3DQuickPress Version 5.2.1

3DQuickTools

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 .................................................................................. 1
Introduction to 3DQuickPress ................................................................ 5
  Design Environment ...................................................................... 6
  Data Flow ..................................................................................... 9
  Design Trees .............................................................................. 10
  Basic Work Flow ...................................................................... 12
  Alternate Workflow .................................................................. 14
Unfold Parts ......................................................................................... 17
  Chapter Introduction ................................................................. 18
  Automatic Unfold of Parts ......................................................... 19
  Semi-Automatic unfold of a part .............................................. 35
  Advanced Unfold Methods ........................................................ 39
  Advanced Fixed Face Options .................................................. 51
  Advanced Unfolding: User Defined Feature ................................ 55
  Advanced Unfolding Using 3DQuickForm .................................. 62
  3DQuickPress Deep Draw Calculator ....................................... 65
  Unfolding Imported STEP model ............................................. 67
Strip Layout Design ........................................................................... 70
  3DQuickPress Strip Layout introduction .................................... 71
Punch Design ......................................................................................... 94
  3DQuickPress Punch Design Introduction ................................. 95
  Manage Part File Properties Quickly with 3DQuickPress .......... 116
Create A Die Set .............................................................................. 119
  3DQuickPress Die Set Design Introduction ............................... 120
  Die Set Creation ...................................................................... 121
Detail Parts ........................................................................................ 136
  3DQuickPress Detail Functions ............................................... 137
Design Change .................................................................................. 147
  ChangE Operations in the Strip layout ..................................... 148
  My Blank Method .................................................................... 153
  Modify Original 3D Part and update 3DQuickPress Data ........ 155
Changeover Tooling ........................................................................... 158
  Changeover Tooling ................................................................ 159
Concurrent Design Environment ....................................................... 175
  Concurrent Design Environment ............................................. 176
Administration and Customization .................................................... 181
  Configure 3DQuickPress ......................................................... 182
  Set up Color Information for unfold features ......................... 187
  Set up Paint Manufacturing Information .............................. 189
  Introduction to Production Ready Libraries - PRL ............... 197
  Creating PRLs ....................................................................... 200
  Cut Punch PRLs ................................................................. 211
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 7**

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 schemes for various case studies vary 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
3DQuickPress can create a SolidWorks Assembly with Horizontal Style Design has five design stages. The documents of each stage are linked together according to the design workflow.

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 for Horizontal made during the die design stage will propagate according to the design workflow below:

- 3D Part
- Unfold
- Strip layout
- Punch design
- Die set design (Re-export and insert updated strip layout)

For design changes performed after the Horizontal die is fabricated, the design change workflow is different. For fast turnaround, it is recommended to change the effected die set component directly instead of driving the change from the ground up. If you choose to make the change from the ground up then follow the workflow below:

- Make design changes to 3D Part
- Rebuild the Unfolded part
- Open The Strip Layout and rebuild
- Open Punch design and rebuild
- Open Die set design re-export and insert updated strip layout, and rebuild
*Horizontal* design continued:

*Depending on the change of the 3D part you may experience some errors in each stage of the design process, so for the best results you may want to consider making a backup before making changes.*

*Vertical* design

3DQuickPress can create a SolidWorks Assembly with *Vertical* Style Design has four design stages. The documents of each stage are linked together according to the design workflow.

These documents are:

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

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

- 3D Part
- Unfold
- Strip layout
- Punch design

*After the design is completed an exported strip will be inserted for a non SolidWorks user to view.*
For design changes performed after the **Vertical** die is fabricated, the design change workflow is different. For fast turnaround, it is recommended to change the effected die set component directly instead of driving the change from the ground up. If you choose to make the change from the ground up then follow the workflow below:

- Make design changes to 3D Part
- Rebuild the Unfolded part
- Open The Strip Layout and rebuild
- Open Punch design and rebuild
- Re-export and insert updated strip layout into Punch design

*Depending on the change of the 3D part you may experience some errors in each stage of the design process, so for the best results you may want to consider making a backup before making changes.*
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.
3DQuickPress follows the SolidWorks user interface standard. The Property Manager 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 objects in the tree have 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 sheet metal part
2. Prepare station layout
3. Set up strip layout parameters
4. Process design
5. Punch design
6. Die set design *(This step is skipped for Vertical design)*
7. Detailing

<table>
<thead>
<tr>
<th>Workflow for 3DQuickPress</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Unfold the sheet metal part</td>
</tr>
<tr>
<td>2. Prepare station layout</td>
</tr>
<tr>
<td>Unfolded part is inserted in strip layout document and all processes are activated according to the stamping process</td>
</tr>
<tr>
<td>3. Set up strip layout parameters</td>
</tr>
<tr>
<td>4. Process design</td>
</tr>
</tbody>
</table>
5. **Punch design**
Add punch design in assembly document to create 3D punch and die.

6. **Die set design** *(This step is skipped for Vertical Assembly)*

7. **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 rework 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
- 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 a 3DQuickForm Professional or a blank can be 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.
STAGE TOOLING: This process is used to create a design for the part below.

1. **Punch Design Assembly**
   
   This is done without the 3DQuickPress unfolding feature recognition method. Instead, use the strip layout with the swap part method.

2. **Use User Defined Component (UDC) to insert Production Ready Library (PRL) for the die block**

3. **Model the form punch**
   
   Use SolidWorks & 3DQuickTools to create the die face.

4. **Die set Design**
   
   Use 3DQuickTools Assembly Utilities to finish.
Chapter 2
UNFOLD PARTS

- Automatic Unfolding
- Semi-Automatic Unfolding
- Advanced Unfolding Methods
- Unfolding Alike Features
- Unfolding Non-Uniform Thickness Parts
- Advanced Fixed Face
- 3DQuickForm
- 3DQuickForm Pro
- Deep Draw Cups
- Importing STEP models
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 UNFOLD 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.

The colors of unfolding models features can be set to easier identify in the unfolded and strip layout, go to Administration and Customization/Setup Color Information for unfolded features in this manual.

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

<table>
<thead>
<tr>
<th>Feature</th>
<th>Diagram</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear Bend</td>
<td><img src="image1" alt="Linear Bend" /></td>
</tr>
<tr>
<td>Lancing</td>
<td><img src="image2" alt="Lancing" /></td>
</tr>
<tr>
<td>Round Extrusion</td>
<td><img src="image3" alt="Round Extrusion" /></td>
</tr>
<tr>
<td>Compound Bend</td>
<td><img src="image4" alt="Compound Bend" /></td>
</tr>
<tr>
<td>Embossing</td>
<td><img src="image5" alt="Embossing" /></td>
</tr>
</tbody>
</table>
Using UAR features

There are two situations User Assist Recognition features are used in 3DQuickPress. UAR features can be used to handle unknown features that cannot be identified by automatic recognition. They can also be used to modify the behavior of the feature if the automatically recognized feature is not suitable for the process.

