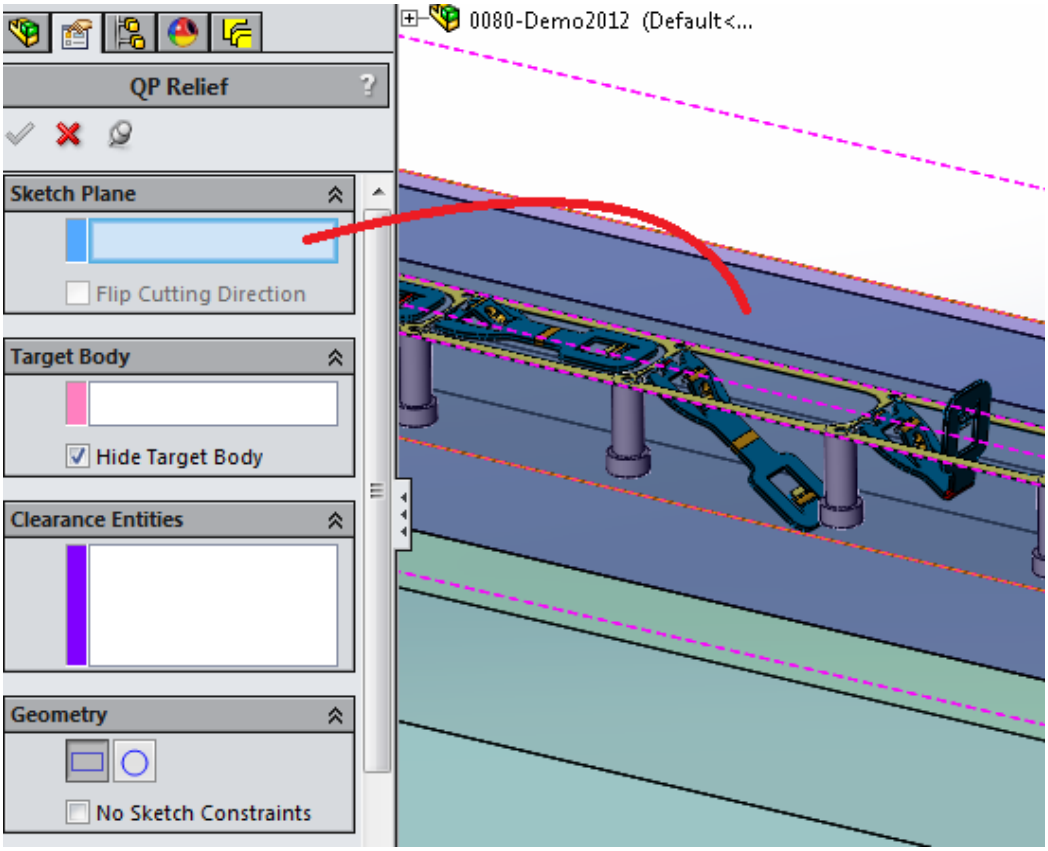


Clearance tool

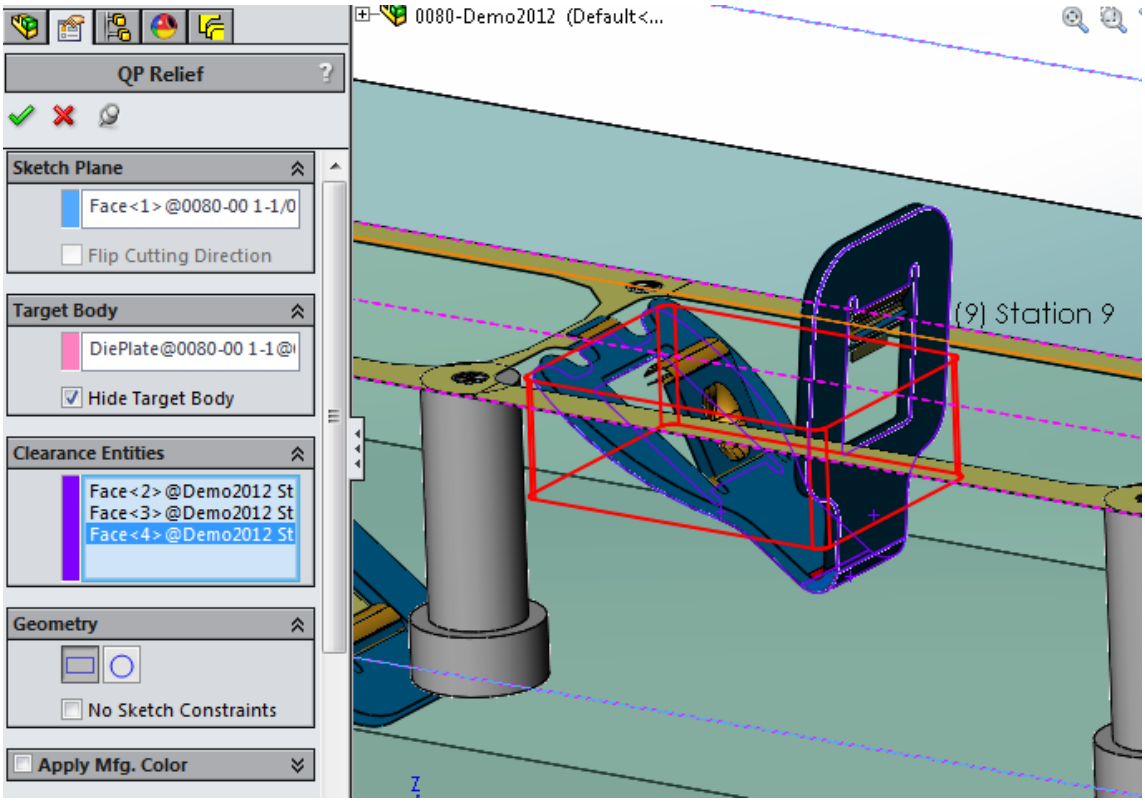
Clearance tool was added to create relief cut and clearance holes for selected entities. This tool is used mainly for handling relief cuts for interference.

To create a clearance pocket using the clearance tool:

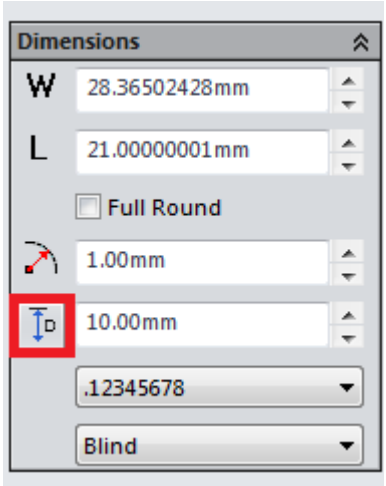
1. Click **Clearance Tools** on the **3DQP DieSet Design Toolbar**
2. Select the top plate of the die plate as the **Sketch Plane**, and the die plate will be selected as target automatically



