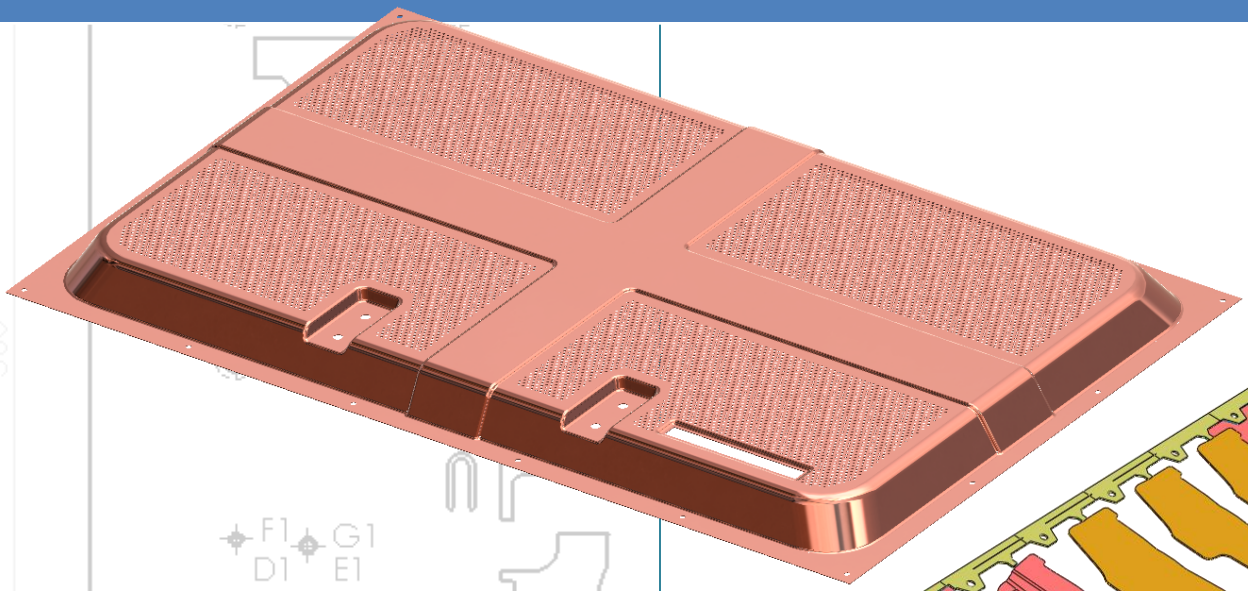
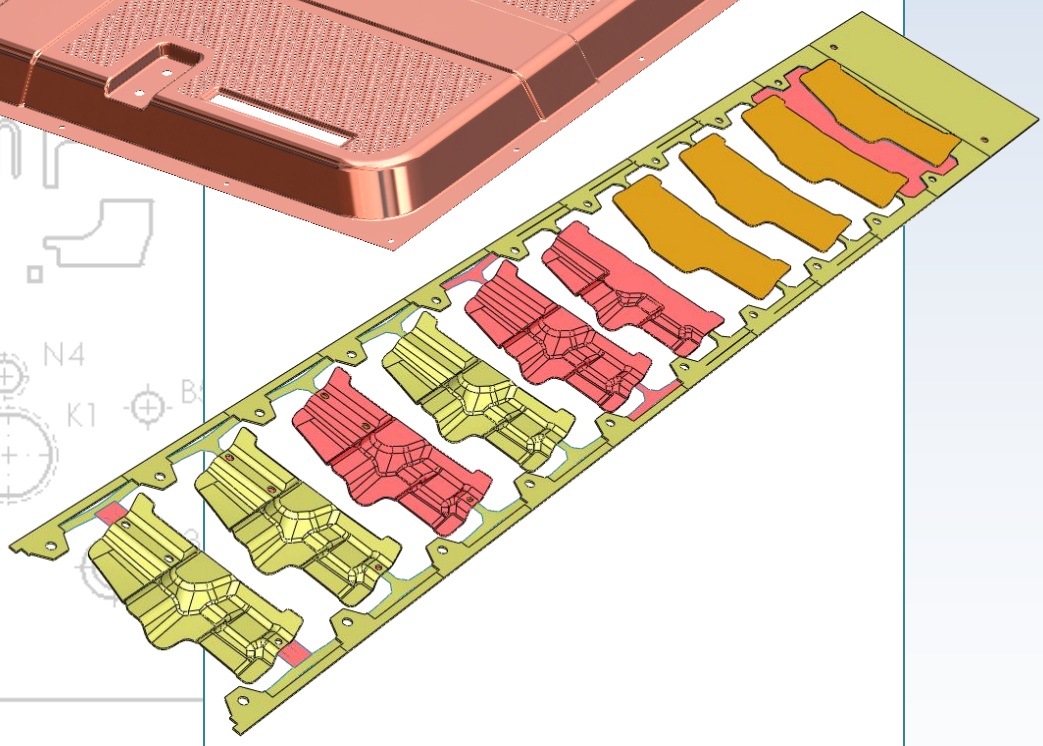
3DQuickPress

What's New in Version 5





3D QuickTools Limited

Contents

3DQuickPress Fundamentals

Special Commands

Open File Without 3DQP Data

Turning on this option, file opened will be loaded without any 3DQP data, like unfold and strip layout information. When opening document, only the SolidWorks data will be loaded.

Disable Automatic SL Opening

Turning on this option, punch design assembly opened will be loaded without the strip layout. When opening the assembly document, only the assembly document is loaded.

Beginner Mode

Turning on this option will enable the 3DQP Beginner Mode, where all external reference will not be created.

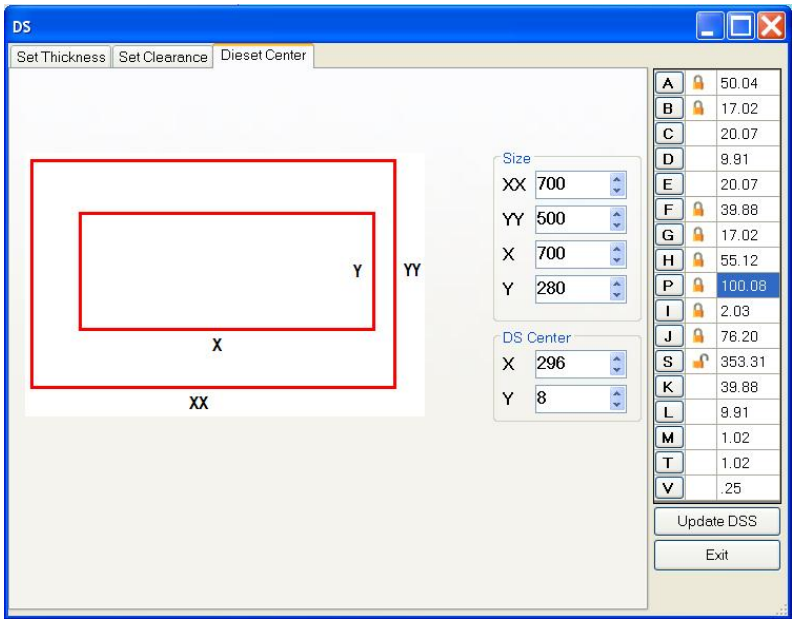
Utilities

Modify Die Set Size and Center

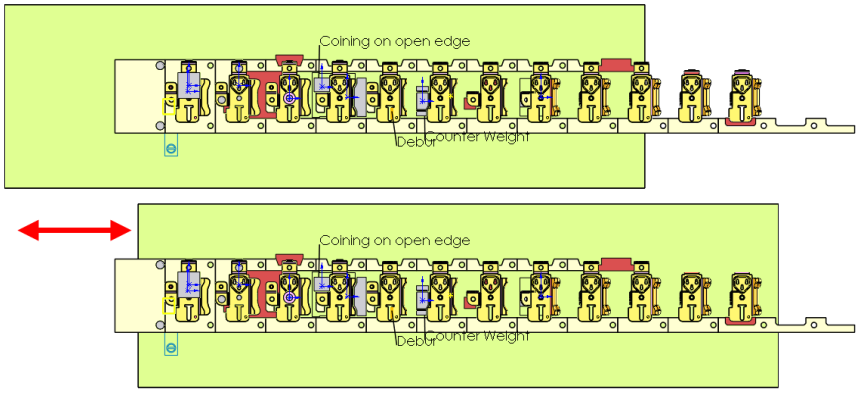
A new interface is added to the Modify Die Set Structure to modify the die set size and die set center.

1. Click **Modify DSS**  from **3DQP Misc. Tool Bar**

2. Click on the **Die Set Center Tab**



3. Modify the position and a preview will be shown in the graphic window



4. Click **Update DSS**
5. Click **Exit**

QT Selection

QT Selection is a new function that speeds up the sketch entities and face selection. There is also function to copy and paste selected faces.

1. Click **QT Selection**  from **3DQP Misc. Tool Bar**

2. The **QT Selection toolbar** below will be shown

2.1. Select all lines in the current sketch

2.2. Select all circles in the current sketch

2.3. Select all arcs in the current sketch

2.4. Select all spline and conics in the current sketch

2.5. Select all circular holes of the selected face

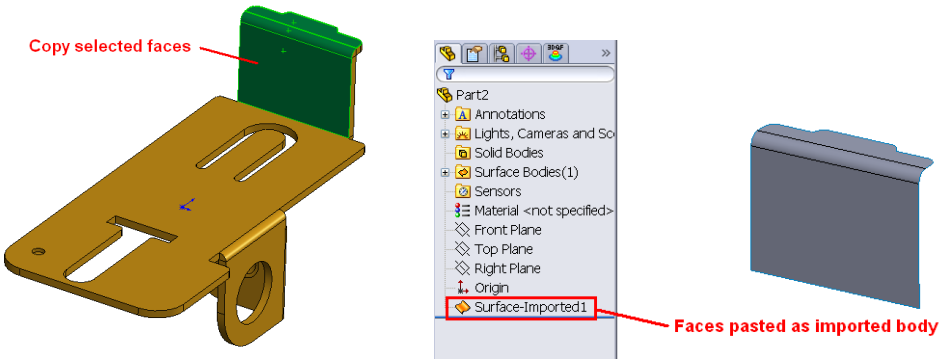
2.6. Select all cylindrical faces of the selected body

2.7. Select all planar faces of the selected body

2.8. Select all faces of the selected body

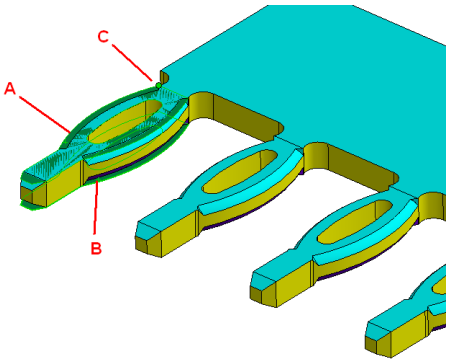
2.9. Copy all selected faces

2.10. Paste copied faces to the current part of a new part

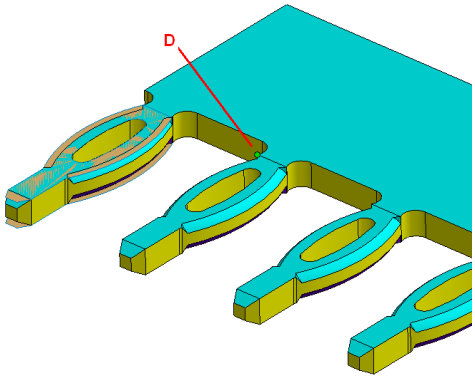


To copy and paste selected faces to another position:

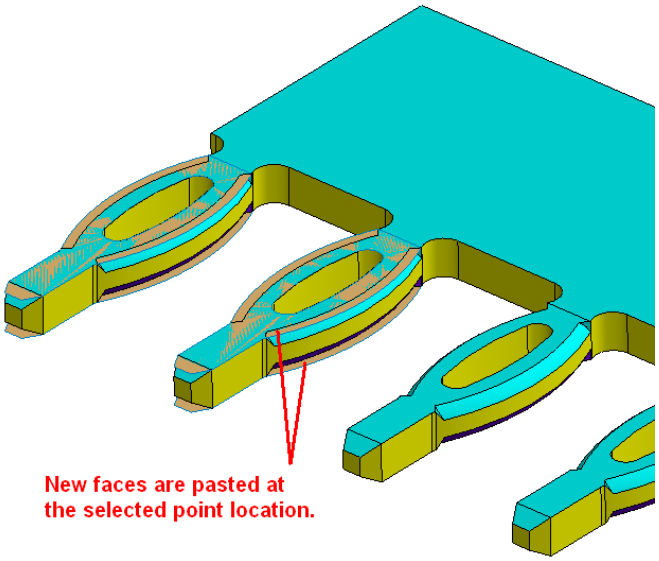
1. Select **Surface Body A and B**, and then select **Vertex C**.



2. Click  to copy the faces with reference to location of **Vertex C**.
3. Select **Vertex D**.



4. Click  to paste the copied faces to location of **Vertex D**.



Multiple Body Design Tool

Multiple Body Design Tool is a set of tools that helps in manipulating multiple body parts and components in assembly.

1. Click **Multiple Body Tool**  from **3DQP Misc. Tool Bar**

2. The **Multiple Body Design toolbar** below will be shown

2.1. Assign random color to bodies of a multi-body part or components of an assembly

2.2. Rename bodies of a part with the part name as prefix

2.3. Create assembly form multi-body part

2.4. Create an indent feature on the target body with the selected body

2.5. Toggle the bodies of a component to be transparent

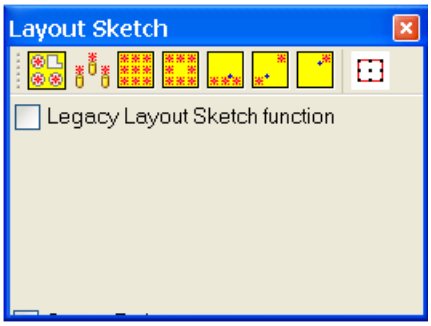
2.6. Create assembly from the select parts

Layout Sketch Tool

Layout Sketch Tool is a set of tools that can quickly draw points on layout sketches.

1. Click **Layout Sketch** from **3DQP Sketch Tools Toolbar**

2. The **Layout Sketch Tool** dialog below will be shown



2.1. Create points on circular holes of a selected face

2.2. Create points on origin of identical components

2.3. Create point pattern on a select face

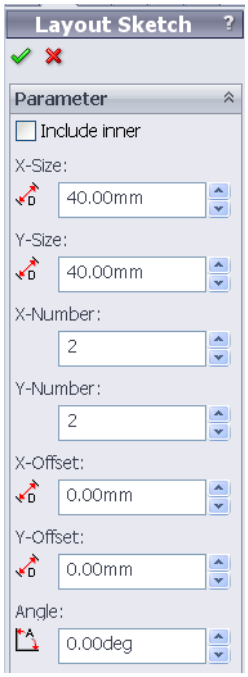
2.4. Create points along the boundary of a selected face

2.5. Create points along the closest edge

2.6. Create two diagonal points

2.7. Create point at the closest corner

2.8. Create points automatically with X and Y dimension entered.

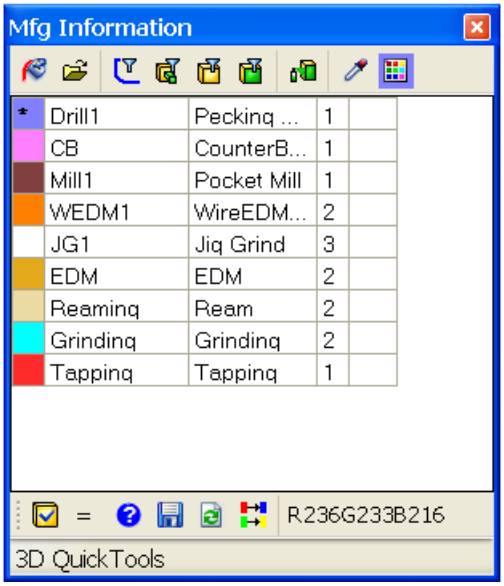


Manufacture Color Tool

Manufacture Color Tool is a set of tools that can set and unset mfg. color onto faces of components.

1. Click **Mfg Color Tool** from **3DQP Misc. Tool Bar**

2. The **Mfg Color Tool** below will be shown



2.1.  Toggle set and unset color to the selected face

2.2.  Load an external manufacturing color data file

2.3.  Set/Unset the Mfg. Color to the selected hole sketch

2.4.  Set/Unset the Mfg. Color to the selected feature

2.5.  Set/Unset the Mfg. Color to the selected body

2.6.  Set/Unset the Mfg. Color to the selected face

2.7.  Assign the color of the body/feature to the face of the body/feature

2.8.  Pick color from the selected face/body/feature

2.9.  Pick color from the color pallet

2.10.  Highlight faces that have color setting

2.11. Highlight faces that have the selected Mfg. Color

2.12.  Identify faces that do not have Mfg. Color setting

2.13.  Save the existing Mfg. Color setting to the part

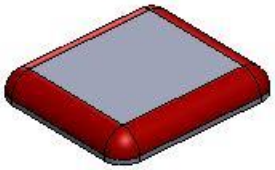
2.14.  Reload Mfg. Color Info. Form the data file

2.15.  Load the Mfg. Color Conversion data file

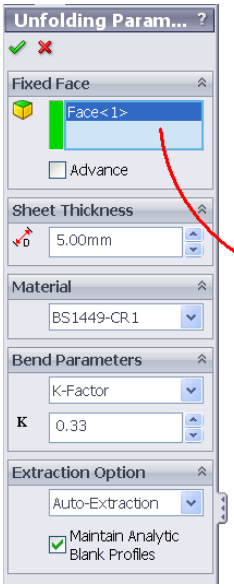
Unfold

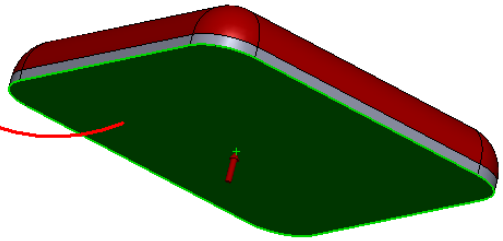
Coining

Coining feature is a new feature induced to handle forming features around the boundary of a part. Coining feature can be identified automatically by 3DQuickPress. The following steps will show how to unfold a part with coining feature.

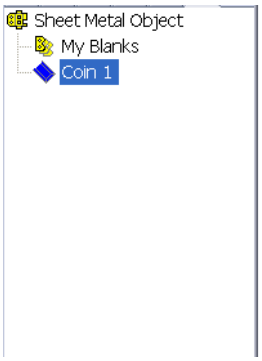


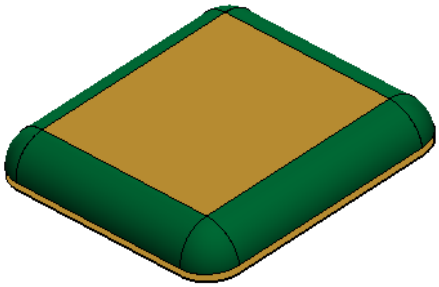
1. Click **Unfold** on the **3DQP Toolbar**
2. Select the highlighted face as the **fixed face** and click **OK**





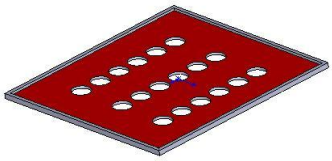
3. The **Coin 1** feature will be identified automatically



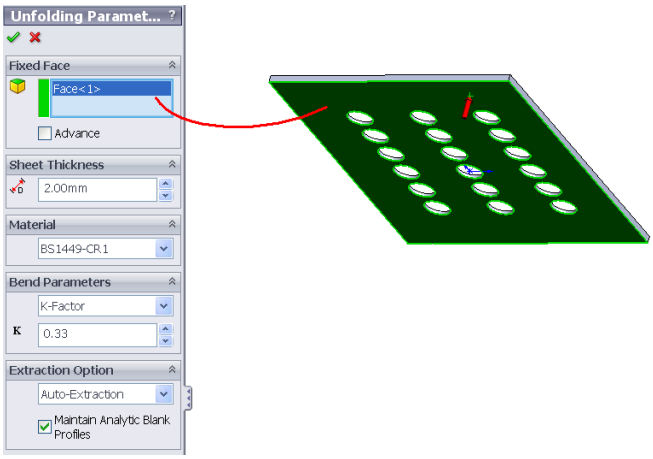


Hole Cluster

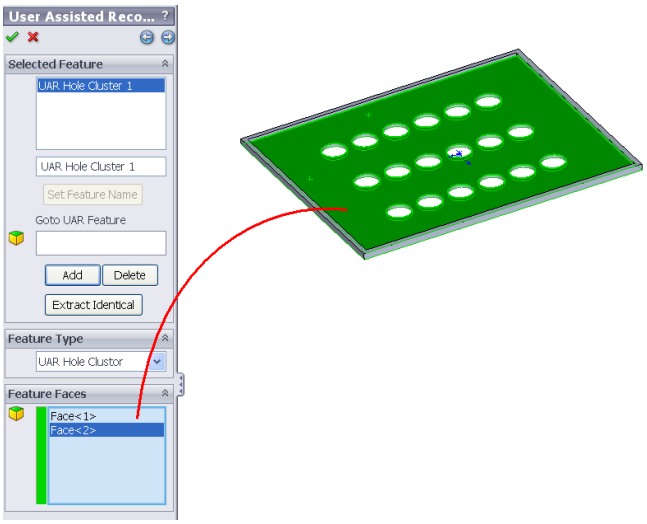
Hole Cluster feature is a group of piercing holes that are having the same size and shape. A single hole cluster feature can be consisted of several thousands of piercing holes. The display of the hole cluster feature is optimized to improve the performance of parts that contains thousands of piercing holes. Below is an example of the use of the hole cluster feature.



