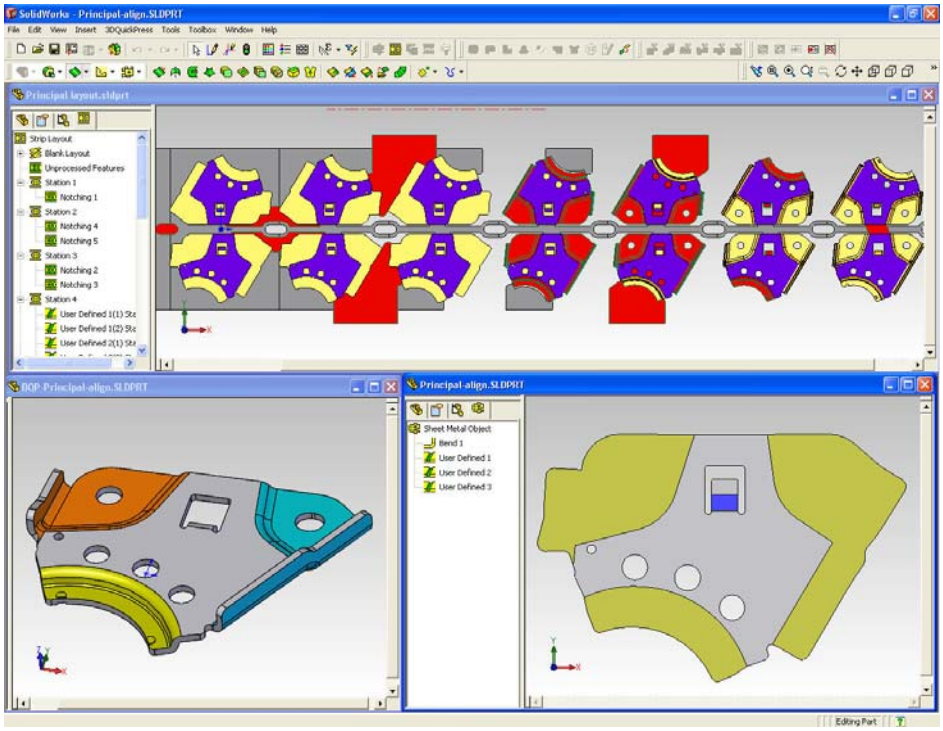
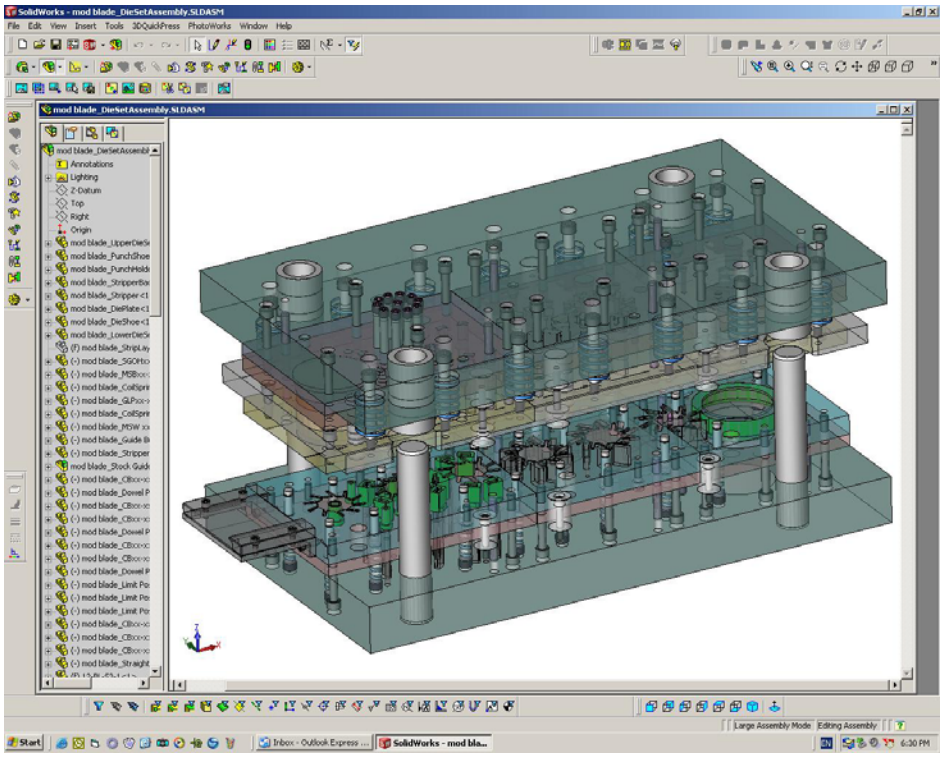
3D QUICKTOOLS LIMITED

3DQuickPress V5 –Training Manual





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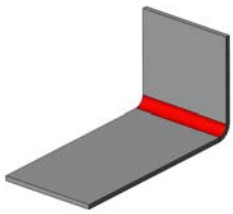
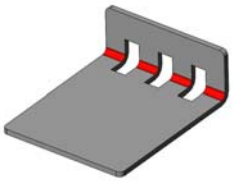
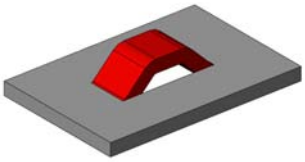
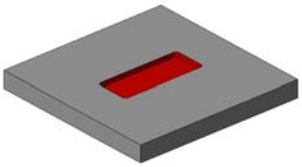
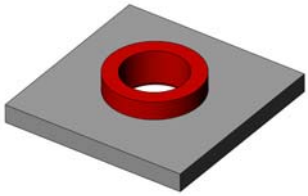
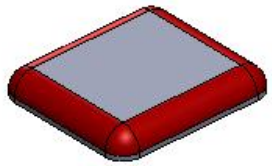
Tutorial 1. Automatic unfolding of parts

Note: For imported parts, it is highly recommended to translate the part closest to the part origin and with press punch direction parallel with Z-axis of the global coordinate system.

These Metal features can be unfolded automatically

	Linear Bend
	Compound Bend
	Lancing
	Embossing
	Round Extrusion

	Coining

Please note throughout this document “RMB” means Right Mouse Button.

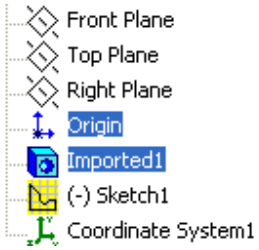
Align Imported Part

1. Open **\training\unfold1\BadOrientation.x_t** by clicking **File**, **Open**, select **ParaSolid (x_t)** as **File Type**.

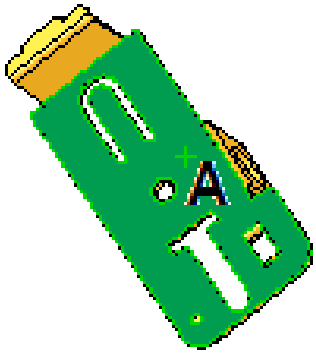
2. Switch between **Front**, **Right**, **Top** view by clicking icon in Standard View Toolbar or the 3DQP View Tool bar. . 

Notice how the part is not orientated to these standard views.

3. CTRL-Select **Origin** and **Imported1** from Feature Manager. Right Click and select  Zoom To Selection. Notice how far away from the part is located to the origin.



4. Click  Insert, Sketch and select Face A..

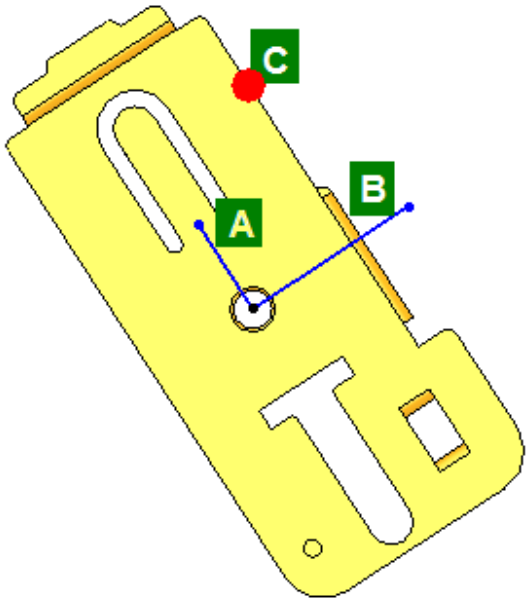


5. Hit **Space Bar** and then select **Normal To**.

6. Click  Line and move cursor over the edge as shown to wake up the center.

7. Click on the woke up center point and click at **A** to create a line segment.

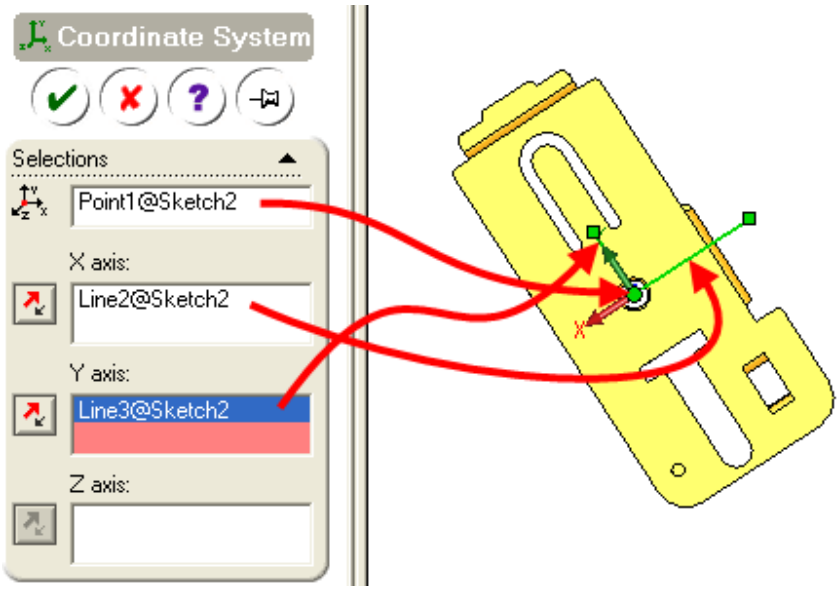
8. Click on the center again and click at **B**.



9. Click  **Add Relation**, select **Line A** and **Edge C,** click  **Parallel,** select **Line A** and **Line B**, click  **Perpendicular.**

10. Click  Exit sketch or **green check** mark in upper right corner of viewport.

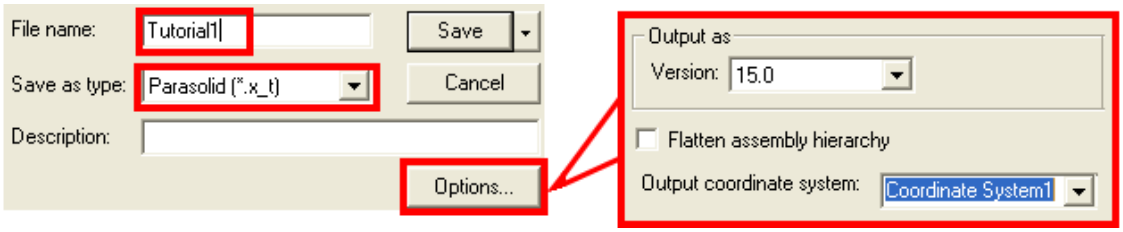
11. Click **Insert**, **Reference Geometry**,  **Coordinate System**, fill in the property manager as follows.



12. Click **OK** to finish

Loop Export Method (SolidWorks Method)

13. Click, **File**, **Save As**, input **unfold1** for file name, select **Save as type** Parasolid (*.x_t), Click **Options** and select **Coordinate System1**



14. Click **Save** to finish and then **Minimize** the badOrientation.sldprt part window.

15. **File/Open** file unfold1.x_t and check the orientation with the view tools. The origin of the part is now equal to the center of the hole.

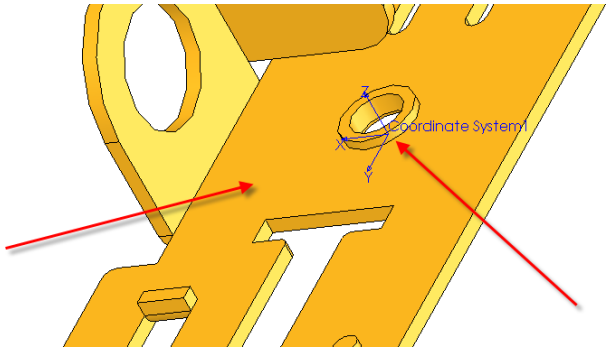
Translate to Origin Method (3DQuickTools Method)

16. Switch windows to the "**badorientation**" part to try one other method. Use CTRL-TAB or Window menu item to select from listed open files.

17. Hide Sketch1 with RMB selection

18. Select "**Translate to World CSYS**" icon

19. Select part and Coordinate System1, then pick OK.

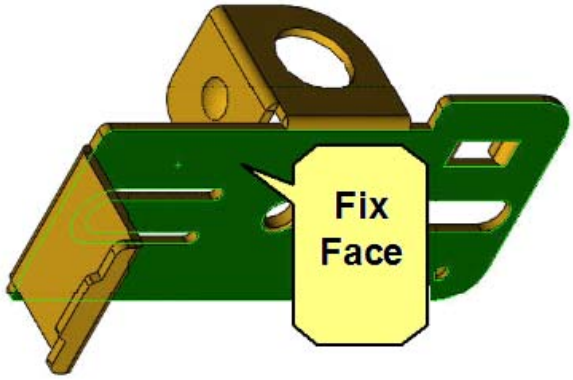


20. Select " **F** " to Fit View and check the orientation with the view tools

21. Save part as **unfold1** in **\training\unfold1** folder

Unfold Part

1. Click **Unfold Part** icon to unfold the part.
2. Select the **fix face** as show.
3. Click **OK** to accept all default values and save the part to **Training\unfold1\unfold1.sldprt.**



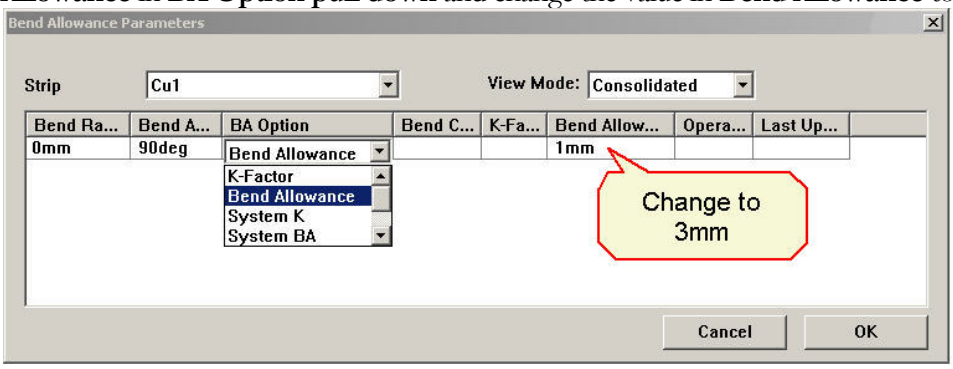
Modify diameter of Round Extrude pre-piercing

4. Right click **Round Extrude 1** from the **Unfold Design Tree** and select **Edit parameters**, change the Unfold diameter to 2mm.
5. Right Click on **Sheet Metal Object** in **Unfold Design Tree** and select **Unfold All**

Modify Bending allowance

3DQuickPress has 3 tables to calculate bending allowance. You can use **3DQuickPress**, **Material Database** from 3DQuickPress pull down menu to modify them. If you have plenty of data to input, you can use Microsoft Access to customize your databases, the file is located in **<3DQuickPress Installation Directory>/Database/PDDS(*).mdb**. 3DQuickPress will try to get the bending allowance from the table **Specific Bend Parameters.** If no matching bends are located, the bending allowance will be interpolated from the table **System Bend Parameters**

6. Right Click on **Sheet Metal Object** in **Unfold Design Tree** and select **Bend Parameters**
7. Select **Bending Allowance** in **BA Option pull down** and change the value in **Bend Allowance** to 3mm. Click **OK**

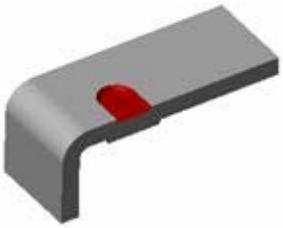
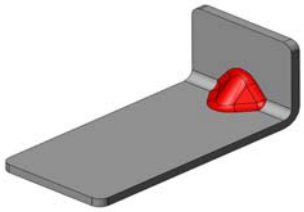


8. Save the part and the tutorial is complete.

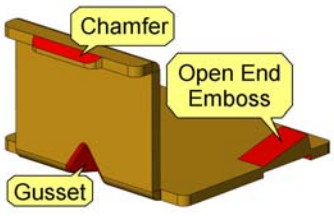
Tutorial 2. Semi-automatic unfold of part

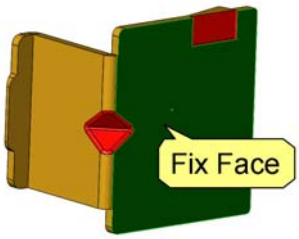
For some features which the material thickness is not even, users have to manually select them to help the system identify and or unfold them.

Semi-Automatic unfold features

	Open End Embossing
	Gusset

The part for this tutorial has 3 features which cannot be unfolded automatically; **Open End Embossing** and **Gusset.**





Semi-Automatic recognition is used to handle these metal features.

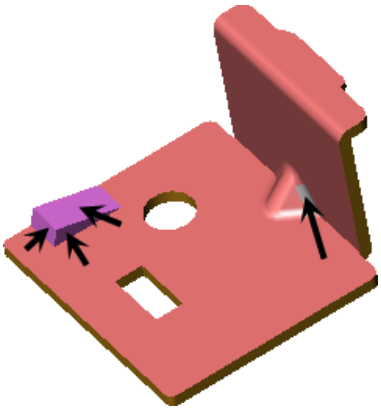
1. Open file **\training\unfold2\unfold2.sldprt.**
2. Select a fix face as show.
3. Click **Unfold Part**  to unfold the part
4. **RMB** select **Sheet Metal Object** in **Unfold Design Tree** and select **Unfold All**

Use User Assist Recognition to recognize the features manually

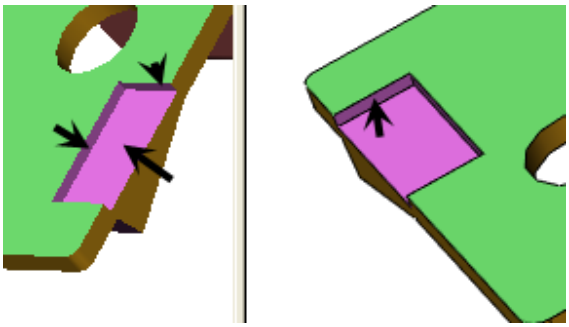
5. **RMB** select **Sheet Metal Object** in **Unfold Design Tree** and select **User Assisted Recognition.**

Manually correct the top and bottom faces

6. Click on the **Previous button** for **Top/Bottom Faces** modification**.**



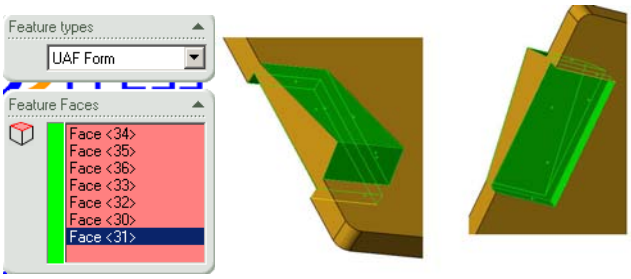
7. Select the faces as indicated above and click **Set Top** button.

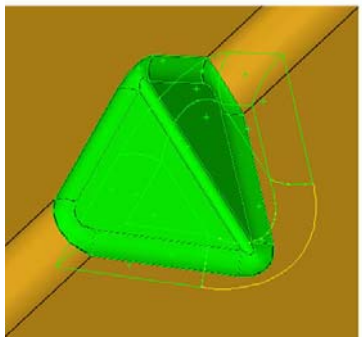
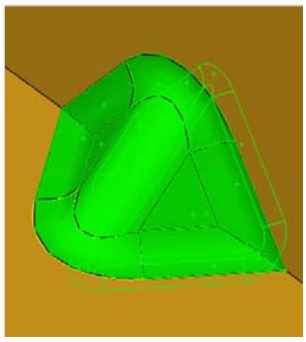


8. Select the faces as indicated above and click **Set Bottom** button.

9. Click **Next Button** to add **UAR** features.

10. Click **Add button** to add a new UAR Feature and select the faces composed of the open end emboss.

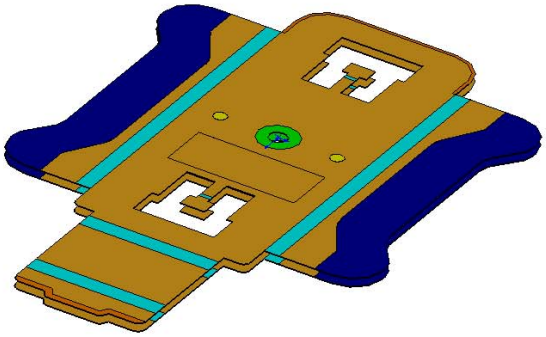


11. Click **Add button** and select **Gusset** for **Feature types** to add another UAR Feature for the Gusset by selecting the faces.  
12. Click **OK** to accept all default values.
13. **RMB** select **Sheet Metal Object** and select **Unfold All**, the part should be unfold
14. Save the part and tutorial is complete.

Tutorial 3. Advanced Unfolding: User Defined Feature

User Defined Feature is a very powerful feature in 3DQuickPress to handle intricate areas of a part related to forming. Through these tutorials you will acquire the technique in handling the following processes:

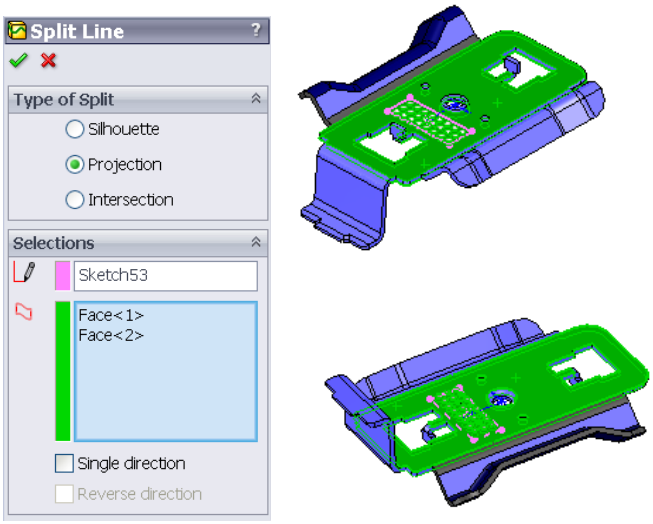
- Forming of undevelopable shapes like curve flange and forming
- Multiple step forming
- Form shape with linear bends and piercing



Prepare the model

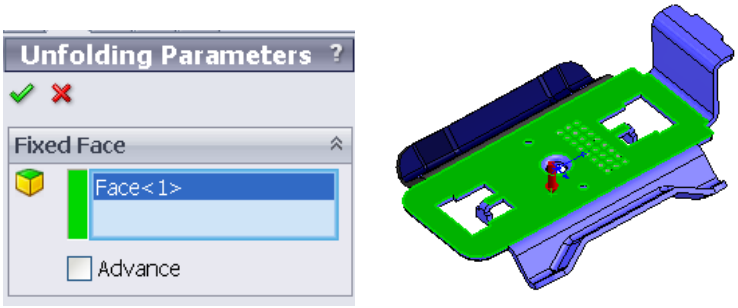
1. Open **c:\training\unfold3\Connectora.SLDPRT**
2. Select **Sketch 53** from **SolidWorks Feature Manager** and click activate **Split Line**  from Tool Bar or **Insert-Curve** pull down menu

3. Select the highlighted surface as shown and click **OK** to finish



4. Click **Unfold Part** icon to unfold the part.

5. Select the green face as the fixed face.



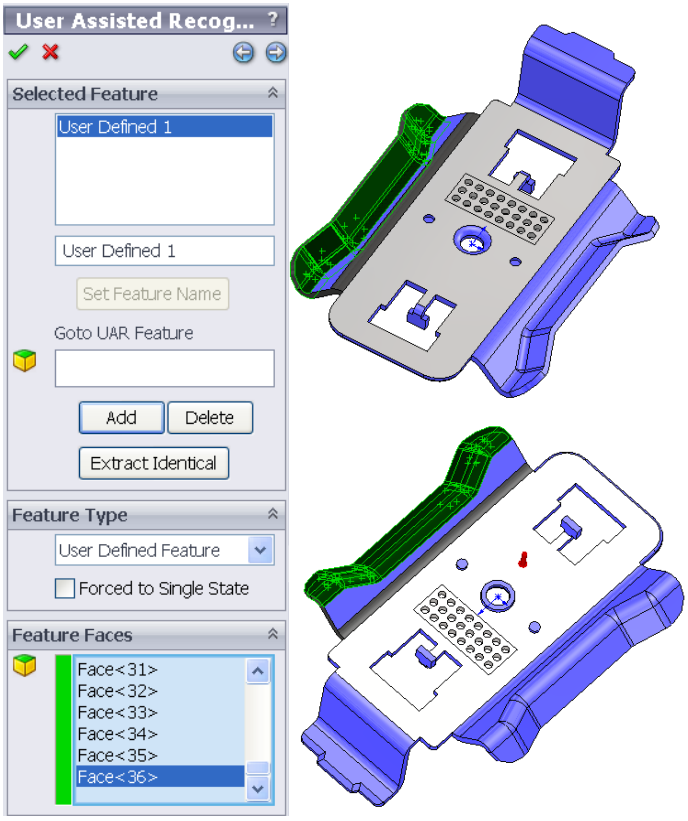
6. Accept all default values and Click **OK** .

