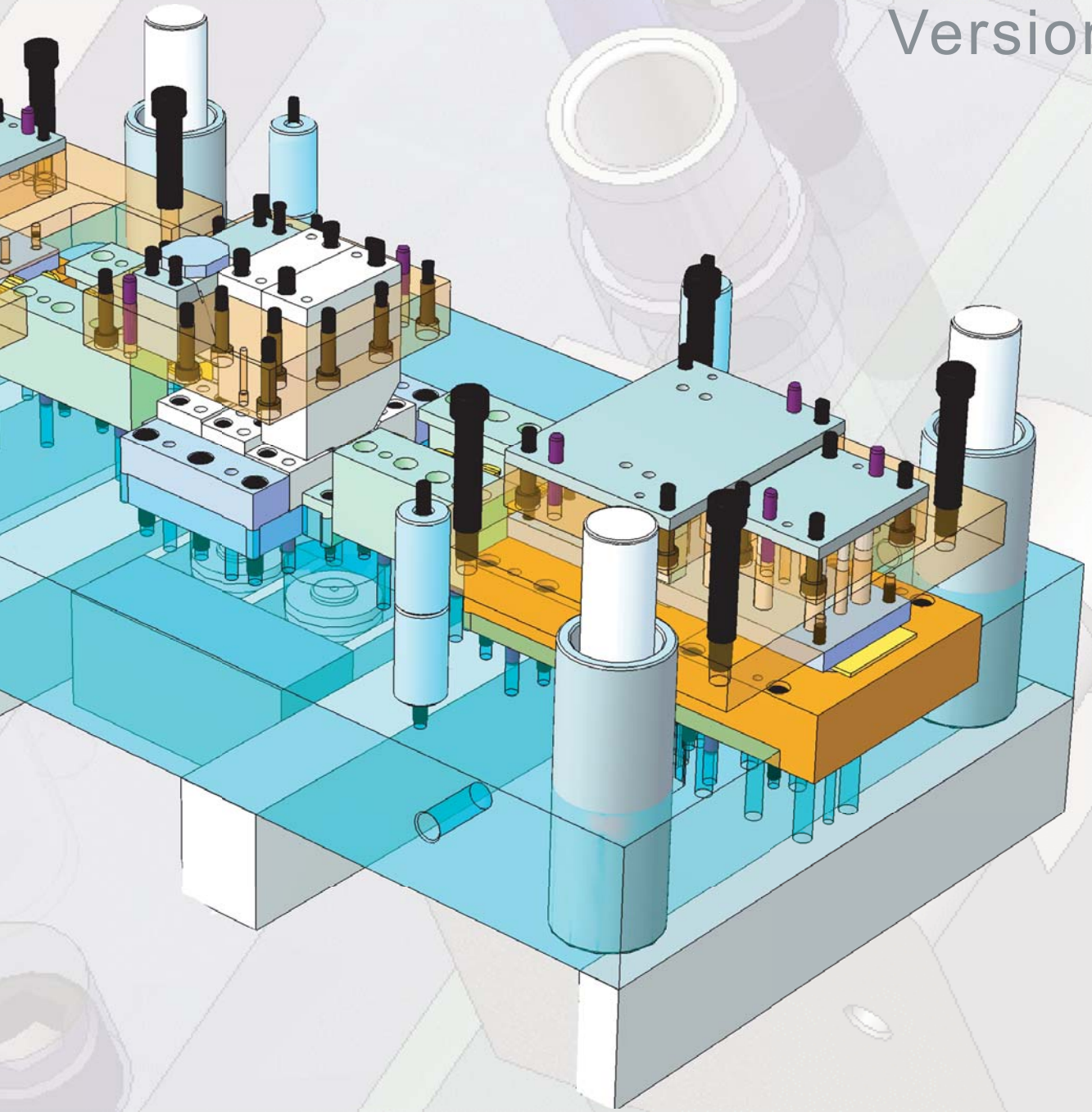
3D QuickTools Limited

Training Manual

3DQuickPress
Version 5



3DQuickPress Version 5

3D QuickTools Ltd.,
Strategic Technology Solutions, Inc.,
and Computer Aided Technology, Inc.
165 Arlington Heights Road
Suite 101
Buffalo Grove, IL 60089
www.cati.com
www.callsts.com
www.3dquicktools.com

Table of Contents

CLASS INTRODUCTION

- About this class
- Prerequisites
- Conventions

About This Course

The goal of this course is to teach the user how to use 3DQuickPress in conjunction with the SolidWorks® software in order to effectively and efficiently design and model stamped components and the progressive die sets that are used to manufacture the stamped components.

Prerequisites

Students attending this course are expected to have the following:

- Experience designing progressive dies.
- Experience with Windows operating system.
- Completed the online tutorials that are integrated in the SolidWorks software. You can access the online tutorials by clicking Help, Online Tutorial.
- SolidWorks Essentials
- SolidWorks Assembly Modeling
- SolidWorks Drawings
- SolidWorks Surface Modeling

Course Length

The recommended minimum length of this course is 4 days.

Course Design Philosophy

This course is designed around a process or task-based approach to training. A process-based training course emphasizes the processes and procedures you follow to complete a particular task. By utilizing case studies to illustrate these processes, you learn the necessary commands, options and menus in the context of completing a task.

Using this Book

This training manual is intended to be used in a classroom environment under the guidance of an experienced 3DQuickTools instructor. After classroom training the manual can be used as a reference. It is not intended to be a self-paced tutorial. The examples and case studies are designed to be demonstrated “live” by the instructor.

About the Training Files

A complete set of the various files used throughout this course can be obtained from the instructor either by copying them directly from the training workstation in class or the instructor sending the files via ftp or other transfer means.

Conventions Used in this Book

This manual uses the following typographic conventions:

Windows XP

The screen shots in this manual were made using the 3DQuickPress software and SolidWorks software running on Windows XP. You may notice differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

Use of Color

The 3DQuickPress and SolidWorks user interfaces make extensive use of color to highlight selected geometry and to provide visual feedback. This greatly increases the intuitiveness and ease of use of the software. To take maximum advantage of this, the training manuals are printed in full color.

Also, in many cases, we have used additional color in the illustrations to communicate concepts, identify features, and to otherwise convey important information.

Color Schemes

Out of the box, SolidWorks software provides several predefined color schemes that control, among other things, the colors used for highlighted items, selected items, sketch relation symbols, and shaded previews of features.

The color scheme for various case studies varies because some colors are more visible and clear than others when used with different colored parts.

In addition, the viewport background has been changed to plain white so that the illustrations are more clearly visible on the white paper of the manual.

As a result, the images you see on your screen may not exactly match those in the book.

Chapter 1

INTRODUCTION TO 3DQUICKPRESS

- Introduction
- Data Flow
- Design Trees
- Basic Workflow
- Alternative Workflow

DESIGN ENVIRONMENT

3DQuickPress has five design stages. The documents of each stage are linked together according to the design work flow.

These documents are:

- Unfold part document
- Strip layout design part document
- Punch design assembly document
- die set design assembly document
- Die component drawings for detailing part and assembly documents

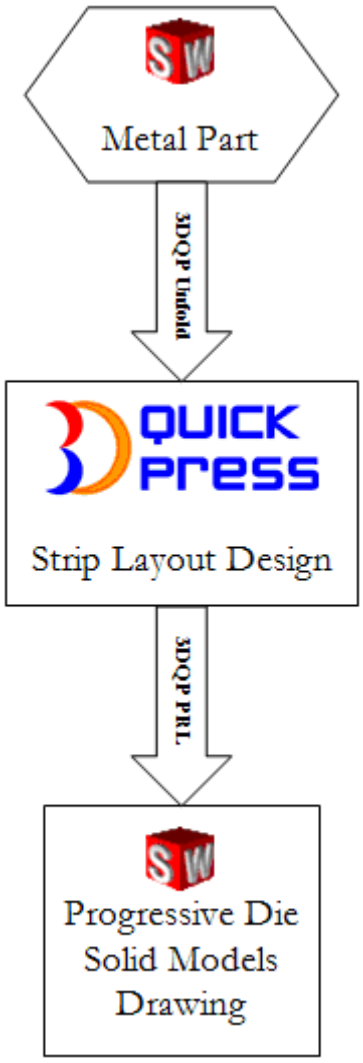
Design process and changes made during the die design stage will propagate according to the design workflow below:

- 3D Part
- Unfold
- Strip Layout
- Punch
- Die Set

For design changes performed after the die is fabricated, the design change workflow is different. For fast turnaround, it is recommended to change the affected die set component directly instead of driving the change from the ground up.

DATA FLOW

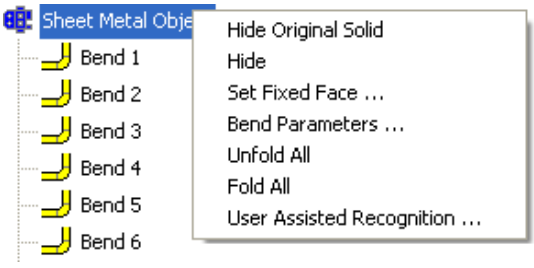
3DQuickPress adds to the SolidWorks Data set by creating a parallel set of data in the unfold part file. This 3DQuickPress data is used to create the strip layout part. This 3DQuickPress strip data is used in the punch design assembly and then converted back to SolidWorks geometry when creating the die set assembly automatically. The final assembly and drawings are created from SolidWorks Models.

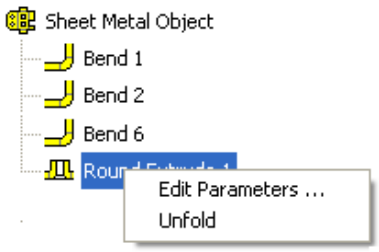


DESIGN TREES

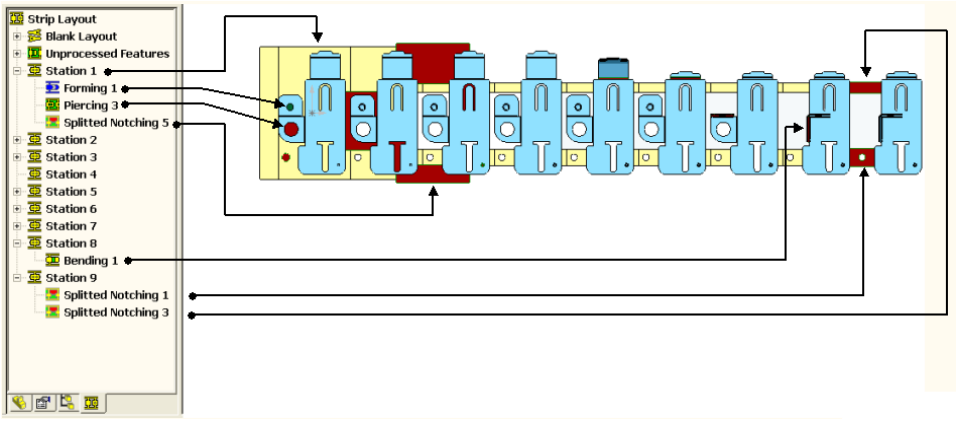
3DQuickPress follows the SolidWorks user interface standard. The PropertyManager and toolbars are employed extensively throughout the application. The Unfold Manager, Design Tree, and Strip Layout Design Tree are all similar to the SolidWorks FeatureManager® design tree.

The Unfold Manager Design Tree is made available after the Unfold part icon , in the 3DQuickPress toolbar, is used to unfold the sheet metal part. This design tree lists all the recognized and unrecognized features of the sheet metal part regarding the manufacturing processes or steps. All of the icons in the tree can be right-clicked to a shortcut menu to edit the feature's properties and change its state to folded or unfolded state.

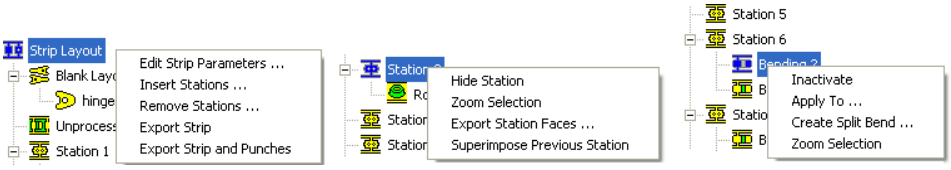




The Strip Layout Design Tree is the core of the stamping design process with 3DQuickPress. This design tree provides an outline view of the active strip layout making it easy to see how the part is processed in each station.



Shortcuts are available for each object in the Strip Layout Design Tree to provide an easy-to-use system to manipulate sheet metal process operations (features).

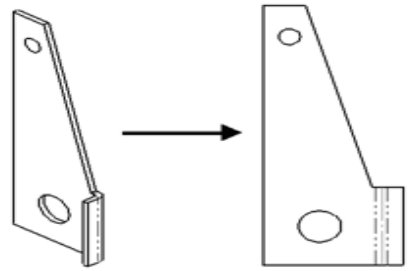
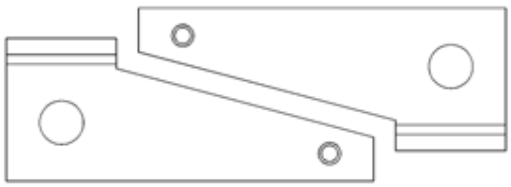
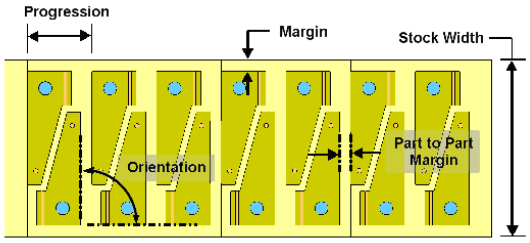
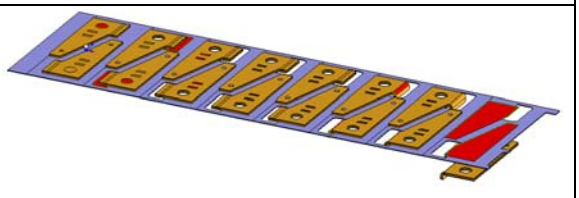


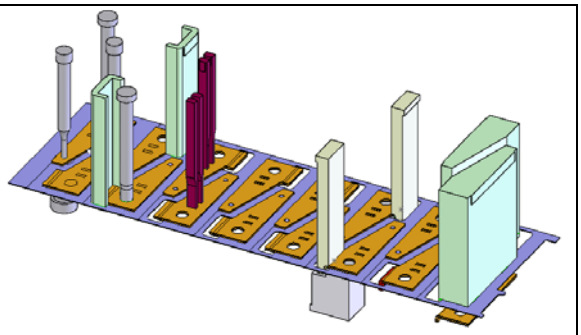
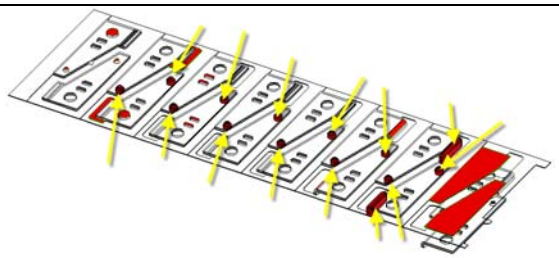
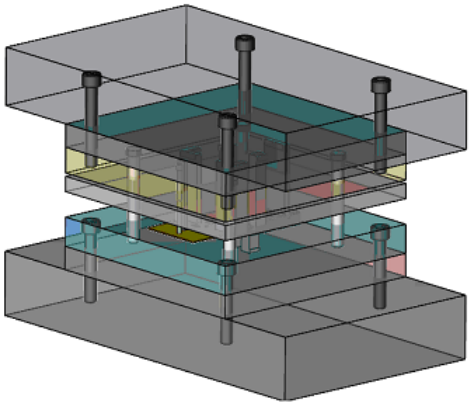
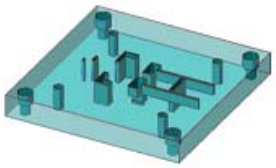
BASIC WORK FLOW

3DQuickPress allows progressive dies to be designed in 3D inside SolidWorks once you have the sheet metal part.

The normal process for progressive die design with 3DQuickPress is:

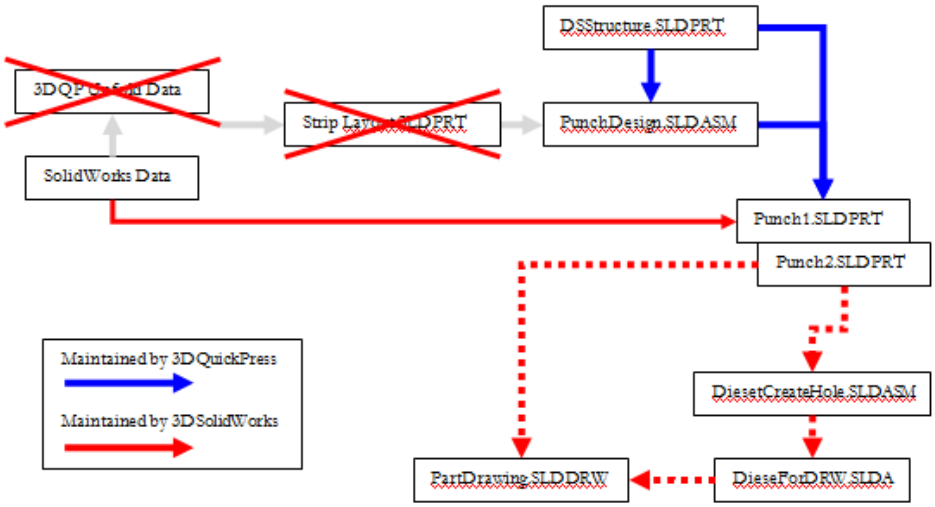
1. Unfold the metal part
2. Prepare station layout
3. Set up strip layout parameters
4. Process design
5. Punch design
6. Recess design
7. Die set design
8. Detailing

Workflow for 3DQuickPress	
1. Unfold Part	
2. Station setup: Unfolded part is inserted to strip layout document and all processes are activated according to the stamping process	
3. Set strip layout parameters	
4. Process design	

5. Punch design: Create 3D punch and die in punch design assembly	
6. Recess design	
7. die set design	
8. Detailing	 

ALTERNATE WORKFLOW

For some tooling, designers may want to utilize 3DQuickPress punch and die set tools while avoiding 3DQuickPress Unfold and Strip layout. 3DQuickPress allows users to start punch designs based on an active SolidWorks Part or Assembly document. This workflow is especially useful for stage tooling, transfer dies or ad-hoc re-work dies.

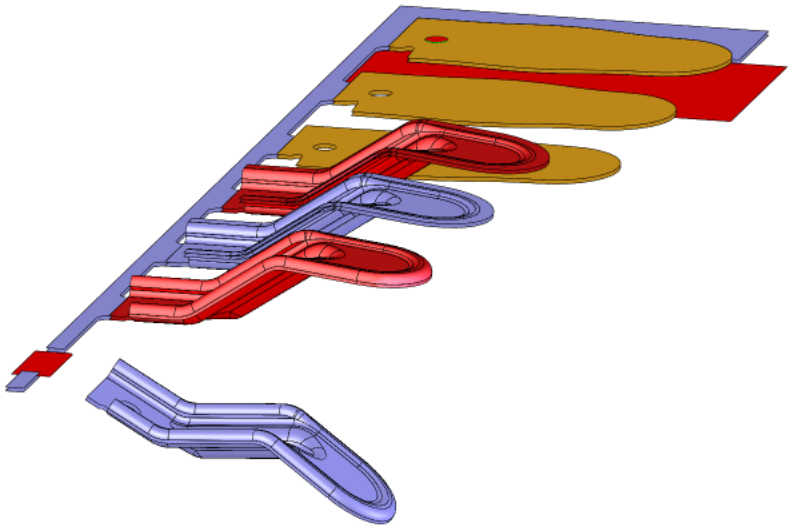


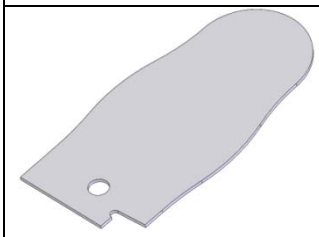
- Use SolidWorks to model the part at different stages of development from blank to final. One part per station is designed manually (it can be single station or complete strip)
- Strip Layout Design
- Punch Design
- Recess Design
- die set design
- Detail drawings of die components.

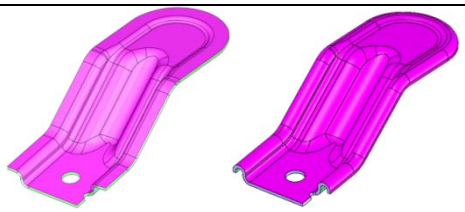
Swap part method

For the unfold and strip layout of complex formed parts, it is suggested that a simple blank be used instead of relying on the 3DQuickPress feature recognition unfolding method to extract sheet metal features and create the blank. A simple blank can be used as the input to strip layout design. Each station is modeled manually with SolidWorks techniques then inserted to the strip layout using the swap part function.

An example of an alternate workflow: Swap part method







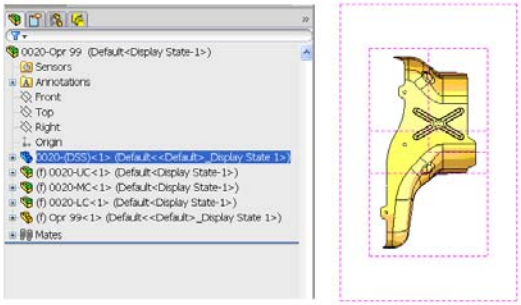
A very simple part representing the blank is used in unfolding. The blank is a result from an FEA program like 3DQuickForm Professional or the blank is designed manually with traditional blank calculation methods.

One or more intermediate steps are created in another SolidWorks part document and 3DQuickPress Swap Part is used to insert it into the strip layout.

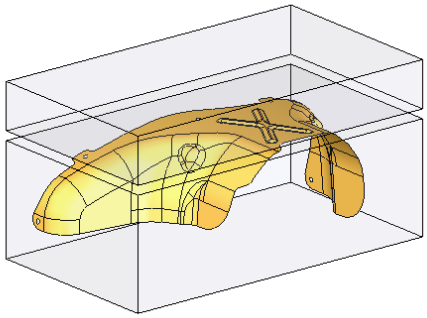
A stage tool is used to fix spring back problems for the part below

1. Punch Design Assembly

   Without the 3DQuickPress unfolding feature, user can also do punch and die design with the part alone.

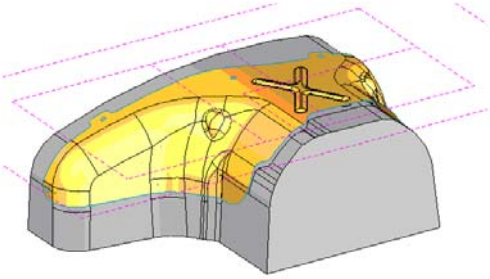


2. Using User Defined Component (UDC) to insert Production-Ready-Libraries (PRL) for the die block.



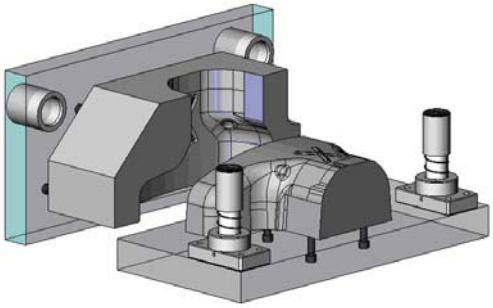
3. Model the form punch

   Use SolidWorks & 3DQuickPress to create the die face.



4. Die set design

   Use 3DQuickPress Assembly Utilities to finish the design.



Chapter 2
UNFOLDING PARTS

- Automatic Unfolding of Parts
- Semi-Automatic Unfolding of a Part
- Advanced Unfolding Methods
- Unfolding Alike Features
- Unfolding Non-Uniform Thickness Parts
- Advanced Fixed Face Options

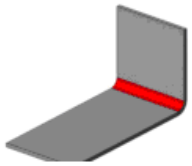
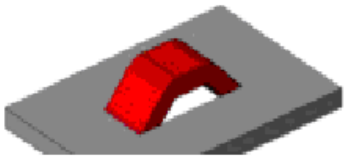
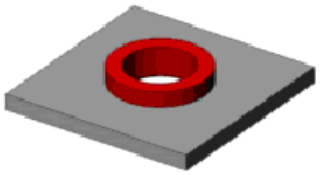
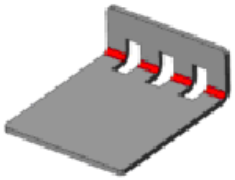
CHAPTER INTRODUCTION

One critical aspect of die design is the creation of an accurate flat of the part to be manufactured by the die. While SolidWorks offers several ways of flattening geometry, its tools are focused solely on uniform thickness sheet metal geometry. 3DQuickPress offers a wide variety of powerful tools to flatten both simple sheet metal parts and complex stamped geometry as well.

AUTOMATIC UNFOLDING OF PARTS

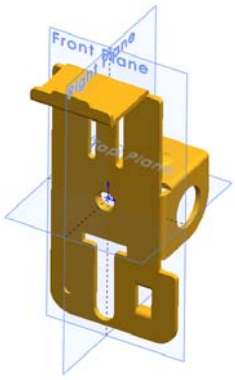
When appropriate, the easiest way to unfold a stamped part in 3DQuickPress is to utilize the automatic unfolding option. The geometry must be simple and/or fall into one of the categories listed below.

These sheet metal features can be unfolded automatically if the features are properly modeled.

Linear Bend	
Lancing	
Round Extrusion	
Compound Bend	
Embossing	

Part Position and Orientation

Frequently, imported parts arrive in the system oriented poorly. They can be not oriented relative to the global coordinate system placed very far from the global origin. In these situations, it is often a requirement to reposition the imported geometry to the part origin and/or reorient the geometry so that the press punch direction is parallel with the Z-axis of the global coordinate system. It is preferred that a part is positioned as close to the part file's global coordinate system (0, 0, 0).

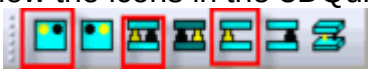


Bad Orientation Good Orientation

Case Study 2 – 1 : Automatic Unfolding of a Part

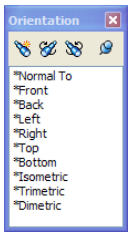
Check orientation and origin location of the part

1. Open part file BadOrientation. File location: `Desktop\Training\Unfolding\Unfold1\BadOrientation.x_t` by clicking **File**, **Open**, select ParaSolid (x_t) as File Type.
2. Run the **Import Diagnostics** to heal any bad faces. Do not proceed with feature recognition.
3. Switch between Front, Right, Top views. 

   Note: Notice how the icons in the 3DQuickPress Standard View toolbar  highlight as well displaying the views in 3DQuickPress that correspond with the standard SolidWorks views. This alternate toolbar provides a better feel for how a part is oriented to the die set.
4. From the FeatureManager design tree, hold **Ctrl** and select both the `Origin` of the part and `Imported 1`.
5. Right-click one of the selected items and select **Zoom To Selection**  . Notice how far away from the part is located to the origin.
6. Zoom in on the part to fill the screen.

7. Right-click on Face A and select **Insert Sketch**.

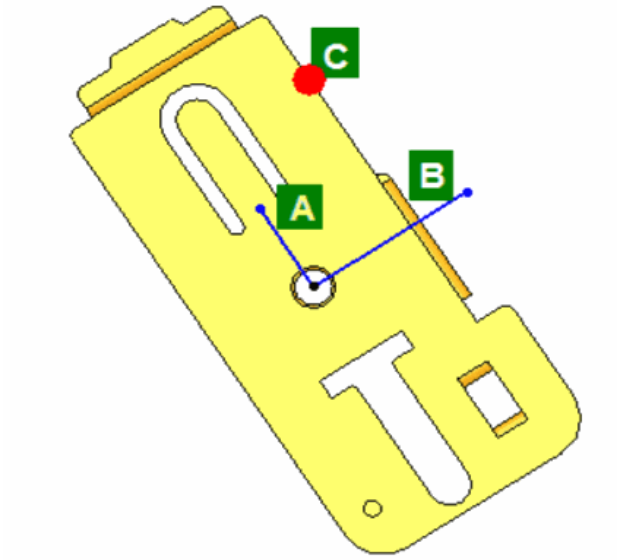
8. With the face still highlighted press the **Space Bar** to bring up the view orientations and then double click on **Normal To**.



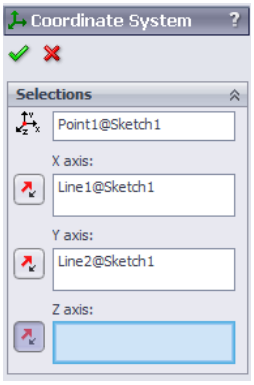
Create construction geometry to orient part

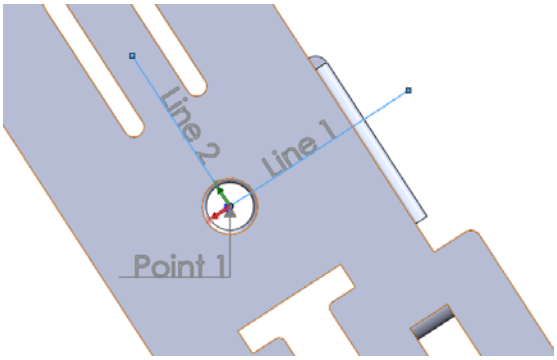
1. From the `sketch` toolbar, select the **line** command. Hold the cursor over the edge of the hole as shown to wake up the center point of the circle.

2. Draw a line from the center point to A and from the center point to B as shown in the picture below.



3. Hold **CTRL**, select Line A and Edge C, and then click **Parallel** from the feature tree.
4. Hold **CTRL**, select Line A and Line B, then click **Perpendicular** from the feature tree.
5. Click **Exit Sketch** from the confirmation corner.
6. Click **Insert**, **Reference Geometry**, **Coordinate System**, to add a coordinate system. The origin, X axis, and Y axis are shown on the picture below.

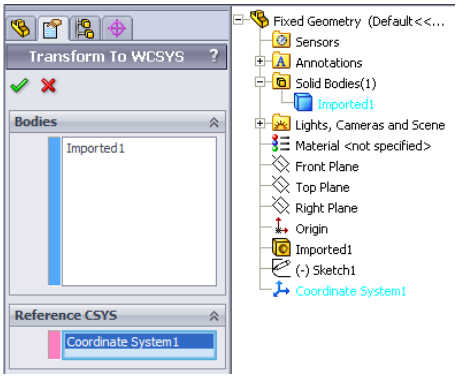


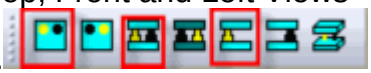


7. Click .
8. Save as Fixed Geometry1.sldprt.

Translate and Orient the Part to the World Coordinate System

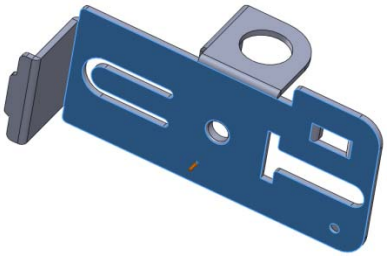
1. Right-click on `Sketch1` and select **Hide**.
2. Click **Transform to World CSYS** on the 3DQP Unfolding CommandManager tab on command manager.
3. In the **Transform To WCSYS** PropertyManager, select `Imported1` for Bodies and `Coordinate System1` for Reference CSYS. Click .



4. Select **F** on your keyboard (zoom to fit). Check the orientation with the tools on the 3DQuickPress Standard View toolbar to insure that the Top, Front and Left Views have been established properly. 
5. Save part as **GoodOrientation** in `Desktop\Training\Unfolding\unfold1` folder.

Unfolding the Part

1. Click the **Unfold Part** icon on the `3DQP Unfolding` CommandManager tab to unfold the part.
2. Select the Fixed Face as shown (Blue Face). Note that the Fixed Face is the same as the die face (the bottom of the strip material).



3. Click ✔ to accept the default values. Click **OK** on the `Bend Allowance Parameters` table.
4. **Save** the Part.

Modifying the diameter of the Round Extrude pre-piercing

Since 3DQuickPress automatically calculated the unfolding of the round extrude feature you may want to change the assumptions that were made. In this case, you can modify the pre-piercing diameter.

1. Right-click **Round Extrude 1** from the 3DQuickPress Feature Manager and select **Edit parameters**, change the Flatten Hole Diameter to 2mm. Press ✔ to complete the command.
2. Right-click **Sheet Metal Object** at the top of the 3DQuickPress Feature Manager and select **Unfold All**.

Options for Calculating Bends in 3DQuickPress

Direct Specification:

The user directly inputs the BA or K factor value that is desired.

System BA or System K factor:

3DQuickPress evaluates the R/T ratio (i.e. the ratio of bend radius / thickness) of a bend, and uses this ratio to query the BA or K factor value from the System Bend Parameters table. If the query is unsuccessful and no data exists in the table that exactly matches the R/T ratio, the following linear interpolation (or extrapolation) method will be used to calculate the BA or K factor.

$$BP = BP1 + (RT - RT1)\frac{(BP2 - BP1)}{(RT2 - RT1)}$$

Where:

BP is the bend parameter BA or K factor,

RT is the R/T ratio of a bend,

Example Case 1:

RT is greater than the maximum R/T ratio in the records of the table.

RT1 is the second maximum R/T ratio in the records of the table.

RT2 is the maximum R/T ratio in the records of the table.

BP1 is the bend parameter in the record corresponding to RT1.

BP2 is the bend parameter in the record corresponding to RT2.

Example Case 2:

RT is less than the maximum and greater than the minimum of the R/T ratio in the records of the table.

RT1 is the maximum R/T ratio value in the records of the table that is smaller than RT.

RT2 is the minimum R/T ratio value in the records of the table that is greater than RT.

BP1 is the bend parameter in the record corresponding to RT1.

BP2 is the bend parameter in the record corresponding to RT2.

Example Case 3:

RT is smaller than the minimum R/T ratio in the records of the table.

RT1 is the second minimum R/T ratio in the records of the table.

RT2 is the minimum R/T ratio in the records of the table.

BP1 is the bend parameter in the record corresponding to RT1.

BP2 is the bend parameter in the record corresponding to RT2.

Database BA or Database K factor

3DQuickPress queries the Specific Bend Parameters table for the BA or K factor value of a bend with the parameters Material Code, Bend Code, Thickness, Bend Radius and Bend Angle. If the query is unsuccessful, the System BA or System K factor will be used.

Macro BA or Macro K factor

3DQuickPress can call a SolidWorks macro for the BA or K factor value with the parameters of the Material Code, Thickness, Bend Radius and Bend Angle. If no such macro exists or the macro returns an invalid bend parameter value, the System BA or System K factor will be used. The Macro is user defined.

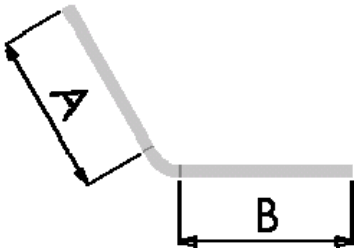
Bend Parameter Calculations

Bend Allowance Calculations

The following equation is used to determine the total flat length when bend allowance values are used:

$$L_t = A + B + BA$$

Where:



- **L_t** is the total flat length
- **A** and **B** are shown in the illustration
- **BA** is the bend allowance value

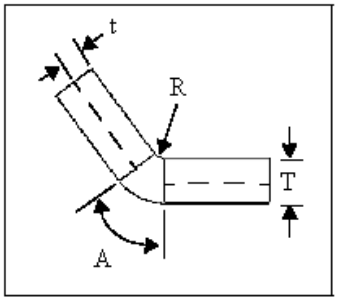
K-Factor

K-Factor is a ratio that represents the location of the neutral sheet with respect to the thickness of the sheet metal part.

Bend allowance with a K-Factor is calculated as follows:

$$BA = 2\pi\frac{A}{360}(R + KT)$$

Where:



BA = bend allowance

R = inside bend radius

K = K-Factor, which is t/ T

T = material thickness

t = distance from inside face to neutral sheet, or a percentage of thickness.

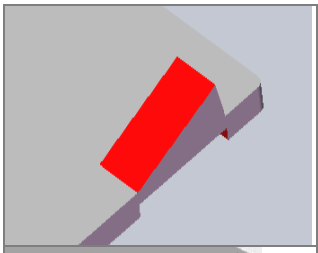
A = bend angle in degrees (the angle through which the material is bent)

SEMI-AUTOMATIC UNFOLDING OF A PART

Some part features will not unfold automatically for several reasons. One of these reasons is due to non-uniform material thickness. Users have to manually select the feature faces to help the system identify, group, and unfold these features.

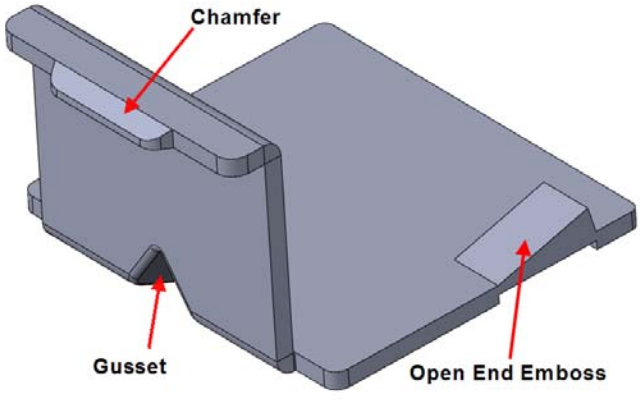
3DQuickPress uses a Semi-Automatic tool to aid in the unfolding of these features.

Semi-Automatic unfold features examples

	Open end embossments
	Gussets

Case Study 2 – 2 : Semi-Automatic Unfolding of a Part

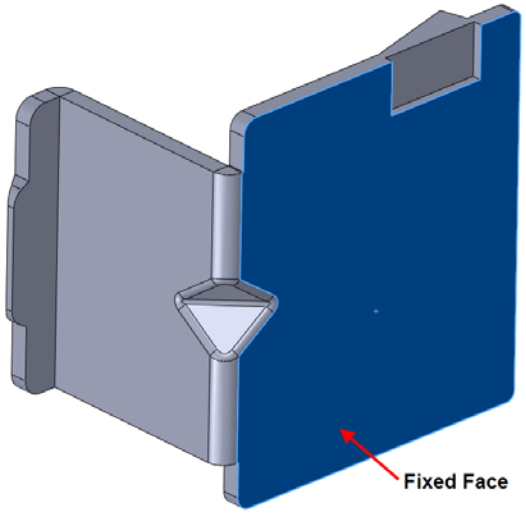
The part for this tutorial has five features to unfold. Two of these features cannot be unfolded automatically-the gusset and open ended embossment. Open ended embossments, gussets and some chamfers (coins) need to be unfolded with assistance by using semi-automatic recognition tools to handle these sheet metal features. The bends will be automatically recognized either immediately or once the other features are recognized.



Perform the Automatic Unfold

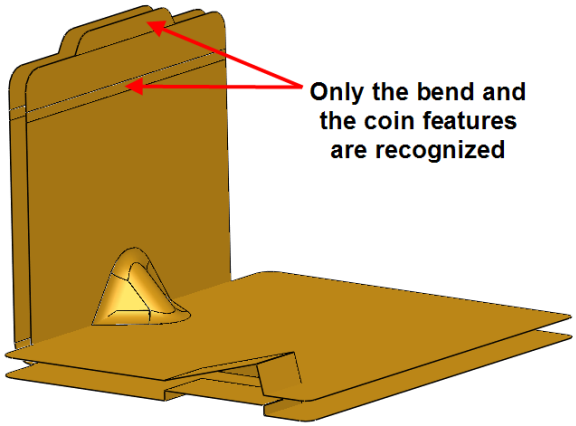
1. Open part file **Unfold2**. File Location: `Desktop\Training\Unfolding\Unfold2\Unfold2.sldprt`.

2. Select the Fixed Face as show.



3. Click **Unfold Part** from the `3DQP Unfolding` CommandManager tab. Use the default settings and press ✔ to unfold the part. Select **OK** to accept the default bend allowance settings.

4. Right-click **Sheet Metal Object** at the top of the 3DQuickPress Feature Manager and select **Unfold All**.



User Assisted Recognition (UAR) to recognize the features

1. Right-click **Sheet Metal Object** at the top of the 3DQuickPress Feature Manager and select **User Assisted Recognition**.

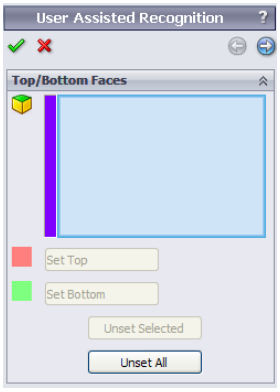
2. Press ✔ to rebuild the part.

Manually correct the top and bottom face group

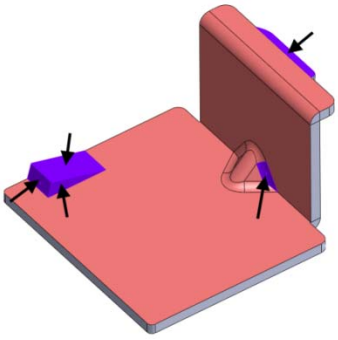
3DQuickPress only calculates the top and bottom faces of a part. It normally will automatically recognize these groups of faces. However, in certain cases, it will not be able to recognize the faces accurately. In these cases the user needs to check and modify the Top and Bottom groups of faces to assist the software.

Note: The material thickness (side faces of the part) is not used for several reasons. First, these faces are not necessary for the unfolding calculations. Second, it would slow down the performance of the software. Third, 3DQuickPress can unfold Solid and Surface type parts.

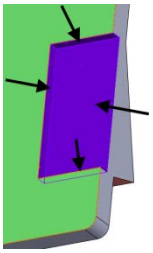
1. Click Back for `Top/Bottom Faces` modification.



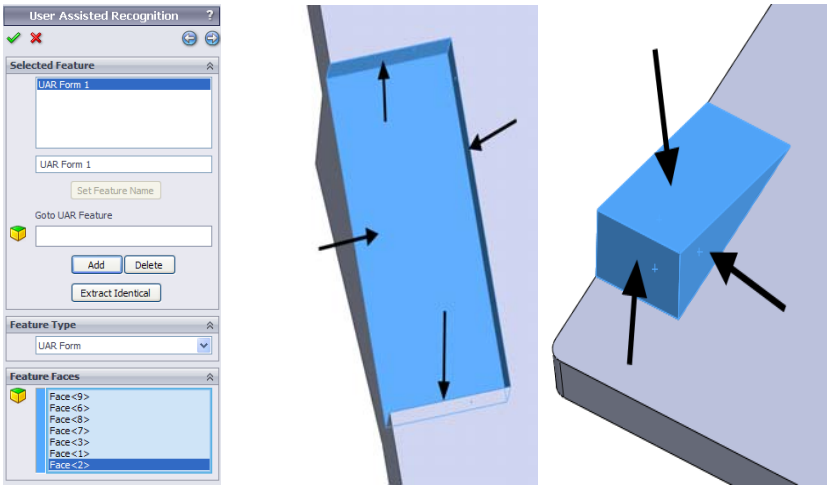
2. Select the faces as indicated then click **SET TOP** to assign the faces to the top material/faces group.



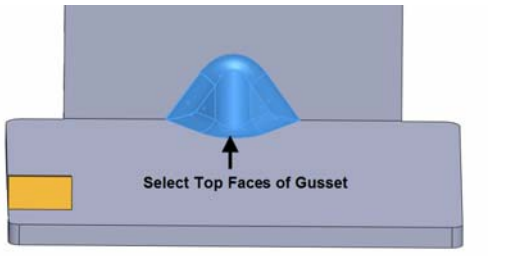
3. Select the faces as indicated then click **SET BOTTOM** to assign the faces to the bottom material/faces group.

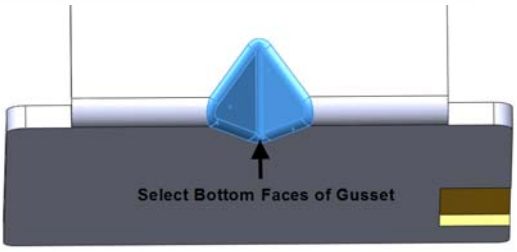


4. Click **Next** to add UAR (User Assisted Recognition) features.
5. Click the **Add** button to add a new UAR feature and select all faces of the open end emboss as shown below.



6. Click **Add** button again and select **Gusset** for Feature type. Select the faces shown below.





7. Click ✔ to accept all default values.
8. Right-click `Sheet Metal Object` at the top of the 3DQuickPress Feature Manager and select **Unfold All**. All features will now unfold.
9. **Save** & **Close** the part.

ADVANCED UNFOLDING METHODS

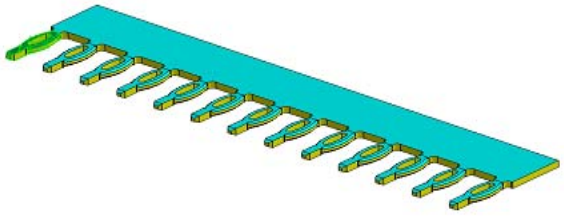
3DQuickPress has the ability to unfold extremely complex geometry beyond standard SolidWorks capabilities. This section explores some of the tools and methods required to tackle the advanced geometry that is found in the progressive die industry.

Case Study 2 – 3 : Unfolding Alike Features

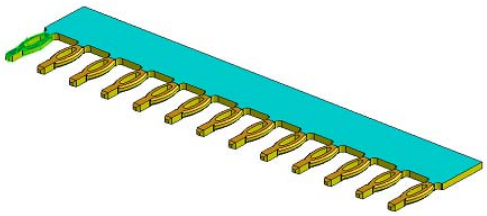
It is very common to have repeated instances of geometry in a part that require the exact same steps to unfold. In these cases, the user can save tremendous amounts of time by patterning the unfolded feature while maintaining order independence in the strip layout.

Find the identical instances

1. Open the part file **ExtractAlikeFeatures UDF**. File location: `Desktop\Training\Unfolding\Alike Features\ ExtractAlikeFeatures UDF.sldprt.` Select NO for SolidWorks feature recognition if prompted.
2. Right-click on `Sheet Metal Object` at the top of the 3DQuickPress Feature Manager and select **User Assisted Recognition**.
3. Select **User Defined 1** from the `Selected Feature` box and click the **Extract Identical** button.



A list of identical user defined features is created automatically.



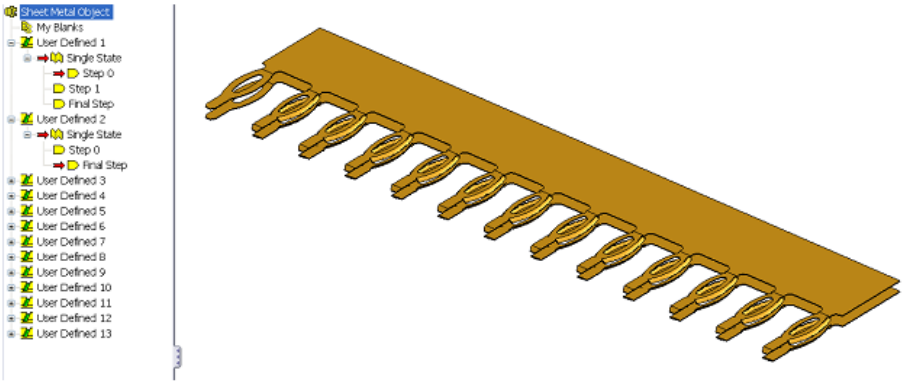
4. Click ✔ to finish the command and rebuild the unfolded part.

Note: The rebuild command inside 3DQuickPress is different from the rebuild tool inside of SolidWorks. This command can be located by right-clicking on `Sheet Metal Object` at the top of the 3DQuickPress Feature Manager and selecting rebuild unfolding model.

Patterning the unfold

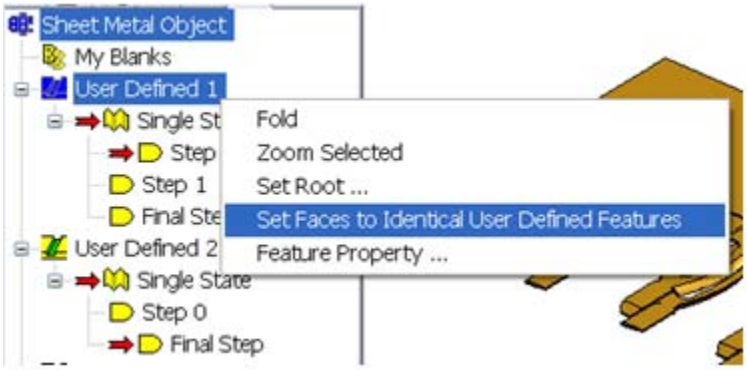
When the same User Defined Feature is used multiple times in an unfolded part the setup of steps and the set faces can be copied from one UDF to all of the identical instances of the UDF.

The steps to copy the settings of User Defined 1 to all other user defined features are as follows:



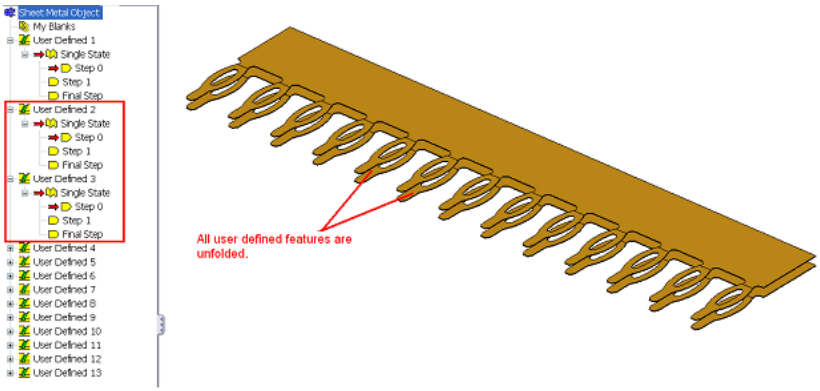
1. Select the completed user defined feature **User Defined 1**.

2. Activate Show Unfolding Model on the `3DQP Unfolding CommandManager tab`. In the feature tree, Right-click on `User Defined 1` and select **Set Faces to Identical User Defined Features**.



3. Right-click `Sheet Metal Object` at the top of the 3DQuickPress Feature Manager and select **Unfold All**. The

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



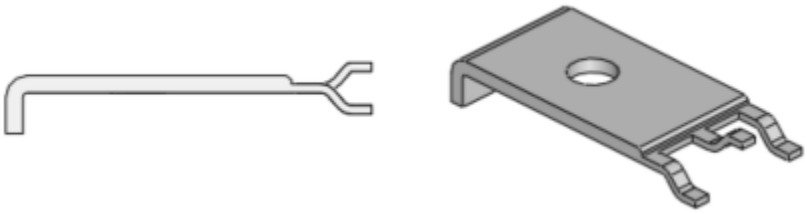
Case Study 2 – 4 : Handling Non-Uniform Thickness

Unfolding Parts with Non-Uniform Thickness

Non-Uniform thickness parts are very difficult for 3D CAD systems to unfold. Most CAD systems will only handle specific situations. 3DQuickPress has tools designed to accommodate these complex features.