There are seven different UAR feature types that can be used in 3DQuickPress.

1. UAR Bend
2. Gusset
3. User Defined Feature
4. UAR Form
5. UAR Round Extrude
6. UAR Hole Cluster
7. 2 Axis Bend

Below are examples showing how to use each type of UAR feature to gain specific behavior of the unfolded part.
Case Study 2 – 1: Unfolding with UAR Features recognitions.

1. Open the part UAR_Features. File Location Desktop\Training\Unfolding\UAR Features\UAR_Features.sldprt.

2. Select **Unfold Part** from the 3DQP Unfolding toolbar.

3. Select the highlighted face as fixed face and click ✔️ to unfold the part. The sheet thickness 1mm is detected automatically. The material and the common bend allowance can both be adjusted.

4. Right-click on the Sheet Metal Object and select Unfold All.
**Grouping a series of bends with UAR Bend**

A UAR Bend feature can be used to **group** the series bend features into one **single feature**. The series of bend features can fold and unfold bends as a single feature.

1. Right-click on **Sheet Metal Object** and select **User Assist Recognition**.

2. Click Add to a UAR feature as shown in the picture to the left.
   - Set Features Type as UAR Bend.
   - Select all of the Bend Features or highlighted faces top and bottom.
   - Click Estimate Parameters, you should see updated radius and angle data to obtain the correct bend parameters.
   - For non-uniform material offset in a radius, see item # 4 thru # 7 to complete unfolding for those material ill-regular conditions.

3. Click ✔ to finish the command.
4. Non-uniform material offset in a radius, if you notice this when the model is completely unfolded and the flanges are not lying flat. Then you will need to add a sketch that represents the inside profile as shown below, straight leading into the radius, the radius, and a flat leading out of the radius is required. SolidWorks Tools\Sketch Tools\Intersection Curve works well for this process, you may need to add a sketch plane that intersects the flanged leg as needed.

5. The user can change the radius size in this sketch if needed, by adding this sketch will make it the driving information for the inside radius without re-modeling. If there are multiple radiiuses that are non-uniformed in the same part, a sketch will be required for each radius.

6. Add a UAR Bend feature for the non-uniform material bend feature or simply re-edit the UAR Bend already created, select the window noted by the icon, then select the newly to created sketch for bend as needed as shown in the example on the left. Repeat this process if the model has multiple Non-uniform material offset in a radius, Bend Allowance can also be updated as needed.

**Grouping patterned hole with UAR Hole Cluster**

A UAR Hole Cluster feature can be used to group a pattern of piercing holes. The grouped holes will become a filled flat surface when the part is unfolded.

1. Right-click on Sheet Metal Object and select User Assist Recognition.
2. Click Add to add a UAR feature.
   Set Feature Type as UAR Hole Cluster.
   Select the highlighted faces.
3. Click ✓ to finish the command.

4. A UAR Hole Cluster feature is created to represent the patterned holes.

```
<table>
<thead>
<tr>
<th>UAR Hole Cluster 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pattern 1: 12 Holes</td>
</tr>
</tbody>
</table>
```

**Grouping top and bottom coin together with UAR Form**

A coin feature will form the chamfer on the top and bottom side of the part using a UAR Form feature.

1. Right-click on Sheet Metal Object and select User Assist Recognition.

2. Click Add to add a UAR feature.

   Set Feature Type as UAR Form, select the highlighted faces. The auto selected coins will be change to a UAR feature.

3. Click ✓ to finish the command.
**UAR Round Extrude - Fixing pre-cut hole for round extrude**

When unfolding the Round Extrude 1 and Round Extrude 2 features, the pre-cut hole is closed. This is because there is no inner radius for the round extrude feature. By using the UAR Round Extrude feature can estimate parameters of the pre-cut hole.

1. Right-click on **Sheet Metal Object** and select **User Assist Recognition**.
2. Click **Add** to add a UAR feature.
   - Set Feature Type as **UAR Round Extrude**.
   - Select the highlighted faces.
   - Click **Estimate Parameters** to obtain the pre-cut hole size and other parameters.
   - Click **Extract Identical** to obtain the UAR Round Extrude 4 feature which has the same parameters.
3. Click ✔️ to finish the command.
Identifying Gusset features
Gusset features enhancement allows across bends identification of a bend feature. By identifying the gusset feature with UAR the bend feature can be recognized.

1. Right-click on Sheet Metal Object and select User Assist Recognition, as shown in the picture on the left.

2. Click Add to add a UAR feature.
   - Set Feature Type as Gusset.
   - Select the indicated Feature Faces.
   - Select the indicated Gusset Base Faces.
   - Click Auto Retrieve to collect all faces of gusset.
   - Click Extract Identical to obtain the second gusset geometry.

3. Click Add to complete the previous gusset recognition function and begin recognizing another gusset feature.
   - Set Feature Type as Gusset.
   - Select the indicated Feature Faces.
   - Select the indicated Gusset Base Faces.
   - Click Auto Retrieve to collect all faces of gusset.

4. Click ✓ to finish the command.
**Customizing blank profile of a Compound Form feature using User Defined Feature**

The **Compound Form 6** feature does not add any consideration of stretch-strain of material needed for the forming of the area. The **Compound Form 6** simply unfolds without development. By using the **User Defined Feature**, we can add develop sketch profile to create surfaces for the develop shape. The surfaces that is required for the **User Defined** for the development has been previously created for this exercise using 3DQuickForm.

![Compound Form 6](image)

1. Right-click on **Sheet Metal Object** and select **User Assist Recognition**.
2. Click **Add** to add a UAR feature, set Feature Type as **User Defined Feature**.
3. Select the highlighted faces.
4. Click **✓** to finish the command.

![User Defined Feature](image)

**Note:** The User Defined Feature is created with a question mark, this denotes that surface or blank faces of the User Defined Feature have not been set yet.

5. From the SolidWorks feature tree, **Show** the two surface bodies in the part.

![Feature Tree](image)
6. From the 3DQuickPress feature tree, Right-click on Step 0 under the User Defined 1 feature, and select Set Active.