To check out the faces that cannot be unfolded automatically:

7. Right Click on **Sheet Metal Object** in **Unfold Design Tree** and select **Unfold All.**

8. **RMB** select **Sheet Metal Object** in **Unfold Design Tree** and select **User Assisted Recognition**

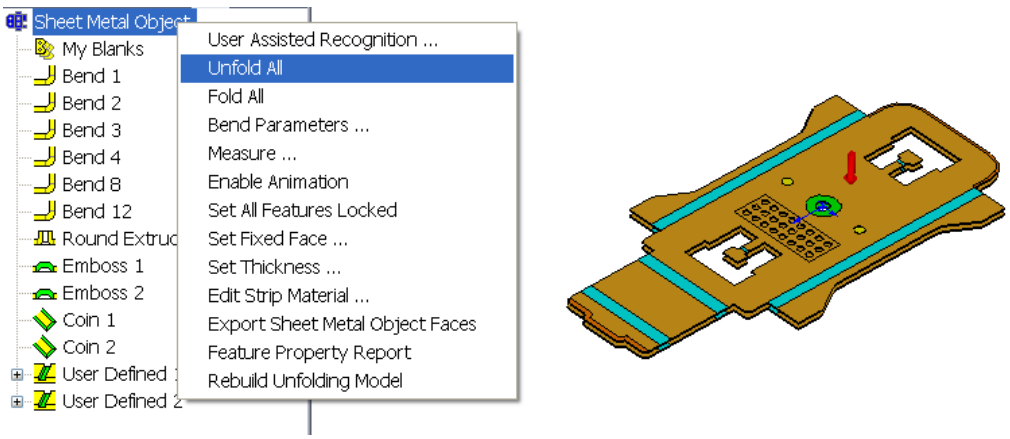
9. Click **Add** button to add a new **UAR Feature**. Also change the **Feature types** to **User Defined Feature** and select the faces as shown in green.



10. Click **Extract Identical** button to extract an identical **User Defined Feature on** the other side of the part

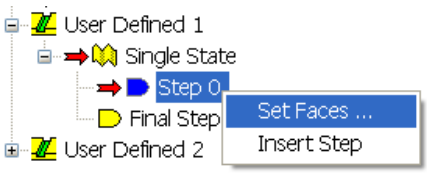
11. Click **OK** to exit.

12. **RMB** select **Sheet Metal Object** and select **Unfold All**, the part should be unfolded as show below.

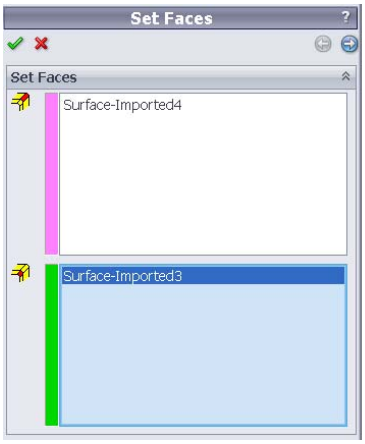


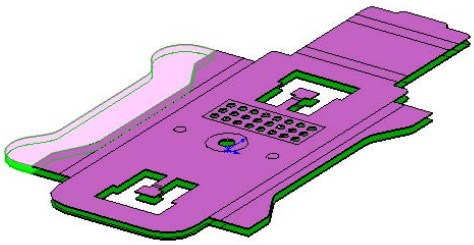
Set the blank surface for the User Defined Features

13. **RMB** Select **Step 0** and Click **Set Faces**



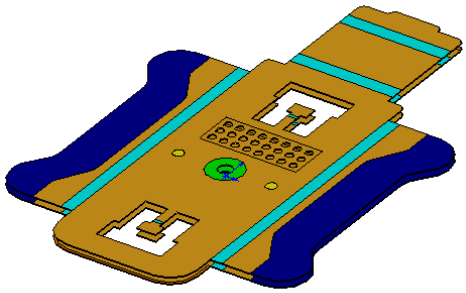
14. Select the imported faces as the blank





15. **RMB** select **User Defined 1** and **Fold** and **Unfold** the feature

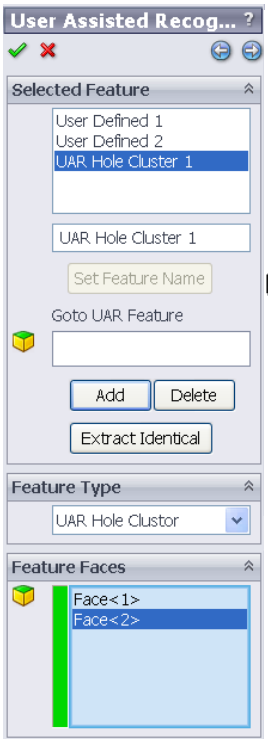
16. **RMB** select **User Defined 1** and **Set Face to Identical User Define Features**

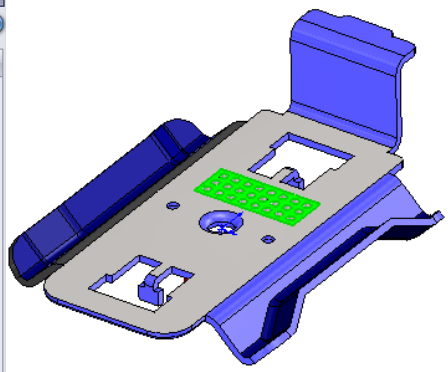


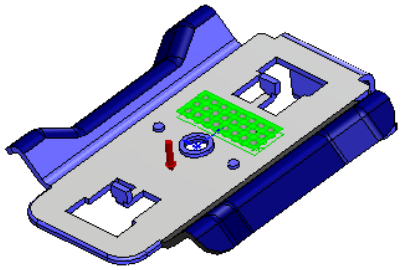
Create a UAR Hole Cluster Feature for patterned holes:

1. **RMB** select **Sheet Metal Object** in **Unfold Design Tree** and select **User Assisted Recognition**

2. Click **Add** button to add a new **UAR Feature**. Also change the **Feature types** to **UAR Hole Cluster** and select the faces as shown in green.

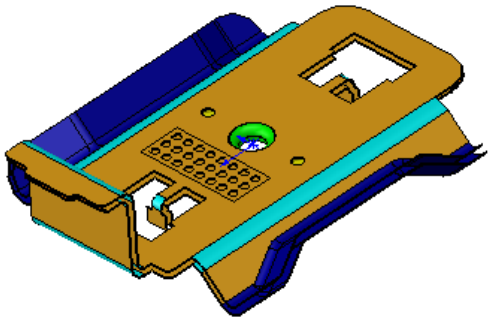


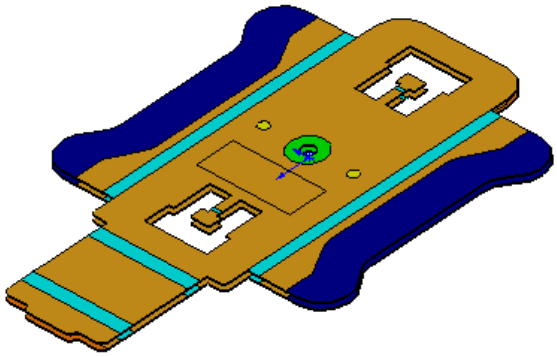




3. Click **OK**  to exit.

4. **RMB** select **Sheet Metal Object** and select **Unfold All**.





Tutorial complete

Tutorial 4. Advanced Unfolding: 3DQuickForm

Introduction

3DQuickForm is a finite element program to solve metal forming problems. Like all finite element software, the main steps are pre-processing; solving and post-processing.

- Pre-processing includes all the procedures to define the problem
- Solver is a black box using the pre-process data as input and outputs the solution
- Post-processing is the interpretation of the results from the solver.

In 3DQuickForm, we further refine the major step as following:

Geometry preparation

- Offset surface
- Simplify surface
- Create mid surface

Pre-processing

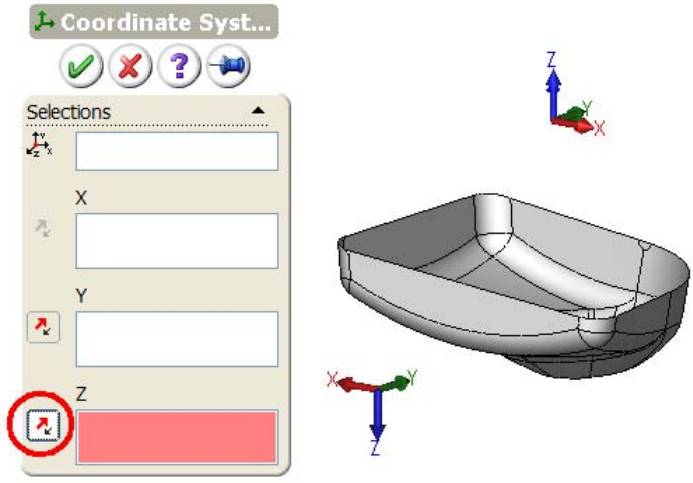
- Input boundary conditions
- Meshing
- Solving

Post-processing

- Blank output
- Thinning report

Define a coordinate system

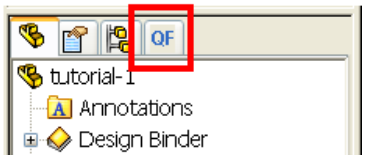
1) Click **File**, **Open**, **C:\Training\ 3DquickForm** select **Parasolid** for **File of Type**. And Select **tutorial-1.x_t**
2) Click **Insert**, **Reference Geometry**, **Coordinate system**. Click **Reverse Axis Direction to** reverse the Z axis direction

i. 

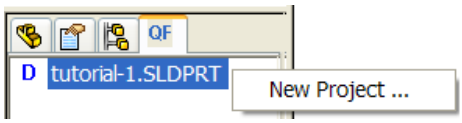
3) Click **OK** when finished

4) Click **File**, **Save** to save the document as **tutorial-1.SLDPRT**

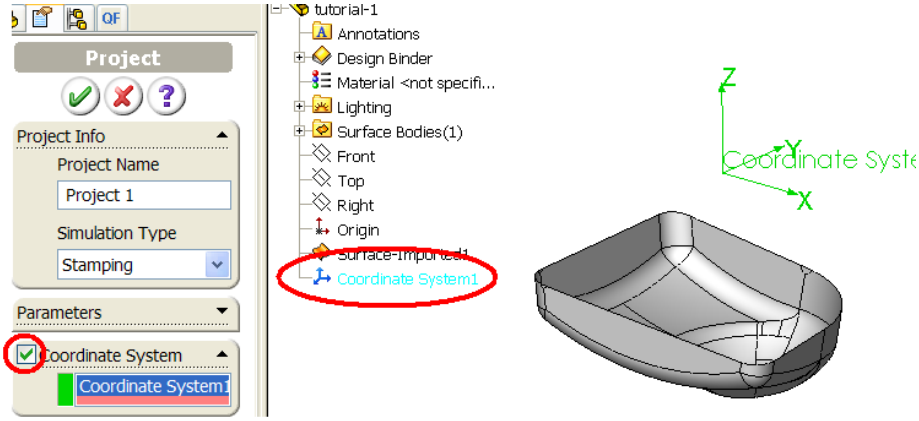
Start a new project

5) Click QF 

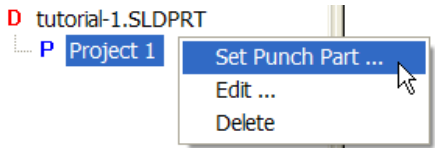
to switch to 3DQuickForm project manager

6) Right click Tutorial-1.SLDPRT 

and select New project, check Coordinate system and select Coordinate System1 from Fly out Feature Manager

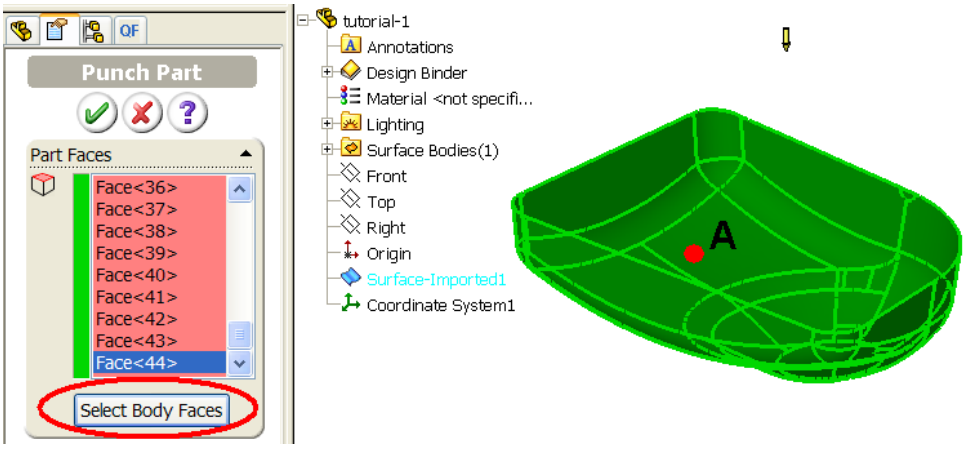


Define punch part



7) Right click **Project1** and select **Set Punch Part**

8) Click on **A** and click **Select Body Faces** to select all faces of the shell body.



9) Input 1.2mm for the material thickness

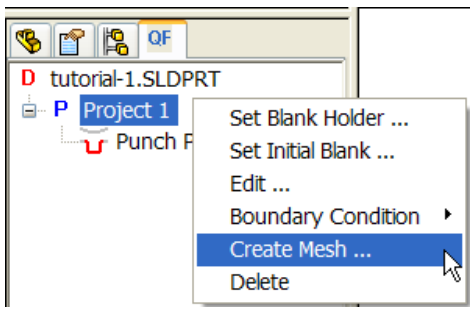
10) Click OK to finish

Create mesh and solve the problem

The mesh size is determined by 2 factors:

- Global mesh size
- Minimum mesh size

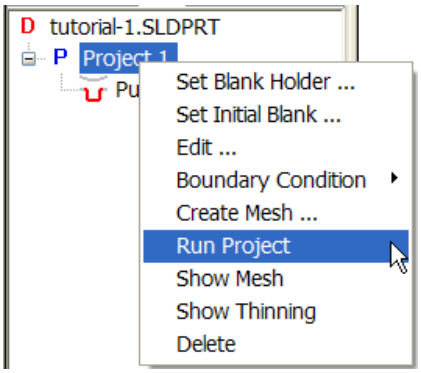
11) Right click **Project1** and select **Create Mesh…**



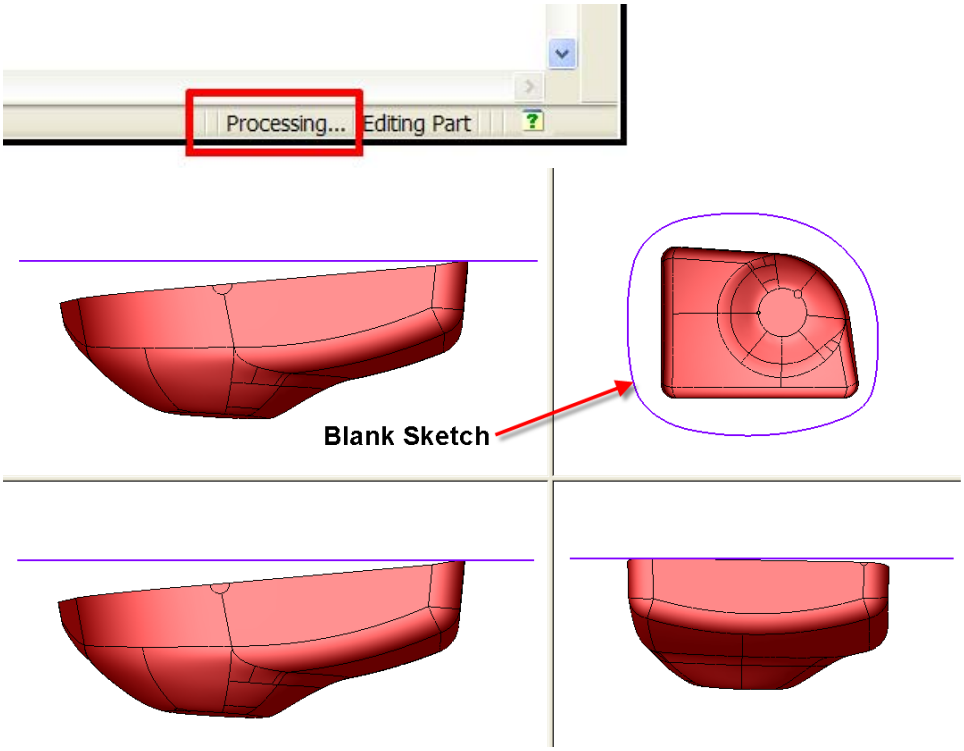
12) Click OK to accept the default mesh settings

Run the project

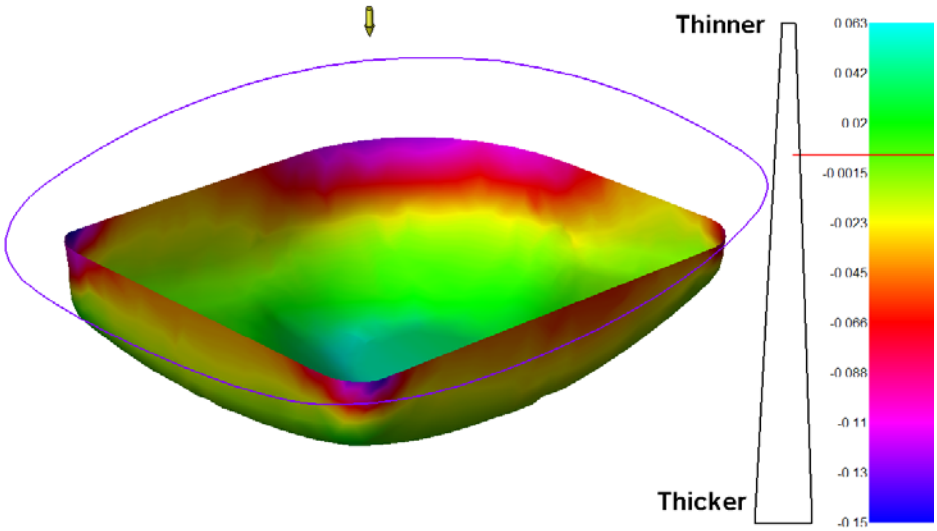
13) Right click Project1 and select Run project



- If the solver is running properly. Processing… will be shown on the Lower left corner of SolidWorks
- The blank is created as a 3d sketch



14) Right click on Project1 and select Show Thinning


Thinner
0.063
0.042
0.02
-0.0015
-0.023
-0.045
-0.066
-0.088
-0.11
-0.13
-0.15
Thicker

Tutorial 5. Strip Layout

After the part is unfolded successfully, we can move to the next step of progressive die design. In strip layout design the following workflow is recommended:

Station preparation

- Re-orientation of the part
- Prepare station for 2 up die

Create strip layout

- Progression
- Stock Width
- Optimization tools
- Automatic process design

Manipulate the strip layout

Assign operations

Split Bend

Split cutting

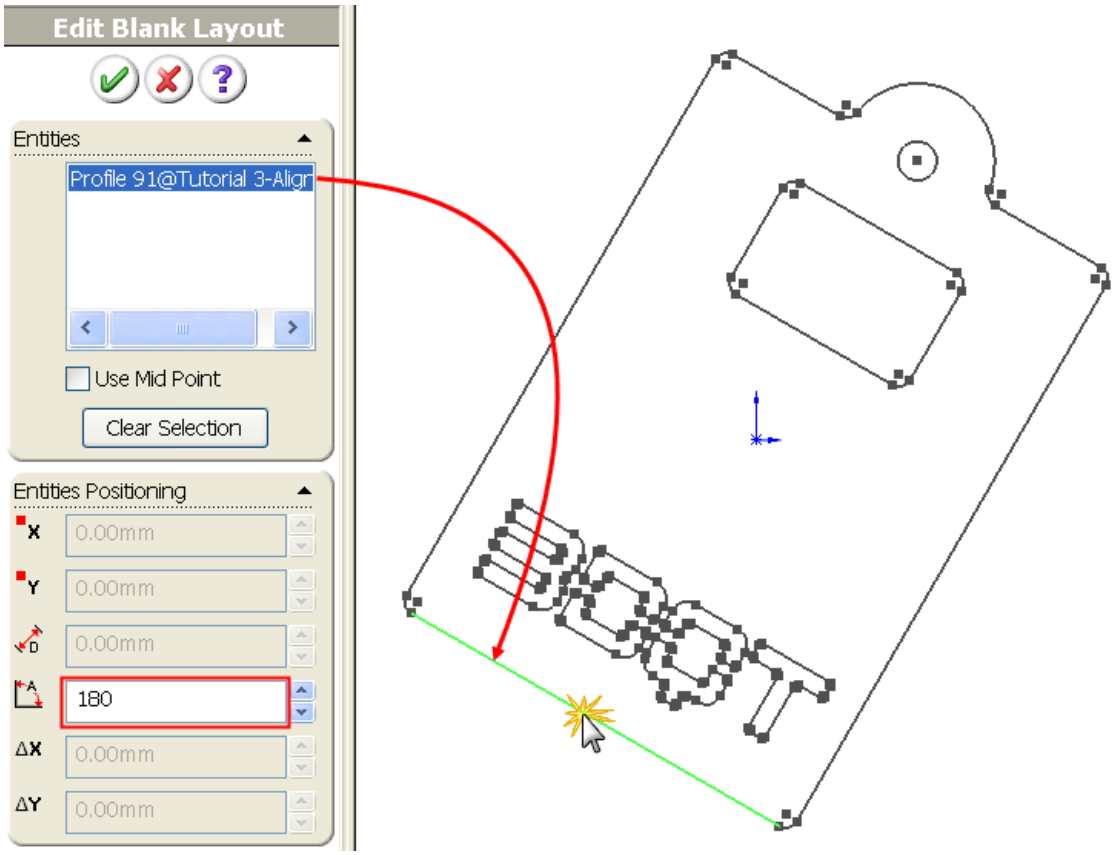
Create notching

Insert/Delete station

Station preparation - Part alignment

1. Open \training\Tutorial 5\Tutorial-Align.sldpart
2. Click **Create Strip Layout**  to create a new strip layout, input file name **T5-1** and click **Save**
3. Click **No** to prevent the existing part to be inserted to the new strip layout
4. **RMB** select **Blank Layout**  from **Strip Layout Design Tree** and **select Insert Sheet Metal Object**
5. In the File Dialog Box, select **Training\Tutorial 5\Tutorial-Align.sldpart**