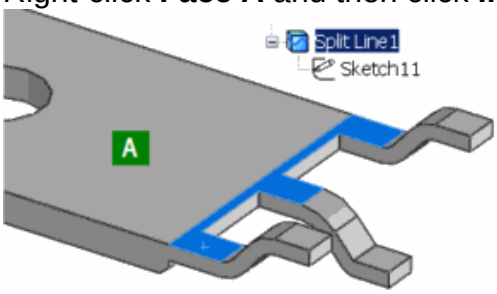
Defining the regions

1. Open the part file Multiple Thickness. File Location: `Desktop\Training\Unfolding\Multiple Thickness\Multiple Thickness.sldprt.`

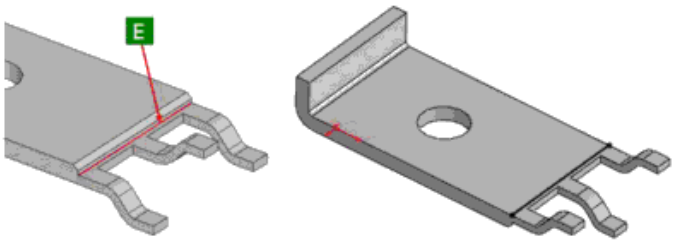


Before starting 3DQuickPress Unfolding, regions or features of different thicknesses must be separated by using the Split Face function of SolidWorks.

2. Right-click **Face A** and then click **Insert Sketch** .



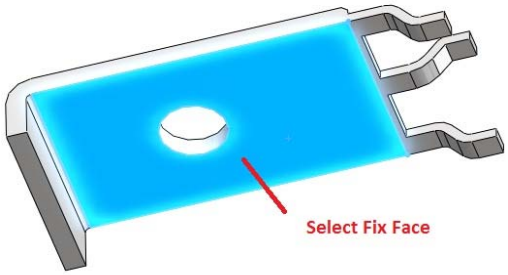
1. Select Edge E, click **Tools**, **Sketch Tools**, **Convert Entities** .



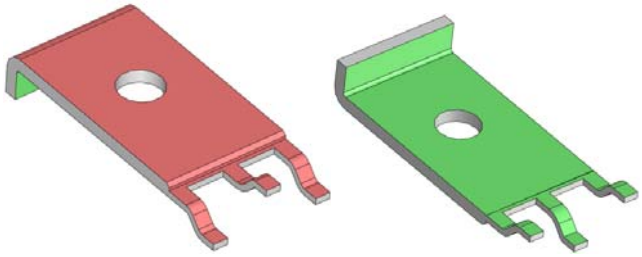
2. Click **Insert, Curve, Split Line**. For `Type of Split` select **Projection**. Highlight the first box under the `Selection` category and select the sketch splitting the face. In the second box select **Face A** as the face that is to be split.
3. Click ✔ to finish the command. The single face should now be 2 separate faces.

Unfold the Part

1. Click **Unfold Part** from the `3DQP Unfolding` CommandManager tab.
2. Select the face noted below for the fixed face. Click ✔ to finish the command.

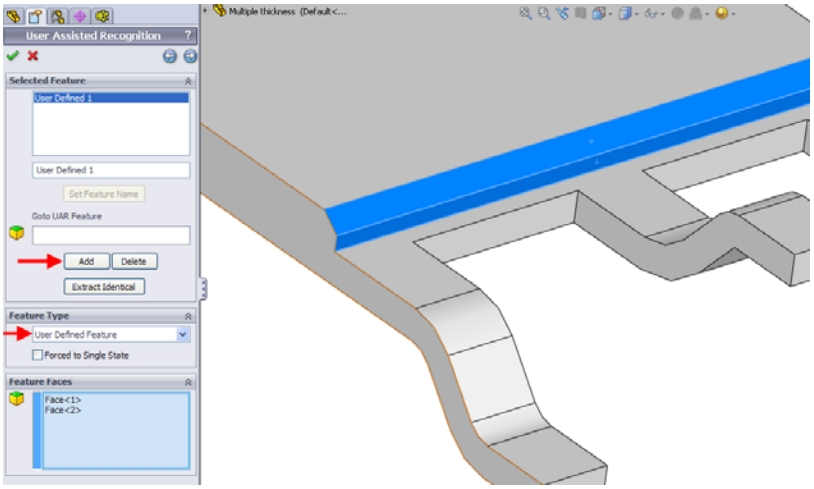


3. Right-click `Sheet Metal Object` at the top of the feature tree and click **User Assisted Recognition**.
4. Click **Back** to set Top (pink) and Bottom (green) Faces properly (see below for proper coloring of faces).

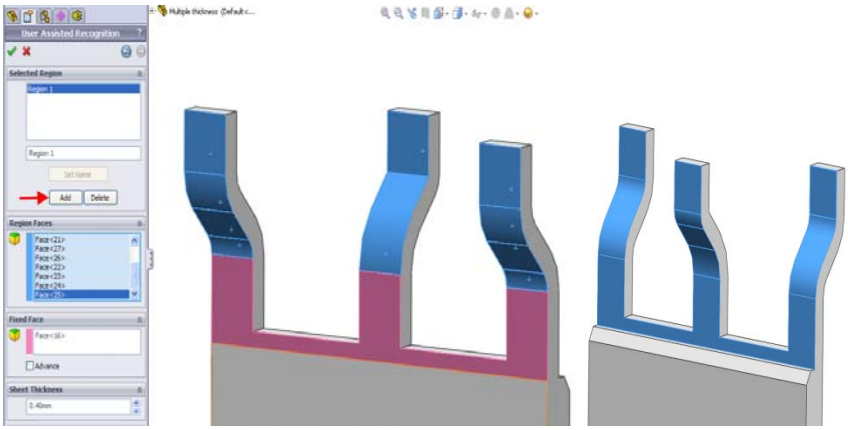


Tip: Right-click on a face and choose Select Tangency from the menu to select all tangent faces to the selected face.

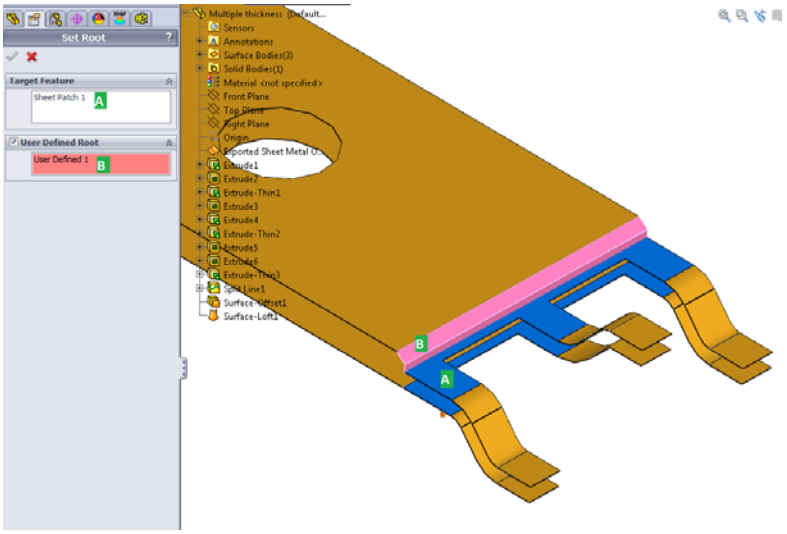
5. Click **Next** and **Add** a **User Define Feature** which will connect the thick and thin regions.



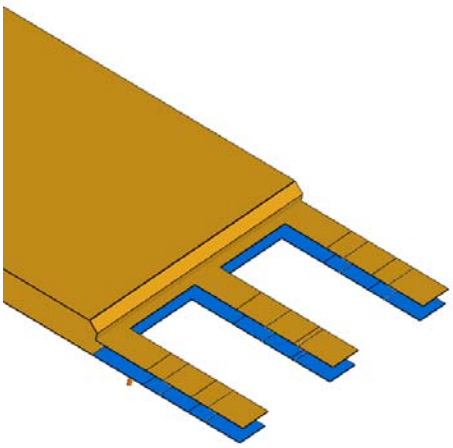
6. Click **Next** and **Add** a new region and select faces as shown. The pink face will be the Fixed Face for the region.



7. Click ✔ to exit UAR mode.
8. Right-click Face A, select **Set Root**, check **User Defined Root**, click Face B.

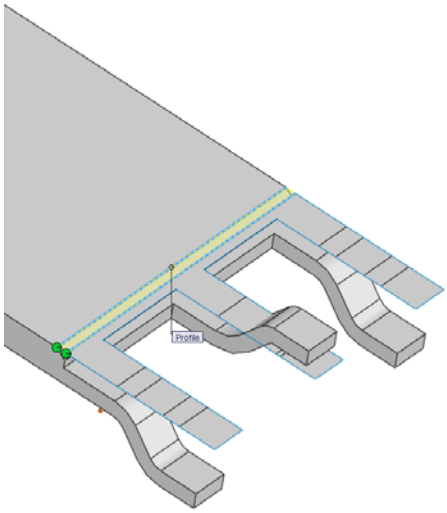


9. Click ✔ to exit the command.
10. Select all bottom faces of region 1 as shown.

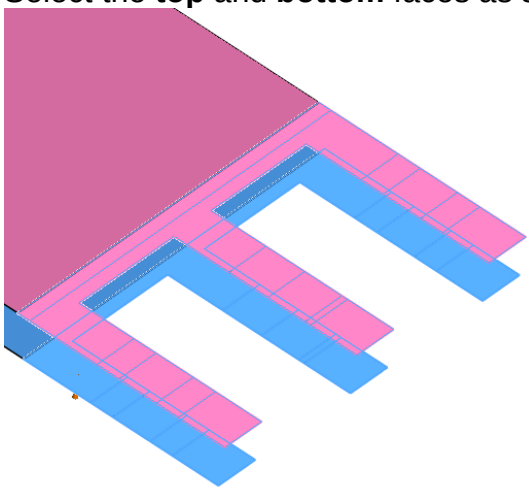


11. Right-click and set **Export selected faces**.
12. Select the exported faces and click **Offset surfaces**.
13. Enter 0.8mm and click ✔.
14. Select **Loft surface**.

15. Select the two highlighted edges and click ✔.



16. Expand User Define 1 feature and right-click on step 0.
17. Select **Set Face**.
18. Select the **top** and **bottom** faces as shown and click ✔.



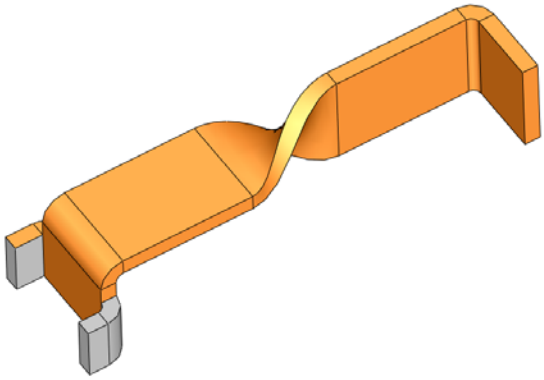
19. Right-click on step 0 again and select **Set Branch Feature Xfrom …**
20. Check the option Suppress Branch and click ✔.
21. **Save** & **Close** the part.

Case Study 2 – 5 : Using a Branch Feature

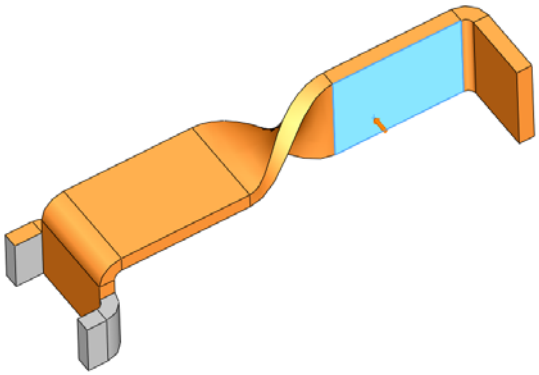
Shown below is a part with analytic (simple) geometry on both ends. User Defined Features will act as a “bridge” when unfolding parts with non-planar geometry that lie between other features that has been unfolded successfully with other methods.

This lesson is a simple example of how to control geometry utilizing a Branch feature with a Vector to Vector or Point to Point method. This lesson will demonstrate how the branch should behave when a User Defined Feature (normally forming feature) is used as the “bridge”.

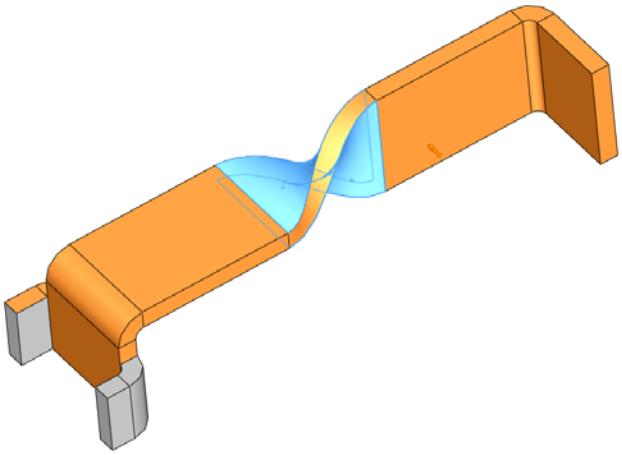
1. Open the part `UDF Branch`. File Location: `Desktop\Training\Unfolding\Unfold4\UDF\`



2. Select **Unfold Part** from the `3DQP Unfolding` CommandManager tab.
3. Select the highlighted face as shown below as the fixed face.



4. Leave the default setting and click ✔ to finish the command.
5. Click **OK** on the Bend Allowance Parameters window.
6. Right-click on `Sheet Metal Object` at the top of the 3DQuickPress Feature Tree and select **User Assisted Recognition.**
7. Under `Selected Feature` press the **Add** button. Set the `Feature Type` to **User Defined Feature**.
8. Select the two highlighted faces as the faces for the User Defined Feature.

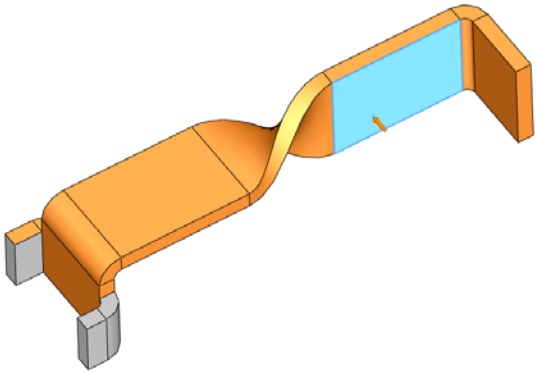


9. Press ✔ to finish the command.
10. Right-click on `Sheet Metal Object` and select **Unfold All**.

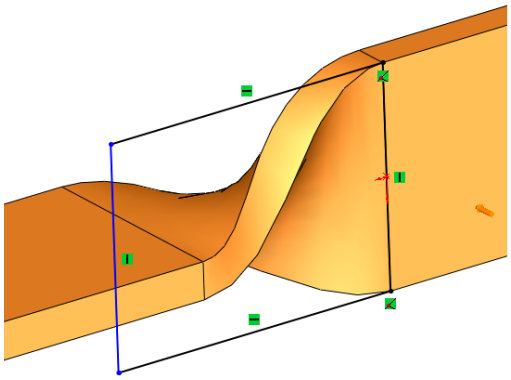
Create the blank faces for the twisted area utilizing standard SolidWorks sketching and surfacing techniques.

11. **Show** the original model by Toggling displays with Display options from the `3DQuickPress Unfolding` toolbar.

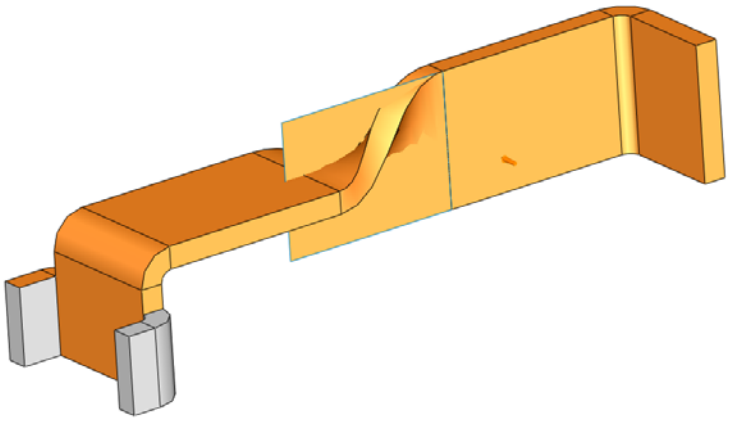
12. Right-click the face below and Insert Sketch **.**



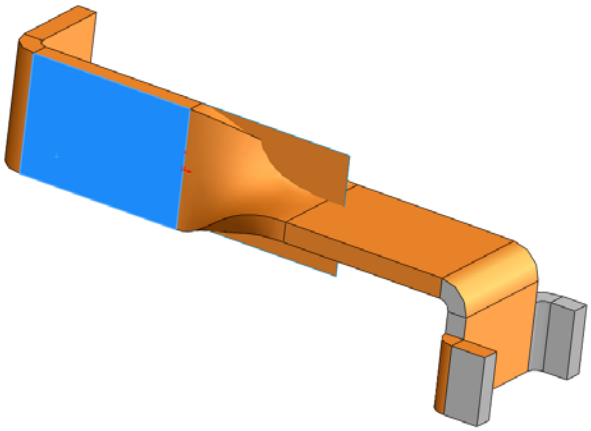
13. Sketch a rectangle of 8mm length. Two corners are coincident to the vertices of the sketch face . The rectangle should be approximately 8mm in length.



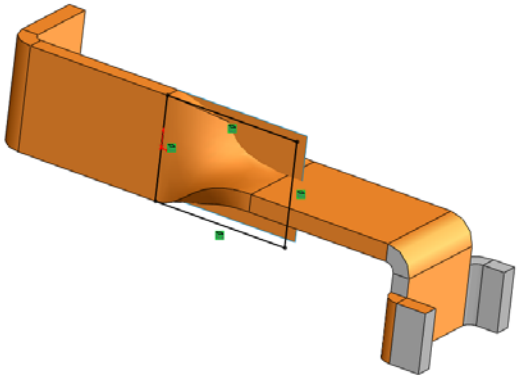
14. Select **Insert, Surface, Planar** . The sketch will be automatically recognized as the Bounding Entity, so press the ✔ to finish the command.



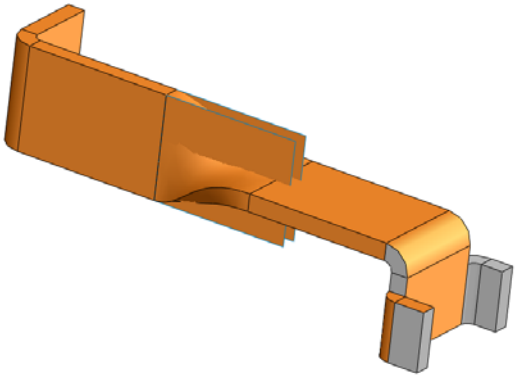
15. Rotate the part view 180° and insert a sketch on the face shown below.



16. From the SolidWorks feature tree, select the sketch of `Surface-Plane1` and **convert entities** from the `Sketch` toolbar. This will project the sketch lines to the new sketch plane. Each segment of the sketch has an On Edge relationship which will update when the original sketch is changed.

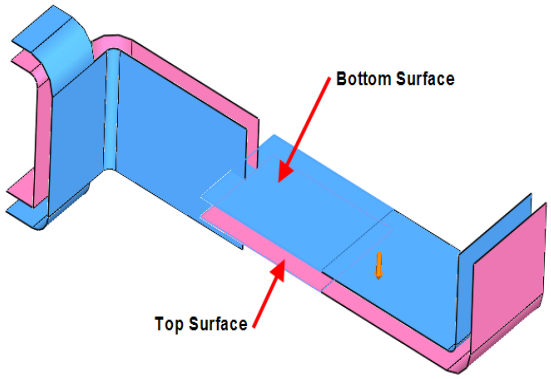


17. Select **Insert, Surface, Planar** . The sketch will be automatically recognized as the Bounding Entity, so press the ✔ to finish the command.

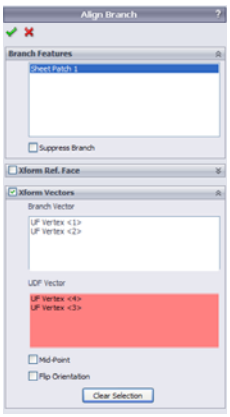


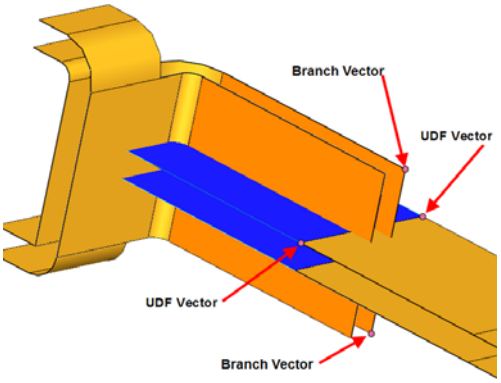
18. **Show** the unfold model by Toggling displays with Display options from the `3DQuickPress Unfolding` toolbar.

19. From the 3DQuickPress Feature Manager, expand `User Defined 1` and `Single State`. Right-click on `Step 0` and select **Set Faces**. The first box in the dialog window is for the Top Surfaces, and the bottom box is for the Bottom surfaces. Press ✔ to finish the command.



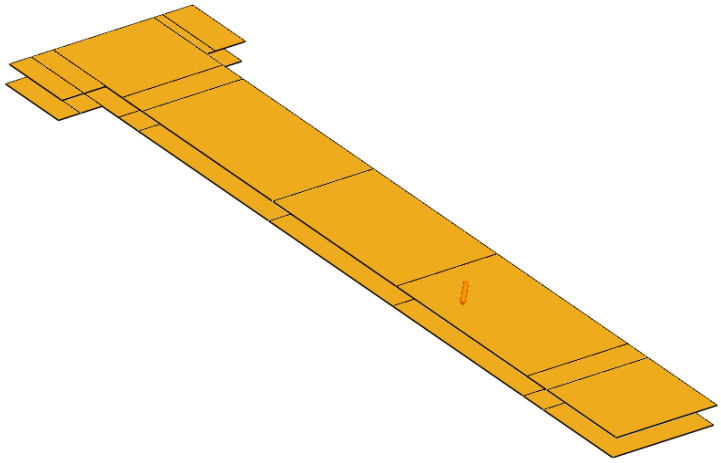
20. Right-click on `Step 0` of `User Defined 1` and select **Set Branch Feature Xform**. Select the vertices as indicated below.





21. Click ✔ to finish the command.

22. Right-click on `Sheet Metal Object` at the top of the feature tree and select **Unfold All** to see the final flattened part.



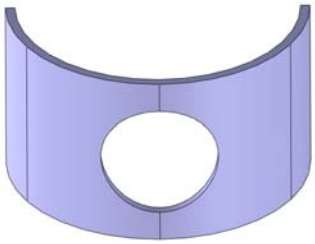
ADVANCED FIXED FACE OPTIONS

3DQuickpress unfolding offers three methods of selecting a fixed face (die face = bottom of strip material). The planar face option has already been explored in the previous exercises. The tangent edge and construction options will be described in this section. This approach is driven by how the part will be formed in the die and by setting the die plane tangent to the arcs on the part. In other words, the tangent of the bottom of the cylindrical part is collinear with the bottom of the stock material when unfolding occurs. The tangent face edge is the only geometry that stays fixed.

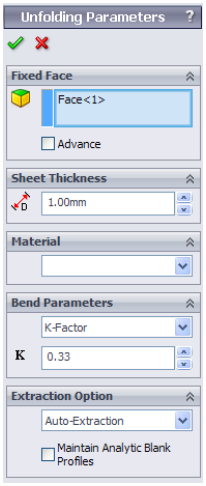
Case Study 2 – 6 : Unfold with Tangent Edge

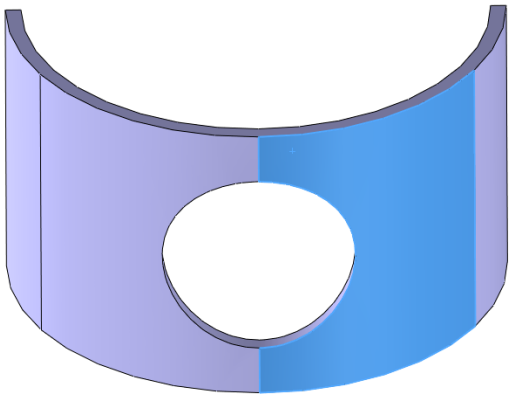
The below part will be unfolded onto a plane which is intersecting with a vertex on the cylindrical Fixed Face and the tangent arcs on the part. In other words, the tangent of the bottom of the cylindrical part is collinear with the bottom of the stock material when unfolding occurs. The tangent face edge is the only geometry that stays fixed.

1. Open part file `cylindrical fix face`. File Location `Desktop\Training\Unfolding\Unfold5\`

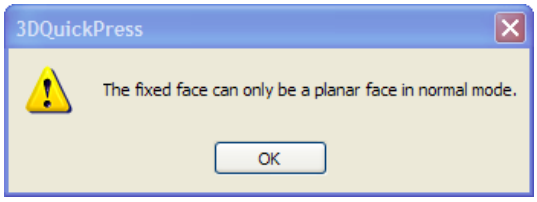


2. **Pre-select** the curved face as shown below, and then from the `3DQP Unfolding` CommandManager tab select **Unfold Part**. A message appears stating that the fixed face can only be a planar face in normal mode.



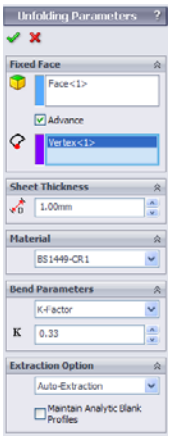


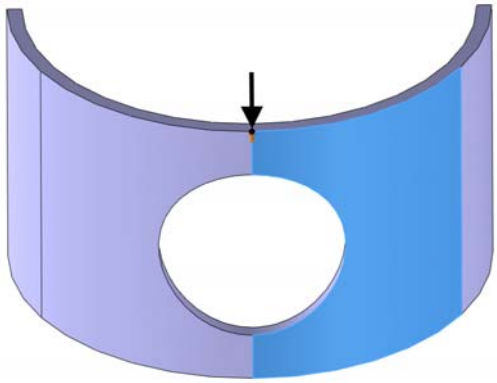
3. Click **OK** to close the warning message.



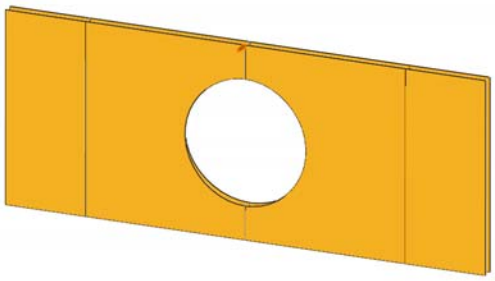
4. Select the **Advance** check box under the `Fixed Face` selection box. Select the same curved face. A new selection box appears in the PropertyManager.
5. Make this box the active window by clicking in the box and select the vertex on the cylindrical fixed face indicated by the arrow below. Press ✔ to finish the command.

Note: A red arrow will appear pointing normal to the bottom fixed face.





6. Select **OK** for Bend Allowance Parameters to accept the default settings.
7. Right-click on `Sheet Metal Object` at the top of the feature tree and select **Unfold All**.



8. Select **Show Unfolding Model** from the 3DQP `Unfolding` CommandManager tab.
9. **Save** & **Close** the part.

Case Study 2 – 7 : My BLANK function

Sometimes designers want to simplify the unfolded results in the blank state (2D editing). My Blank can also address “design for manufacturing” requirements such as simplification of a conic profile to an arc profile in the blank profile without changing the 3D part model. This may be required to reduce machining time and cost. Therefore, My Blank allows the designer to make changes in the blank 2D sketch easily when the 3D part modification would be very difficult.

When creating the Strip Layout with this part the My Blank sketch will be used for Piercing and Notching creation by suppressing the original unfolded part data. The original 3D Model is not changed. If the change can be made to the original 3D part, the change will update the unfolded blank and MY BLANK is not required.

Create a MY BLANK feature

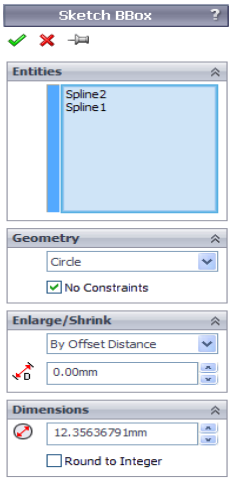
1. Right-click on `My Blanks` in the feature manager and select **Add My Blank**.

A My Blank Sketch is now available to modify the unfolding results.

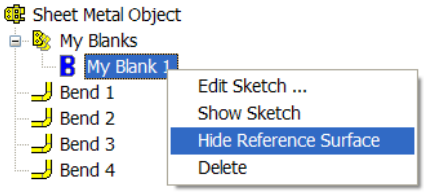
2. Right-click **My Blank 1** in the feature tree under `My Blanks`. And select **Edit Sketch**.

Note: The hole in this part is actually made up of 2 splines *not arcs* since the hole is circular before the unfold was calculated.

3. Orient the part normal to the sketch by clicking the **Normal To** button.
4. Hold the cursor over the edge of the hole and the spline symbol will appear next to the cursor.
5. Click **Bounding Box** from the `3DQP QTools` CommandManager tab, `QPSketch Tools` dropdown menu. Select the two splines that make up the holes.



6. Under `Geometry` select **circle** and under `Enlarge/Shrink` select **By Offset Distance** and set the distance to **0.00 mm**.
7. Click ✔ to finish the command.
8. Hold **CTRL** and select the 2 splines that make up the original hole. In the feature tree, check the box **For Construction**.
9. **Exit Sketch** in the confirmation corner.
10. Right-click on `My Blank 1` and select **Hide Reference Surface**. If the reference plane is still shown, Right-click the plane in the graphic area and select hide .



11. Select **Toggle Unfolding Display** from the `3DQP Unfolding` CommandManager tab to hide the unfolded model and show the original solid.
12. **Save** & **Close** the part.

Chapter 3
STRIP LAYOUT DESIGN

- Strip Layout Introduction
- Feature Based Method
- Swap Part Method
- Station Layout
- Strip Layout Parameters
- Rearranging Strip Operations
- Adding and Modifying Strip Operations

3DQUICKPRESS STRIP LAYOUT INTRODUCTION

Creating a strip layout can be done with two methods, Feature Base and Swap Part methods. Both of these methods require an unfolded part or multiple unfolded parts to start the strip layout. The strip layout file is a new part file that will contain the unfolded part(s) data — this is not an Assembly file type. The strip layout part is primarily used to represent the metal forming operations. The result is an accurate representation of the operations of each station that will occur during running the die. Both 2D and 3D features are created to represent the metal forming operations. Changes to these operations are quick, easy, and logical to the die designer.

Overview of Steps:

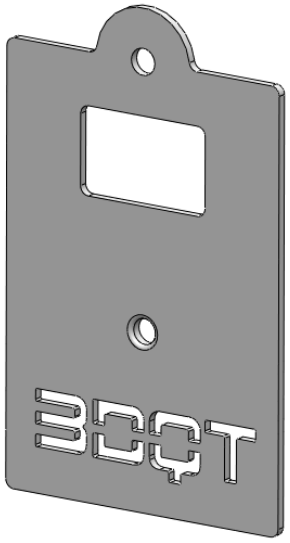
- Station Layout
- Strip Parameters
  - Progression
  - Stock width
  - Optimization material utilization
- Manipulate the strip layout
  - Assign operations
  - Modifying existing operations
- Adding Operations

Case Study 3 – 1 : Feature Based Method

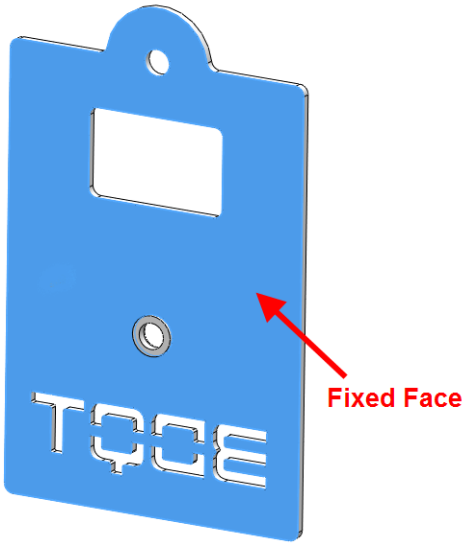
In this case study a simple part is unfolded and a 2-up strip layout is created. After the initial layout is completed, several types of cutting punches will be created using 2D features to represent these operations. These operations will then be manipulated in several ways including operation order, activation state, and mirroring operations. The result is a representation of the metal forming process in a single SolidWorks part file. Speed of creation and manipulation are the key elements to using the 3DQuickPress approach.

Initial Station Layout

1. Open the file named `Tutorial-Align` File Location: `Desktop\Training\Strip Creation\`

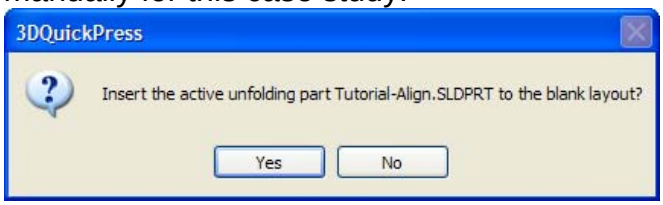


2. Click **Unfold Part** on the `3DQP Unfolding` CommandManager tab. Select the back face as the fixed face.

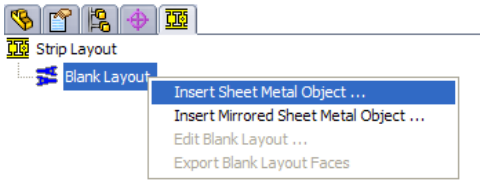


3. Click **Strip Layout Design** from the 3DQP Strip Layout CommandManager tab to create a new strip layout, input **T5-1 Strip Layout** as file name and click **Save**.

Note: Click **No** to prevent the existing part from be inserted automatically to the new strip layout. It will be inserted manually for this case study.

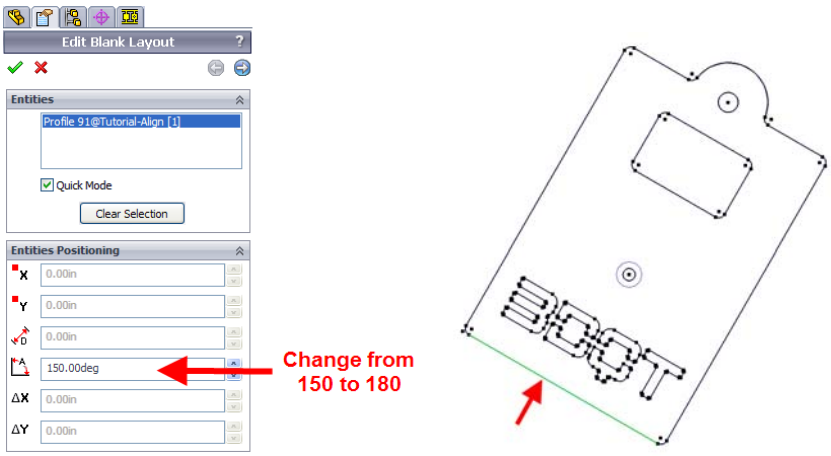


4. Add an unfolded part to the strip. Right-click **Blank Layout** from the Strip Layout Design Tree and select **Insert Sheet Metal Object**.



5. From the Open Dialog Box, select `Titorial-Align` File Location: `Desktop\Training\Strip Creation\` Now the part appears in the strip layout but needs to be properly oriented.

6. The Edit Blank Layout mode is active upon inserting the part. To orientate differently than the default location: Select the line indicated below and change the line angle from 150 to **180**.



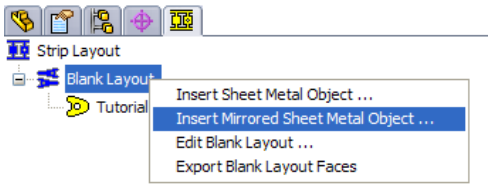
7. Click ✔ to finish the command.

Adding a 2nd Part to the Strip Layout

In many cases, a right-hand or left-hand version of a part is required. When this occurs, a mirrored version of the part can be inserted rather than creating a second unfolded part.

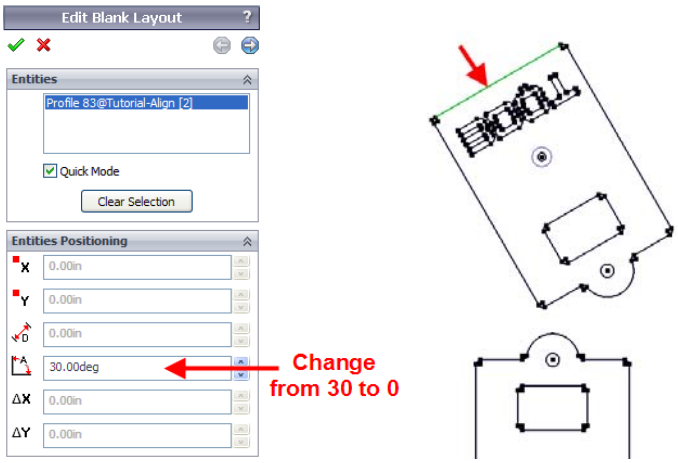
3DQuickPress has an option to do this for you automatically from the original part.

1. To add a mirrored version of the original part, right-click **Blank Layout** from the Strip Layout Design Tree and select **Insert Mirrored Sheet Metal Object**.



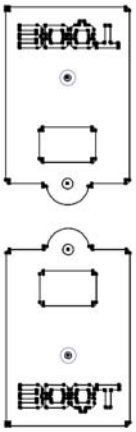
2. From the File Open Dialog Box, select `Training-Align`
   File location: `Desktop\Training\Strip Creation\`

3. Now align the 2nd blank. Select the line indicated below and change the line angle from 30 to **0**.



4. Drag the mirrored blank to a position directly above the first part.

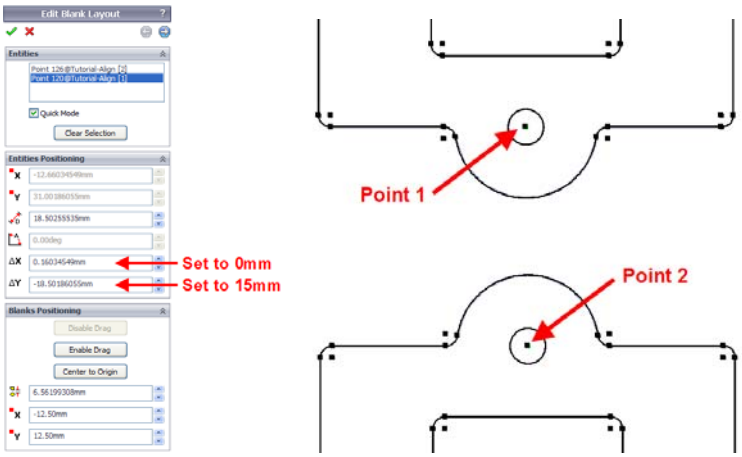
Note: By selecting the profile, or the upper blank, you can drag it to the approximate location. The next steps will allow you to position it precisely.



Additional Alignment

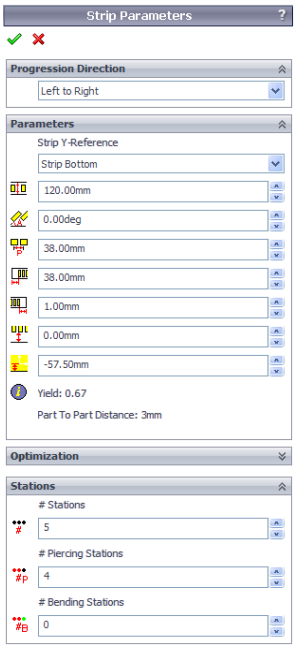
Additional alignment is required to position the blanks relative to each other and to the origin of the strip layout. 3DQuickPress can assist in the alignment of the parts by selecting reference points from each individual blank profile and manipulating the X, Y distances between these points.

1. Zoom in close to the area shown below. Under the Entities section in the PropertyManager, click **Clear selection**. Select **Point 1** and **Point 2**, change `delta X` to **0**, `delta Y` to **15mm**.



2. Next, make sure the blank layout is positioned properly relative to the origin of the Strip (0, 0, 0). Under `Blank Positioning`, click **Center to Origin**.

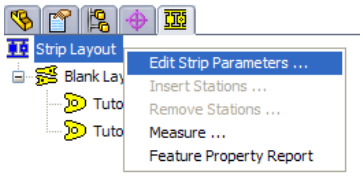
3. Click ✔ to finish the command.



Editing the Strip Layout Parameters

Now that the initial station layout is complete, the Strip Parameters can now be edited. This includes many options such as Progression Direction, Progression Distance, and Number of Stations to name a few. These options may be changed at anytime and the strip will update appropriately.

1. Right-click `Strip Layout` from the tree and select **Edit Strip Parameters**.



2. To quickly set material utilization (Yield), the Optimization tools are used. Under `Optimization`, input **0mm** for `Width Margin` and `Station Clearance`, click **Optimize** button.

This will provide a preview of the results while allowing you to continue to edit the parameters. The Yield is listed above the Optimization Box. For instance: (.67) is reporting that 67% of material is currently utilized or 33 % is wasted material. The Yield amount updates immediately as the Parameters of the Strip are changed.

3. Update the parameters of the strip. Set the following values for the `Parameters` and the `Stations` properties.

Parameters

Progression Direction	Left to Right
Strip Y Reference	Strip Middle
Width of strip	120 mm
Station Rotation Orientation	0 deg
Inter-Station Pitch	38 mm
Start Margin (Extra material representation before station 1)	38 mm
End Margin(Extra material representation after last station)	1 mm
Strip Y Position	-60 mm

Stations

Number of Stations	5
Number of Piercing Stations	4
Number of Bending Stations	0

4. Click ✔ to finish the command.
5. **Save** the part.

The first time the number of stations is set for Piecing and Bending operations and the Finish button is hit; existing operations in the unfolded part will be automatically distributed and activated throughout the stations listed. This is done for convenience. All other processes, such as Form Features, are NOT distributed.

This only happens the first time Edit Strip Parameters is edited. Going forward, all operations can be activated and inactivated using standard right-click selections in the Feature Manager or by right-clicking on the strip operation.

For instance, since “Number of Piercing Stations” is set to 4; all piercing operations that exist in the unfolded part will be automatically activated and distributed into the first 4 stations. Operations can then be rearranged as required. Since “Number of Bend Stations” is set to 0, all bends, if there were any, would stay inactive or unprocessed until activated in a station of choice.

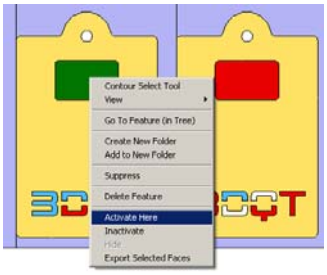
Rearranging Strip operations

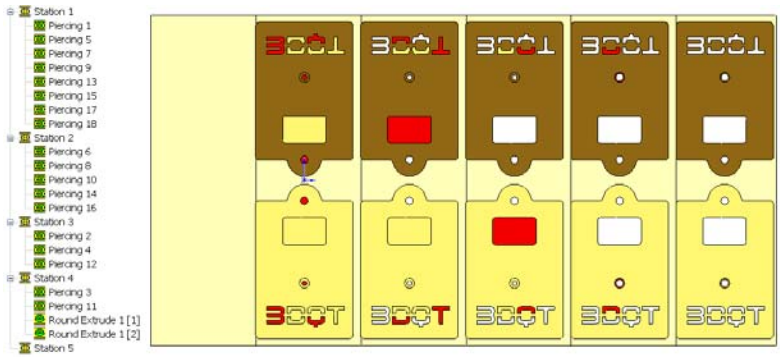
After the initial strip is created, the piercing operations can be reordered.

1. Click **Filter Features** from `3DQP Strip Layout` CommandManager tab.

Note: With this option enabled, the user can only select the 3DQuickPress features. The Filter Features option is employed to make it easier to select 3DQuickPress features while avoiding inadvertent selection of SolidWorks geometry. This optional command may be helpful when selecting features while the model is zoomed out and the feature you are trying to select is difficult to pick from that zoom scale.

2. Right-click a 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.

4. Click **Filter Features** from `3DQP Strip Layout` CommandManager tab to deactivate the filtering option.

Adding Strip operations

Several types of operations can be created to manipulate the strip layout. Notching punches are created to remove material from the outside of the blank profiles. Internal Notching features are created to remove material inside a part profile that was not recognized as a piercing or needs to be done in a separate process that are not belongs to the part profile. For instance, a piercing punch will be required to be split into two internal notching punches to process in 2 different stations.

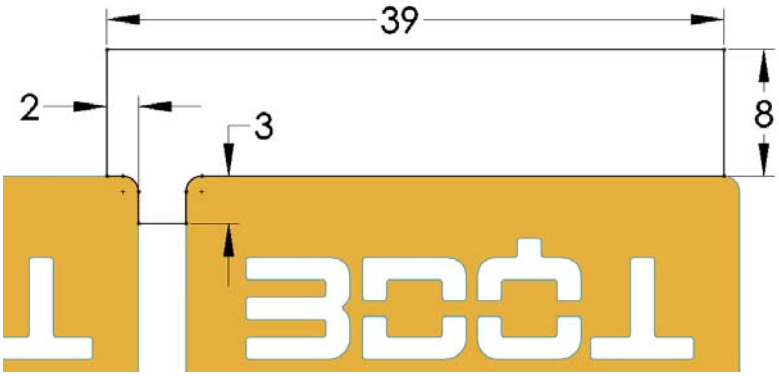
Create a Notching punch

1. Right-click `Unprocessed Features` from the Strip Layout Design Tree and select **Create Notching Feature**.

   Note: All punches are created as 2D representations for simplicity and manipulation speed. The basis for these features is 2D sketches. All the standard sketching tools plus many additional 3DQuickPress sketching tools are available. A reference station is shown with the blanks for design reference (Station 0, which only exists while in punch creation or editing mode).

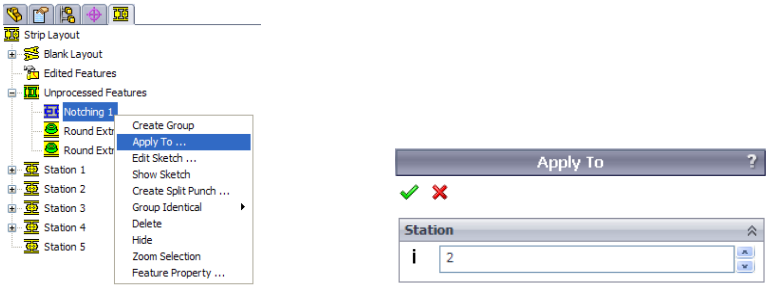
2. Click **Rectangle** and sketch a rectangle as shown below on the left. Click on **QuickSketch** on `3DQP Strip Layout` toolbar. **QuickSketch** will automatically adjust the rectangle sketch to match the contour of the part.

3. Detail the sketch.

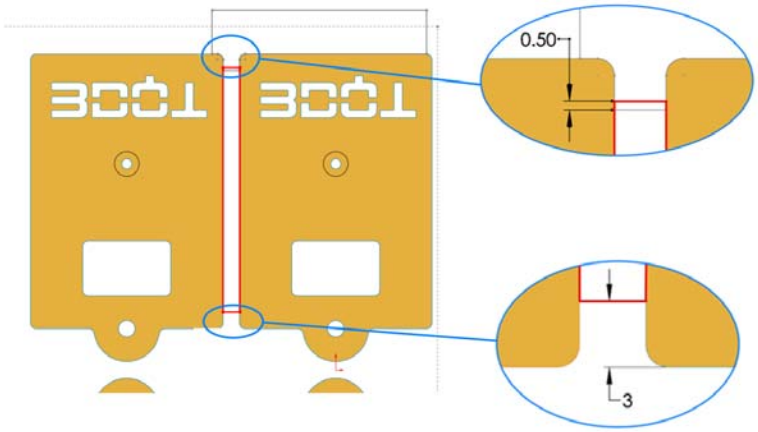


4. Exit the sketch to create the punch

5. Now activate the punch in Station 2. Right-click `Notching 1` in the feature tree and select **Apply To** Station **2**.

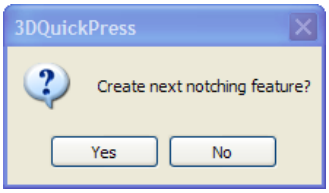


6. Create another Notching punch that is a rectangular shaped. Right-click **Unprocessed feature** from the **Strip Layout Design Tree** and select **Create Notching Feature**.
7. In order to reference (add relationships and dimension to the other punch sketches) other punches to create new punches you can turn them on and off when necessary. Right-click in the empty space of the Graphic Window and select **Show All Notching Sketches**.
8. Click **Rectangle** and create a rectangle as shown.

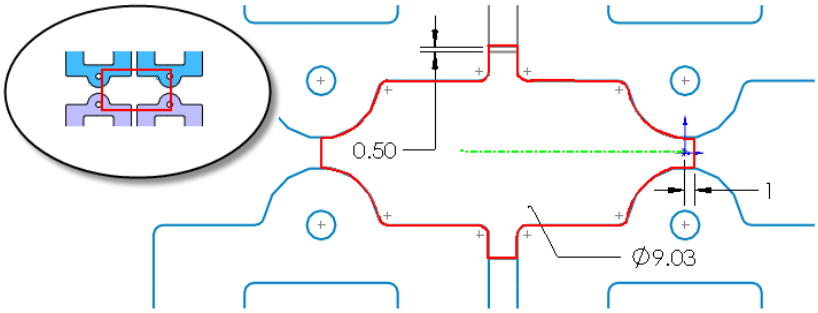


When creating several notching punches many steps can be saved by proceeding directly to the next notch creation and then activating the punches later in the process. This function is called Create Notch Batch Mode.

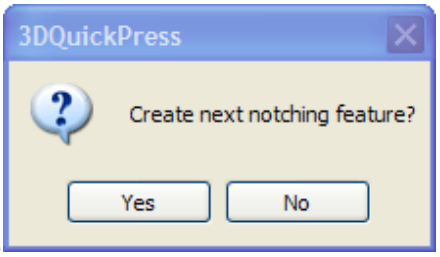
9. Right-click in the empty space of the Graphic Window and select **Create Notch Batch Mode**.
10. **Exit Sketch** and click **Yes** to continue the design of next notching feature



11. Create another punch for cutting off the part. Click **Rectangle** and create a rectangle as shown in the detail below. Click **QuickSketch** on `3DQuickPress Sketch Tools` toolbar and detail it as shown.



12. **Exit sketch**. No additional notching punches are needed at this time so click **No** to exit the notching punch design.

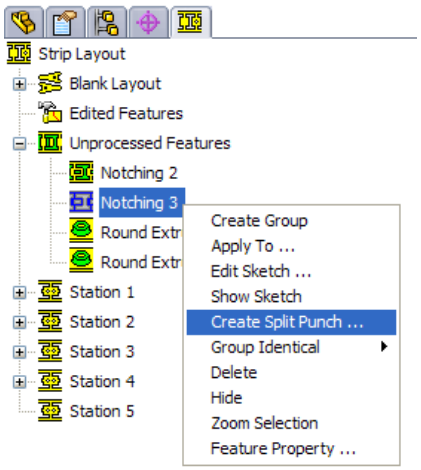


Splitting an operation into multiple operations

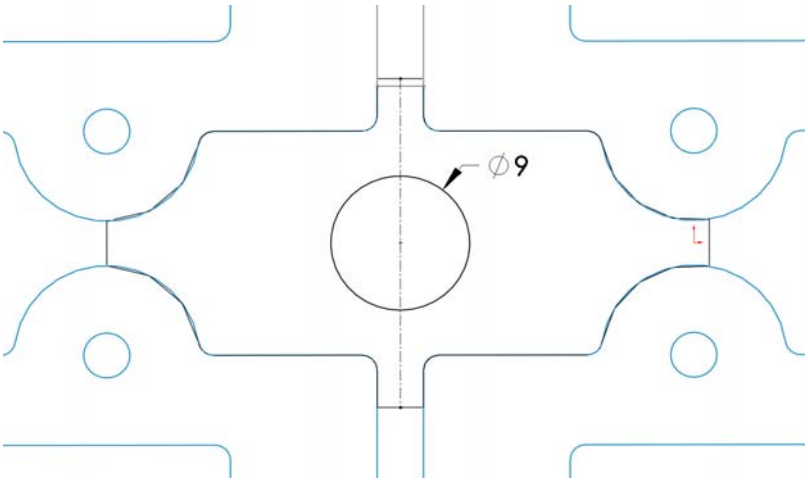
There are many cases when an existing punch needs to be modified or divided into many operations. For this we can use the Split Punch approach in lieu of creating individual punches from scratch.