3. Select the surface highlighted as the **Clearance Entities**, and a preview of the clearance pocket will be shown

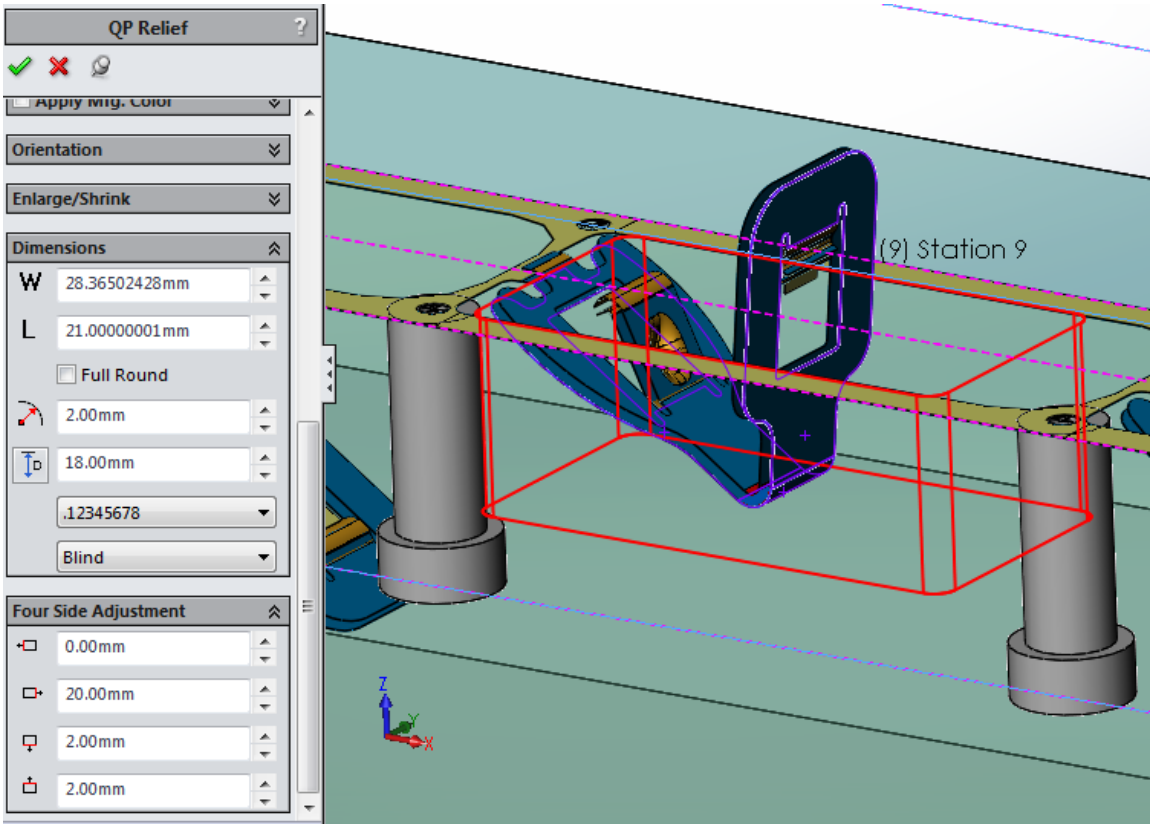


4. In the **Dimensions** tab, enter 2mm as the **radius for auto fillet**
5. Click **Evaluate Depth** button to find the depth of the clearance pocket, and round up the depth as 18mm