6. Select the inclined line below 3DQT and change the line angle from 150 to **180,** Press Enter to update.



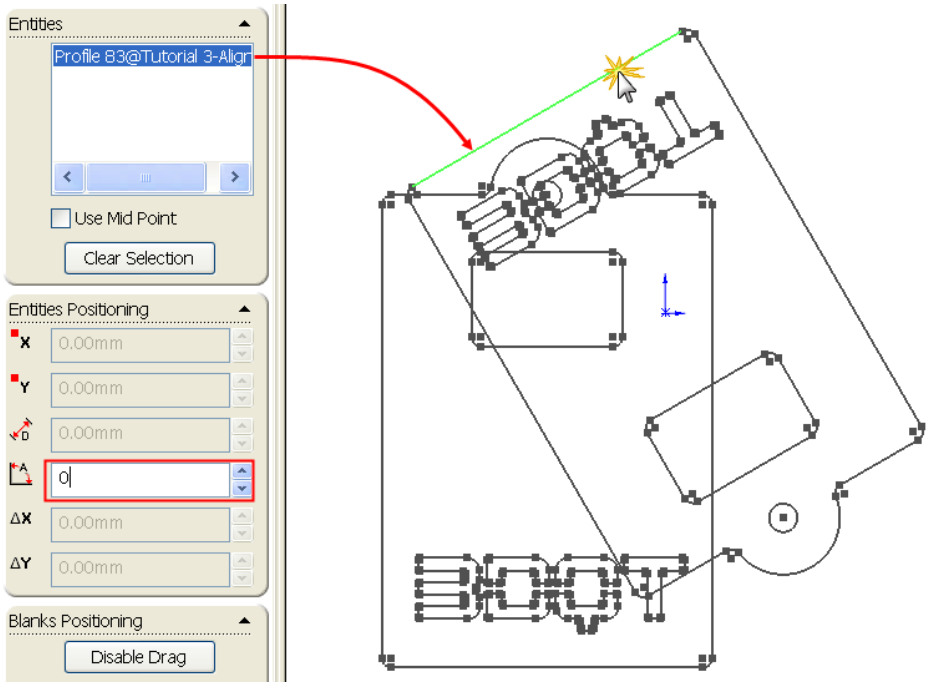
7. Click **OK** to finish

Mirror unfold part

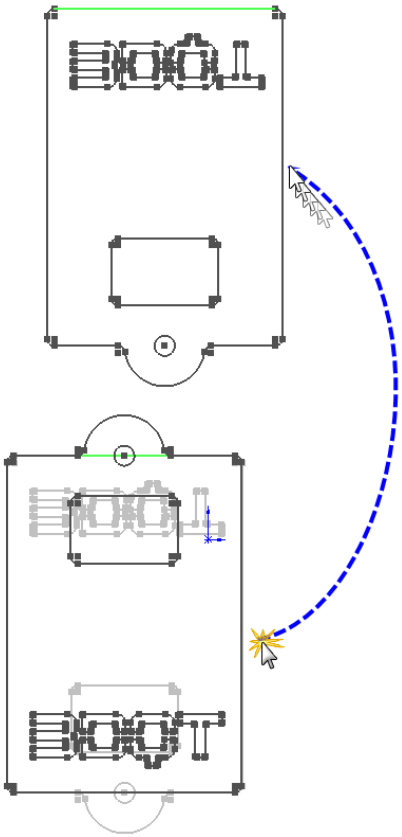
1. **RMB** select **Blank Layout** from **Strip Layout Design Tree** and **select Insert Mirrored Sheet Metal Object**

2. In the File Dialog Box, select **Training/Tutorial 5/Tutorial-Align.sldpart**

3. Select the inclined line above 3DQT and change the line angle from 30 deg to **0 deg,** Press Enter to update.

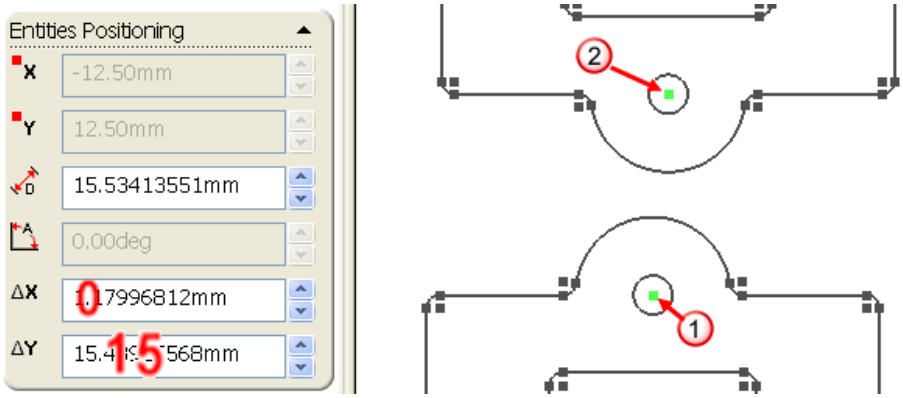


4. The horizontal should still highlight, drag to the position as shown



Align the blanks to position

5. Zoom in close to the area shown below. Click **Clear selection**. select **Point [1]** and **Point [2]**, change **delta X** to **0**, **delta Y** to **15mm**



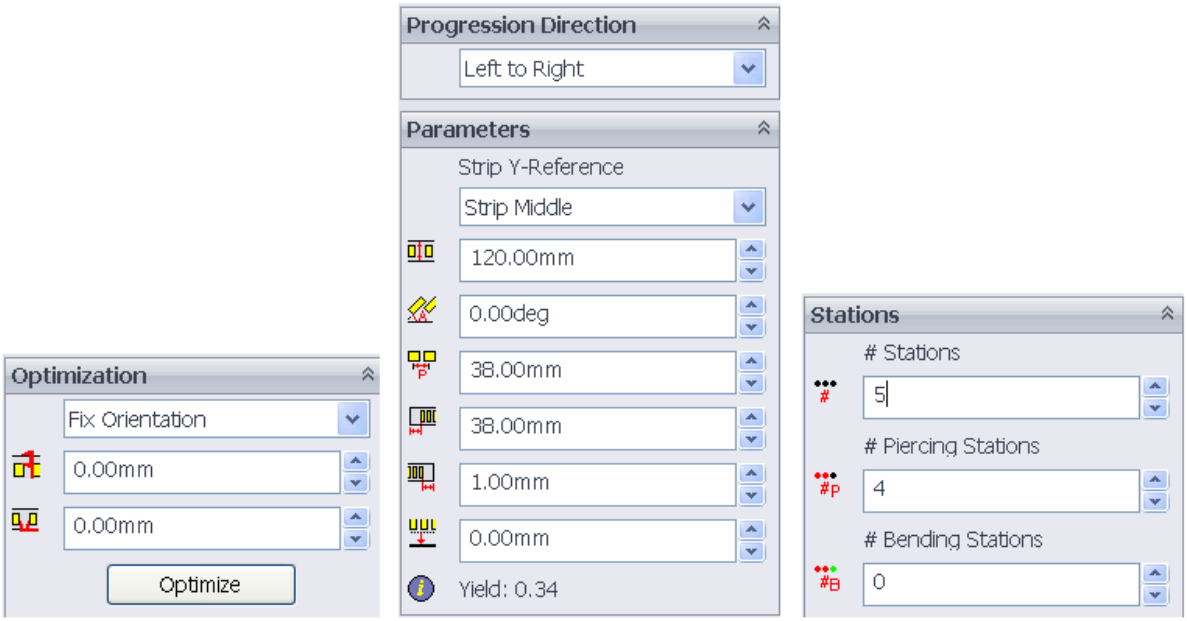
6. The 2 points should remain selected, Click **Center to Origin** from **Blank Positioning**

7. Click **OK**  to finish

Prepare strip layout

1. **RMB** select  from the tree and select **Edit Strip Parameters**

2. Expand the **Optimization** property sheet and input 0 for **Width Margin** and **Station Clearance**, click **Optimize** button



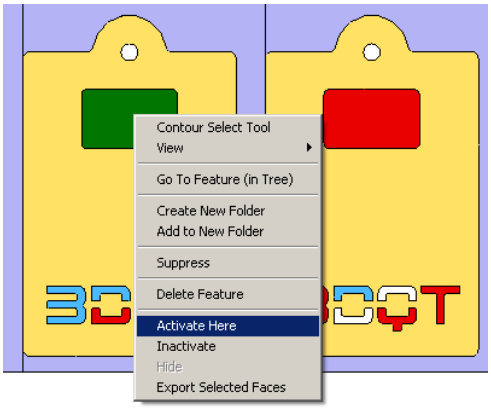
3. Input the values as shown in the Parameters and then **Stations** property sheets

4. Click **OK** button to finish

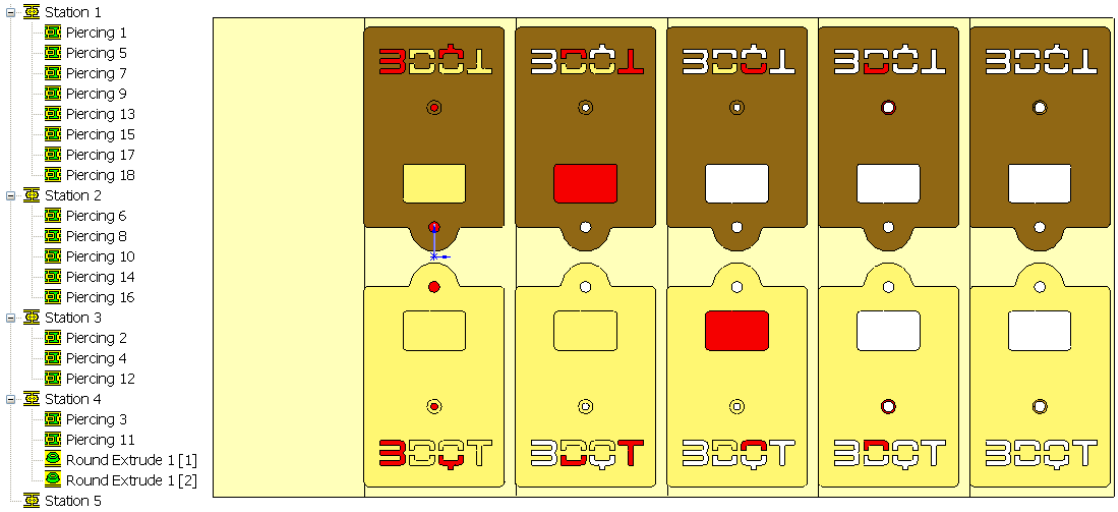
5. **Save** the part

Manipulate the features

1. Click on the  **Filter Features** from display tool bar



2. **RMB** select the feature from the graphic window, use **Activate Here** or **Inactivate** to modify the strip layout to the one as show below.

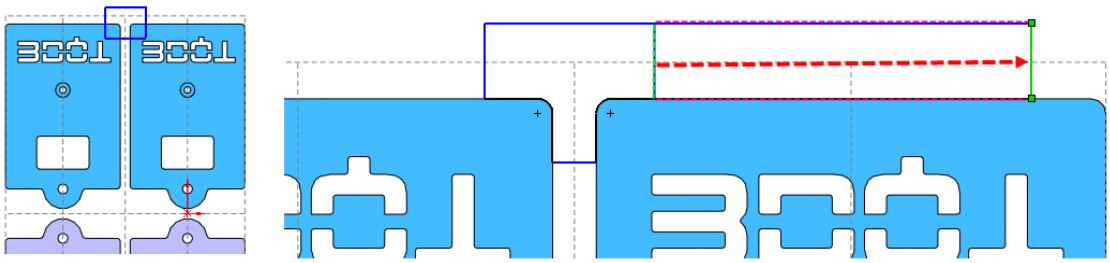


3. **Save** the file when finished.

Create Notching

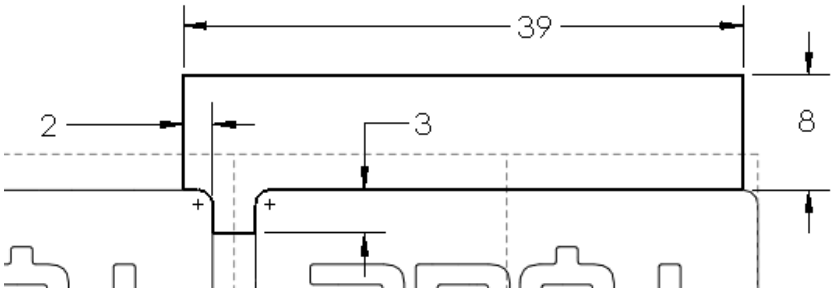
1. **RMB** select **Unprocessed feature** from the **Strip Layout Design Tree** and select **Create Notching Feature.** Press " **F** " to fit the blank in the graphic area

2. Click **Rectangle** and create a rectangle as show and click on **QuickSketch** from toolbar.

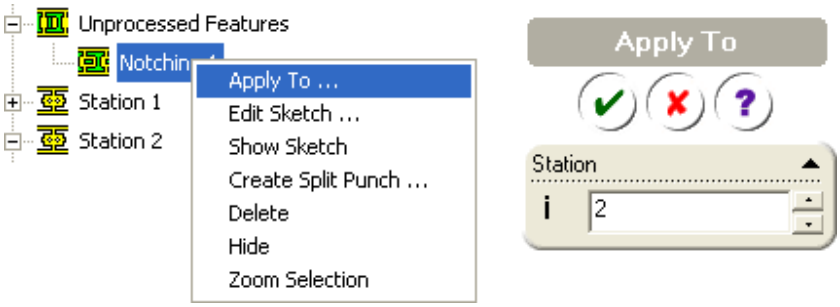


3. Drag the line segment as show above

4. Detail the sketch and Exit sketch mode when finished



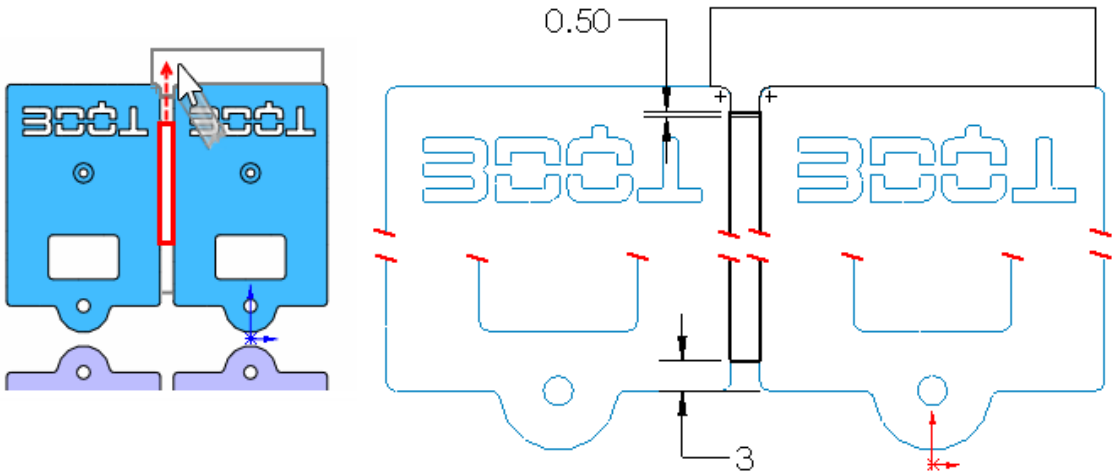
5. **RMB** select the newly created **Notching punch** and **Apply To** Station2



6. Repeat Step 1 and create a rectangular notching punch and a notching punch for cutoff

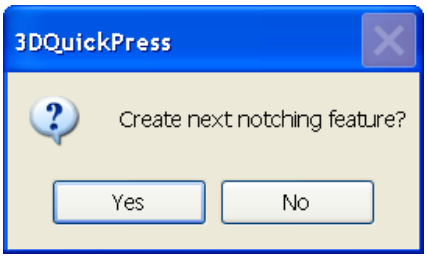
7. Click **RMB** in empty space of Graphic Window and select **Show All Notching Sketches**.

8. Click **Rectangle** and create a rectangle as show, drag it up to the arrow position and detail the sketch

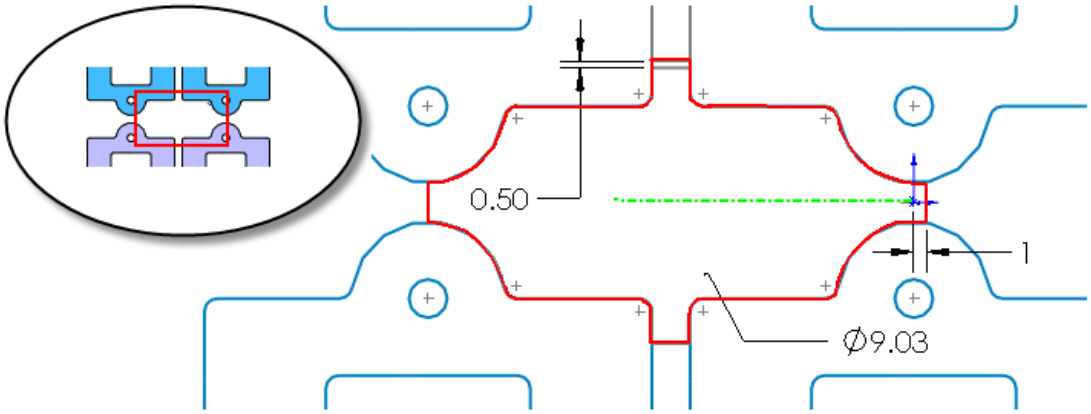


9. Click **RMB** in empty space of Graphic Window and select **Create Notch Batch Mode**.

10. Exit Sketch Mode and click Yes to continue the design of next notching feature



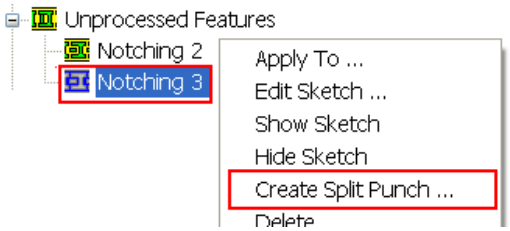
11. Click **Rectangle** and create a rectangle as show, click **QuickSketch** and detail it as show below for the cutoff punch.



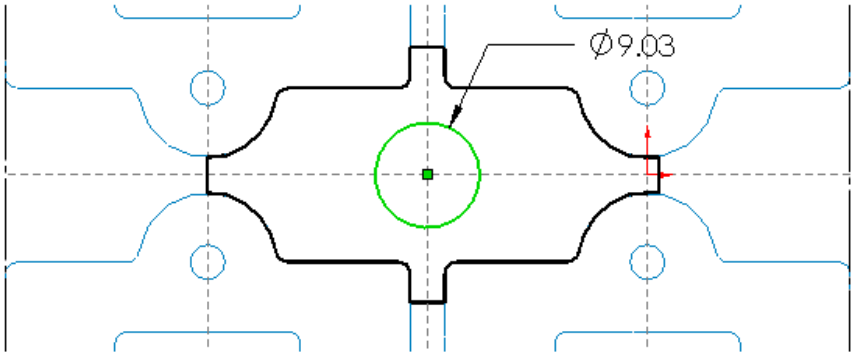
12. **Exit** sketch and click **No** to finish all standard notching punch design

Create Split Punch

1. RMB select the cutoff punch and select **Create Split Punch …**



2. Click **Circle** and create the following 9mm diameter round hole.



3. **Exit Sketch** and Click **OK** for split punch Property Manager

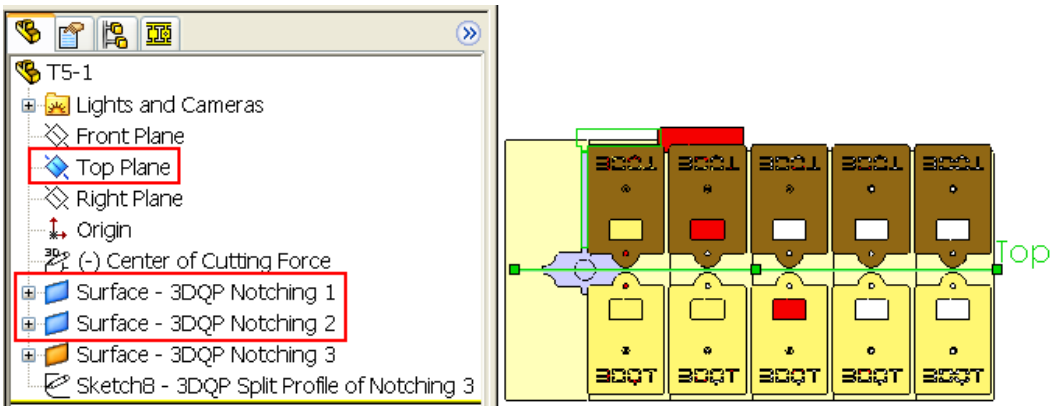
4. Save the document.