Create a Split Punch

1. Right-click `Notching 3` in the feature tree and select **Create Split Punch**.



2. Click **Circle** and sketch a **9mm** diameter circle in the center of `Notching 3`.



Tip: It may be helpful to show the sketch of `Notching 3` to draw a centerline from top to bottom. The midpoint of the centerline will be the center of the Splitting Punch.

3. **Exit Sketch** and click on the Split Punch PropertyManager.
4. **Save** the part.

Mirroring an operation

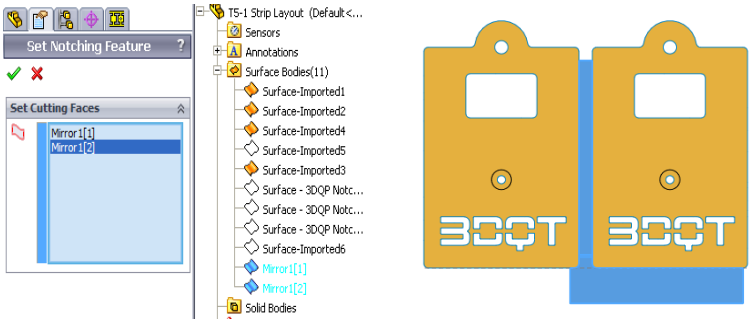
Mirroring a 3DQuickPress punch requires the SolidWorks surface geometry of the punch to be mirrored first. Inside 3DQuickPress, the surface geometry is defined as a punch, allowing the manipulation of station location and active state independent of the original geometry.

For every 3DQuickPress feature (operation) there is a SolidWorks associated equivalent surface (also known as Faces). However, since 3DQuickPress manages this operation, displaying these surfaces is not often necessary. Occasionally it is appropriate to access these associated hidden surfaces.

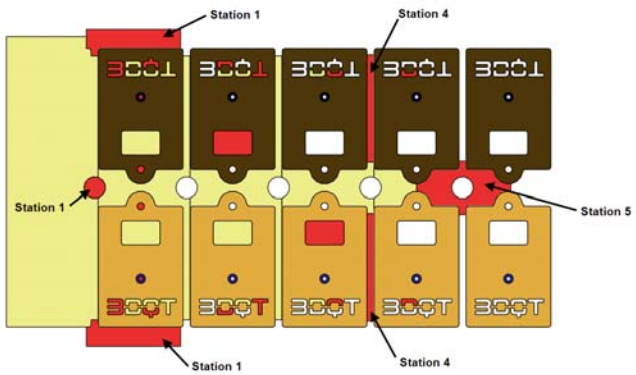
This exercise focuses on using native SolidWorks functions (I.e. Pattern/Mirror) to edit or manipulate 3DQuickPress created features. The ability to use SolidWorks functions to edit and manipulate 3DQuickPress features leverages the power of SolidWorks and avoids a need to duplicate existing SolidWorks commands in the 3DQuickPress environment.

1. Show all 3DQuickPress features in the SolidWorks Feature Manager. Right-click `T5-1 Strip Layout` at the top of the SolidWorks feature tree and select **Show All 3DQP features**.

2. Mirroring the SolidWorks Surface: Hold Ctrl and select Top `Plane, Surface – 3DQP Notching 1, & 3DQP Notching 2`. Click **Insert, Pattern/Mirror, Mirror** and click ✔.
3. To display the 3DQuickPress punch surfaces: click **Notching Punch Design Mode** on the `3DQP Strip Layout` CommandManager tab to show the notching design faces.
4. Set the SolidWorks Surface to become a 3DQuickPress Punch. Right-click `Unprocessed Features` from the 3DQuickPress feature tree and select **Set Notching Feature**.
5. Select the mirrored faces from the Surface Bodies folder of the flyout FeatureManager design tree and click ✔.



6. To hide the 3DQuickPress punch surfaces: click **Notching Punch Design Mode** on the `3DQP Strip Layout` CommandManager tab to toggle back to the display of strip layout.
7. Station assignment of the Punches: Right-click on the notching features in the feature tree and using **Apply To** move the punches to the positions indicated below. Right-click on the features indicated in red and select **Activate Here**.



8. **Save** the part.

Note: This part will be used later in the manual for the Design Change chapter case studies.

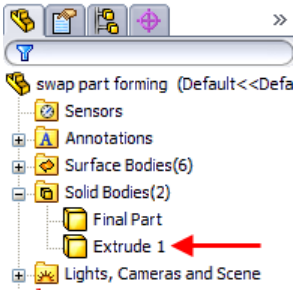
Case Study 3 – 2: Strip Layout Swap Part Method

For highly formed parts where feature recognition does not apply, the swap part feature can assist with strip layout creation. For this case study, In-place Cutting Features will be applied for trimming and piercing operations that are not in the original blank station or that occurs after forming operations. Translate and rotate will be used to rotate the part coplanar to the cut opening. These operations can be used in conjunction with the feature recognition methods. The 3DQuickForm Professional software module was used to create the forming steps and blank development for this case study.

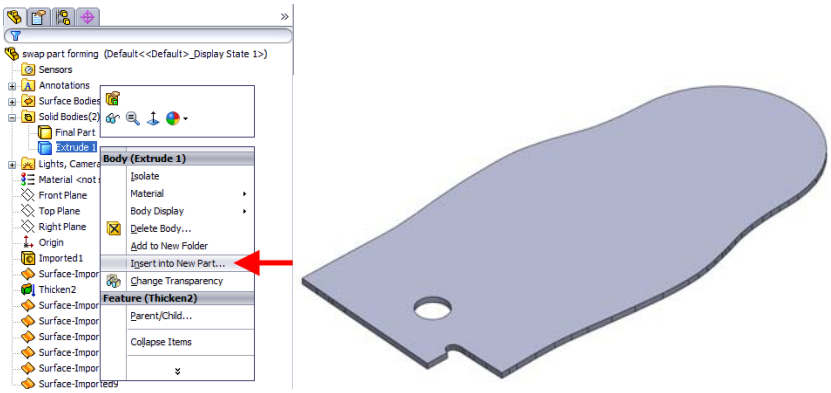
Saving the Flat Blank Solid

1. Open part file `swap part forming`. File location: `Desktop\Training\Strip-Swap part\`.

2. Change view to **ISO 2**.
3. Under the Solid Bodies folder right-click Extrude 1 and select show.

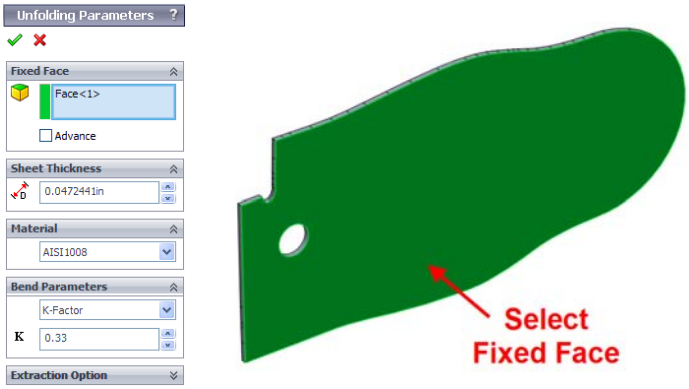


4. Right-click on the `Extrude 1` body and select **Insert into New Part**. Save the part in the `Strip-Swappart` folder and name it **swap part forming-blank.SLDPRT**.

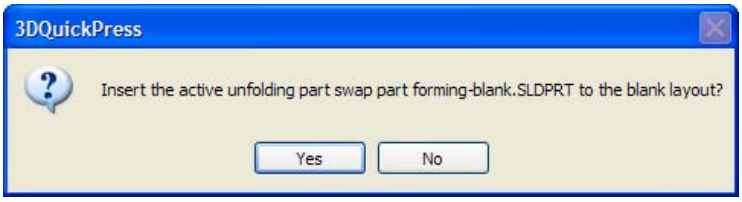


Create Unfold and Strip Layout

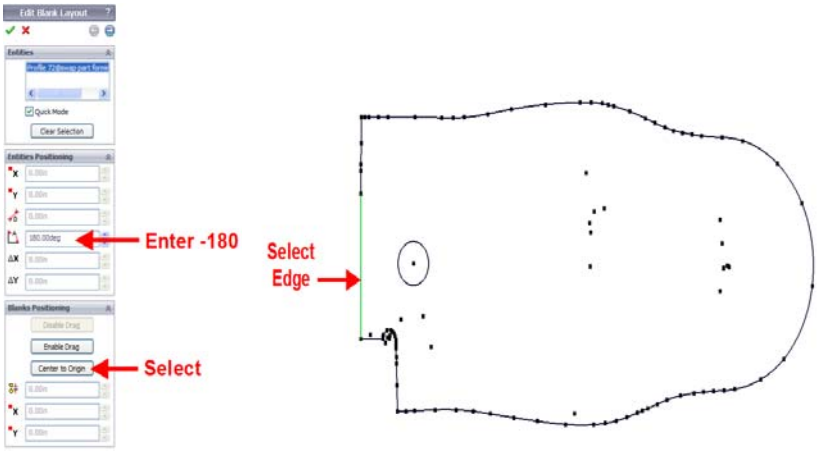
1. Select **Unfold Part** from the 3DQP Unfolding CommandManager tab. Select the highlighted face as the fixed face.

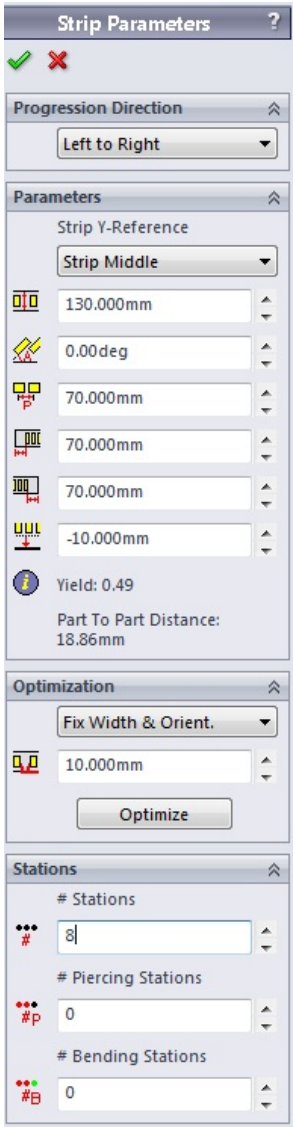


2. Click ✔ to finish the command. Click OK to close the `Bend Allowance Parameters` window.
3. Select Strip Layout Design from the 3DQP Unfolding CommandManager tab. Save the part in the `Strip-Swappart` folder and name it **swap part forming-strip layout.SLDPRT.**
4. Select **Yes** on the warning. Insert the active unfolding part into the blank layout?



5. Select the edge indicated below. Under `Entities Positioning` enter **-180 deg** for the angle. Select **Center to Origin.** Click ✔ to finish the command.





6. Right-click on `Strip Layout` at the top of the 3DQuickPress feature tree and select **Edit Strip Parameters**.

Parameters

- Select the y-reference option as **Strip Middle**
- Input the width of the strip – **130mm**
- Input the blank orientation with respect to the strip – **0.00deg**
- Input the inter-station pitch value – **70.00mm**
- Input the start margin for the strip – **70.00mm**
- Input the end margin for the strip – **70.00mm**
- Input the strip y position – **-10.00mm**

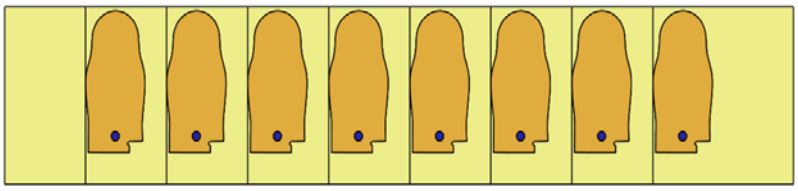
Optimization

- Select the optimization option – **Fix Width & Orientation**
- Input the width between the strip and blank layout – **10.00mm**
- Input the clearance of the components in adjacent stations – **10.00mm**

Stations

- Input the number of stations in the strip – **8**
- Input the number of piercing stations in the strip – **0**
- Input the number of bending stations in the strip – **0**

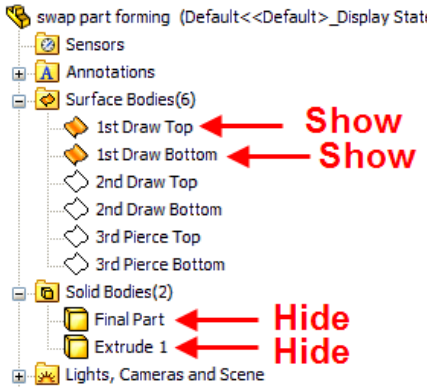
7. Click ✔ to finish the command.



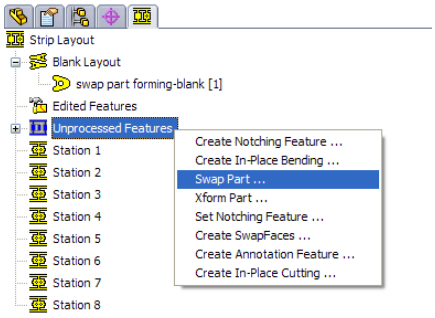
8. **Save** the part.

Modify Stations with Swap Part

1. Return to the `swap part forming` part.
2. **Hide** solid bodies `Final Part` and `Extrude 1`, and **show** surface bodies `1st Draw Top` and `1st Draw Bottom`.



3. Return to the `swap part forming-strip layout` part.
4. Right-click Unprocessed Features in the 3DQuickPress feature tree and select **Swap Part**.



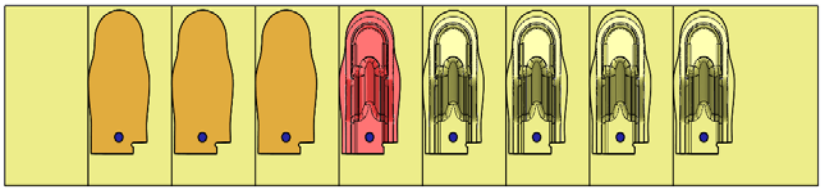
5. Select the **swap part forming.sldprt** from the `strip-swappart` folder.
6. Check the box next to `Face Groups` in the feature tree. Set the color of the `Top Surfaces` to **light green.** Set the color of the `Bottom Surfaces` to **dark green**.

Note: All surfaces can be turned on and inserted at one time, but it is often easier to control if this is done one step at a time.

7. Click ✔ to finish the command.

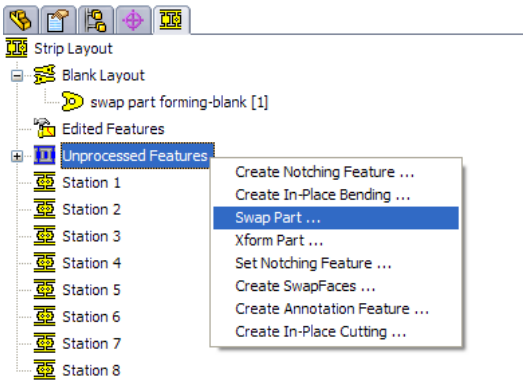
Swap Part for next form station

1. **Drag and drop** the swap part forming [1] from the Unprocessed Features folder to Station 4.



2. Return to the swap part forming part.
3. **Hide** surface bodies 1st Draw Top and 1st Draw Bottom, and **show** surface bodies 2nd Draw Top and 2nd Draw Bottom.

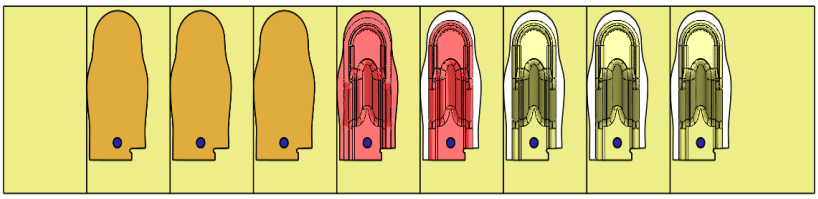
4. Return to the swap part forming-strip layout part.
5. Right-click Unprocessed Features in the 3DQuickPress feature tree and select **Swap Part**.



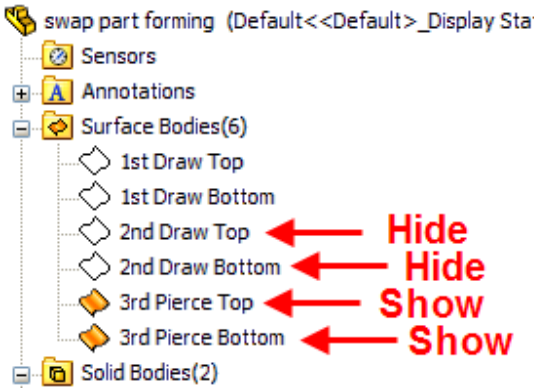
6. Select the swap part forming.sldprt from the strip-swappart folder.
7. Check the box next to Face Groups in the feature tree. Set the color of the Top Surfaces to **light purple.** Set the color of the Bottom Surfaces to **dark purple**.
8. Click ✔ to finish the command.

Swap part for third form station

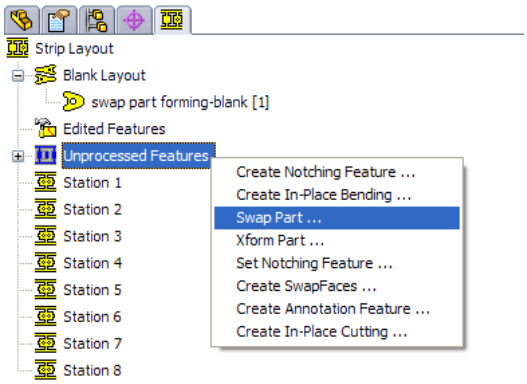
1. **Drag and drop** the swap part forming [2] from the Unprocessed Features folder to Station 5



2. Return to the `swap part forming` part.
3. **Hide** surface bodies `2nd Draw Top` and `2nd Draw Bottom`, and **show** surface bodies `3rd Pierce Top` and `3rd Pierce Bottom`.

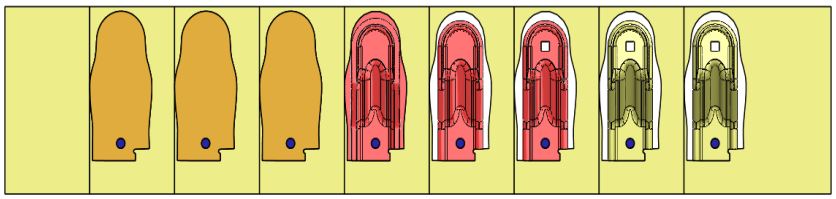


4. Return to the `swap part forming-strip layout` part.
5. Right-click **Unprocessed Features** in the 3DQuickPress feature tree and select **Swap Part**.



6. Select the **swap part forming.sldprt** from the `strip-swappart` folder.
7. Check the box next to `Face Groups` in the feature tree. Set the color of the `Top Surfaces` to **light blue.** Set the color of the `Bottom Surfaces` to **dark blue**
8. Click ✔ to finish the command.

9. **Drag and drop** the swap part forming [3] from the Unprocessed Features folder to Station 6.

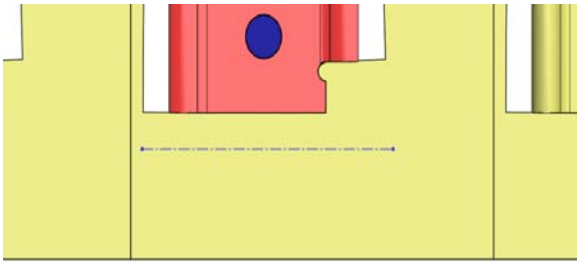


10. **Save** the part.

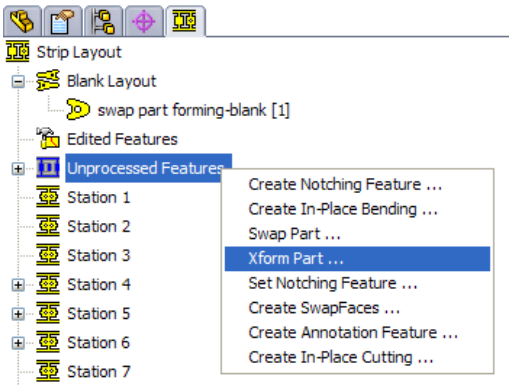
Editing the Swap Part Stations

The part will need to be rotated to allow a normal cut of the square opening. This will be accomplished using the 3DQuickPress Xform command to rotate the part so that the surface is coplanar with the stock material. The part will then be pierced with an In-place Cut Feature.

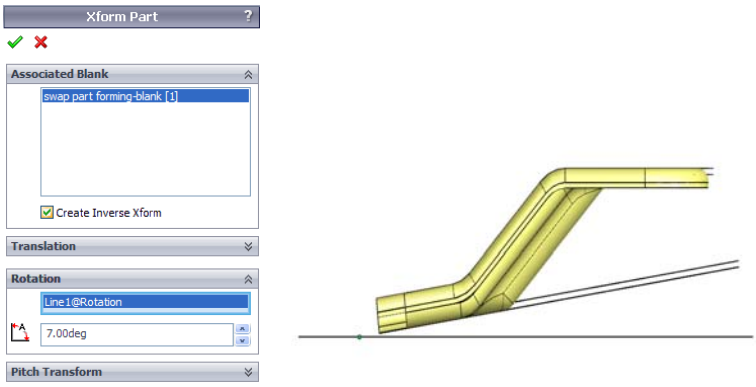
1. From the SolidWorks feature tree, right-click the `Front` plane and **Insert Sketch**.
2. Sketch a horizontal construction line beneath the part in station 6, and exit the sketch.



3. **Rename** the sketch to **Rotation.**
4. From the 3DQuickPress feature tree, right-click **Unprocessed Features** and select **Xform Part**.



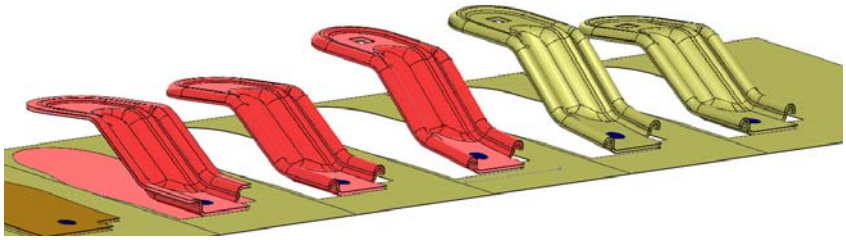
5. Under `Associated Blank` check the box for **Create Inverse Xform**, under the `Rotation` parameter select the **centerline** previously sketched and enter **7 deg** for the rotation angle.



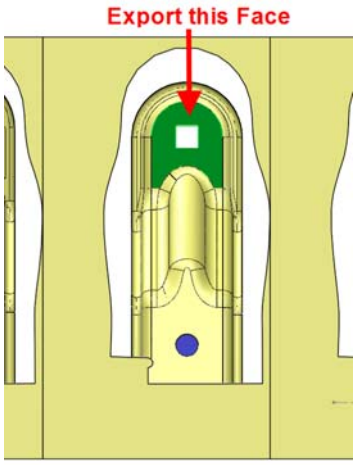
6. Click ✔ to finish the command.

Creating an In-Place cut operation

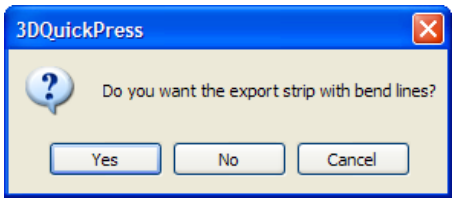
1. **Drag and drop** the `Xform Part 1` from the `Unprocessed Features` folder to `Station 6`, and `Reverse of Xform Part 2` to `Station 8`.



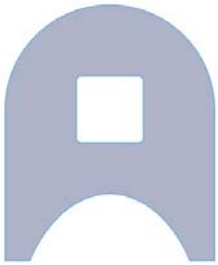
2. Click **bottom view** on the `3DQuickPress Standard View toolbar`.
3. Establish the pierce opening in station 7. Right-click on the face indicated below and selects **Export Selected Faces**.



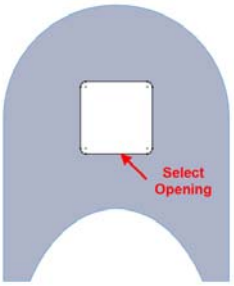
4. Select No to the message: Do you want to export the strip with bend lines?



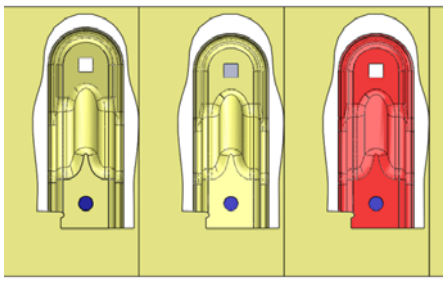
5. Click **Hide All** on the `3DQP Strip Layout` CommandManager tab.



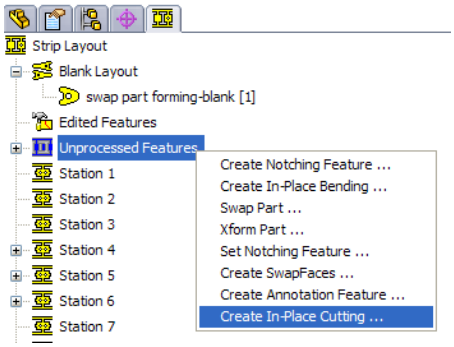
6. Right-click on the surface and Insert Sketch .
7. Select the edges of the hole in the center of the face and click Convert Entities from the Sketch toolbar.



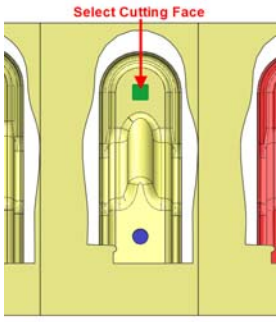
8. Select **Insert, Surface, Planar** to create a new surface bounded by the converted entities.
9. Right-click `Bottom Face 1` from the SolidWorks FeatureManager design tree and **Hide** .
10. Click **Show All** on the `3DQP Strip Layout toolbar.`



11. Right-click Unprocessed Features from the 3DQuickPress feature tree and select Create In-Place Cutting.



12. Under `Cutting Faces`, select the newly created square surface. Under `Show Start From` input **6**.

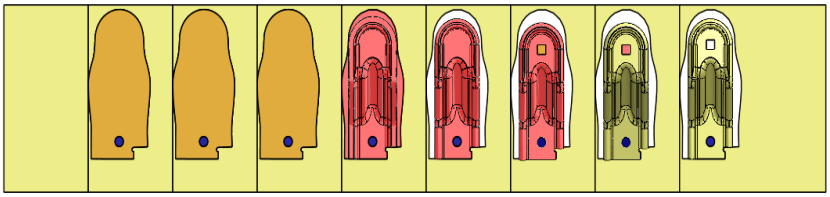


Note: By select the face of station 6 it will fill the opening for the pierce in station 6 up to the end of the strip.

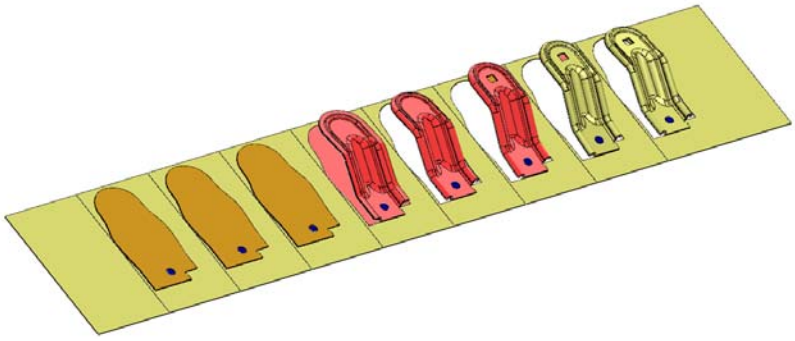
13. Click ✔ to finish the command.

Activate the In-Place cutting operation

1. **Drag and drop** the `In-Place Cutting 1` from the `Unprocessed Features` folder to `Station 7`.



2. **Save** & **Close** all opened the part.



Chapter 4
PUNCH DESIGN

- Creating a new Punch Design
- Create Cutting Punches
- Create Lancing
- Create L-Bend
- Create Pilots
- Create Stock Guides
- Translate Components
- Separate Components
- Using 3DQP File Properties

3DQUICKPRESS PUNCH DESIGN INTRODUCTION

Punch design is the third step in the process of creating a die design with 3DQuickPress. Unfolding and Strip Layout were the first two steps. With the Strip created you are now going to proceed to add the 3D components that will represent the tooling that will make contact with the strip layout design operations. In essence, you are designing around the strip, "wrapping it" with tooling to define the components that form and or guide the strip. The next step will be to "wrap" the Punch Design with the die set to complete the 3D design portion of the Tool.

Punch Design assists you with the creation and organization of SolidWorks Assemblies and Parts. The process is highly automated to maintain standards while allowing for custom input for job specific parameters. Many new tools are introduced to assist you with creating and editing these components and die settings.

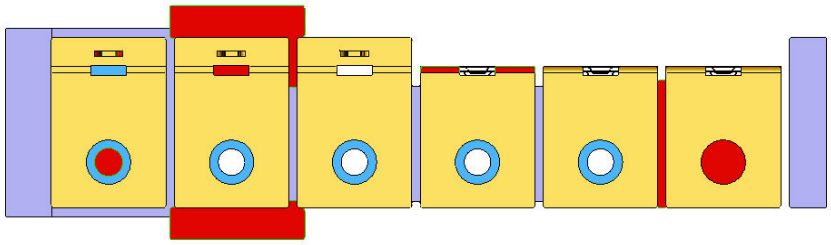
The Process

- Create the punch design assemblies
- Create punches automatically and semi-automatically.
- Modify punches
- Apply and modify User Defined Components (UDCs) for lancing, embossing and forming operations.
- Apply UDCs to add stock guides and pilot punches
- Translate and Separate components
- Check for interference and modify for clearance.
- Add non-graphical data (Properties) to components in groups to automate organization, selection, detailing and Bill or Materials.

Case Study 4 – 1 : Punch Design

In this chapter, you will take an existing strip layout part and create a punch design assembly. You will define the die set parameters for the tool so that the die set itself will be automatically created in the following design steps. You will use automatic punch design and semi-automatic punch design tools to accomplish placing pilots, lancing, piecing, notching, and bending operations. You will also be introduced to some productivity tools to move, copy, and edit components. Finally you will add addition intelligence by applying SolidWorks Properties in a group selection technique.

This tutorial will based on the following strip layout:

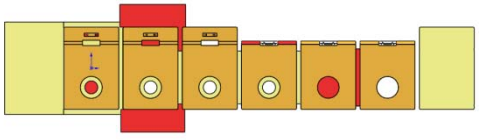


Pilot Hole Lancing	Notching Piercing		Bending		Piercing Cutoff

Starting a new Punch Design project

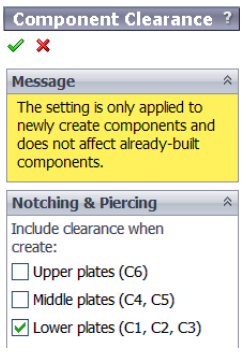
Starting a new punch design is an automated process with the ability to adjust the parameters of the project. The process starts with an existing strip layout design and the software will lead the user through a series of questions requesting project specific information. Then 3DQuickPress will create the new files (parts and assemblies). The user will continue the design process by adding custom components for each die operation.

1. Open the file named `P1 Strip Layout`. File Location: `Desktop\Training\Punch Design Creation\`



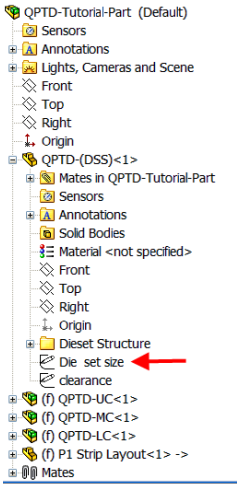
2. Click **Create Punch Design Assemby** from the 3DQP Punch Design CommandManager tab.

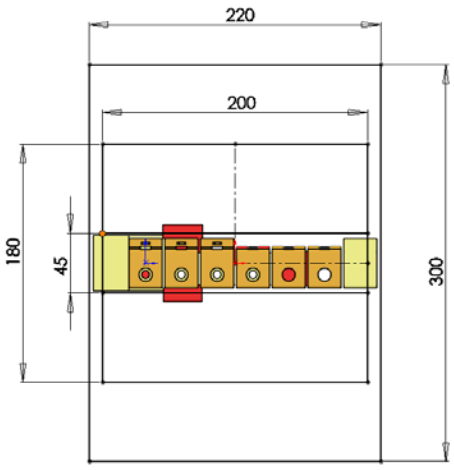
3. Input the project information in the `Edit Project Info` PropertyManager.
   - Input Project Name = **QPTD**
   - Project No. = **1**
   - Naming Convention = ***1-*3**
   - Die Set Template = **L1**
   - Round Punch (Straight) = **Misumi Straight**
   - Round Punch (Step) = **Misumi Round Step**
4. Leave all other settings at the default values and click ✔ to finish the command.
5. The Component Clearance PropertyManager should automatically open. Uncheck Upper plates and Middle plates under the Notching & Piercing parameter. Leave all other settings at the default values and click ✔ to finish the command.



By un-checking these two items, clearance bodies will not be created for punches in the Upper and Middle plates even though there is a clearance value greater than 0.000 entered. This is a Shop Practices decision that should be known before punch design continues otherwise the clearance holes for these punches through these plates will need to be manually created and or the punches will need to be recreated with 3DQuickPress commands. After the die set is created and die set Holes are run, the clearance will be 0.000 for these plates. Manufacturing process can then use the nominal opening and add clearance compensation during the CAM programming stage for the true clearances.

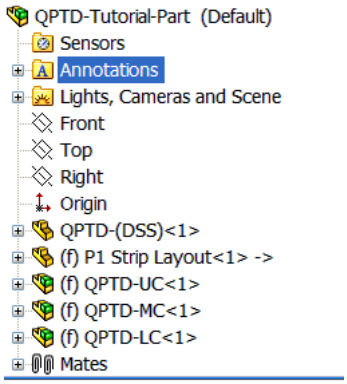
6. From the SolidWorks FeatureManager design tree, expand `QPTD-(DSS)`, right-click `Die set size` sketch and select Edit Sketch and change the die set dimensions. This is editing the actual die set even though it has not been created in 3D yet. The DSS (die set Structure) part contains the 2D driving sketches that will eventually be used to create the 3D die set parts and assemblies.





7. Exit edit sketch and exit **Edit Component** Mode.

8. Reorder the components in the FeatureManager design tree. Left-click and drag `P1 Strip Layout` and drop it after `QPTD-(DSS)`. This is a best practice since you will be accessing the DSS file often. Therefore, having `P1 Strip Layout` at the top of the FeatureManager design tree makes it easy to find and select.

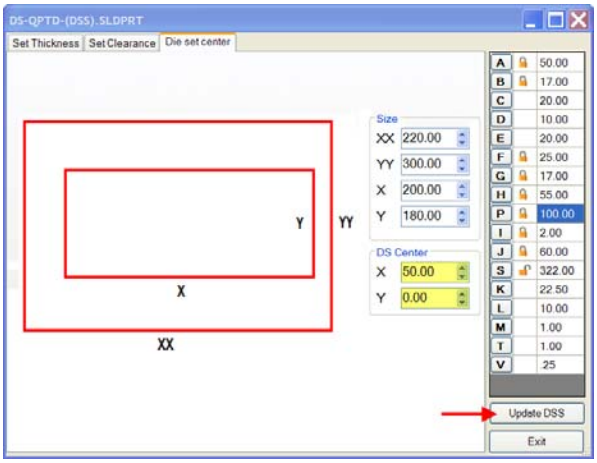


Adjust die set Center

1. Select **Modify DSS** from the 3DQP QTools toolbar.

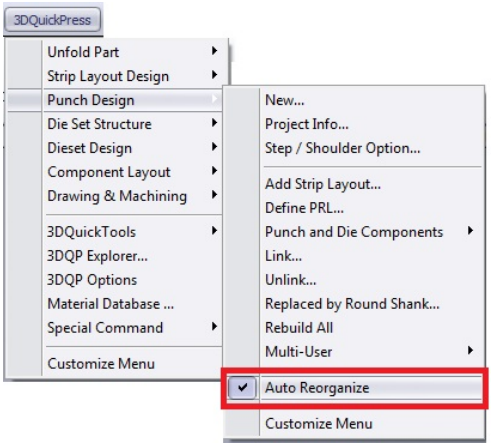
2. Select the `Die set center` tab at the top of the new window.

3. Change X to **50mm** and Y to **0mm** under the `DS Center` section. Select **Update DSS** to confirm the change. **Exit** the window.



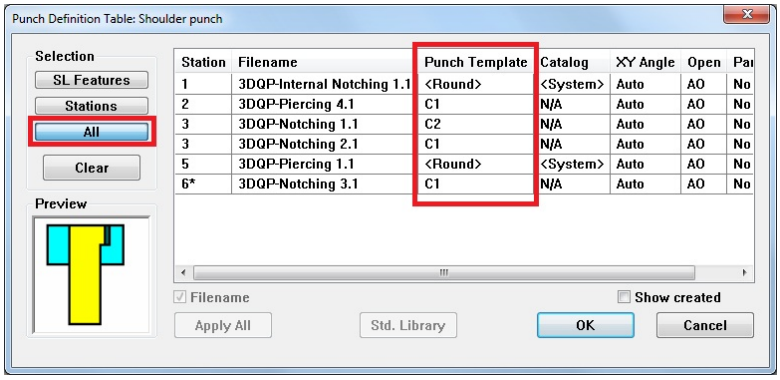
Create Cutting Punches

1. Turn on Auto-reorganize so components are automatically placed in correct sub-assemblies (UC/MC/LC) by selecting **3DQuickPress, Punch Design, Auto Reorganize**



2. Define the Punches. Click **Punch Definition Table** from the `3DQP Punch Design` CommandManager tab.

3. Under `Selection` click **All** and update **Punch Template** column and click **OK** to finish.



The option **Show created** is added to the Punch Definition Table to show those cutting features for which punches have been previously created.

Cutting features of the primary strip layout will be shown in a dark color while cutting features of other strip layout will be shown in grey.

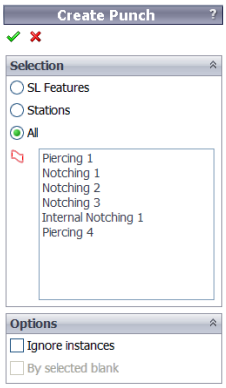
Default cutting punch components will be named using the format “project number-cutting feature name.x”, where “.x” represents the strip layout to which the cutting features belongs.

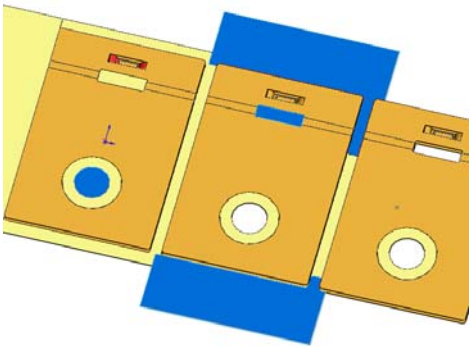
When there is a patterned cutting feature, a bracket with the instance number of the cutting feature is added to the station number.

<u>Create the Cutting Punches Automatically</u>

1. Click **Create Cutting Punch** from the `3DQP Punch Design` CommandManager tab.

2. Select **All** under the `Selection` parameter. Click ✔ to finish the command. 3DQuickPress will automatically create the punches.



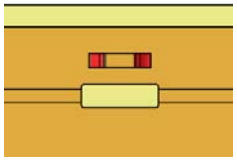


Create Lancing Punch Semi-Automatically

In this section we will create a punch with a User-Defined Component (UDC) that already exists. This will get us started very quickly yet further design changes will need to be made to this Punch to complete.

1. Go to a **Front View** .

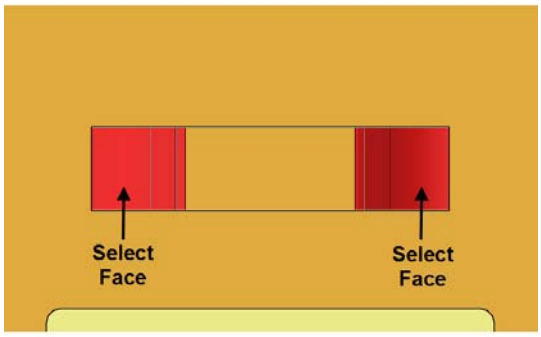
2. Zoom in to the upper part of station 1.



3. Click **Create User-Defined Component** from the 3DQP `Punch Design` CommandManager tab.

4. Under the `Definition` parameters select **Rect. Blank Punch** and select PRL **RECT01**, select the 2 faces indicated below, and select **Rectangular Type**.

Note: The 2 faces selected drive the punch size automatically.

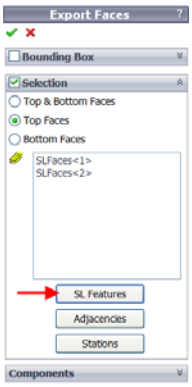


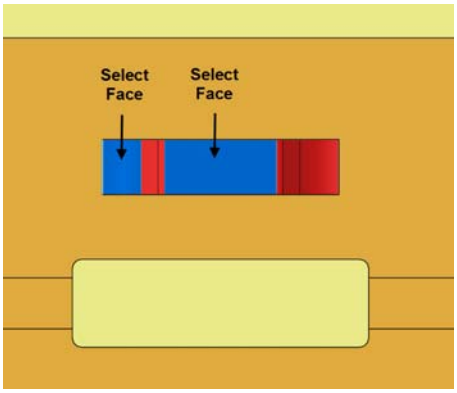
5. Click to finish the command.

6. Transfer the Lancing Faces from the Strip part to the Punch part for ease of design and performance. Click **Export Faces** the 3DQP Punch Design CommandManager tab.

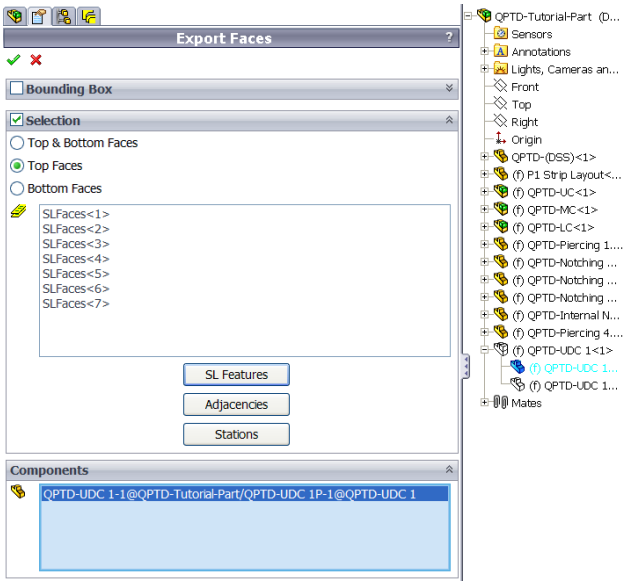
7. Check `Selection` parameter, and choose Top Faces.

8. Select the 2 faces below, and then click **SL Features**.

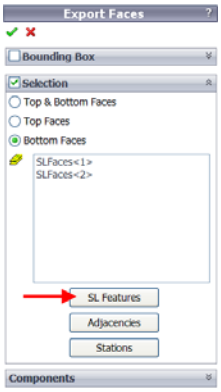


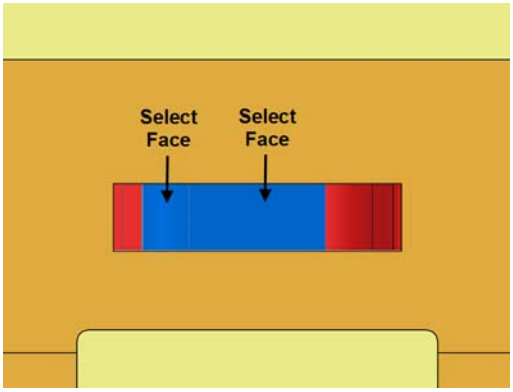


9. Under `Components`, Click **QPTD-UDC 1P** from the fly out tree.

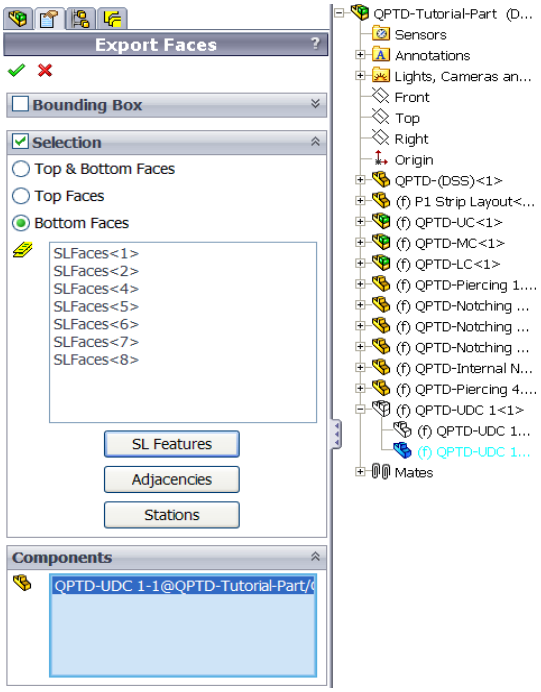


10. Click ✔ to finish the command.
11. Click **Hide/Show Top** from the `3DQP Punch Design` CommandManager tab, `QP Display` dropdown menu.
12. Click **Export Faces** from the `3DQP Punch Design` CommandManager tab.
13. Check `Selection` parameter, and choose Bottom Faces.
14. Select the 2 faces below, and then click **SL Features**.





15. Under `Components`, Click **QPTD-UDC 1D** from the fly out tree.



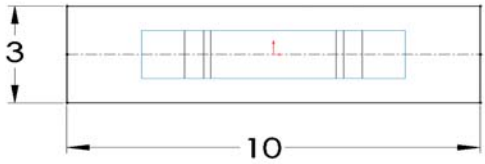
16. Click ✔ to finish the command.

There are two reasons why exporting the face to the part is important.

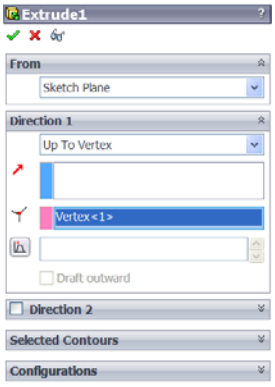
a. With the faces in the punch part, the user can open the punch on its own to further design in the part mode in lieu of working in the assembly to do part design. The typical user will design more efficiently in SolidWorks' Part Mode since it is easiest.

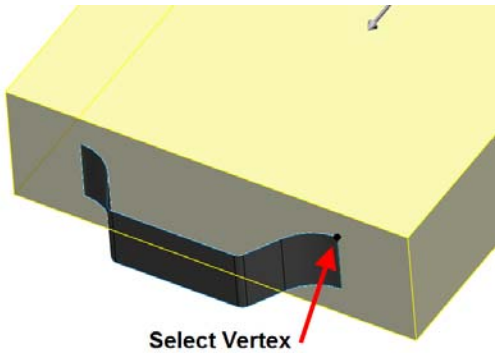
b. Avoid IN-CONTEXT ASSEMBLY relationships when appropriate. In-Context relationship features are very powerful. However, they complicate the design and reduce performance greatly.

17. Now you will design in the part mode to further design the Lance forming faces. Right-click `QPTD-UDC 1D` and click **Open Part** .
18. If necessary, reorder `Exported Sheet Metal Object Top face 1` so that it is before `Extrude 1` in the feature tree.
19. Edit `Sketch 1 of Extrude 1` and change dimensions to **3mm** and **10 mm**. Then exit the sketch .

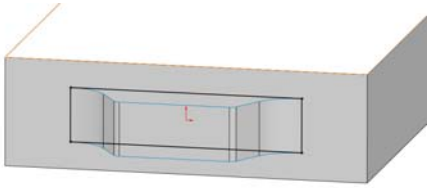


20. Right-click **Extrude1** and click **Edit Feature**.
21. Change the end condition to **Up To Vertex,** and select the vertex of the exported surface.

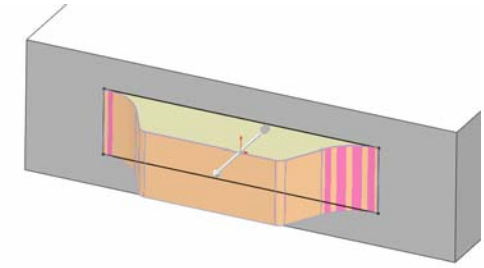




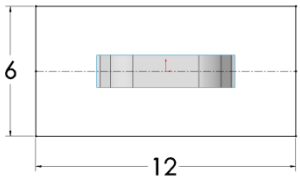
22. Click to finish the command.
23. Right-click on the end face of `Extrude 1`, **Insert Sketch**
24. Sketch a **Rectangle** , coincident to the corners of the exported surface.



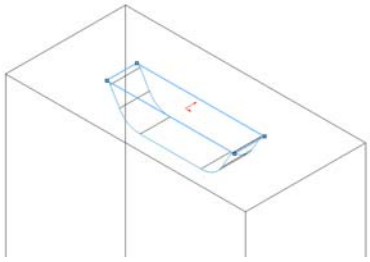
25. Click **Boss Extrude** , set the end condition as **Up to Surface**, and select `Exported Sheet Metal Object Top Face 1` as the surface.



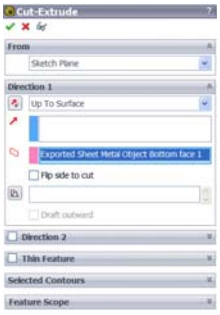
26. Click ✔ to finish the command.
27. **Save** & **Close** the Part.
28. From the QPTD-Tutorial-Part feature tree, right-click on QPTD-UDC 1D and click Open Part.
29. Edit `Sketch 2 of Extrude 1` and change dimensions to **6mm** and **12 mm**. Then exit the sketch.

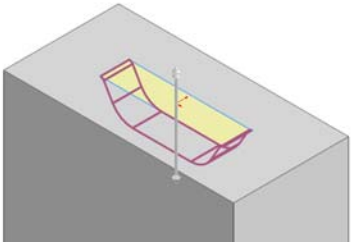


30. Change to an **Isometric View** , right-click on the end face of `Extrude 1`, **Insert Sketch** .
31. Change the display style to **wireframe** . Sketch a **Rectangle** , coincident to the corners of the exported surface.