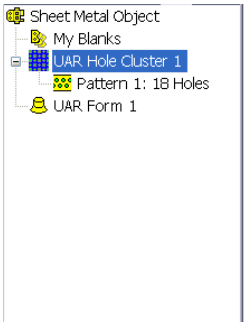
1. Click **Unfold** on the **3DQP Toolbar**
2. Select the highlighted face as the **fixed face** and click **OK**

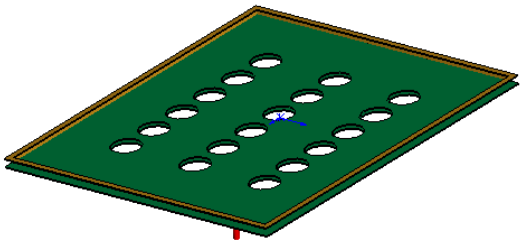


3. **RMB** on **Sheet Metal Object** and select **User Assisted Recognition…**
4. Click **Add** to add feature
5. Select the feature type to be **UAR Hole Cluster**
6. Select the highlighted faces as the feature face



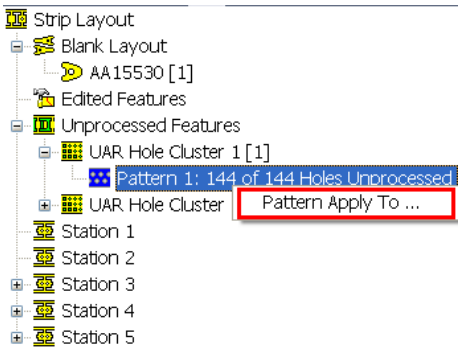
7. Click **OK** to rebuild the unfold part
8. **UAR Hole Cluster 1** Feature with 18 piercing holes is created



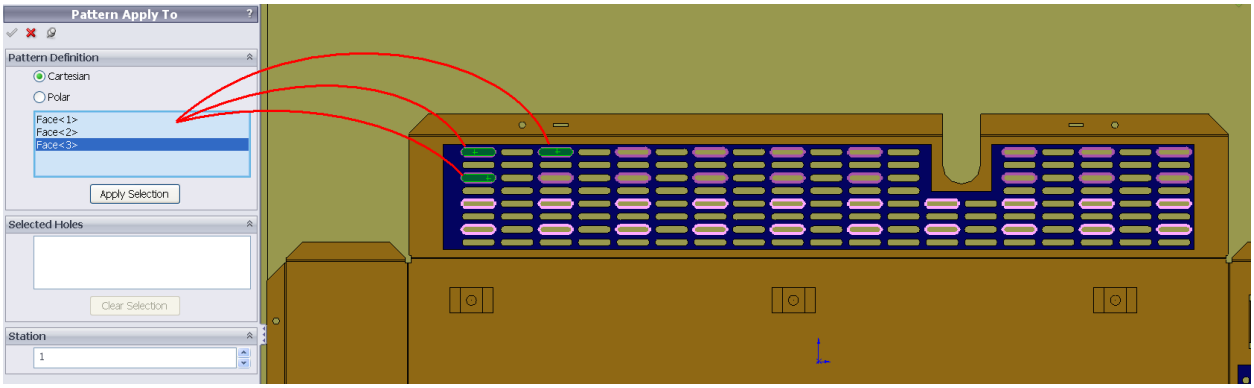


Activating hole cluster in strip layout:

1. Expand **Unprocessed Features** on the strip layout feature tree.
2. Expand **UAR Hole Cluster 1** Feature.
3. RMB on **Pattern 1** and select **Pattern Apply To…**

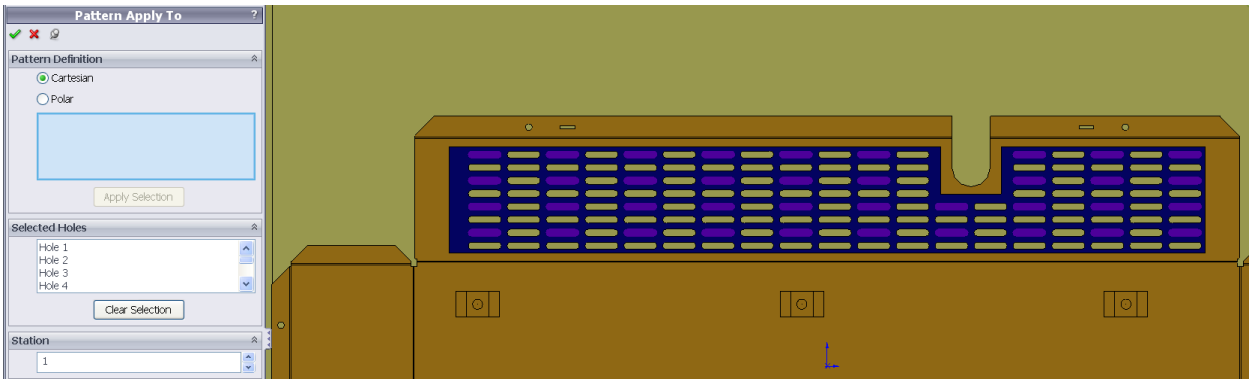


4. Check **Cartesian** option and select the three highlighted précising holes to the **Pattern Definition** area.

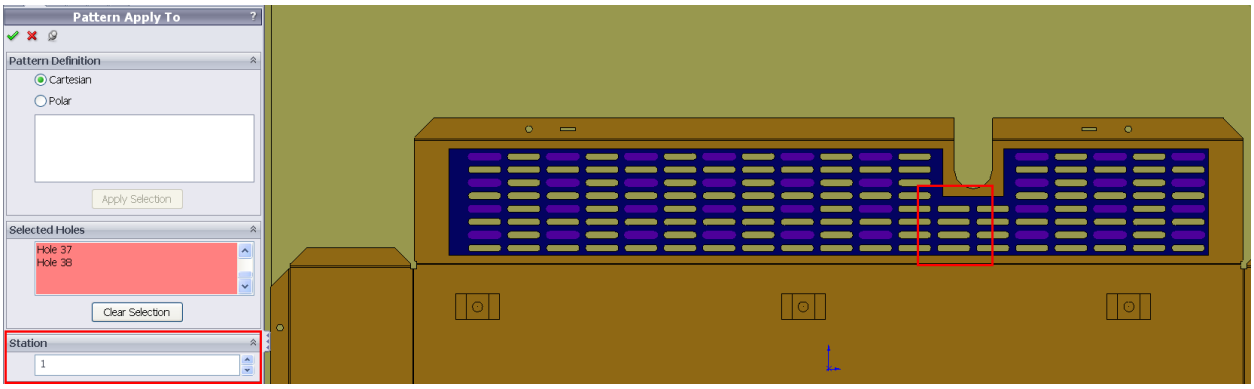


5. The pattern will propagated automatically to select précising holes along the X and Y direction according to the selected pattern seed features.
6. Click **Apply Selection** to select all highlighted précising holes.

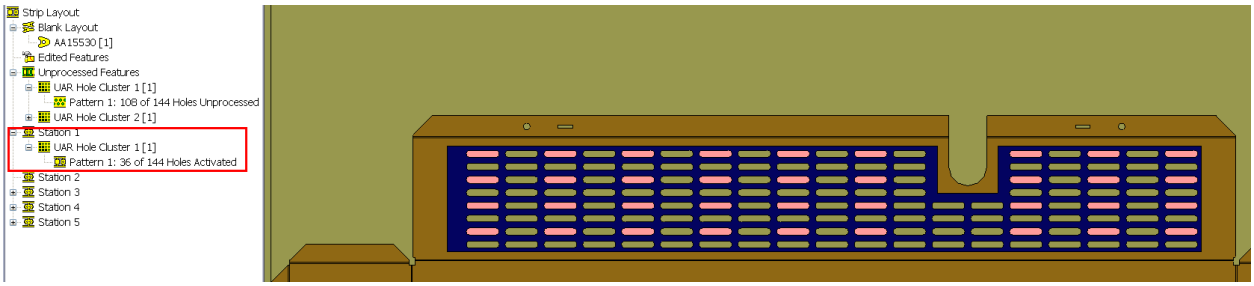
7. All selected pattern précising holes are listed in the **Selected Holes** area and colored in purple.



8. Deselect hole 7 and 17 from the graphic area.

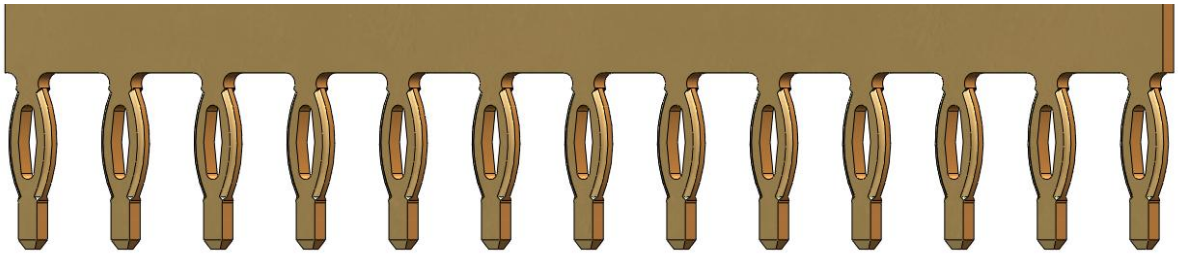


9. Set to apply selected holes to **Station 1**.
10. Click **OK** to apply the selected holes to **Station 1**.

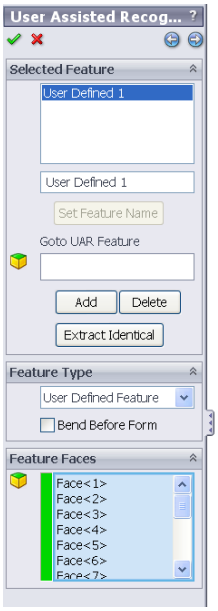


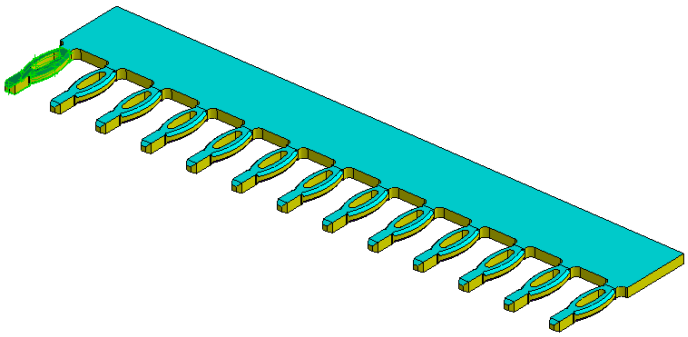
Extract Identical Features

A new function is introduced to **User Assisted Recognition** interface to extract identical UAR features. By clicking on the **Extract Identical** button, features identical to the selected UAR features on the unfold part will be created automatically. All UAR features will support the Extract Identical Feature function. The below steps are used to extract those identical User Defined Features in the example.

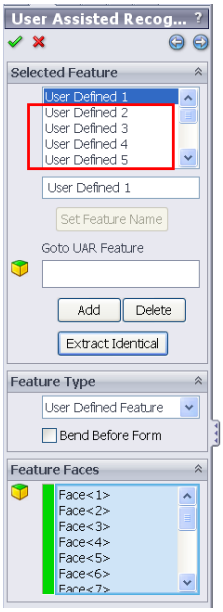


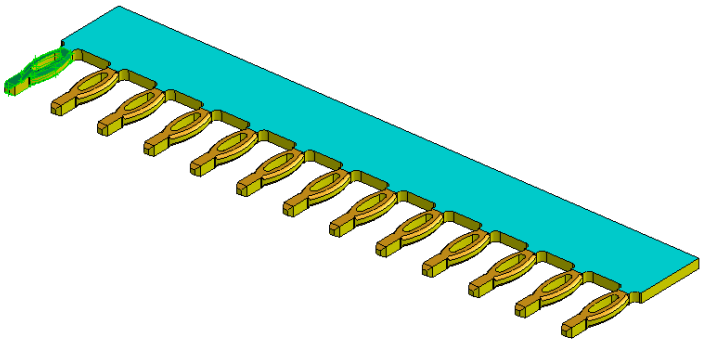
1. **RMB** on **Sheet Metal Object** and select **User Assisted Recognition…**
2. Select **User Define 1** and click **Extract Identical**





3. A list of identical user defined feature is created automatically.

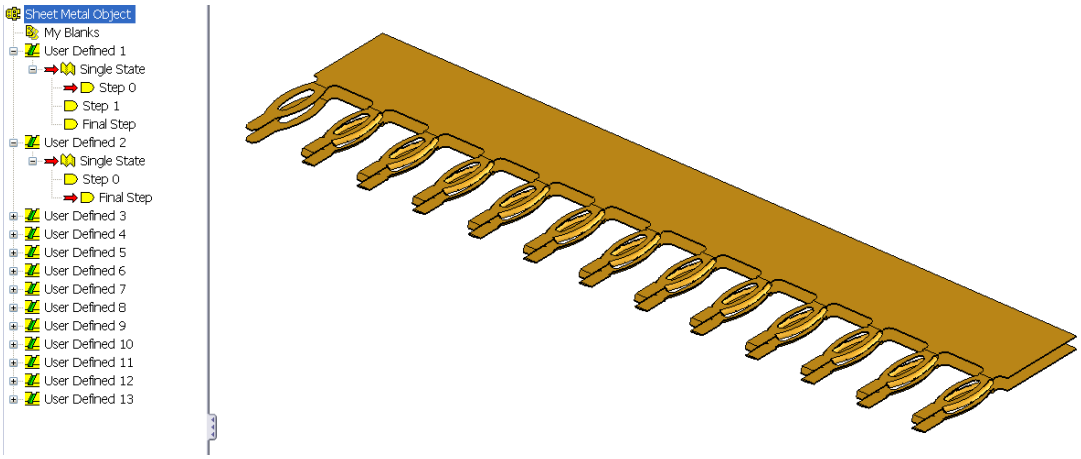




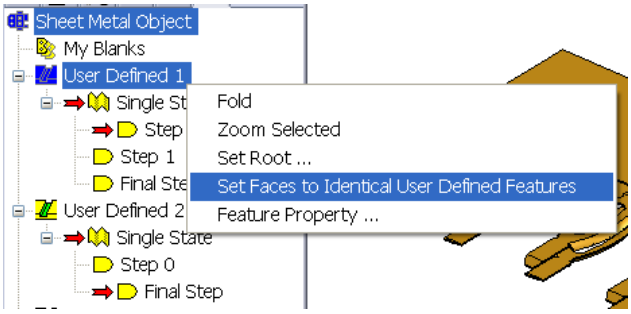
4. Click **OK** to rebuild the unfold part.

Copy Face to Identical User Defined Features

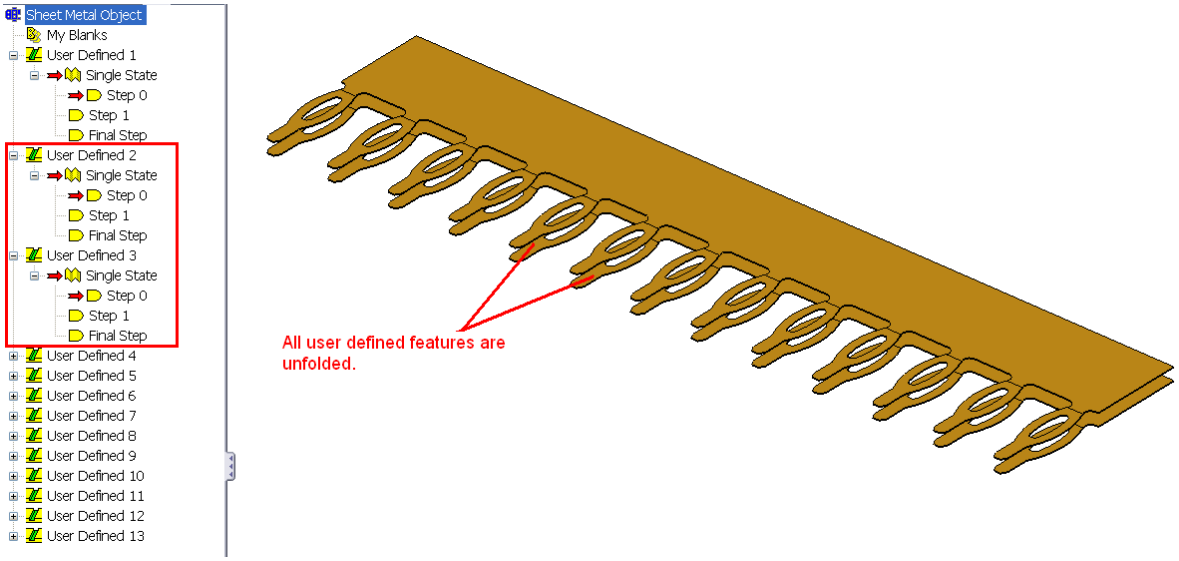
For identical User Defined Features in an unfold part, a new function is introduced copy the settings of a selected user defined feature to all identical user defined features. Both the setup of steps and the set faces will be copied. Below are the steps to copy the setting of User Defined 1 to all other user defined features.



1. Select the completed user defined feature **User Defined 1**.
2. **RMB** on **User Defined 1** and select **Set Faces to Identical User Defined Features**.

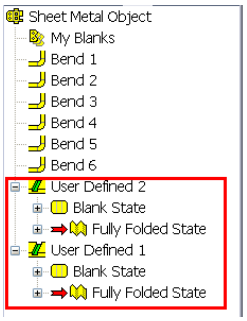


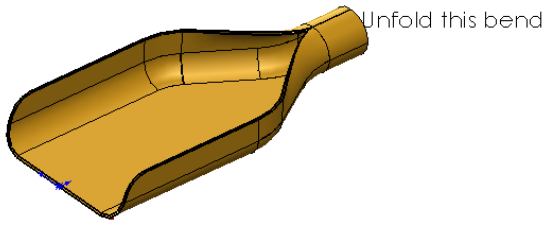
3. The steps and the set faces are then copied to other identical features.



Single State User Defined Feature

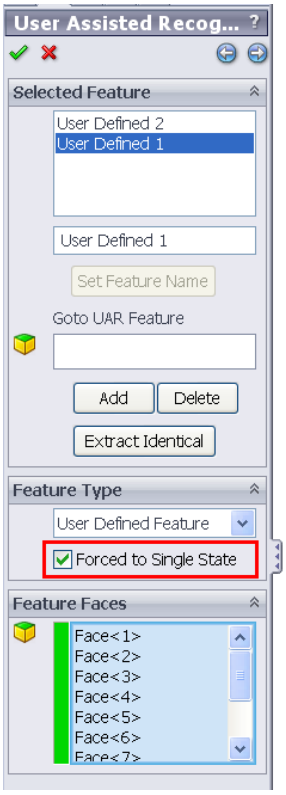
New operation allows user to set a multiple state user defined feature to a single state user defined feature. A single state user defined feature bypasses the control of the states on the near-by features, and let the user to manipulate the steps of a user defined feature directly.

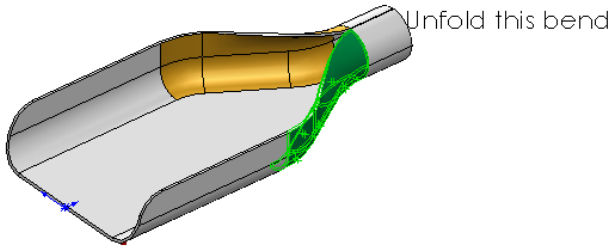




Steps to change the above two multi-state user defined feature to a single state one:

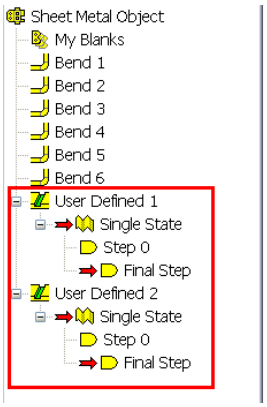
1. **RMB** on **Sheet Metal Object** and select **User Assisted Recognition…**
2. Select **User Defined 1.**
3. Check the **Force to Single State** option.

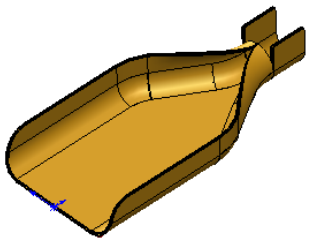




4. Repeat **Steps 2** and **3** for **User Defined 2.**
5. Click **OK** to rebuild the unfold part.

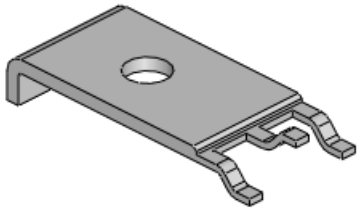
6. The two user defined features now becomes single state. Steps can be added and faces can be set to each steps.



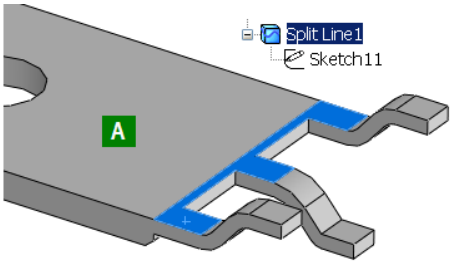


Unfold Multiple Thickness Part

To handle parts that have the thinning feature, function to unfold part that has multiple thickness is introduced. Below is an example of the steps to unfold a multiple thickness part.

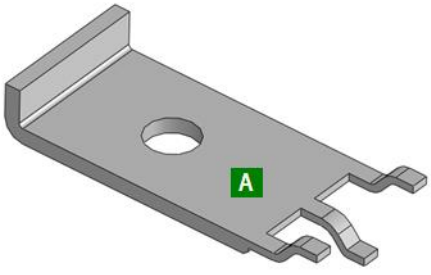


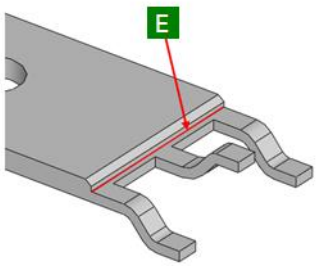
Before start 3DQuickPress Unfold, the regions of different thickness must be separated by

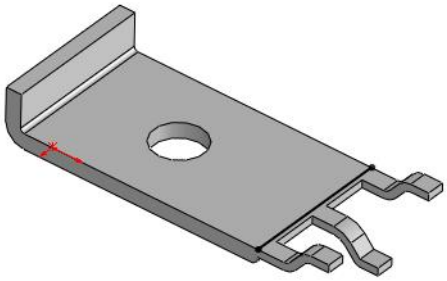


Split Face.