Mirror Notching Punch

1. Click **SolidWorks Feature Manager Tab**

2. **RMB** select Top of Tree and select **Show All 3DQP features**

3. Pres-Select **Top Plane**; Surface **– 3DQP Notching 1** & **3DQP Notching 2,** Click **Insert**, **Pattern Mirror**, **Mirror** and click **OK**

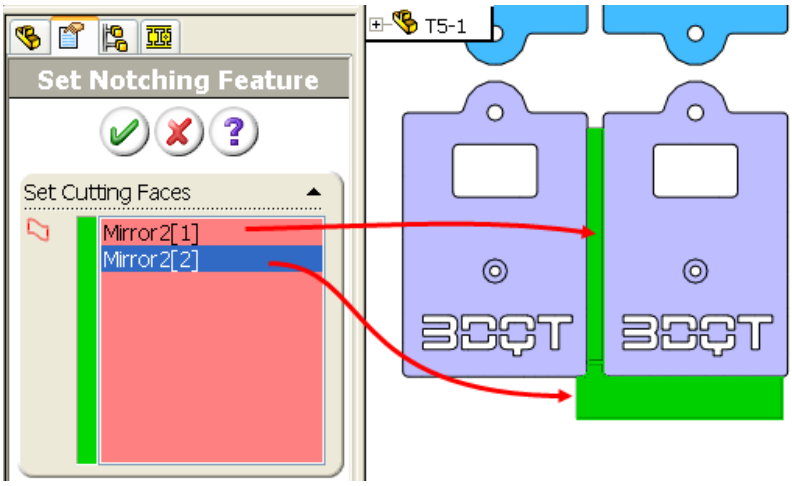


4. Click **Show Stations only** to show the notching design faces.

5. Click **Strip Layout Design Manager Tab**

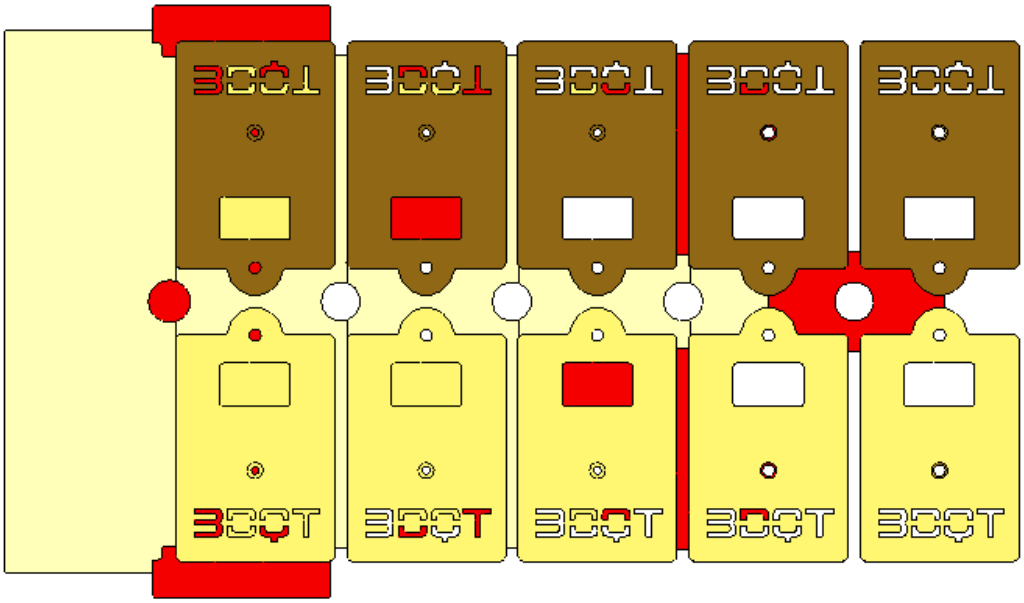
6. **RMB** select **Unprocessed feature** from the **Strip Layout Design Tree** and **Set Notching features…**

7. Select the mirrored faces and click **OK**



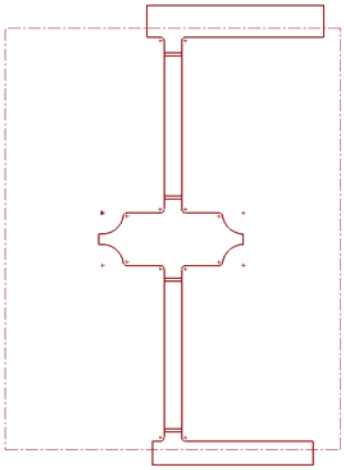
8. Click **Show Stations only** to toggle back to the display of strip layout.

9. Using **Apply To** or **Drag** & **Drop** activate the notches to create a strip layout as show below



Tips:

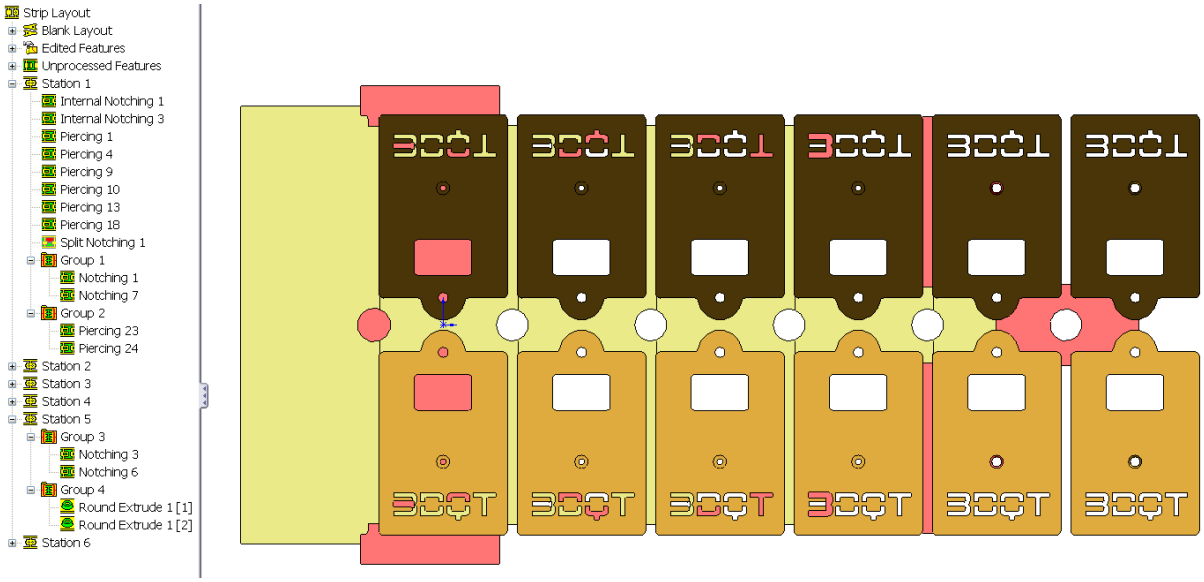
RMB select the **Notch Punch** and select **Show Sketch** to show the underlying Sketch to assist undercut design



Grouping the notching features

10. Select **Notching 1** and **7** together by holding the **Ctrl** key
11. **RMB** on selected **Notching Features** and **Create Group**

12. **Create Group** for other feature pairs and changed the strip layout as shown below



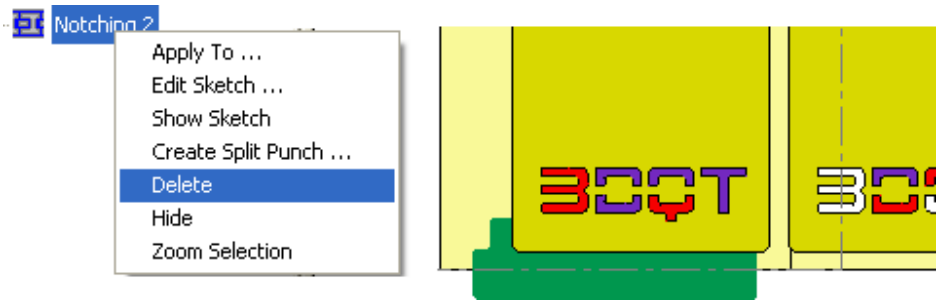
13. Select **Group 3** and drag to **Station 4**

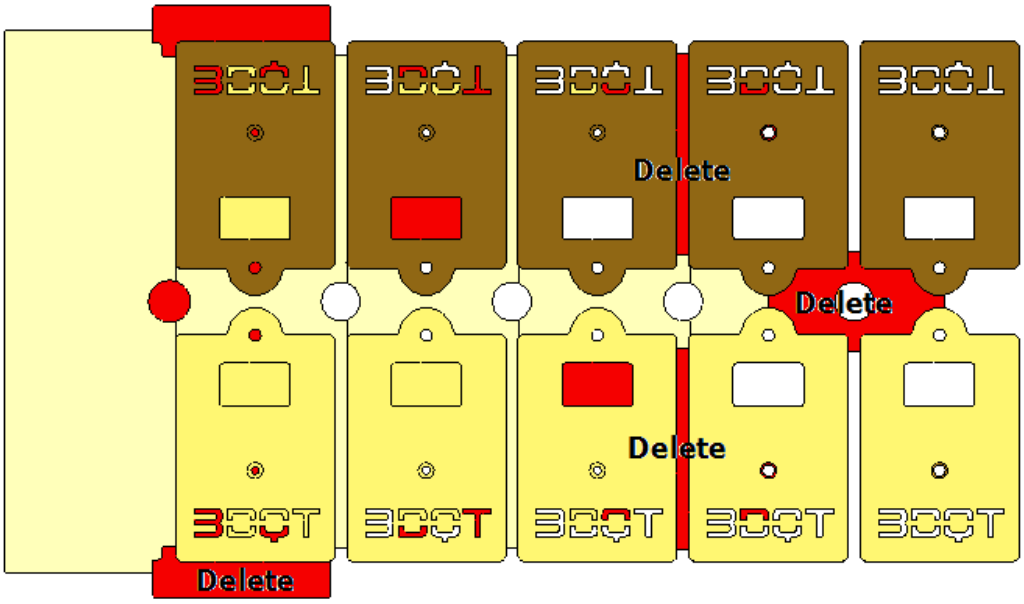
14. Save the document

Design Change - Create Blanking punch process

1. Click "**Save as"** and name the strip file name **Blankoff.sldprt**

2. Delete most of the notching punches from **Strip Layout Manager** Short Cut menu. We want to delete all except **Notching 1** and the **Pilot**.

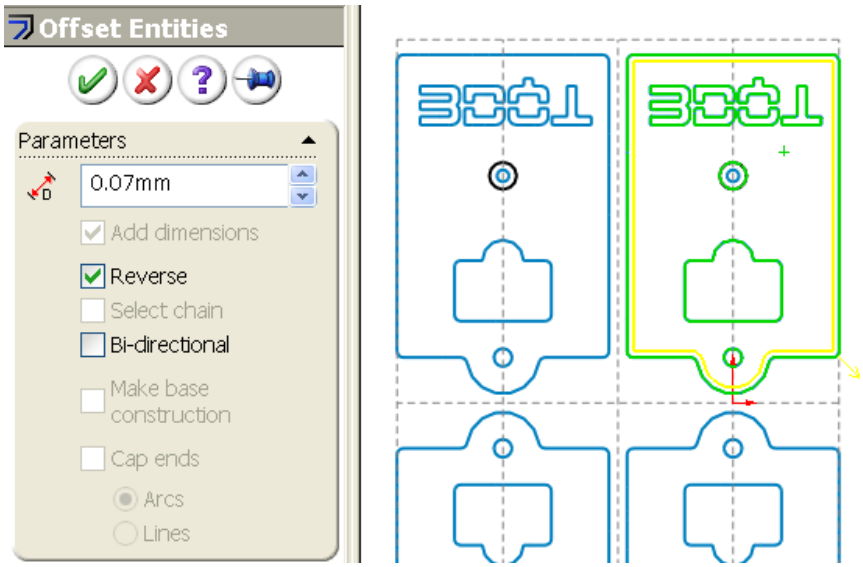




3. **RMB** select Notching 1 from **Strip Layout Design Tree** and click **Edit Sketch**

4. Delete all the sketch entities.

5. Select the upper blank and Offset it 0.07mm inward for cutting clearance of the blanking punch.



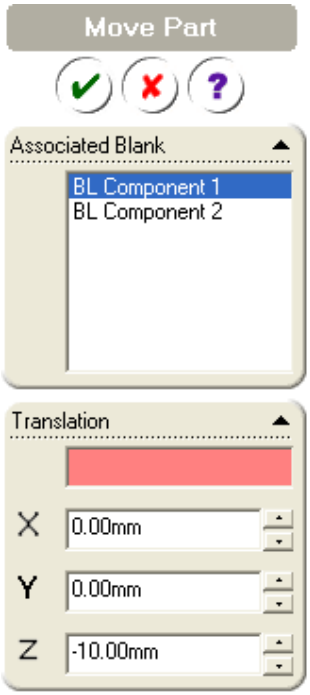
6. Click **OK** to finish offset and Exit Sketch

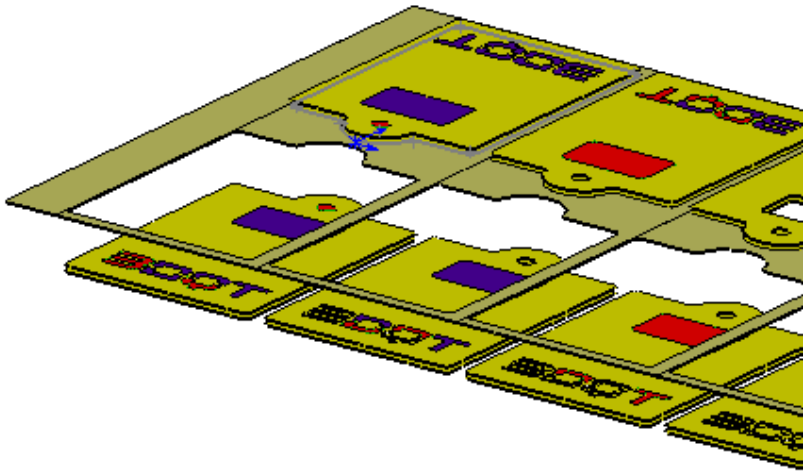
7. **RMB** select **Unprocessed feature** from the **Strip Layout Design Tree** and select **Create Notching Feature.** Use similar technique to create blanking punch for the lower blank.

8. Rename the 2 notching punches to **Blank1** and **Blank2** respectively.

9. **Apply Blank1** and **Blank2** to the last station.

10. **RMB** select **Unprocessed feature** from the **Strip Layout Design Tree** and select **X-Form Part**… Move **BL Component 1** downward **10mm in Z-direction**.





11. Click **OK** when finish

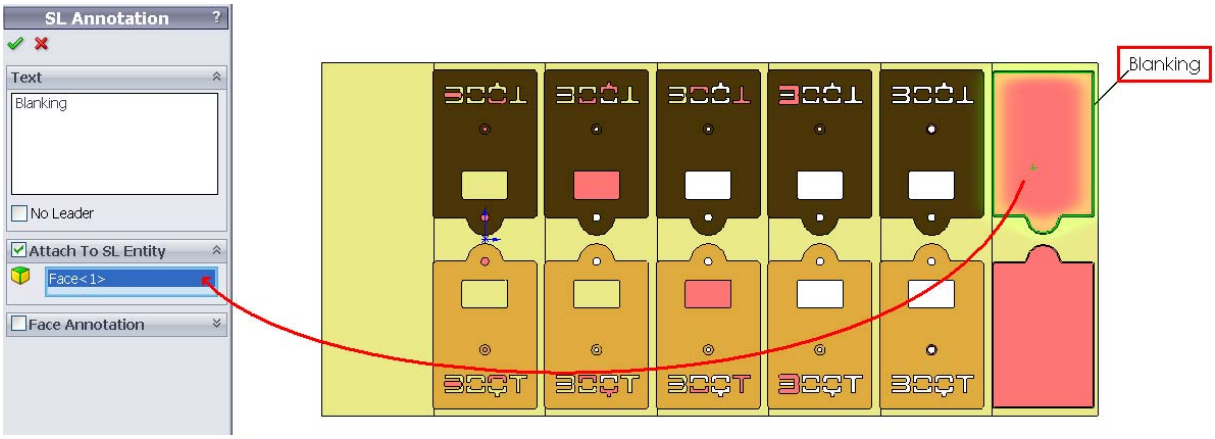
12. Repeat this for **BL Component 2**

13. Apply **Move Part 1** and **Move Part 2** to the last station

14. **RMB** select **Unprocessed feature** from the **Strip Layout Design Tree** and **Create Annotation Feature…**

15. Select the face for the **blanking feature**, and type "**Blanking**" in the text box

16. Click the graphic window to location the annotation label

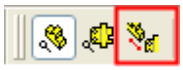


17. Click **OK** button to finish

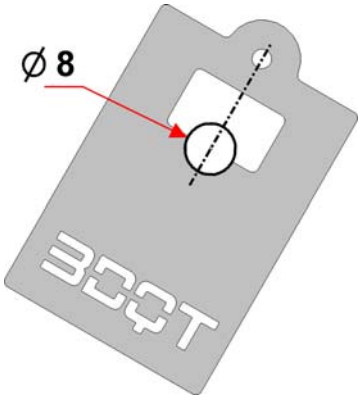
18. Repeat Steps **14** to **17** for the other blanking feature

19. Save the document

Parametric Design Change

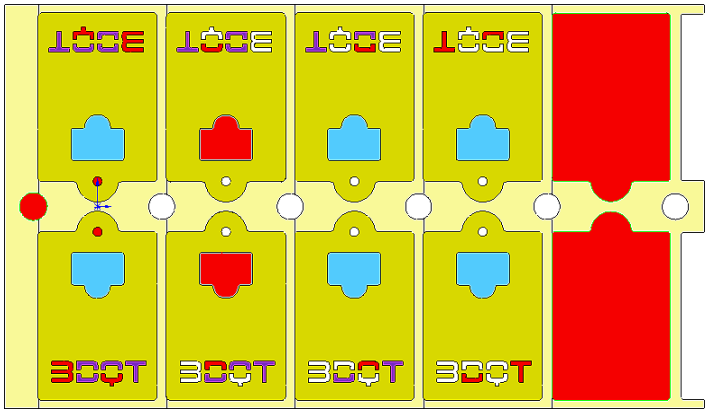
1. Show original solid model only—selecting the display icons of 3DQP to either show/hide Solid, show/hide 3DQP Model, or Toggle view. 

2. Make a cut in the original part by starting a sketches and making a circle with a 8mm diameter with center on the midpoint of the line



3. Rebuild Unfold Model by Shortcut menu

4. Switch to Strip layout window and Rebuild Strip Layout by Shortcut menu of design tree...



5. Re-apply the piercing

End of Tutorial

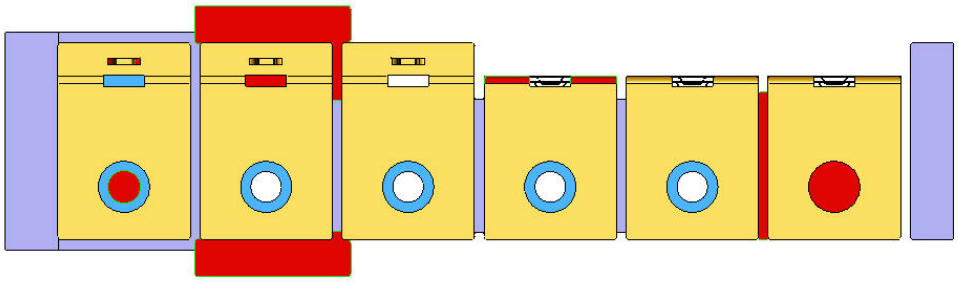
Tutorial 6. Punch design

You should have assembly design concept and basic SolidWorks assembly design knowledge for this tutorial.

After strip layout design is finished, the following workflow is used for the punch design

- Create punch design assembly environment
- Create all punch components
- Modify punch
  - Shoulder location
  - Convert straight punch to stepped
- Create User Duefined Components for lancing; embossing and forming
- Use User Defined Components for stock guide and pilot punches
- Check interference and fix

This tutorial will based on the following strip layout



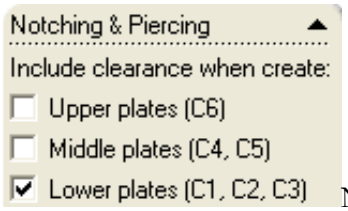
Pilot Hole Lancing	Notching Piercing		Bending		Piercing; Cutoff

Preparation for punch Design

1. Open file **Training\Tutorial 6\P1 Strip Layout.SLDPRT**
2. Click **New Punch Design**  from Tool Bar to start a new Punch Design Assembly Document.

3. Input Project Name = **QPTD**, Project No. = **1**, Naming Convention = ***1-*3**, Die Set Template = **0**, Round Punch (Straight) = **Misumi Straight**, Round Punch (Step) = **Misumi Round Step**, Click **OK** to accept the default settings in other pages

4. Click **3DQuickPress**, **Die set Structure**, **Clearances**, uncheck **Upper plate** and **Middle plate.**

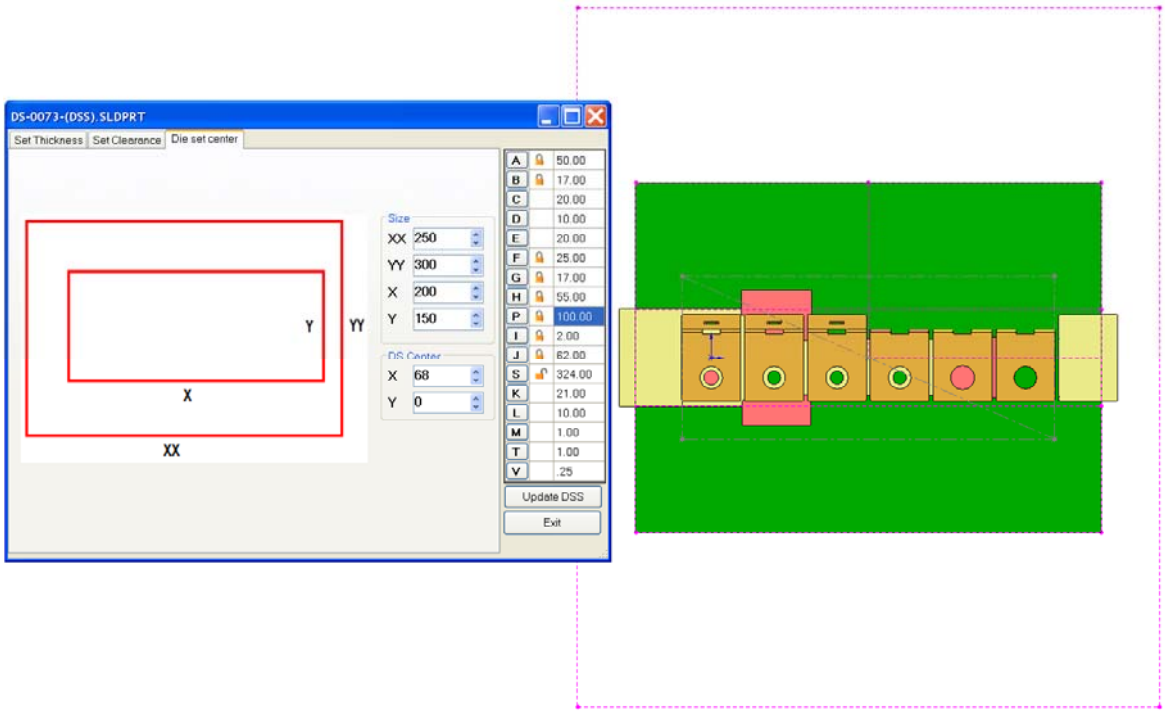


Note: by deselecting these two items clearance bodies will not be created.

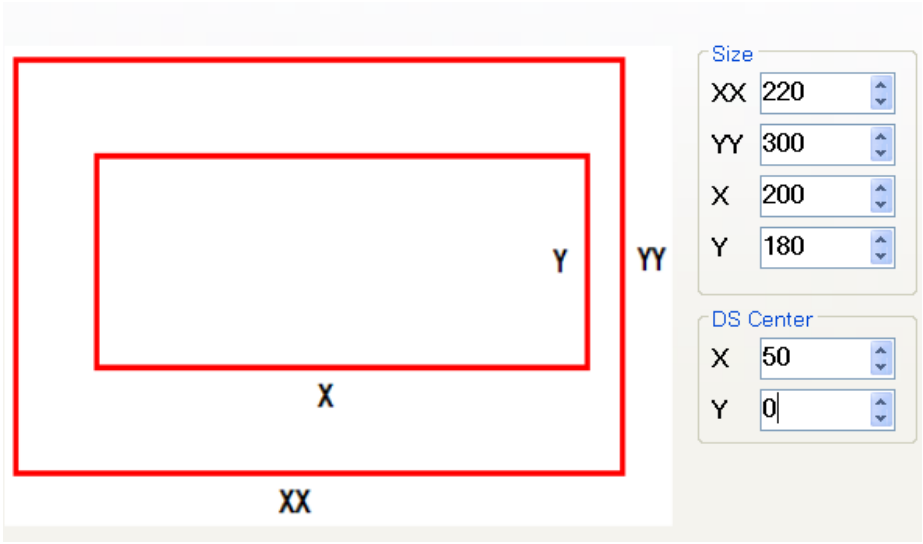
Adjust Die set Center and Die Set Size with Modify DSS function.