32. Click **Extrude Cut** , set the end condition as **Up to Surface**, and select `Exported Sheet Metal Object Bottom Face 1` as the surface.

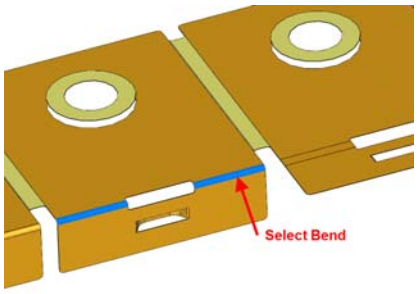




33. Click ✔ to finish the command.
34. **Save** & **Close** the Part. This will return you to the Punch Design Assembly.
35. Right click on **QPTD-UDC 1** subassembly and click **Show**
36. **Save** the assembly.

Create L-Bend Assembly automatically

1. Select **Create L-Bend** from the 3DQP Punch Design CommandManager tab.
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**.



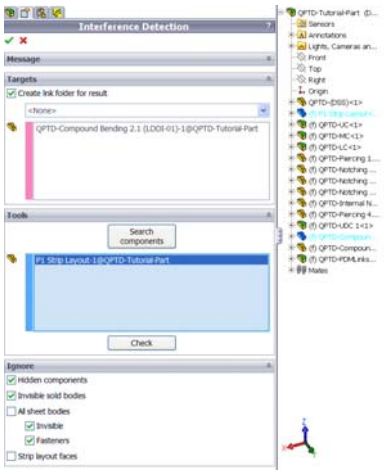
3. Click ✔ to finish the command.

Checking for Interference with 3DQP tools

3DQP Interference is a separate command from the standard SolidWorks command. It adds the ability to check for interference between solids and surfaces. Also, it gives you the ability to automatically transfer the interfering faces and or bodies to the part level. This allows the user to see and design at the part mode around the actual interferences. The interferences are listed in the FeatureManager design tree and are time stamped. They can be deleted, hidden, and are not associated with the original part that from which they were transferred to avoid complex relationships. This is done for reasons similar to those for Export Faces.

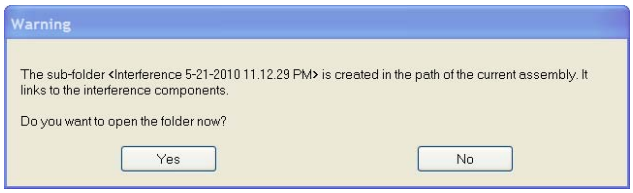
1. Click **Interference Detection** on `3DQP QTools` CommandManager tab.

2. From the fly out feature tree, select **QPTD-Compound Bending 2.1(LDDI-01) -1** under the `Targets` selection box, and select **P1 Strip Layout** under the `Tools` selection box.



3. Click ✔ to finish the command.

4. Click **YES** to the warning message. This will open Windows Explorer. The folder will contain links for parts that have interference.



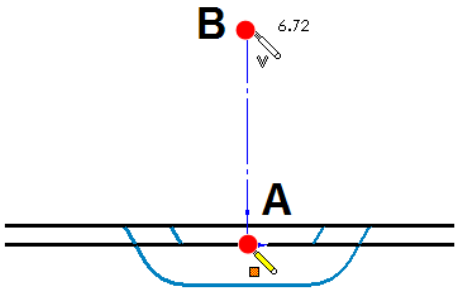
5. Double-click the link to open the document.

6. Change to an **Isometric View** .

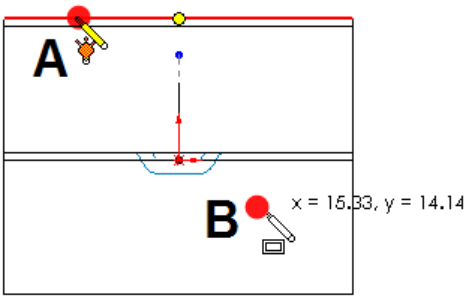
7. Right-click the top face and **Insert Sketch** . Change the display style to **wireframe** .

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

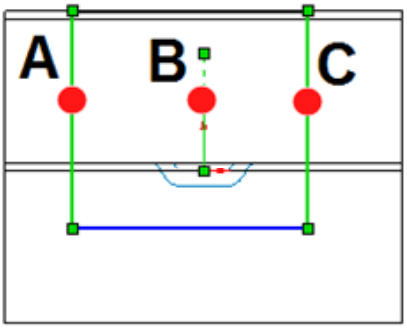
   Sketch a **Center Line** and from A to B**.**



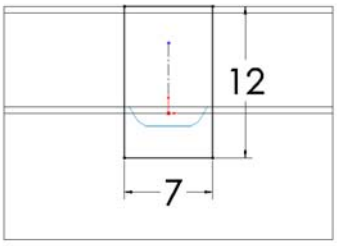
9. Sketch a **Rectangle** from point **A** to point **B**.



10. Hold Control and select lines **A, B, & C** then click **Symmetric** in the `Add Relations` PropertyManager.



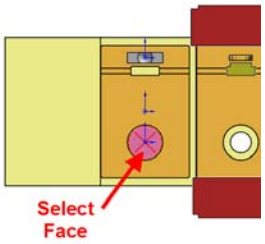
11. Click **Smart Dimension** . Detail the sketch as shown below.



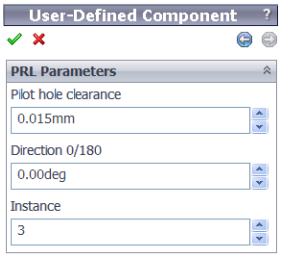
12. Click **Extrude Cut** , change the end condition to **Through All**. Click to finish the command.
13. Change the Display Style back to **Shaded with Edges** .
14. Right-click the Surface Bodies folder, select Hide .
15. **Save and close** the file.
16. **Save** P1 the QPTD-Tutorial-Part assembly.

Create a Pilot with a UDC

1. Change to a **Front View** .
2. Click **Create User-Defined Component** from the `3DQP Punch Design` CommandManager tab.
3. Under the `Definition` parameter, select **Pilots** for PRL type, select **Pilot-01** for the Pilot type, select the **Round Type** radio button, and select the face indicated below.



4. Click Next , Set the pilot hole clearance to **0.015mm** and the Instance to **3**.



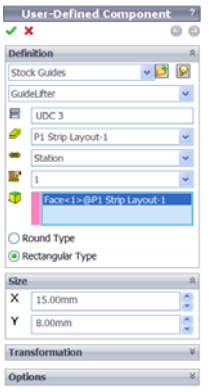
5. Click to finish to finish the command.

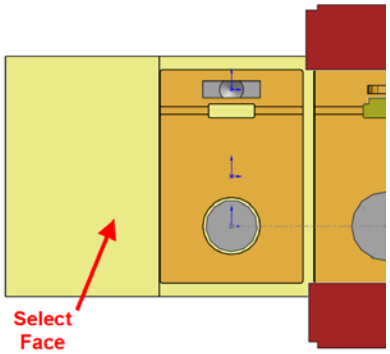
Create Stock Guides with a UDC

1. Change to a **Front View** .

2. Click **Create User-Defined Component** from the 3DQP `Punch Design` CommandManager tab.

3. Under the `Definition` parameter, select **Stock Guides** for PRL type, select **GuideLifter** for the Stock Guide type, select the **Rectangular Type** radio button, and select the face indicated below. Under the `Size` parameter set X to **15mm** and Y to **8mm**.



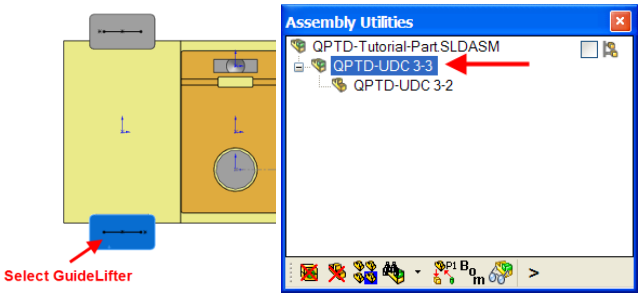


4. Click to finish the command.

Translating Components

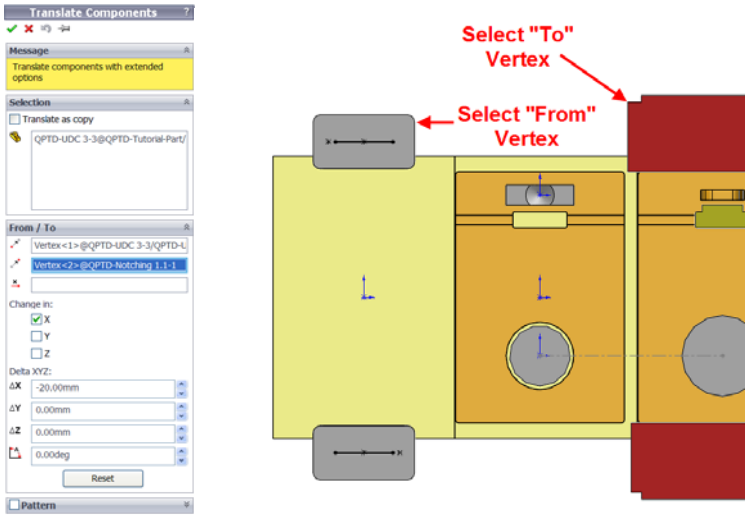
1. Click **Assembly Utilities** from the 3DQP `QTools` CommandManager tab.

2. Select one of the GuideLifter parts from the graphic window. In the Assembly Utilities window select the GuideLifter assembly. Close the Assembly Utilities window.



3. Click **Translate Component** from the 3DQP QTools CommandManager tab.

4. Under the `From/To` parameters select the vertex of the GuideLifter as the From point and select the vertex of the punch as the To point. Change only in the **X** direction and set the delta X value to **-20mm**.

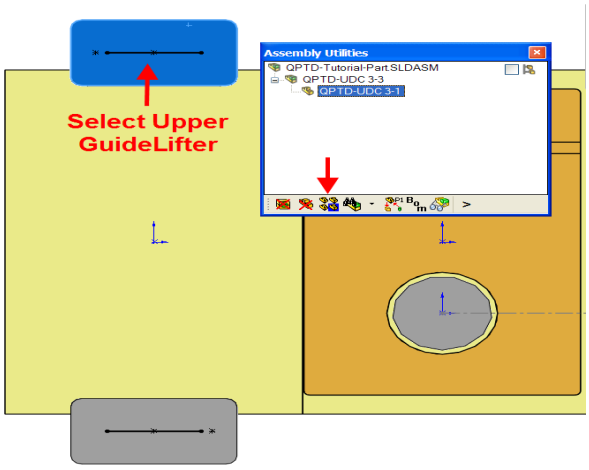


5. Click ✔ to finish the command.

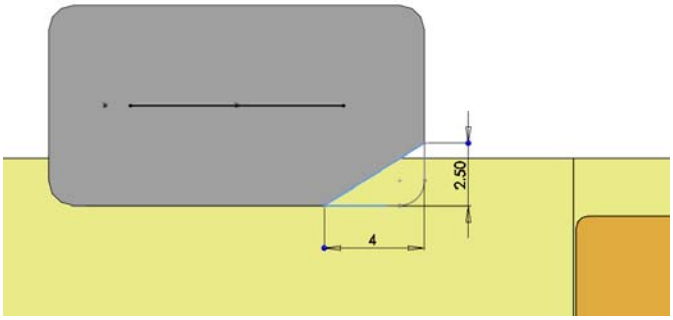
Separate Component with Assembly Utilities

The insert stock guides come from the same file `QPTD-UDC 3.1.SLDPRT`. The upper guide will need to be cut for clearance purposes while the lower guide remains the same. This will require a standard part to be slightly customized and documented appropriately without changing the original instance(s) in this design and the Library component for all designs. 3DQuickPress has an automated tool to achieve this called Separate Component.

1. Select upper guide and click **Assembly Utilities** from the 3DQP QTools CommandManager tab.



2. Click Separate Component and Input file name QPTD-UDC 1A.SLDPRT. Click Save.
3. Right-click the upper guide and select **Edit Part** .
4. Cut the corner with dimension **4mm x 2.5 mm.**

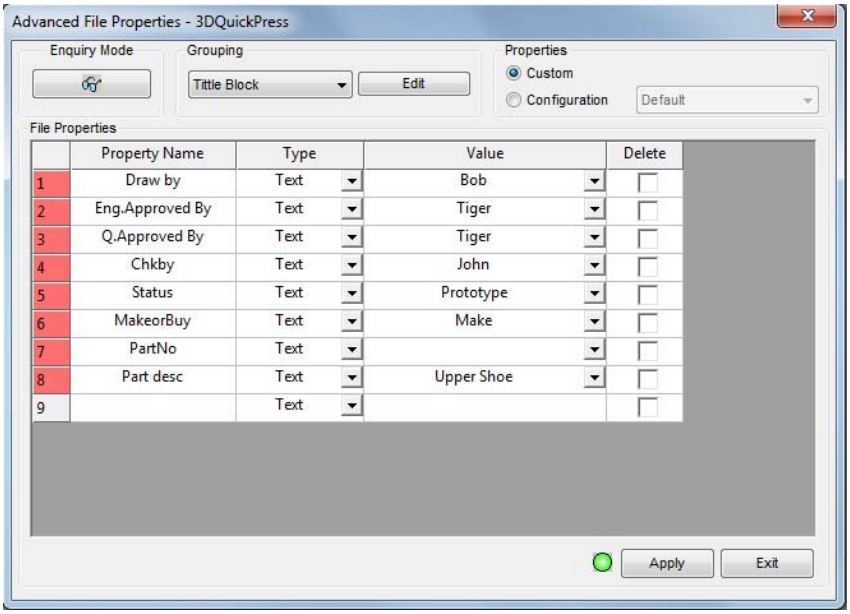


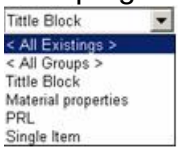
5. Exit edit part mode.
6. **Save** the part.

Note: The original part is not modified even though the separated part is modified.

MANAGING PART FILE PROPERTIES QUICKLY WITH 3DQUICKPRESS

Editing file properties in the context of assembly is not easily achieved with SolidWorks basic capabilities. Therefore 3DQuickPress has enhanced the ability to add properties to Parts while in the Assembly design mode with group methods. File properties are very important to the design process for many reasons, such as, advanced selection and change, detail drafting, and Bill of Material automation just to name a few. The 3DQuickPress Edit File Properties tool can also be customized to predefine your standards and avoid manual entry of text.

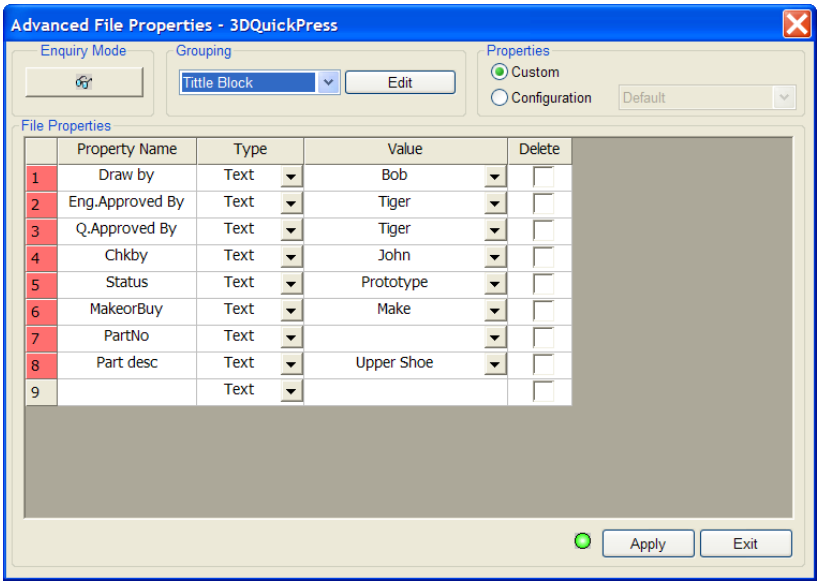


Enquiry Mode	When the Enquiry Mode button is depressed the Properties of selected components are shown in the dialog box table immediately.
Grouping 	Grouping Filters allows the user to organize and simplify the file properties by a name. This named group can help the user see only the properties relevant to a function or need (a group is user defined) • <All Existings> – Display all existing file properties in the selected component • <All Groups> – Display an editing page containing all groups • Title Block, Material properties, and so on are examples of a user defined groups. When this is selected, only the properties connected to the title block of a drawing are shown.

Edit	See Chapter 9, Administration and Customization.
Properties	Edit configuration specific properties or File properties.
Apply	Apply the table content to select components and keep editing.
Exit	Exit the function.
Legends 	The color of the dialog property item number acts as a key. It tells the user how the current group compares to the selected part file(s) existing properties. For instance, if the number is red, that file property does not currently exist in the selected part. If the number is gray then the property exists in the selected part and the current value is displayed. If you choose to Apply (add) the group properties that currently do not exist in the currently selected part file(s) the field will update to green to indicate that the Apply was successful.

Assigning Material Properties to multiple parts with 3DQuickPress

1. Click **File Properties**  from the 3DQP QTools CommandManager tab to launch File Properties Editor.



2. Set the `Grouping` to **Title block.**
3. Fill in the Values using the existing pull down options or type in some example information like listed above.
4. Select any component to apply the properties to.

Tip: Use window select to pick groups of components.

5. Click **Apply**.

How to check file properties of component

1. Click **File Properties**  from the 3DQP QTools CommandManager tab to launch the File Properties Editor.
2. Click the `Enquiry Mode`  button (depressed) and click on any component. The selected component will display its existing File Properties.

Chapter 5
CREATING A DIE SET

- Introduction
- Creating the die set
- Designing with Layout Sketches
- Adding Standard Components
- Interference and Clearance Design
- File Property Automation
- Cutting die set Holes

3DQUICKPRESS DIE SET DESIGN INTRODUCTION

die set design is the fourth step in the process of creating a die design with 3DQuickPress. Unfolding, Strip Layout and Punch Design were the first three steps. With a Punch Design assembly started you are now ready to create a new die set design assembly that will incorporate the strip and portion of the punch design assemblies.

die set creation assists you with the creation and organization of SolidWorks assemblies and parts. The process is highly automated to maintain standards while allowing for modifications. Many new tools and techniques are introduced to assist you with creating and editing the Die Set, standard components, and clearance holes.

The die set assembly is not the simple addition of all parts and assemblies created thus far. It is a unique *combination* of *existing* part/assemblies and *new* parts/assemblies and can also include drawings.

The die set creation is a special combination of some of the punch design assemblies. Not all assemblies are inserted to the die set Assembly. Only the UC/MC/LC sub-assemblies are inserted to the die set design from the punch design. The top level punch design assembly is not inserted into the die set design. In addition, the master or “Operation-Based” Strip Layout part is not inserted into the die set assembly.

The “Operation Based” Strip Layout Part is converted to a new, associated (linked), simplified part file that represents the strip for performance and visualization purposes. The strip is represented by the upper and lower material surfaces. Performance is also enhanced since it is not actively associative (checking for updates on every rebuild) but, passively associated to the original strip layout part. Passive associativity is controlled by 3DQuickPress to allow the user to choose when a rebuild and/or revision of the strip layout part is to be checked for, and introduced to, the die set design.

After the creation of the die set the addition of standard components is automated with layout techniques and tools for ease of insertion and modification.

Finally, die set holes and clearances can be manually and automatically calculated for the user for both standard and project specific components. These clearances are calculated and created with intelligence (File Properties) in an active and passive timing depending on the tools used. These tools aid the user in ensuring accuracy which in turn reduces days of tedious design tasks to minutes. This allows the designer to focus on fine-tuning and checking the design.

DIE SET CREATION

Starting with an existing punch design, this case study will take you through the process of creating a Die Set. The steps of creating layout sketches, adding standard components, checking for and resolving interference, adding file properties for estimating, and creating the die set holes and clearances will all be covered through these exercises.

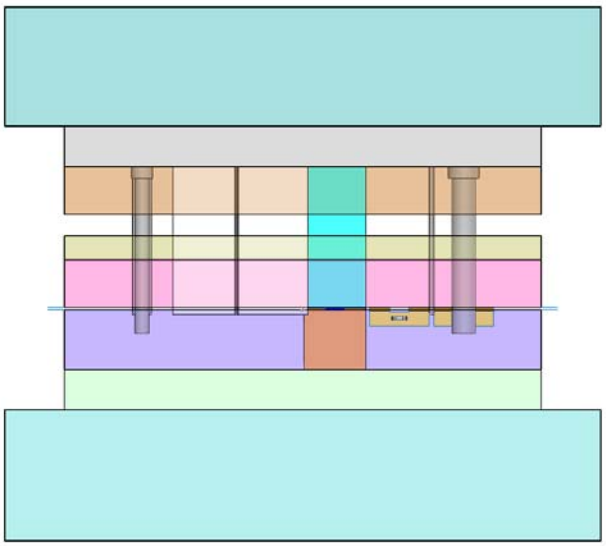
Case Study 5 – 1 : Initiate die set Creation

1. Open the file named `T5 Strip Layout-T5 Part.sldasm`. File Location: `Desktop\Training\Dieset Creation\`

2. Click **Create Dieset**  from the 3DQP Punch Design CommandManager tab to create the die set assembly.
3. Set the die set Templates to **00**.
4. Click ✔ to finish the command.

3DQuickPress will automatically insert the components of the die based on the PRL template.

5. Input **T5 Strip Layout-01** for the filename and click Save.

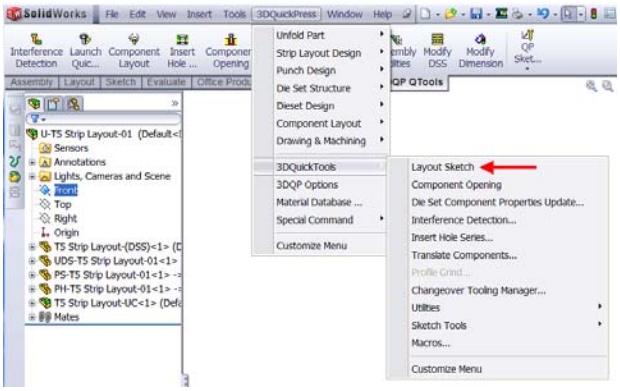


Case Study 5-2: Adding Standard Components

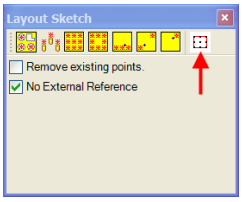
Adding standard components can be done using many techniques with SolidWorks and 3DQuickPress. This technique is a 3DQuickPress method known as Insert Component method. The Sketch is created in any way you would like. However, 3DQuickPress offers many options to automate common design layouts. The sketch is a series of point entities along with dimensions and relationships. These points are used to pattern the components. This Layout sketch can also be reused for other components that will need to be added later. It is recommended to name the sketch for ease of modification. If change is needed, the user simply edits a point in the sketch to add or remove instances of the related part(s).

Insert Socket Head Cap Screws to the Upper Subassembly

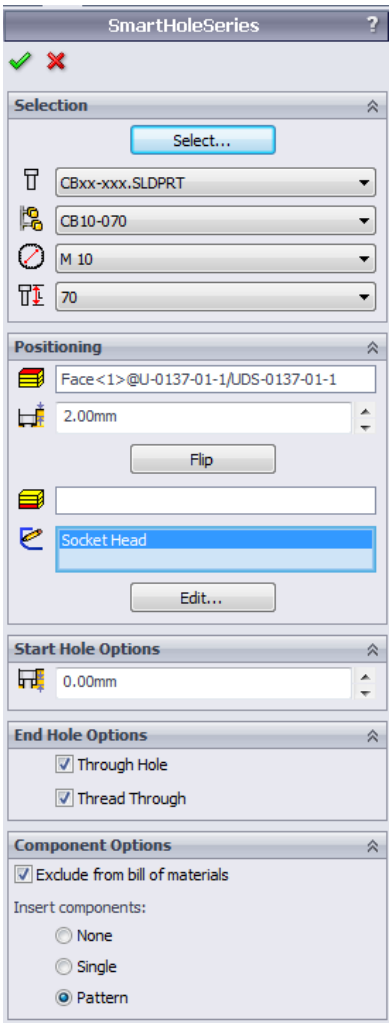
1. Right-click **U-T5 Strip Layout-01** and select **Open Assembly**.
2. Select the **Front Plane** from the feature tree. Select **Layout sketch** on the `3DQP Punch Design` CommandManager tab**,** `QPSketch Tools` dropdown menu.



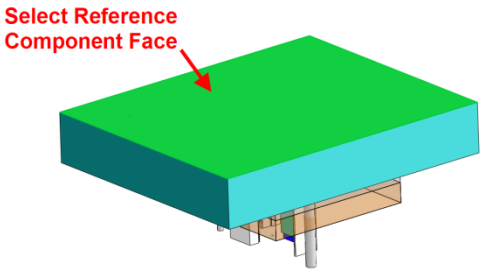
3. Select the **Legacy Layout Sketch Function**.



4. Input `X-Size` **150mm** and `Y-Size` **120mm**.
5. Click to finish the command.
6. Rename sketch1 to **Socket Head.**
7. Click **Insert Component** from `3DQT Die Set Design` CommandManager tab.



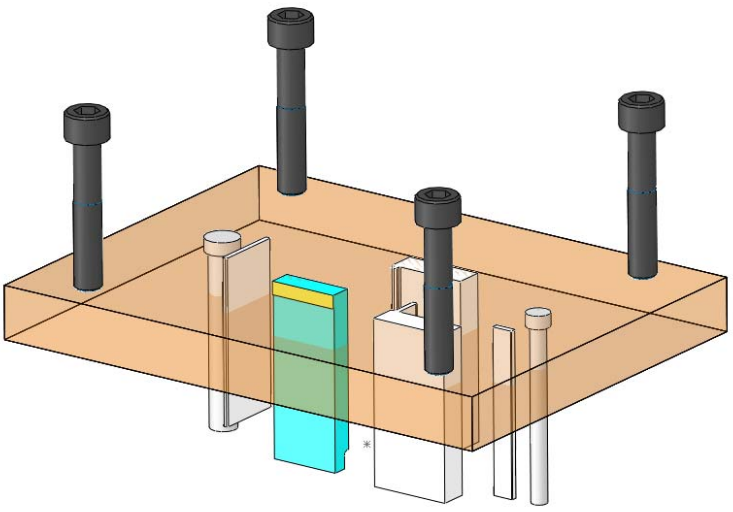
8. Under `Selection` press the **Select** button and browse to `C:\Program Files\3DQuickPress\Standard Components\Misumi\Socket Head Screw`. Select **CBxx-xxx.**
9. Under `Part Configuration` select **CB10-070.**
10. Under `Position` select the face indicated below and set the `distance from reference location` to **2mm.** (Note: You may need to click the FLIP button if the screw is not going into the plate)



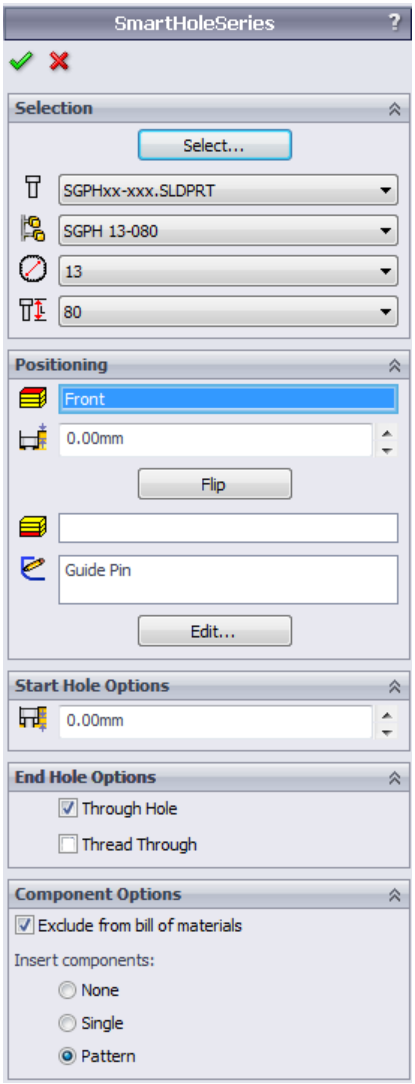
11. Under `Layout Sketch` select the **Socket Head** sketch from the feature tree.
12. Under `End Hole Options` check **Through Hole** and **Thread Through**.
13. Click ✔ to finish the command.

Insert Guide Pins to the Upper Subassembly

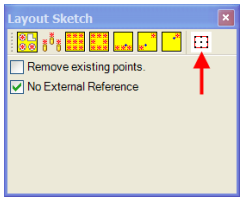
1. Hide the top plates. Right-click `UDS-T5 Strip Layout-01 & PS-T5 Strip Layout-01` and select **Hide**.



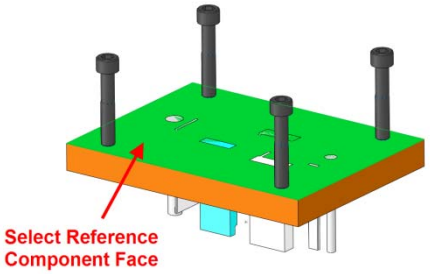
2. Right-click `T5Strip Layout-(DSS)` and select **Open Part**.



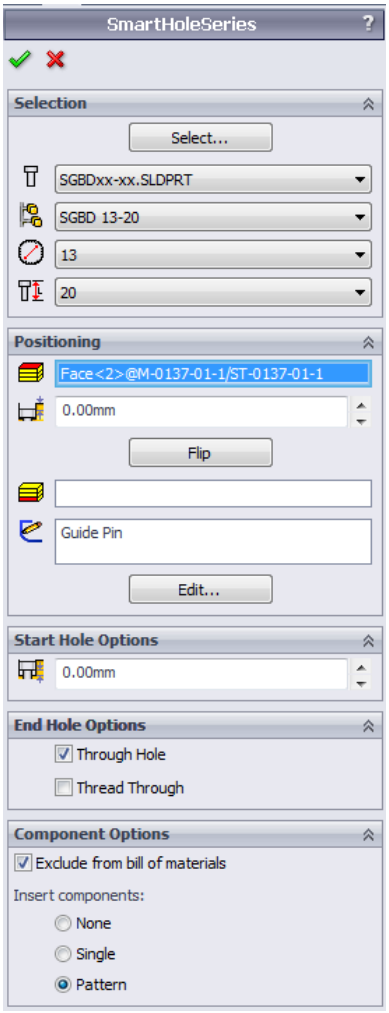
3. Create a Layout Sketch in the DSS part. Select the **Front Plane** from the feature tree. Select **Layout sketch** on the `3DQP Punch Design` CommandManager tab**,** `QPSketch Tools` dropdown menu.
4. Select the Legacy Layout Sketch Function.



5. Input `X-Size` **120mm** and `Y-Size` **100mm**.
6. Click to finish the command.
7. Rename sketch1 to **Guide Pins**.
8. **Save** and **Close** the `T5Strip Layout-(DSS)Part` window.
9. Click **Insert Component** from `3DQT Die Set Design` CommandManager tab.
10. Under `Selection` press the **Select** button and browse to `C:\Program Files\3DQuickPress\Standard Components\Misumi\Guide Pin`. Select **SGPHxx-xxx.**
11. Under `Part Configuration` select **SGPH 13-080.**
12. Under `Position` select the face indicated below and set the `distance from reference location` to **0mm.**



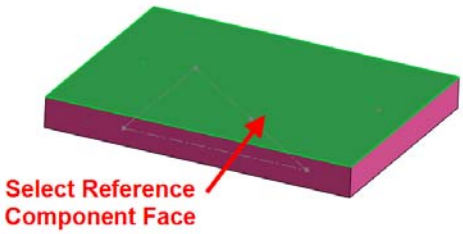
13. Under `Layout Sketch` select the **Guide Pins** sketch from the feature tree under `T5Strip Layout-(DSS)`
14. Under `End Hole Options` check **Through Hole** and uncheck **Thread Through**.
15. Press to finish the command.
16. **Save** & **Close** the `U-T5 Strip Layout-01` assembly window.



Insert Guide Pin Bushes to the Middle Subassembly

In this scenario you will reuse an existing layout sketch from the common DSS part. Organization and reuse are the benefits to this approach. Remember, all layout sketches can be found quickly by going to the common part to create and edit sketches even though the parts are located in different sub-assemblies. This allows the user to work in smaller assemblies versus working at the master assembly which will always be the largest and most complicated.

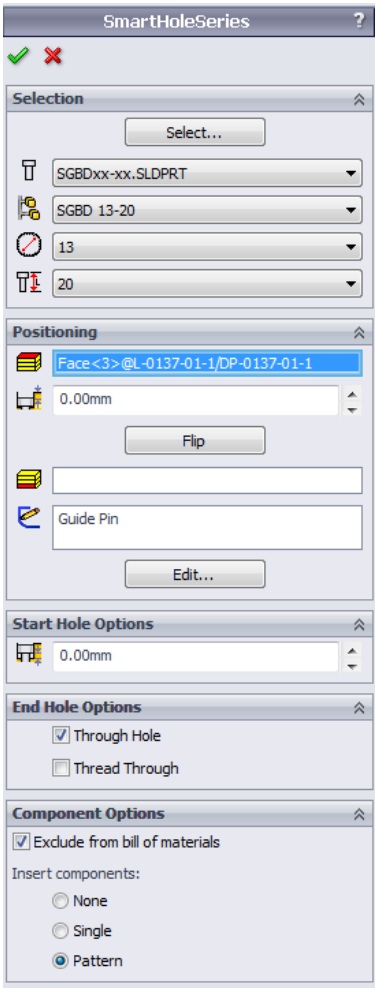
1. **Open** `M-T5 Strip Layout-01` subassembly from the T5 Strip Layout -01 assembly.
2. Right-click `SB-T5 Strip Layout-01` and select **Hide**.
3. Click **Insert Component** from `3DQT Die Set Design` CommandManager tab.
4. Under `Selection` press the **Select** button and browse to `C:\Program Files\3DQuickPress\Standard Components\Misumi\Guide Pin Bush`. Select **SGBDxx-xx.**
5. Under `Part Configuration` select **SGBD 13-20.**
6. Under `Position` select the face indicated below and set the `distance from reference location` to **0mm.**



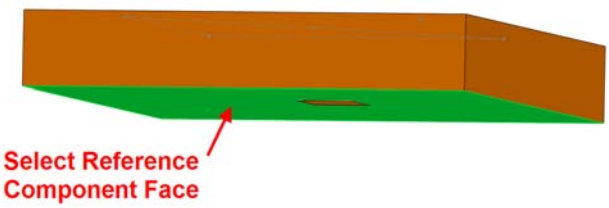
7. Under `Layout Sketch` select the **Guide Pins** sketch from the feature tree under `T5Strip Layout-(DSS)`
8. Under `End Hole Options` check **Through Hole** and uncheck **Thread Through**.
9. Press ✔ to finish the command.
10. **Save** & **Close** the `M-T5 Strip Layout-01 assembly` window.

Insert Guide Pins to the Lower Subassembly

1. **Open** `L-T5 Strip Layout-01` subassembly from the T5 Strip Layout -01 assembly.
2. Right-click `DS-T5 Strip Layout-01` & `LDS-T5 Strip Layout-01` and select **Hide**.



3. Click **Insert Component** from `3DQT Die Set Design` CommandManager tab.
4. Under `Selection` press the **Select** button and browse to `C:\Program Files\3DQuickPress\Standard Components\Misumi\Guide Pin Bush`. Select **SGBDxx-xx.**
5. Under `Part Configuration` select **SGBD 13-20.**
6. Under `Position` select the face indicated below and set the `distance from reference location` to **0mm.**



7. Under `Layout Sketch` select the **Guide Pins** sketch from the feature tree under `T5Strip Layout-(DSS)`
8. Under `End Hole Options` check **Through Hole** and uncheck **Thread Through**.
9. Click ✔ to finish the command.
10. **Save and Close** the `L-T5 Strip Layout-01` assembly window.

Case Study 5 – 3 : Interference and Clearance design

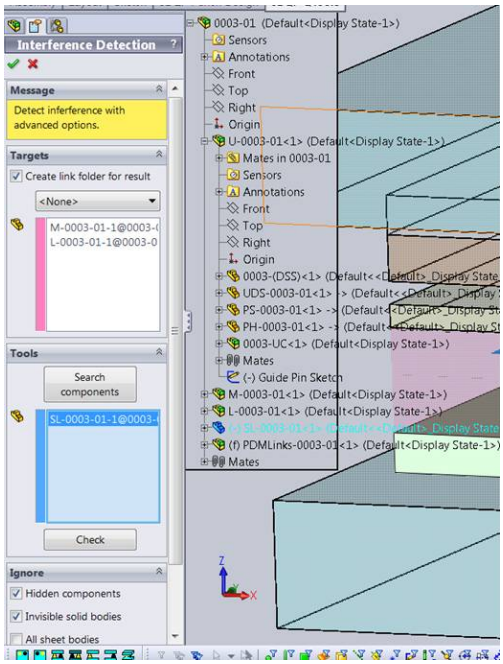
Interference and clearance design is crucial for a tool to function the first time it is manufactured. Therefore, 3DQuickPress has enhanced the interference detection capabilities for the die designer to quickly find and resolve these areas of collision. The added capabilities of 3DQuickPress interference detection include:

- Solid to surface interference
- Interference file short cuts to track and resolve which components have the interference
- Insertion of interference bodies into the component with a time stamped folder per interference situation
- Selective targets and tools to control the scope and rebuild time per calculation

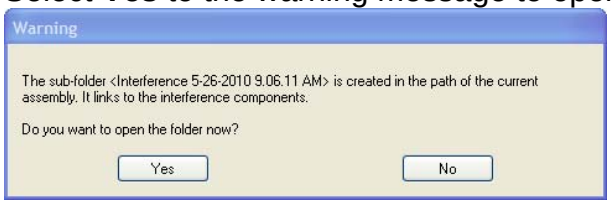
Check for Interference between the Strip and the Stripper Plate and the Die Plate.

1. In the `T5 Strip Layout-01` assembly, click **Interference Detection** from the `3DQP QTools` CommandManager tab.

2. Select the die plate (`DP-T5 Strip Layout-01.sldprt`) and the stripper plate (`ST-T5 Strip Layout-01.sldprt`) so they are listed in the `Targets` selection box. Select from the Flyout Feature Manager the Strip Layout part called `SL-T5 Strip Layout-01` while the `Tools` selection box is active.



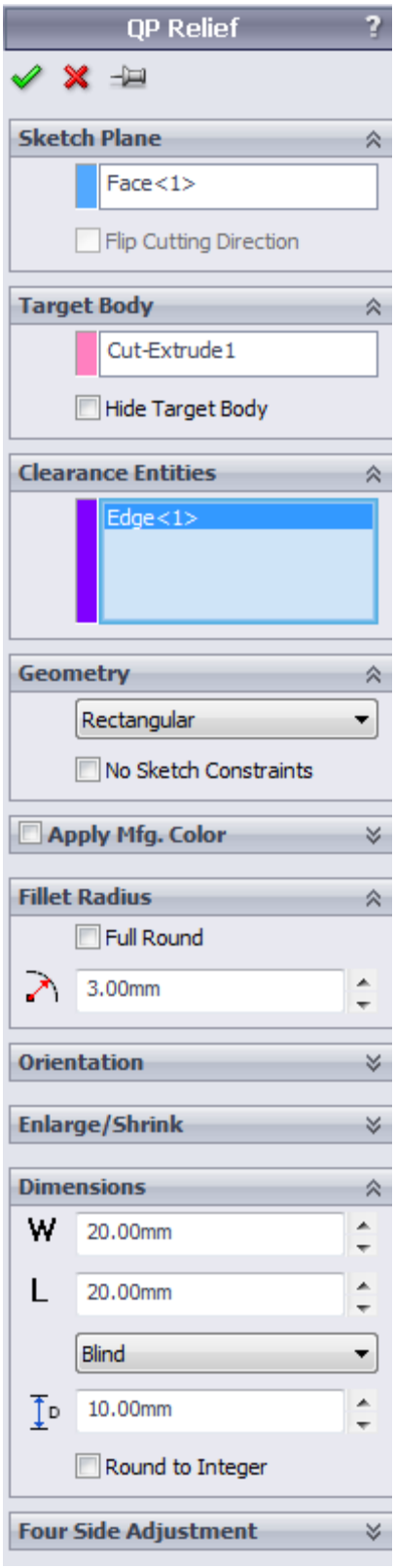
3. Select **Yes** to the warning message to open the folder.



4. Double-click on the link to open the interfering parts then set display to wireframe.

Note: These are Windows shortcuts to the files that have interference in them (the Targets), to keep track and easily locate the files with interferences. It commonly occurs that many parts interfere and each must be edited to design proper clearance.

5. Open the `DS-T5 Strip Layout-01` part file. The interference bodies (surfaces and or solids) are located in

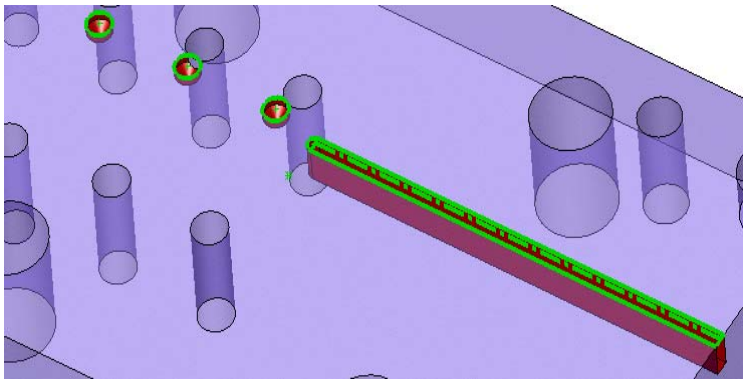


the part as reference and are also listed in FeatureManager design tree with a time stamped folder name. To make clearance for these interferences, use the **Clearance Tools** in the `DieSet Design Toolbar`.

6. In the `T5 Strip Layout-01` assembly, **Clearance Tools** in the `DieSet Design` CommandManager tab.
7. Select the top face of the `DS-T5 Strip Layout-01` part file.
8. Select the interference bodies into the **Clearance entities**.
9. Set the `Geometry` to **Rectangular**.
10. Set the `Fillet Radius` to **3mm**.
11. Set the `W` to **200mm** and `L` to **3mm**, and `Depth` to **10mm**.
12. Click ✔ to finish the command.

After the clearances are created, you can hide the interference bodies by right-clicking them in the FeatureManager design tree and select **Hide**.

Note: If preferred, the interference bodies can also be suppressed or deleted. However, standard relationship dependencies apply since you potentially could have created a sketch relation to edges of the interfering body. This is easily resolved by editing the sketches that have dangling relations after the bodies have been deleted or suppressed. Suppression of these bodies is recommended since they will be needed in the file for revision purposes. Suppression will allow the bodies to remain in the file without affecting the performance of the software.



Adding and Updating File Properties for current Die Set

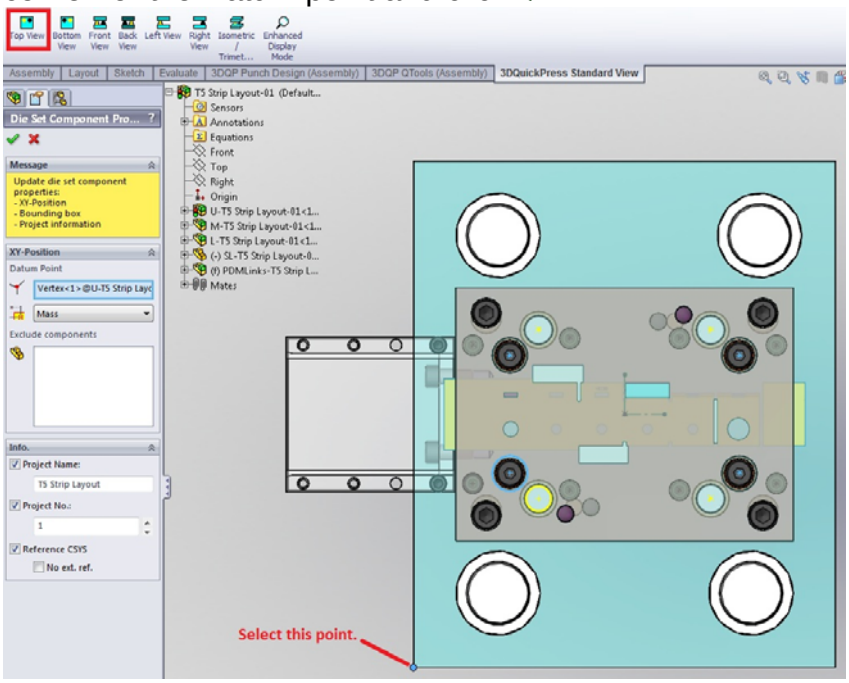
Users often require information about the custom (non-standard) components in a die design. This information is necessary for estimating material and other reasons. Since the design is changing at a rapid pace the user should request these properties be created and updated regularly if changes are occurring. This

can be laborious to do component by component so 3DQuickPress has an automation tool to calculate and manage these file properties.

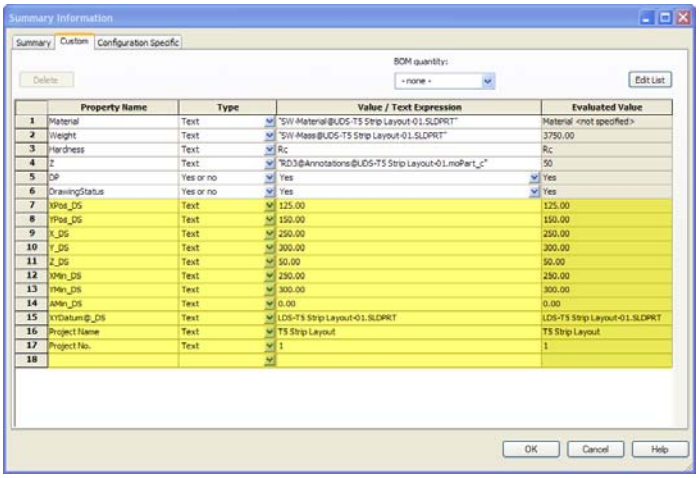
These include:

- Part X & Y, location from top of die from selected point
- X, Y, Z envelopes (horizontal rectangle volume and rotated rectangle volume)
- Part selected for Datum point
- Project Name
- Project Number

1. In the `T5 Strip Layout-01` assembly, click `3DQuickPress, 3D QuickTools, Die set Components Properties Update from the menu.` Change to the top view of the Die and select the lower left corner for the Datum point and click ✔ .

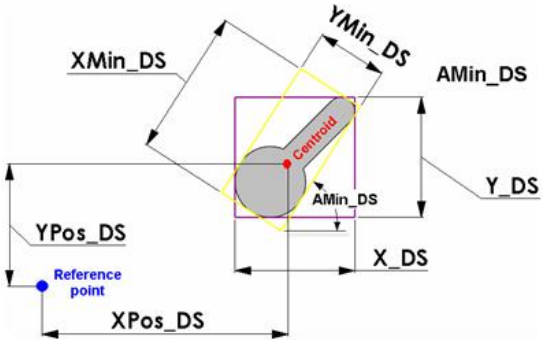


Tip: To view the results of this command, open any non-standard component and select **File**, **Properties**. Select the `Custom` tab and review XMin_DS (X dimension), Ymin_DS (Y dimension), AMin_DS (Rotation Angle) for the minimum Stock Material. Other properties added are highlighted below.



Note that these file properties will not update automatically if a part(s) changes in any way. The Component Properties Update function needs to be run again to update all parts.

- **XPos_DS** – X coordinate of the part centroid of the shown body relative to selected datum
- **YPos_DS** – Y coordinate of the part centroid of the shown body relative to selected datum
- **X_DS** – Width of minimum bounding box parallel to X-axis
- **Y_DS –** Height of minimum bounding box parallel to Y-axis
- **Z_DS** – Length of minimum bounding box parallel to Z-axis
- **XMin_DS** – Width of Actual bounding box
- **YMin_DS** – Length of actual bounding box
- **AMin_DS** – Orientation of actual bounding box relative to X-axis



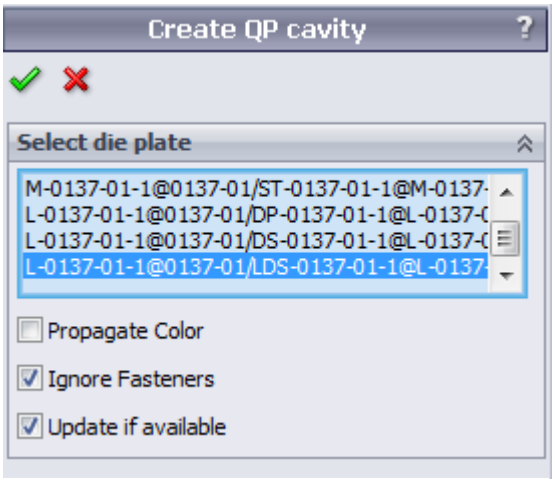
<u>Create die set holes (clearance body automation)</u>

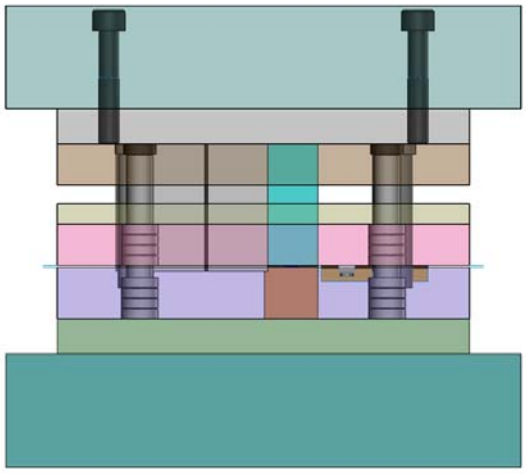
There is option to create holes and clearances components upon insertion of these components. User still have the option to not creating holes and clearances to save time and improve performance. Since the true clearance bodies are not known until the full assembly is created, individually creating the clearances is not a recommended design strategy.

The recommended design strategy is to combine all of the die assemblies and components as fast and easy as possible; then allow 3DQuickPress to determine which components need clearance, which components do not need clearance, and which components need to get clearance externally and internally. This is primarily based on a part's File Properties, file location, and sub-assembly location.

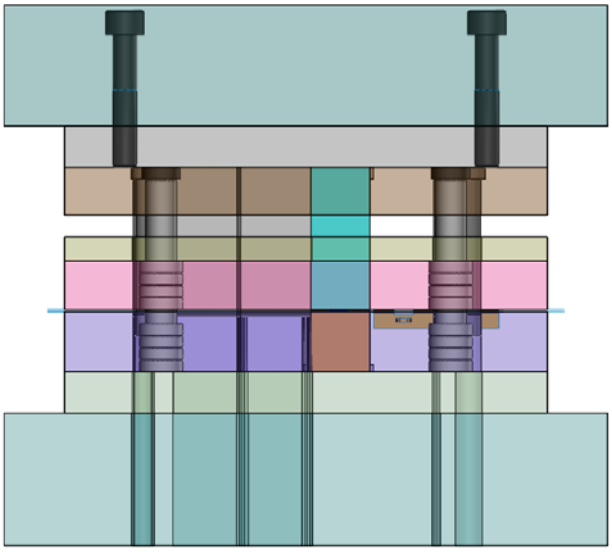
This is a very powerful cavity (a.k.a. subtraction and or cutting) function. It is executed in a batch mode to maximize effectiveness and save time. The function calculates the many scenarios that exist since the combinations and logic must be considered for many different component types in one calculation.