7. Right-click on Step 0 under the User Defined 1 feature, and select Set Faces.

8. Select Surface-Offset1 as the top face and Surface-Plane19 as the bottom face.

9. Click ✅ to finish the command.

10. Right-click on Sheet Metal Object and select Unfold All.

Note: By adding a check for Extract Gussets options the gussets unfold automatically if modeled correctly.
Part Position and Orientation

Frequently, imported parts arrive in the system oriented poorly. They can be oriented poorly relative to the global coordinate system or 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).

Case Study 2 – 2: Automatic Unfolding of a Part

In this exercise, we will reorient the imported body and unfold the part using 3DQuickPress tools.

**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 Import Diagnostics to heal any bad faces. Do not proceed with feature recognition if prompted.
3. Turn on the 3DQuickPress Standard View toolbar.

![3DQuickPress Standard View](image)

4. Switch between the Front, Right and Top views using SolidWorks viewing tools.
5. Switch between the Front, Right and Top views using the 3DQuickPress Standard View toolbar. This alternate tool bar provides a better feel for how a part is oriented to the die set.
6. From the Feature Manager design tree, hold Ctrl and select both the **Origin of the part** and **Imported 1**.

7. Right-click one of the selected items and select **Zoom To Selection**. Notice how far away from the part is located to the origin.

8. Zoom in on the part to fill the screen.

9. Create a sketch on Face A and orient that face normal to the screen.

---

**Create Construction Geometry to Orient Part**

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

2. Add a Parallel relation between line A and edge C.

3. Add a Perpendicular relation between line B and edge C.

   Exit the sketch.
4. Add a coordinate system. The origin, X axis, and Y axis are shown on the picture below.

5. Add a coordinate system. The origin, X axis, and Y axis are shown on the picture below.

6. Save as Fixed Geometry1.sldprt in the desktop\Training\Unfolding\unfold1 folder.

**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 3DQuickPress Sketch Tools toolbar.

3. In the Transform to WCSYS Property Manager, 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 the Desktop\Training\Unfolding\unfold1 folder.
Unfolding the Part

1. Click the Unfold Part icon on the 3DQuickPress toolbar 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 Unfold Design Tree 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 UNFOLD 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 faces of features that create non-uniform material thickness 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

<table>
<thead>
<tr>
<th>Open End Emboss</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gussets</td>
</tr>
</tbody>
</table>

Case Study 2 – 3 : 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 End Emboss. Open End 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 Semi-Automatic.sldprt File Location: Desktop\Training\Unfolding\Unfold2\  

2. Select the fixed face as shown.

3. Click **Unfold Part** from the 3DQuickPress toolbar. 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 Unfold Design Tree and select **Unfold All**.

**Manually Recognize Features with User Assisted Recognition (UAR)**

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. Right-click **Sheet Metal Object** at the top of the Unfold Design Tree and select **User Assisted Recognition**.

2. Click **Back** for **Top/Bottom Faces** modification.

3. Check the face colors on the part to ensure all faces that should be top faces are pink and all faces that should be bottom faces are green. If any faces are not properly assigned, select those faces and pick the **Set Top** or **Set Bottom** buttons accordingly.

4. Click **Next** to add UAR (User Assisted Recognition) features.

5. Click the **Add** button, select **UAR Form** as the Feature Type, and select all faces of the open end emboss as shown below (7 faces total).
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 feature tree and select **Unfold All**. All features will now unfold.

9. **Save** and **Close** the part.
ADVANCED UNFOLD 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 – 4 : 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 Extract Alike Features UDF.sldprt File location: Desktop\Training\Unfolding\Like Features

2. Right-click on Sheet Metal Object at the top of the feature tree 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 tree and selecting Rebuild Unfolding Model.

**Patterning the Unfold**

When the same User Defined Feature (UDF) 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 3DQuickPress Unfolding toolbar.
3. In the feature tree, Right-click on **User Defined 1** and select **Set Faces to Identical User Defined Features**.

When all question marks have disappeared from the User Defined items in the tree, the function has completed. The steps and the set faces are then copied to other identical features.
4. Save and Close the part.

Case Study 2 – 5: 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

10. 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.

11. Insert a sketch on Face A.
12. Flip the part over to select Edge E on the other side of the part. Use Convert Entities on Edge E.

13. Add a split line. Select the active sketch for Sketch to Project. Select Face A for Faces to Split. Click to split Face A into 2 faces.

14. Save the part.

**Unfold the Part**

15. Click Unfold Part from the 3DQuickPress toolbar.

16. Select the face shown below for the fixed face. Click to finish the command. Select OK to accept the default bend allowance settings.

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

![Diagram of Top and Bottom Faces](image)

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

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

![User Define Feature](image)

20. Click **Next** and click **Add** to define the thin region.

21. For **Region Faces**, select the blue faces as shown in the images below.

![Region Faces](image)
22. For **Fixed Face**, select the purple face indicated below. Note: The Fixed Face is also selected as a Region Face.

![Fixed Face Image]

23. Click ✔️ to exit **User Assisted Recognition mode**. 3DQuickPress now recognizes the bends in this non-uniform part.

24. Right-click **Sheet Metal Object** at the top of the feature tree and click **Unfold All**.

25. When geometry is adjacent to more than one feature, an ambiguous condition may occur. The **Set Root** function will solve this problem. Right-click Face A, select **Set Root**, check **User Defined Root**, click Face B.

![Set Root Image]

26. Click ✔️ to exit the command.

27. Expand User Define 1 feature and right-click on step 0.

28. Select **Set Face**.
29. Select the **top** and **bottom** faces as shown and click ✓.

30. Right-click on step 0 again and select **Set Branch Feature Xfrom **...

31. Check the option Suppress Branch and click ✓.

32. **Save & Close** the part.

33. **Save** and **Close** the part.
Case Study 2 – 6: 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 lies between other features that have 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”.

Perform the Automatic Unfold

1. Open the part UDF Branch.sldprt. File Location: Desktop\Training\Unfolding\Unfold4\n
2. Select Unfold Part from the 3DQuickPress toolbar.

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 faces shown for Feature Faces

9. Press ✓ to finish the command.

10. Right-click on Sheet Metal Object and select Unfold All.

Create Surfaces for Twisted Area

The next steps will create the blank faces for the twisted area utilizing standard SolidWorks sketching and surfacing techniques. These surfaces will then be connected to the UDF.

1. Select Show Original Solid 📝 and toggle off Show Unfolding Model 📒 with the 3DQuickPress Unfolding toolbar icons.

2. Insert a sketch on the face as shown 📒.
3. Sketch a **Rectangle** of approximately 8mm length. Two corners are coincident to the vertices of the sketch face.