5. Start **Modify DSS** function by clicking the **Modify DSS** button on the **3DQP Mics. Tools Toolbar**

6. Switch to **Die set center** tab and the preview of the die plates will be shown on the graphic area.



7. Set the die set size and die set center location as below.



8. Click **Update DSS** to update the parameters.

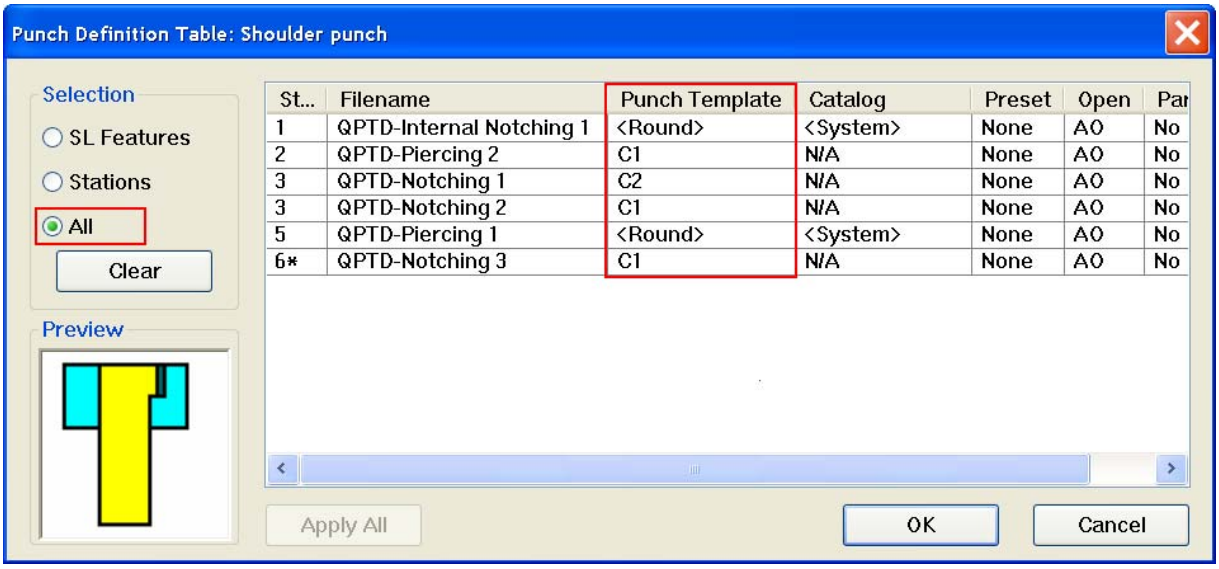
9. Click **Exit** to return to the punch design assembly, and the assembly will be rebuit.

Note: All 3DQuickPress Punch Design features must be defined and created in Assembly Editing mode. If you notice that the icon is in depressed mode  **, click on it to turn it off.** 

Create cutting punches

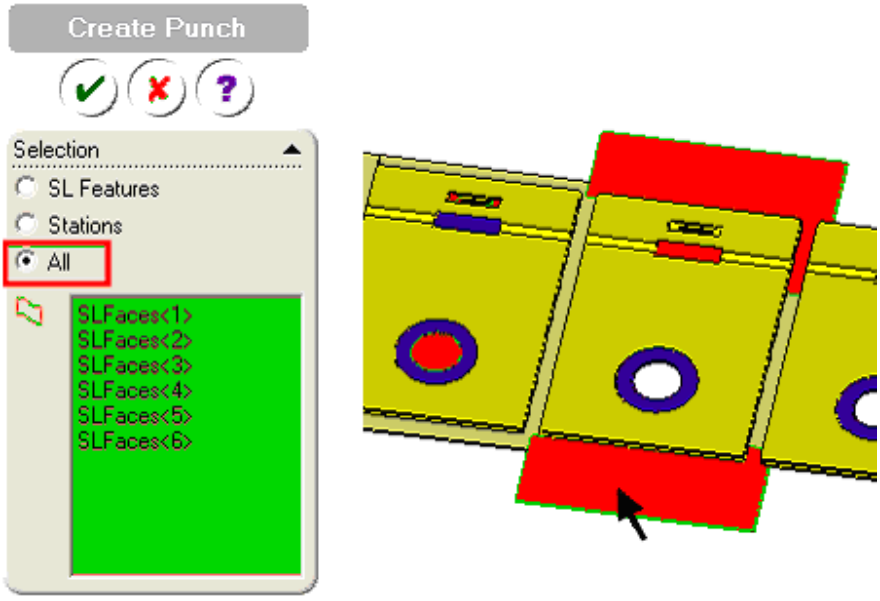
1. Click **Punch Definition Table**  from PRL tool bar

2. Click **All** and select **Punch Template** column and click **OK** to finish



3. Click **Create Cutting punch**  from PRL tool bar

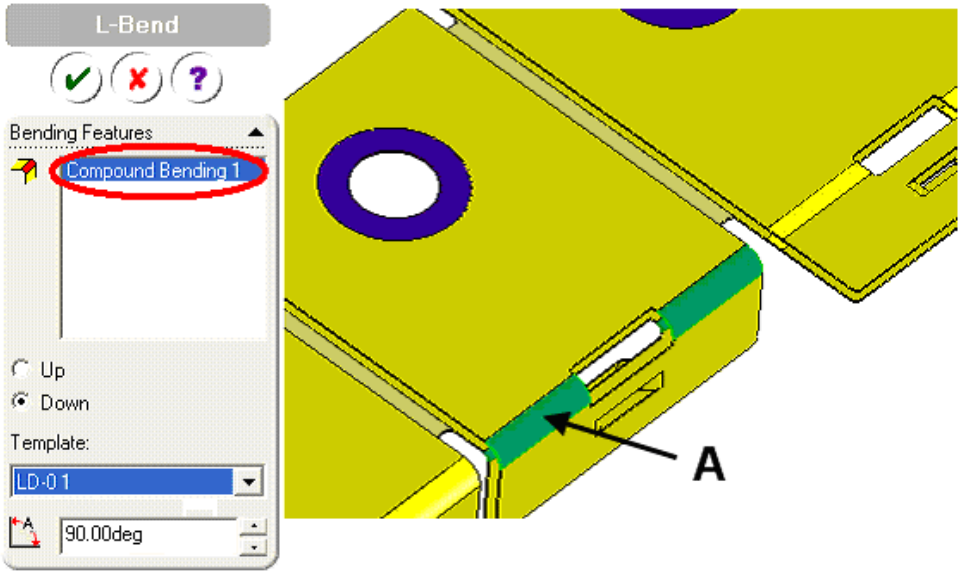
4. Check the **All** option and Click **OK** and wait until all punches are created



Create L-Bend

1. Select **Create L-Bend**  from Punch Design Tool Bar

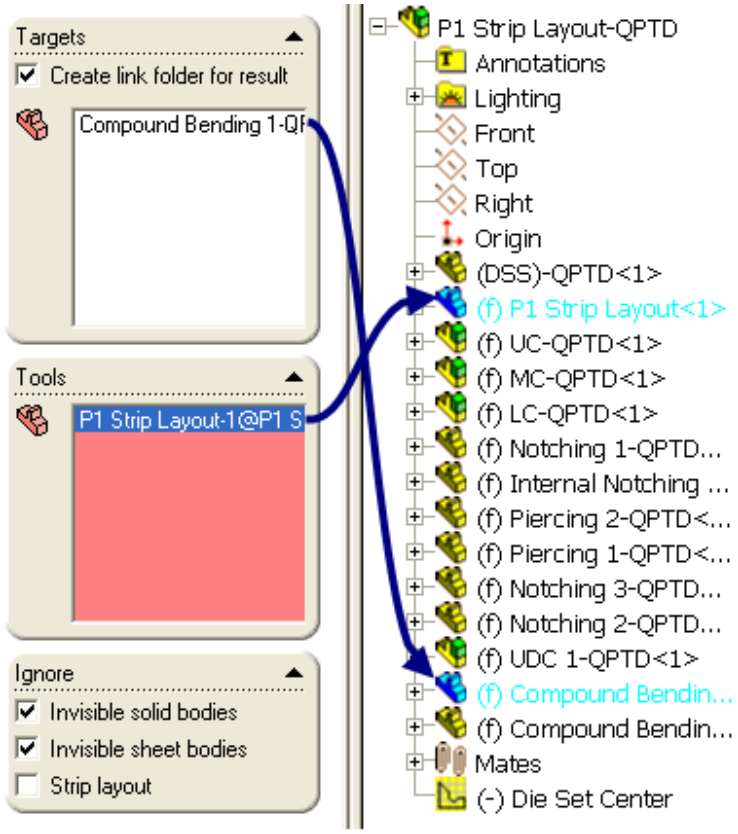
2. Click **A** to select the bend in the graphics area. Also click on **Compound Bending 1** inside Bending Features selection box of property manager and select template **LD-01**.



3. Click **OK** to finish

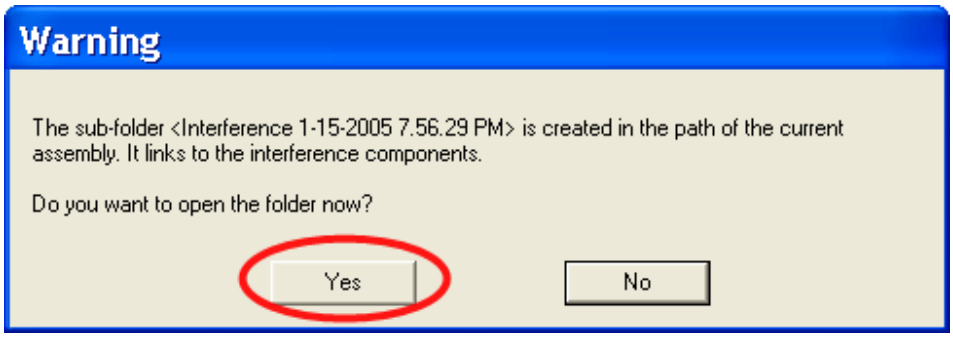
4. Click 3DQuickPress, 3D QuickTools, Interference Detection

5. Select **Compound Bending 1-QPTD (LDDI-01)<1>** to **Target** selection box, select **P1 Strip Layout** from Fly Out Feature tree to **Tools** selection box.



6. Click **OK**  to finish

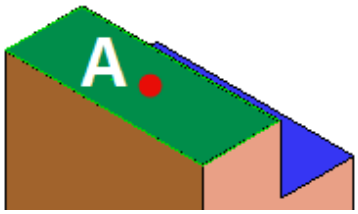
7. Click **OK** to launch Windows Explorer for the folder containing links of part having interference.



8. Double click the link 

to open the document

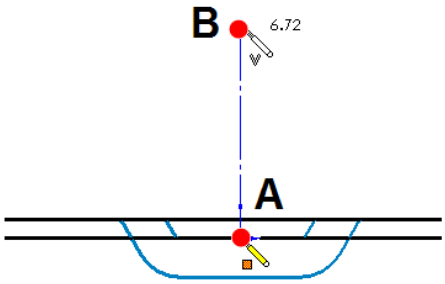
9. Click **Isometric** to set to isometric view wire frame

10. Click on Face **A** 

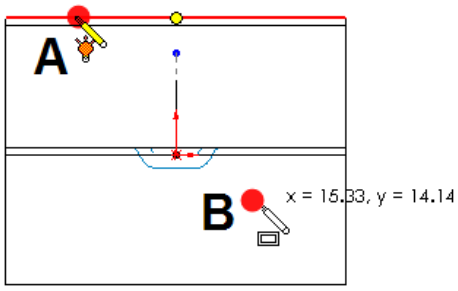
and click **Insert Sketch**

11. Click **Wire frame** to set the display to wire frame

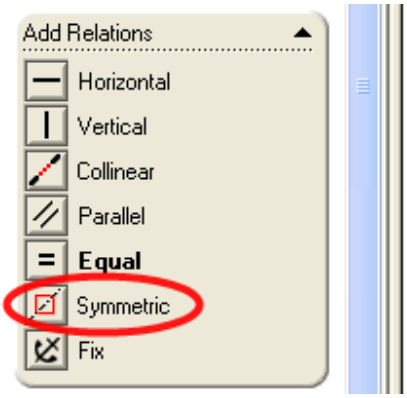
12. Click **Normal To** to set the view to the sketch plane

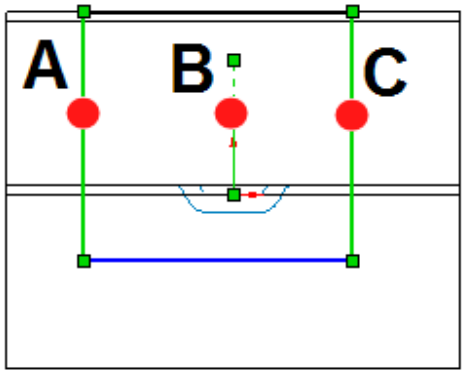
13. Click **Center Line** and click **A – B** 

14. Click Rectangle on the Sketch toolbar, or Tools, Sketch Entities, Rectangle.

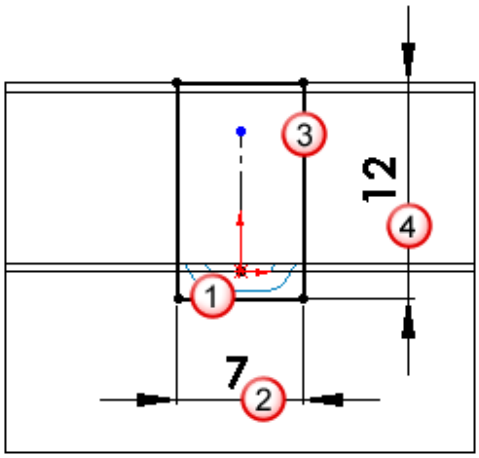
15. Click on **A** and then **B** to create a rectangle. 

16. Ctrl-Select **A-B-C** and click **Symmetric** in Add Relations Property Manager

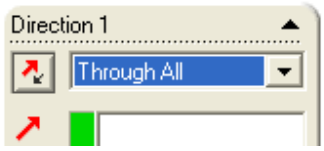




17. Click **Smart Dimension** , Click Line **1** and then click location **2**, input **7 mm**. Click Line **3** and then click location **4**, input **12 mm**



18. Click **Extrude Cut** or click **Insert**, **Cut**, **Extrude**

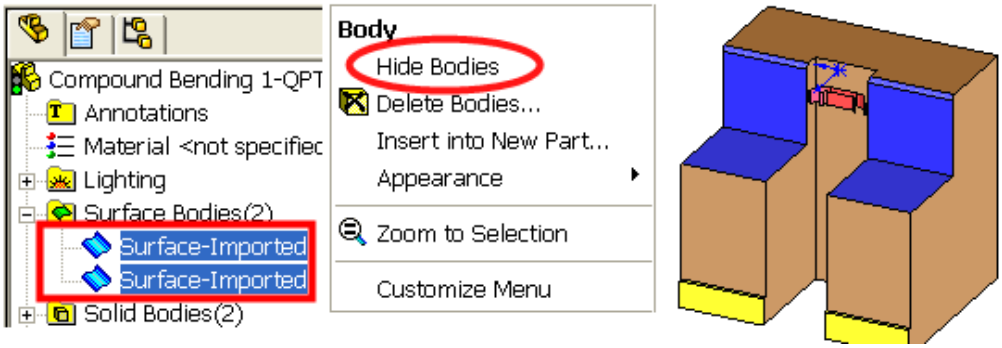
19. Select **Through All**. 

and Click **OK** to finish

20. Click **Shaded with Edges** to set the display back.

21. Click **Rotate View** and rotate the view to see the interference bodies.

22. Control Select the **2 imported surfaces** in **Surface Bodies folder**, right click and select **Hide Bodies**



23. **Save** and then **close** this document and go back to **P1 Strip Layout-QPTD.SLDASM**

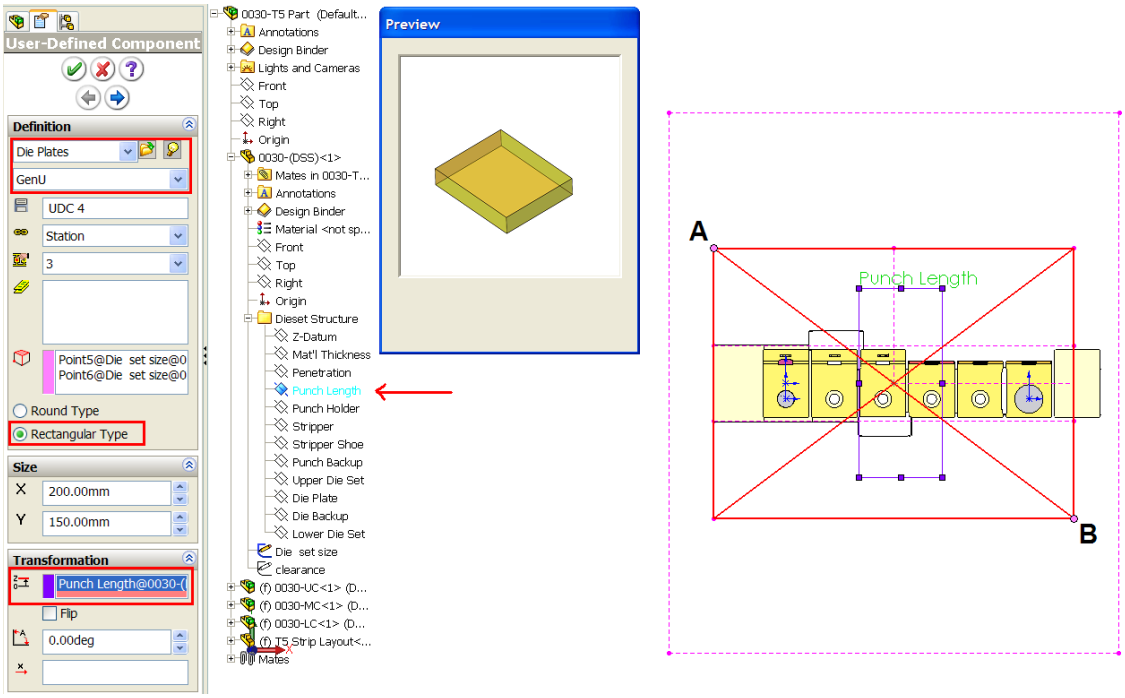
24. Save **P1 Strip Layout-QPTD.SLDASM**

Create Die Plate

1. Click **Front** 

2. Click **Create User-Defined Component**  from Punch Design Tool Bar.

3. From Assembly Tree, expand **(DSS)-QPTD**, right click and show the Sketch **Die Set size**.

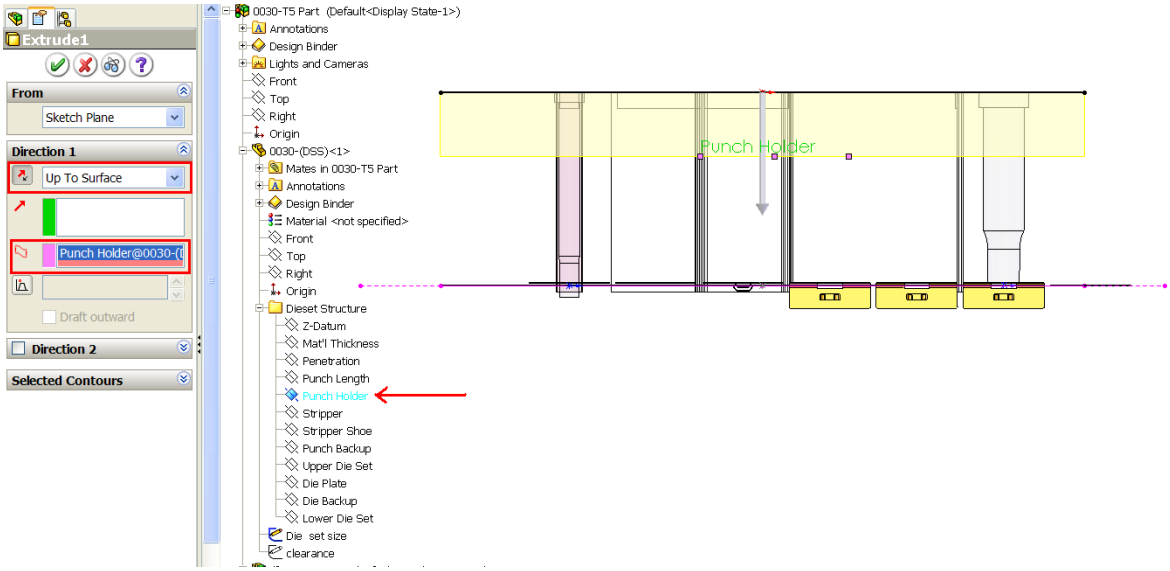


4. Click **A** & **B** to Selection. Select Function Group **Die Plates** and select PRL **GenU**, Check **Rectangular Type**, select **Punch Length** in **(DSS)-QPTD** as reference plane.

5. Click **OK** to insert the die plate.

6. Right Click on the die plate **UDC1-QPTD** and select **Edit Part**.

7. Right Click on feature **Extrude1** and select **Edit Feature**.



8. Changes extrude type to **Up To Surface** and select **Punch Holder** from **(DSS)-QPTD** as the target surface.

9. Click **OK** to confirm the changes.

10. Click **Create User-Defined Component** again from Punch Design Tool Bar.

11. Click **A** & **B** to Selection. Select Function Group **Die Plates** and select PRL **GenM**, Check **Rectangular Type**, select **Stripper** in **(DSS)-QPTD** as reference plane.

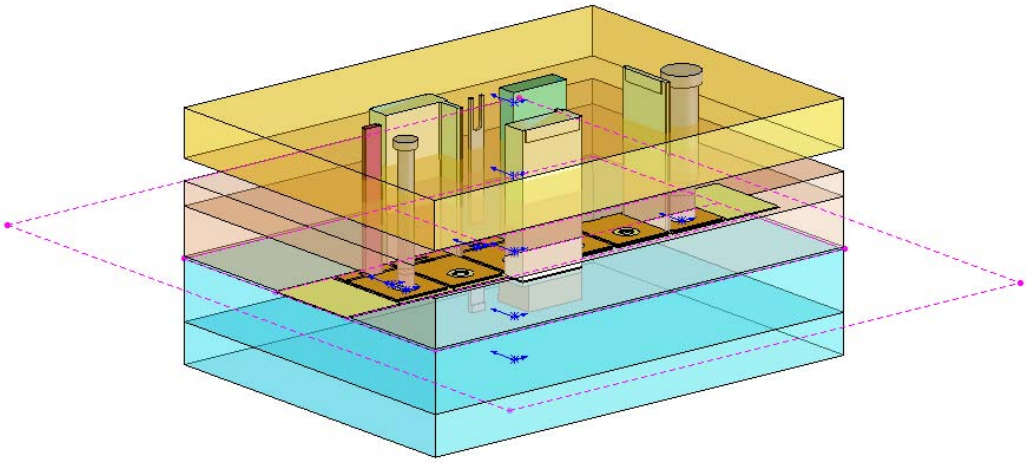
12. Click **OK** to insert the die plate.