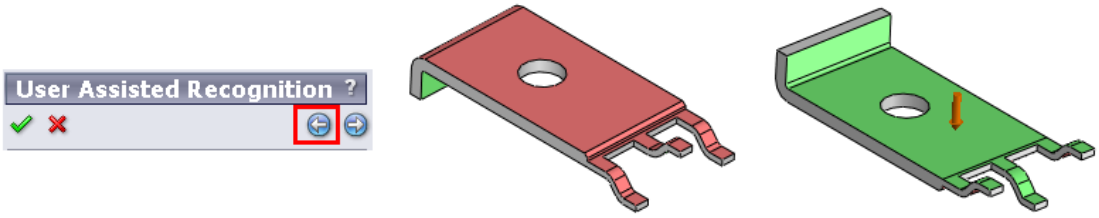
1. Click Face A and click **Insert**, **Sketch**



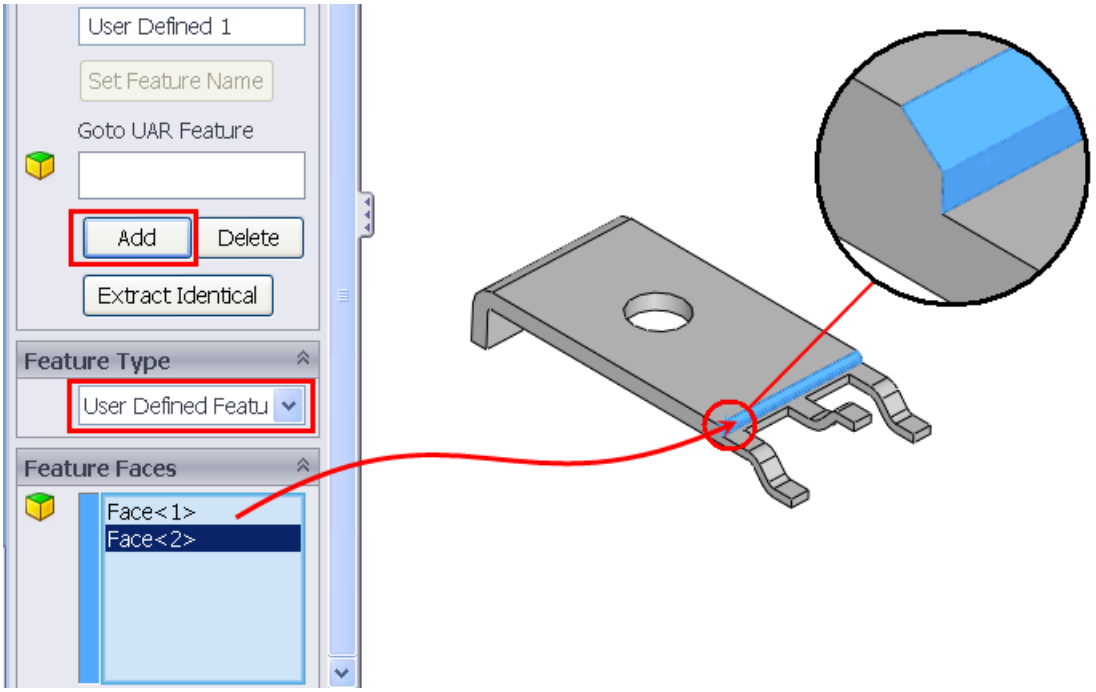




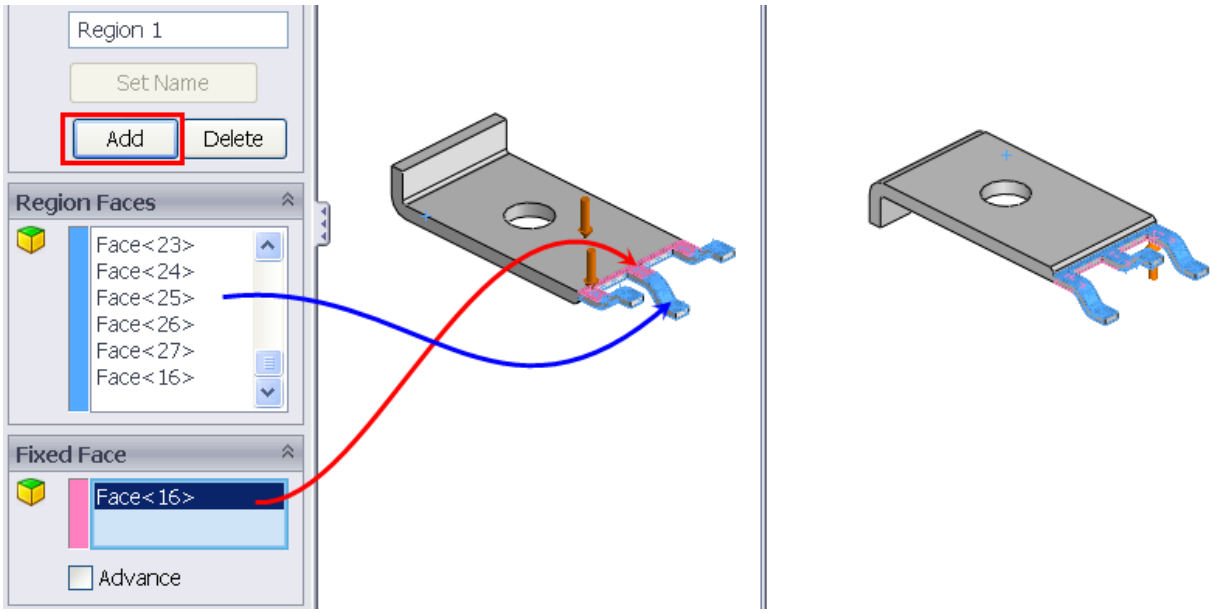
2. Select Edge E, click Tools, Sketch Tools, Convert Entity
3. Click **Insert**, **Curve**, **Split Line**, select Face A to Faces to Split
4. Click OK to finish the separation of regions
5. Click **3DQuickPress**, **Unfold Part**, **Unfold Part** to start unfold
6. Select Face A for Fixed Face
7. RMS **Sheet Metal Object** and click **User Assisted Recognition**
8. Click **Previous Button**  to set Top and Bottom Face properly



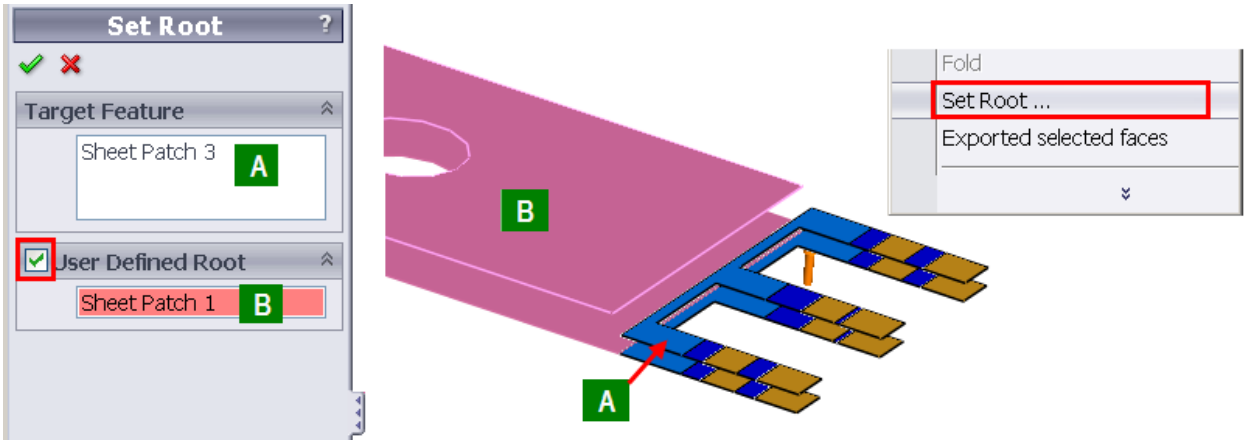
9. Click **Next Button**  and Add an User Define Feature which connect the thick and thin region



10. Click Next Button  and add a new region



11. Click OK  to exist UAR mode.
12. RM select Face A, click Set Root, check User Defined Root, click face B

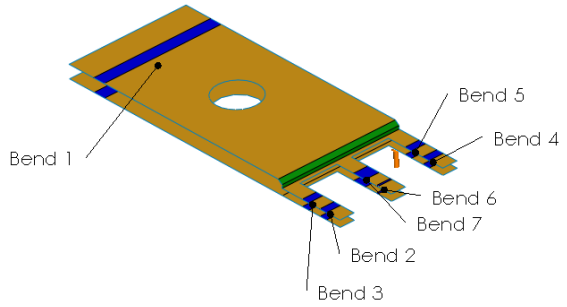


Export Face with SolidWorks Annotation

Export face of unfold part now will also export the feature name annotation.

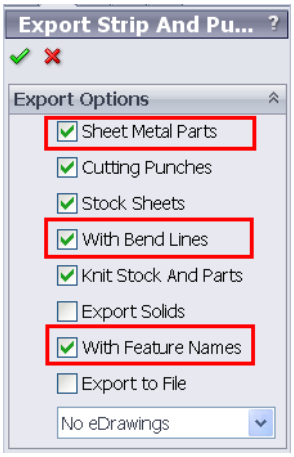
1. **RMB** on **Sheet Metal Object** and select **Export Sheet Metal Object Faces.**
2. Click **Yes** when asked to export with bend lines.

3. Object faces with feature name annotation will be exported.

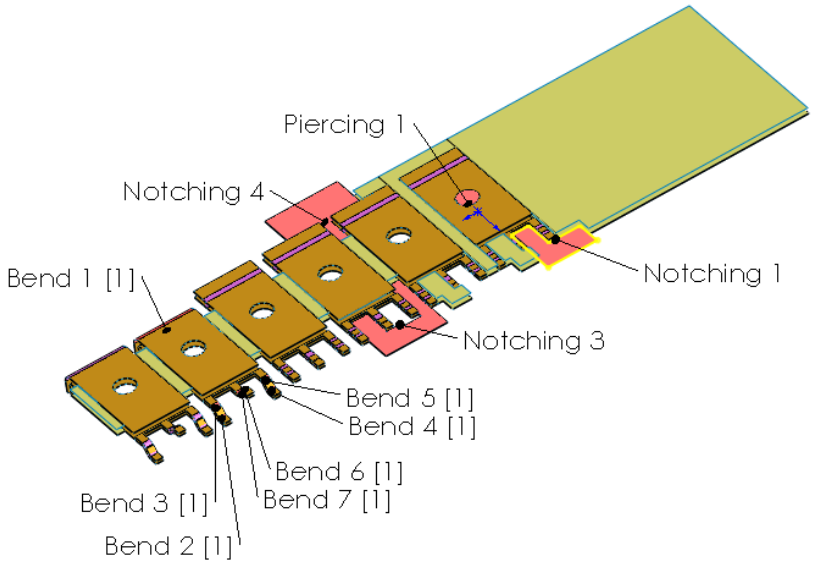


Export face with feature name is also supported in strip layout.

1. **RMB** on **Strip Layout** and select **Export Strip and Punches.**
2. Enable the options **Sheet Metal Parts**, **With Bend Lines**, and **With Feature Names**.



3. Click **OK** to export the strip layout with feature names.

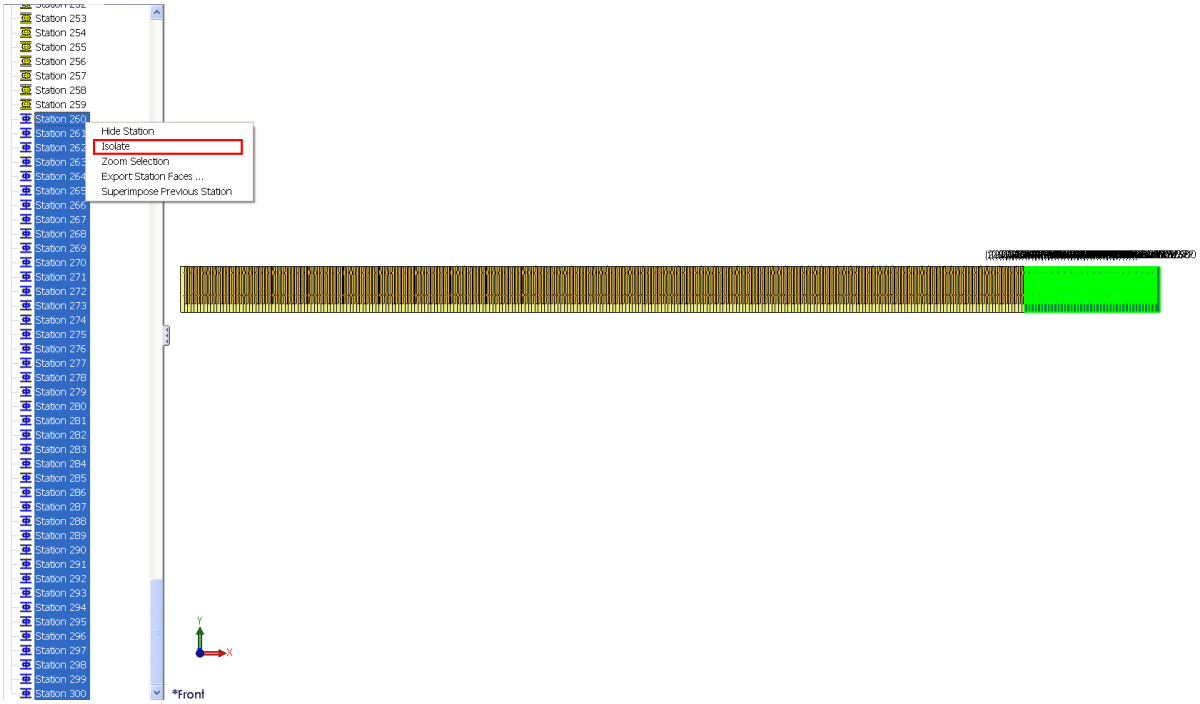


Strip Layout

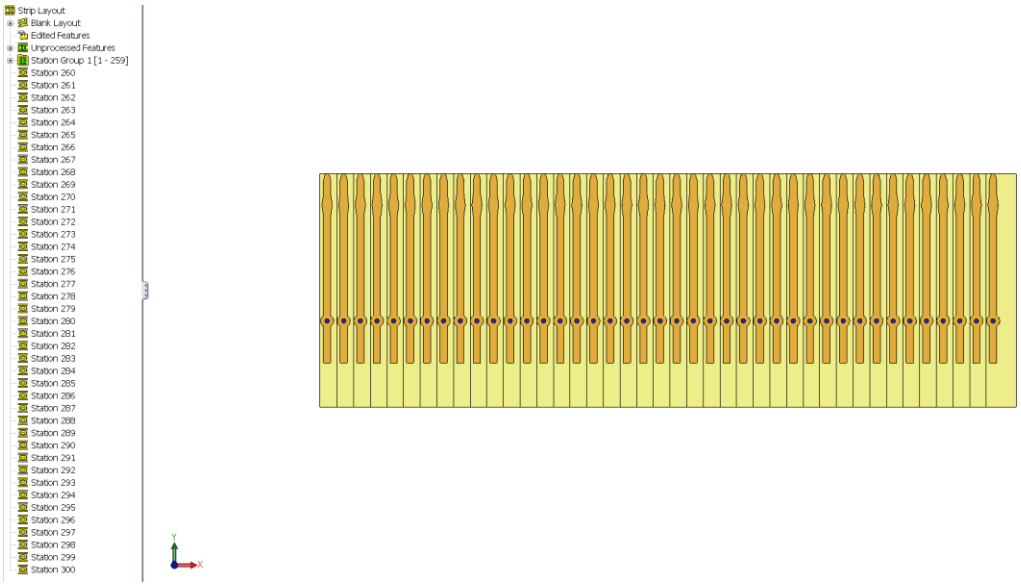
Isolate Stations

Isolate stations allows user to view the selected stations on the strip layout only, and hiding the others.

1. Select the stations **260** to **300** that are going to be isolated.
2. **RMB** on the selected stations and select **Isolate**.



3. The strip layout and the feature tree will be shortened to show only the selected stations.



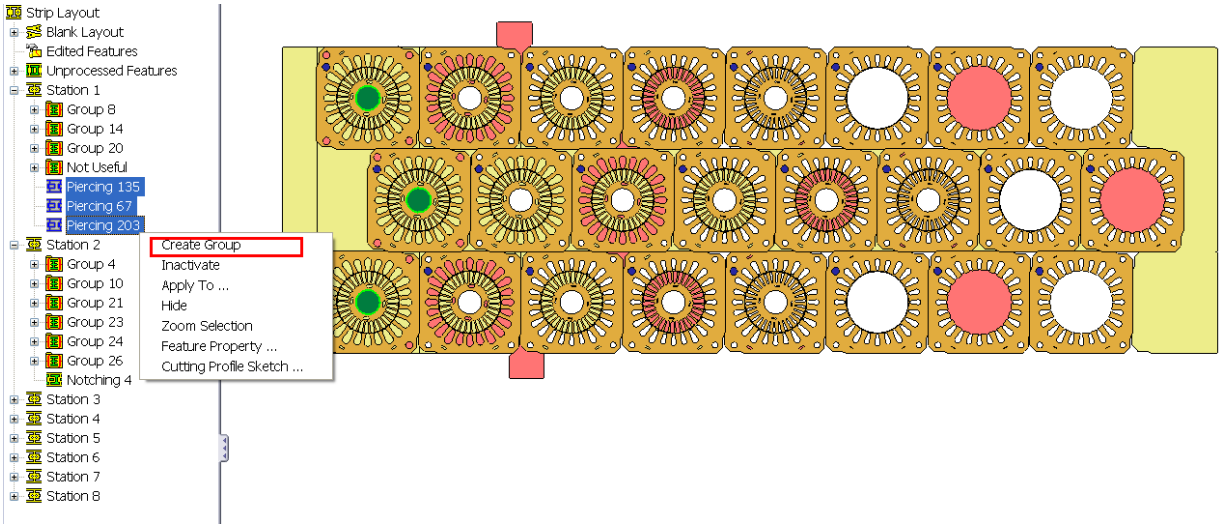
4. To un-isolate the stations, select stations **260** to **300** again.
5. **RMB** on the select stations and select **Unisolate**.

Group Features

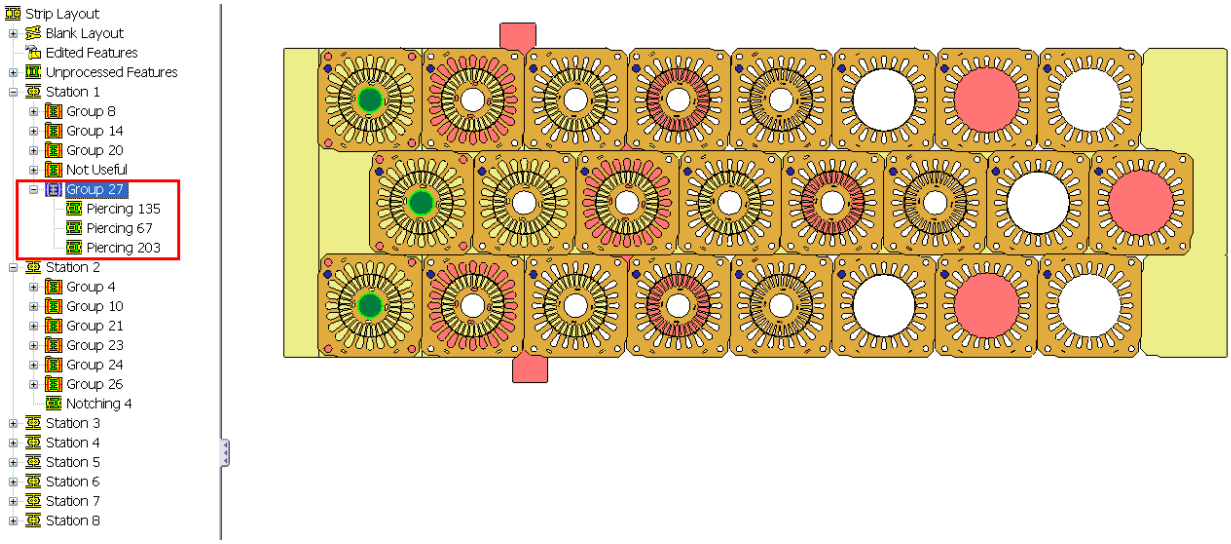
Group feature is used to group the strip layout feature together. It will then allow user to activate and inactivate a group of strip layout features together. Components created in punch design can also be linked to a group feature.

To create a group:

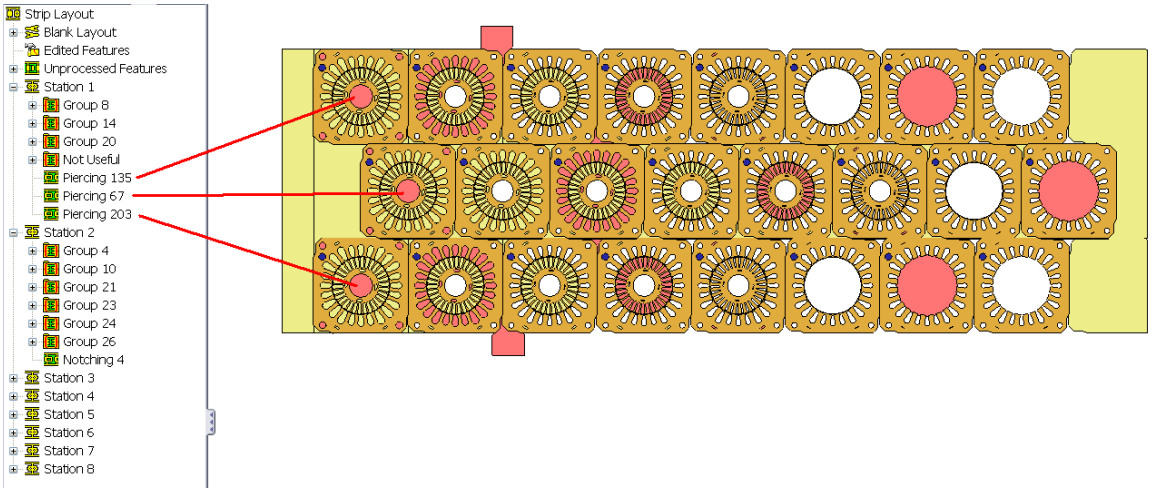
1. Select the features that are going to be grouped.
2. **RMB** on the selected feature and click **Create Group**.



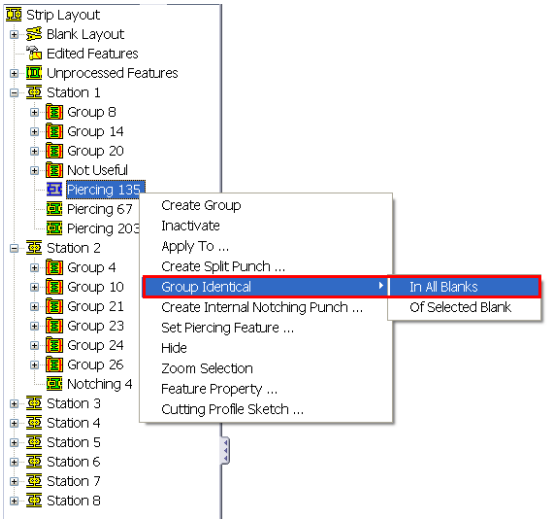
3. A group feature is created.



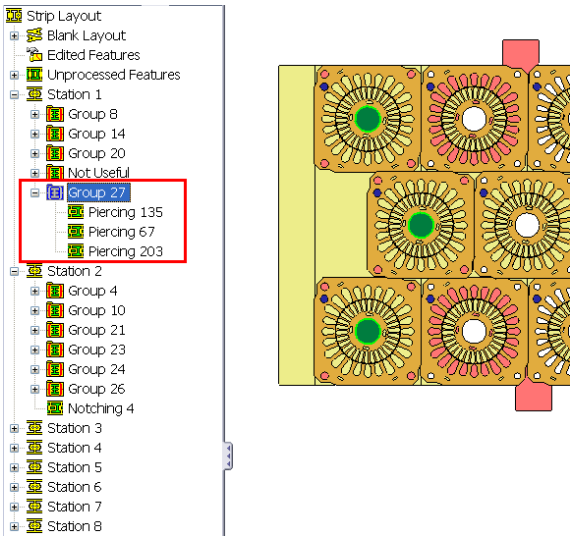
To create a group from identical features:



1. **RMB** on cutting feature **Piercing 135**.