4. Create a **Planar Surface** from the current sketch.

5. Rotate the part view 180° and insert a sketch on the face shown below.
6. Use **Convert Entities** to convert the sketch of Surface-Plane1 onto the current sketch.

![Image](image1.png)

7. Create a **Planar Surface** from the current sketch.

![Image](image2.png)

8. Toggle on **Show Unfolding Model** from the 3DQuickPress Unfolding toolbar. Toggle off **Show Original Solid**.

9. Right-click on **Sheet Metal Object** and select **Fold All**.

10. From the 3DQuickPress Feature Tree, expand **User Defined 1** and **Single State**. Right-click on **Step 0** and select **Set Active**.

![Feature Tree](feature_tree.png)

11. Right-click on **Step 0** again 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.

![Set Faces](set_faces.png)
12. Right-click on Step 0 of User Defined 1 and select **Set Branch Feature Xform**. Select the vertices as indicated below.

13. Click ✔️ to finish the command.

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

15. **Save** and **Close** the 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 case study. 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 – 7 : Unfold with Tangent Edge

In this case study, we need to unfold to a vertex on the cylindrical fixed face that intersects the tangent arcs on the formed 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.sldprt. File Location Desktop\Training\Unfolding\Unfold5\n
2. Pre-select the curved face as shown below, and then from the 3DQuickPress toolbar 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 you tried earlier. A new selection box appears in the feature tree.

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. **Save** the part.
Case Study 2 – 8 : 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.
   
   Note: 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. 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. Hold the cursor over the edge of the hole and the spline symbol will appear next to the cursor.

4. Click Bounding Box on the 3DQuickPress Sketch Tools toolbar. Select the two splines that make up the holes.
5. **Under Geometry** select circle and under Enlarge/Shrink select **By Offset Distance** and set the distance to **0.00 mm**.

6. Click ✔️ to finish the command.

7. **Select the converted circle** that makes up the original hole. In the feature tree, check the box **For Construction**.

8. **Exit Sketch 📔** in the confirmation corner.

9. Right-click on **My Blank 1** and select **Hide Reference Surface**. If the reference plane is still shown, right-click the plane in the graphics area and select **Hide 🍎**.

10. Select **Toggle Unfolding Display 🏗️** from the 3DQuickPress Unfolding toolbar to hide the unfolded model and show the original solid.

11. **Save** and **Close** the part.
User Defined Feature is a very powerful feature in 3DQuickPress. This feature is used to handle intricate and or set by set process the manufacturing areas of a part related to forming. Through these tutorials you will learn techniques to handle the following processes:

- Forming undevelopable shapes like emboss forms
- Multiple step forming
- Form shapes with linear bends and piercings

Case Study 2 – 9: Using UDFs for Unfolding

![Image of UDFs](image)

Note: Shown in the left picture is the standard User Defined (UDF) automatically created by 3DQPress with 2 standard States (Blank, Fully Folded) inside both States there is a Step 0, also included is a Final Step within the Fully Folded State.

? On the User Define 1, indicates that no surfaces have been set.

**RED Arrows** – Arrows indicates the position of the model, if the model is folded to another State then the arrows will disappear indicating that another State will need to be created to continue.

**Step** – A step is added to an already created State when the shape surrounding UDF doesn’t change, but the shape inside the UDF feature does.

**State** – A state is added when the shape surrounding UDF changes.

1. Open the part **UAR_Features.sldprt**. File Location Desktop\Training\Unfolding\UAR Features\
2. Select User Assist Recognition, select Gusset 3 and change the feature type to a User Defined then **Unfold All**.

3. Start by creating the surfaces for the **Blank State\Step0**.

4. The area that surfaces are requires are within the model bounders, select **Patch Hole** from the 3DQPress Unfolding menu.

   Note: The block SWX selection will keep you from picking a non-exposed SolidWorks Surface.

5. Select the surrounding surface of the opening, and then select any 1 edge for the Reference Edge. 3DQPress will create the surface for you in this instance.

6. Repeat this process for the bottom of the unfolded model.

   Note: If the model is bad then there are times you will need to revert to SolidWorks to create these surfaces.

7. From the 3DQPress Feature Manger Tree scroll down to the **User Define 1\Blank State\Step0** and set faces as shown in the picture on the left.

8. Right click on **Blank State** of the UDF 1 and add a new Step.

9. 3DQPress will add a Step 1 to the already created Step 0.

10. All the faces will disappear, use SolidWorks to create the next operation.

11. Toggle to show Solid Model, by using SolidWorks you can copy faces of the emboss face to help create the next Step of the gusset.

12. Copy the surfaces Top and Bottom and offset by .000, since you will need 2 copies of this emboss area then repeat the process. Note: These surfaces have previously been created.

13. Go to the SolidWorks Feature Manger Tree and expand the surfaces folder and rename the surfaces to later identify them.

14. Toggle back to the 3DQPress unfolded part, right click on the surrounding faces, right click and export selected faces.
15. Toggle off both the SolidWorks and 3DQPress Unfolded Models.

16. This will expose the surfaces, use SolidWorks\Copy/Move to move the bottom form surfaces into position.

17. Then Loft the surfaces between the forms emboss faces Top and Bottom, toggle the 3DQPress Unfolded surface on. Note: These surfaces have previously been created.

18. Go to the 3DQPress Feature Manger Tree, UDF 1\Blank State\Step 1 right click to set faces Top and Bottom.

19. Currently the arrows are set to UDF 1\Blank State\Step 1, as we fold the outside radius the State of the model has change, and arrows has disappear.

20. Right click on the UDF 1 and create a new State.

21. Repeat steps 9 thru 15. Note: These surfaces have previously been created.

22. Expand UDF1\State 1\Step 0, right click and set faces.

23. Save and Close part.

Case Study 2 – 10: UDFs using Split Line for Unfolding

1. Open the part Split Line.sldprt. File Location Desktop\Training\Unfolding\Unfold3\ Note: Steps 2 and 3 are already completed and are listed for reference.

2. Select Sketch 1 from SolidWorks Feature Manager and click Insert, Curve, Split Line.
3. Select the highlighted faces as shown and click ✔️ to finish the command.