13. Right Click on the die plate **UDC2-QPTD** and select **Edit Part**.

14. Right Click on feature **Extrude1** and select **Edit Feature**.

15. Changes extrude type to **Up To Surface** and select **Stripper Shoe** from **(DSS)-QPTD** as the target surface.

16. Click **OK** to confirm the changes.
17. Repeat Steps **10** to **16** to insert the die plate that extrude from datum plane **Mat'l Thickness** to **Stripper** of **(DSS)-QPTD.**
18. Click **Create User-Defined Component** again from Punch Design Tool Bar.
19. Click **A** & **B** to Selection. Select Function Group **Die Plates** and select PRL **GenL**, Check **Rectangular Type**, select **Die Plate** in **(DSS)-QPTD** as reference plane.
20. Click **OK** to insert the die plate.
21. Right Click on the die plate **UDC4-QPTD** and select **Edit Part**.
22. Right Click on feature **Extrude1** and select **Edit Feature**.
23. Changes extrude type to **Up To Surface** and select **Z-Datum** from **(DSS)-QPTD** as the target surface.
24. Click **OK** to confirm the changes.
25. Repeat Steps **18** to **24** to insert the die plate that extrude from datum plane **Die Backup** to **Die Plate** of **(DSS)-QPTD.**



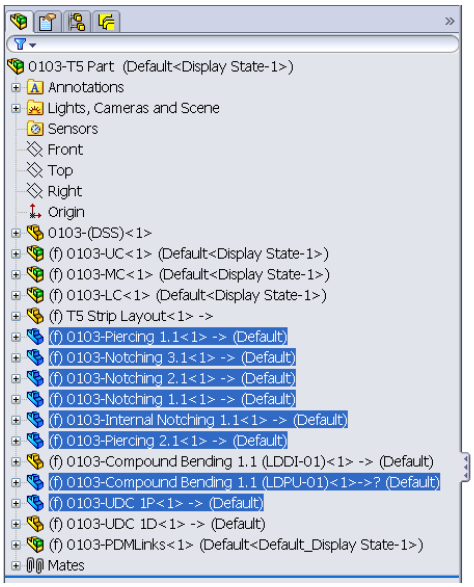
Tutorial 7. Create die set

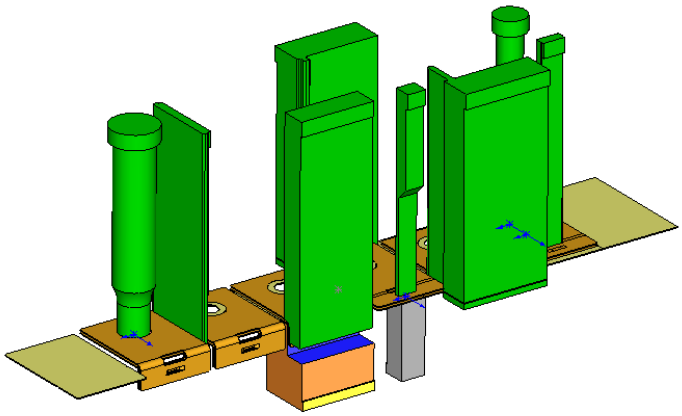
1. Open file **training\Tutorial 7\0103-T5 Part\0103-T5 Part.SLDASM**

   The sub-assemblies UC, MC & LC are used to hold the punches and die for the die set U, M & L die set subassemblies. If the PRL is a subassembly, you may need to dissolve it with SolidWorks functions.

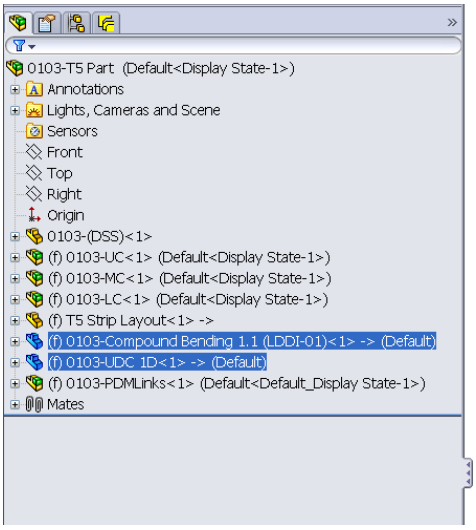
2. **RMB** select the sub-assembly **0103-UDC 1-T5<1>** and select **Dissolve Sub-assembly**

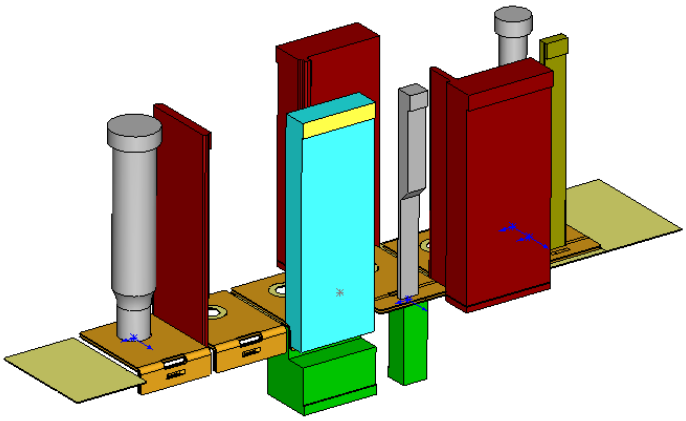
3. **Ctrl-Select** all the punches as shown below and drag & drop it to **0103-UC**<1>





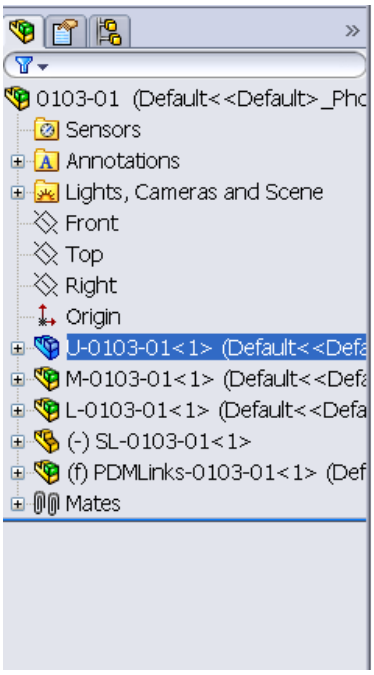
4. **Ctrl-Select** all the inserts as shown below and drag & drop it to **0103-LC**<1>

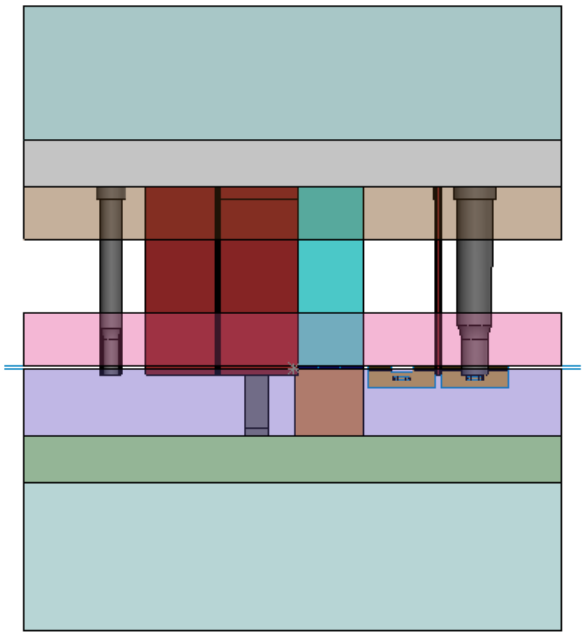




5. Expand 0103-(DDS)<1> and double click on Die set size, change **250** to **200**; **300** to **250**

6. Select **3DQuickPress**, **Die set Design**, **Create Die set** to create die set assembly





Insert Socket Head Cap Screw

7. **RMB** select **U-0103-01<1>** and select **Open Assembly** from shortcut menu

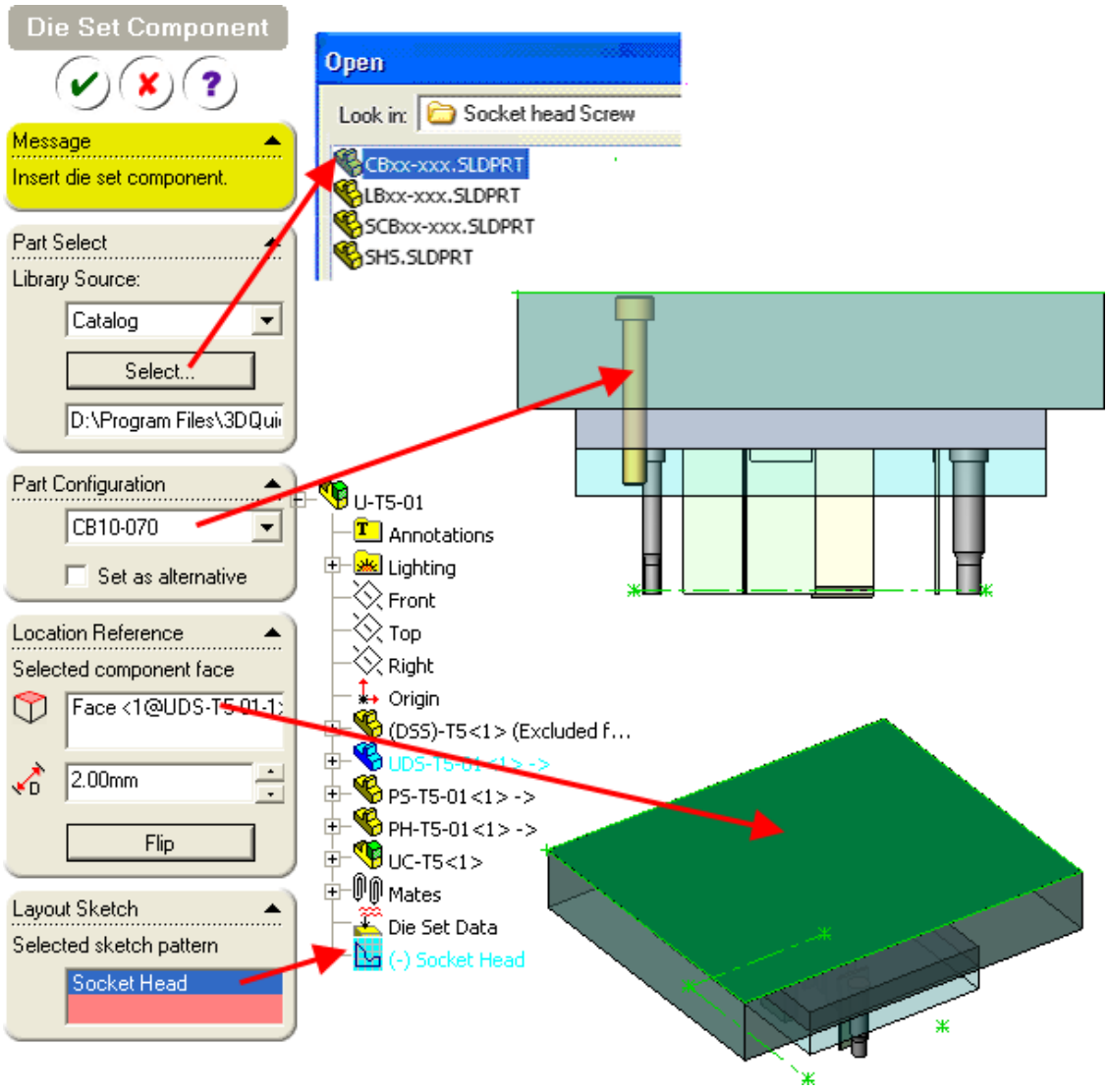
8. Select datum plane **Front from** assembly tree**.** From the **3DQuickPress** pull down menu, select **3DQuickTools->Layout Sketch**

9. Input X-Size 150mm; Y-Size 120mm, click OK when finished

10. Select the newly create sketch from Feature Manager – Hold the LMB until the sketch name is high lighted 

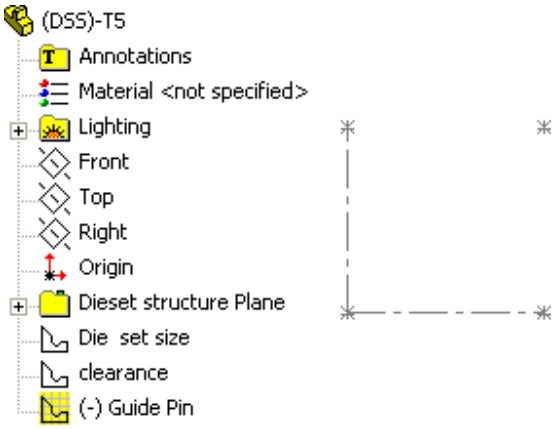
, rename the sketch to **Socket Head**

11. Click Component Layout from 3DQuickPress tool bar, after select the Location Reference face, switch to Bottom view to get the best visualization.



Create a sketch which is shared amongst M and L assembly

12. **RMB** select 0103-(DSS)<1>, and select Open Part from shortcut menu

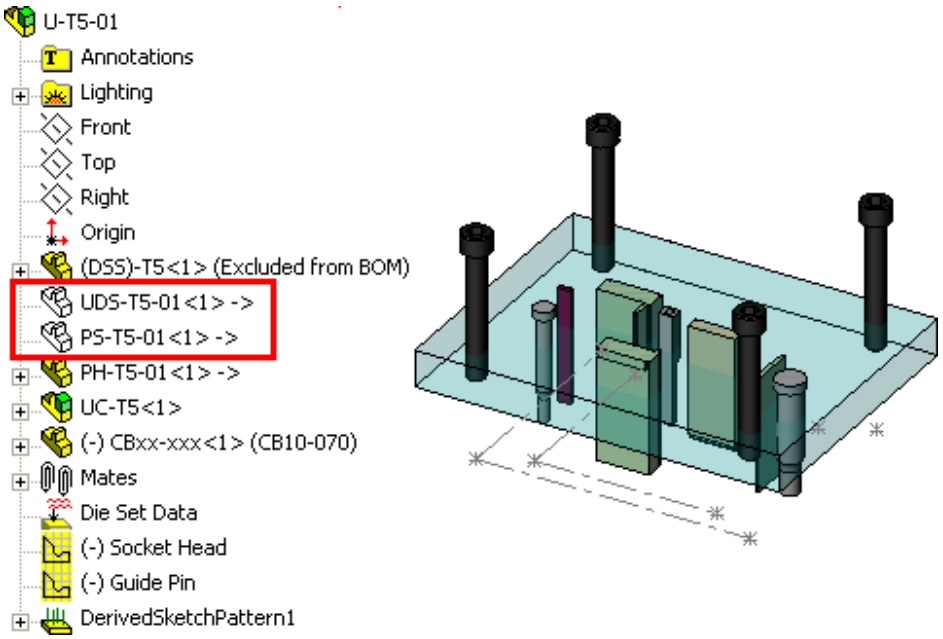


13. Repeat **step 8** to create another layout sketch for Guide Pin, X-size = 100, Y-size = 95

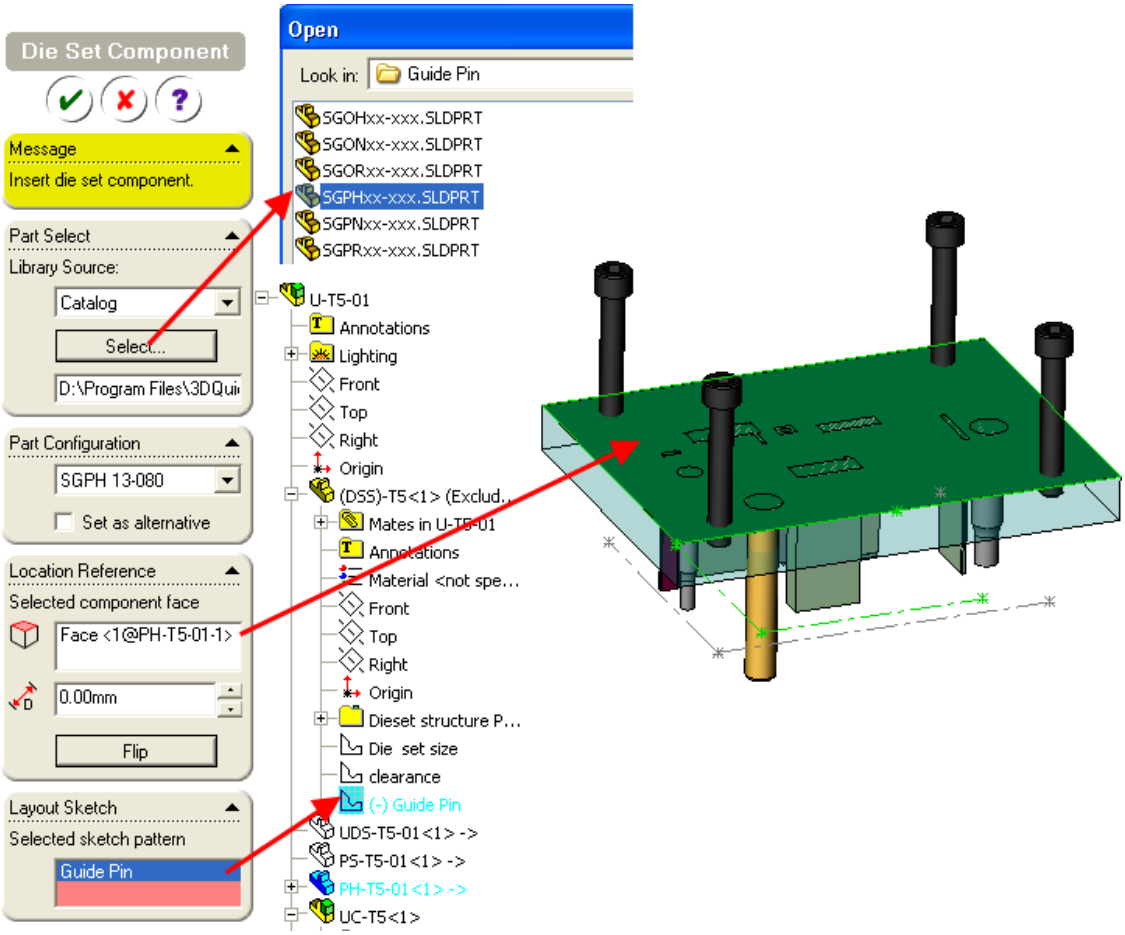
14. Rename the sketch to **Guide Pin**

15. Click **Control-Tab** to back to **U-0103-01 Assembly**

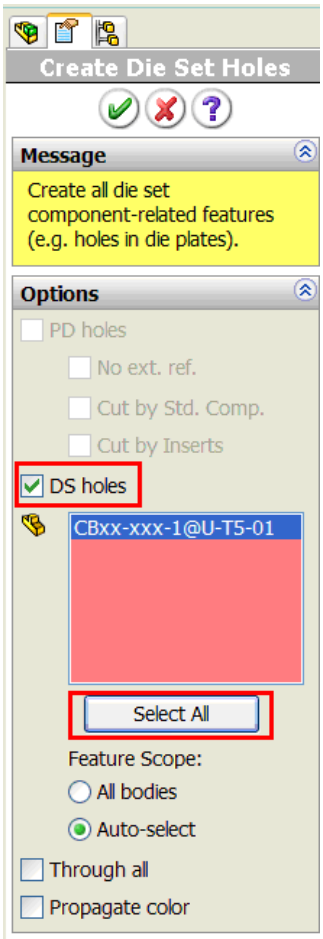
16. Hide the top plates



17. Repeat step 11 and select Guide Pin and the Top Face of Punch Holder



18. From Menu, select 3DQuickPress->Dieset Design->Create Die Set Holes…

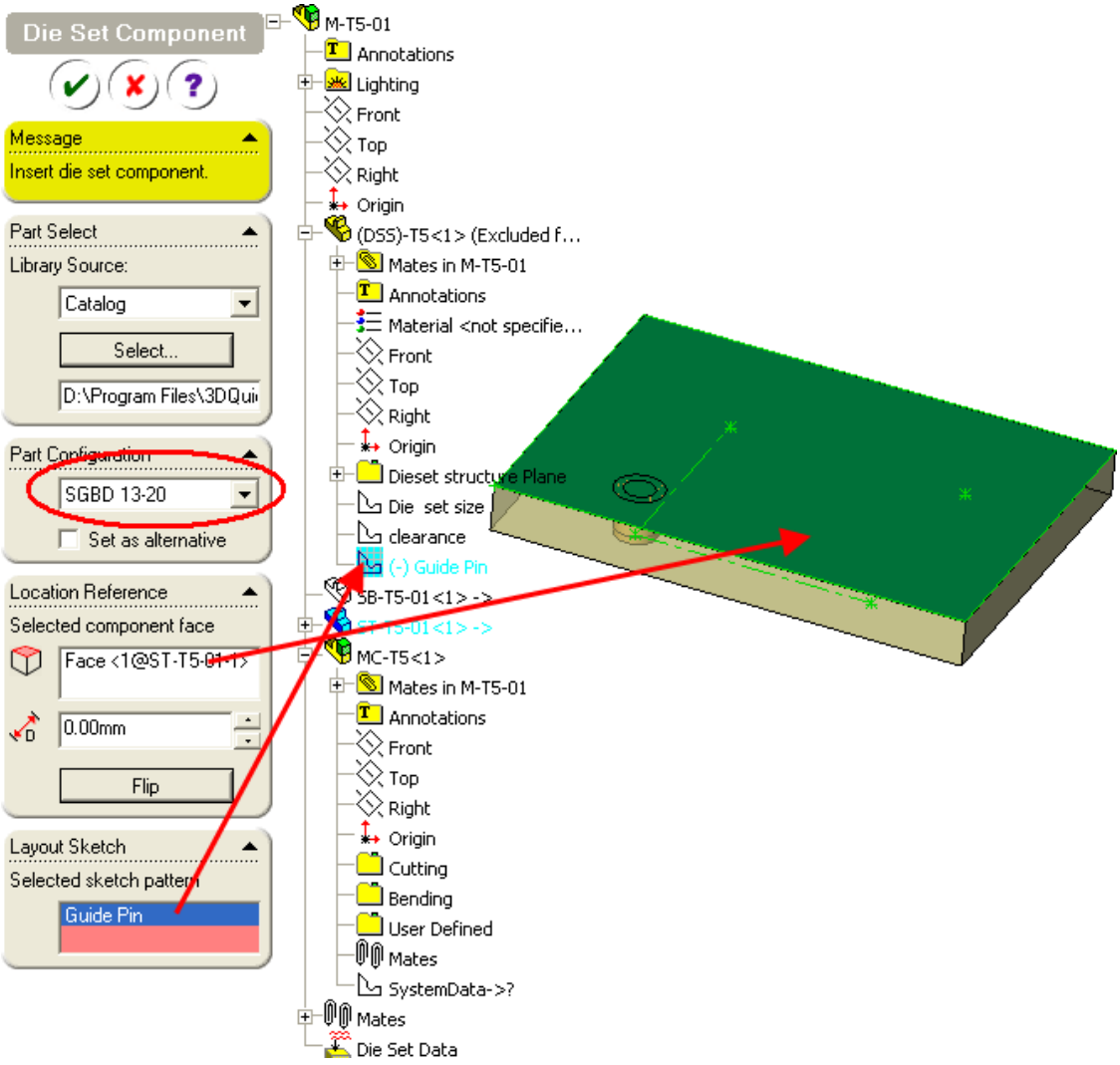


19. Check DS Holes, and then click the "Select All" button to select the screw.

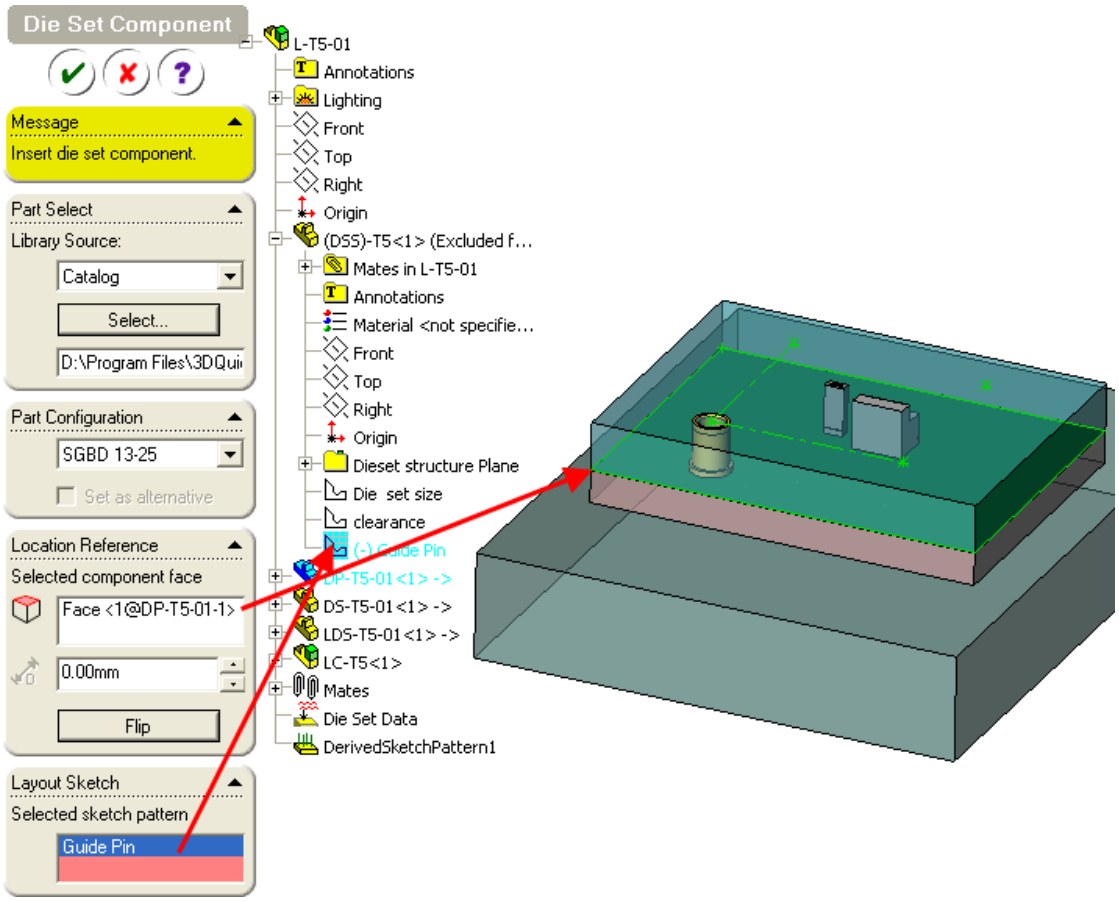
20. Click OK to confirm create hole.