2. Select **Group Identical->In All Blanks**



To delete a group:

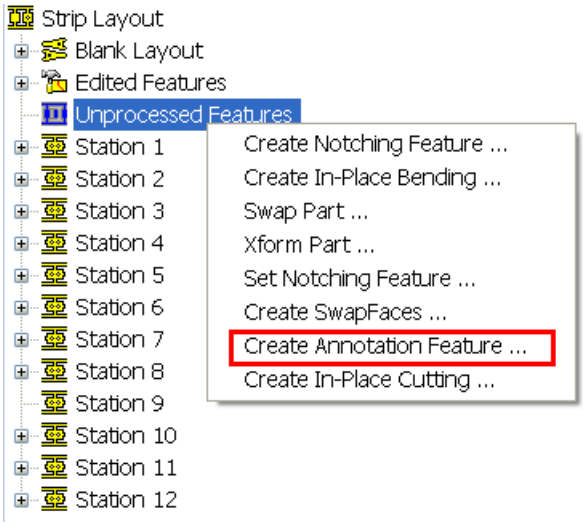
1. **RMB** the group feature and select **Delete Group.**

Annotation Feature

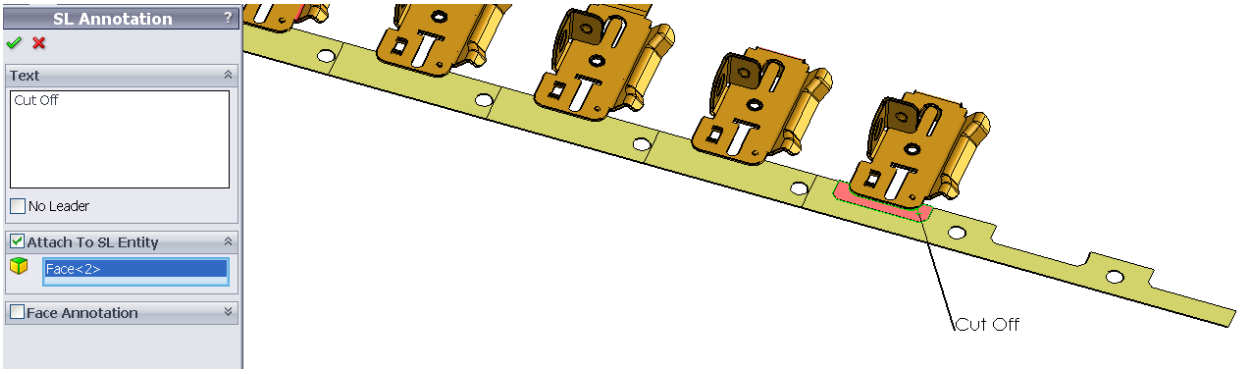
Annotation feature is a note attached to the strip layout entity or to strip layout face. An annotation feature can be activate and inactivate on any stations.

To create an annotation feature attached to strip layout entity:

1. **RMB** on the **Unprocessed Feature** on the strip layout feature tree.
2. Select **Create Annotation Feature…**



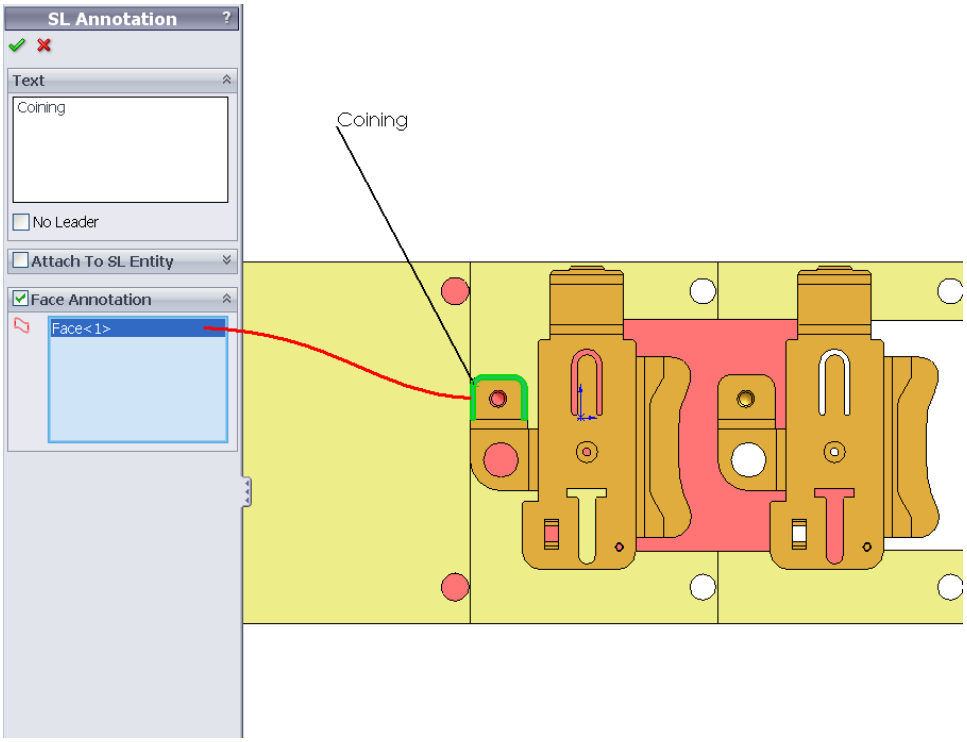
3. Type the text “Cut Off” to the text area.
4. Check **Attach To SL Entity**.
5. Select the **notching 4** feature on **station 12**, and click on the location where the note is placed.



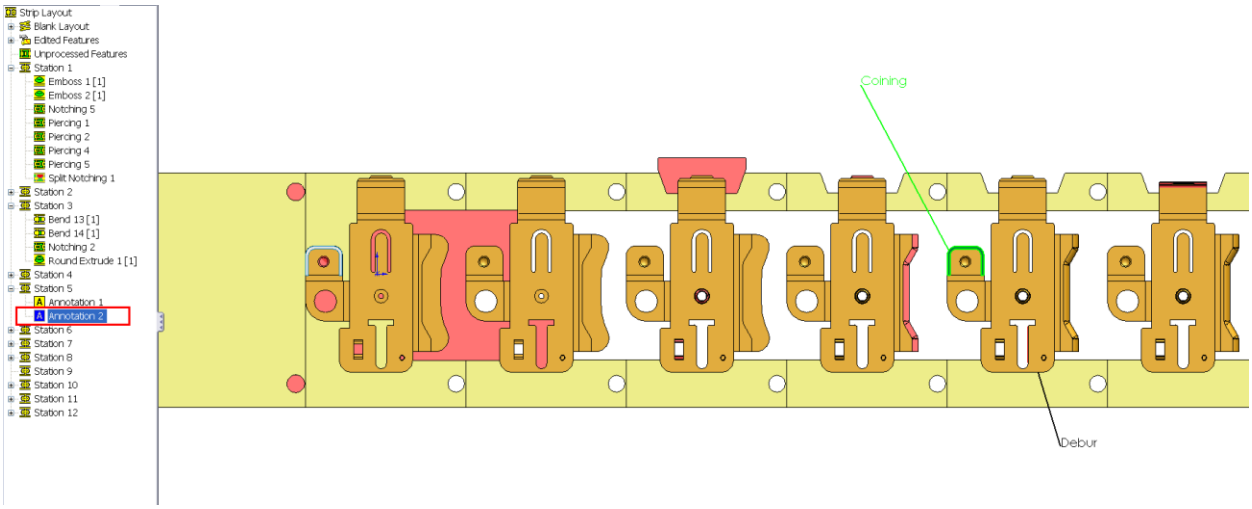
6. Click **OK** to create the annotation feature.

To create an annotation feature attached to a face:

1. **RMB** on the **Unprocessed Feature** on the strip layout feature tree.
2. Select **Create Annotation Feature…**
3. Type the text “Coining” to the text area.
4. Check **Face Annotation**.
5. Select the highlighted face created on **Station 1**, and click on the location where the note is placed.



6. Click **OK** to create the annotation feature.
7. Apply the **Annotation 2** feature to **Station 5**.

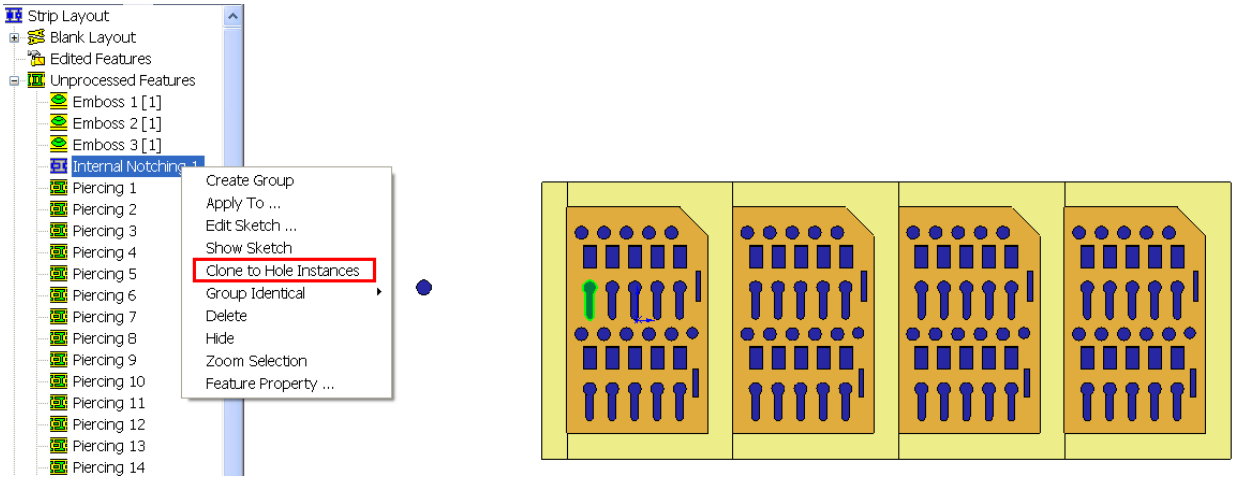


Clone Internal Notching

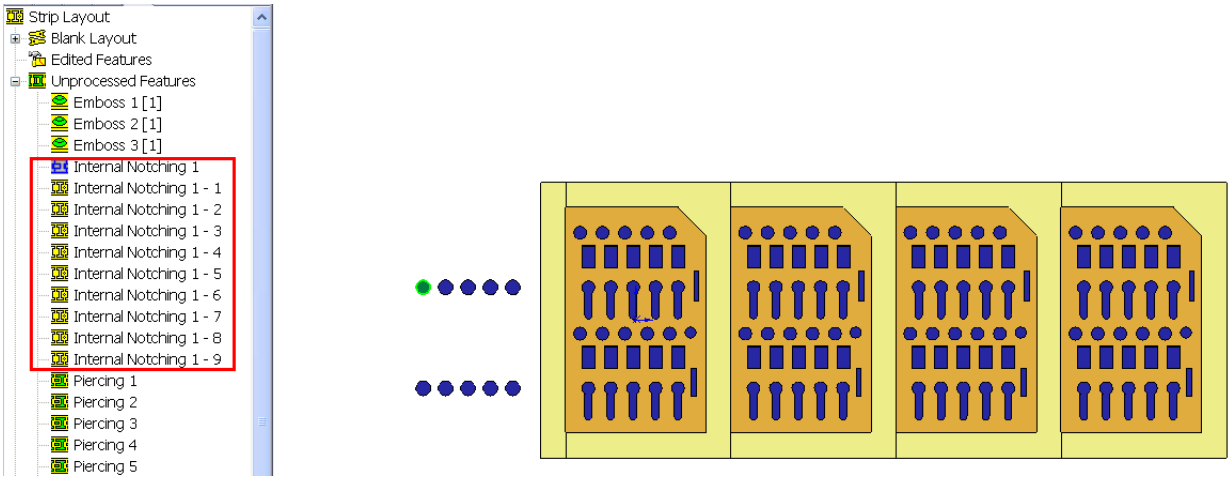
Internal notching feature can be cloned to identical piercing features. Cloned internal notching features will update according to the parent internal notching feature.

To clone internal notching feature:

1. Create the parent internal notching **Internal Notching 1** from **Piercing 27**
2. **RMB** on **Internal Notching 1** and select **Clone to Hole Instances**



3. A group of internal notching features same as **Internal Notching 1** is cloned.



Swap Part Apply Up To

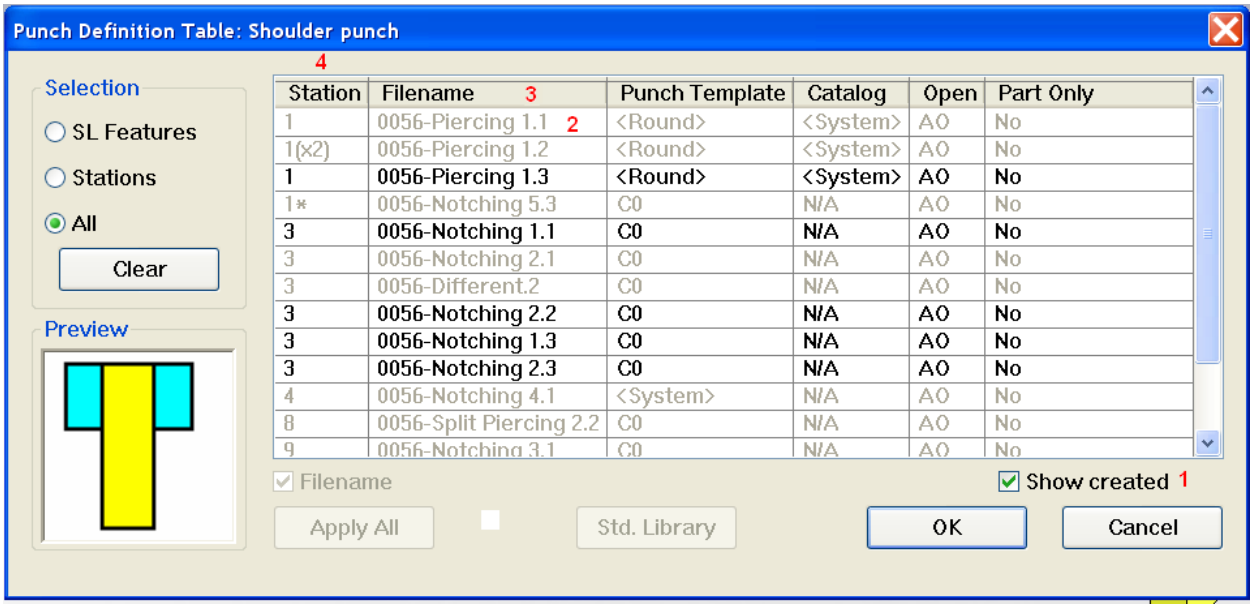
Swap Part feature can apply up to a target station. Swap Part feature can be applied within a range of stations.

Edit Swap Face & In-Place Bend

Edit of Swap Face feature and In-Place Bend features is supported.

Punch Design

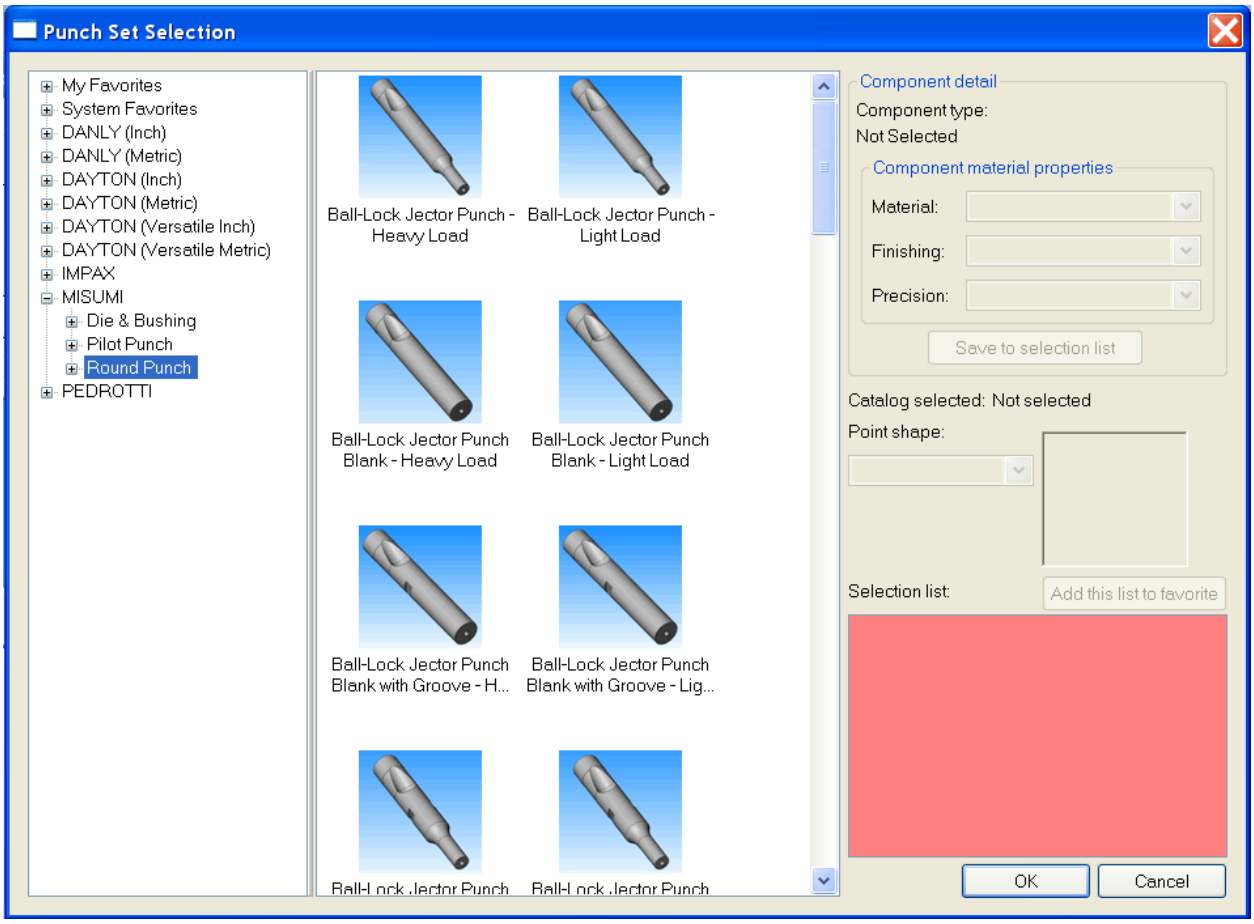
Changes on Create Punch



1. **Show created** option is added to **Punch Definition Table** to show those cutting features that punches have already been created.
2. Cutting features of the primary strip layout will be shown in dark color and cutting features of other strip layout will be shown in grey color.
3. Default cutting punch component name will be named in a format “project number-cutting feature name.x”, where the “.x” represent which strip layout is the cutting features belongs to.
4. When there is patterned cutting feature, a bracket with the number of instance cutting feature will be added to the station number.

Ball Lock Round Punch Standard

Heavy duty and light duty ball lock punches, pilots and dies are added to standard catalog round punch database for catalog DANLY (Inch), DANLY (Metric), DAYTON (Inch), DAYTON (Metric), IMPAX, and MISUMI.

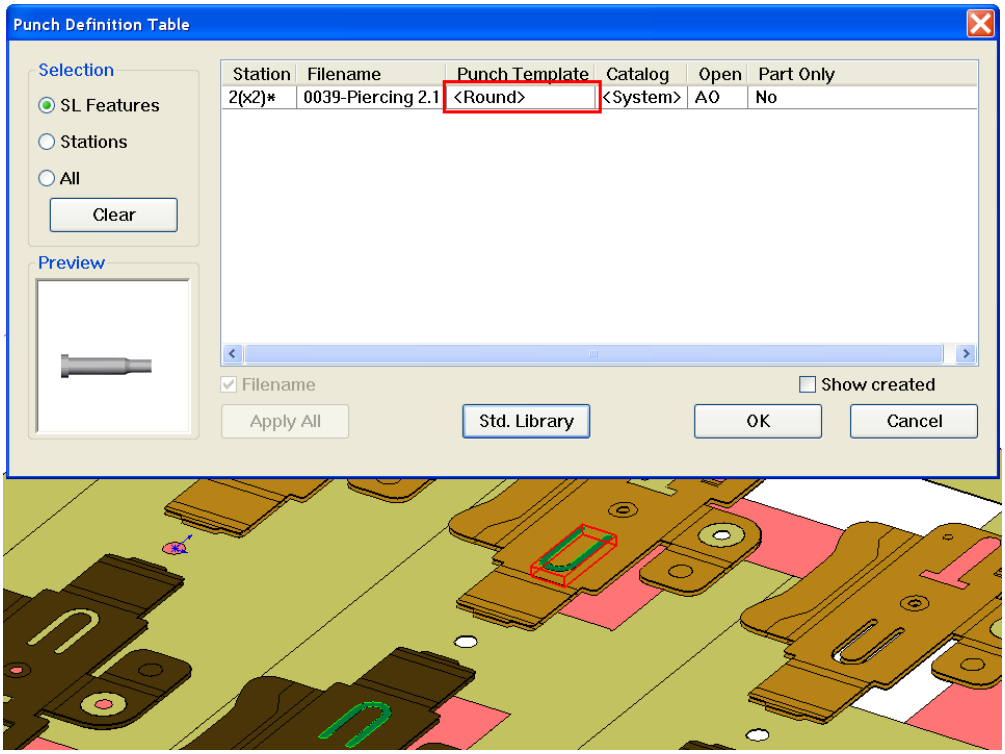


Round Punch for Ob-round Shape

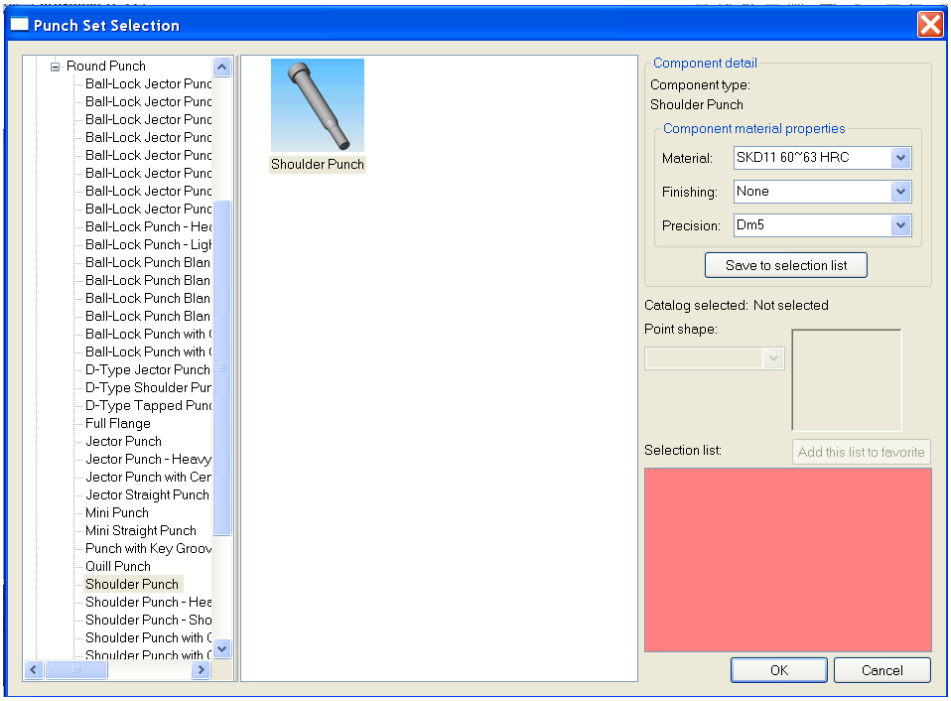
For Ob-round shape cutting features, standard round punch from the standard catalog can be selected as the punch body. The head shape of the punch created with have the shape of the ob-round cutting hole. Both round punch and die of ob-round shape can be created. And Component Library can be used to update the body size of the standard punch even the punch is having an ob-round shape. The steps below show the procedures to create a standard round punch with ob-round shape cutting head.