1. Click **Create QP Cavity** from `3DQT Die Set Design` CommandManager tab.





2. Check options **Ignore Fasteners** and **Update if available**.
3. Select all die plates that require the cavity.
4. Click ✔ to finish the command. The die set Holes will be cut automatically.



Note: All options available for using QP cavity are described below:

- **Propagate color** – Transfers the manufacturing information (coloring of individual faces) to all holes that are cut with the Create die set Holes command. The coloring of faces is useful to a MFG group for the automation of CAM programming. The color of the face implies a tolerance and or type of machining for that face or surface. The CAM system must offer a Select By Color option when programming the part for this to be utilized. Note: The setup of the color key, naming, and definitions for the manufacturing functions of 3DQuickPress are covered in the Administrative Setup and Customization chapter later in this training manual.
- **Ignore Fasteners** – Any component that has the IsFastener file property will be ignored in the cavity.
- **Update if available** – If previous cavity feature is found in the die plates, the cavity feature will be updated instead of creating a new cavity feature.

Chapter 6
DETAILING

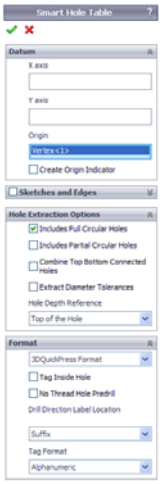
- Smart Hole Table
- Section Line Tools
- Smart Hole Callout
- Edge Annotation
- Advanced Ordinate Dimensioning
- Apply Mfg. Information to Components
- Sort Balloons

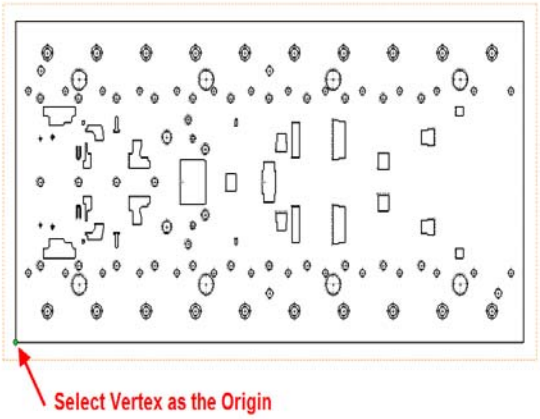
Case Study 6 – 1 : 3DQuickPress Detailing

This case study will cover the detailing functions provided by 3DQuickPress, customization of the manufacturing color database, and setting up components with manufacturing color information.

Create 3DQuickPress Smart Hole Table

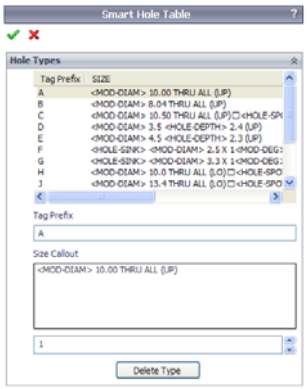
1. **Open** the drawing file `DP-0052-01`. File location: `Desktop\Training\Detailing\Hole Table\DP-0052-01.slddrw`.
2. Select **Smart Hole Table** from the 3DQuickPress Drawing and Machining toolbar.
3. Select a vertex as the origin.



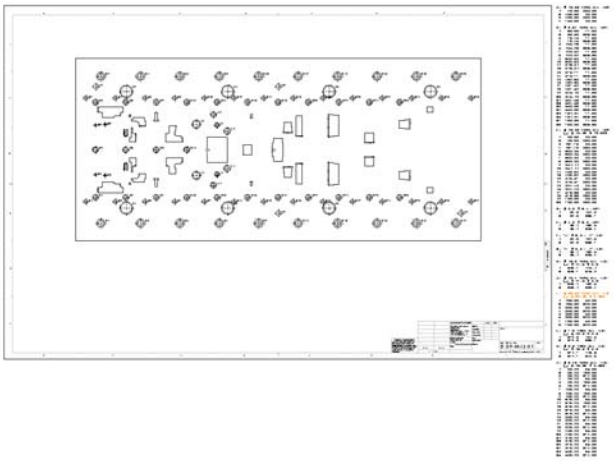


4. Under the Format parameter select **3DQuickPress Format**.
5. Click ✔ to finish the command.

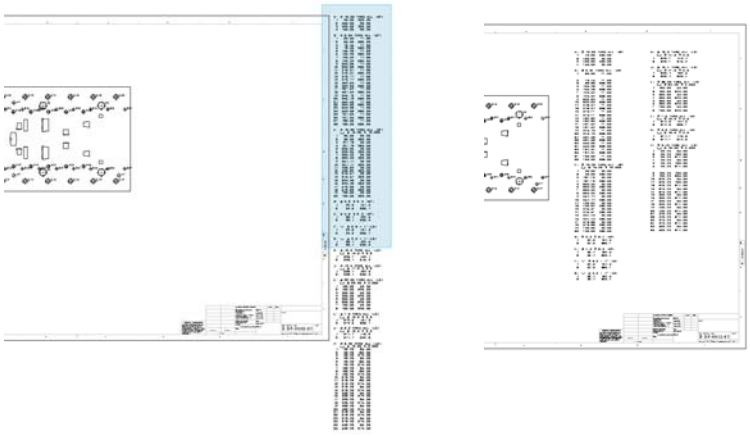
The 3DQuickPress Smart Hole Table will identify and count different types of holes. The hole data is displayed in the feature tree for review and modification before completing the table. Selecting one of the identified hole types in the top selection box under the Hole Type parameter will display the tag prefix, size callout, and starting item number. These parameters can be changed for each hole. Holes can also be removed from the table by selecting the hole from the Hole Type list and pressing the **Delete Type** button.



6. Click ✔ to finish the hole table dialog. A text based hole table will be generated. The tag of each hole will be placed next to the hole in the drawing view.



7. Drag a window around the first half of the hole table and drag it onto the drawing. Repeat this process for the second half of the table.



8. Below shows some of the result of the hole table created.

A: ⌀ 10.00 THRU ALL (UP)

1	40.00	305.00
2	400.00	55.00
3	400.00	305.00
4	760.00	55.00

B: ⌀ 8.04 THRU ALL (UP)

1	20.00	77.80
2	20.00	282.20
3	78.46	77.80
4	78.46	282.20
5	136.92	77.80
6	136.92	282.20
7	195.39	77.80
8	195.39	282.20
9	253.85	77.80
10	253.85	282.20

H: ⌀ 10.0 THRU ALL (LO)
⌴ ⌀ 14.0 ↧ 5.0

1	298.7	143.1
2	298.7	216.9

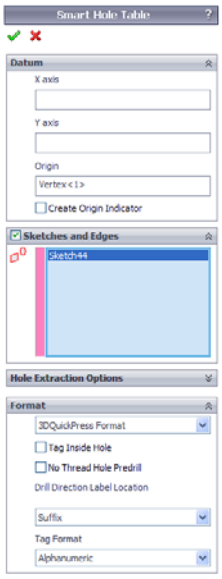
J: ⌀ 13.4 THRU ALL (LO)
⌴ ⌀ 17.0 ↧ 5.0

1	238.1	129.6
2	238.1	230.4

K: ⌀ 22.00 THRU ALL (LO)
⌴ ⌀ 25.50 ↧ 3.000

1	100.00	65.00
2	100.00	295.00
3	300.00	65.00
4	300.00	295.00
5	500.00	65.00
6	500.00	295.00

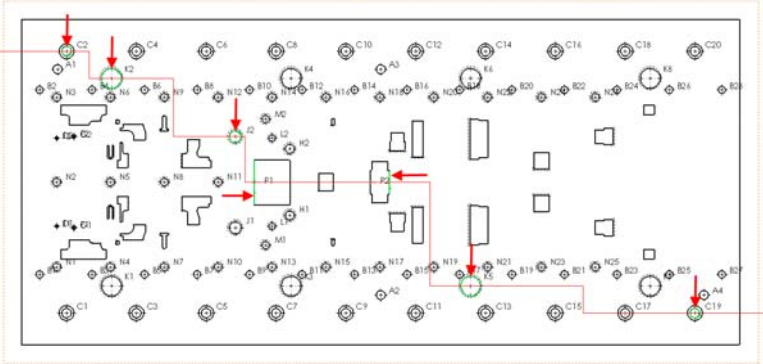
Add EDM sketch holes to the hole table



1. Expand Drawing View 1 in the feature tree.
2. Expand the part DP-0052-01.
3. Right-click Sketch 44 and select Show.
4. Select Smart Hole Table from the 3DQuickPress Drawing and Machining toolbar.
5. Select the lower left corner of the die plate as the origin.
6. Under the Format parameter select 3DQuickPress Format.
7. Enable the Sketches and Edges option.
8. Select the EDM hole sketch “Sketch 44” into the Sketches and Edges area.
9. Click ✔ to proceed.
10. Change the Tag Prefix to P.
11. Click ✔ to finish the command.
12. Move the new table onto the drawing.
13. Save the drawing.

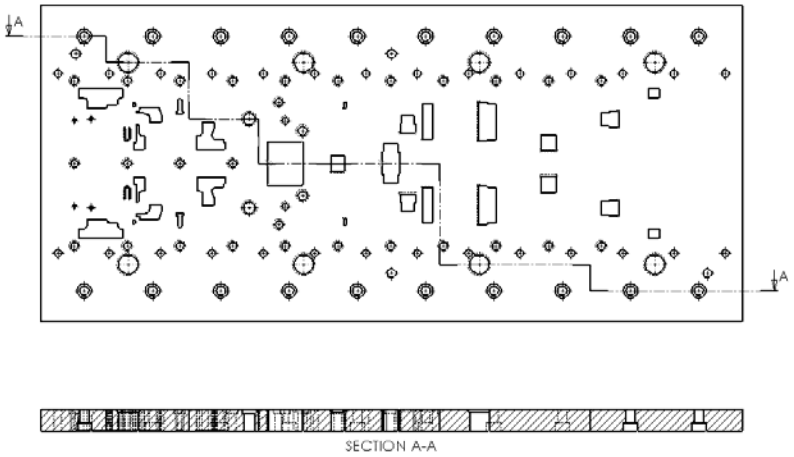
Create 3DQuickPress Section Line

1. Select **3DQP Section Line** from the 3DQuickPress Drawing and Machining toolbar.
2. Select entities to be sectioned through. A red line will be show for the preview of the section line.



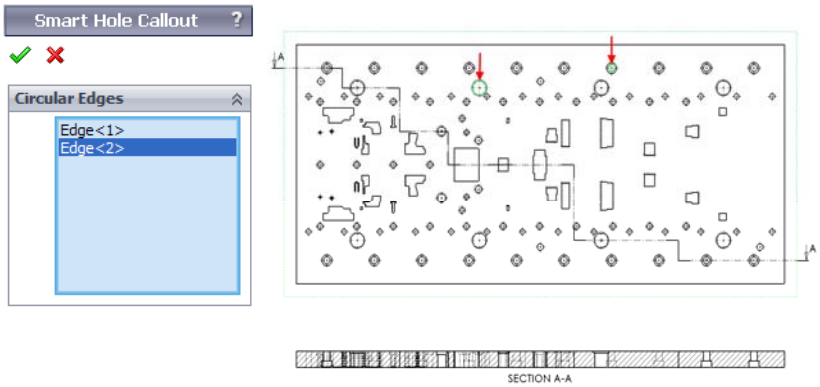
3. Click ✔ to finish the command.
4. Click and drag vertical edges of this sketch to make any adjustments.
5. Select the section line, and click **Section View** from the view layout toolbar.

6. Place the section view below the current drawing view.

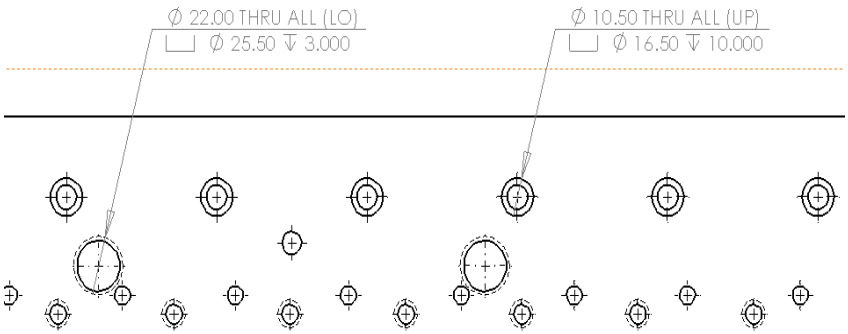


Create 3DQuickPress Smart Hole Callout

7. Select **Smart Hole Callout** from the 3DQuickPress Drawing and Machining toolbar.
8. Select the two edges indicated below.

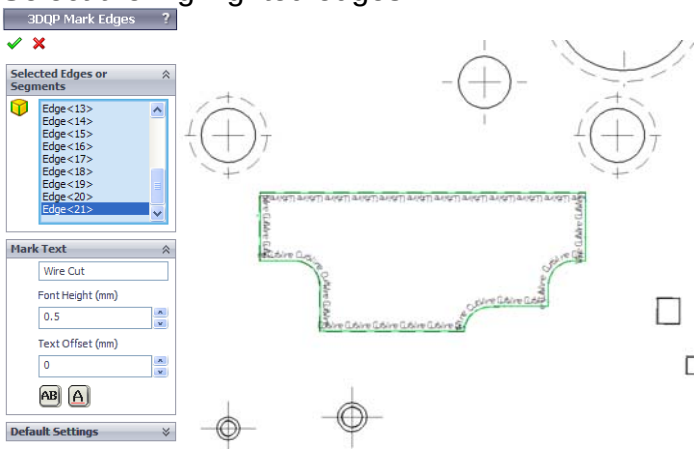


9. Click ✔ to finish the command. The hole callout with the size and depth information will be created automatically.



Create Edge Annotation

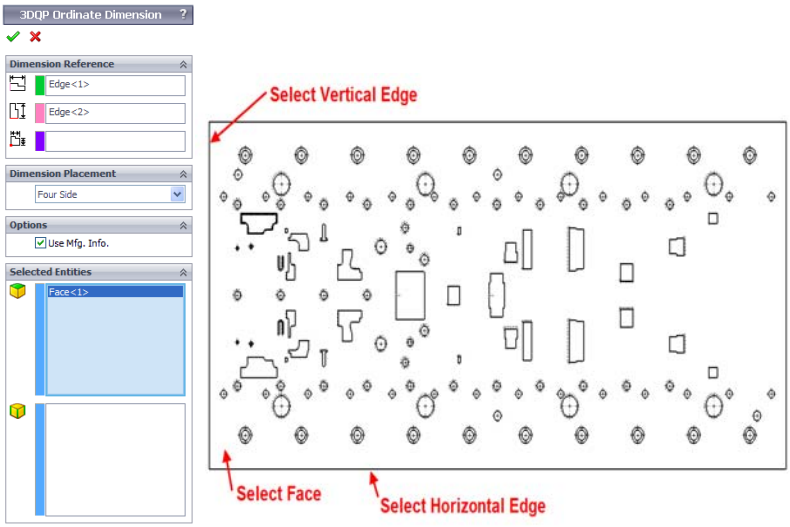
1. Select **3DQP Mark Edge** from the 3DQuickPress Drawing and Machining toolbar.
2. Select the highlighted edges.



3. Enter **Wire Cut** for Text, **0.5** for Font Height, and **0** for Text Offset.
4. Click ✔ to finish the command.

Create 3DQP Ordinate Dimension

1. Select **3DQP Ordinate Dimension** from the 3DQuickPress Drawing and Machining toolbar.
2. Select the horizontal and vertical side edges as the references.
3. Select the top face in the drawing view as selected entities.



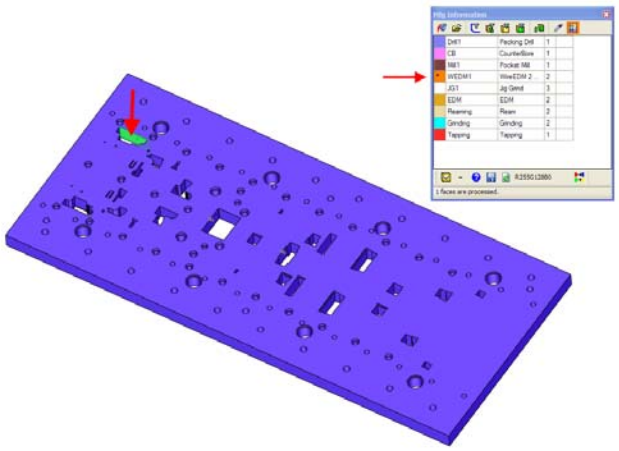
4. Select **Four Side** for Dimension Placement**.**
5. Enable Use Mfg. Info.

6. Click ✔ to finish the command.

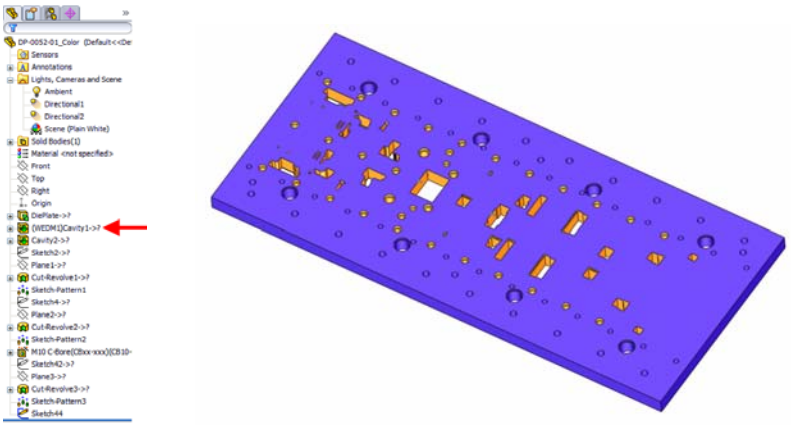
Apply Mfg. Color Info. to components and drawings

Applying manufacturing color is a pre-processing function to add manufacturing information to geometry by applying specific colors to model faces. Setting up the Mfg Colors is covered in Chapter 11 Setup and Administration of 3DQuickPress.

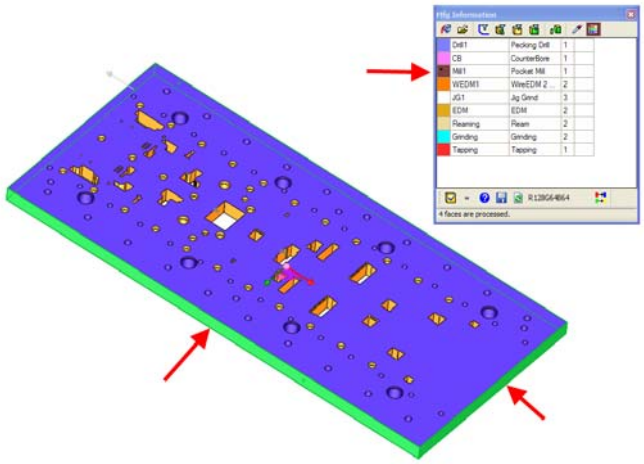
1. **Open** the drawing `DP-0052-01_Color.slddrw`. File location: `Desktop\Training\Detailing\Hole Table\`
2. Right-click **Drawing View 1** and select **Open Part**.
3. Click **Mfg. Color Tool** from the 3DQuickPress Misc. Tools toolbar.
4. Select **WEDM1** as the manufacturing information for the cavity cut feature.
5. Select the indicated face below to select the cavity feature. Click **Set color for selected feature** to assign color for the cavity feature.



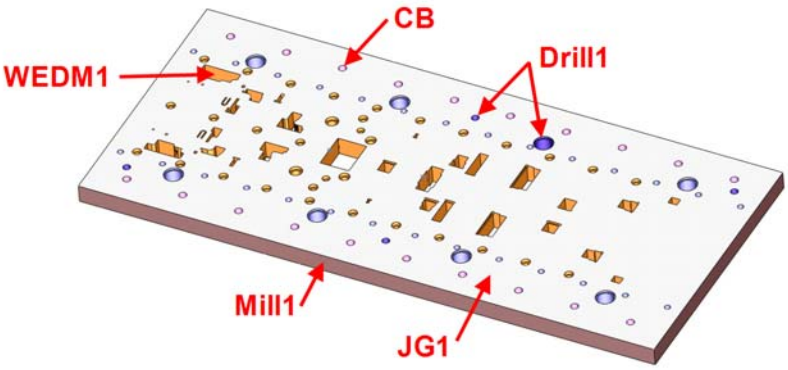
6. The mfg. code “WEDM1” will be added to the front of the cavity feature and all faces of the cavity feature will be colored. This may require you to rebuild the part.



7. Hold control and select the 4 outside faces of the plate, and select **Mill1** as the manufacturing information.



8. Click **Set color for selected face** to assign color for side faces.



9. Click **Assign Color to Face** to assign existing display color to face.

   Continue applying manufacturing methods to the part as indicated above.

10. Select **Sketch44** on feature tree, and select **Drill1** as the Mfg. information. Click **Set color for selected sketch** to assign color for the sketch.

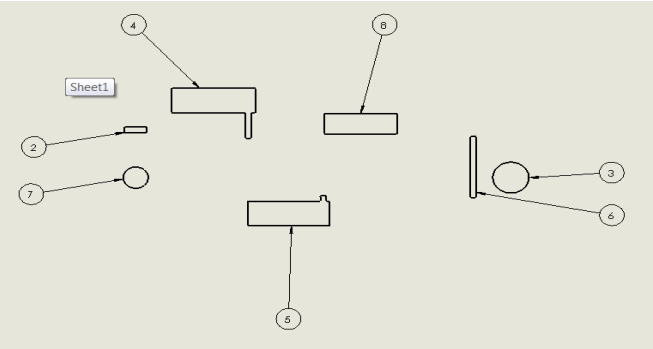
11. **Save and close** the part.

Sort BOM Balloons for assembly drawing

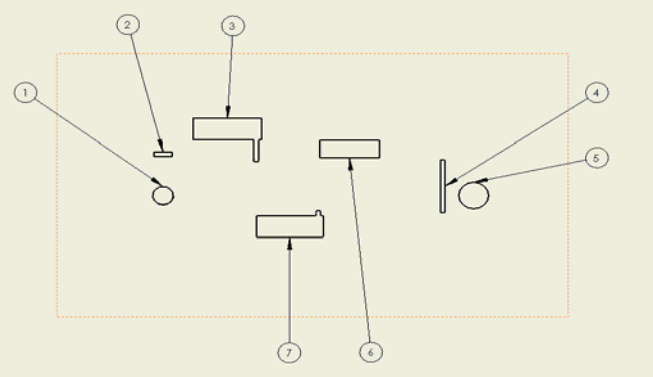
The Sort Balloons command is used to modify BOM balloons that SolidWorks creates. It is an editing function to sort the balloon in a logical order for ease of blue print reading. The balloon number is a new file property which is created and substituted once the sort command is utilized. The new file property is added to the part files that are unique to the current project.

To sort SolidWorks BOM balloons:

1. To get started **Open** the drawing `U-T5 Strip Layout-01.SLDDRW` File location: `Desktop\Training\Detailing\Balloons\`
2. Pre-select all balloons with **Notes** and **Balloons SolidWorks Filter** turned on.
3. Click **3DQuickPress**, **Drawing & Machine**, **Sort Balloon**.
4. Select the BOM template, browse to `Program Files\3DQuickPress\3DQP Templates\BOM Template &` Click OK
5. Click Yes to create and place the BOM



6. The balloons are now in a Clockwise sequence after Sort is executed.



Note: The standard components are identified as “S#”; S for Standard, then the number callout. Since these are shared files amongst many projects 3DQuickPress does not add a unique file property to the part itself. Standard components are tracked with a text file database per project noting the qty and the description of these components. The definition of a standard component is any component with a FILE PATH located outside the current working directory and or Project Folder.

7. **Save** & **Close** file.

Chapter 7
DESIGN CHANGE

- Changing Operations in a Strip Layout
- My Blank Method for easy 2D Changes
- Modifying the Original 3D Part and Updating 3DQuickPress data

CHANGING OPERATIONS IN THE STRIP LAYOUT

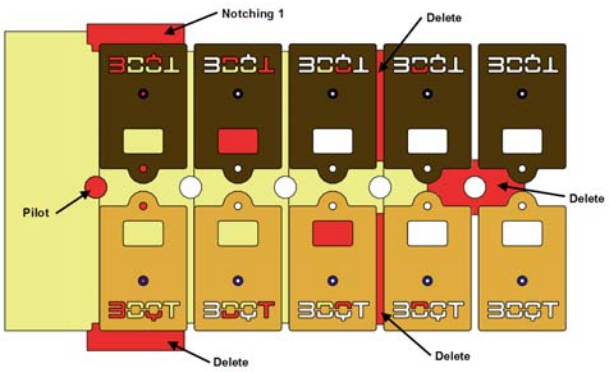
Engineering changes happen in every aspect of die design. The ability to easily and rapidly make changes to a tool design is a key capability of the SolidWorks/3DQuicktools software. Whether you are changing the original 3D part model or if you are modifying the die set drawings, the associative nature of the software was designed with the change process in mind. In this chapter, you will study several different case studies to illustrate these techniques. There are many ways to make changes to designs and with that in mind the user should understand that there are different approaches depending on the status of the release process. Another consideration to making changes is the time available. Some changes could be made at the product design stage or the part level. Other changes may be more practical if they are made at the flat blank tooling level, while others can be made at the detail drawing level. 3DQuickPress offers many tools and techniques to approach change with the design and manufacturing processes.

Case Study 7 – 1 : Blanking Punch Process

In this scenario, an engineering change has been issued to make a change to the type of operations to develop the strip layout. Hiding, deleting, reordering, and creating new operations are covered in this case study.

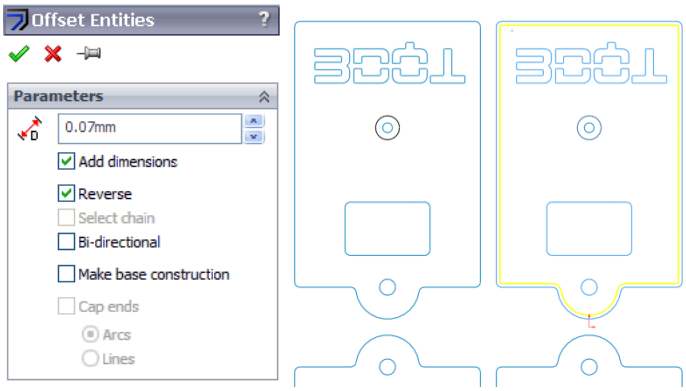
Creating the new punching operation

1. Open strip layout part named `T5-1 Strip Layout.sldprt` location: `Desktop\Training\Strip Creation\`
2. Click **File**, **Save as** and save the part as `Blankoff`.
3. Delete all notching punches except `Notching 1` and `Split Notching 1`. To delete a notching feature, right-click the notching feature in the feature tree and select **Delete**.

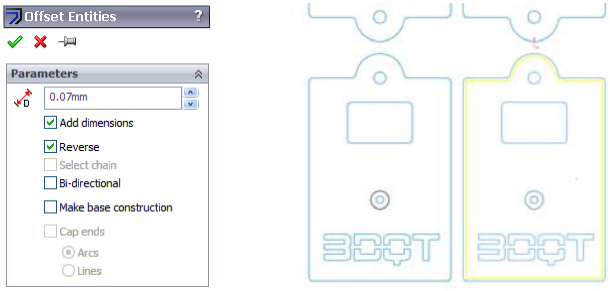


4. Right-click `Notching 1` from Strip Layout Design Tree and click **Edit Sketch.**

5. **Delete** all the sketch entities.
6. Establish cutting clearance for the blanking punch. Select the upper blank and **Offset Sketch Entities** a distance of **0.07mm** to the inside.

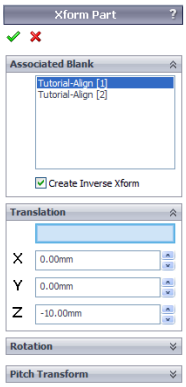


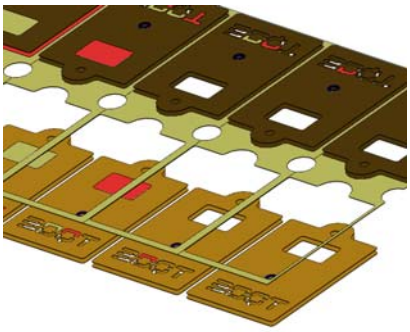
7. Click ✔ to finish.
8. **Exit Sketch**.
9. Right-click `Unprocessed Features` from the Strip Layout Design Tree and select **Create Notching Feature.**
10. Create a blanking punch for the lower blank. Select the lower blank and **Offset Sketch Entities** a distance of **0.07mm** to the inside.



11. Click ✔ to finish offset command. **Exit Sketch**.
12. Rename `Notching 1` to **Blank1** and `Notching 2` to **Blank2**.
13. Right-click `Blank 1`, select **Apply To**, set to station **5**. Repeat for `Blank 2`.

14. Right-click `Unprocessed Features` from the Strip Layout Design Tree and select **X-Form Part**. Select the `Tutorial-Align [1]` component in the Associated Blank window of the feature tree and set the Z distance to **-10mm** under Translation.





15. Click ✔ to finish the command.

16. Right-click `Unprocessed Features` from the Strip Layout Design Tree and select **X-Form Part**. Select the `Tutorial-Align [2]` component in the Associated Blank window of the feature tree and set the Z distance to **-10mm** under Translation.

17. Click ✔ to finish the command.

18. Right-click `Xform Part 1`, select **Apply To**, set the station to **5**. Repeat for `Xform Part 2`.

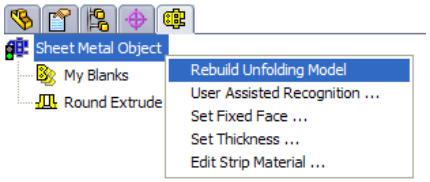
19. **Save** the part.

Changing the original solid part geometry

Part designs change constantly. Therefore, 3DQuickPress takes advantage of the parametric nature of SolidWorks to allow these changes to update the strip layouts automatically.

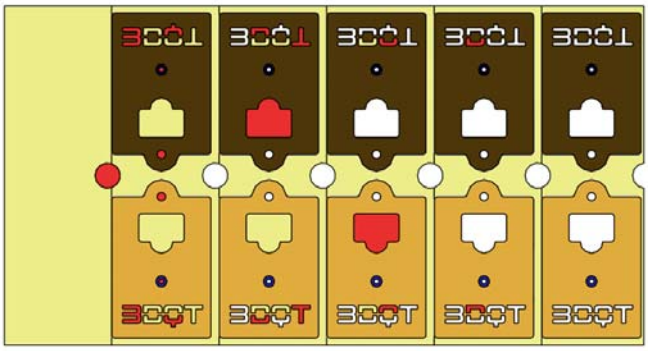
1. Open and modify the original unfold part. Sketch an **8mm** diameter circle on the midpoint of the rectangular opening in the part. **Extrude cut**, **Through All**.

2. Right-click `Sheet Metal Object` from the 3DQuickPress feature tree, and select **Rebuild Unfolding Model**.



3. Switch to Strip Layout window. Right-click on `Strip Layout` at the top of the 3DQuickPress feature tree and select **Rebuild StripLayout Model**.

   Activate any piercing geometry that may have changed.



4. **Save** the part.

MY BLANK METHOD

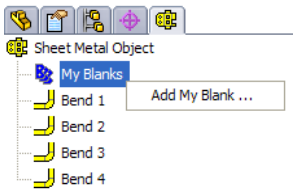
The 3DQuickPress My Blank option is used for two reasons: first it eliminates the task of fixing bad models that have been imported, and second it allows for quick editing of part data to aid design for manufacturing. When making a design change using the My Blank option, changes can be made to the My Blank Layout sketch. If changes must be made to the part model, then those changes also need to be made to the My Blank sketch for the updates to show in the Strip Layout, Punch Design, or die set Design.

Case Study 7 – 2 : My Blank Process

Creating a My Blank

1. Open the part file **Cylindrical fix face.** File Location: `Desktop\Training\My Blank\ Cylindrical fix face.sldprt.`

2. Right-click `My Blanks` from the 3DQuickPress feature tree and select **Add My Blank**. This creates a new blank sketch to modify unfolding results.



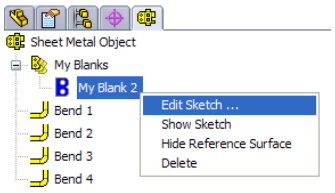
3. **Hide** the 3DQuickPress model by toggling off **Show Unfolding Model** from the `3DQP Unfolding` toolbar.

Modifying the My Blank

In this example two areas of the My Blank will be modified.

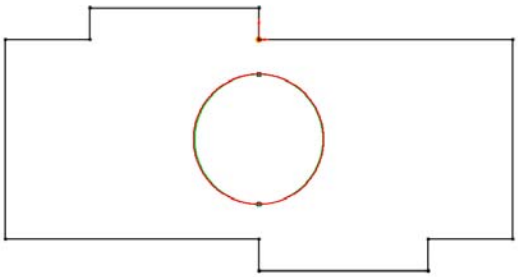
- First, the inside hole was created when the part was in the folded position. For manufacturing purposes the hole will be pierced in the flat blank layout.
- Second, fillets will be added to the corners of the tabs.

4. Right-click `My Blank 2` and select **EDIT SKETCH**.

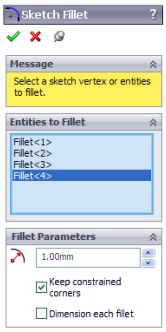


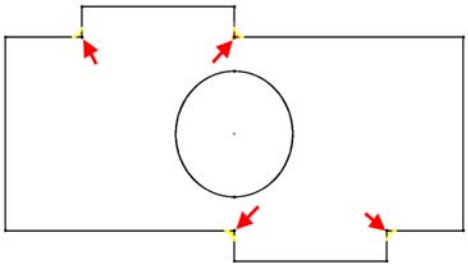
5. Hold control and select both segments of the circle.

6. Select **Bounding Box** from the 3DQP QTools toolbar, QP Sketch Tools Menu. Under Geometry select **Circle**, and under Enlarge\Shrink select By **Offset Distance** and set the value to **0.00mm**. Click ✔ to finish the command.

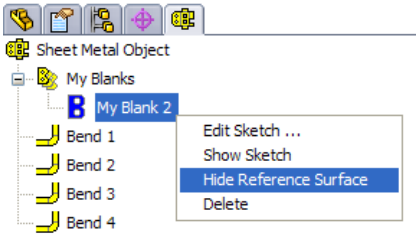


7. Select the original spline segments and check the box **For Construction** in the feature tree.
8. Select the new sketch circle and add a Fixed relationship to lock it in place.
9. Add a **Sketch Fillet** with radius of **1mm** to the inside corners of the blank. The addition of the fillet will update the punches in the Punch Design.





10. Click ✔ to finish the command.
11. **Close** the sketch from the confirmation corner.
12. **Show** the 3DQuickPress model by selecting **Show Unfolded Model** from the 3DQP Unfolding toolbar to see the updates made to the My Blank surface.
13. Right-click My Blank 2 from the 3DQuickPress feature tree and select **Hide Reference Surface**.



14. **Save** and **close** the part.

MODIFYING THE ORIGINAL 3D PART AND UPDATING 3DQUICKPRESS DATA

Making changes to the original 3D part is a common task. In this case study, the unfolded part, strip layout design, and punch design assembly are already completed. A design change from the product designers calls for the tooling to be updated to reflect these part changes. Using sketching tools, the user will make some modifications to the original solid model part. Following the change, the 3DQuickPress data needs to be updated for the unfolded part, strip layout part and the punch design assemblies.

Case Study 7 – 3 : Design Changes

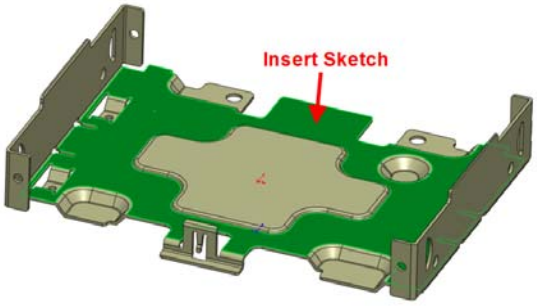
Making changes to the 3D product design model

1. Open punch design assembly `0108-Front_Plate`. File location: `Desktop\Training\Design Changes\0108-Front_Plate\`

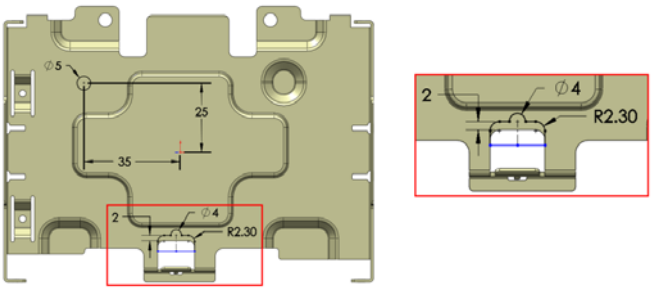
2. Switch to `Front_Plate.SLDPRT`

3. Click **Toggle Unfolding Display** from the 3DQP Unfolding toolbar to display the solid model of the part.

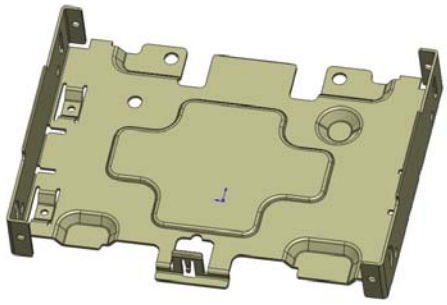
   Right-click on the face below and Insert Sketch.



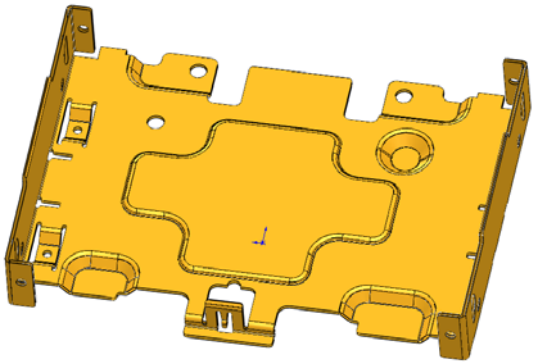
4. Complete the sketch as shown below.



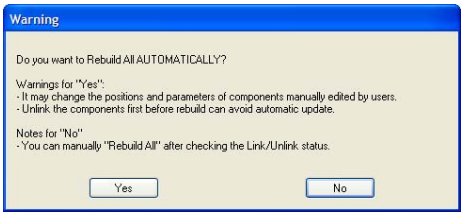
5. Click **Insert, Cut, Extrude** and set the end condition to **Through All.** Click ✔ to finish the command.



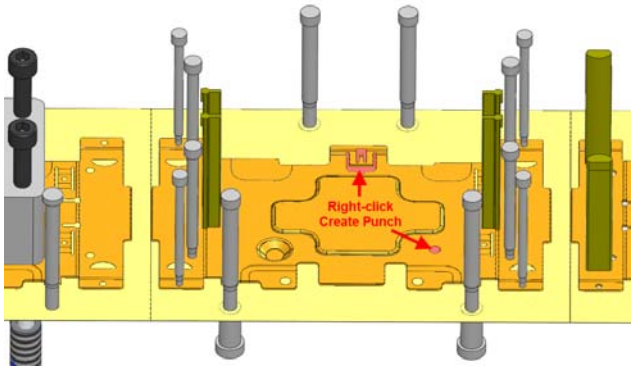
6. Click **Toggle Unfolding Display** from the 3DQP Unfolding toolbar to display the sheet metal part.
7. Right-click **Sheet Metal Object** from the 3DQuickPress feature tree and select **Rebuild Unfolded Model**.



8. Switch to `Front_Plate Strip Layout.SLDPRT`.
9. Right click **Strip Layout** from the 3DQuickPress feature tree and select **Rebuild Strip Layout Model**.
10. **Drag and drop** `Piercing 27` and `Piercing 28` from Unprocessed Features to **Station 2** to activate these features.
11. Switch to `0108-Front_Plate.SLDASM`.
12. Click **Yes** when asked to rebuild the punch design.



13. Right-click `0108-Piercing 24.1<1>` in the FeatureManager design tree and select **Hide**, then right-click the red cutting punch surface for Piercing 27 and select **Create Punch**.



14. Right-click the red cutting punch surface for `Piercing 28` and select **Create Punch**.
15. **Save** and **close** the assembly.

Chapter 8
CHANGEOVER TOOLING

- Changeover Tooling Manager (COT)
- Add Multiple Strips in Punch Design
- Share Multiple Operations within the Design

CHANGEOVER TOOLING

Changeover tooling is a common practice in the industry to produce similar parts using the same tooling through proper tool setup. 3DQuickPress simplifies the design of changeover tooling and produces tooling setups automatically by adding die components names to the PRL's or on the fly in a current design by using an XML file to customize.

What is a XML Notepad file

XML (Extensible Markup Language) data file that uses tags to define objects and object attributes, formatted much like an HTML document, but uses custom tags to define objects and the data within each object; can be thought of as a text-based database.

The software can be downloaded to a machine by following the path- http://www.microsoft.com/downloads/en/details.aspx?FamilyID=72d6aa49-787d-4118-ba5f-4f30fe913628&displaylang=en

Customize the XML Notepad file

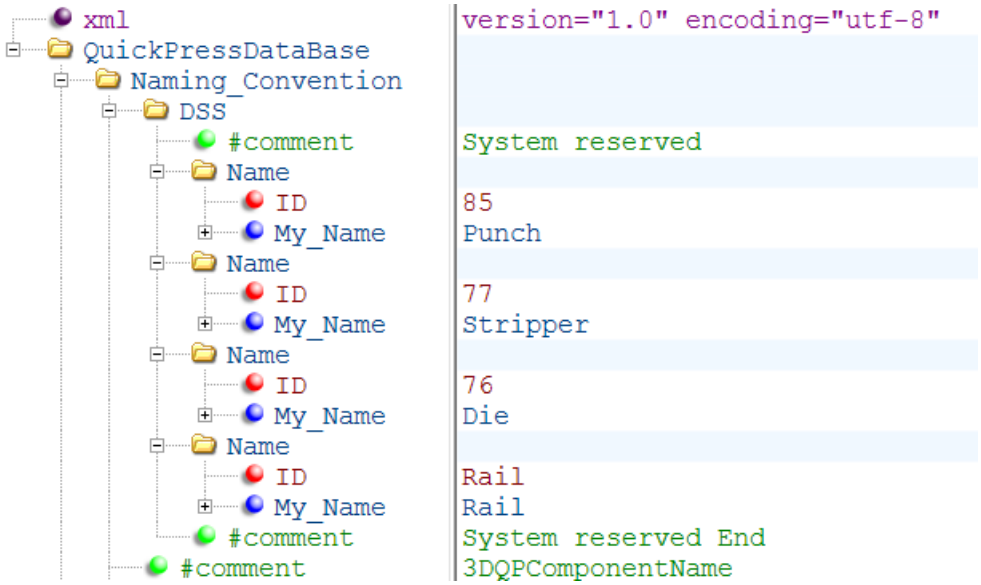
To add new or change process names XMT NotePad

Die set structure and component naming is controlled by an xml file. The XML file has 4 sections set up **Naming Convention**, **Process, Part and Assembly document type.** For each section, 3DQP reserved some IDs which must not be changed.

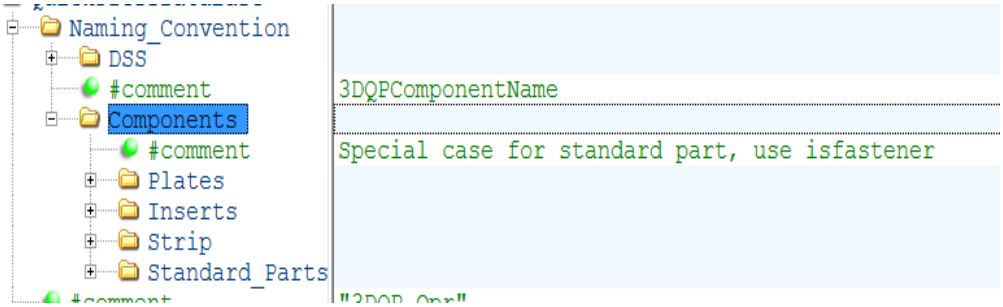
Section	Restricted IDs	File Properties
Naming Convention\DSS	85, 77, 76	3DQPComponentName
Naming Convention\Components	All IDs are free to change	3DQPComponentName
Process	All IDs are free to change	3DQP Operation
Part Type	DieSetStructure.SLDPRT<4>, 22	3DQP DocType
Assembly Type	33, 4, 85U, 77M, 76L, 3, 85, 77, 76, 12, 11, 14	3DQP DocType

Launching XML NotePad file for Customizing

1. Launch XML NotePad and open 3DQP.XML. File location: `Program files\3DQP\Database\3DQP.xml`
2. Expand DSS folder

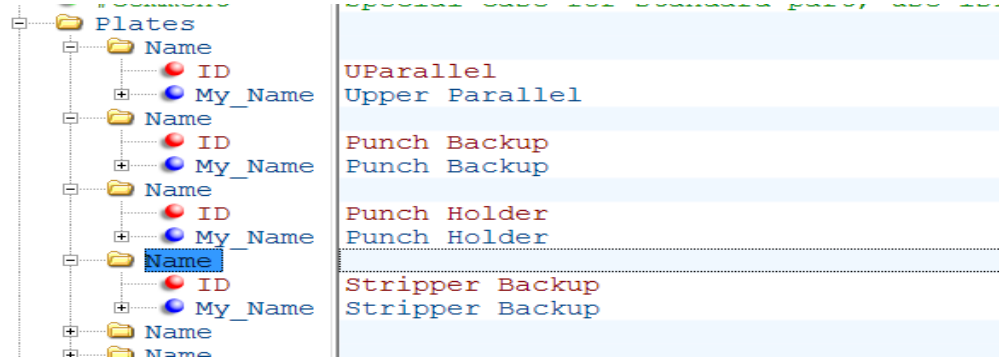


3. The files in the DSS are restricted ID's.
4. Expand the Component folder.



- Inside of the Components folder you will find 4 sub-folders called **Plates, Inserts, Strip, and Standard parts.**

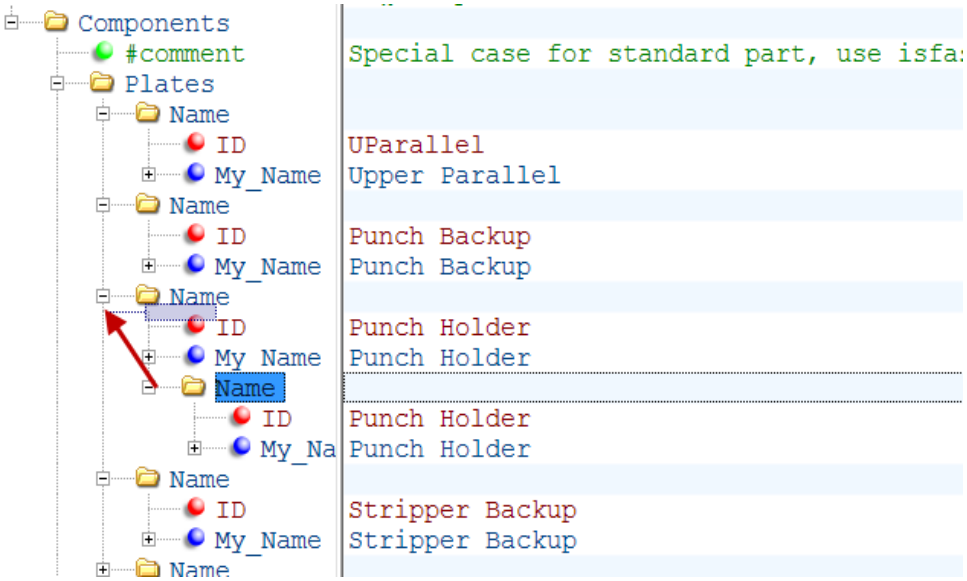
5. Expand the plates folder



The this folder you will find descriptions of plates that are in the die design, any or all of these names can be changed to meet your design standards, but keep in mind that current PRL's may already been set with the current names. The plate group is already setup top to bottom similar to a cross-section of a die design, so when adding a new plate place it in its correct location.

- **<u>Add a New Plate to the Group.</u>**
-

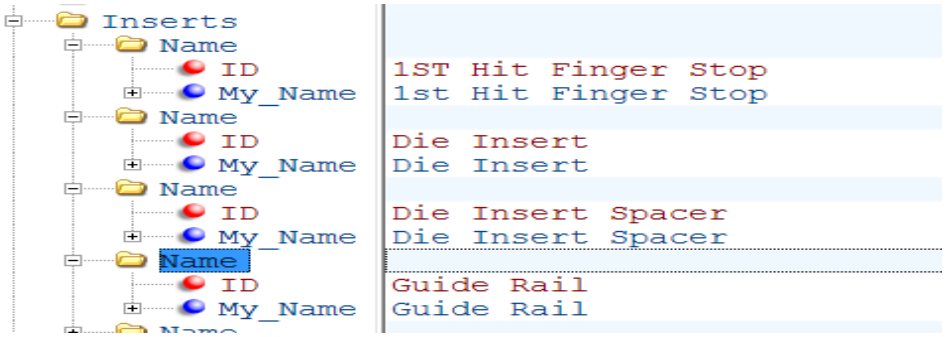
6. Right click on the folder above or below the location you would like to add a plate and select **Copy.**
7. Right click again at the same location and **Paste**.
8. Once you have pasted the new folder at the same location, drag it to the correct tree location and drop it as shown by the arrow below.