6. Expand the **Four Side Adjustment** tab

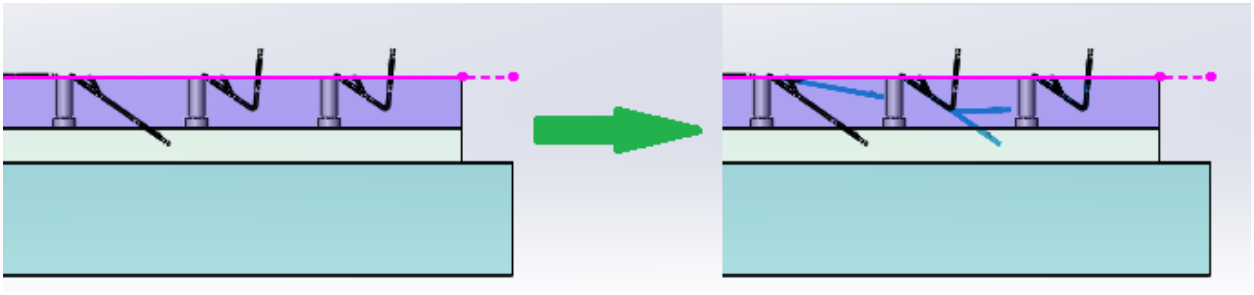
7. Adjust the sides offset as below



8. Click **OK** to create the clearance pocket

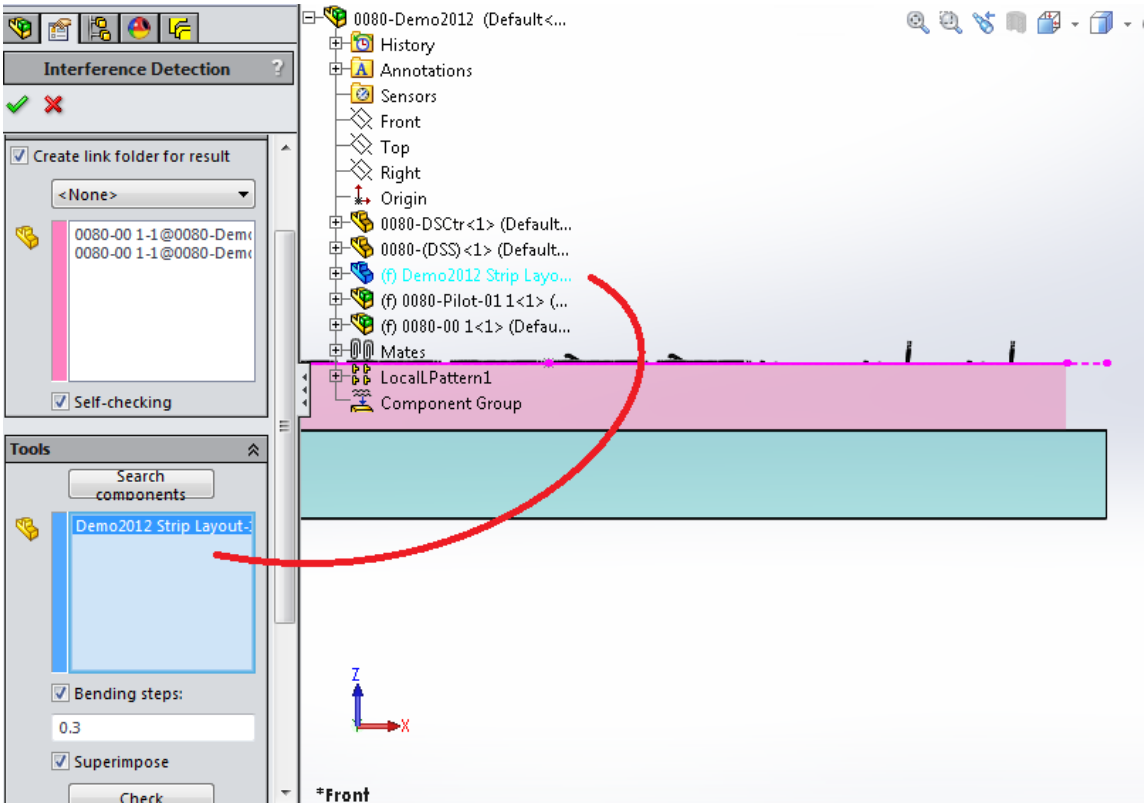
Bend feature intermediate steps for interference detection

Interference detection tool was enhanced to show the intermediate steps of a bending feature. User can show multiple steps of a bending feature during the process of it forms for interference check.

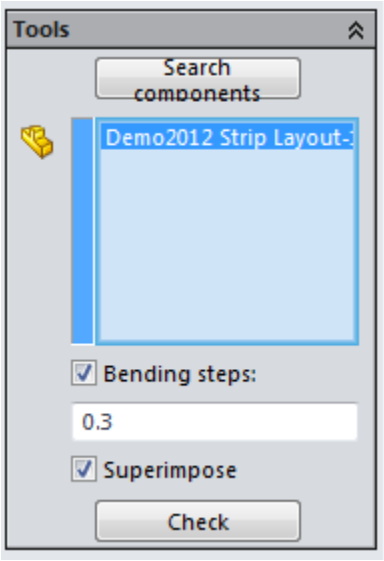


To check interference of a bending feature with die plate:

1. Click **Interference Detection** on the **3DQP Misc. Tool Toolbar**
2. Select the die plates that needs to check interference into the **Target** field



3. Select the strip layout part into the **Tools** field from the assembly feature tree
4. Turn on the Bending Steps option and enter 0.3 as step (0 < step valve < 1.0, e.g. 0.3 is one third of the full bend angle)



5. Click **OK** to do interference detection and output the intersected surface with intermediate steps of bending features