1. Click **Punch Definition Table** on the **3DQP PRL Toolbar**

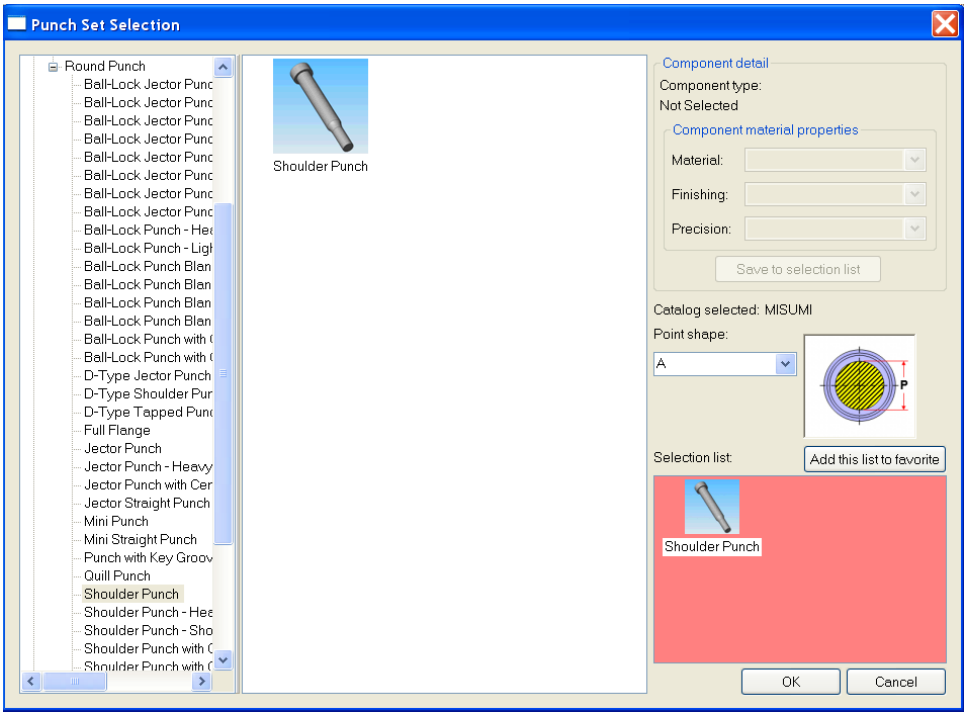
2. Select the strip layout feature **Piercing 2** and select **<Round>** as the **Punch Template.**



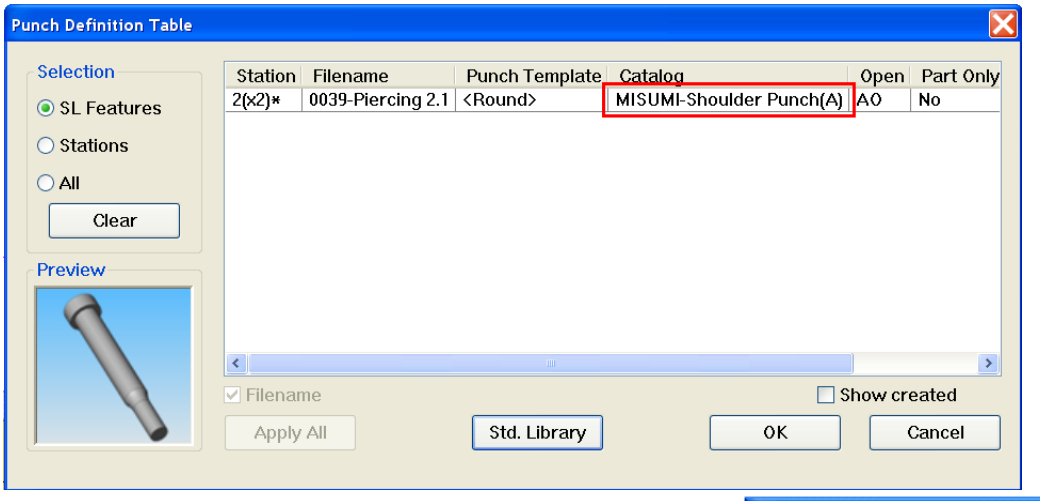
3. Click **Std. Library** button to start Standard Library selection.
4. Select MISUMI->Round Punch->Shoulder Punch



5. Select material and click **Save to selection list button**.

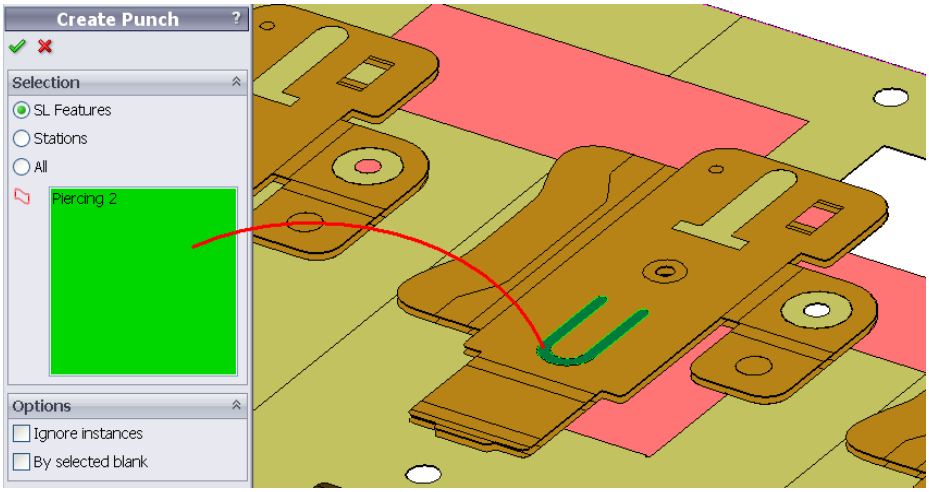


6. Click OK to confirm the selection.
7. The Catalog selected will be updated with the standard punch type, then click OK to complete the punch selection.

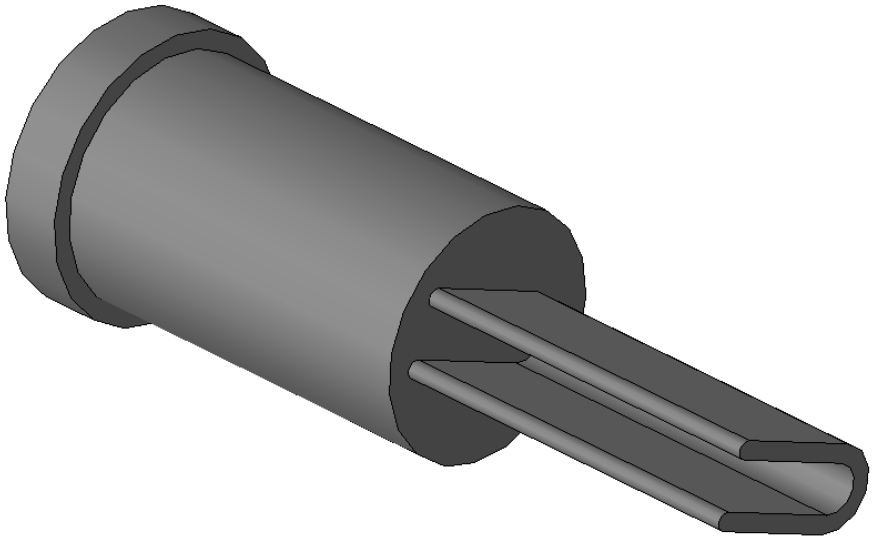


8. Click **Create Cutting Punch** on the **3DQP PRL Toolbar**

9. Select the strip layout feature **Piercing 2**.



10. Click **OK** to create the cutting punch.
11. Open the created cutting punch 0039-Piercing 2.1, the cutting head of the round punch is the shape of the cutting feature on the strip layout, and the size of the punch body follow the data in the catalog.



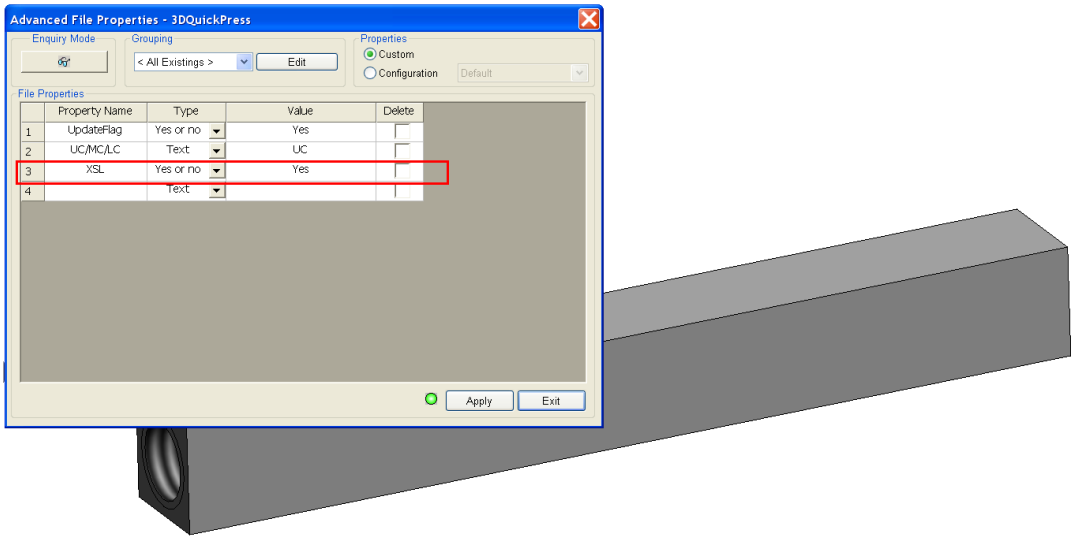
Export Face in User-defined Components

Strip layout faces can be exported during the inserting of User-defined Components. Only faces that are within the bounding box of the User-defined Component will be exported to the new component.

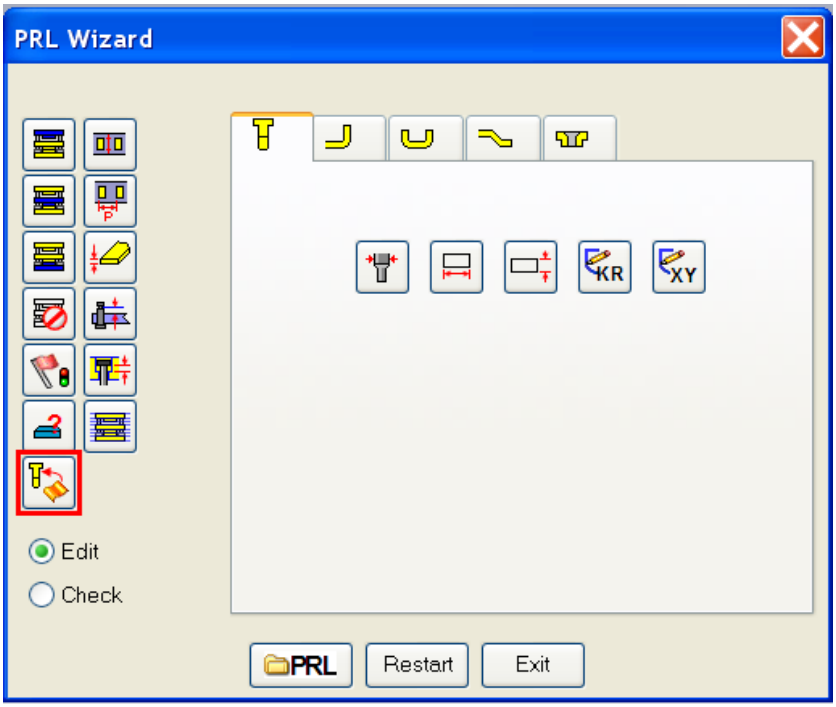
Prepare PRL to support the export strip layout faces when insert:

1. Open components in the PRL that is going to contain the exported strip layout faces when insert to the punch design.
2. Click **File Properties** on the **3DQP Misc. Toolbar**

3. Add the file property **XSL** to the component with **"Yes or no"** type, and set the value to **Yes**.



4. Click **Apply** to update the component file property.
5. Or use the **Export Faces to Part** button of **PRL Wizard** to add the file property.

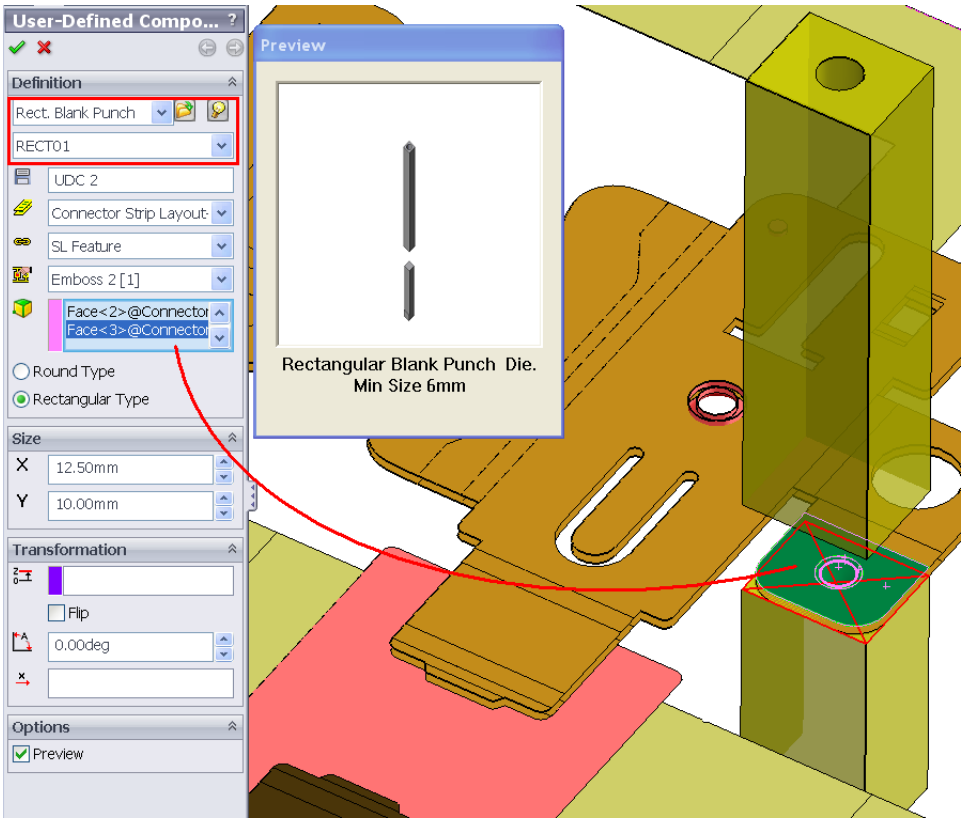


6. Save and close the component.
7. Repeat **Steps 2** to **5** for other components of the PRL that will have the strip layout faces exported.
8. Save and close the RPL assembly after updating all the components.

To insert User Defined Component with exported faces:

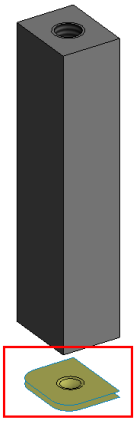
1. Click Insert User-defined Component on the 3DQP PRL Toolbar

2. Select the PRL prepared above (RECT01) and select the strip layout faces for the area of User-defined Component.



3. Click **OK** to insert the User-defined Component.
4. Open the inserted component 0039-UDC2.

5. The selected strip layout faces within the bounding box of the User-defined Component is imported to the inserted component.

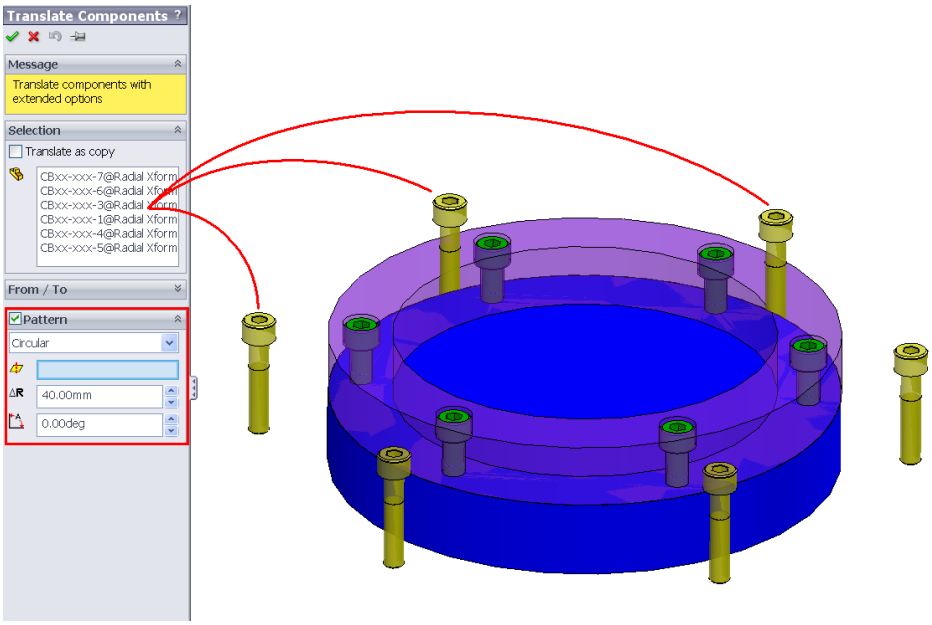


Translate Component in Polar Direction

Translate Component supports the translation of components along the polar direction specified.

1. Click **Translate Components** on the **3DQP Misc. Toolbar**

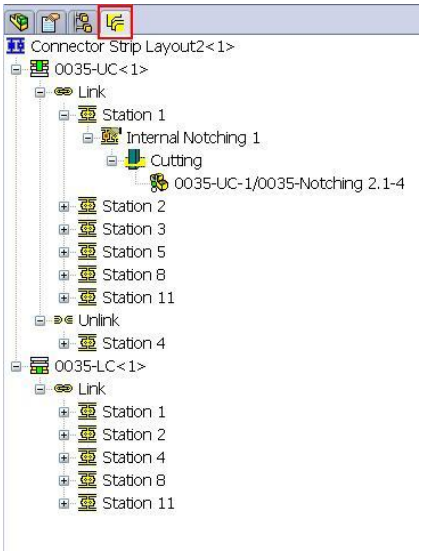
2. Select those six CBxx-xxx screws as components to be translated.
3. Check the **Pattern** input area, select **Circular** as translation type, and enter **40mm** as amount of translation in the polar direction.



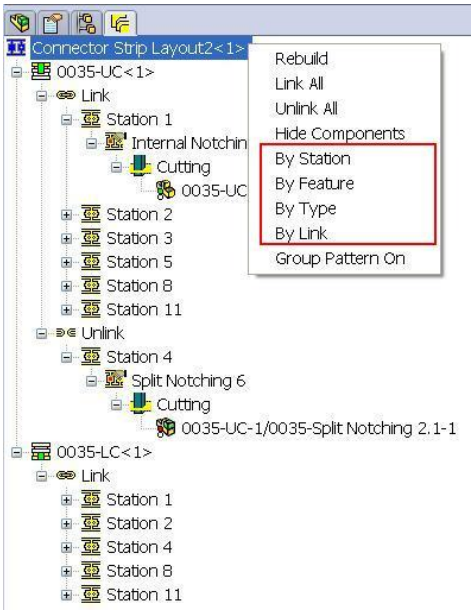
4. Click **OK** to translate the components.

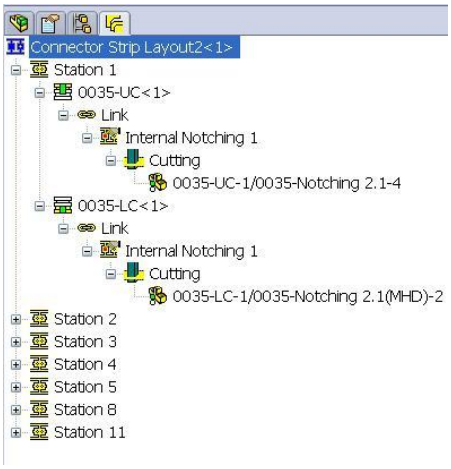
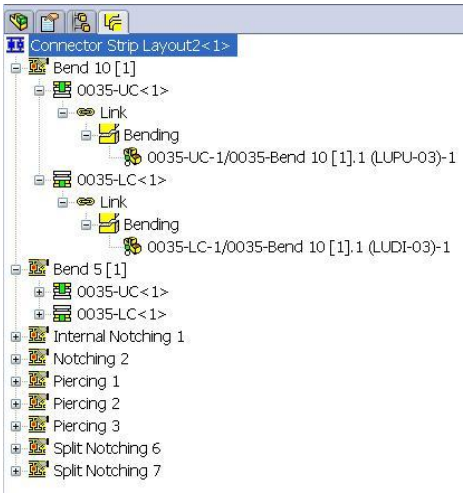
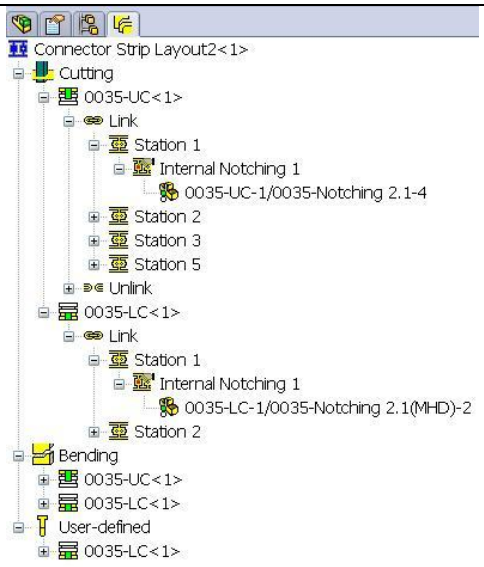
Link Manager

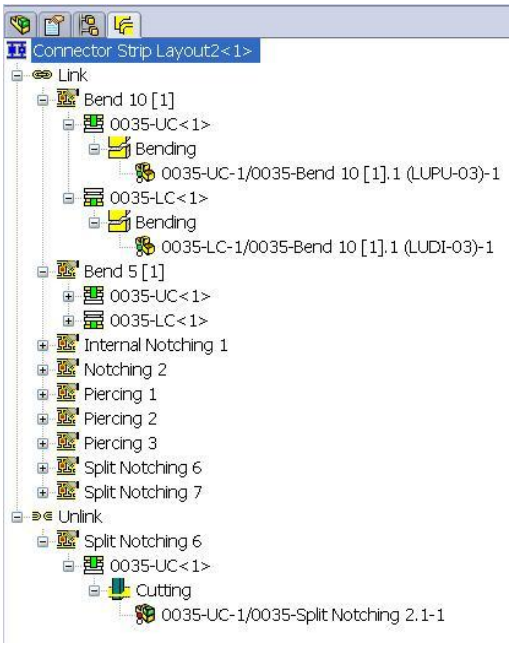
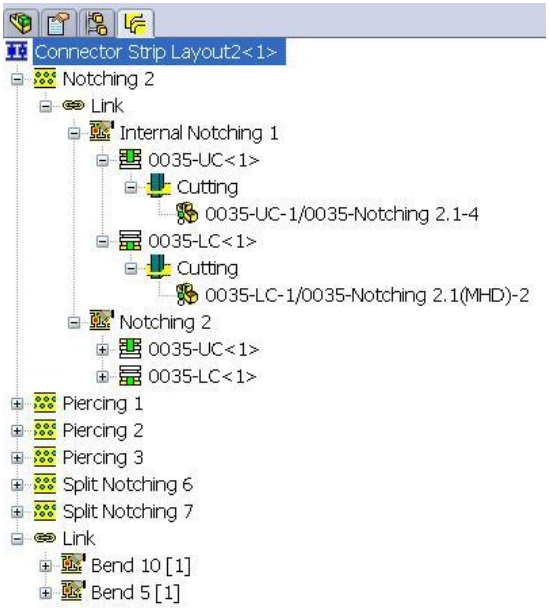
Link Manager is formed automatically after the punch design assembly is created. It lists all the link information between any created component and strip layout features. All icons in the tree have shortcut menu to link/unlink the component.