4. Click Unfold Part 🛠️ from the 3DQ Unfolding toolbar.

5. Select the indicated face as the fixed face.

6. Accept all default values and click ✔️ to finish the command.

7. Click OK on the Bend Allowance Parameters window.

**View unrecognized bend geometry:**

1. Right-click on Sheet Metal Object at the top of the feature tree and select Unfold All.

2. Right-click on Sheet Metal Object and select **User Assisted Recognition**.

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

4. Click ✔️ to finish the command.

5. Right-click on Sheet Metal Object and select **Unfold All**. The part should be unfolded as show below.
Import the blank surface for the free form region

1. Select **Imported Geometry** 🍂 from Insert, Features menu, and select **Blank.x_t**. File Location Desktop\Training\Unfolding\Unfold3\Blank.x_t

2. Click **Offset Surface** 🍂 from Surfaces toolbar and offset the imported blank to a distance of **1mm**.

Note: Check the direction so that it is consistent with material direction.

3. From the 3DQPress Feature Manager, right-click Step 0 and select **Set Faces**.

4. Select the imported face and the offset surface as the blank. Click ✔️ to finish the command.

5. Right-click **User Defined 1** in the feature tree and select **Fold**.
Create an intermediate step for forming – there is no limitation on the number of steps:

1. Select **Imported Geometry** 🍃 from Insert, Features menu, and select **Multi-Step.x_t.** File Location Desktop\Training\Unfolding\Unfold3\Multi-Step.x_t.

2. Click **Offset Surface** 🍃 from Surfaces toolbar and offset the imported blank to a distance of **1mm**.

3. Click ✔️ to finish the command.

4. From the 3DQPress Feature Manager, right-click **Final Step** and select **Insert Step**.

5. Right-click on **Step 1** and select **Set Faces**.

6. Select the faces as indicated, then click ✔️ to finish the command.
7. Right-click on User Defined 1 and select Fold. You can fold and unfold each step to see the entire forming sequence.
ADVANCED UNFOLDING USING 3DQUICKFORM

3DQuickForm is a finite element program to solve metal forming problems. Like all finite element software, the main steps are pre-processing; solving and post-processing. Pre-processing includes all the procedures to define the problem; the solver uses the pre-process data as input and generates a solution; and post-processing displays the interpretation of the results from the solver.

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

Geometry preparation
Offsetting surfaces
Fixing invalid surfaces
Creating mid surfaces
Pre-processing
Inputting boundary conditions
Defining initial blank
Meshing
Solving
Post-processing
Blank output
Thinning report
Stress and strain report

Case Study 2 – 11: Using 3DQuickForm form Unfolding

Define a coordinate system

1. Open the part Tutorial-1. File Location Desktop\Training\Unfolding\3DQuickForm\Tutorial-1.x_t.

2. Click Insert, Reference Geometry, Coordinate system. Click the Z axis box to make this field active, and then click Reverse Axis Direction to reverse the Z axis direction.

3. Click ✓ to finish the command.

4. Save the file as Tutorial-1.SLDPRT.
Start a new project 3DQUICKForm

1. Turn on the 3DQuickForm add-in from the Tools, Add-ins menu. 3DQuickForm will appear as a new tab at the top of the FeatureManager.

2. Right-click Tutorial-1 in the 3DQuickForm Feature Manager and select New Project.


4. Under Output Options choose Fit Bi-arc.

Note: Fit Bi-arc will remove all splines created by the unfolding and replace them with arcs and lines per the fitting tolerance.

Define punch part

1. Right click Project1 and select Set Punch Part.

2. Select the faces of the body.

Tip: Select one face on the part and use Select Body Faces to automatically collect the rest of the faces.

3. Input 1.2mm for the material thickness.

4. Click ✓ to finish the command.
**Create mesh and solve the problem**

The mesh size is determined by 2 factors:

**Global mesh size**

**Minimum mesh size**

1. Right-click **Project 1** and select **Create Mesh**.

2. Select **Run Solver after Meshing** and use the default values.

3. When the solver is running, **Processing...** will be shown on the Lower right corner of **SolidWorks**.

4. The blank is created as a sketch in the **SolidWorks Feature Tree**.

5. Right-click on **Project1** and select **Show Thinning** to see the amount of material thinning when the part is formed.
The Deep Draw Calculator is designed to estimate deep drawn results for five types of cup. It estimates the minimum number of deep drawing stages, and calculates all dimensions of the cup in each stage, by generating a table of numeric results and SWP (Macro) files. The SWP files generated from the calculator can automatically create cups in SolidWorks of each stage. With 3DQuickPress, users can easily insert them into a strip layout and rapidly proceed to punch designs and die sets.

Case Study 2 – 12 : Using the Deep Draw Calculator

1. Launch the Deep Draw Calculator from Start, All Programs, 3DQuickPress, Deep Draw Calculator.

2. Choose a Cup Type, using the <<< or >>> buttons. There are 5 different types of round cup to choose from: Straight Cup, Sloped Cup, Straight Cup (Flank), Sloped Cup (Flank), or Bi-Step Straight Cup (Flank).

3. Choose Straight Cup (Flank) for this example.

4. Set Convert Units to SI (MMKS).

5. Set the Material Properties. The Deep Draw Calculator provides 5 material types: Steel, Stainless Steel, Aluminum, Copper, or Bronze. All properties are customizable.

6. Choose Steel for this example
7. Cup dimensions can be entered under the Primary, Secondary, and Miscellaneous fields.

Note: Use the default values for Straight cup for this example.

8. Adjusting the Deep drawing suitability increases or decreases the steps required to create a drawn cup.

<table>
<thead>
<tr>
<th>Deep drawing suitability:</th>
<th>Additional steps:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poor</td>
<td>Good</td>
</tr>
</tbody>
</table>


Note: There are 3 folders of templates: Punch, Stretch Webs template, and template (No Stretch Web).

10. Choose Stretch Web Templates for this example.

11. Set a location to save the file.

12. Enter a SWB file name.

13. Select Estimate and Show Results.

14. Check to see if the results Pass.

Select Generate *.SWB Files.

15. Inside SolidWorks select Tools, Macro, Run.

16. Browse to C: \Temp and select Demo-All Steps.swb.

17. The profile of the Stretch Web can be customized by editing Sketch1 of Surface-Plane1.
Case Study 2 – 13: Importing and Unfolding a Step File

**Importing STEP file**

1. Open part file FP12.1_1. File location:
   Desktop\Training\Unfolding\STEP Model\FP12.1_1.STEP by clicking File, Open, select STEP AP203/214 (*.step,*.stp) as File Type.

2. Click **Yes** when asked to perform Import Diagnostics.

3. Select Attempt to Heal All.

4. Save and Close the part FP12_1_1.SLDPRT.

5. Start a new part.

6. Select Insert, Part.

7. Select FP12_1_1.SLDPRT.

8. Select Solid bodies for the insert part options and click ✓.

9. Right-click on FP12_1_1 in the feature tree and select List External Refs.

10. Click Lock All to temporary breaking the link between the imported part and the unfold part. When engineering changes are necessary, the link can be resumed and update the unfold part.