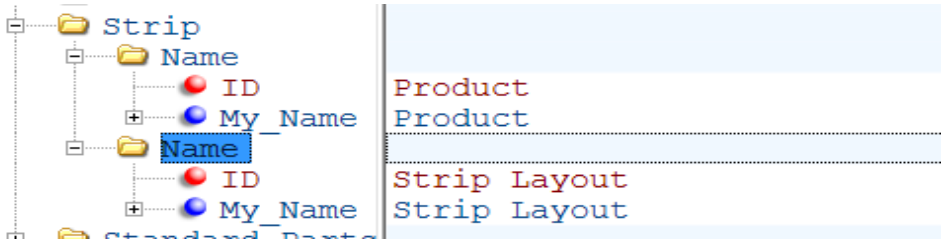
-
-

9. Then rename both the ID and the My_Name comments as needed.
10. The groups called "Plates" are set for all the plates inserting into the **Die Set Design.**
11. Inserts are all Items inserted into the **Punch Design** with the **Guide Rails being the exception.**

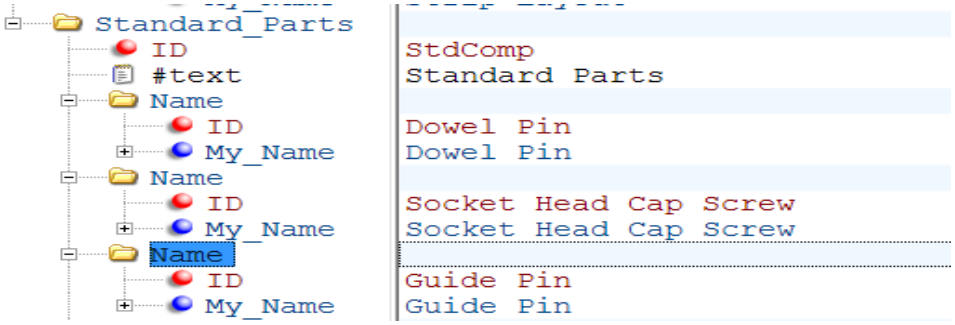
-



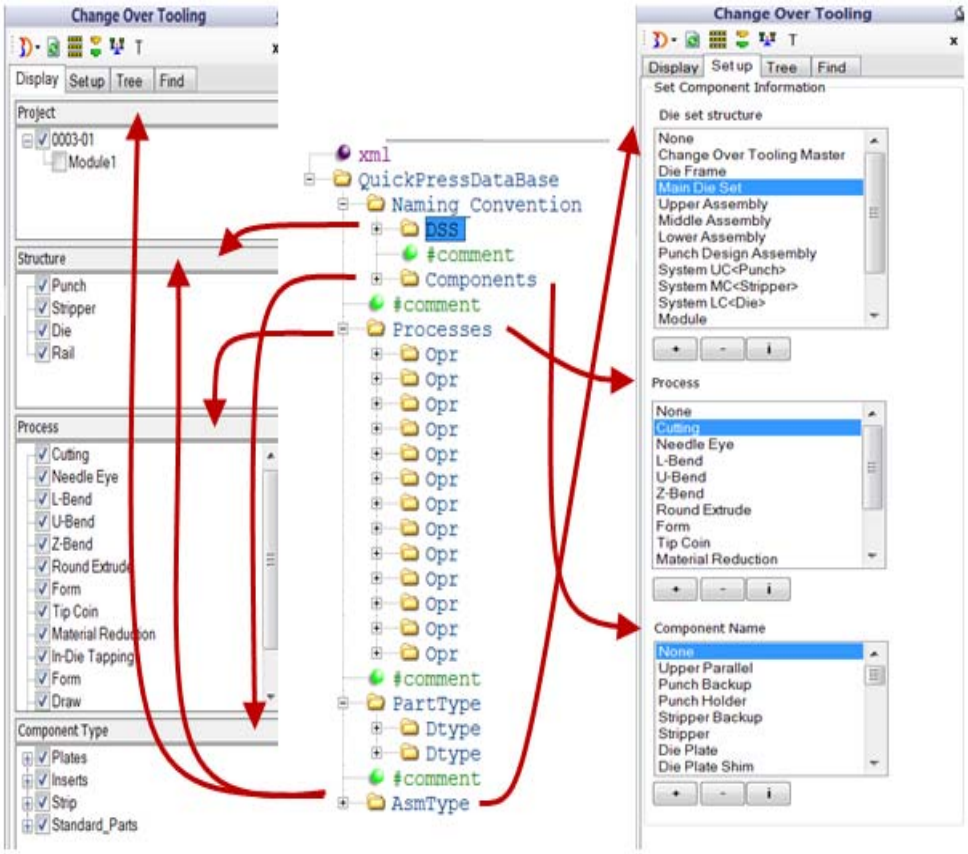
12. The Strip folder will have the **Strip Layout** and the **Unfolded Model.**

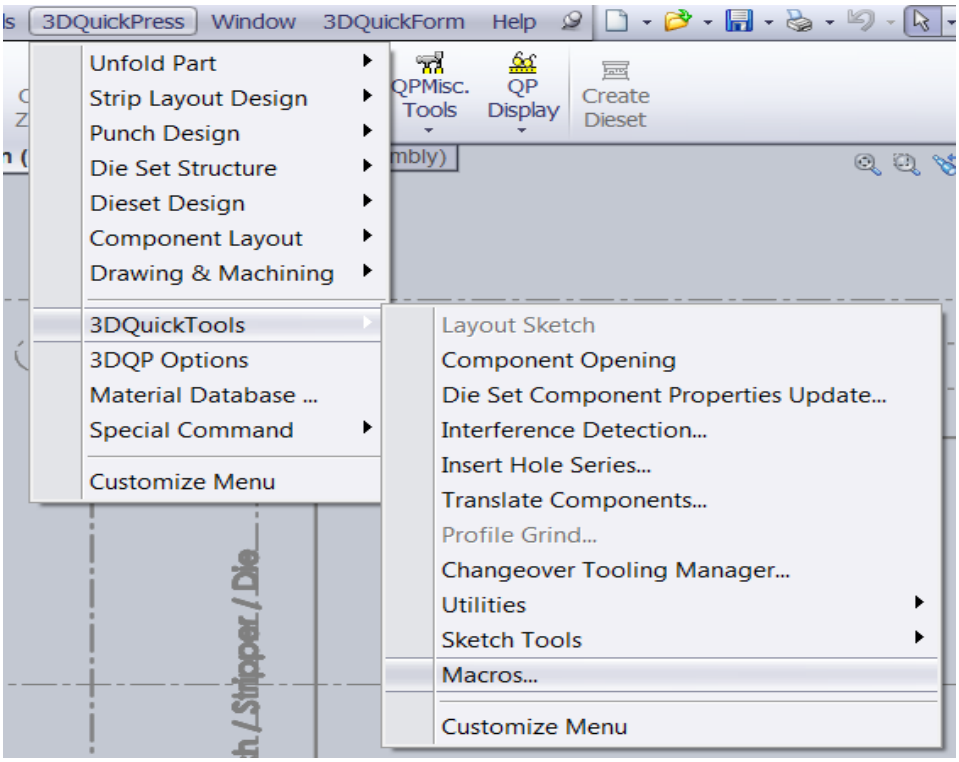


13. **Standard Components** are all features that are inserted from the standards folder, SHCS, Guide Pins, ect.

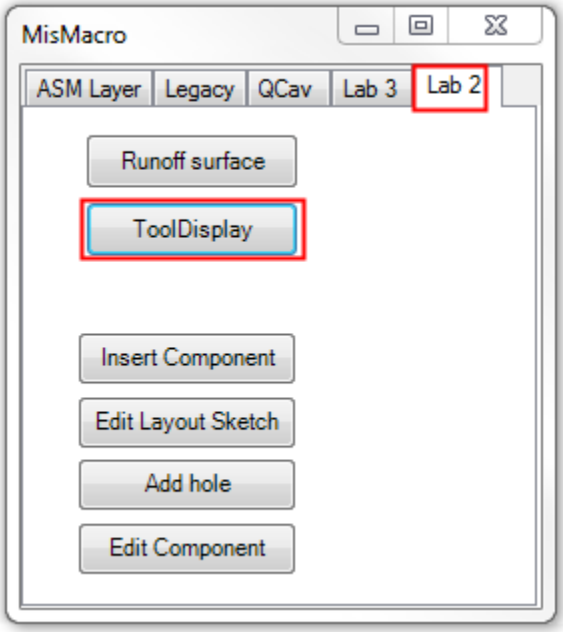


14. Relationship between **3DQP.xml** and the **COT Manager**

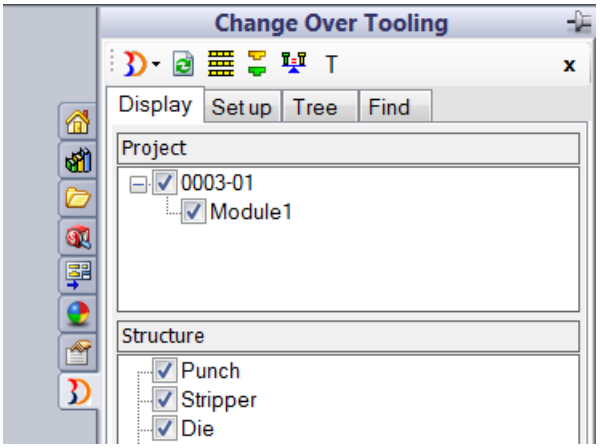


- **<u>Launching the COT Manager</u>**
  - 1. Change over Tooling Manager COT is launched by selecting the menu 3DQPress\3DQTools\Macro.
  - 

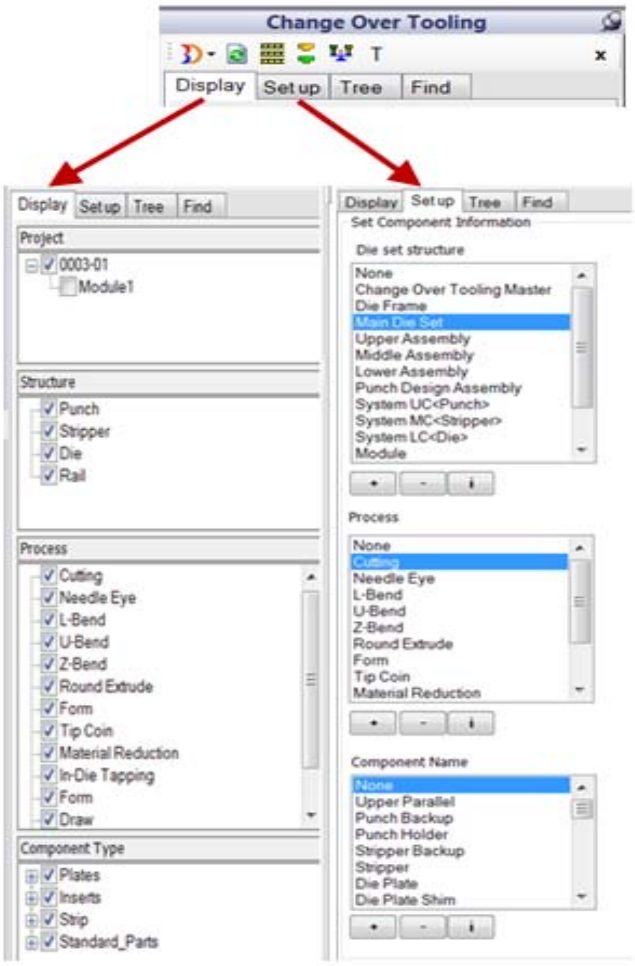
2. Select Lab 2 Tool Display Tab.



3. The COT tab will automatically create on the SolidWorks Task Manager Pane.

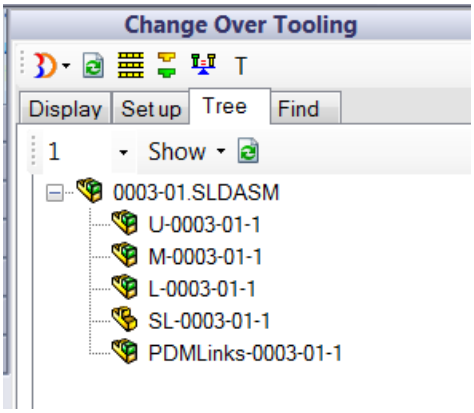


Main User Interface

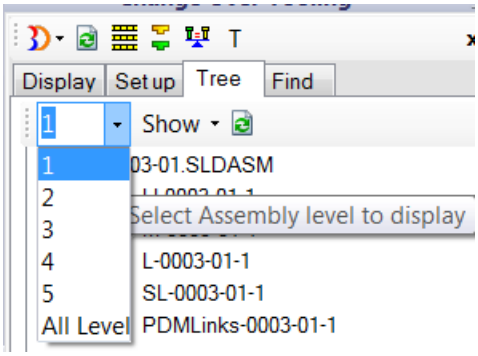


The Simplified Tree Manager

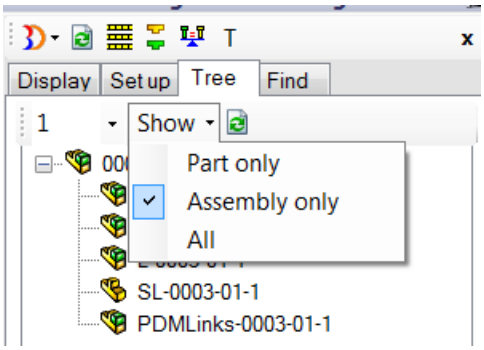
1. By selecting the tree manager Icon T this will allow the user to control how many levels you will see in the SolidWorks Feature Manager Tree from the COT Manager.



2. Selecting the 1st pull down tab allows you to decide how many **Levels Deep** you will see in the feature manager tree.



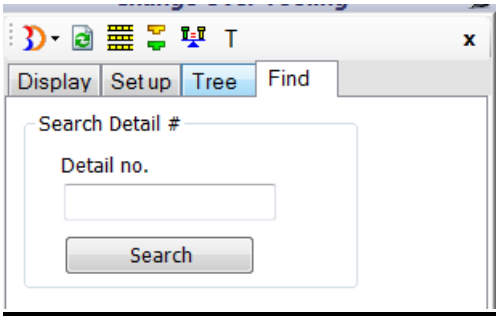
3. Selecting the "**Show**" pull down this will allow you to choose to see Assemblies, Part or all files.



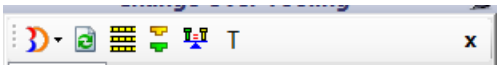
4. The **Refresh Button** simply refreshes the tree displayed.

Search for Detail Number

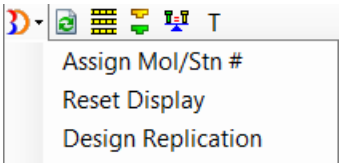
1. The **Find Tab** allows the user to search for a specific detail after a design has been completed, the detail number will be entered into the box, by selecting search, 3DQPress will Isolate the detail.



Utilities Tool Bar



1. The **3DQP** Tools pull down tab.



2. **Assign** – Setting details as Module Number or Station Numbers.
3. **Reset Display** – Show all top level components in the assembly except those suppresses in the graphics window.
4. **Design Replication** – Reuse current or older design for a **New Strip Layout** or **New Punch Design Assembly**.

- **<u>3 different processes are handled</u>**

a) New Strip Layout for an old or new Punch Design.
b) New Strip Layout for an old or new Die Set Design.
c) New Punch design for an old or new Die Set Design.

Refresh Tab - After a new property has been assigned to the assembly, by pressing the Refresh Tab the tree in the COT Manager will update.

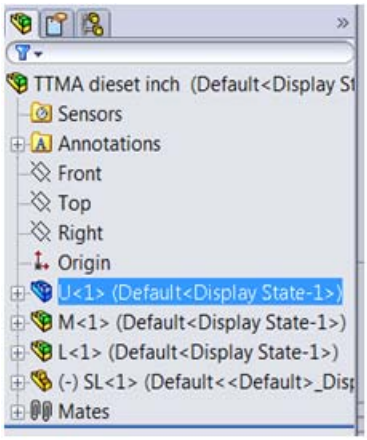
Explode Tab – After an assembly is made from multiple designs then the explode tab can be used to separate the designs for viewing.

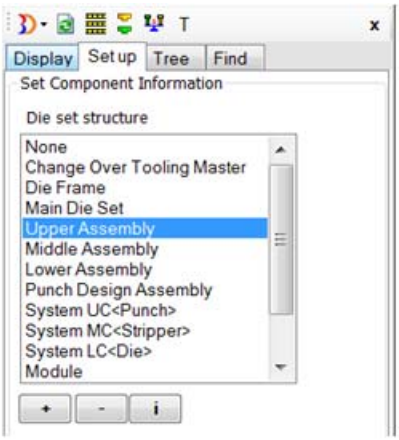
Share Component Tab – This will allow you to share a component from current design to a new design.

Compare Tap – Allows you to compare components to each other to see if they are identical.

+ (Add) – (Subtract) i (Check) Will adding the properties to the **Die Set Structure**, a **Process**, or a **Component Name**. By selecting the SolidWorks part or assembly then selecting one of these Icons you can **Add** a property, **Subtract** a property or **Check** to see what property has already been added if any.

PRL Customization Check List

- **Die Set PRL**
  - All die set PRL's must have the file property "**3DQP DocType**" assigned to **U,M and L** assemblies with values **85U, 77M and 76L** respectively.
  - Note: In order to use the **Copy Die Set** function of the COT manager the above properties must be set, otherwise the U, M and L will retain the old name. If this happens the user can use the 3DQP utilities to separate them and continue to design, although the die set template should be updated before it is used again.
  - 

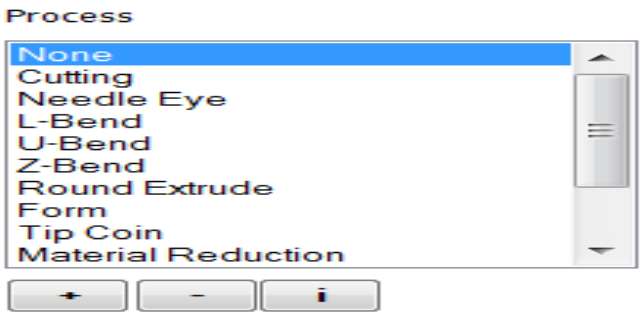


  - 1. Open the die set template and select the "U" assembly from the feature manager tree of SolidWorks, open the COT Manager select the **Setup Tab**.
    - 2. 1st test to see if the assembly has a property set already by clicking on the **Check Tab** [i]. If a property has been already set, features will high light inside the **Set Component Information**.
    - 3. If nothing high lights then continue by selecting the **Add Tab** [+]. By selecting the **Add** icon will add the property of **DocType – 85U**.
    - 4. To finish setting up the die set template continues with the "M" and "L" with the appropriate selection to complete the settings.
- **Setting of other PRL's**

- All other PRL's must be set with properties that describe a Component Name or an operation, some PRL's may have both setting inside them.

- **Setting a Process**

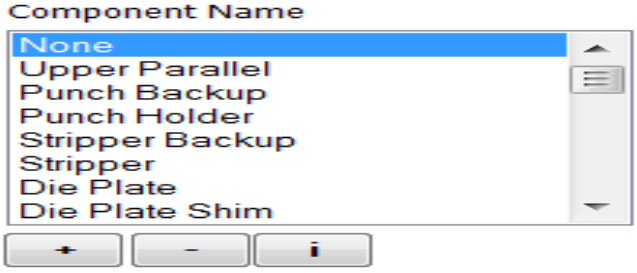
1. A **Process** is described as a main assembly of multiple assemblies or part files of a PRL file.
2. A **Process** can be added on the fly in a design setting after the PRL has been inserted.
3. The PRL should be marked and updated appropriately before using again.



4. By setting the main assembly this will allow the user to show / hide the full assembly in the design.

- **Setting a Component Name**

1. A **Component** is described as a sub-assembly or a part file inside of a main assembly, by setting them as component names the user will be able to show / hide individual components from the U, M, L.
2. This **Component Name** can also be added on the fly in a design after the PRL has been inserted.
3. The PRL should be marked and updated appropriately before using again.



- **Creating a Design Replication**
-

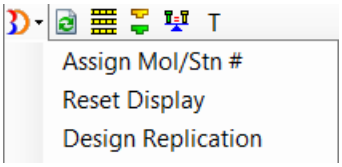
1. New Strip Layout for an old or new Punch Design.
2. New Strip Layout for an old or new Die Set Design.
3. New Punch design for an old or new Die Set Design.

-
-

- **2nd Option – New Strip Layout**
-

1. Open a completed Die Set Design.
2. Select the Design Replication Tab.

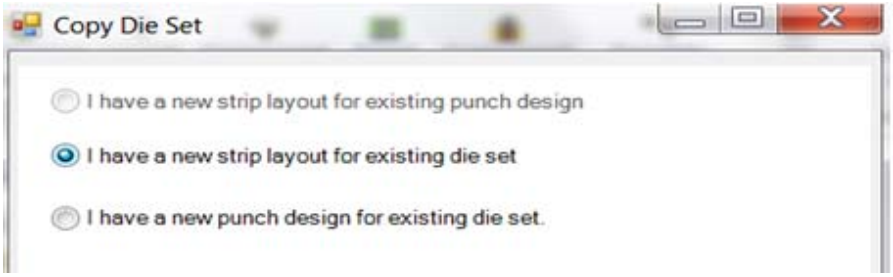
-



-

3. Select option 2 (I have a new strip layout for existing die set).

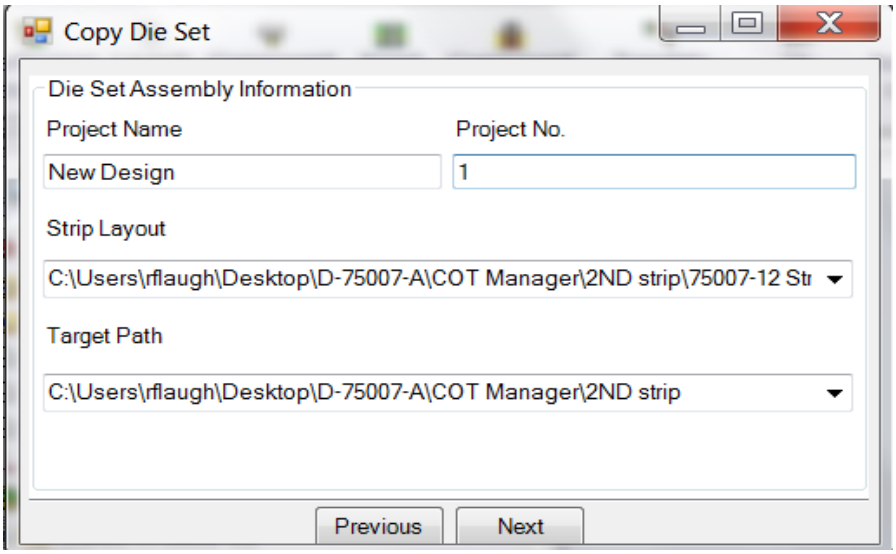
-



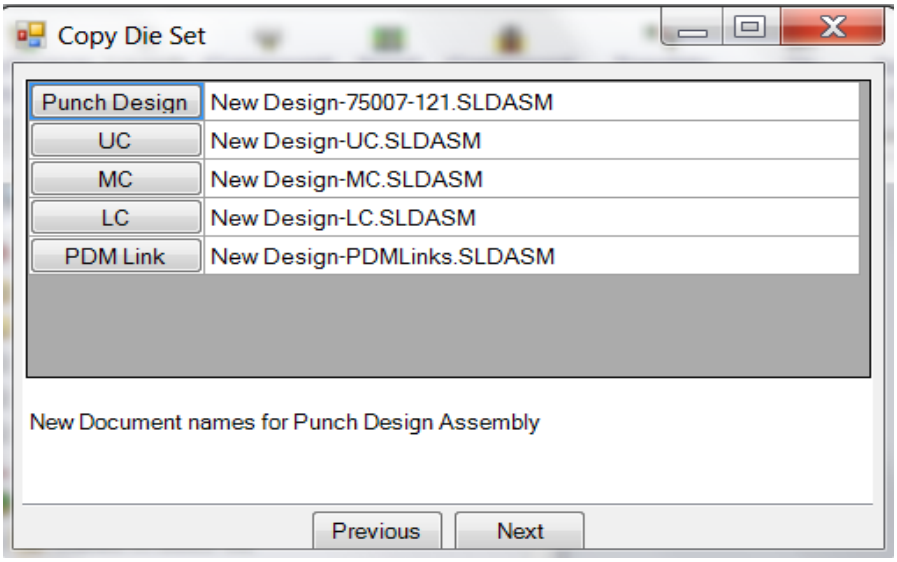
-

4. Insert the new Project Name, Project Number, Strip Layout Location, and the target path where the files will be save at.

-



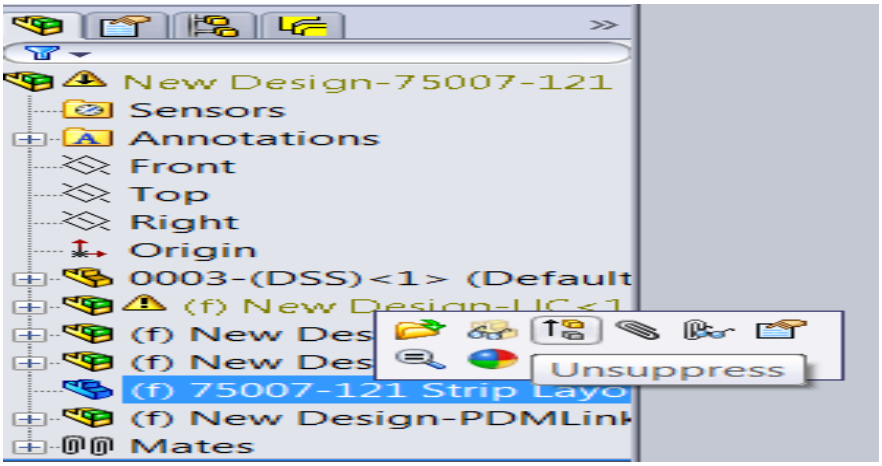
5. The COT Manager continues re-naming the UC, MC and LC assemblies inside the new design. Keep in mind the parts and assemblies that it is coping are still sharing.



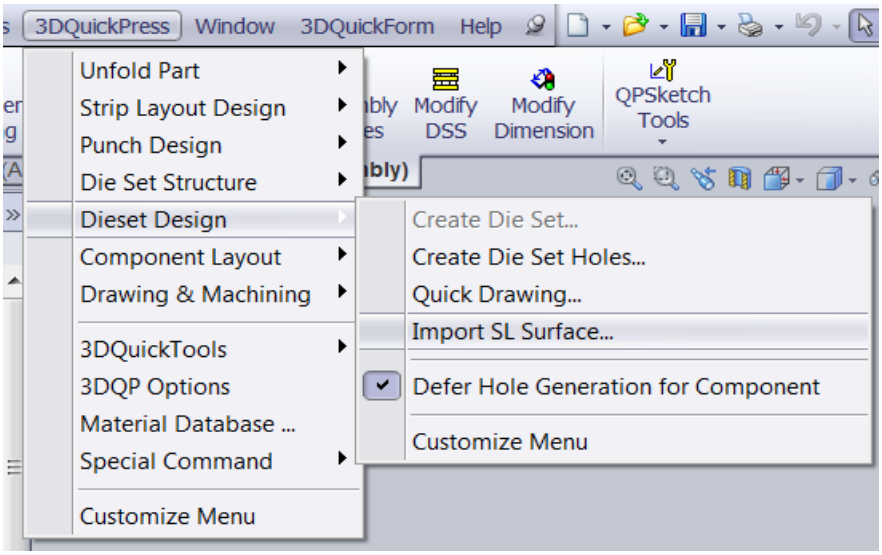
6. The COT Manager continues re-naming the U, M and L assemblies inside the new design. Keep in mind the parts and assemblies that it is coping are still sharing.

7. After the copy is complete the user must open the **Punch Design** and find the **New Strip Layout**

8. Go to the feature manager tree and un-suppress the Strip Layout, SolidWorks will ask you to locate the new strip, browse to the location of the new strip layout and select.



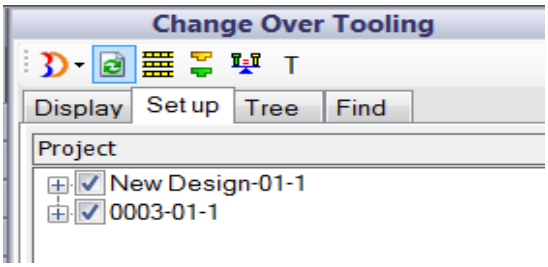
9. Go to the main die set design and insert the new strip layout.



Create Change Over Tooling Master

1. While you are in the current Die Set Design, open a new Assembly template and save it as the Main Assembly.

2. Select the setup tab of the COT manager, select the new main assembly in the feature manager tree and the Change Over Tooling Master, then set the property.
3. Insert the 2nd die set design into the new Main Assembly.
-
4. Select the refresh icon on the COT Utilities bar.
-



5. After refreshing the 2nd die assembly will be setup in the COT Manager.

Chapter 9

CONCURRENT DESIGN ENVIRONMENT

- Setting up a Concurrent Design
- Understanding the Concurrent Design
- Adding Designers to a project
- Removing Designers from a project
- Combining Designers

CONCURRENT DESIGN ENVIRONMENT

The 3DQuickPress Concurrent Design Environment is used to aid tool designers who are working collaboratively on the same project. This allows designers and engineers to work continuously through the duration of the project without the threat of interference from others working on the same project.

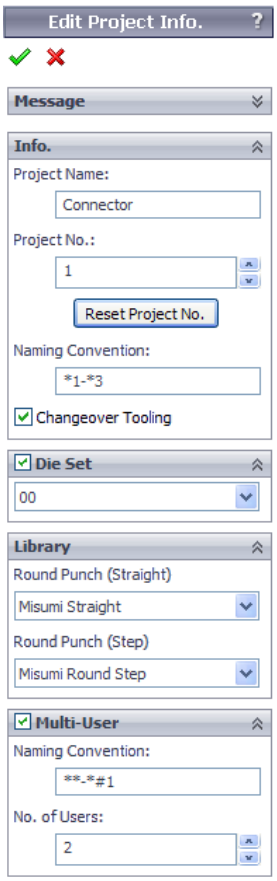
A master punch assembly is created along with sub-punch assemblies which are distributed to different designers. Any changes on the sub-punch assemblies will update the master punch design assembly. Sub-punch assemblies have their own strip layout display which can be controlled independently of other sub-punch assemblies and the master punch assembly. This mechanism is what allows multiple designers to work concurrently but safely on the same project.

Case Study 9 – 1 : Punch Design with Multiple Users

Setting up and managing the concurrent design process.

Setting Up The Environment

Starting the project with two users on one tool design.



1. Open the file named `Concurrent.sldprt`. File location: `Desktop\Training\ Concurrent Design\`

2. Click **Create Punch Design** from the 3DQP Punch Design Toolbar. Enable the **Multi-User** option.

Info.

- Project Name – Concurrent
- Project No. – 1
- Naming Convention – *1-*3
- Check Changeover Tooling
- die set – 00

Library

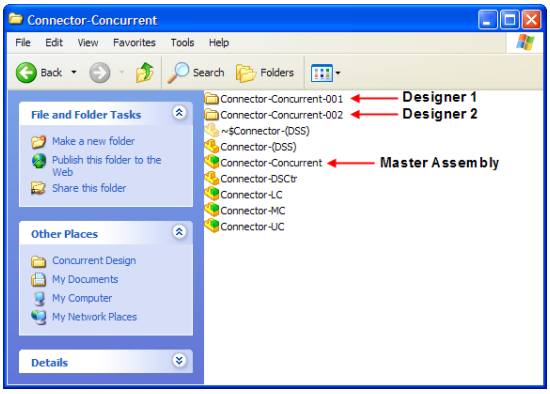
- Round Punch (Straight) – Misumi Staight
- Round Punch (Step) – Misumi Round Step
- Multi-User
- Naming Convention – **-*#1
- No. of Users – 2

3. Click ✔ to finish the command.

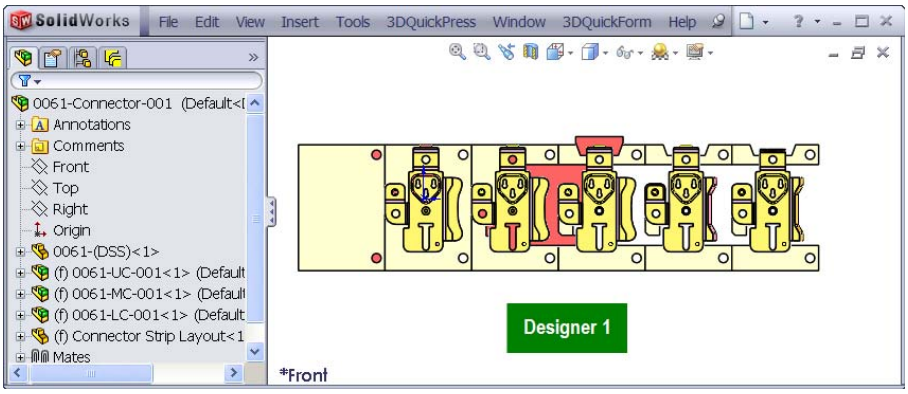
4. Click ✔ to close the Component Clearance dialog accepting the default values.

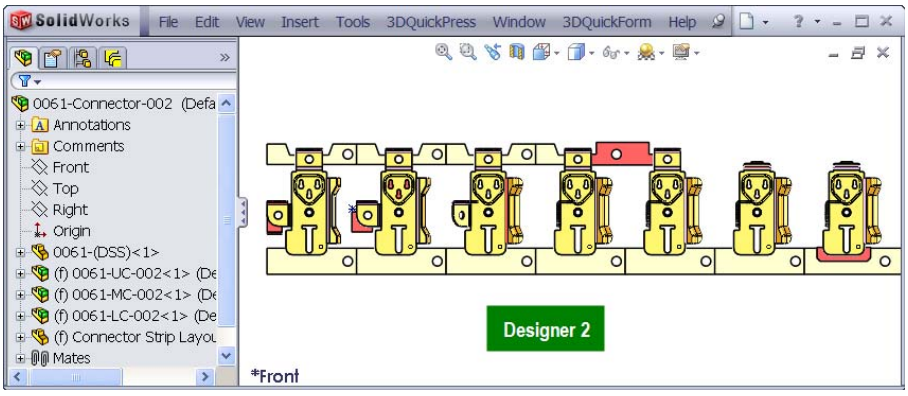
Understanding the Concurrent Design file structure

Sub-folders are created inside the project folder for each of the assigned designers. Engineers will store their work in their assigned folder. The master assembly will reflect the combined results of all sub-punch designs. Only one individual will have the rights to modify the master strip layout.

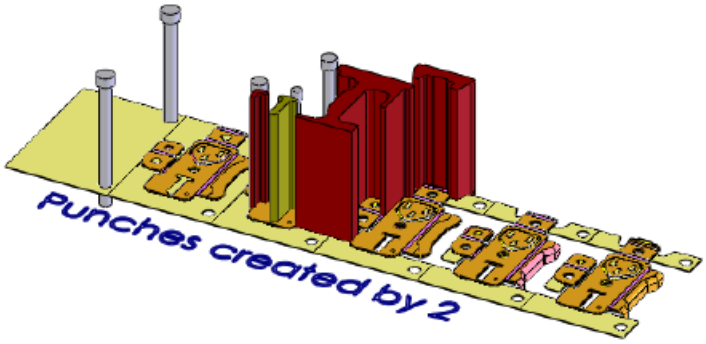


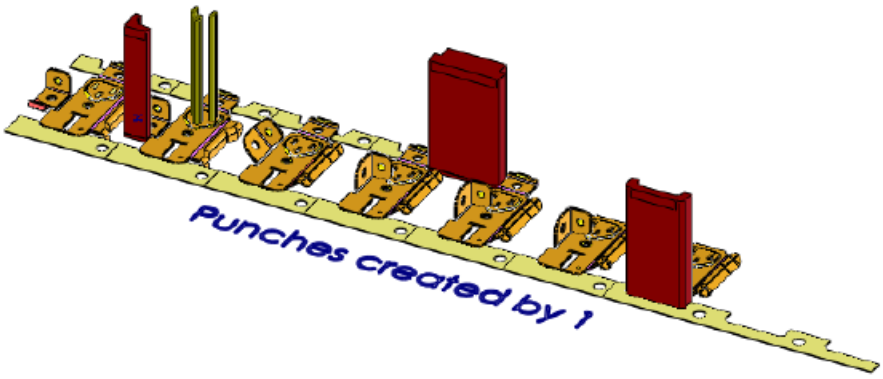
Each individual sub-punch design will be limited to a range of stations. The display of the strip layout for each sub-punch design can be controlled by the individual user.

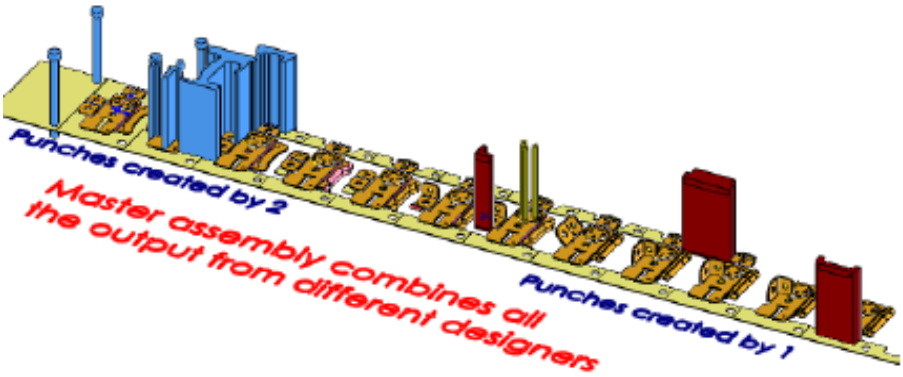




Components that are inserted in a sub-punch design assembly can only be seen inside that sub-punch design assembly and the master punch design assembly.





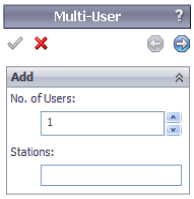


Case Study 9 – 2 : Managing Users (sub design assemblies)

The concurrent design environment supports the addition, deletion, and combining of multiple users' work at any stage of the design process. This can be done by going to **3DQuickPress**, **Punch Design**, **Multi-User**.

Adding Users

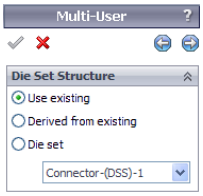
1. Click 3DQuickPress, Punch Design, Multi-User, Add.



   - Add
     - No. of Users – number of users to be added to current master punch design
     - Stations – define stations to be controlled by each new user

2. Click to go to the next screen.

   - die set Structure
     - Use existing – use the existing die set structure of the punch design
     - Derived from existing – use a new die set structure that is derived from the existing one
     - die set – set the die set structure to use in the list



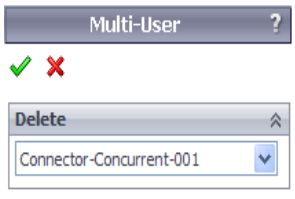
3. Click to go to the next screen.

   - die set Center
     - Use existing – use the existing die set center of the punch design
     - Derived from existing – use a new die set center that is derived from the existing one
     - die set – set the die set center to use in the list

4. Click to finish the command.

Deleting Users

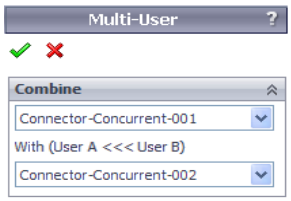
1. Click **3DQuickPress**, **Punch Design**, **Multi-User**, **Delete**



2. Select the sub-punch design.
3. Click to finish the command.

Combining Users (sub designs)

1. Click **3DQuickPress**, **Punch Design**, **Multi-User**, **Combine**.
2. Select the two sub-punch designs that will be combined.
3. Click ✔ to finish the command.



Chapter 10
TROUBLESHOOTING

- This Section is currently under development

Chapter 11

ADMINISTRATION AND CUSTOMIZATION

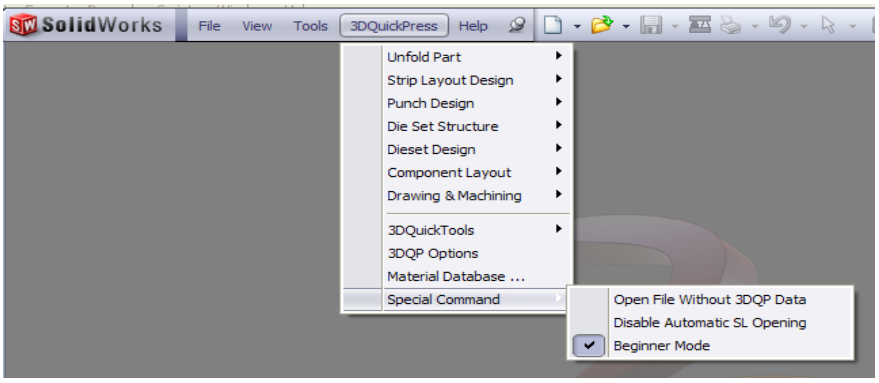
- Adjusting File Behaviors
- General Environment Setup
- Introduction to PRLs
- Creating PRLs
- Creating Die Sets PRLs

SETTING UP 3DQUICKPRESS

The proper setup and configuration of 3DQuickPress is essential to effective progressive die design. File behavior, custom properties, and manufacturing information are all specific areas of 3DQuickPress that are configurable to the needs of an organization.

Adjusting File Behaviors

The Special Command option within the 3DQuickPress pull-down menu offers several options that allow the user to file behavior as they are loaded into the working environment.



Open File without 3DQP Data

With this option turned on, any file opened will be loaded without any 3DQuickPress data, such as unfold and strip layout information.

This function is very helpful when troubleshooting a technical problem between SolidWorks software and the 3DQuickPress Add-in. By disabling the 3DQP data from a technical support scenario, the software can identify where and why a problem may be occurring.

Disable Automatic Strip Layout (SL) Opening

With this option turned on, a punch design assembly opens without the strip layout. Only the assembly document is loaded. This option will allow users to save system resources if needed on large assemblies.

Beginner Mode

With this option turned off, SolidWorks will allow the creation of external references. An external reference is created when one document is dependent on another document for its solution. If the referenced document changes, the dependent document changes also. Upon installation this option is turned on by default.

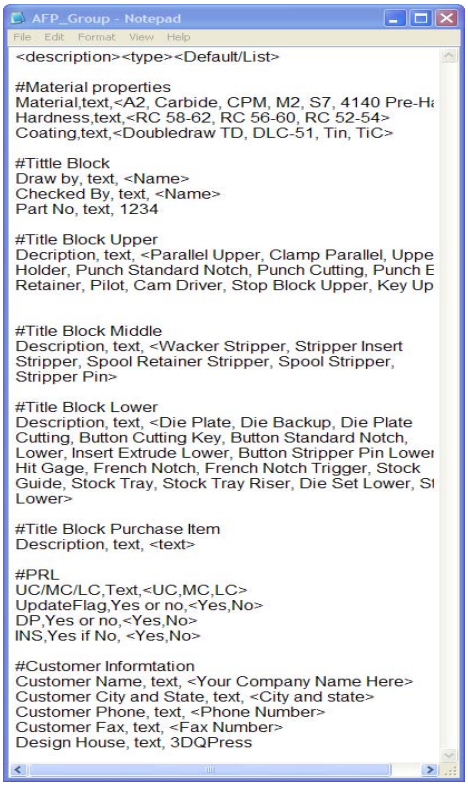
Refresh Testing Flags

From time to time, 3DQuickPress developers will allow users to preview and use “soon-to-be-released” enhancements. These functions are not officially released, so the user may choose to disable or enable these commands or options. This function allows the user to turn these functions on or off. A Testing Flag is a registry entry that enables or disables a function or capability of the 3DQuickPress software. This function eliminates the need for registry modification by the user. See Release Notes for further description of enhancements and fixes to the software located under the HELP menu.

3DQuickPress Custom Properties Manager

The 3DQuickPress Custom Properties tab is found on SolidWorks command manager under the 3DQP QTools CommandManager tab called `File properties`. By enabling this function, 3DQuickPress automates entries of properties such as company standards, reducing tedious repetitive data entry. All properties are added to SolidWorks custom properties. Below is the process to set the file up.

These properties are configured using the AFP-Group text file located in: `C:\Program files\3DQuickPress\DataBase\AFP_Group.txt`. This file can also be accessed inside of SolidWorks on the 3DQP QTools toolbar. From the File Properties command, select **edit** to access the text file.



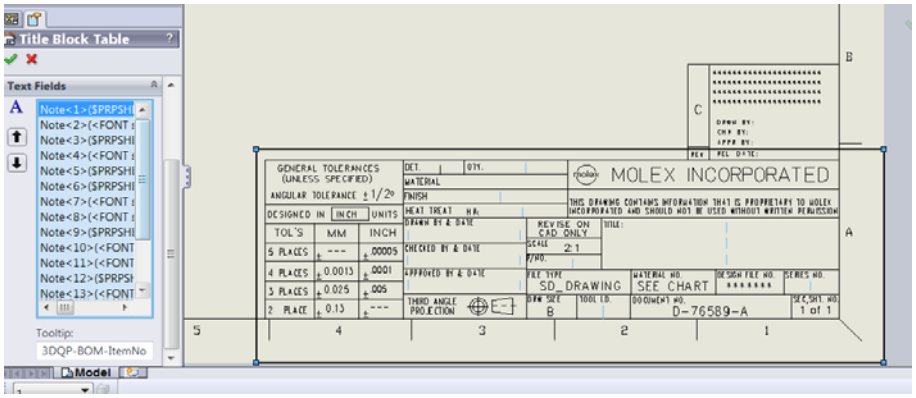
As shown above these properties are organized in groups. The # symbol indicates a group. The description of the group directly follows the # symbol. The user can create as many groups as needed to effectively organize the data.

Each property line following a group callout specifies a property name found in the company's SolidWorks title block. The syntax for these callouts is as follows:

[SolidWorks Property Name], text, <option1, option2, option3>

The < > symbols around the options are only necessary if there are multiple options entered. Otherwise, they may be excluded.

Shown below is a sample of the custom information in the title block. This is the information that will propagate from the SolidWorks Custom Properties of each detail. Please consult the SolidWorks Drawing Training manual on "How to setup."



3DQuickPress File Properties Editing

This command is meant for the customization and setup of the file properties group data.

Note: In a large implementation of 3DQuickPress this only should be modified by the administrator of the system.

Editing the group configuration file which is located in `<3DQPinstallation path>/AFP_Group.txt`

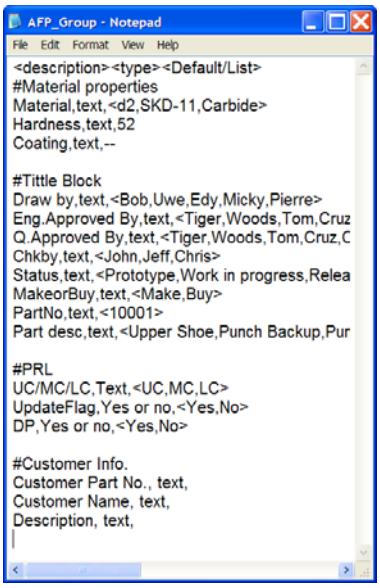
This is a text file with simple definition and should be used to customize the group name and the content of the group. The parameters that make up this content are:

- Properties that are listed in the group
- The default value per property
- Pull-down list values
- These values are available for the user to quickly change with a mouse pick versus manually typing text values helping to avoid typos and assure acceptable company best practices (standard tables) are utilized

Case Study 11 – 1 : How to customize the 3DQuickPress File Properties

To add a group for specific customer related properties

1. Click **File Properties** from the 3DQP QTools toolbar to launch File Properties Editor.
2. Click **Edit** button
3. Add the following lines in Note Pad.
   - #Customer Info.
   - Customer Part no.,text,
   - Customer name,text,
   - Description,text,



4. Save and exit notepad.
5. Click **Exit** button to exit File Properties Editor.
6. Click **File Properties** from the 3DQP QTools toolbar to launch File Properties Editor re-initializing the group file.
7. Click the dropdown under Grouping and Customer Info will now be listed.

SETTING UP PAINT MFG. INFO

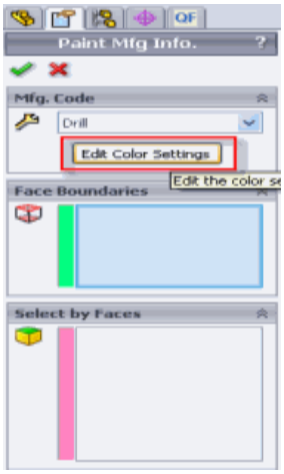
Manufacturing Information

3DQuickPress can color code the model based on manufacturing methods, providing direct visual cues as to the manner of these manufacturing methods. These can be customized to your specifications under the Paint Mfg. Info menu. The color coding of part faces is extremely helpful when the part is transferred to CAM systems. Masking is used for selections during programming.

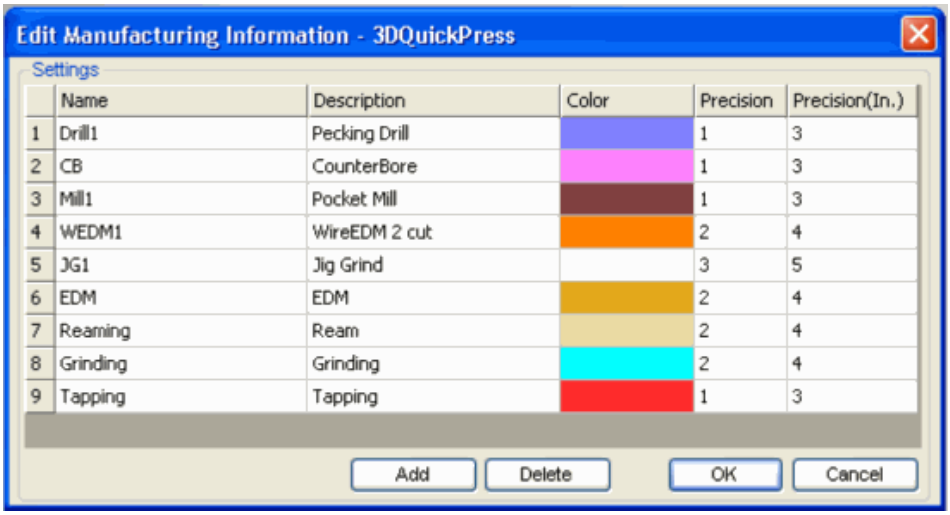
Note: Not all translators support color, nor do all CAM systems support selection by color.

Where to find it: **3DQuickPress, Paint Mfg. Info**

Open the Paint Mfg. Info command, and select Edit Color Setting.



In the Edit Color setting window you can add/delete manufacturing information. The “name” should be a shortened description of the operation. The color can be set by double clicking inside the cell for each method. The precision will control the number of decimal places in the hole chart when using the 3DQuickPress hole chart function.



	Name	Description	Color	Precision	Precision(In.)
1	Drill1	Pecking Drill		1	3
2	CB	CounterBore		1	3
3	Mill1	Pocket Mill		1	3
4	WEDM1	WireEDM 2 cut		2	4
5	JG1	Jig Grind		3	5
6	EDM	EDM		2	4
7	Reaming	Ream		2	4
8	Grinding	Grinding		2	4
9	Tapping	Tapping		1	3

Tip: For the best results, plan the machining operations before adding information. This will keep operations in order. See the example above.

Note: For the manufacturing color information to be propagated to all part walls automatically, this information must be added to all current and new Production Ready Libraries prior to working in 3DQuickPress.

The manufacturing information must be added to hardware files inside of 3DQuickPress located `C:\Program files\3DQuickPress\Standard components\all fasteners`. Manufacturing information need not be applied to all hardware files, only the files you will use.

The Mfg. Color Info database file

Setting up manufacturing color information is a pre-processing function to add manufacturing information to geometry by changing face color. The color code map to machining code is set up in a text file located in `C:\Program Files\3DQuickPress\DataBase\3DQP_AHT_MfgInfo.txt`.

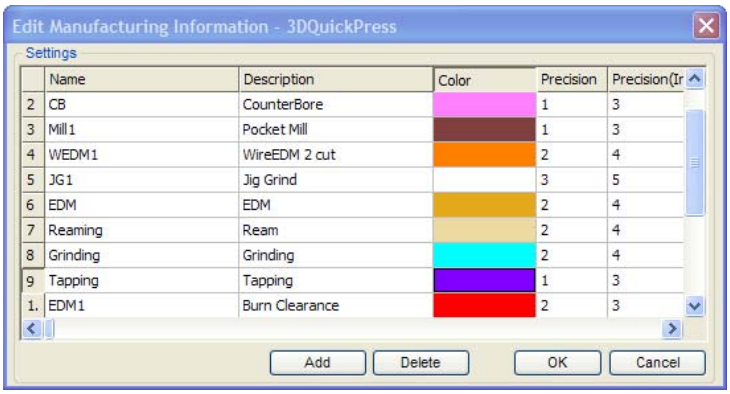
The format of the data file is shown below. The precision for metric and inch unit can be defined for each manufacturing process. A single RGB color code is assigned to each manufacturing process.