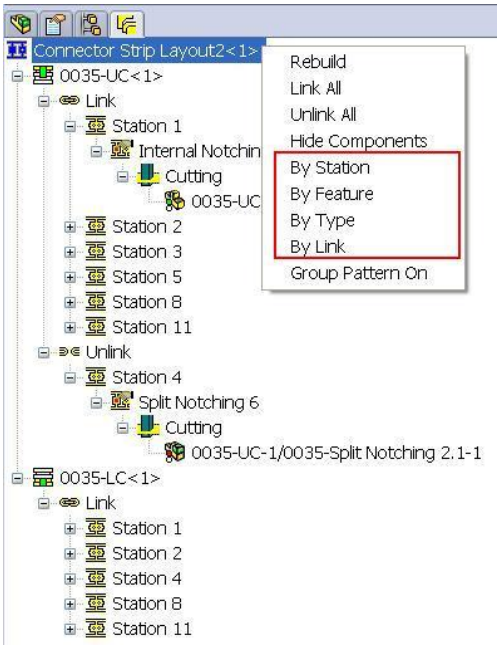
It can list the link information in different sorted type.

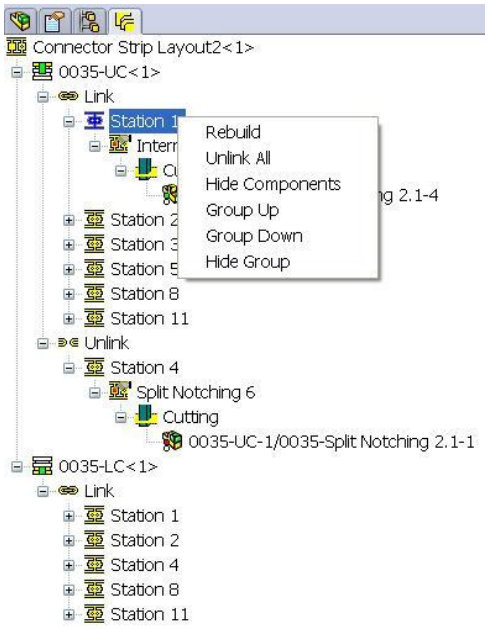


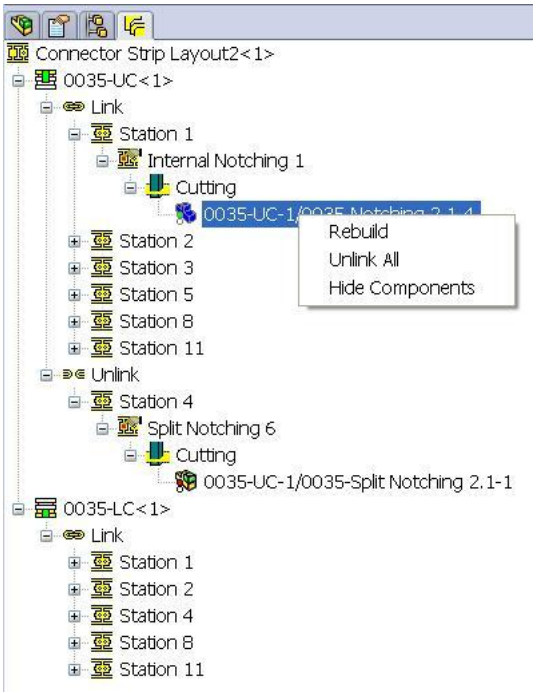
By **Station**	
By **Feature**	
By **Type**	

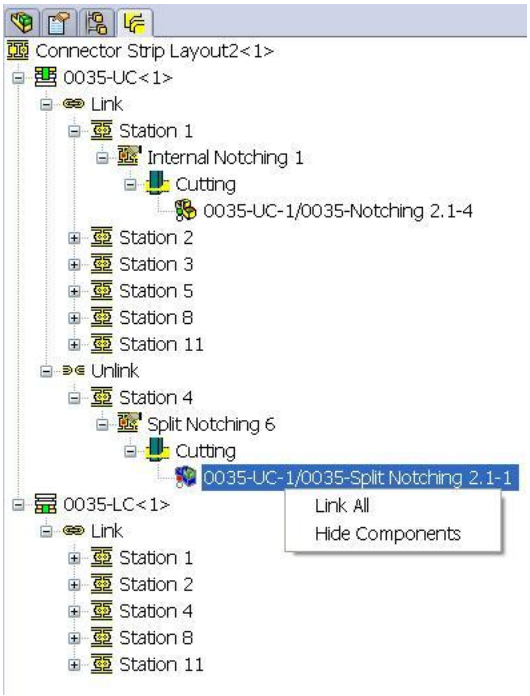
By **Link**	
Group Pattern On	

Most of the functions for link manager are accessed through the shortcuts. Link/unlink and hide/show functions for individual link and component are available for each item in the tree.









To access Link Manager functions:

1. **RMB** on Link Manager Design Tree Item
2. Select any function from the popup menu

Concurrent Design Environment

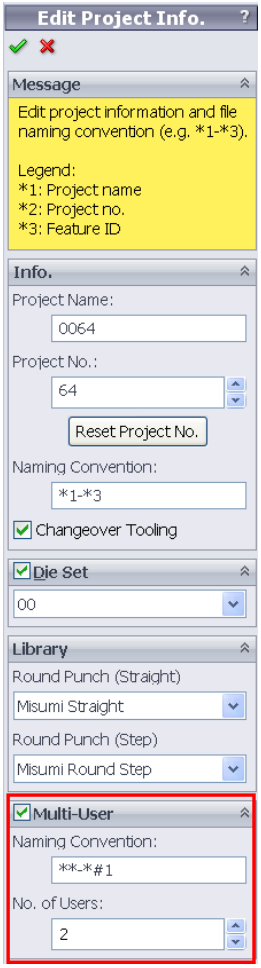
Concurrent design environment is introduced to facilities the collaboration between a group of tool designers who are working on the same project. A concurrent punch design project allows multiple users to work together. This can fully utilizes the expertise of each designer.

A master punch design assembly is created and numbers of sub-punch design assemblies are created to be distributed to different users. Any changes on the sub-punch design assemblies will be reflected on the master punch design assembly. And each sub-punch design assembly will have its own strip layout display and will not affect the display of strip layout on other sub-punch design assembly and master punch design assembly.

To create a punch design with multiple users:

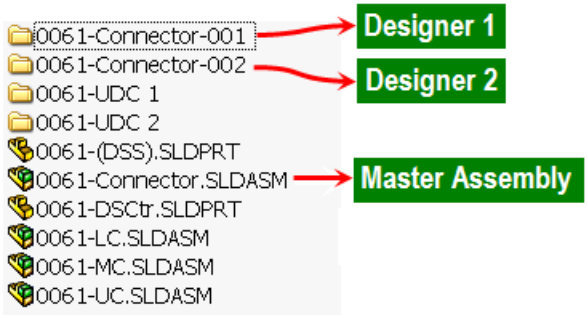
1. Click **Create Punch Design** from the **3DQP Toolbar** 

2. Enable **Multi-User** option and set the **No. of Users** to **2**

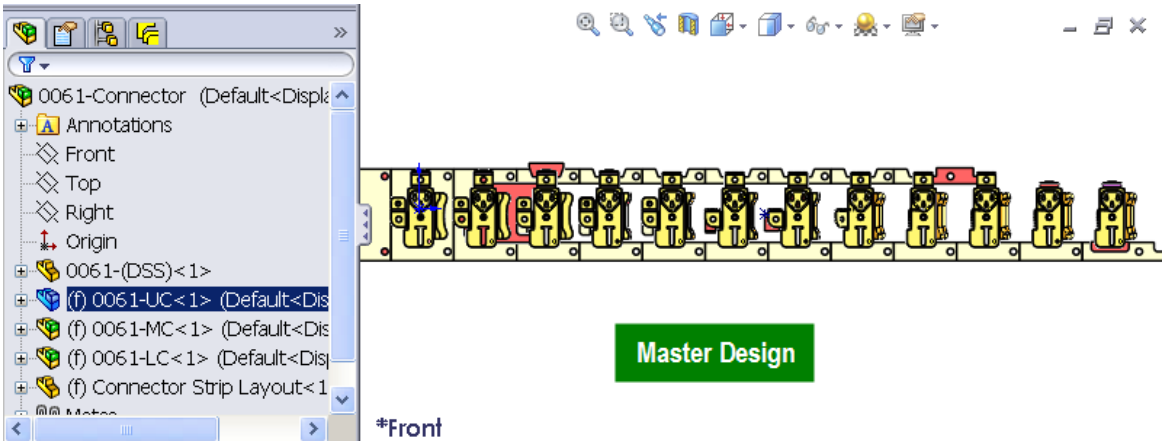


3. Click **OK** for the clearance setting, and the punch design will be created.

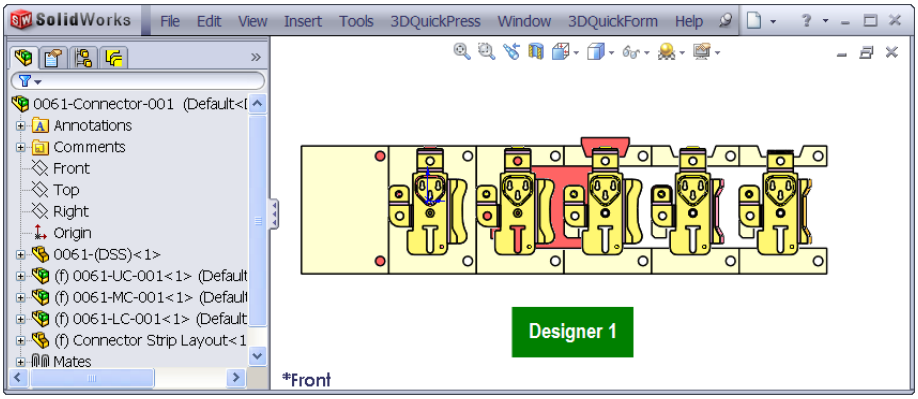
After creating the punch design for concurrent design environment, the project folder for the sub-punch designs will be under the project folder of the master punch design.

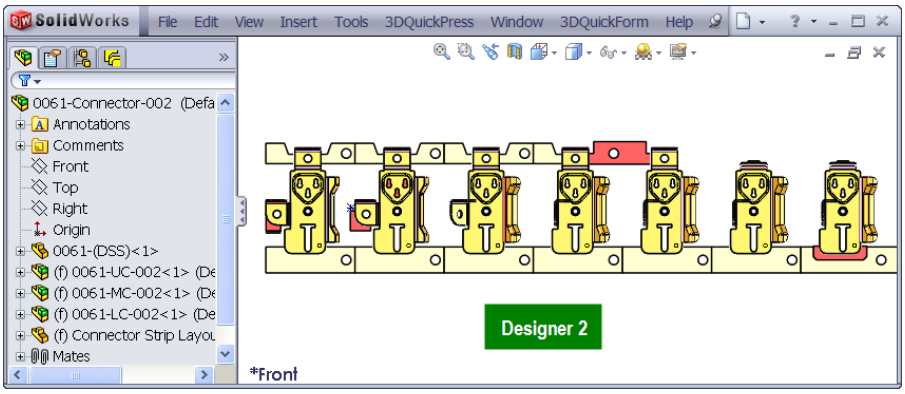


The master punch design will reflect the combined results of all the sub-punch designs. And only the master user will have the right to edit the strip layout.

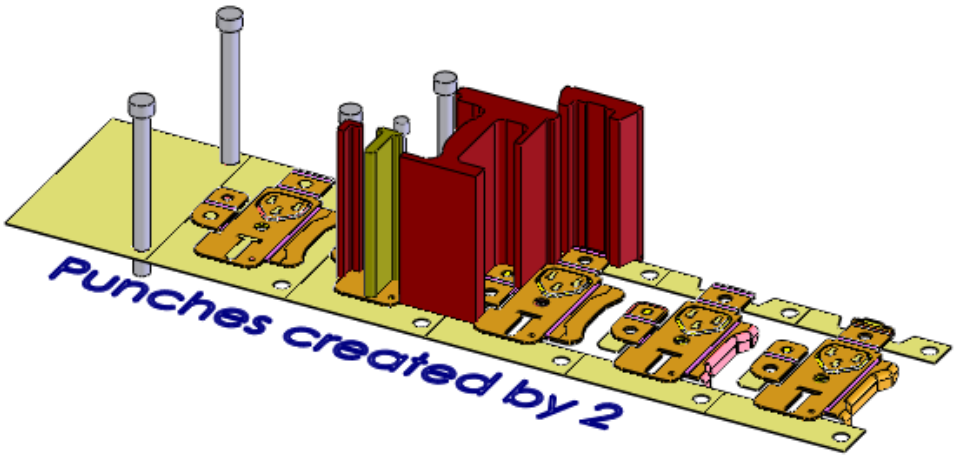


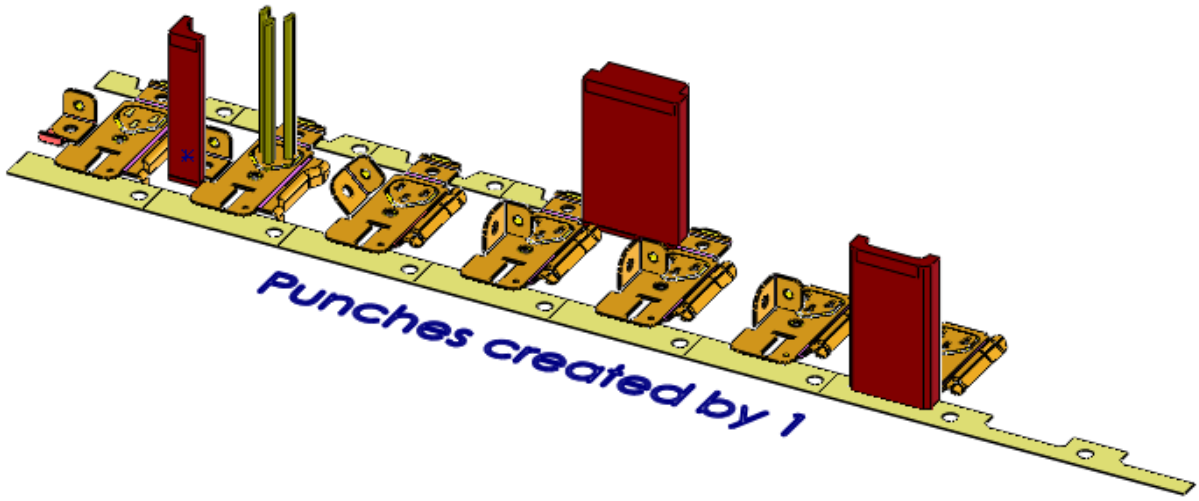
Each individual sub-punch design will be limited to a range of stations that the user is allowed to work on. Display of the strip layout for each sub-punch design can be controlled by individual users.

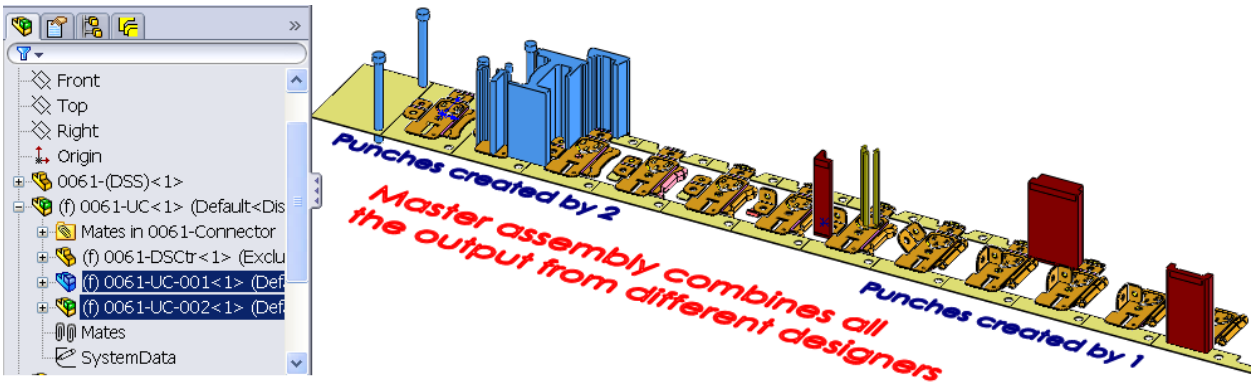




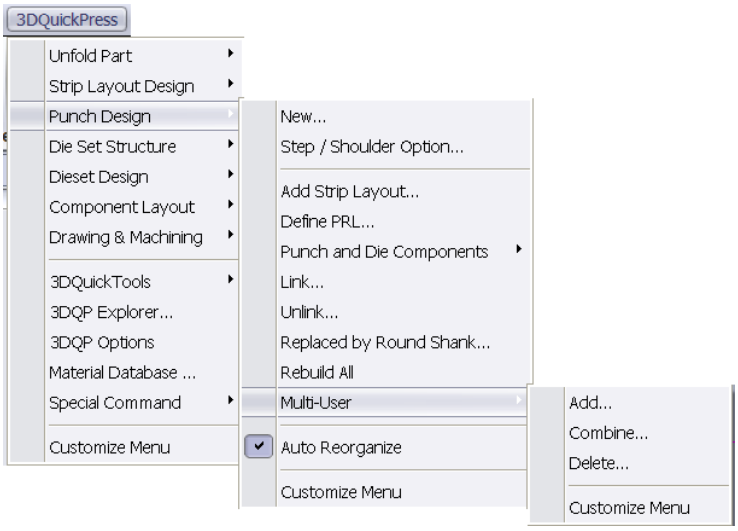
A component inserted in one sub-punch design assembly can only be seems inside that sub-punch design assembly and the master punch design assembly. Other sub-punch design assembly will not see this component.







Concurrent design environment also supports add, delete and combine users at any time during the punch design.

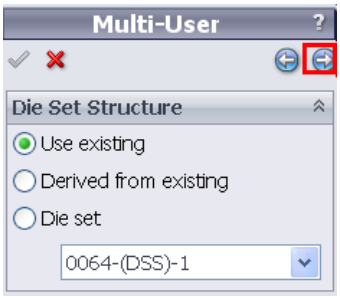


Adding a new user to the master punch design:

1. Click menu item **3DQuickPress->Punch Design->Multi-User->Add…**
2. Enter the **No. of Users** as 1
3. Click the right arrow to proceed

4. Select **Use existing** Die Set Structure

5. Click right arrow to proceed

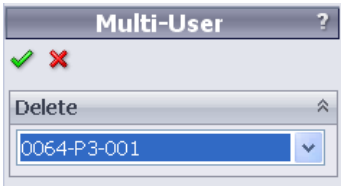


6. Select **Use existing** Die Set Center

7. Click **OK** to add a new user to the master punch design assembly.

Deleting a user from the master punch design:

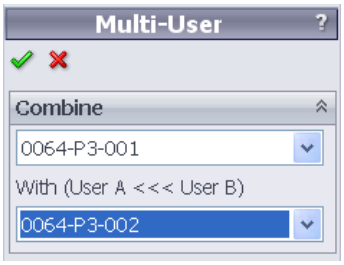
1. Click menu item **3DQuickPress->Punch Design->Multi-User->Delete…**
2. Select the sub-punch design **0064-P3-001**.



3. Click OK to delete the sub-punch design **0064-P3-001** from the master punch design.

Combining two users:

1. Click menu item **3DQuickPress->Punch Design->Multi-User->Combine…**
2. Select the two sub-punch designs **0064-P3-001** and **0064-P3-002** that are going to be combined.



3. Click **OK** to combine **0064-P3-002** into **0064-P3-001.**

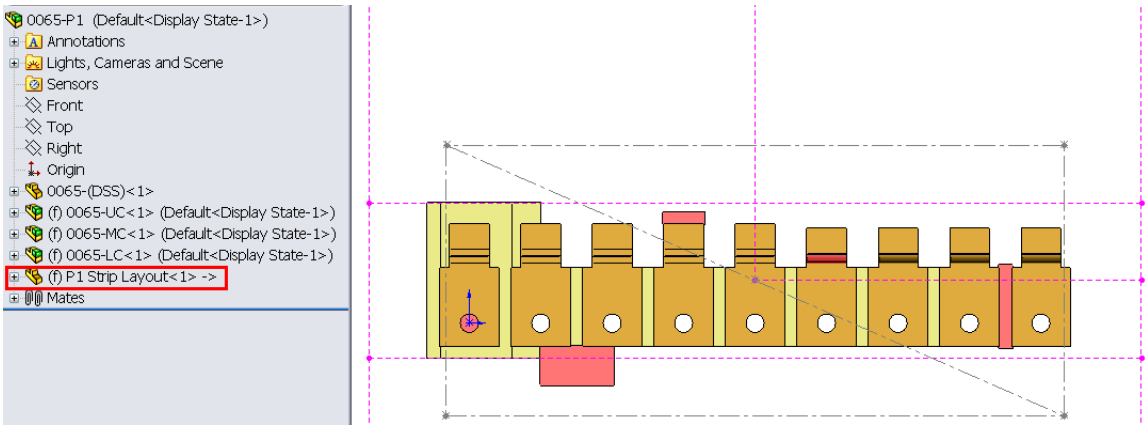
Changeover Tooling Manager

Changeover tooling is the common practice in this industry to produce similar parts by the same tooling through proper tool setup. 3DQuickPress Version 5 enables the design of change over tooling easier and produce tooling setup automatically.