11. **Save** the part as FP12_1_1_Unfold.SLDPRT.
Unfold the inserted part

1. Click the Unfold Part icon on the 3DQP Unfolding toolbar to unfold the part.

2. Select the highlighted face as fixed face and click OK to unfold. The sheet thickness 0.1524mm is detected automatically. The material and the common bend allowance can both be adjusted.

3. Change the K-Factor to 0.5 for 90deg bends with bend radius equal to 0.25mm and click OK.

4. As there is an unknown feature on the feature list, that feature has to be unfolded with User Assist Recognition (UAR). Right-click on Sheet Metal Object and select User Assist Recognition.

5. Click Left Arrow to modify the top and bottom face setting.

6. Select the highlighted surface and click Unset Selected.

7. Click ✔ to finish the command.

8. Right-click on Sheet Metal Object and select Unfold All.
Chapter 3

STRIP LAYOUT DESIGN

- Strip Layout Introduction
- Feature Based Method
- Swap Part Method
- Station Layout
- Strip Layout Parameters
- Rearrange Strip Operations
- Add and Modify Strip Operations
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 not an Assembly. It is a part file that will contain the unfolded part(s) data. 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 while 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

**3DQuickPress Cutting Features**

There are four types of cutting features in 3DQuickPress: piercing, internal notching, notching and in-place cutting.

**Piercing** – a 2D cutting profile defined by the 3DQuickPress system. The piercing feature is commonly used to create internal holes on a blank.

**Internal Notching** – a 2D cutting profile defined by user. The cutting feature is created from a piercing feature.

**Notching** – a 2D cutting profile defined by user on the Z-datum. The cutting feature is created from the unprocessed feature.

**In-place Cutting** – a 2D or 3D cutting profile defined by user at any level in the die set. The cutting feature is created from the unprocessed feature.

Each cutting feature supports several operations. Depending on the type of cutting feature, the operations supported will be different. Below is a list of operations that can be used with cutting features.

**Create** – Creates a cutting feature by defining a 2D cutting profile in a sketch. The 3DQuickPress QuickSketch function can be used to help create the cutting profile. For in-place cutting features, create uses a user defined surface instead of a sketch.
**Set** – Creates a cutting feature referencing a user created planar surface. The cutting profile will follow the outside profile of the planar surface.

**Split** – Splits up a cutting feature referencing user defined split lines in a sketch. The 3DQuickPress QuickSketch function cannot be used with split operation.

**EditSketch** – Changes the cutting profile by changing the cutting profile sketch inside the split line sketch.

**Edit** – Changes the cutting profile by changing the user defined surface in the set operation or create operation (for in-place cutting feature).

**Update** – Updates the cutting profile if the user defined surface in the set operation or create operation (for in-place cutting features) is changed.

Operations supported by each type of cutting feature.

<table>
<thead>
<tr>
<th></th>
<th>Piercing</th>
<th>Internal notching</th>
<th>Notching</th>
<th>In-place Cutting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Set</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Split</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>EditSketch</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Edit</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Update</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

![Sketch is created by user.](icon)

![Surface is created by user.](icon)

![QuickSketch function can be used in the operation.](icon)

**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.sldprt` location: Desktop\Training\Strip Creation\.

2. Click Unfold Part on the 3DQuick Press toolbar. Select the back face as the fixed face. Leave the default setting and click ✓ to finish the command. Click OK on the Bend Allowance Parameters window.

3. **Save** the file.

4. Click Strip Layout Design on the 3DQuickPress toolbar. Type **T5-1 Strip Layout** for File name and Desktop\Training\Strip Creation\ for location and click **Save**.
Note: Click **No** to prevent the existing part to be inserted automatically to the new strip layout. It will be inserted manually for this case study.

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

6. From the **Open** Dialog Box, select **Tutorial-Align** file location: Desktop\Training\Strip Creation\. Now the part appears in the strip layout but needs to be properly oriented.

7. 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** degrees.

8. Click ✔️ to finish the command.

**Adding a 2\textsuperscript{nd} 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 Strip Layout Design Tree and select Insert Mirrored Sheet Metal Object.

2. From the File Open Dialog Box, select Tutorial-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 degrees.

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

   **Note:** By selecting the upper profile, 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 Property Manager, click **Clear selection**. Select **Point 1** and **Point 2**, change **delta X** to 0, **delta Y** to -15mm.

![Diagram showing Point 1 and Point 2 with annotations](image)

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**.

![Dashboard showing Edit Strip Parameters window](image)

2. To quickly set material utilization (Yield), the Optimization tools are used. Under **Optimization**, select **Fix Width** from the drop down, type **0mm** for **Width Margin** and **Station Clearance**, then click **Optimize**.

![Optimization window showing Fix Width and Station Clearance](image)
Note: 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: (.64) is reporting that 64% of material is currently utilized or 36 % 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 the strip 120mm
- Blank orientation with respect to strip 0deg
- Inter-station pitch value 38mm
- Start margin for the strip 38mm
- End margin for the strip 1mm
- Strip Y position 0mm

Note: Start margin is the extra material representation before station 1. End margin is the extra material representation after last station.

Stations:

- Number of stations of the strip 5
- Number of piercing stations of the strip 4
- Number of bending stations of the strip 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 3DQuickPress Display Options toolbar.

   **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 3DQuickPress Display Options toolbar 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 internal to a part profile that was not already recognized as a piercing. They can also be created when a separate process is needed to remove material that is not part profile specific. An example of this is splitting a punch into two punches.

**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 **QuickSketch** on the 3DQuickPress Sketch Tools toolbar. QuickSketch will automatically adjust the rectangle sketch to match the contour of the part.
3. Detail the sketch.

![Sketch Image]

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**.

![Feature Tree Image]

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) other punch sketches 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 and detail the sketch.

![Rectangle Image]
Note: 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. With the **Select** tool active, 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 the 3DQuickPress Sketch Tools toolbar and detail it as shown.

12. After adding **Centerlines** as shown, make the left end line **Coincident** to the vertical centerline, then add a **Symmetrical** relational ship to the horizontal centerline.

13. **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 Property Manager.

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 case study 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 FeatureManager design tree and select **Show All 3DQP features**.

2. Mirror the surfaces. Select **Top Plane, Surface – 3DQP Notching 1, and 3DQP Notching 2** using the **Ctrl** key. Click **Insert, Pattern/Mirror, Mirror** and click ✔.