21. Open M-0103-01.SLDASM.

22. Repeat step 11 and select Guide Pin Bush/SGBD 13-20 and the top face of stripper plate.



23. Open **L-0103-01.SLDASM**., repeat step 11 and select Guide Pin Bush/SGBD 13-20 and the bottom face of DP-T5.

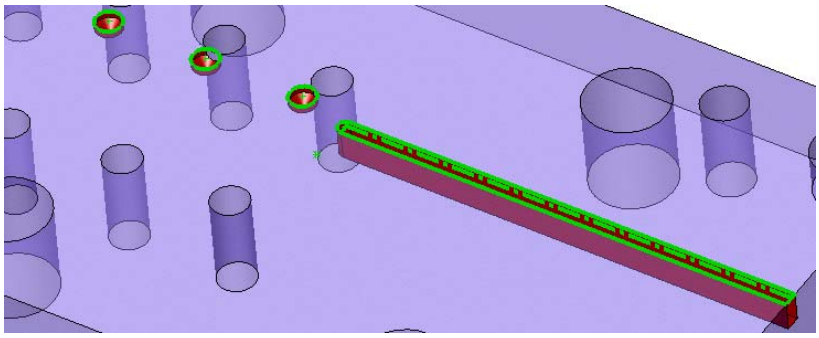


24. From **3DQuickPress** pull down menu **Die set Design** select **Create Die Set Holes**

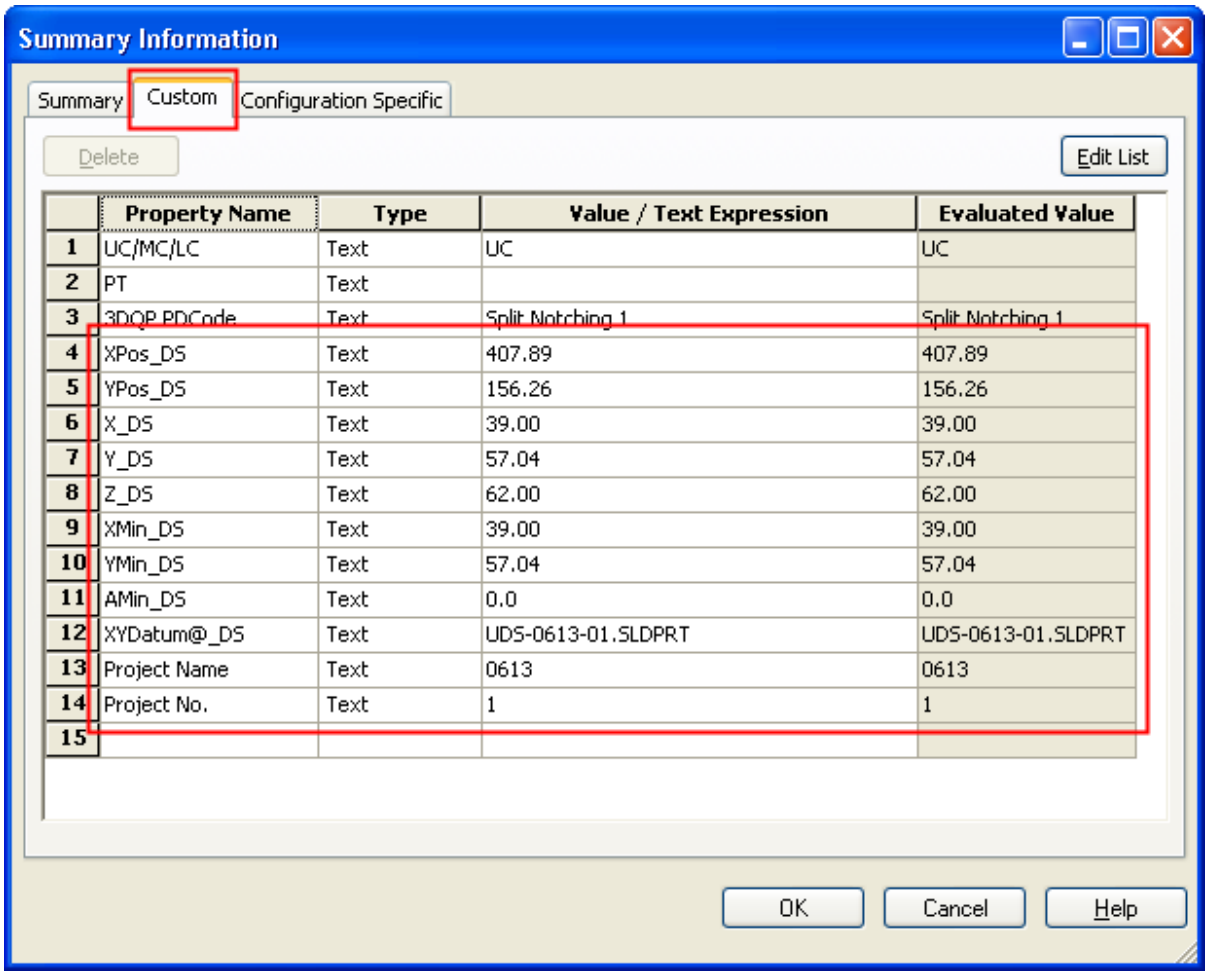
25. Click **3DQuickPress**, **3DQuickTools**, **Interference Detection**, select die plate and stripper to **Targets** selection box, strip layout to **Tools** selection box.

26. Double click on the link to open the interference parts, set display to wire frame

27. Create a sketch on top face of die plate and use 3DQuickPress Box to create circle or rectangle to create the clearance.



28. Back to die set assembly, click **3DQuickPress**, **3D QuickTools**, **Die set Components Properties Update.** Change to the top view of the Die and select the lower left corner for the Datum point**.** To view the results of this command, open any non-component and select FILE/PROPERTIES/CUSTOM Tab and review XMin_DS, Ymin_DS, Z_DS for the minimum Stock Material. Other properties added are highlighted below.



	Property Name	Type	Value / Text Expression	Evaluated Value
1	UC/MC/LC	Text	UC	UC
2	PT	Text		
3	3DQP PDCode	Text	Split Notching 1	Split Notching 1
4	XPos_DS	Text	407.89	407.89
5	YPos_DS	Text	156.26	156.26
6	X_DS	Text	39.00	39.00
7	Y_DS	Text	57.04	57.04
8	Z_DS	Text	62.00	62.00
9	XMin_DS	Text	39.00	39.00
10	YMin_DS	Text	57.04	57.04
11	AMin_DS	Text	0.0	0.0
12	XYDatum@_DS	Text	UDS-0613-01.SLDPRT	UDS-0613-01.SLDPRT
13	Project Name	Text	0613	0613
14	Project No.	Text	1	1
15				

29. The output below is a BOM created with SolidWorks to illustrate how the File Properties can be used in a drawing.

ITEM NO.	PART NUMBER	DESCRIPTION	XMin_DS	YMin_DS	Z_DS	QTY.
1	UDS-0027-01		300.00	300.00	50.00	1
2	PS-0027-01		150.00	250.00	17.00	1
3	PH-0027-01		150.00	250.00	20.00	1
4	Default		11.00	11.00	62.00	1
5	P3-0027		8.00	2.88	62.00	1
6	CB1-0027P		8.50	26.00	65.50	1
7	Default		16.00	16.00	62.00	1

8	NT1-0027	30.00	21.69	62.00	1
9	NT2-0027	29.09	13.31	62.00	1
10	NT3-0027	2.50	26.00	62.00	1
11	Lan1-0027	9.00	5.00	60.50	1
12	CB10-070				4
13	MS12-80				4
14	SGOH 16-080				4
15	SB-0027-01	150.00	250.00	10.00	1
16	Spring-0027-01	25.00	25.00	57.30	6
17	ST-0027-01	150.00	250.00	20.00	1
18	SGBD 16-22				4
19	MSB 13-80				6
20	CB 6-25				10
21	MS12-30				2
22	DP-0027-01	250.00	150.00	25.00	1
23	SGBL 16-25				4
24	Stock Guide1-0027-01	100.00	62.20	18.00	1
25	LBracket-0027-01	62.20	25.00	17.00	1
26	CB 8-20				2
27	DS-0027-01	150.00	250.00	17.00	1
28	LDS-0027-01	300.00	300.00	55.00	1
29	1-0027	26.00	20.50	25.00	1
30	UDC1-0027	10.00	8.00	25.00	1
31	CB10-055				4
32	GLH10-25.00	13.00	13.00	60.00	8
33	MSW 16				8
34	SL-0027-01				1

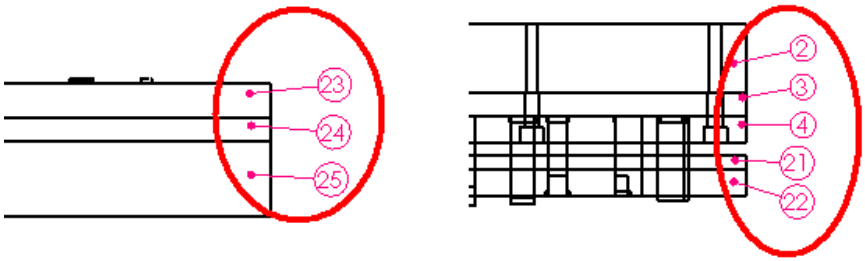
Tutorial End

Tutorial 8. Detailing

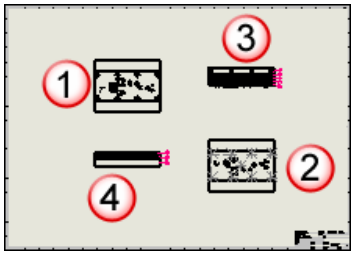
Lab 1 Sort Balloon

The die set assembly for this Lab has 2 display states for Upper assembly and lower assembly. Use display state over configuration is mainly due to better performance.

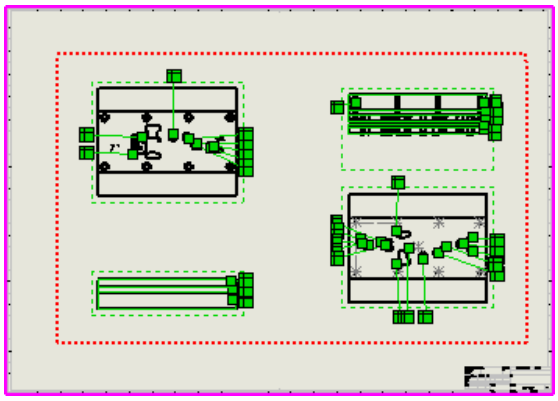
1. Open drawing 0025-01.slddrw in **\training \Detailing\BOM**
2. Select any view and click Insert, Tables, Bill of Material, browse for bom template **3dqpballoon.sldbomtbt in \training \Detailing\BOM**
3. Click Edit, Undo to remove the BOM. (Step 2 is used to set SolidWorks default template, we don't use this table)
4. Click Balloon to manually assign balloon to die plates



5. Click view **1** and then click **AutoBalloon**, click view 2 , 3 & 4 to add them to selection



6. Window select all balloon and click **3DQuickPress**, **Drawing & Machining**, **Sort Balloon**. Click **OK** to accept the default setting. Click **Yes** to create the BOM table.

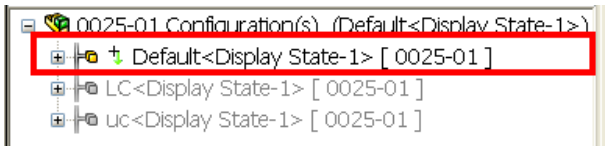


7. RM select BOM Item no* and select Sort, Ascending

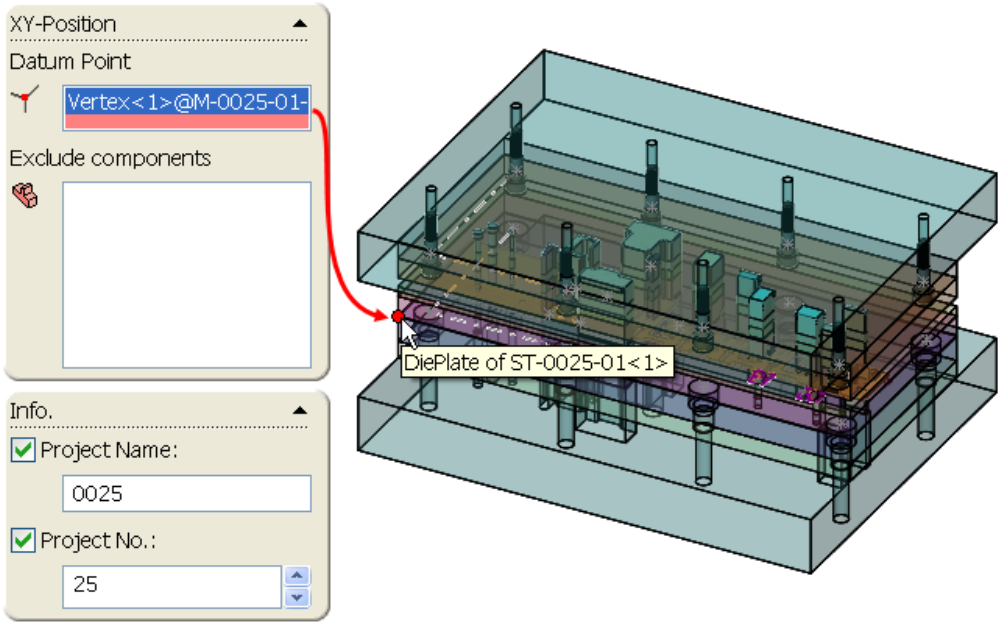
	A	B	C	D	E	F
1	Item No.	PART NUMBER	DESCRIPTION	STOCK SIZE	Position in Die Set	QTY.
2	23	25-01		XX	,	1
3	24	PS-0025-01		XX	,	1
4	25	PH-0025-01		XX	,	1

Right Click

8. Open **0025-01.SLDASM** and switch back to Default display state



9. Click **3DQuickPress**, **3DQuickTools**, **Die Set component Properties Update.** Select the corner form die plate as show. Click **OK** to finish

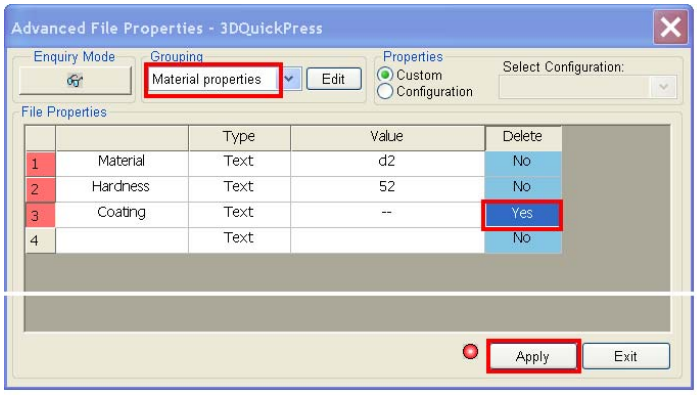


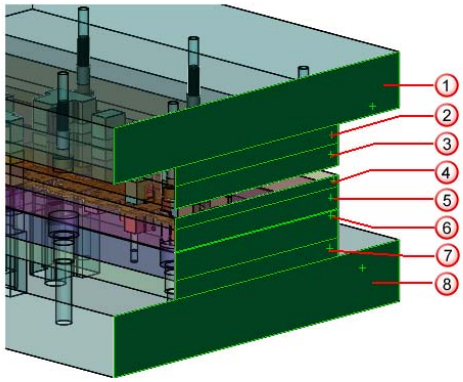
10. Switch back to drawing and click Rebuild the stock size and position will be updated.

ItemNo	PART NUMBER	DESCRIPTION	STOCK SIZE	Position in Die Set	QTY.
1	Default		5X5X62.0000	66.2592,5.0000	1
2	Default		7X7X62.0000	41.4489,7.0000	1
3	0025-Piercing 2		9X18X62.00	110.26,18.00	1
4	0025-Split Notching 5		45.58X21.81X62.00	176.87,21.81	1
5	0025-Split Notching 6		19X9.21X62.00	394.51,9.21	1
24	PS-0025-01		5X21X59.00	364.18,21.00	
25	PH-0025-01		450X200X20.00	225.88,200.00	1
26	SB-0025-01		450X200X10.00	225.64,200.00	1
27	ST-0025-01		450X200X20.00	225.51,200.00	1
28	DP-0025-01		450X200X25.00	227.79,200.00	1
29	DS-0025-01		200X450X17.00	226.6,200.00	1
30	LDS-0025-01		350X450X55.00	226.2,350.00	1

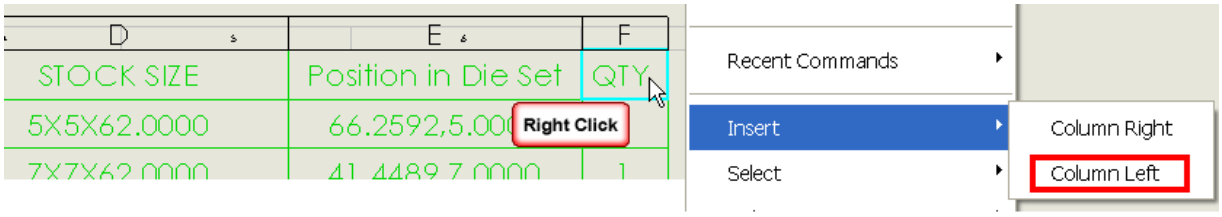
11. Switch back to **0025-01.SLDASM** window and click **File Properties** 

12. Set up the table as shown and select all the die plate 1-8, click Apply button. And then Exit.

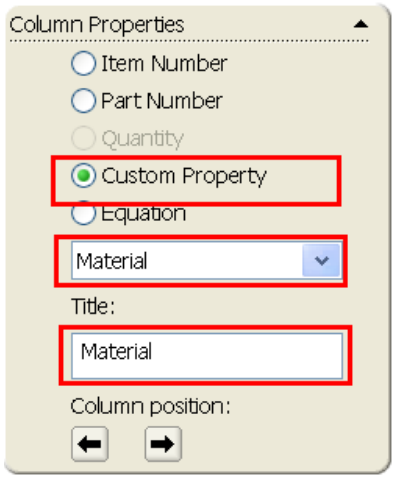




13. Switch back to **0025-01.SLDDRW** window. RMB select **Qty** column and click **Insert**, **Column Left**.



14. Check **Custom Property**, select **Material** and column Title = **Material**



15. The information created in Step 12 was filled.

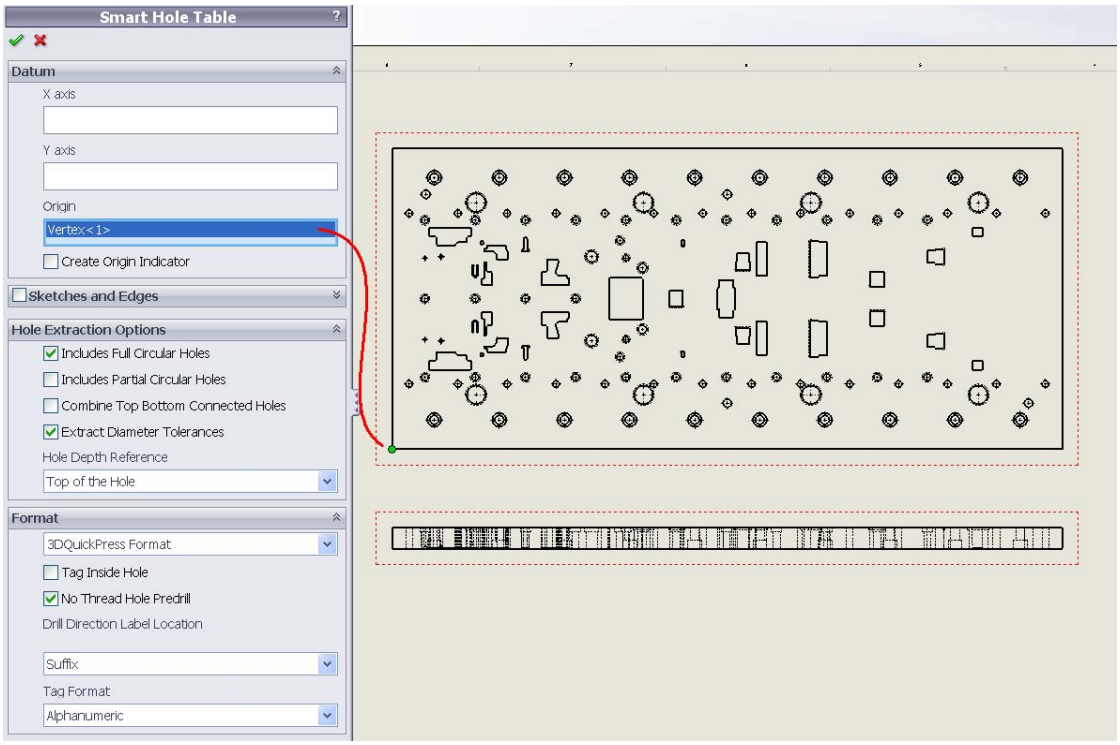
20	0025-Bend 3 [1] (LIIPP-01)		[illegible]			
21	0025-UDC 2		42.5X62.42X25.00	133.18,62.42	Material <not specified>	1
22	CB10-050		7X7X34.00	146.15,7.00	XXX	8
23	UDS-0025-01		350X450X50.00	225,350.00	d2	1
24	PS-0025-01		200X450X17.00	225,200.00	d2	1
25	PH-0025-01		450X200X20.00	225.58,200.00	d2	1
26	SB-0025-01		450X200X10.00	225.64,200.00	d2	1
27	ST-0025-01		450X200X20.00	225.51,200.00	d2	1
28	DP-0025-01		450X200X25.00	227.79,200.00	d2	1
29	DS-0025-01		200X450X17.00	226.6,200.00	d2	1
30	LDS-0025-01		350X450X55.00	226.2,350.00	d2	1