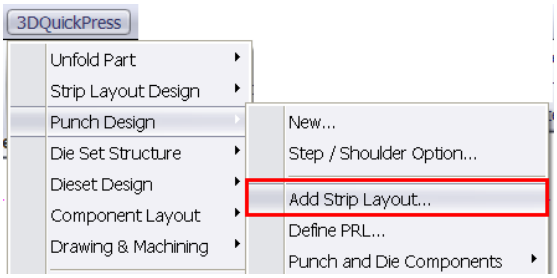
Punch design supports inserting multiple strip layouts. After creating the punch design from the first strip layout, strip layouts with similar parts can be inserted to this punch design to design the changeover tooling.

To insert a second strip layout:

1. Create the punch design using the first strip layout **P1 Strip Layout**.

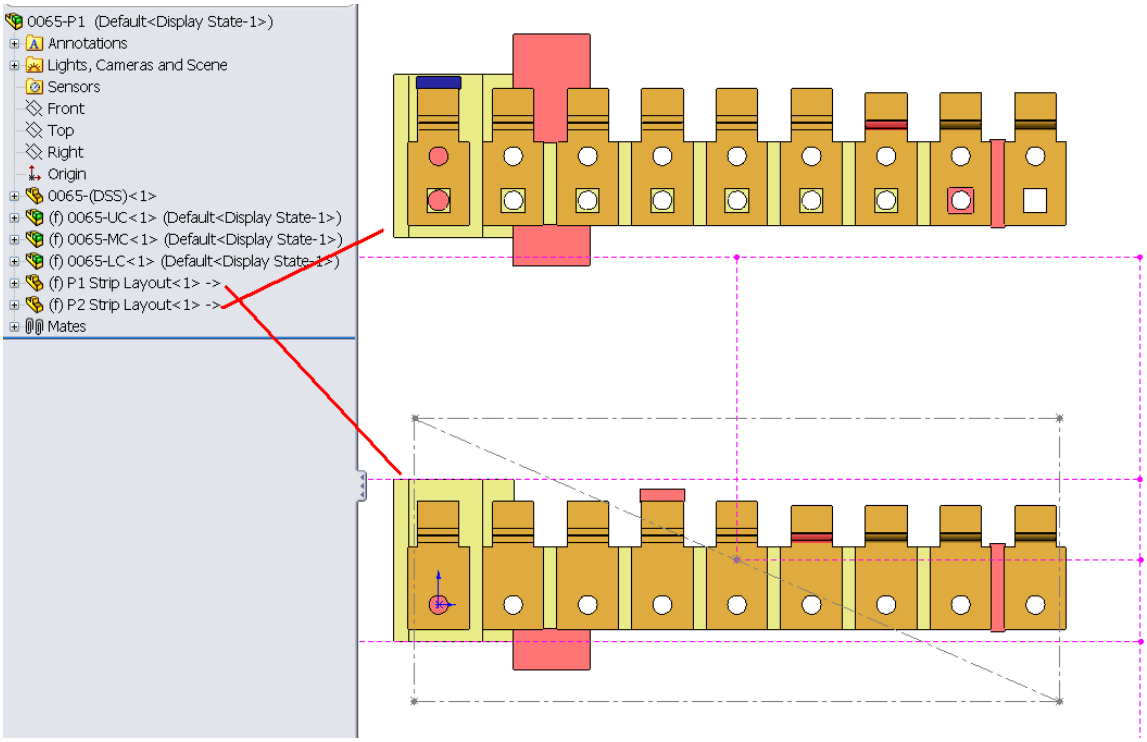


2. Select from menu **3DQuickPress->Punch Design->Add Strip Layout…**



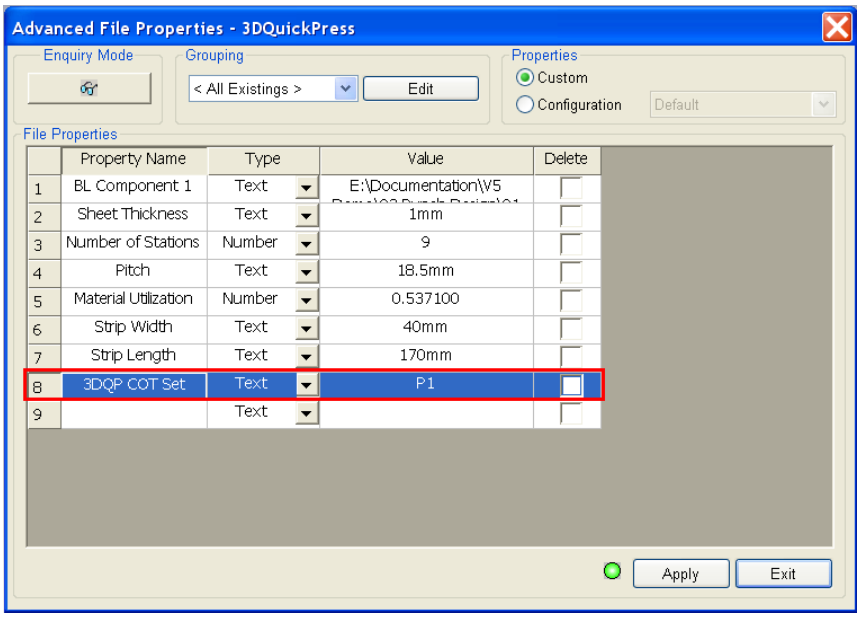
3. Browse for the second strip layout **P2 Strip Layout.**

4. Enter the **Y** offset as **100mm**, and select the option **Use existing** Die Set Structure.
5. The second strip layout **P2 Strip Layout** was inserted to the same punch design with an offset of **100mm** in **Y** direction.



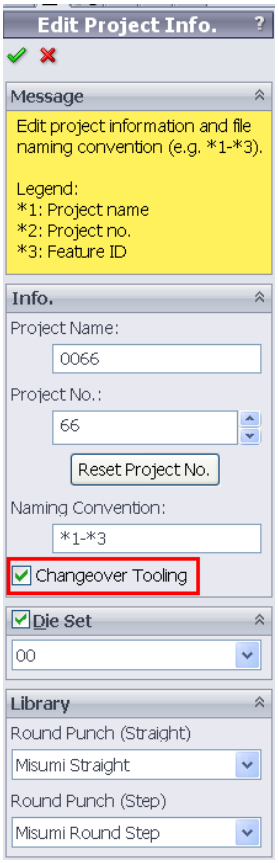
Set up changeover tooling modules by strip layout:

1. Add the file property **3DQP COT Set** with the value **P1** to the strip layout **P1 Strip Layout**.



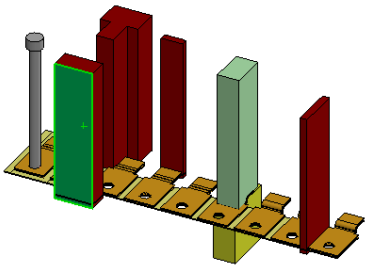
2. Add the file property **3DQP COT Set** with the value **P2** to the strip layout **P2 Strip Layout**.

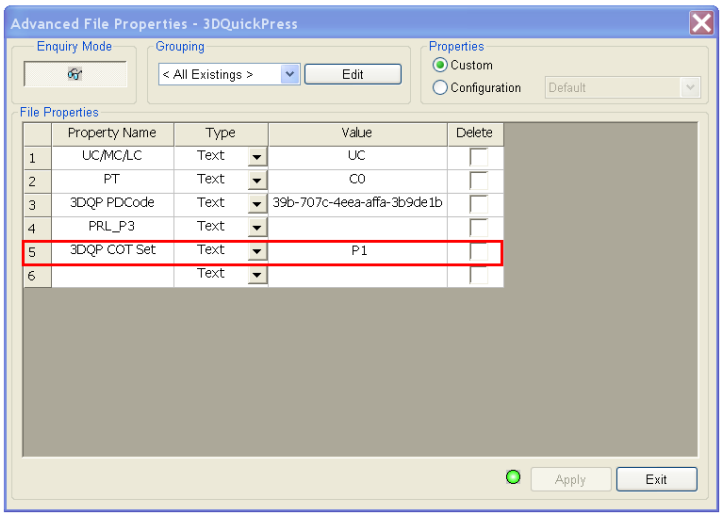
3. Create punch design using **P1 Strip Layout**.
4. Enable option **Changeover Tooling** when create punch design.



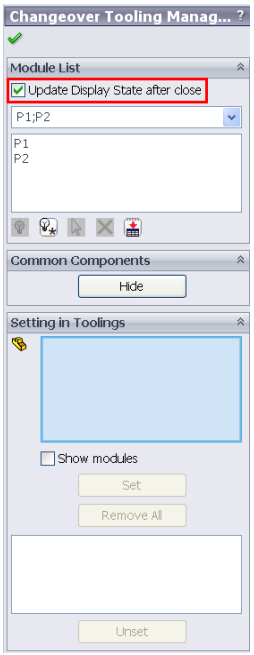
5. Add **P2 Strip Layout** to the created punch design.
6. Any components created with links to **P1 Strip Layout** features will have the file property **3DQP COT Set** valued as **P1**.

7. Any components created with links to **P2 Strip Layout** features will have the file property **3DQP COT Set** valued as **P2**.





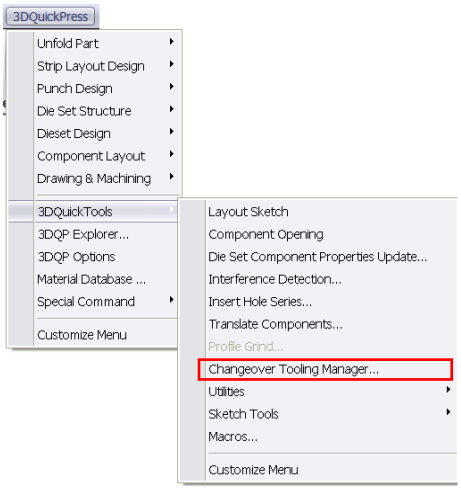
8. Select menu item **3DQuickPress->3DQuickTools->Changeover Tooling Manager…**
9. The list of the different modules is set up according to the file property of the strip layouts.



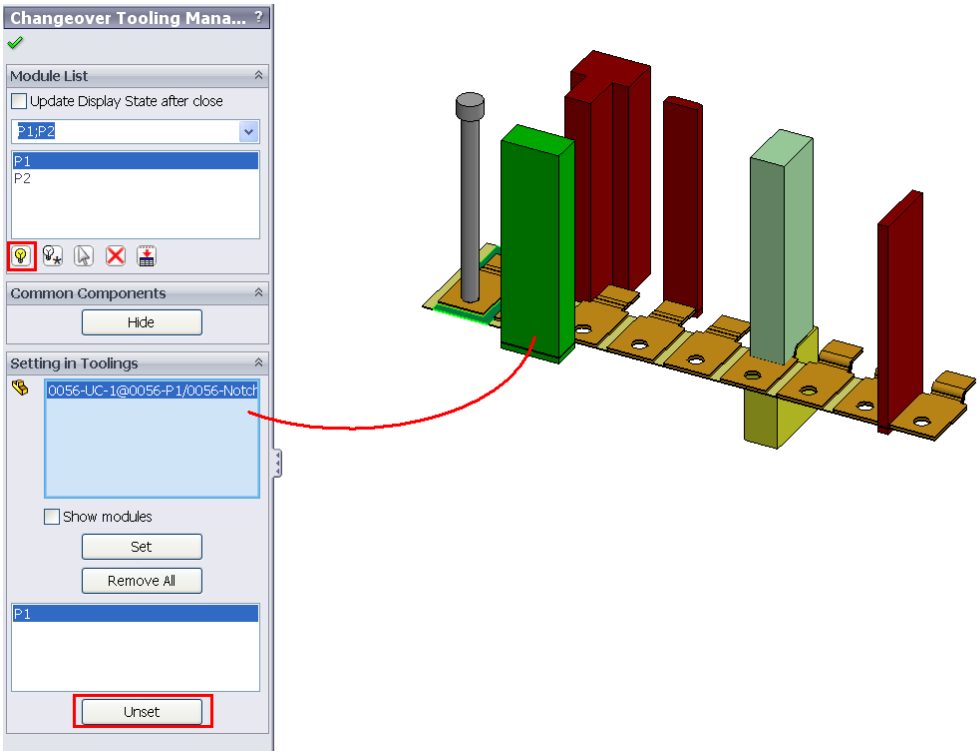
10. Enable the **Update Display State after close** option.
11. Click **OK** to close the **COT manager**.
12. Display state of different module set is created.

To change a component from module P1 to P2 using COT manager:

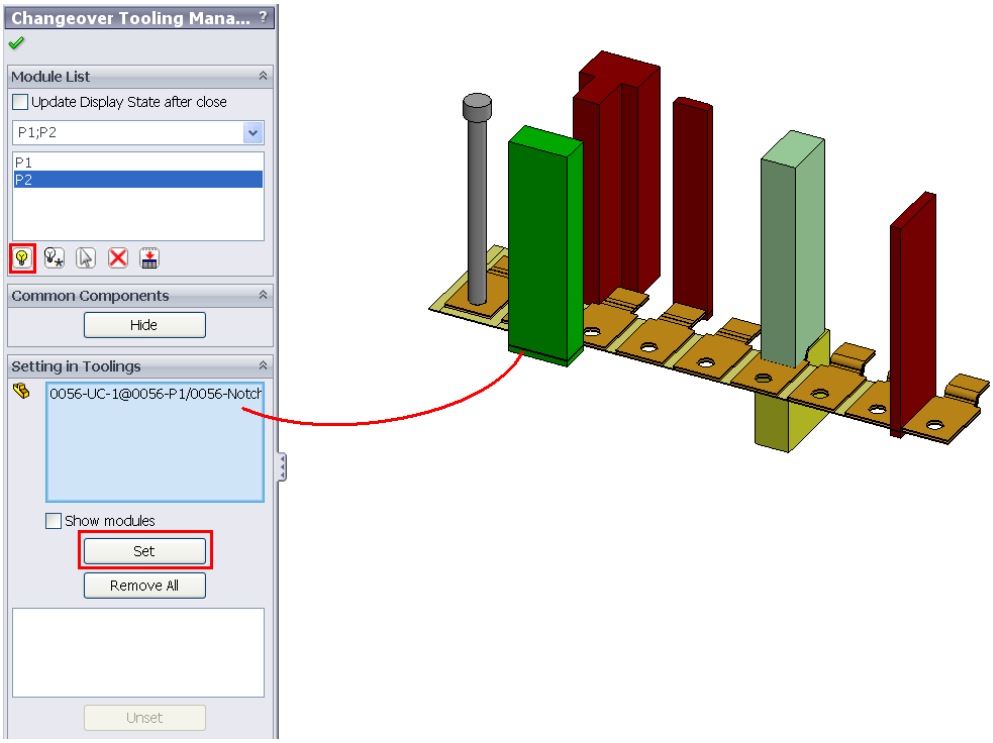
1. Select menu item **3DQuickPress->3DQuickTools->Changeover Tooling Manager…**



2. Select **P1** form the **Module List**.



3. Click **Show**  to show only components and strip layout of **P1**.
4. Select the cutting punch **0056-Notching 2.1**.
5. Select **P1** in the list of modules under the **Setting in Toolings** group.
6. Click **Unset** to remove the cutting punch form the module **P1**.
7. Select **P2** form the **Module List**.



8. Click **Show**  to show only components and strip layout of **P2**.
9. Show and select cutting punch **0056-Notching 2.1.**
10. Click **Set** to add the cutting punch to module **P2**.
11. Click **OK**  to close the **COT Manager**.

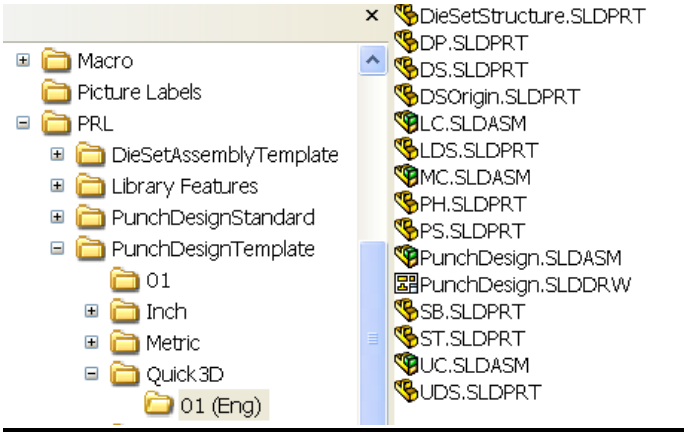
Die Set Design

3DQuickPress MKII

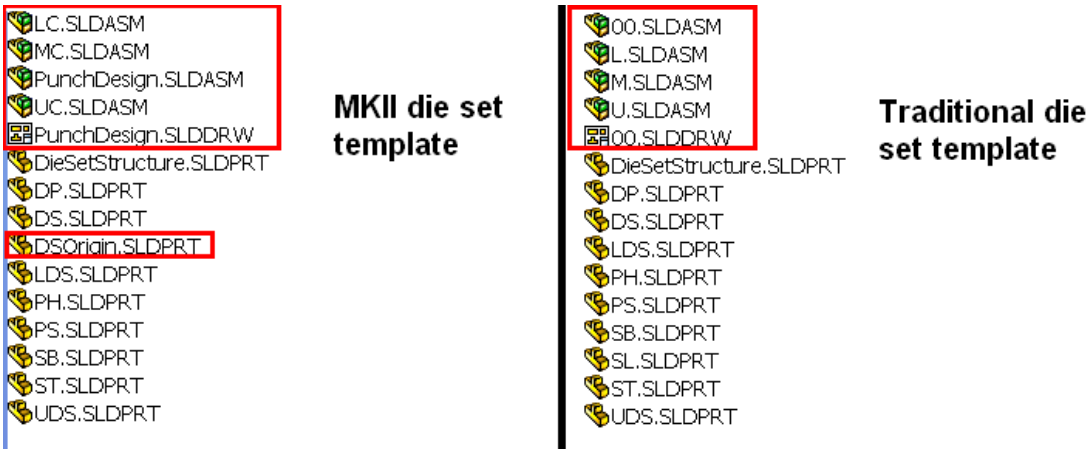
The 3DQuickPress MKII scheme allows users to create a punch design and die set without unfolding a part and designing the strip layout. From a simple SolidWorks part document, user can create punch design assembly with die plates and die set components directly. This scheme is particularly suitable for stage tooling design.

Prepare the die set template for 3DQuickPress MKII scheme:

1. The template folder of the MKII scheme should be located under the folder \\PRL\PunchDesignTemplate\Quick3D.

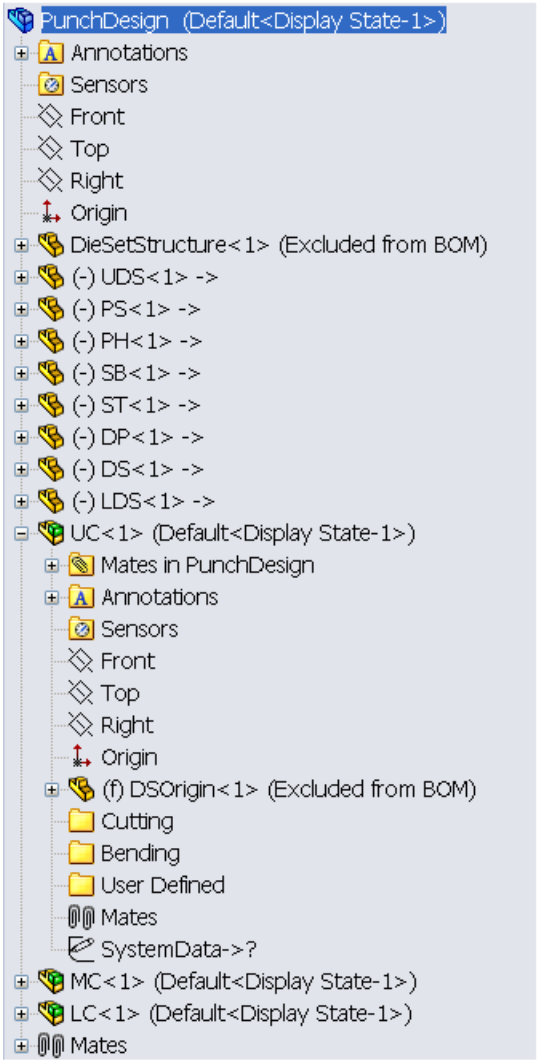


2. The template assembly name for the MKII scheme is “PunchDesign.SLDASM” instead of using the name of the template as the assembly name as a normal die set template.
3. Same as the template drawing file, the drawing file name should be “PunchDesign.SLDDRW”.
4. The U/M/L assemblies in the traditional die set template should be replaced by UC/MC/LC assemblies.

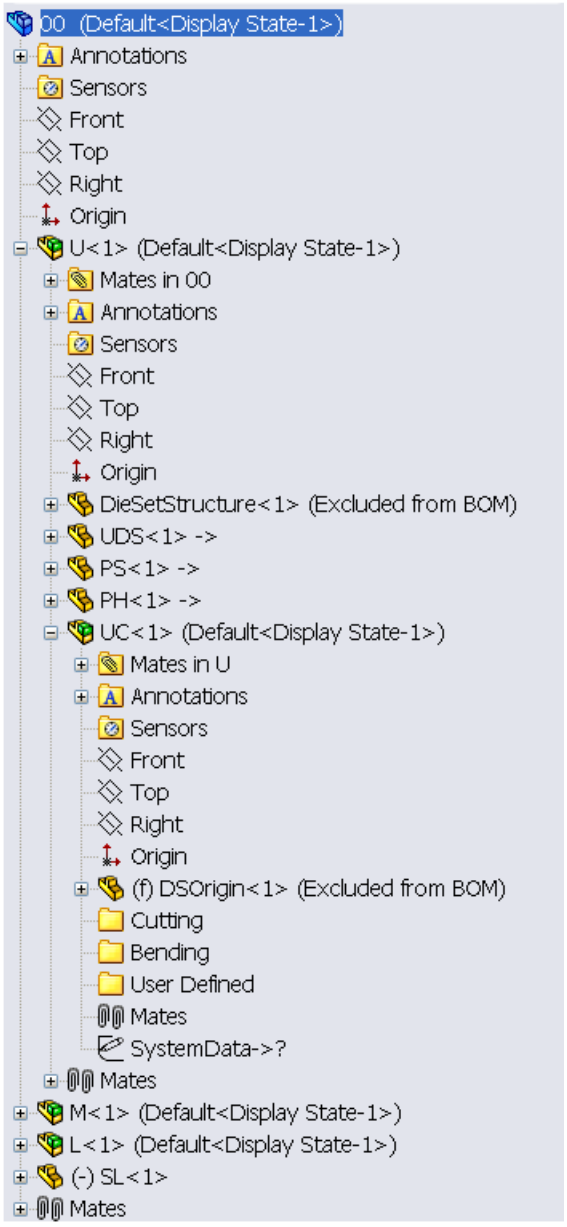


5. A sketch part DSOrigin.SLDPRT is added to locate the center of the die set in the MKII template.
6. The assembly structure is also different between a normal die set template then a MKII die set template. The die plates parts in the MKII die set template are all listed under the PunchDesign main assembly, where normal die set template should be under the U/M/L assemblies.
7. In normal die set templates, the UC/MC/LC assembly should be under the U/M/L assembly respectively. In the MKII die set template, only the UC/MC/LC assembly are presented.