3. To display the 3DQuickPress punch surfaces: click **Notching Punch Design Mode** on the 3DQuickPress Display Options toolbar 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 fly out FeatureManager design tree and click ✔.

6. To hide the 3DQuickPress punch surfaces: click **Notching Punch Design Mode** on the 3DQuickPress Display Options toolbar 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.
8. Right-click on all the features indicated in red and select **Activate Here** if it not already invoked.

9. **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. Xform part function will be used to rotate the part planar to the cut opening. All forming steps has been created for the user with 3DQuickform Pro software including blank development for this case study.

**Saving the Flat Blank Solid**

1. Open part file **swap part forming**. File location: Desktop\Training\Strip-Swappart\. 

2. Right-click on the **Extrude 1 body** from the solid body folder 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 3DQuickPress toolbar. Select the fixed face as shown.

2. Click ✓ to finish the command. Click OK to close the *Bend Allowance Parameters* window.

3. Select **Strip Layout Design** from the 3DQuickPress toolbar. 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

- Strip Y-Reference – **Strip Middle**
- ![ ] Input the width of the strip – **130.00mm**
- ![ ] 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 & Orient.**
- 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.sldprt** part.
2. **Hide solid bodies** Final Part and Extrude 1, and **show** surface bodies 1\textsuperscript{st} Draw Top and 1\textsuperscript{st} Draw Bottom.

3. **Return to the** swap part forming-strip layout.sldprt 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.

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

---

**Swap Part for Next Form Station**

1. Return to the swap part forming.sldprt part.

2. **Hide** surface bodies 1st Draw Top and 1st Draw Bottom, and **show** surface bodies 2nd Draw Top and 2nd Draw Bottom.


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 purple**. Set the color of the **Bottom Surfaces** to **dark purple**.

7. Click ✅ to finish the command.

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

---

**Swap Part for Third Form Station**

1. Return to the **swap part forming** part.

2. **Hide** surface bodies 2\(^\text{nd}\) Draw Top and 2\(^\text{nd}\) Draw Bottom, and **show** surface bodies 3\(^\text{rd}\) Pierce Top and 3\(^\text{rd}\) Pierce Bottom.


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 blue**. Set the color of the **Bottom Surfaces** to **dark blue**.

7. Click ✔️ to finish the command.

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

9. **Save** the part.

**Editing the Swap Part Stations**

The part will need to be rotated to allow a cut of the square opening. The part will then be pierced with an in-place cut.

1. Using the SolidWorks tools, insert a sketch on the Front plane 📹.

2. Sketch a horizontal construction line ![construction line](image) 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 🌃 using the 3DQuickPress Standard View toolbar.
3. Establish the pierce opening in station 7. Right-click on the face indicated below and select **Export Selected Faces**.

![Export Selected Faces](image)

4. Click **Hide All** on the 3DQuickPress Display Options toolbar.

![Hide All](image)

5. Insert a sketch on the surface.

6. **Convert** the entities of the hole in the center of the surface.

![Convert](image)

7. Create a **Planar Surface** bounded by the converted entities.

8. **Right-click** **Bottom Face 1** from the FeatureManager design tree and **Hide**.

9. **Click Show All** on the 3DQuickPress Display Options toolbar.
10. Right-click **Unprocessed Features** from the 3DQuickPress feature tree and select **Create In-Place Cutting**.

![Feature Tree](image)

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

![Select Cutting Face](image)

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

12. 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.

![Station 7](image)

2. Save and Close all open parts.
Chapter 4

PUNCH DESIGN

- Create a New Punch Design
- Create Cutting Punches
- Create Lancing
- Create L-Bend
- Create Pilots
- Create Stock Guides
- Translate Components
- Separate Components
- Use 3DQuickPress File Properties
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 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 case study you will take an existing strip layout part and create 2 Punch Design assembly, Horizontal and Vertical. 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.
You will find that creating a Horizontal or a Vertical Design are very similar process. 3DQuickPress will create (UC/MC/LC) sub-assemblies in the Horizontal Design were as the Vertical Design has no assemblies created.

This tutorial will be based on the following strip layout:

![Strip Layout Diagram](image)

**Starting a New Punch Design Project (Horizontal Design)**

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.sldprt**. File Location: Desktop\Training\Punch Design Creation\
2. To create the Horizontal Design the user should turn **on** the **Auto Reorganizer** from the **Special Commands** menu under 3DQuickPress\Punch Design\Auto Reorganizer, this function is a toggle **On** or **Off**.

3. Create **Punch Design Assembly** ![Icon] from the 3DQuickPress Command Manager tab.

4. Input the project information in the **Edit Project Info** Property Manager.
   - Project Name: **QPTD_H**
   - Project No.: **1**
   - Naming Convention: **1-3**
   - Die Set: **L1**
   - Punch Template: Default
   - Round Punch (Straight): **Misumi Straight**
   - Round Punch (Step): **Misumi Round Step**

5. Leave all other settings at the default values and click ✓ to finish the command.

6. The Component Clearance Property Manager should automatically open. Punch plates and Stripper plates and Die plates under the Notching & Piercing parameter. Leave all settings at the default values and click ✓ to finish the command.
Note: By un-checking any of these three items, clearance bodies will not be created for punches in the Punch holder (Upper) plates, Stripper (Middle) plates and Die (Lower) 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 to the punch for these plates. Manufacturing process can then use the nominal opening and add clearance compensation during the CAM programming stage for the true clearances.

7. From the SolidWorks FeatureManager design tree, expand QPTD-(DSS), right-click Die set size sketch and select Edit Sketch MODIFY. Change the die set dimensions according to the image below. 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.

8. Exit the Sketch and Edit Component MODIFY Mode.

9. Reorder the components in the FeatureManager design tree so that P1 Strip Layout is located below QPTD_H-(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.
Note: UC/MC/ LC are (Horizontal Design) naming convention used by 3DQuickPress. The sub-assemblies serve as component holding assemblies that represent the Upper (Punch) set, Middle (Stripper) set, and Lower (Die) set respectively.

**Starting a New Punch Design Project (Vertical Design)**

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.

10. Open the file named **P1 Strip Layout.sldprt**. File Location: Desktop\Training\Punch Design Creation\

11. To create the **Vertical Design** the user should turn off the **Auto Reorganizer** from the special commands menu under 3DQuickPress\Punch Design\Auto Reorganizer, this function is a toggle On or Off.

12. Create **Punch Design Assembly** from the 3DQuickPress Command Manager tab.
13. This time we will need to **Un-Check** the selection by **Die Set**.