<R>	<G>	<B>	<NAME>	<DESCRIPTION>	<Precision>	<Precision(In.)>
#128	128	255	Drill1	Pecking Drill	1	3
#255	128	255	CB	CounterBore	1	3
#128	64	64	Mill1	Pocket Mill	1	3
#255	128	0	WEDM1	WireEDM 2 cut	2	4
#255	255	255	JG1	Jig Grind	3	5
#228	168	27	EDM	EDM	2	4
#235	219	163	Reaming	Ream	2	4
#0	255	255	Grinding	Grinding	2	4
#255	43	43	Tapping	Tapping	1	3

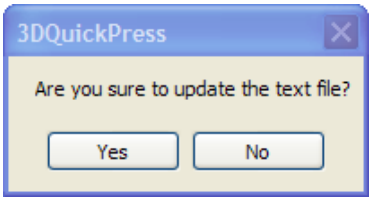
Case Study 11 – 2 : Setting up Mfg. Color Info.

To customize color codes

1. Click **3DQuickPress**, **Drawing & Machining**, **Paint Mfg. Info.**
2. Click **Edit Color Setting** to launch the Color Settings window.



3. Click **Add** to insert a new line and input **EDM1** for Name and **Burn Clearance** for Description. Select **Red** color.
4. Click **OK** and then **Yes** to confirm the modification of setup file.



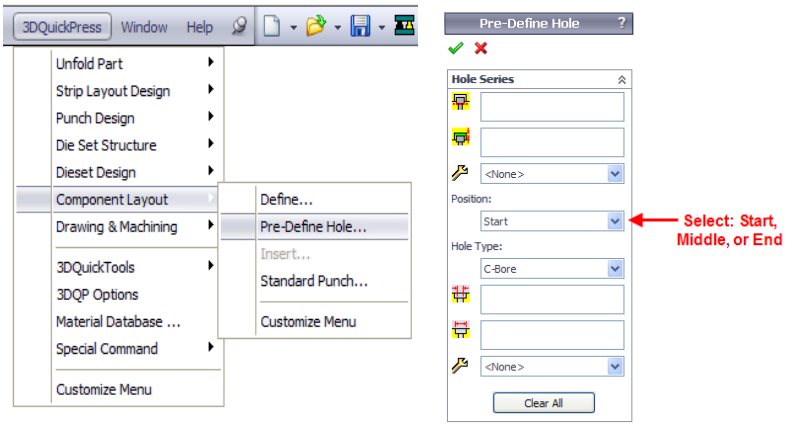
5. Click ✔ to finish the command.

Applying to standard components with Pre-defined holes

Manufacturing information can be automatically inserted when using the 3DQuickPress standard hardware library. This has to be setup for each fastener supplier that will be used. This information is setup using predefined hole information.

Defining manufacturing information is also supported on predefined holes of standard components. The manufacturing color and data will be applied when the hole is inserted into the part if the definitions were set up prior to the insertion.

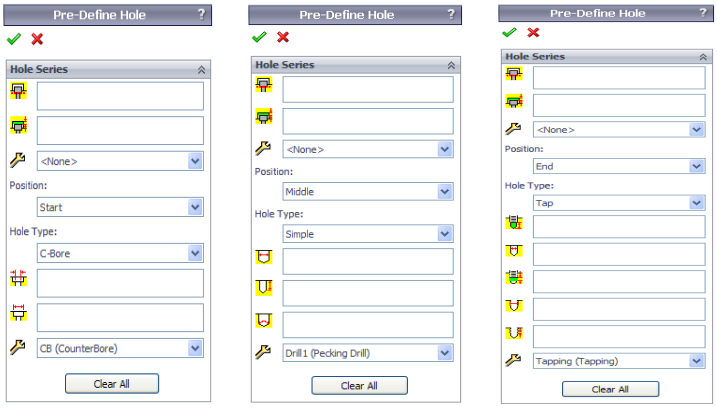
Where to find it: **3DQuickPress, Component Layout, Pre-Defined Hole**



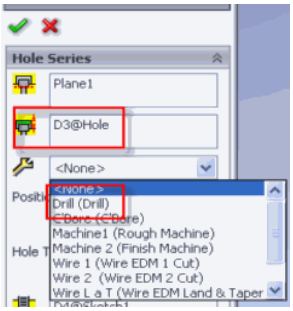
In the Pre-Defined Hole menu, there are 3 positions that you need to set; the **start, middle** and **end.**

As an example:

- Select the **start** position.
- Then select the drop down menu next to the wrench.
- Set this to Counterbore.
- Repeat this process for the middle and end positions using drill and tap respectively.



- For Socket Head Cap Screws (SHCS) you will need to set the clearance for the head. Select drill from the wrench dropdown menu located above the position field.



REPEAT this with all hardware in 3DQuickPress to set up the appropriate shop practices (standards) for your company.

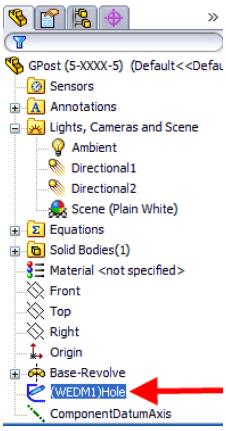
Applying to standard components with hole sketches

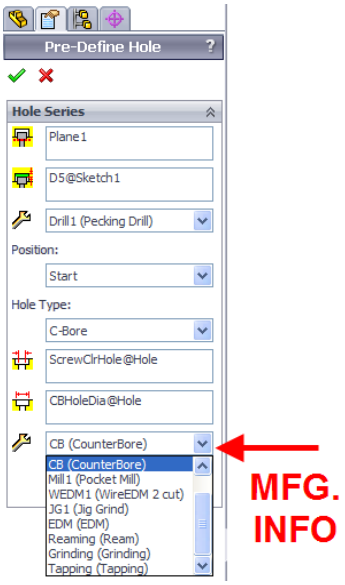
Pocketing holes for cylindrical components can be defined with a sketch that is a cross section of the hole and is used to create a revolved cut feature by 3DQuickPress. The revolved cut feature will be painted with the predefined manufacturing color information.

1. Open the part file `GPost (5-XXXX-5)`. File location: `C:\Program Files\3DQuickPress\Standard Components\Danly\Guide Posts\GPost`

2. Click **Mfg. Color Tool** from the 3DQuickPress Misc. Tools toolbar.

   Select **Hole** sketch in the feature tree, and select **WEDM1** as `Mfg. Color Info`. Click **Set color for selected sketch** to assign color for the sketch.





MFG. INFO

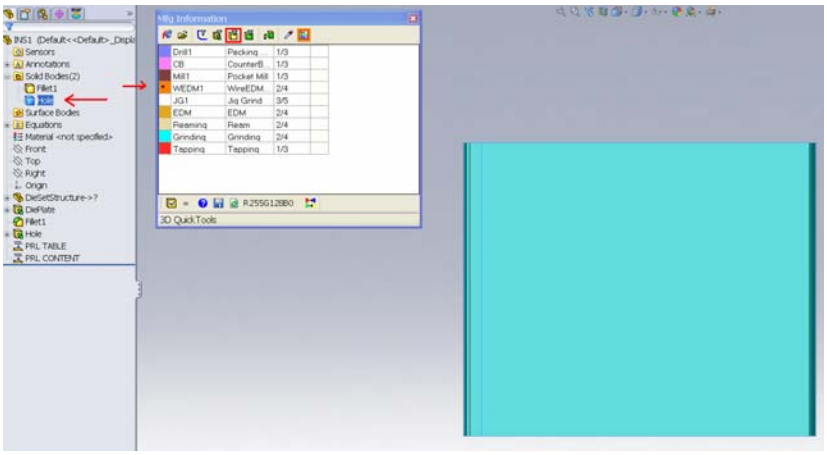
Applying to PRL components with hole bodies

Pocketing holes for PRL components are defined with hole bodies. The cavity feature will be painted with the predefined manufacturing color information.

1. Open the part file `INS1` File location: `C:\Program Files\3DQuickPress\PRL\PunchDesignTemplate\Inch\User Defined\Inserts\INS1\`

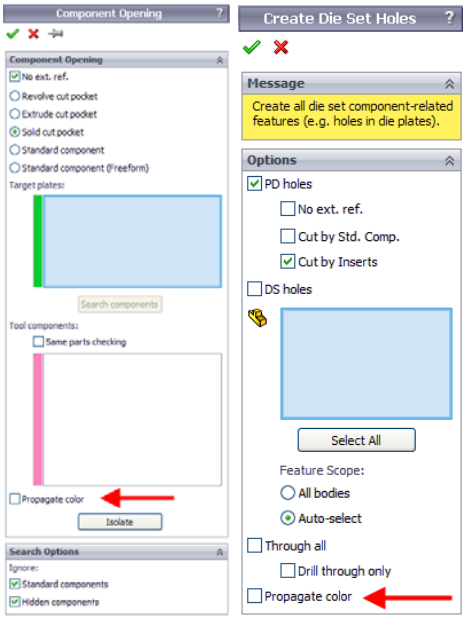
2. Click **Mfg. Color Tool** from the 3DQuickPress Misc. Tools toolbar.

3. Select the **Hole** Body in the feature tree and select **WEDM1** as `Mfg. Color Info`. Click **Set color for selected body** to assign color for the body.



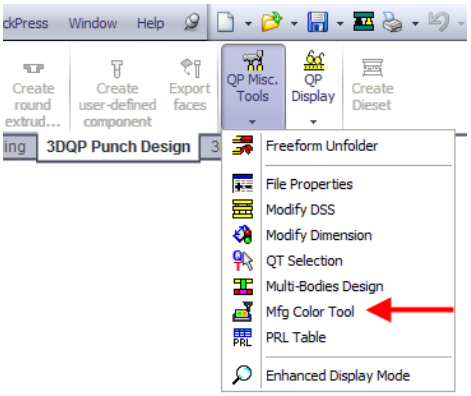
Using Mfg. Color Info. when creating cutting pockets

In both the Create die set Hole and Component Opening functions of 3DQuickPress, there is a propagate color option to carry the mfg. color info. From the standard component or PRL component to the pocketing holes. Once the propagate color option is turned on, pocketing hole created will have the same mfg. color as the color defined in the standard component or the PRL component.

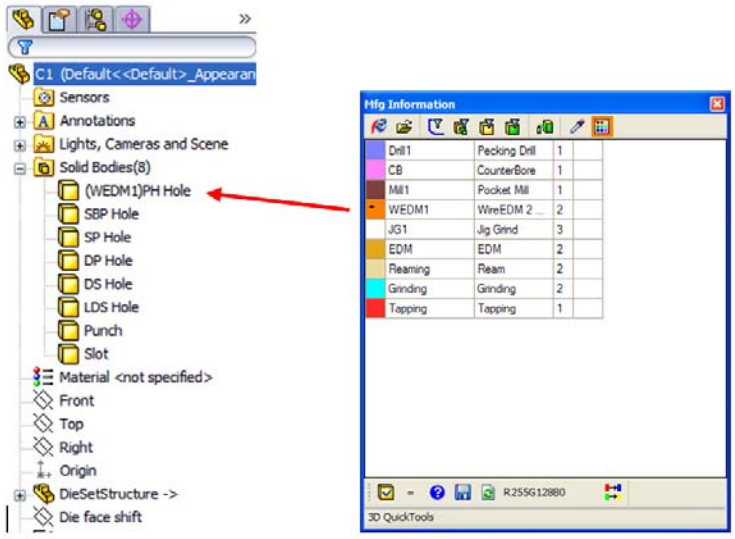


Using Manufacturing Information to define features manually

3DQuickPress also gives us the ability to manually define the manufacturing information for features created in the model. This option is located on the `3DQuickPress Punch Design toolbar, QP Misc. Tools, Mfg Color Tool`.



Example: If you open a cutting punch and go into the Manufacturing Info selection menu, you can select the manufacturing method and apply it using the selection filters at the top of the window. The colors on your model will update after you select your manufacturing type.



Setting up SolidWorks File Locations for 3DQuickPress data

3DQuickPress utilizes Library Features and 3DQP Templates to facilitate automated operations inside of 3DQuickPress. Library Features is the location of files that are created for 3DQuickPress to use for the Insert Hole Series tool, it mimics the SolidWorks hole series. The 3DQuickPress hole series has the advantage of assigning Library Features to each plate and has better

performance when the assembly is edited. It is also able to identify common sketches to eliminate die set top level assembly context.

The 3DQuickPress Templates are set up to create quick drawing files with views and station by station information. Some default templates are provided, but all templates can be customized to meet a company's need.

File locations for the Library Features and Templates need to be set inside SolidWorks by setting **Tools, Options, System Options, File Locations.** Each of the following locations should be added to the existing lists.

Click Tools, Options, System Options, File Locations. Add the follow paths:

Show folders for:	**Folders:**
Design Library	C:\program files\3dquickpress\PRL\Library Features
Document Templates	C:\Program Files\3DquickPress\3DQP Templates

INTRODUCTION TO PRODUCTION READY LIBRARIES - PRL

Production Ready Libraries (PRLs) are the cornerstone capability available in the 3DQuickPress environment. The PRL technology is a toolbox of concepts and techniques used to automate and leverage SolidWorks and 3DQuickPress data in order to speed up the creation and editing of parts, assemblies, and drawing files according to a designer's and or a companies' standard best practices.

PRLs are designed to capture design intelligence for specific 2D & 3D tasks while automatically adapting to the specific and current die design parameters of the job at hand. Many redundant and tedious tasks are automated so that the designer can concentrate on the more difficult design decisions and tasks.

PRLs are not standard components libraries although can incorporate standard components such as purchased items. PRL's are unique in many ways and are an advanced topic to author and maintain properly. No API or programming knowledge is required. However, extensive SolidWorks knowledge (parametrics, solid and surface modeling, etc) along with die design knowledge and company shop practices are very important in order to truly grasp and implement the capabilities of PRLs.

PRLs are primarily used in the 3D Punch and Die set Assembly steps of 3DQuickpress design flow. PRLs can automatically create drawings related to the 3D parts and assemblies being incorporated in to unique die designs. Some additional benefits include maintaining standard naming conventions and file locations for standard parts (i.e. purchased components). In addition, job specific (custom) parts and assemblies can be uniquely named and stored in the proper locations with company specific conventions. File properties and manufacturing information can also be incorporated into a PRL so that the BOM is accurately filled in and updated.

Once a set of PRLs are created the basic 3DQuickPress user can quickly create dies. Keep in mind, PRLs can always be edited on the fly and or modified in the master library for long term reuse. We advise the beginner to get started, use them and or have designers use them, take feedback then refine and enhance. If you need enhancements and or technique advice, do not hesitate to contact your support team.

Properties for PRL's

There are several properties that control the behavior of PRLs. The properties are set outside of SolidWorks. You can find this under **Programs**, **3DQuickPress**, **3DQuickTools**, **Utilities**, **File Properties**

	Property Name	Type	Value	Delete
1	UC/MC/LC	Text	UC	
2	UpdateFlag	Yes or no	Yes	
3	DP	Yes or no	Yes	
4	INS	Yes or no	Yes	
5	XSL	Yes or no	Yes	
6	F-SH	Yes or no	Yes	
7		Text		

UC/MC/LC - You can control where Punch PRLs will be inserted using this property. This property is only valid in Punch Design. The possible values are:

LC - Lower Container (assembly)

MC - Middle Container (assembly)

UC - Upper Container (assembly)

F-SH (Yes|No)– Is for extra small PRL's.

Update Flag (Yes|No) – This enables or disables the PRL parameters to be updated is a 3DQuickPress specific parameter is changed and the die is told to update.

DP (Yes|No) – Die Plate. This property is ONLY for details that will reside in die set design, such as stripper and punch holder plates that will be cut by EVERYTHING in the punch design. If set to YES, these parts will automatically be cut when Create die set Holes is executed. They will be cut by ANY component from the UC, MC, LC assemblies that interfere with them.

Note: Create die set Holes is a powerful timesaving automated capability.

INS (Yes|No) – punch design inserts in the lower die block.

XSL (Yes|No) – Set to Yes to extrude up to surface for the shape of a form insert. When inserted with this property the surface will automatically export to all part bodies in the PRL. This functionality minimizes unnecessary assembly associativity to reduce rebuild times in SolidWorks.

3DQP COT SET – With TEXT as the type and set the value P1 to

the strip layout P1 Strip Layout, P2 for P2 Strip Layout. This is only an example of what you will get if you check.

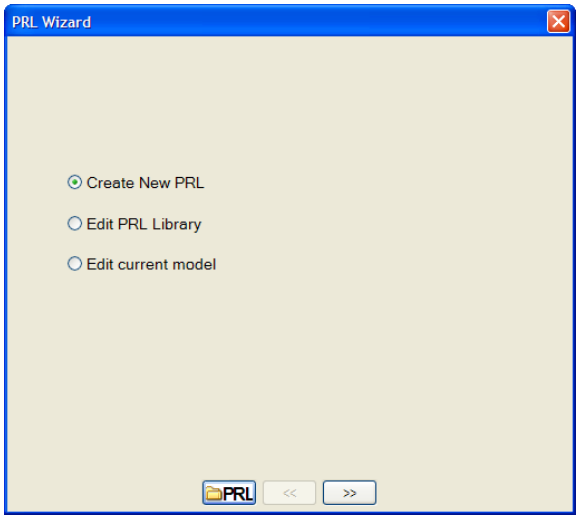
Case Study 11 – 3 : Create Rectangular PRL Punch

In this case study we will create a new PRL for a rectangular punch. The process will create a SolidWorks part that represents the punch and we will configure it as a PRL. When completed, this custom punch can be inserted into a punch design assembly.

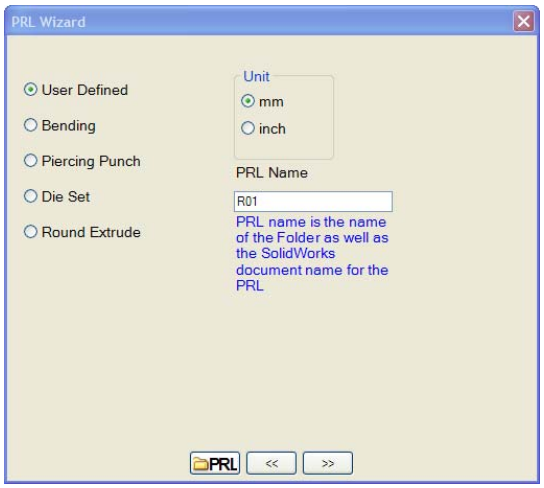
Establish the new PRL

1. Use the PRL Wizard. Click from the top menu bar **3DQuickPress**, **3DQuickTools**, **Utilities**, **PRL Wizard**.

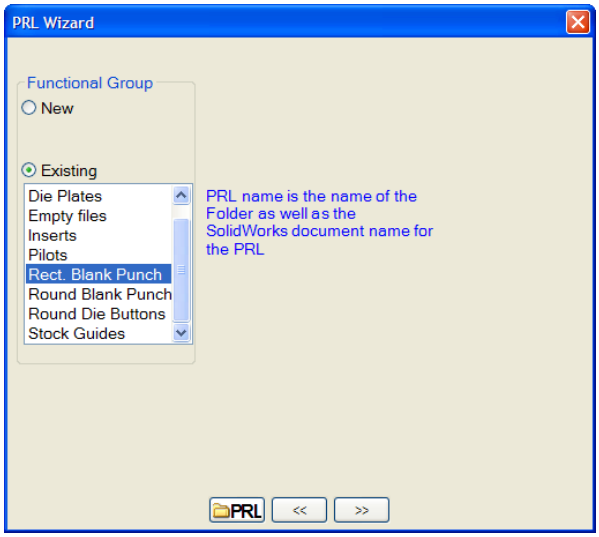
2. Select **Create New PRL** and then click **>>** button



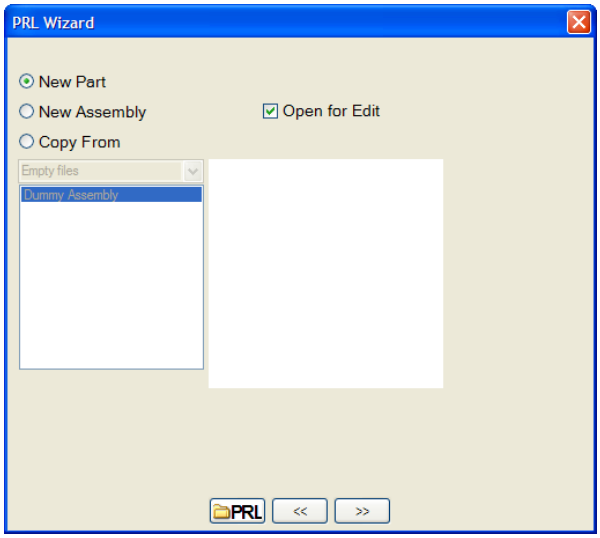
3. Select **User Defined**, set the `unit` to **mm**, and input `PRL Name` as **R01** and then click **>>** Button. This will create a folder and a SolidWorks part with this name.



4. Specify the Functional Group for this PRL. Select **Existing** and select Rect. Blank Punch and then click **>>** Button.



5. You will now specify where the part used to create this PRL comes from. Select New Part and check Open for Edit to open the file in SolidWorks. Click the >> button to continue.

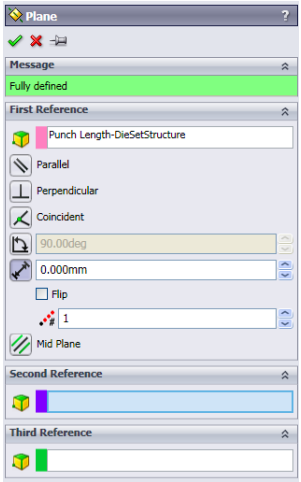


6. Click the **Insert die set Structure (DSS)** icon, then click **exit** to finish.

Note: The DSS is a standard part that is inserted into the current part and is extremely important to the 3DQuickPress automation. By inserting the DSS it will allow the new rectangular punch to have intelligence to automatically adjust to any die design. For instance, different plate thicknesses, shut heights and other standard variables that change from job to job are updated in the PRL while inserting it with 3DQuickPress commands.

A new part is automatically created in SolidWorks named `R01` with all the required reference geometry. This file is located in:

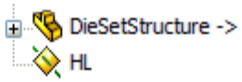
`C:\Program Files\3DQuickPress\PRL\PunchDesignTemplate \Metric\User Defined\Rect. Blank Punch\.`



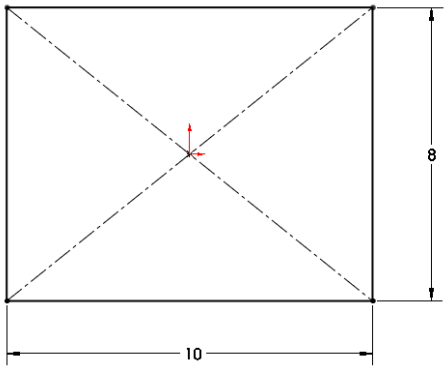
Create the initial geometry

1. In SolidWorks, insert a reference plane offset from the existing reference plane named: `Punch Length-DieSetStructure`.
2. Select the reference plane named: Punch Length-DieSetStructure
3. From the SolidWorks Menu, select: Insert, Reference Geometry, Plane, Offset.
4. Check the direction of the offset to assure it is in the direction of the punch.
5. Set the distance of the offset to **0.00** and click ✔ to finish the command.

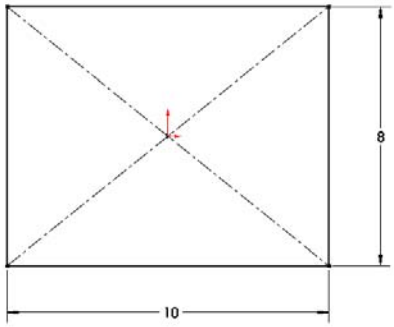
6. Rename the datum plane to **HL**.



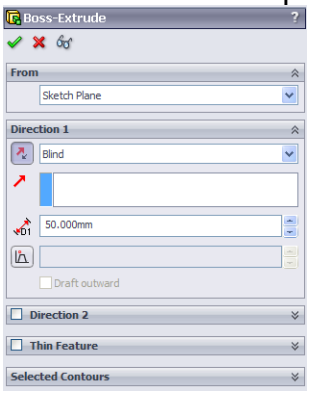
7. Right-click HL plane and **Insert Sketch** .



8. From the Sketch toolbar, select **Rectangle** . Create a **10mm x 8mm** rectangle centered on the origin. Rename the sketch to Base Sketch.



9. **Extrude** the sketch. Set the end condition to **Blind** and set the depth to **50mm**. The extrusion occurs in the same direction of a punch.



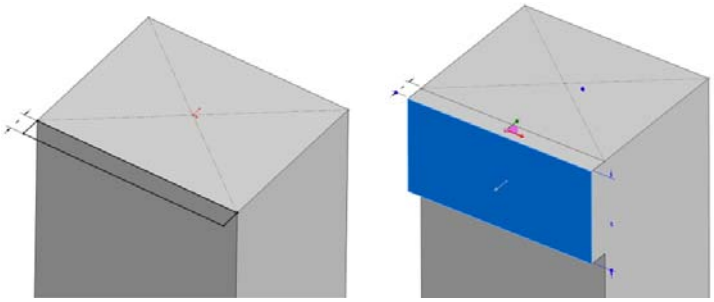
Creating the cutting geometry

1. Show `Base Sketch` in the feature tree by expanding the feature `Boss-Extrude1`.
2. Right-click `Base Sketch`, and select the eye glass icon.

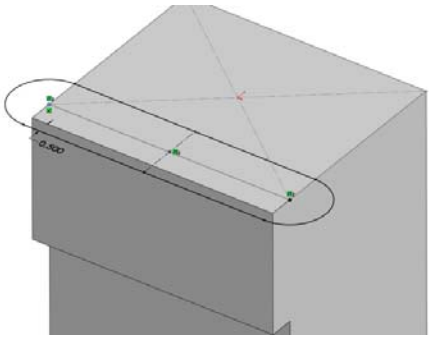
Note: You may also want to hide the extruded body and work with the sketch.

3. Pre-right-click `HL` plane, and **Insert Sketch**.
4. Sketch a **Rectangle** with a **1mm** width and end points coincident with the 10mm x 8mm rectangle sketch, **Extrude** the new sketch to make a foot on the punch **5 mm** thick as shown below.

Tip: Be sure to merge the main body with the foot.



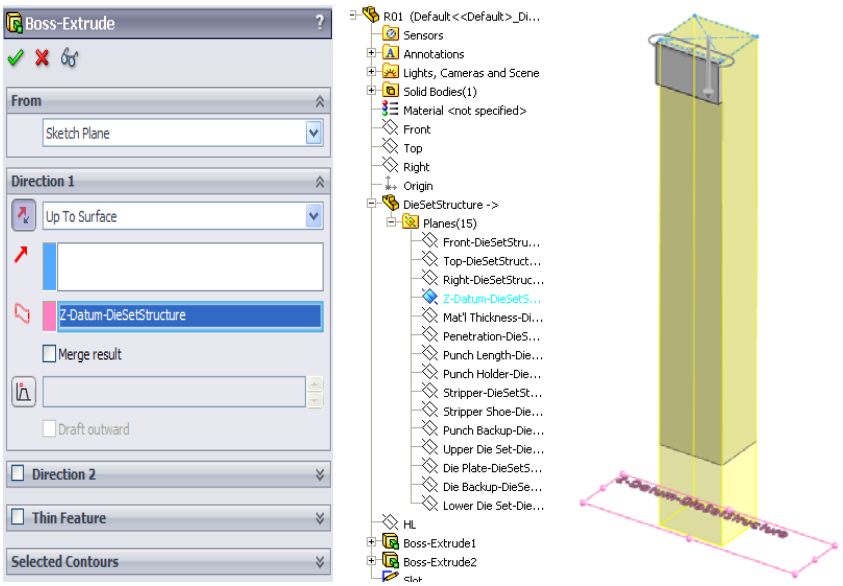
5. Right-click HL plane, and Insert **sketch**. Preselect the sketch line between the main body and foot.
6. Insert a Slot by selecting the 3DQuickPress, QuickTools, QuickPress Sketch Tools.
7. Delete the dimensions and constrain one of the end center line points of the foot sketch and add a **.5 mm** dimension as shown below.



8. Click to finish.
9. Rename the sketch to Slot.
10. Create a Clearance Hole Body for the main extrude. Automatic clearance will be added when the body is inserted

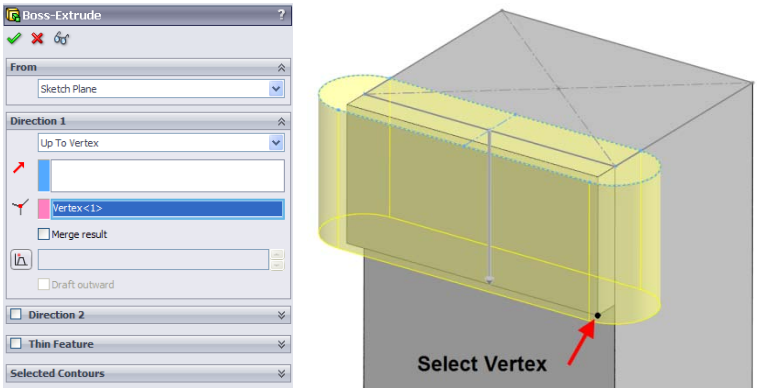
into a Die Set. Select Base Sketch from the feature tree and click Extrude Boss.

11. Extrude the boss up to the Z-Datum-DieSetStructure plane in the DieSetStructure part. Do not merge results.

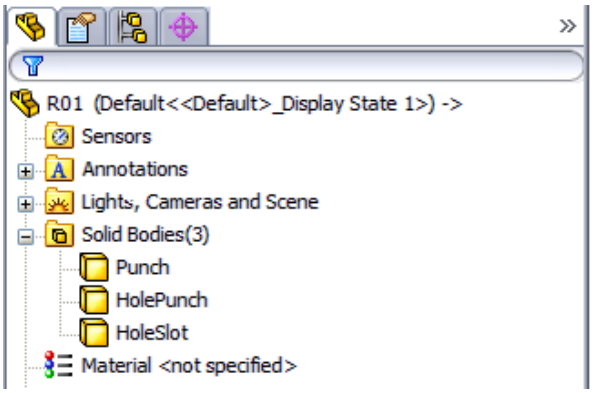


12. Create Clearance Hole Body for the foot. Automatic clearance to be added when inserted to a Die Set: Select Slot sketch and Extrude up to Vertex as shown below.

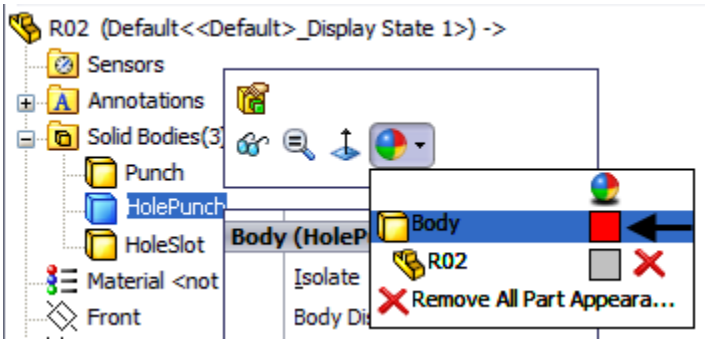
Note: Do not merge results. This will cause the slot extrude to be a separate solid body.



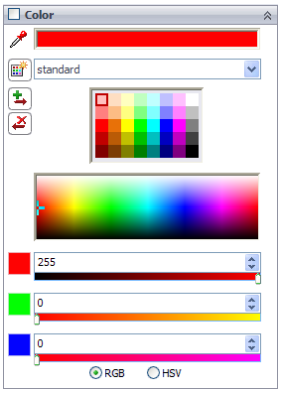
13. Rename the new bodies in the Feature Manager to Punch, Hole Slot, and Hole Punch.

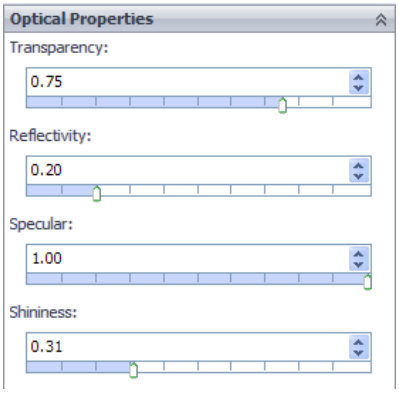


14. Set the hole color to transparent red by right clicking on the solid body, selecting the appearance icon, and click the square for the Body.



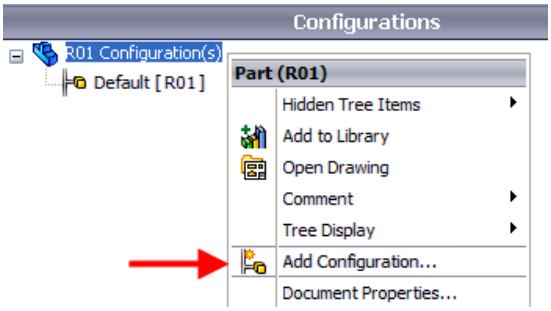
15. Select red from the color grid, and change the transparency under the optical properties.





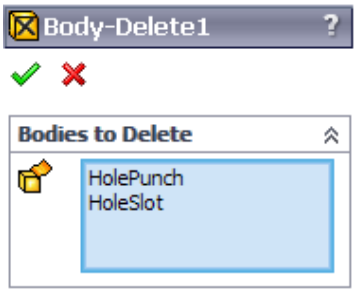
16. **Save** the part.

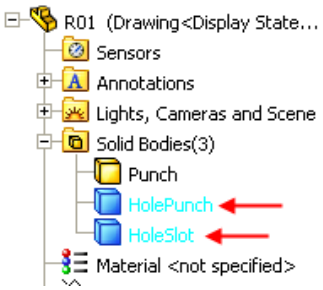
17. Click the configuration tab, add a configuration called Drawing by right-clicking on the `R01 Configuration` at the top of the tree and selecting **Add Configuration**.



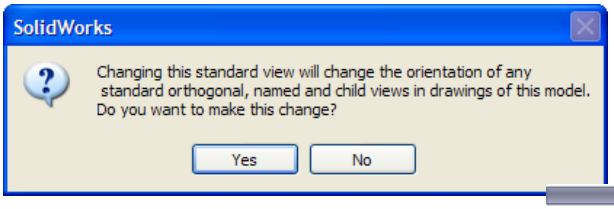
18. Click **Insert**, **Feature**, **Delete Body**, select the `Hole Punch & Hole Slot bodies`. Press ✔ to finish the command.

This feature will delete the unneeded bodies from the solid bodies folder and create a feature in the feature tree called body delete. It is recommended that you rename this feature with an appropriate description of what was deleted.





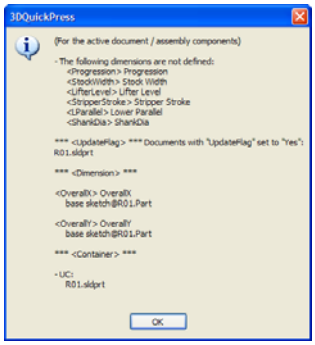
19. Update the standard views. Go to a **Front view** and press **Space Bar**. Select **Top** and click the **Update Standard View** icon. Click **Yes** when the warning window pops up. This will now orient the model so that the Front view is now the Top view.



20. Click **3DQuickPress**, **Punch Design**, **Define PRL** to define the punch as a PRL.

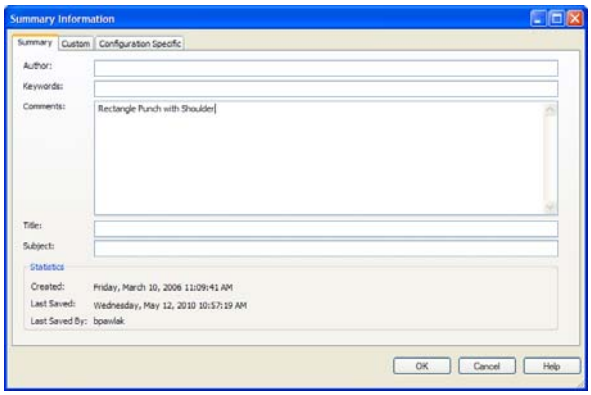
21. Double click on the `Boss-Extrude1` to bring up the dimensions of the feature.
22. Select dimension 10 and click **OverallX**.
23. Select dimension 8 and click **OverallY**.
24. Under the `container` property, select **UC** from the drop-down menu and click the **Set** button.
25. Under the Update Flag property, select **Yes** from the dropdown menu and click the **Set** button.

26. Click the **Report** button to check that all parameters have been set. Click **OK**.



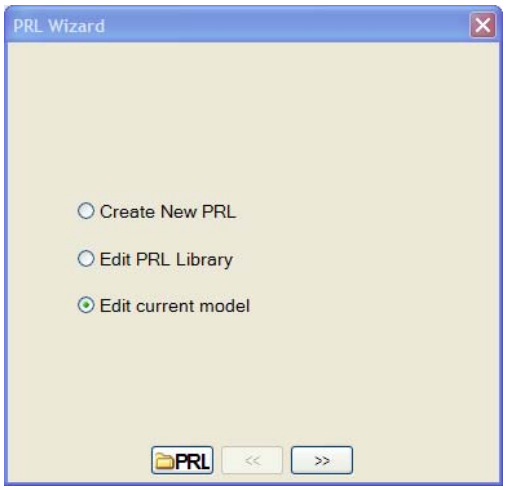
27. Click the ✔ to finish this operation.

28. Add a description for the PRL Preview Window by going to File, Properties and select summary tab. Add the following to the Comments field: **Rectangle Punch with Shoulder**.

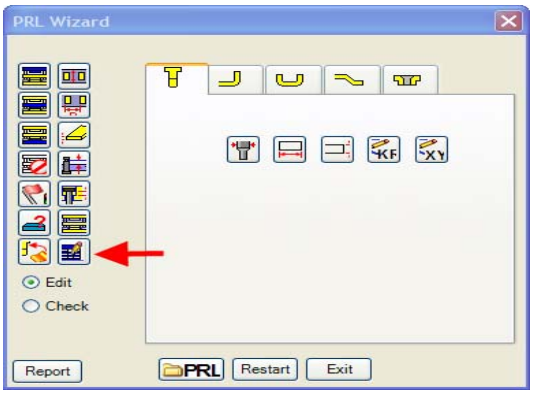


29. Click **3DQuickPress**, **3DQuickTools**, **Utilities**, **PRL Wizard**.

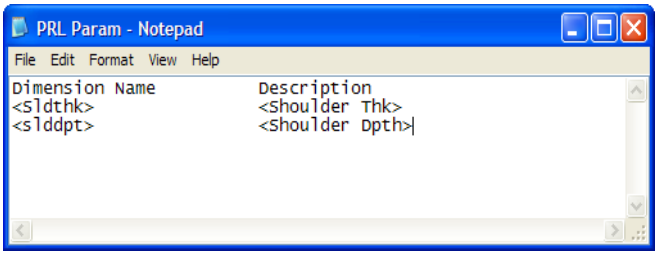
30. Select **Edit Current Model** and then click the **>>** button.



31. Select **Insert PRL Param.def file**, and click **Exit** to finish.



Add the information as follows, and save to the `R01` folder. These parameters will be available the next time you define a PRL.

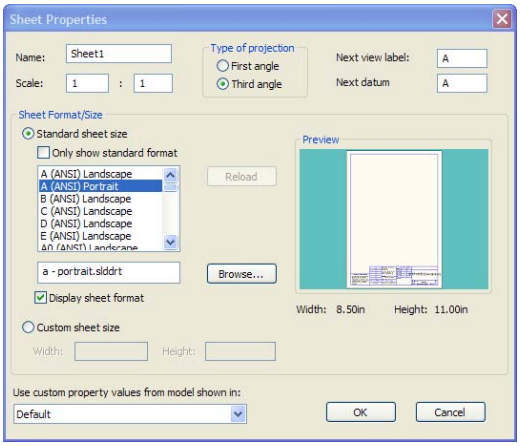


32. Select **3DQuickPress**, **Punch Design**, **Define PRL**, repeat steps 20 thru 27 to connect the two new parameters (sldthk & sldprt) to the proper dimensions related to `Boss-Extrude2`. This renames the dimension to link them to the PRL interface. When the PRL is inserted into a die, the user can enter new values for these dimensions with the PRL insert command.

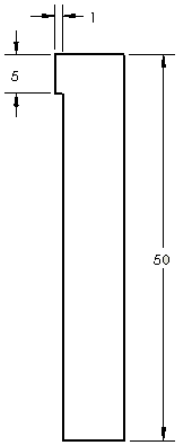
Note: Do not select the same dimensions as step 22 & 23.

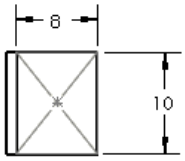
33. Click **File**, **New**, **Drawing Template** and then select the `A4-Portrait` sheet size. Press the **ESC** key if Model View property manager opens up.

34. Right-click on the drawing and go to **Properties**. Set the projection to **Third Angle** and **1:1** scale.



35. On the View Layout toolbar, select **Model views**, select R01 from the open documents window, then click the **Next** button. Check the option to create multiple views, then select both the **Right** and the **Bottom** views.





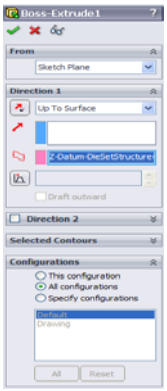
36. Click **Insert**, **Block** and insert the XYPos. File location: `C:\training\annotations\XYPos.sldblk block` to the drawing.
37. **Save** the drawing as `R01.sldrw` under the same folder as the part.
38. Return to the `R01` part, set the part configuration back to Default.
39. **Save** all files.

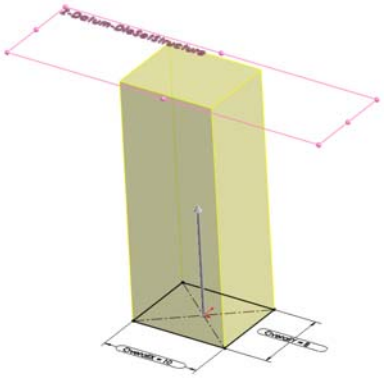
Case Study 11 - 4 Create Rectangular PRL Assembly

In the previous case study a part PRL for a punch was created. In order to make a PRL for both the punch and die side to be inserted, we will create another PRL and use the existing punch to quickly create the die side part. Then create an assembly PRL that combines the punch and die into a new PRL.

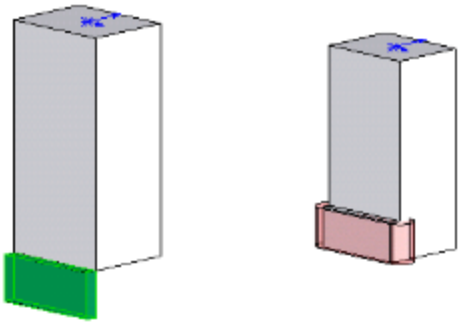
Duplicate the punch and use it as a starting point for the die

1. Start the **PRL wizard 3DQuickPress, 3DQuickTools, Utilities, PRL Wizard**.
2. Click **Create New PRL** and then click **>>** button.
3. Click **User Defined, mm**, Input PRL Name = **R02** and then click **>>** Button.
4. Click **Existing** and select Rect. Blank Punch and then click **>>** Button.
5. Click **Copy From** and select R01 and check **Open for Edit** and then click **Finish** Button.
6. Click **Exit** Button.
7. Current document should be the R02 part, Save as `R02D.sldprt`. All file operations are inside newly created folder called `R02`.
8. Right-click on plane HL, select **Edit Feature**. Change the reference plane to `Die Plate-DieSetStructure`. This will move the start plane to the correct position for a die.
9. Right-click on feature `Extrude1`, select **Edit Feature**, change `End condition` to **Up to surface** and select `Z-Datum-DieSetStructure` of the DieSetStructure part. Press ✔ to finish the command. This will cause the die to extend to the correct position.





10. The direction of the features used to create the Shoulder and the Hole Slot may need to be reversed (please reverse if needed).



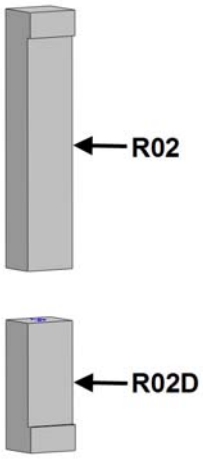
11. Switch to the drawing configuration.
12. **Save** the file.
13. Create a drawing for `R02D.SLDPRT` and **Save** the drawing.
14. Switch to `R02D` part, switch configuration to `default`, and hide `hole bodies`.

The directory `R02` now contains Part PRLs for the punch and die called `R02` (punch) and `R02D` (Die).

Create the Assembly PRL

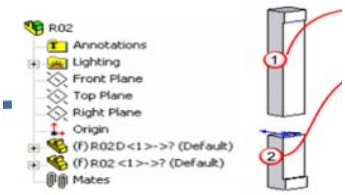
1. Create a new assembly in Folder R02 and save the assembly as `R02.SLDASM`
2. Insert `R02.SLDPRT` and `R02D.SLDPRT` to this assembly at the origin of the assembly-this is very important for locating at the same origin!

Tip: Use Insert Component, select your part to insert, and go straight to the green checkmark. SolidWorks will automatically align the part origin to the assembly origin.



3. Select **3DQuickPress**, **3DQuickTools**, **Utilities**, **PRL Wizard**
4. Make sure that your current window is `R02.SLDASM`, click **Edit Current Model** and then click **finish**.

5. Select `R02` in the feature tree and click **UC Button** on the PRL Wizard, select `R02D` in the feature tree and click **LC Button** on the PRL Wizard. Then **Exit** the PRL Wizard.



6. Go to **File**, **Properties**, **Summary tab**. Add “Simple Punch & Die” to comments field.
7. **Save** the assembly.

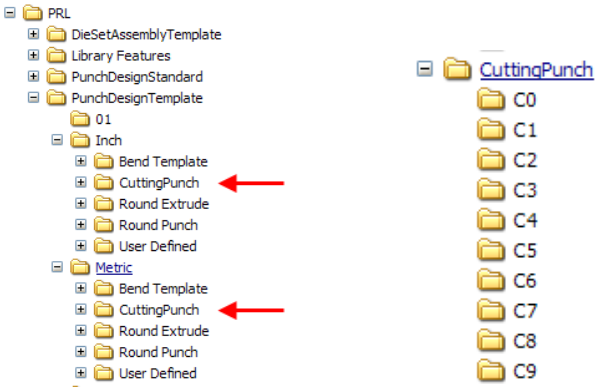
Cutting Punch Customization

Topics covered:

- Understanding cutting punch PRL structure
- Fine tune cutting punch parameters
- Customize a cutting punch by creating a variation
- Add ejector
- Punch with shear
- Create preview bitmap
- Cutting punch location

The data for Cutting PRLs is located under `Program Files`, `3DQuickPress`, `PRL`, `Punch Design Template`, `Inch/Metric`, `Cutting Punch`.

Tip: You should always customize PRLs in one unit type folder. Then copy your work to the alternate unit folder and then change the units in those files with SolidWorks.

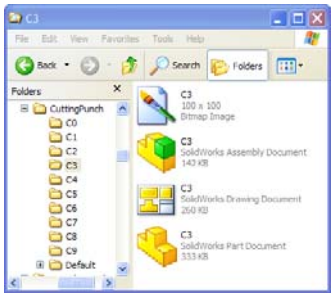


Each folder under CuttingPunch represents a different cutting punch type.

Guidelines for cutting punch files:

The folder name and top level component name must be the same; this applies to all 3DQuickPress PRLs.

Tip: For cutting punches, a good practice is to name the main punch part document the same as the folder name.

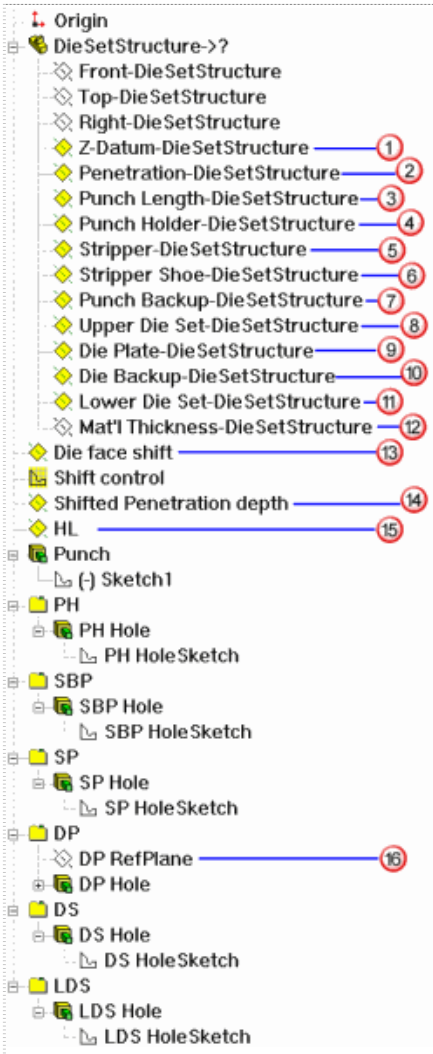


Folders should contain a bitmap file for a preview of the cutting punch during insertion. If this file does not exist, 3DQuickPress will use the SolidWorks file preview bitmap.

Cutting Punch Details

Reference planes

Cutting punches are driven by reference planes.

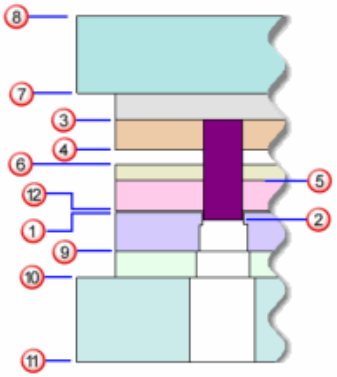


Datum planes are imported from **DieSetStructure** and used to determine the default size and holding level of cutting punches

Length of cutting punch is the distance between **(3) & (2)**

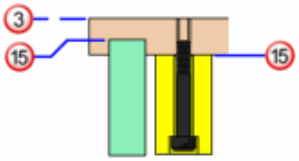
Penetration depth is the distance between die face and punch tip

Punch is retained by punch holder plate

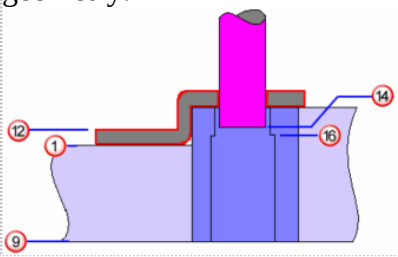


Datum planes created inside the cutting punch document are used to handle variations.

HL(15) is used to control holding level variation

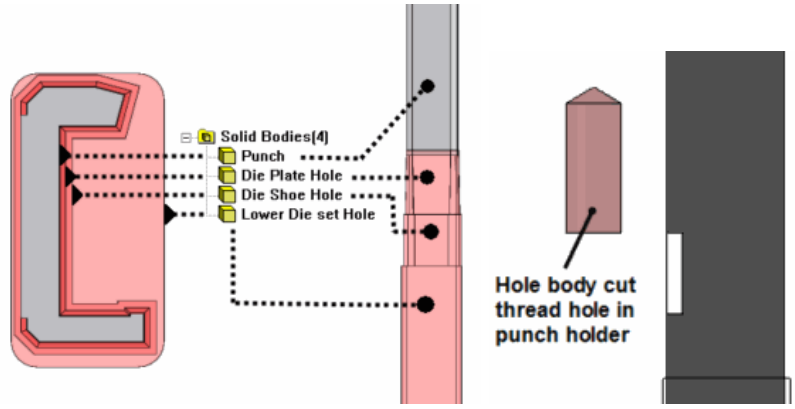


Die face shift (**13**) is used to control cutting level which depends on the part geometry.