Lab 2. Smart Hole Table

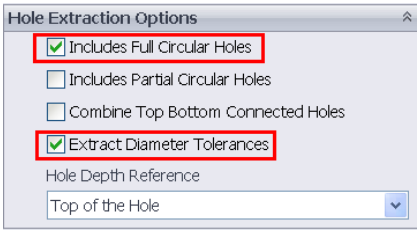
1. Open drawing DP-0052-01.SLDDRW in **\training \Detailing\ HoleTable**

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

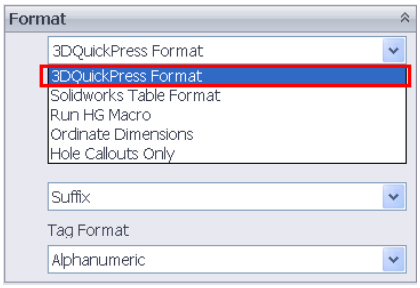
3. Select a vertex as the origin



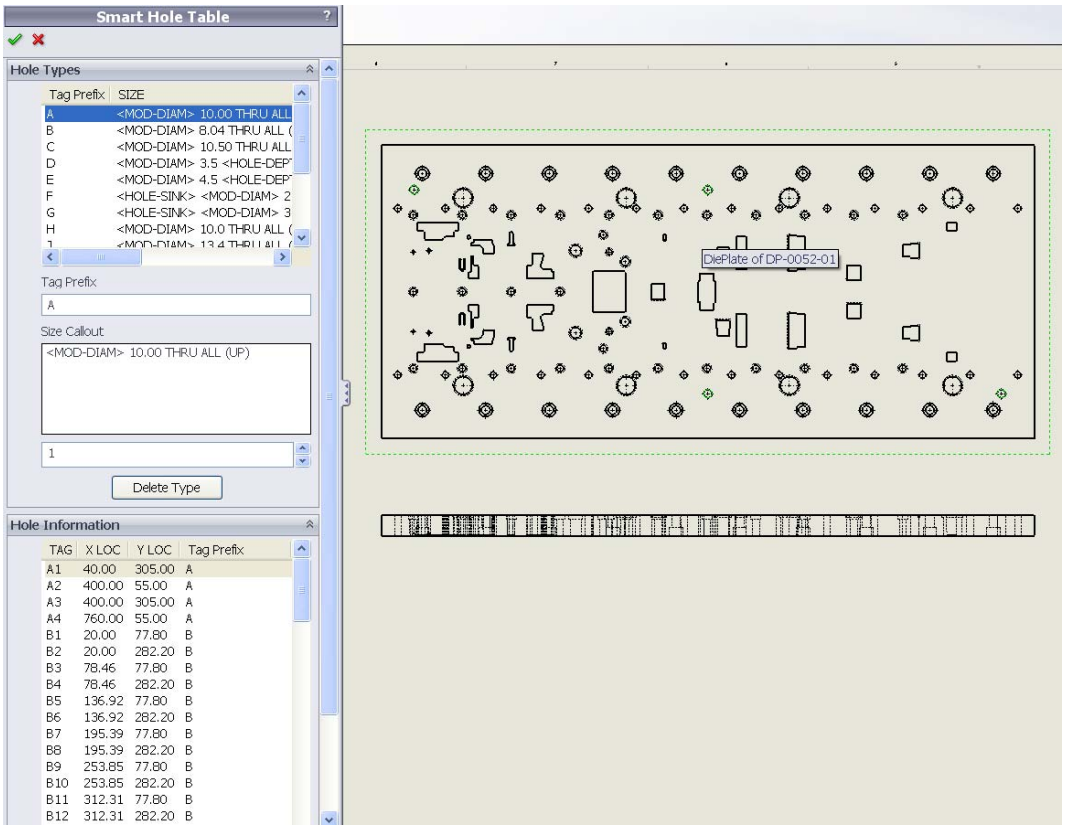
4. Select the hole extraction options **Includes Full Circular Holes** and **Extract Diameter Tolerances**.



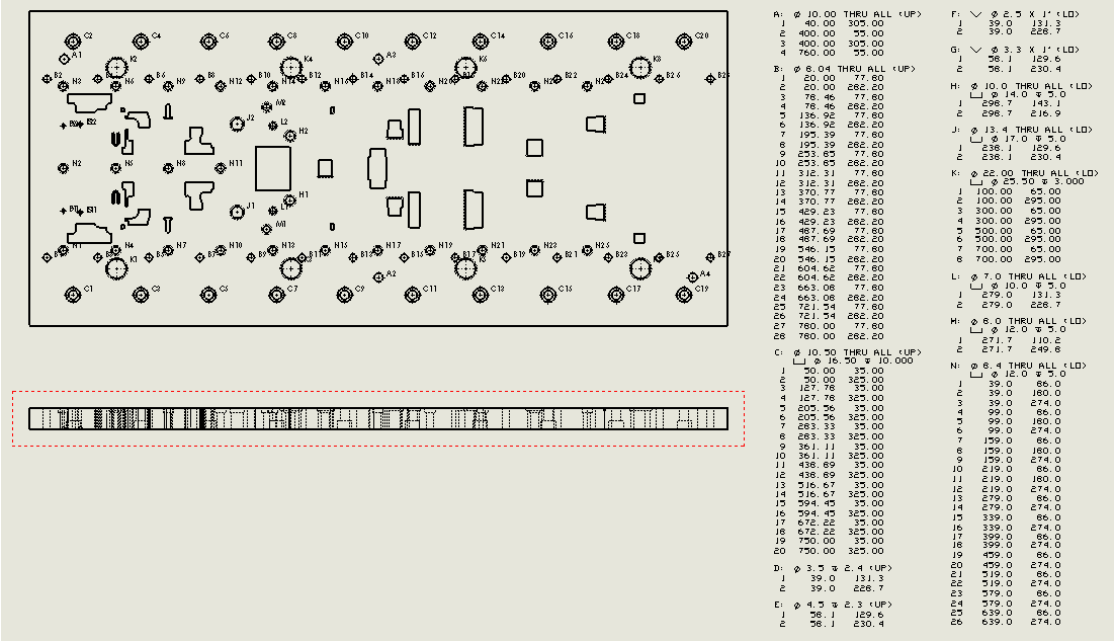
5. Select hole table format **3DQuickPress Format**.



6. Click **OK** to proceed.
7. 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.

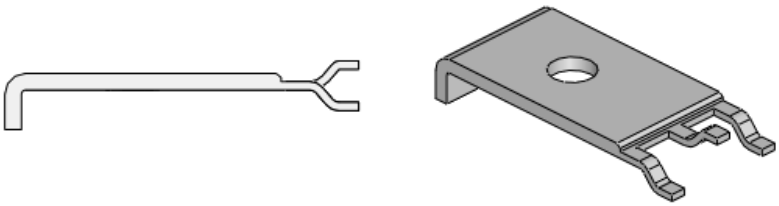


8. Click **OK** to create the hole table.

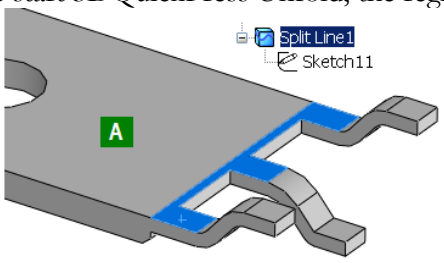


Tutorial 9. Self Study: Advanced Unfolding: Multiple Thickness Part Unfolding

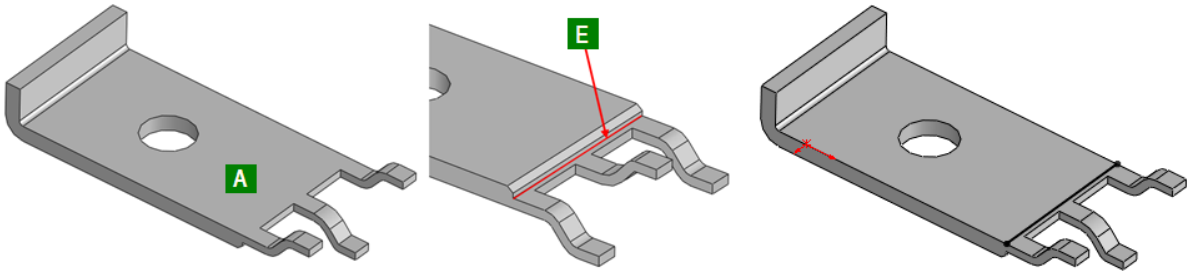
This tutorial is to show how to handle parts that have the thinning feature, function to unfold part that has multiple thickness is introduced. Below is an example of the steps to unfold a multiple thickness part.



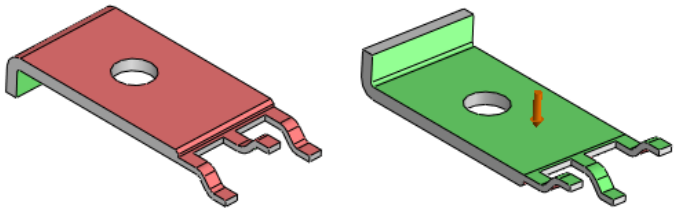
Before start 3DQuickPress Unfold, the regions of different thickness must be separated by **Split Face.**



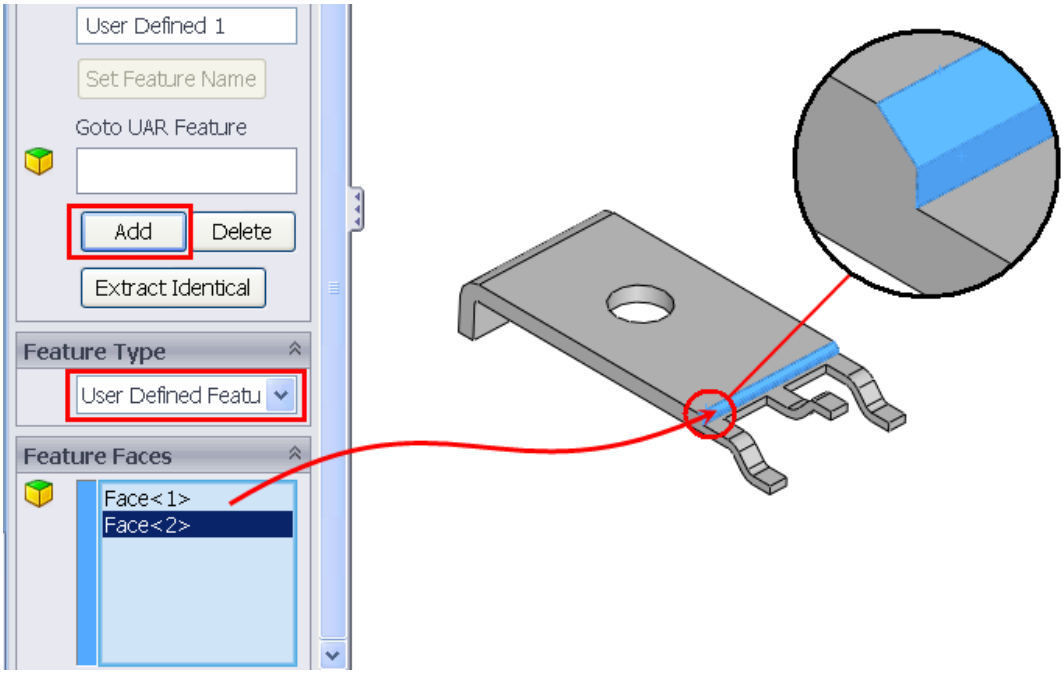
1. Click Face A and click **Insert**, **Sketch**



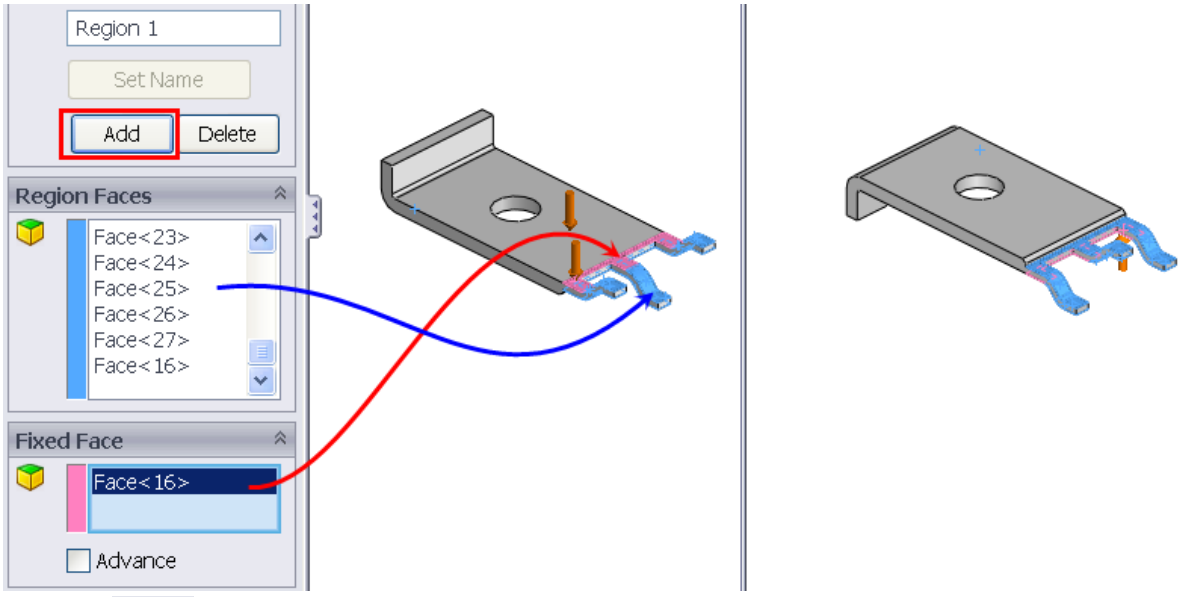
2. Select Edge E, click Tools, Sketch Tools, Convert Entity
3. Click **Insert**, **Curve**, **Split Line**, select Face A to Faces to Split
4. Click OK to finish the separation of regions
5. Click **3DQuickPress**, **Unfold Part**, **Unfold Part** to start unfold
6. Select Face A for Fixed Face
7. RMB **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

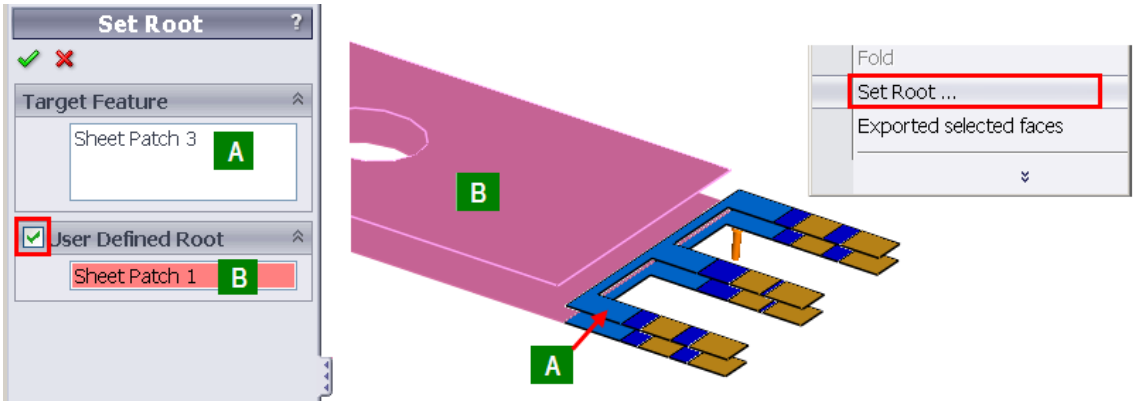


10. Click Next Button  and add a new region

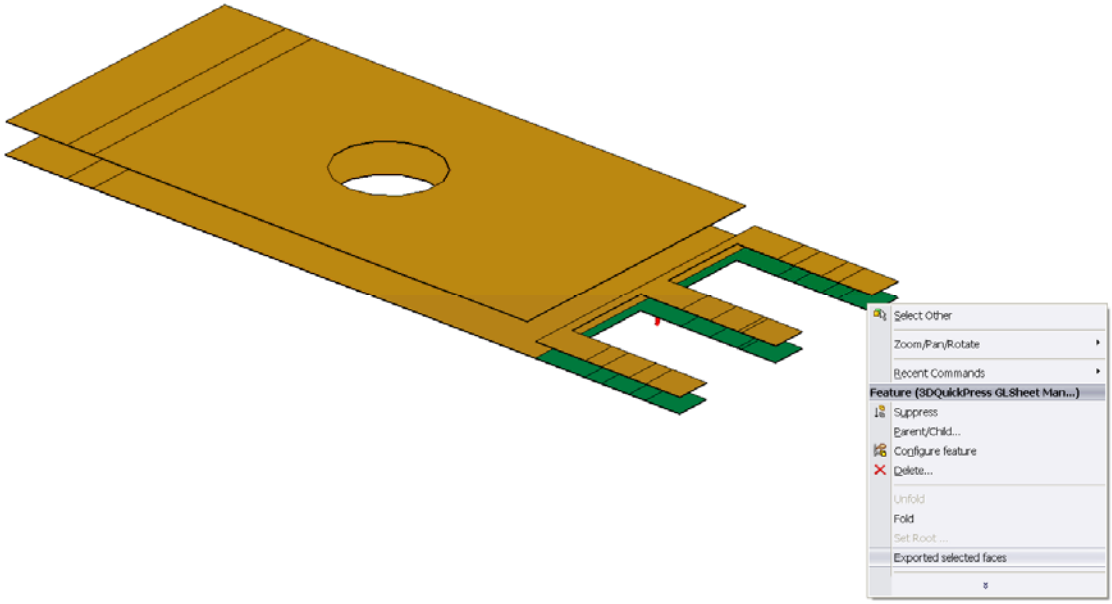


11. Click OK  to exist UAR mode.

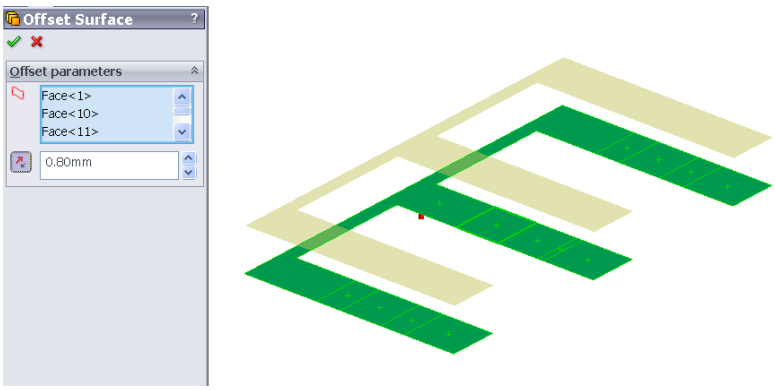
12. RMB select Face A, click **Set Root**, check **User Defined Root**, click face B



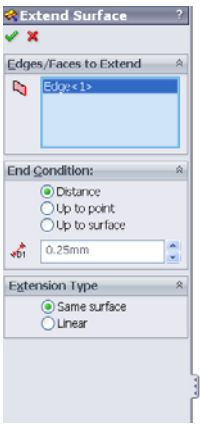
13. RMB **Sheet Metal Object** and click **Unfold All**
14. Select the green faces and RMB select **Export selected faces**

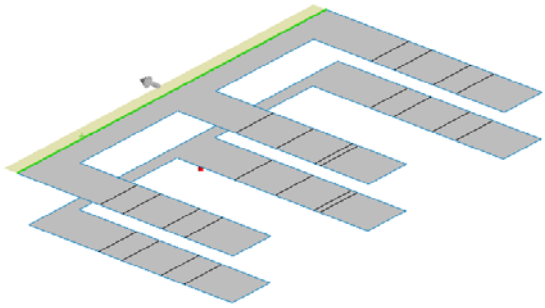


15. Hide the unfold part
16. Click **Insert**, **Surface**, **Offset**, select the exported faces and set offset value to 0.8mm upwards

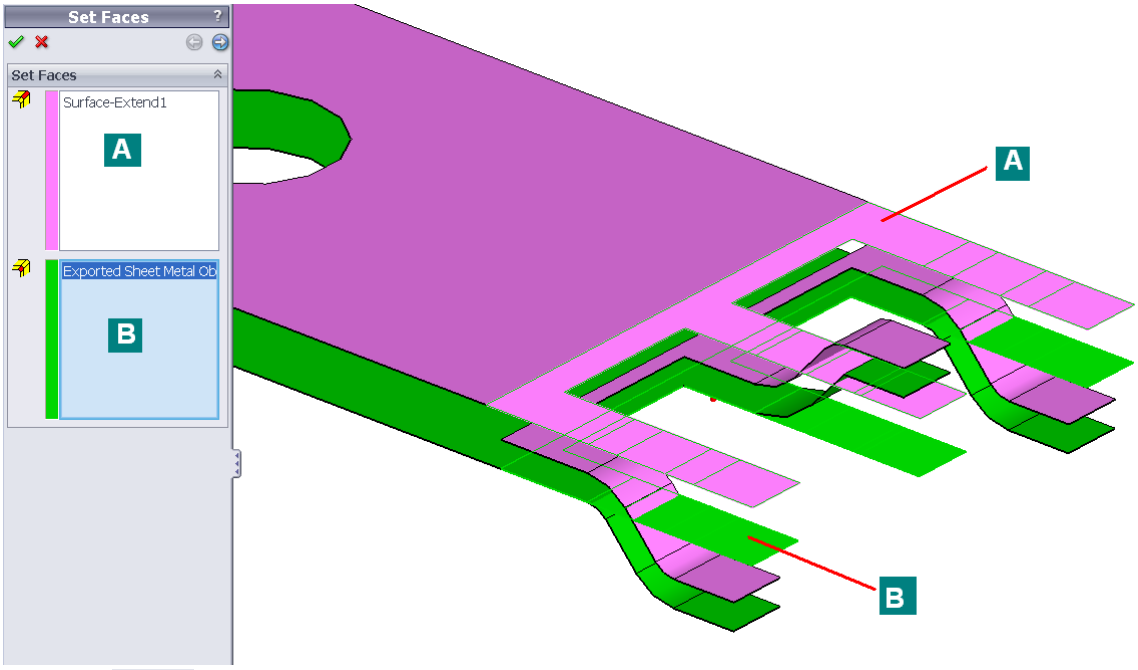


17. Click **Insert, Surface, Extend Surface**, select the edge on the offset face and set extend value to 0.25mm.



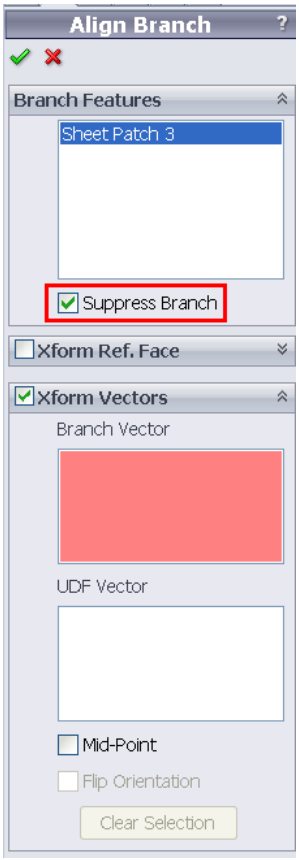


18. Show the unfold part
19. RMB on **Step 0** of **User Defined 1** Feature on the feature tree, and select **Set Faces**
20. Select face A as top and face B as bottom



21. Click OK  to exit
22. RMB on **Step 0** of **User Defined 1** Feature and select **Set Branch Feature XForm**

23. Select **Suppress Branch**



24. Click OK  to exit

Tutorial finished