MKII Die Set Template structure



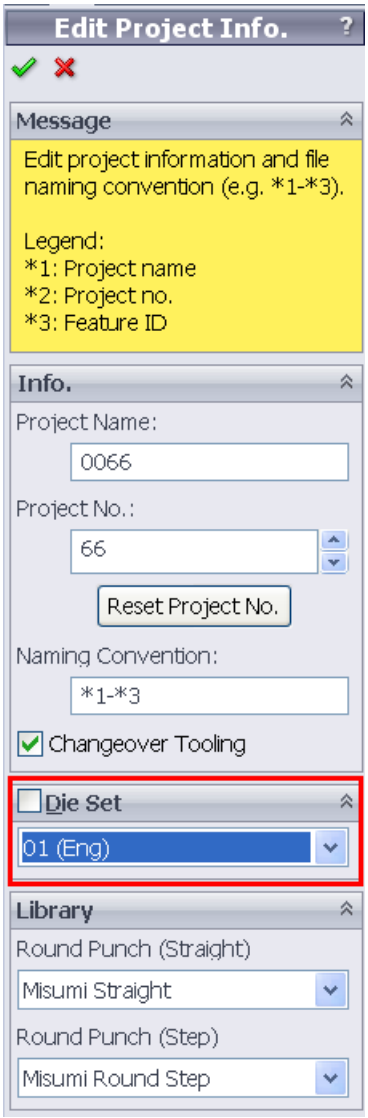
Traditional Die Set Template structure



Creating the MKII project for die set design:

1. Click **Create Punch Design** from the **3DQP Toolbar** 

2. Uncheck **Die Set** Option.



3. Select the MKII die set template is the die set selection list.
4. Click **OK** to create the die set assembly with the MKII template.

Detailing

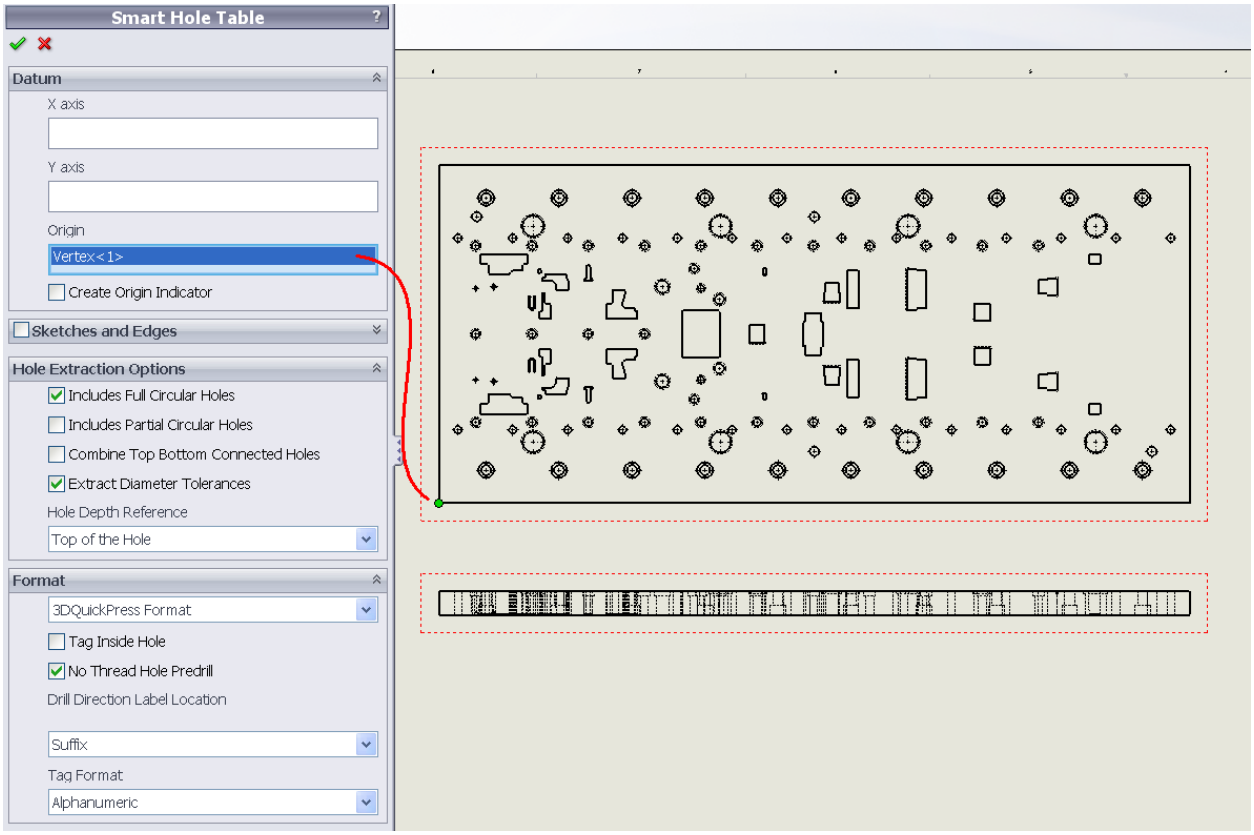
Smart Hole Table

The Smart Hole Table of 3DQuickPress can automatically identify circular cutting holes on a part and create the hole table. The Smart Hole Table is able to handle all difficult cases. It can select number of decimal places for the hole position and size according to the Mfg. Color setting, and it can recognize far side hole automatically. It also supports fillet and broken holes. Format of the hole table can be customized by employing a user defined macro.

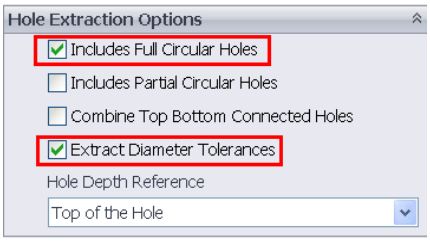
To create 3DQuickPress Smart Hole Table:

1. Click Smart Hole Table  on the 3DQP Drawing & Machining toolbar

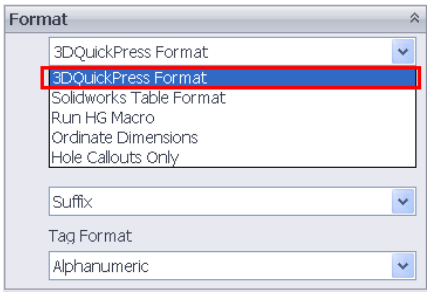
2. Select a vertex as the origin



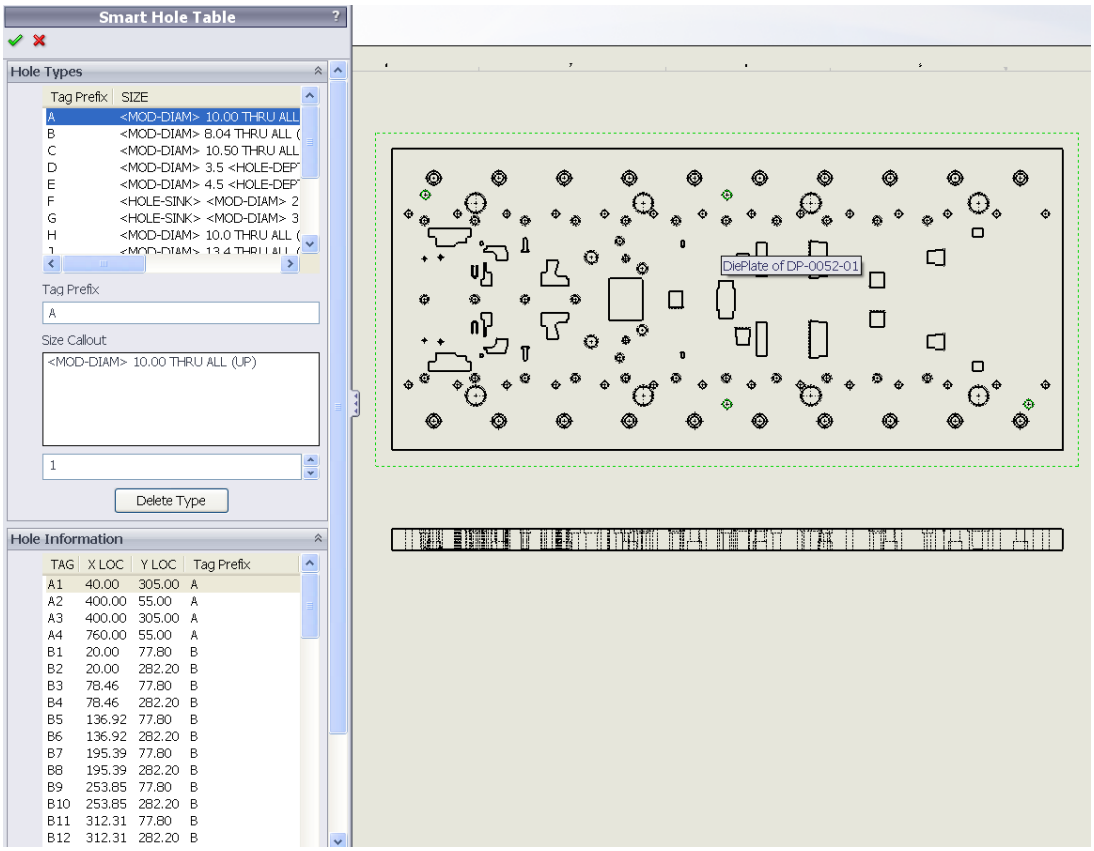
3. Select the hole extraction options **Includes Full Circular Holes** and **Extract Diameter Tolerances**.



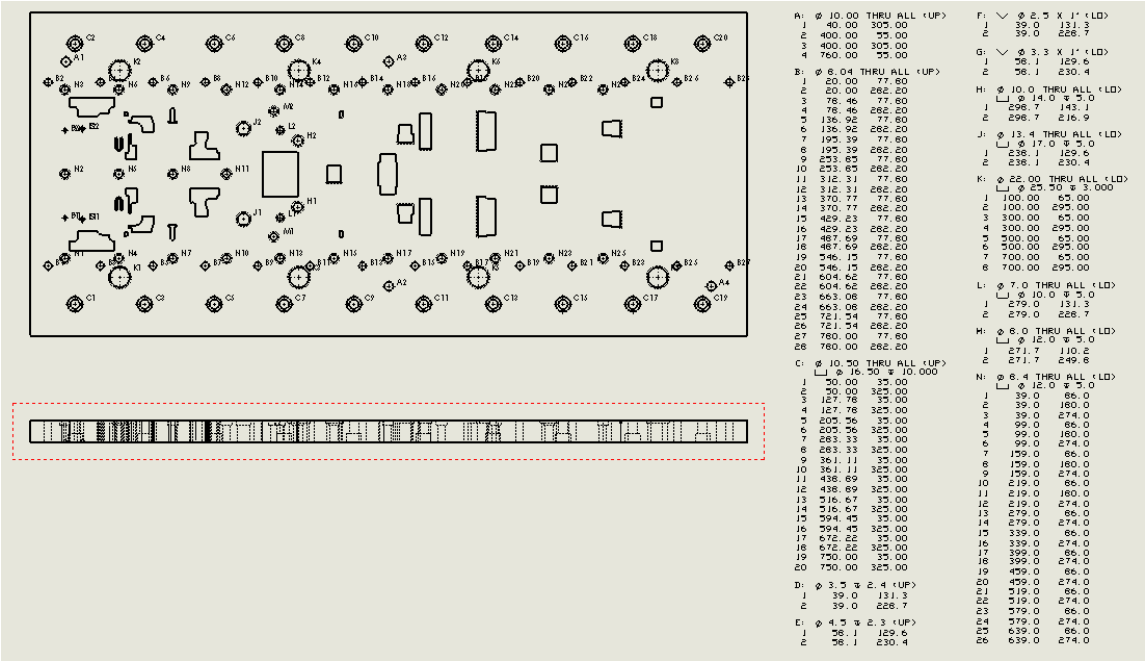
4. Select hole table format **3DQuickPress Format**.



5. Click **OK** to proceed.
6. Select the type of hole, and the hole will be highlighted in the drawing view. The hole callout and the hole location are listed. The hole callout can be edited to the desired format.



7. Click **OK** to create the hole table.

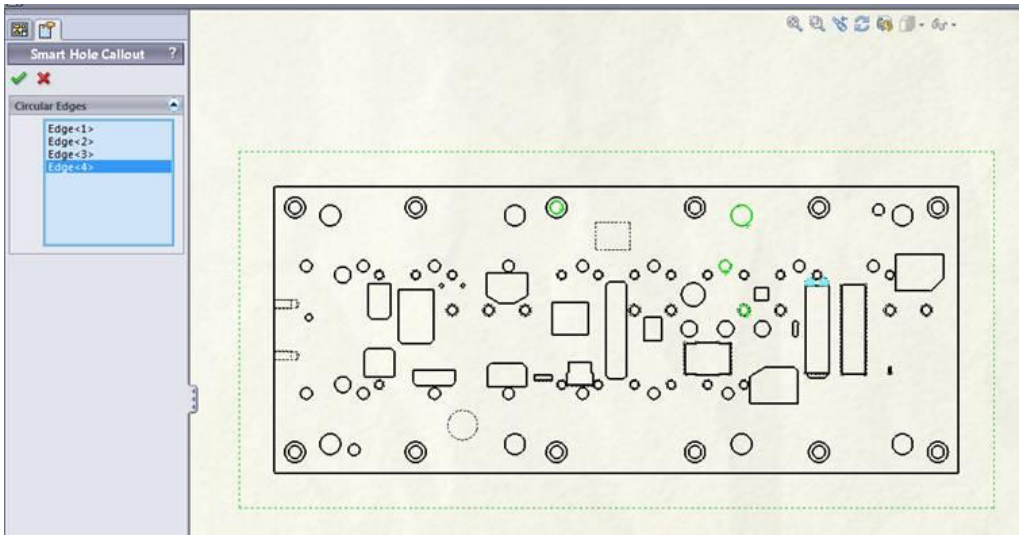


Smart Hole Callout

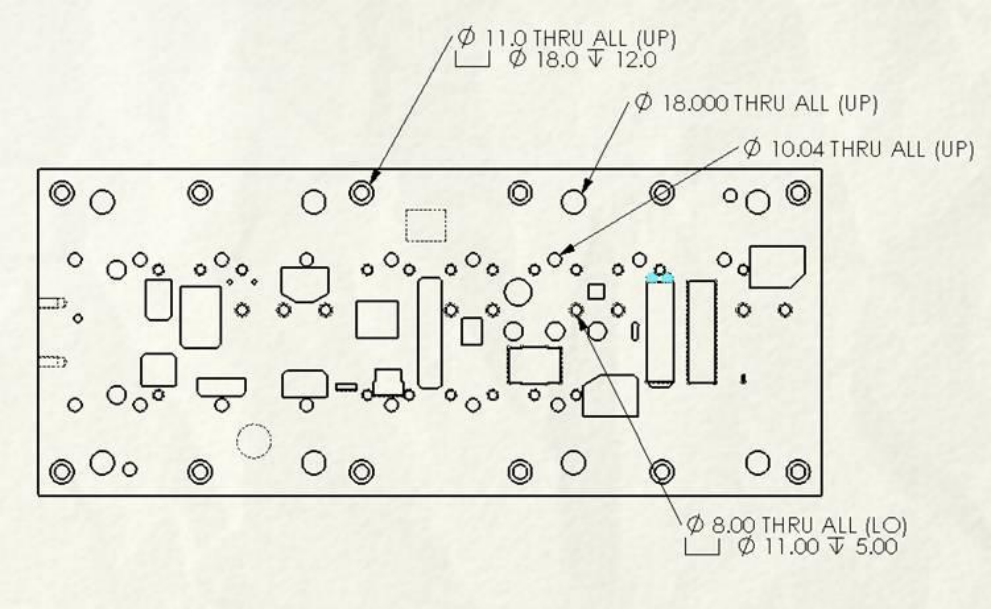
Smart hole callout is similar to the smart hole table but it just dimension one single hole.
To create hole callout:

1. Click **Smart Hole Callout** on the **3DQP Drawing & Machining toolbar**

2. Select the edge of the cutting holes, then click **OK** to finish.



3. Hole Callout created with precision information. Far side and near side hole is also identified.

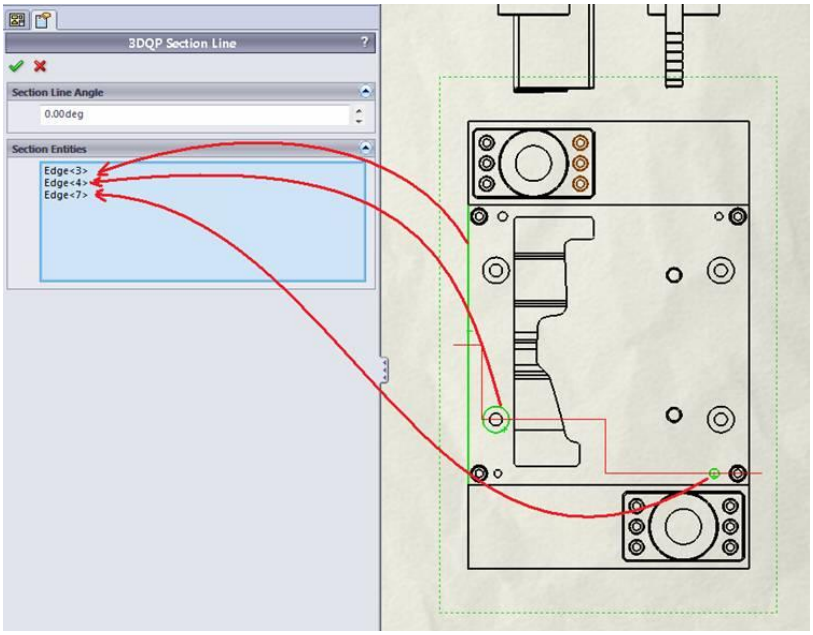


Section Line Tool

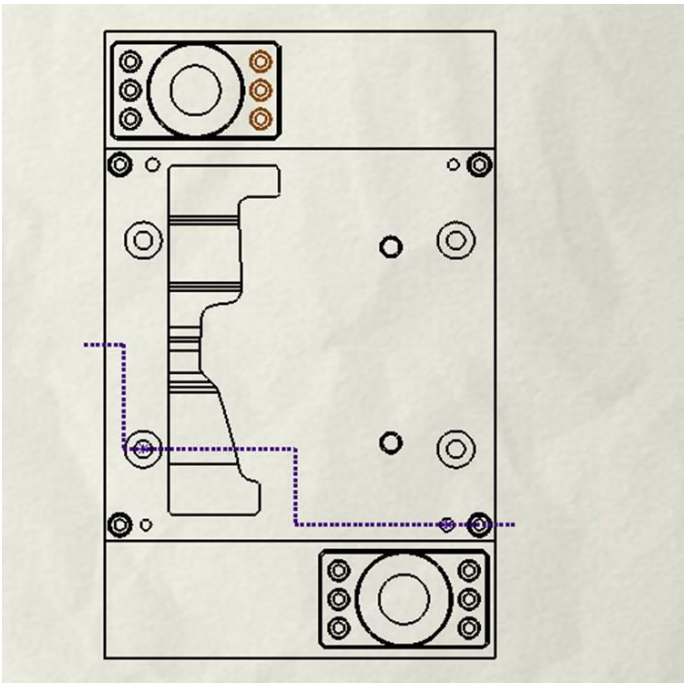
To create section lines in a Drawing SolidWorks Document.

1. Click **3DQP Section Line** on the **3DQP Drawing & Machining toolbar**

2. Select section entities.



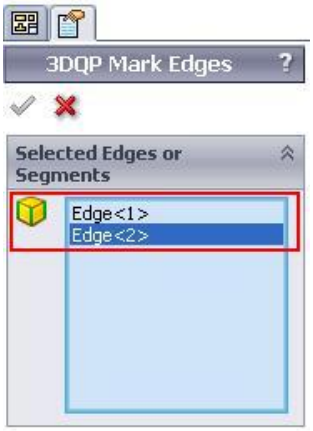
3. Click  to finish



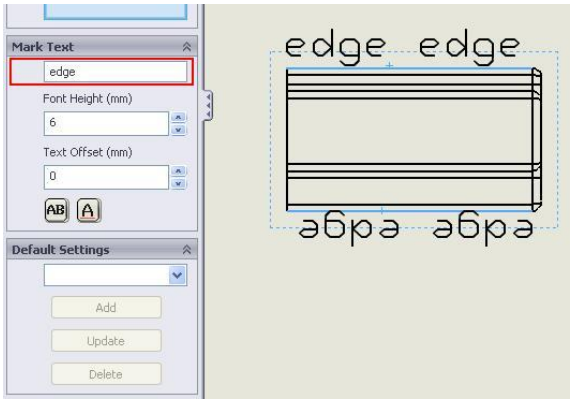
Edge Annotation

1. Click **3DQP Mark Edges**  on the **3DQP Drawing & Machining toolbar**

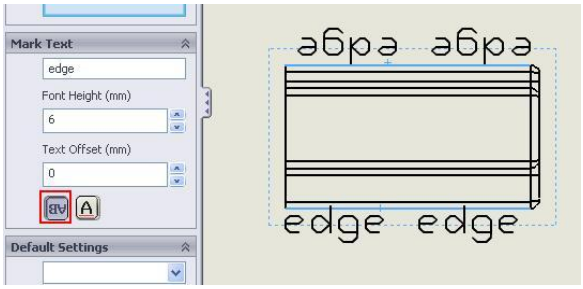
2. Select edges to be marked.

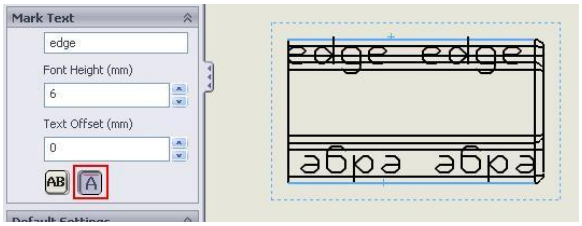


3. Input text.



4. Adjust text direction and location by filling in the Font Height and Text Offset and by clicking  and 





5. Click **OK**  to complete.