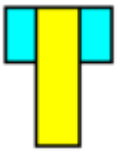
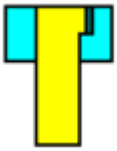
Clearance Hole bodies

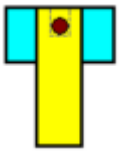
Hole bodies in a cutting punch are used to automatically cut the die plates when using Create die set Holes. These bodies will be used to cut the proper clearance for the components.



Retaining Methods and Step Punch Standards in 3DQPress

The Chart below depicts the standard Cutting Punch Retaining options that 3DQuickPress allows for custom punches. The folder name is also the name of retainer type.

Folder	**Description**	**Preview**	**Controlled by**
C0	Plain		Nothing
C1	Flange		KR
C2	Groove type by Key		KR
C3	Fixed by screw from top		Nothing
C4	Full Flange type by Key		KR

C5	Pin		KR
C6	Shoulder by Screw		XY
C7	Stepped punch		XY
C8	Screw from bottom		Nothing

The Controlled By column displays which sketch is used to control the retainer.

The retaining location and holding profile is controlled by sketches KR (Key Retainer) & XY respectively. Sketches and relationships are created on top of these 2 sketches to model retaining features like grooves and shoulders. The holding profile is controlled by XY sketch and is a rectangle by default. For example, if you want to create a round shank, you can create a circle with diameter driven by X.

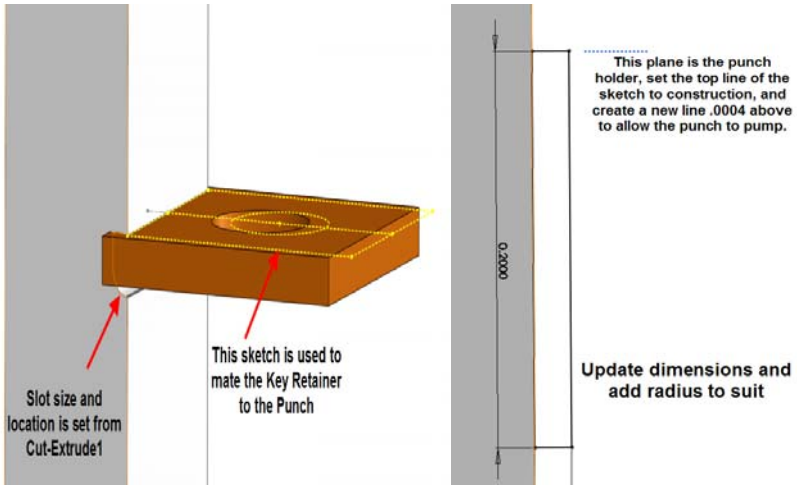
Case Study 11 – 5 : Creating a Cutting Punch

Create a New Style Cutting Punch C2 Square Keeper

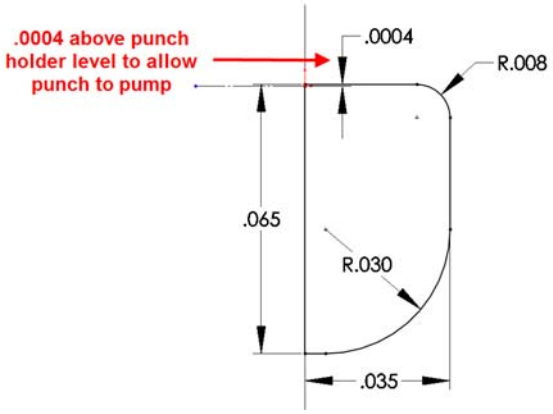
1. From the examples above select the cutting punch that is the closest to the new style. For this example we are going to use C2.
2. Start the PRL wizard 3DQuickPress, **3DQuickTools**, **Utilities**, **PRL Wizard**.
3. Click **Create New PRL** and then click **>>** button.
4. Click **Piercing Punch**, **Inch**, Input PRL name = **C2 Square** then click **>>** button.
5. Select **Cutting Punch** and then click **>>** Button.
6. Click **Copy From** select C2 and check **Open for Edit** and then click **>>**.
7. Click **Exit** button.

Shown below (Left) are the features that will be adjusted to your standards.

Shown below (Right) is the open punch in its own window and edit the Cut Extrude1

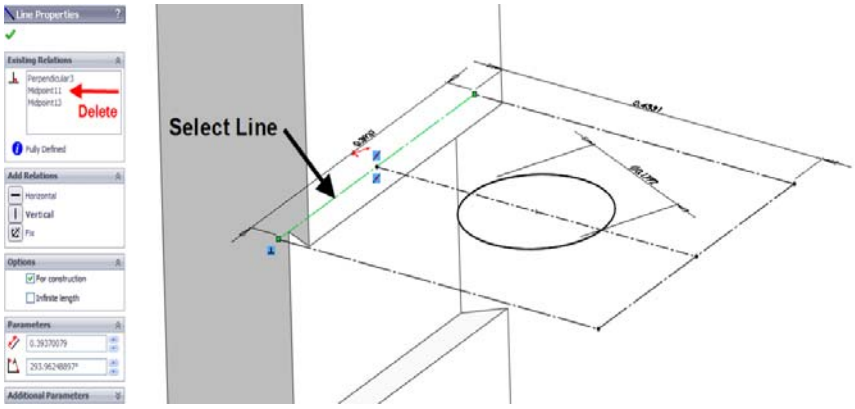


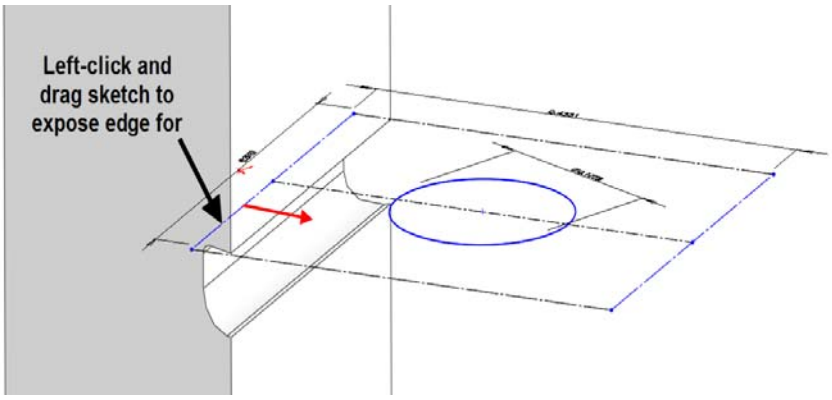
8. Right-click `Cut-Extrude1` and select **Edit Sketch** . Change the sketch to match the picture below. Change the slot height **.065in**, slot depth to **.035in**, and then add 2 radii **.03in** radius at the bottom **.008in** at the top of the slot as shown below.

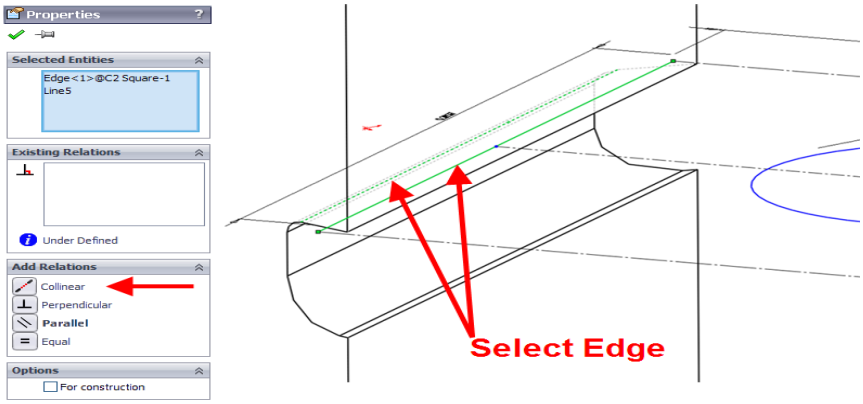


9. Right-click `Key Layout` sketch and select **Edit Sketch** . **Delete** the `Midpoint11` relations as show in the first picture below.

This will allow the sketch to be adjusted to new slot mounting as shown in the 2nd and 3rd screen shot. Add a collinear relation to sketch and slot as shown in the 3rd screenshot.

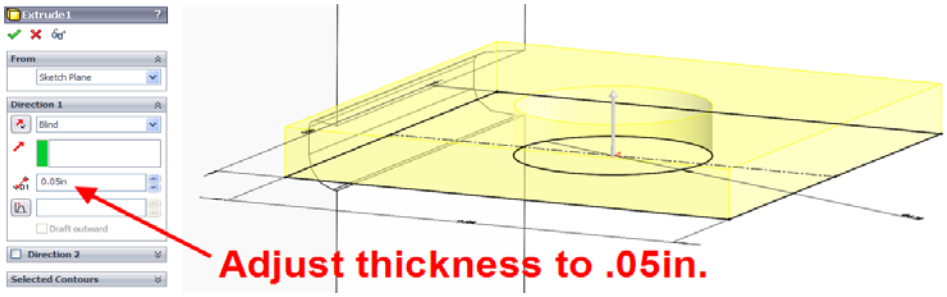






10. Exit **Edit Component** mode.

11. Open the assembly of the new C2 Square punch then adjust the thickness of the keeper (`boss extrude 1`) to **.05** inch, rebuild. **Save** and **close**.



Note: In this assembly they will be similar to standard components, therefore they will **NOT** be renamed as they are inserted and will act as instances. They are located in the following path `C:\Program Files\3DQuickPress\Standard Components\Accessories`.

<u>Create a New Style Cutting Punch C2 Round Keeper</u>

1. Select the cutting punch that is the closest to the new style. For this exercise we will use `C2 Square`.

2. Start the **PRL wizard 3DQuickPress**, **3DQuickTools**, **Utilities**, **PRL Wizard**.

3. Click **Create New PRL** and then click **>>** button.

4. Click **Piercing Punch, Inch**, Input PRL name = **C2 Round** then click **>>** button.

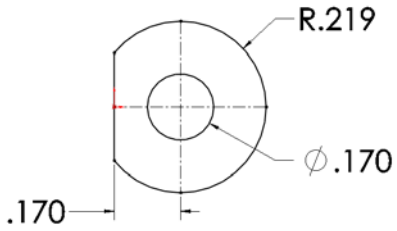
5. Select **Cutting Punch** and then click **>>** button.

6. Click **Copy From** select `C2 Square` and check **Open for Edit** and then click **>>**.

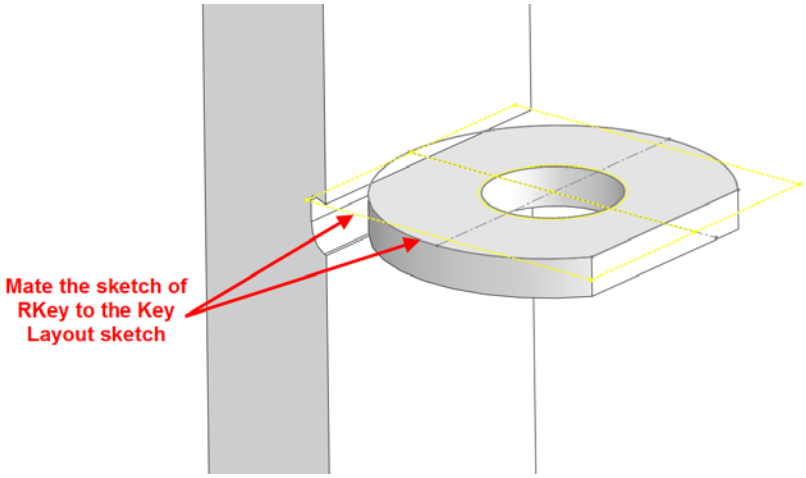
7. Click **Exit** button.

8. Because the slot is already updated, we only have to replace the keeper with a round one. **Delete** the keeper, and the mates that have errors.

9. Create new round keeper. Start a **new part** and insert a sketch as illustrated below.



10. **Extrude** the sketch, **Blind**, **.05 in**.
11. **Save** the new round keeper as `RKey` to `C:\Program Files\3DQuickPress\Standard Components\Accessories`.
12. **Mate** the `C2 Round` part sketch to the `RKey Keeper` sketch.



13. **Save** and **Close**.

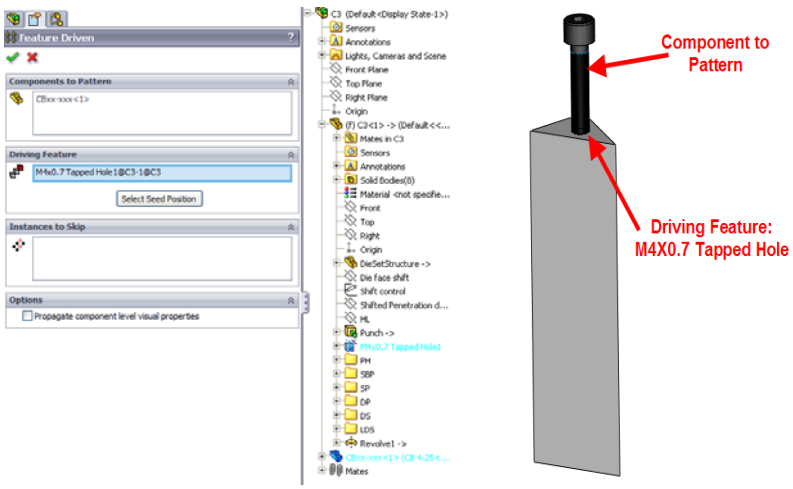
Enhance an existing Punch PRL

Modify the C3 Punch PRL to allow for a variable amount of fasteners after insertion:

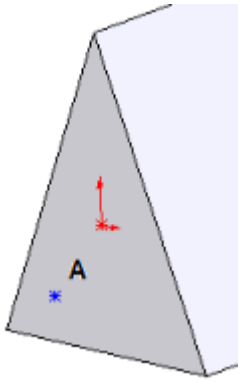
- Modifying the hole body to cover the thread hole
- Pattern the hole body
- Delete the first instance of the pattern as it overlaps with the original body
- Pattern the Socket Head Cap Screw

1. **Open** the `C3.sldasm` assembly file. File location: `C:\Program Files\3DQuickPress\PRL\PunchDesignTemplate\Metric\CuttingPunch\C3`.
2. Click **Insert**, **Component Pattern**, **Feature Driven**.

3. Select **CBxx-xxx** and **M4x0.7 Tapped Hole** for `Component to Pattern` and `Driving Feature` respectively, click ✔ to finish.



4. Switch to `C3.SLDPRT`
5. Right -click on `Sketch3` of the `M4x0.7 Tapped Hole` and click **Edit Sketch**.
6. Insert a **Point** at A and click ↵ to confirm and close the sketch.



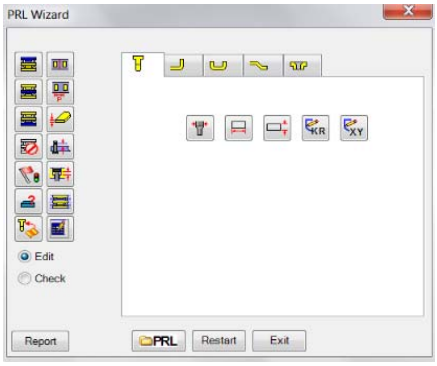
7. Exit **Edit Component** mode.
8. Switch back to the `C3.SLDASM`. The number of screw should be updated to **2**.
9. Switch back to the `C3.SLDPRT`
10. Right-click on `Sketch3` of the `M4x0.7 Tapped Hole` and click **Edit Sketch**.
11. Select `Point A` and press **Delete**.
12. **Save** and **Close** all files.

Create a New Cutting Punch

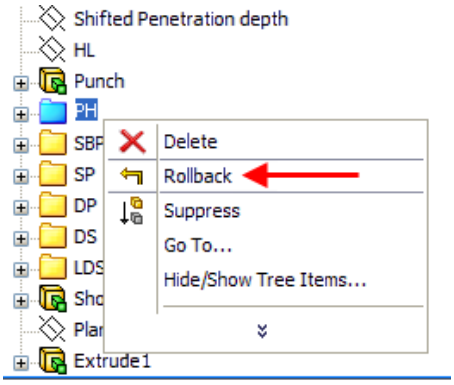
Create a new cutting punch from existing cutting punch by:

- Use Solid Works Explorer to duplicate and rename C1
- Add XY sketch
- Modify die plate hole body
- Modify Lower die set hole body

1. Click **3DQuickPress, 3DQuickTools, Utilities, PRL Wizard.**
2. Click **Create New PRL** and then click **>>** button.
3. Click **Piercing Punch**, **mm**, `Input PRL name` = **C9** and then click **>>** Button.
4. Click **Cutting Punch** and then click **>>** Button.
5. Click **Copy From** and select `C1`, click **Open for Edit** and then click **>>** Button. `C9` is open in SolidWorks and ready to add a XY sketch. Leave the PRL Wizard open.

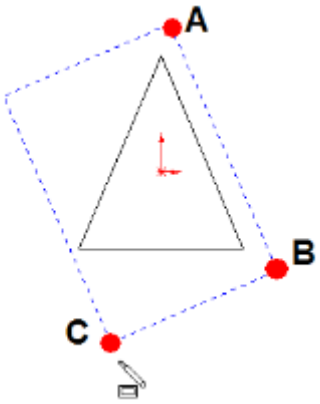


6. Right click on Folder PH in the feature tree and click **Rollback**.



7. Right-click on plane `HL` and click **Insert Sketch**

8. Click **Tools, Sketch Entities, Parallelogram**, click on A , B and then C to create a parallelogram.



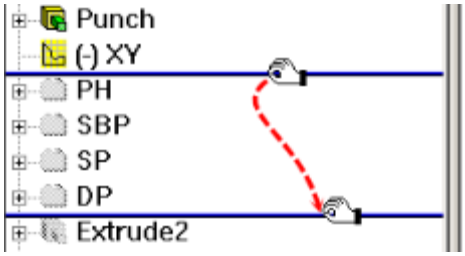
9. Click **Exit Sketch** from the confirmation corner.

10. Rename the sketch to **XY** or select the sketch in the feature tree and click the **XY button** on PRL Wizard.

11. **Close** the PRL Wizard.

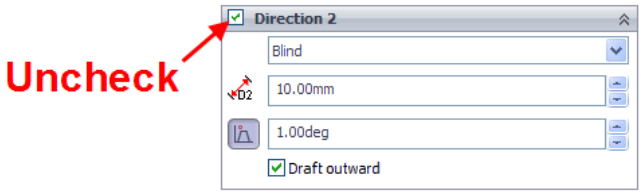
The default die plate hole is tapered and we are going to change it to stepped punch.

12. Drag the roll back bar just after folder DP

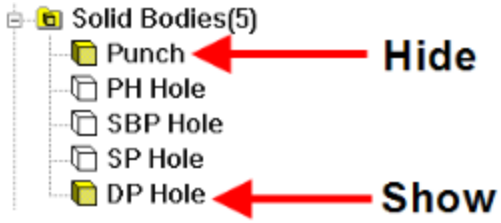


13. Right-click `DP Hole` and click **Edit Feature**.

14. **Uncheck** `Direction 2` and click to finish the command.

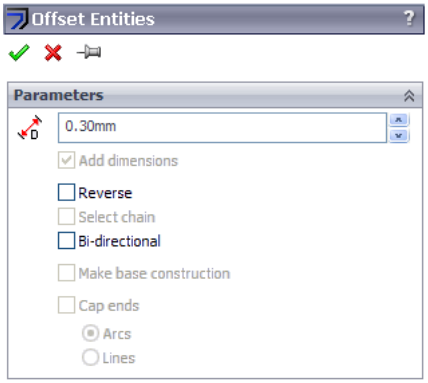


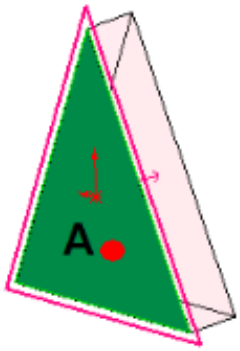
15. Expand Solid Bodies Folder, Right-click on `Punch` and select **Hide**, Right-click on `DP` and select **Show**.



16. Right-click on `DP RefPlane` and click **Insert Sketch**.

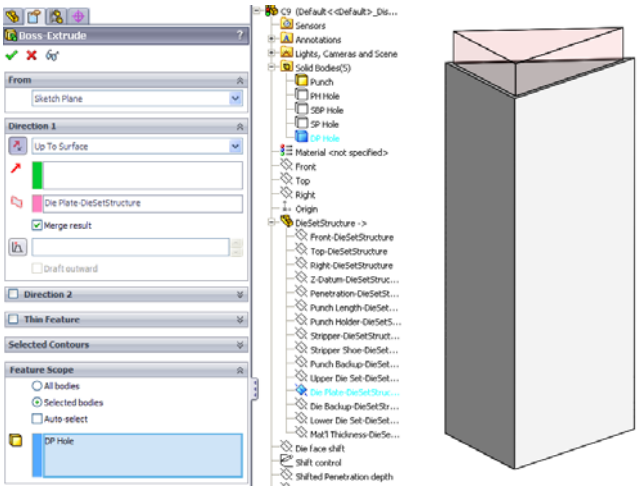
17. Select Face A and click **Offset Entities**. Input **0.3mm** for the offset distance, and make sure the offset is to the outside.





18. Click ✔ to finish the command.

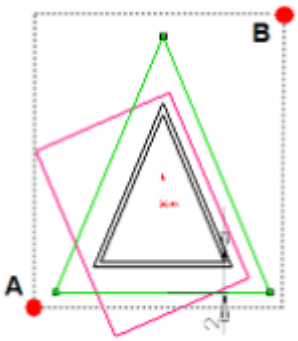
19. Click **Insert, Boss/Base, Extrude** , select **Up to Surface** for the end condition, Select **Die Plate-DieSetStructure**, check `Selected Bodies` and select the **DP Hole** body.



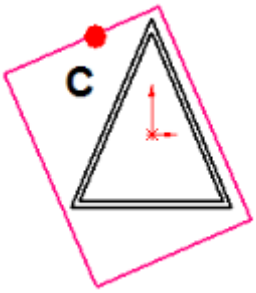
20. Click ✔ to finish the command.

Now modify the hole body of Lower Die Set. The original design is an offset of the cutting profile and now we will make use of the XY sketch to create a rectangular opening.

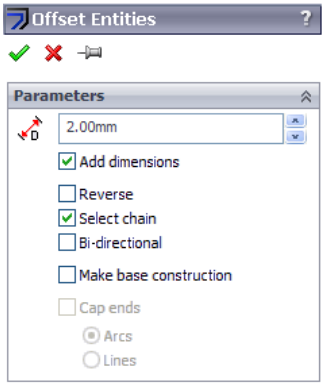
21. Right-click the roll back bar and select **Roll to End**.
22. Expand `LDS` folder, expand the `LDS Hole` feature and right-click `LDS HoleSketch` and select **Edit Sketch**.
23. Delete all entities by window selection at A, B and press **Delete** on the keyboard**.**

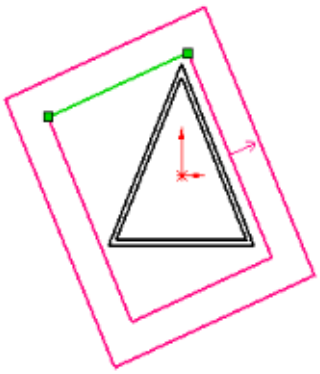


24. Select line entity from the XY sketch at C and click Offset Entity. Input **2mm** for the offset distance.

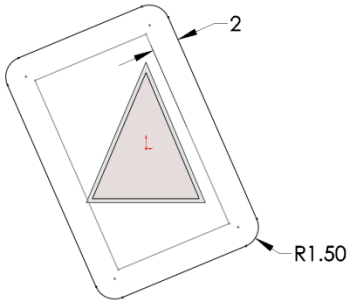


25. Check **Select chain** to offset the whole profile to the outside.





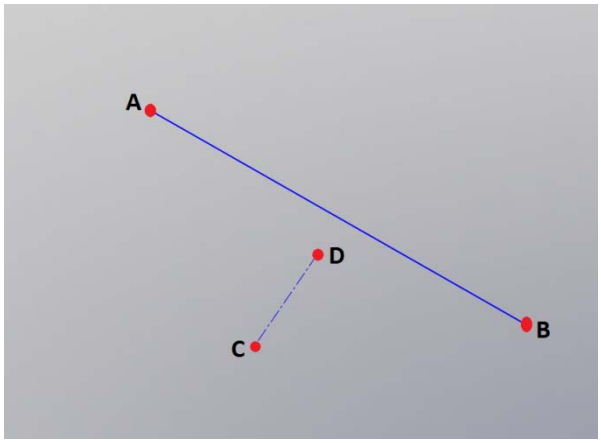
26. Click ✔ to finish the offset command.
27. Add a **sketch fillet** of **1.5mm** to all 4 corners.



28. **Exit** the sketch.

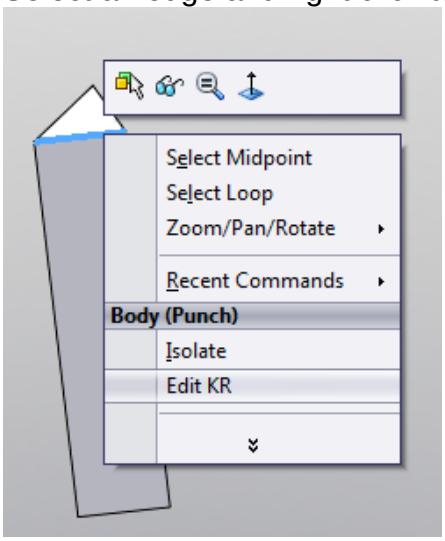
<u>Adding a shoulder to a cutting punch</u>

29. Click on plane HL and click **Insert Sketch** .
30. Set the view normal to the HL plane by clicking **Normal To**
    - Select **Line** and click on A, B to create a line.
    - Select **Centerline** and click on C, D to create a center line.

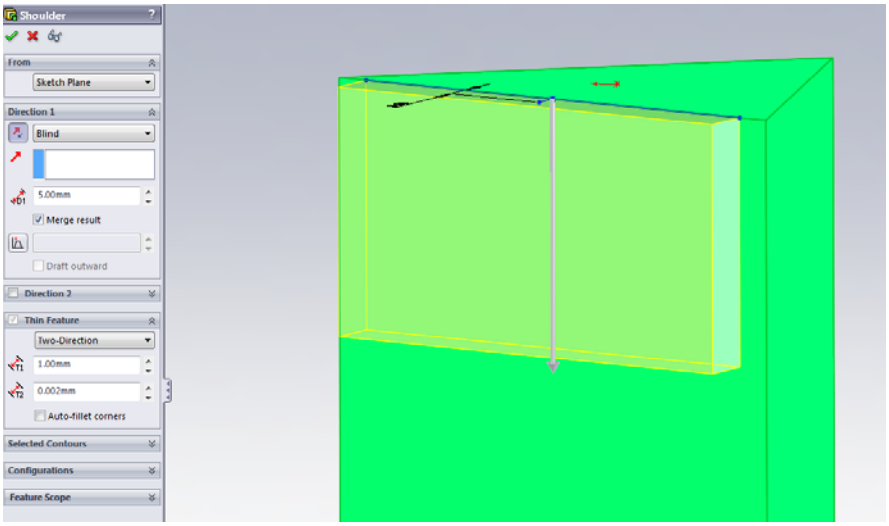


31. Click **Add Relation** ,
    - Select **line AB** and **endpoints D** and click **Midpoint**
    - Select **line AB** and **Centerline CD** and click **Perpendicular**
32. Click **Smart Dimension**, add a **0.5mm** dimension between **Point C** and the **line AB**.
33. Click **Exit Sketch** from the confirmation corner.
34. Rename the sketch to **KR** or select the sketch in the feature tree and click the **KR button** on PRL Wizard.

35. **Close** the PRL Wizard.
36. Select an edge and right click to select **Edit KR**.



37. Click **Insert, Boss/Base, Extrude** , select **Blind** for the end condition, and enter **5mm**, check `Thin Features` and select **Two-Direction** with **T1** as **1mm** and **T2** as **0.002mm**.

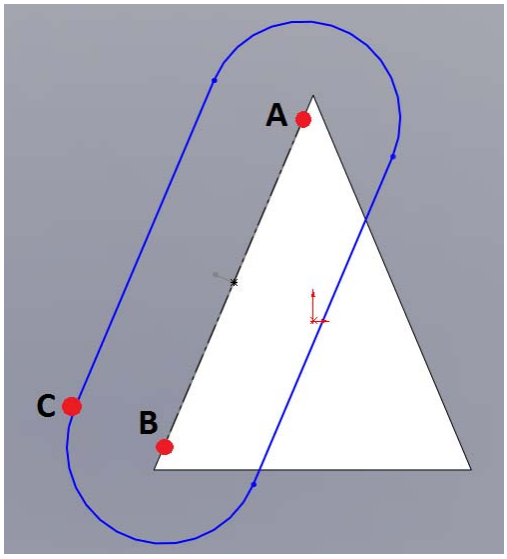


38. Click ✔ to finish the command.

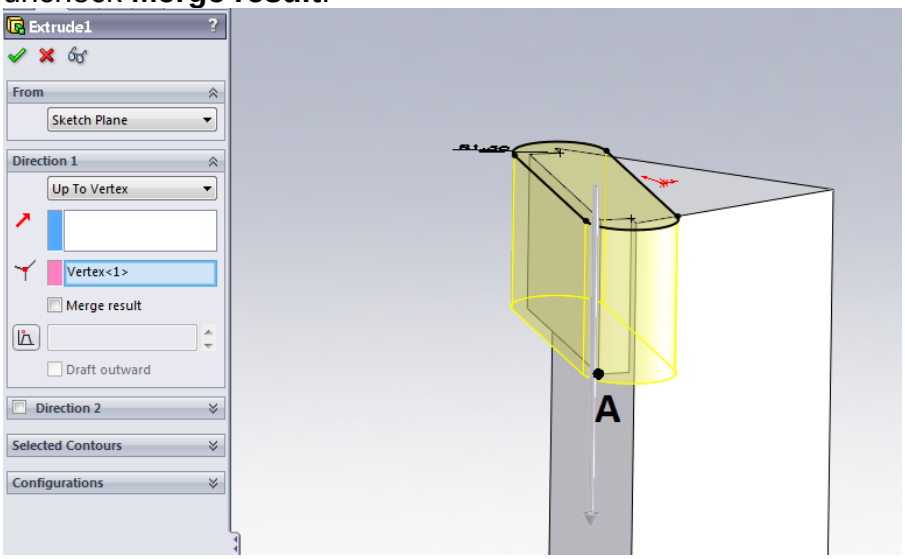
After creating the shoulder, the key slot hole body for the shoulder should be created to open the key slot in die plates.

39. Click on plane `HL` and click **Insert Sketch** .
40. Set the view normal to the `HL` plane by clicking **Normal To**

- Select **Straight Slot** and click on A, B and then C to create a slot.

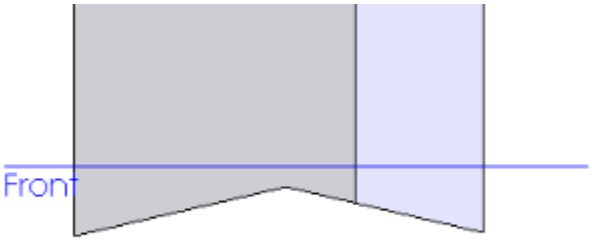


41. Click **Smart Dimension**, add a **1.5mm** dimension at the arc of **Point C**.
42. Click **Insert, Boss/Base, Extrude** , select **Up to Vertex** for the end condition, and select **Point A**, and then uncheck **Merge result**.

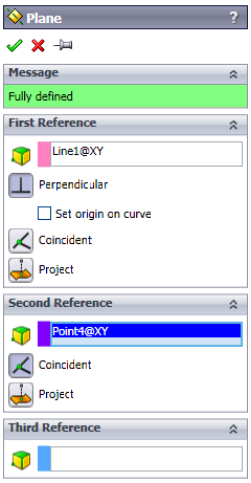


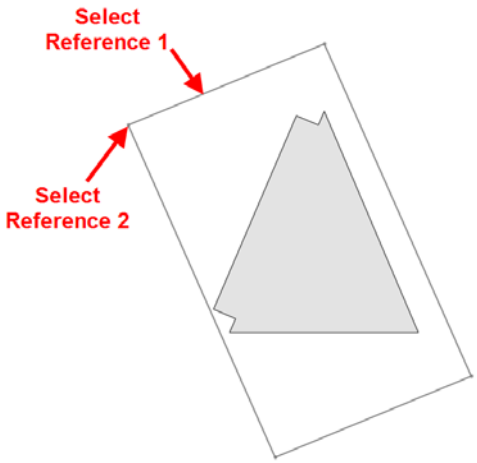
43. Click to finish the command.
44. Expand Solid Bodies Folder, Right-click on `Extrude1` and select **Hide**.

Modify the punch with shear to reduce cutting force



1. Click **Insert, Reference Geometry, Plane** and select the line entity from sketch XY for referance1 and the endpoint of the line for reference 2.





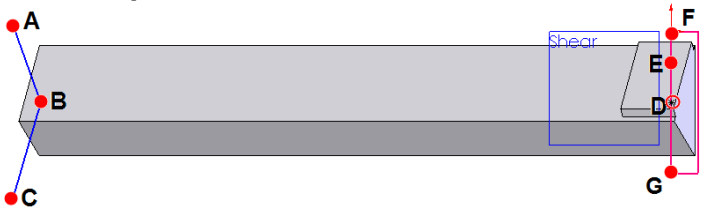
2. Click ✔ to finish the command.
3. Rename the plane to **Shear**.
4. Click on plane Shear and click **Insert Sketch** .

5. Set the view normal to the `Shear` plane by clicking **Normal To**

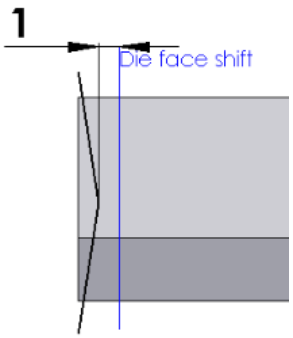
   - Select **Line** and click on A, B, C to create 2 lines.
   - Select **Point** and click on D to create a sketch point.

6. Click **Add Relation**,

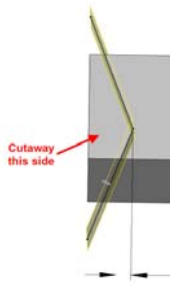
   - Select **endpoints A & C** and click **vertical**
   - Select **point D** and **endpoint B** and click **Horizontal**
   - Select **Sketch XY** and **point D** and click **Midpoint**
   - Select **endpoint A & F** and click **Horizontal**
   - Select **endpoint C & G** and click **Horizontal**



7. Click **Smart Dimension**, add a **1mm** dimension between `endpoint B` and the `Die face shift` plane.



8. Click **Insert, Cut, Extrude**. Flip the side to cut.



9. Click ✔ to finish the command.
10. **Save** and **Close** the file.

Case Study 11 – 6 : Create die set PRL

Topics covered:

- Create your own die set library based on system die set assembly.
- Create a SolidWorks Part document to hold all the Layout Sketches for the positioning standard components and holes to pass through multiple plates.
- Insert component to the die set structure based on the Layout Sketches.

Case Study 11 – 7 : Creating a die set from system template

1. Click **3DQuickPress, 3DQuickTools, Utilities, PRL Wizard**.
2. Click **Create New PRL** and then click **>>** button.
3. Click **Die Set**, **mm**, Input PRL name = **DS06** and then click **>>** Button.
4. Click **Copy From,** from the existing die sets select **00**, then click >> Button.
5. **Exit** the PRL Wizard.

Case Study 11 – 8 : Create a Layout Sketch Part for Holes Series

Layout sketch parts (common parts) should be used to maintain the hole-position for components that pass through different sub-assemblies. A typical example is a guide pin which passes through the U, M, L assemblies. By using this technique, guide pin layout sketches are modified by changing point locations, or adding and removing points to adjust quantities. The associated holes and components will be updated. The following steps are used to create 2 layout sketches, Guide Pins, and Stripper bolts in a common part.

When developing a sketch for layout purposes, it is helpful to know which location is the parent location. By placing a circle at this location in lieu of a point it is easier to locate this parent location. The circle has 2 purposes:

1. To have an idea of the hole size.
2. It is used to mate the 1st component which is going to be patterned.

Note: There is no need to insert a point at the center of the circle. This extra point will create problems when using Feature Driven Component pattern.

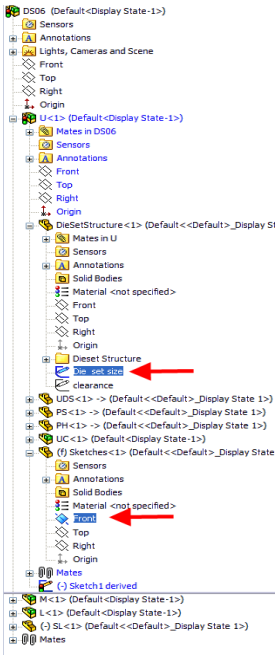
Create and add the common sketch part to all main subassemblies

1. Click **File**, **New** select the metric part template.
2. **Save** the part as **Sketches** and save in the C:\Program Files\ 3DQuickPress\PRL\Dis Set Assembly Template\DS06 folder.
3. Minimize the part window and return to the main DS06 assembly window.
4. Right-click the U assembly and select Edit Assembly.

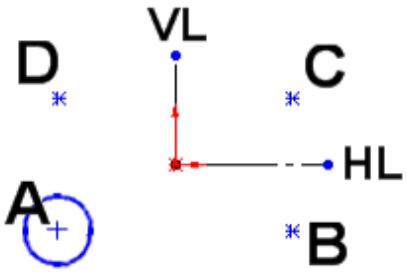
5. Select **Insert, Component, Existing Part** and select **Sketches** form the list and select ✔ to place at origin and fix part inplace.
6. **Reorder** the sketches part below the `DieSetStructure` part.
7. **Exit** edit component mode.
8. Right-click the `M assembly` and select Edit Assembly.
9. Select **Insert, Component, Existing Part** and select **Sketches** form the list and select ✔ to place at origin and fix part inplace.
10. **Reorder** the sketches part below the `DieSetStructure` part.
11. **Exit** edit component mode.
12. Right-click the `L assembly` and select Edit Assembly.
13. Select **Insert, Component, Existing Part** and select **Sketches** form the list and select ✔ to place at origin and fix part inplace.
14. **Reorder** the sketches part below the `DieSetStructure` part.
15. **Exit** edit component mode.
16. **Save** the Assembly.

Derive the "die set size" sketch into the "Sketches" part for localized reference

1. From the `DS06 assembly`, expand the feature manager tree for the `U assembly` by selecting the plus symbol.
2. In the `U assembly`, expand the feature manager tree for the `DieSetStructure` part by selecting the plus symbol.
3. From the `DS06 assembly`, expand the feature manager tree for the `sketches part` by selecting the plus symbol.
4. Right click on the `Sketches` part and select **Edit Part**.
5. Hold down **Ctrl**, and select the `Front Plane` in the `Sketches` part and `Die set size` sketch of the `DieSetStructure` part. Release the CTRL key and select **Insert, Derived sketch**. Close the sketch.

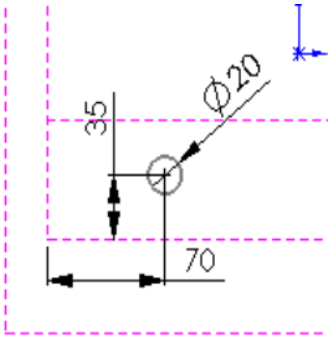


6. Exit **Edit Assembly** .
7. **Save** the assembly.
8. Right-click on `Sketches` and **Open** part.
9. Rename `Sketch1->derived` to **Die set size**. This sketch will update anytime the parent sketch is modified in the `DieSetStructure` part.
10. Right-click `Front Plane`, select **Insert Sketch** .
11. Add **vertical and horizontal centerlines** at Origin. See VL and HL below.

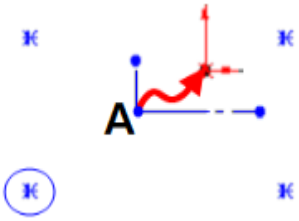


12. Sketch a **Circle** at A.
13. Create a **Point** at B.
14. Add a **symmetric** relationship. Select line VL, the Circle Center point (not circumference) & Point B and add the relationship.
15. **Mirror** Point B about line HL to create Point C.
16. **Mirror** Point C about line VL to create Point D.

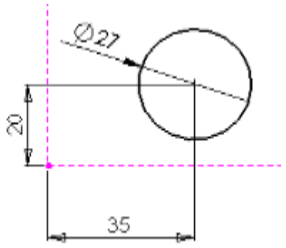
17. **Show** sketch `die set Size` and dimension the hole center as shown below.



18. **Exit Sketch** and rename the sketch to **Guide Pins**.
19. Select sketch `Guide Pins,` click **Ctrl-C** (copy), select `Front Plane` and click **Ctrl-V** (paste).
20. Rename the new sketch to **Stripper bolt**.
21. Right-click the `Stripper Bolt` sketch and select **Edit Sketch**. Constrain `Point A` to the Part `Origin` by dragging `Point A` away from the origin and then back to the origin until it picks up a **coincident** constraint.



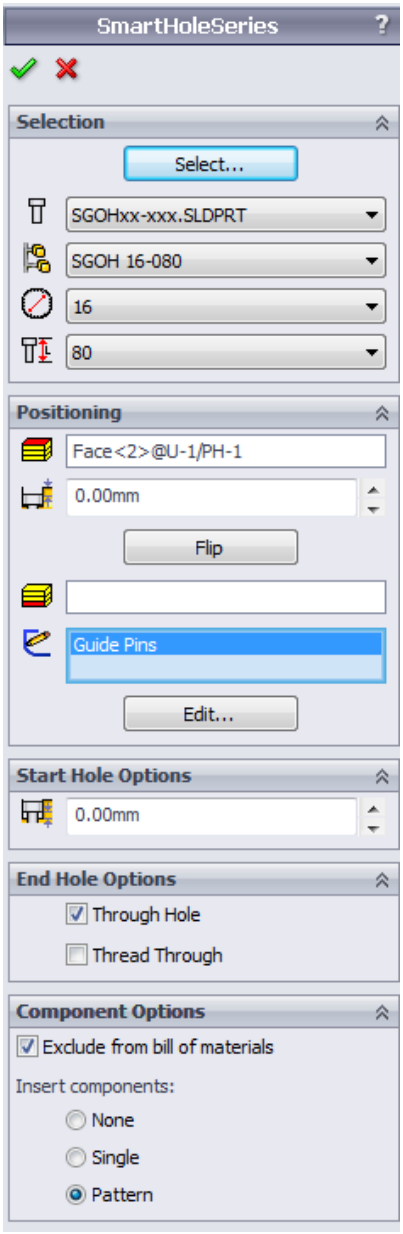
22. **Dimension** the circle in this sketch to the same left corner of the `Die set size` sketch.



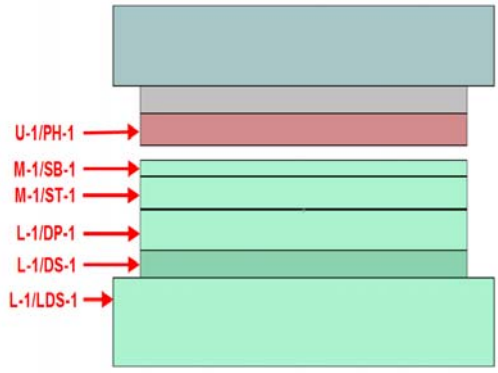
23. **Exit Sketch** .
24. **Hide** all the sketches in the feature manager.
25. **Save** and **Close** the part.

Adding Guide Pin Holes to a die set PRL

1. Open the `DS06` assembly.
2. Open `U` assembly.

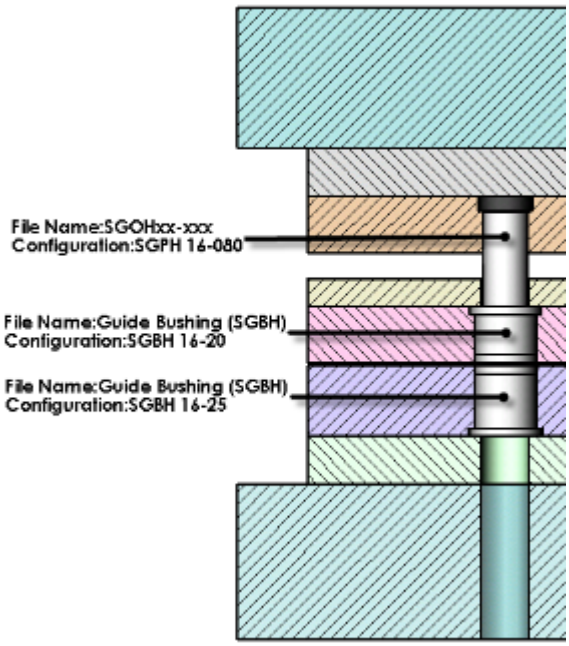


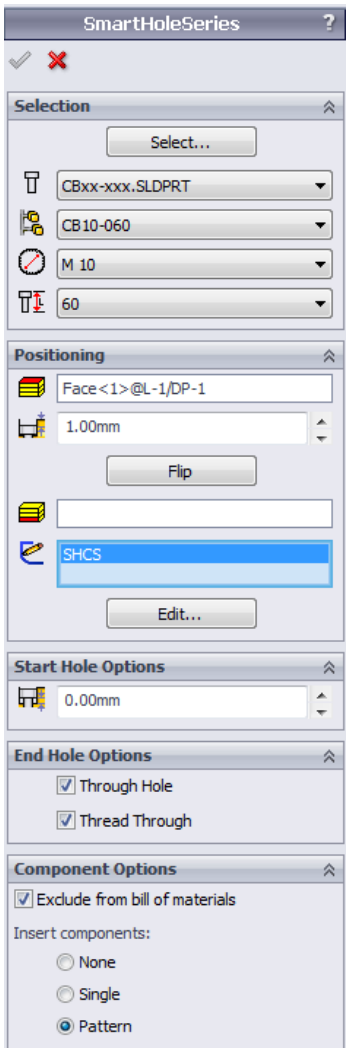
3. Click **Insert Component** from `3DQT Die Set Design` Toolbar. Select the `Guide Pins` sketch under the `U assembly, sketches` part.
4. Under `Selection` press the **Select** button and browse to `C:\Program Files\3DQuickPress\Standard Components\Misumi\Guide Pin`. Select **SGOHxx-xxx.**
5. Under `Part Configuration` select **SGOH 16-080.**
6. Under `Position` select the top face of `U-1/PH-1` and set the `distance from reference location` to **0mm.**
7. Under `End Hole Options` check **Through Hole** and uncheck **Thread Through**.
8. Press ✔ to finish the command.



9. Close `U` assembly.
10. Open `M` assembly.
11. Click **Insert Component** from `3DQT Die Set Design` Toolbar. Select the `Guide Pins` sketch under the `M assembly, sketches` part.
12. Under `Selection` press the **Select** button and browse to `C:\Program Files\3DQuickPress\Standard Components\Misumi\Guide Pin`. Select **SGBHxx-xxx.**
13. Under `Part Configuration` select **SGBH 16-20.**
14. Under `Position` select the top face of `M-1/ST-1` and set the `distance from reference location` to **0mm.**
15. Under `End Hole Options` check **Through Hole** and uncheck **Thread Through**.
16. Press ✔ to finish the command.
17. Close `M` assembly.
18. Open `L` assembly.

19. Click **Insert Component** from `3DQT Die Set Design` Toolbar. Select the `Guide Pins` sketch under the `L assembly, sketches` part.
20. Under `Selection` press the **Select** button and browse to `C:\Program Files\3DQuickPress\Standard Components\Misumi\Guide Pin`. Select **SGBHxx-xxx.**
21. Under `Part Configuration` select **SGBH 16-25.**
22. Under `Position` select the bottom face of `L-1/DP-1` and set the `distance from reference location` to **0mm.**
23. Click the **Flip** button to turn the component up.
24. Under `End Hole Options` check **Through Hole** and uncheck **Thread Through**.
25. Press ✔ to finish the command.
26. Close the `L` assembly.
27. Select the guide pin **SGOH 16-080.**
28. Click **Rebuild Holes** from `3DQT Die Set Design` Toolbar. Holes will be created on the SB-1, DS-1 and LDS-1 plates.
29. Save all documents.

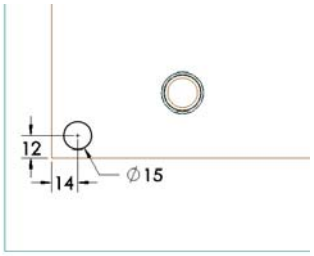




Create Fasteners for Die Set

If the components only have common holes in the subassembly, the layout sketch can be created in the assembly instead.

1. Open the `L` assembly.
2. Select the **Guide Pins** sketch from `Sketches` part and press **ctrl-c** (Copy).
3. Select **Front** plane of the `L` assembly and press **CTRL-V** (Paste).
4. **Rename** the sketch to **SHCS**.
5. Right-click the SHCS sketch and select Edit Sketch.
6. Add a **coincident** relation between Origin and the center point of the sketch by dragging the point away from the origin and dropping it on to the origin.
7. **Dimension** the sketch as shown below.



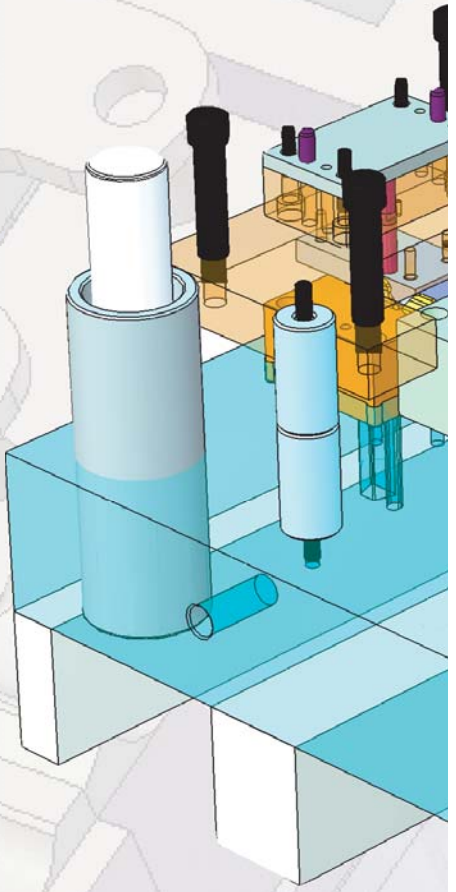
8. **Exit** the sketch.
9. Click **Insert Component** from `3DQT Die Set Design` Toolbar. Select the SHCS sketch under the `L assembly`.
10. Under Selection press the **Select** button and browse to C:\Program Files\3DQuickPress\Standard Components\Misumi\ Socket head Screw. Select **CBxx-xxx.**
11. Under `Part Configuration` select **CB 10-60.**
12. Under `Position` select the top face of `L-1/DP-1` and set the `distance from reference location` to **1mm.**
13. Under `End Hole Options` check **Through Hole** and **Thread Through**.
14. Press to finish the command.
15. **Close** and **Save** the `L` assembly.

GLOSSARY

- This Section is currently under development

3D QuickPress V5

www.3dquicktools.com



Unit 1711, Grand City Plaza, No. 1-17 Sai Lau Kok Road, Tsuen Wan, N.T., Hong Kong

Tel: +852 2788 2832 | Fax: +852 2777 6676 | Web: www.3dquicktools.com