![Die Set]

14. Input the project information in the **Edit Project Info.** Property Manager.

- Project Name: **QPTD_V**
- Project No.: **1**
- Naming Convention: ***_1-*3**
- Die Set: **V_1 (Metric)**
- Punch Template: Default
- Round Punch (Straight): **Misumi Straight**
- Round Punch (Step): **Misumi Round Step**

15. Leave all other settings at the default values and click ✓ to finish the command.

16. The Component Clearance Property Manager should automatically open. Punch plates and Stripper plates and Die plates under the Notching & Piercing parameter. Leave all settings at the default values and click ✓ to finish the command.

![Message]

**Note:** By **un-checking** any of these three items, clearance bodies will not be created for punches in the Punch holder (Upper) plates, Stripper (Middle) plates and Die (Lower) 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 to the punch for these plates. Manufacturing process can then use the nominal opening and add clearance compensation during the CAM programming stage for the true clearances.

**Adjust Die Set Center (Horizontal Design)**

1. From the Command Manager select 3DQuickPress QTools tab\ Modify DSS.
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.

![Adjust Die Set Center (Horizontal Design) Diagram]

**Adjust die set Center (Vertical Design)**

1. Select Translate Component from the 3DQP QTools toolbar.
2. Select the strip layout in the graphic window.
3. Change X to 25mm. Click OK to confirm the change.
Note: Just a reminder on Punch Design Selection, to create the Horizontal Design the user should turn **on** the **Auto Reorganizer** from the **Special Commands** menu. To create the Vertical Design the user should turn **Off** the **Auto Reorganizer** from the **Special Commands** menu under 3DQuickPress\Punch Design\.

**Create Cutting Punches**

1. Define the Punches, click **Punch Definition Table** from the 3DQuickPress PRL toolbar.

2. Under Selection click **All** and update the **Punch Template** column according to the image below. 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.1”, where “.x.1” 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.

**Create the Cutting Punches Automatically**

1. Click **Create Cutting Punch** from the 3DQuickPress PRL toolbar.
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 third of station 1.
3. Click **Create User-Defined Component** from the 3DQuickPress PRL toolbar.

4. Under the Definition parameters select **Rect. Blank Punch** and select 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. Right-click on **QPTD-UDC 1P** (found under the **QPTD-UC** subassembly) and click **Hide Component**.

7. Transfer the Lancing Faces from the Strip part to the Punch part for ease of design and performance. Click **Export Faces** the 3DQuickPress PRL toolbar.

8. Check the **Selection** parameter and pick **Top Faces**.

9. Select the 2 faces below, and then click **SL Features**.
10. Under Components, Click **QPTD_( )-UDC 1P** from the fly out tree.

11. Click ✔️ to finish the command.

12. Click **Hide/Show Top** 🍃 from the 3DQuickPress Display Options toolbar.

13. Right-click on **QPTD-UDC 1D** (found under the **QPTD-LC** subassembly) and click **Hide Component** 🍃.

14. Click **Export Faces** 🍃 from the 3DQuickPress PRL toolbar.

15. Check the **Selection** parameter and pick **Bottom Faces**.

16. Select the 2 faces below, and then click **SL Features**.
17. Under Components, click QPTD_() - UDC 1D from the fly out tree.

18. Click ✓ to finish the command.

There are two reasons why this 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.
19. Now you will design in the part mode to further design the Lance forming faces. Right-click QPTD-UDC 1P (found under the QPTD-UC subassembly) and click Open Part.

![Diagram of Lance forming faces]

20. If necessary, reorder Exported Sheet Metal Object Top face 1 so that it is before Extrude 1 in the FeatureManager design tree.

21. Edit Sketch 1 of Extrude 1 and change dimensions to 3mm and 10 mm. Then exit the sketch.

![Sketch dimensions]

22. Right-click Extrude1 and click Edit Feature.

23. Change the end condition to Up To Vertex, and select the vertex of the exported surface.

![FeatureManager with Up To Vertex selected]

24. Click to finish the command.

25. Right-click on the end face of Extrude 1, Insert Sketch.

26. Sketch a Rectangle, coincident to the corners of the exported surface.
27. Click **Boss Extrude**️, set the end condition as **Up to Surface**, and select **Exported Sheet Metal Object Top Face 1** as the surface.

![Boss Extrude](image)

28. Click ✔️ to finish the command.

29. **Save** and **Close** the part.

30. From the **QPTD-Tutorial-Part FeatureManager** design tree, right-click on **QPTD-UDC 1D** (found under the **QPTD-LC** subassembly) and click **Open Part**️.

![QPTD-UDC 1D](image)

31. **Edit** Sketch 2 of **Extrude 1** and change dimensions to **6mm** and **12 mm**. Then exit the sketch 🆐.

![Sketch 2](image)

32. Change to an **Isometric View**️, and insert a sketch on the top end face of **Extrude1**.

33. Change the display style to **Wireframe**️. Sketch a **Rectangle**️, coincident to the corners of the exported surface.
34. Click **Extrude Cut** , set the end condition as **Up to Surface**, and select **Exported Sheet Metal Object Bottom Face 1** as the surface.

35. Click ✔️ to finish the command.

36. **Save** and **close** the part. This will return you to the punch design assembly.

37. Right click on **QPTD_( )-UDC 1P** part (found under the **QPTD-UC** subassembly) and click **Show**

38. **Save** the assembly.

---

**Create L-Bend Assembly automatically**

1. Select **Create L-Bend** from the 3DQuickPress PRL toolbar.

2. Click the face shown in the image below to select the bend. Also click on **Compound Bending 2** inside Bending Features selection box of PropertyManager and select template **LD-01**.

3. Click ✔️ to finish the command.
Checking for Interference with 3DQuickPress tools

3DQuickPress 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.


2. From the fly out feature tree, select QPTD_( )-Compound Bending 2.1 (found under the QPTD-LC subassembly) 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 QPTD-Compound Bending 2.1 (L00).SLDPRT 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.

![Center Line](image)

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

![Rectangle](image)

10. Add a **Symmetric** relationship between lines A and C about line B.

![Symmetric Relationship](image)

11. Detail the sketch as shown below.

![Detailed Sketch](image)

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 QPTD_( ) assembly file.

Create a Pilot with a UDC

1. On the QPTD-Tutorial-Part assembly, change to a Front View.
2. Click Create User-Defined Component from the 3DQuickPress PRL toolbar.
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 3DQuickPress PRL toolbar.

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 3DQuickPress Misc. Tools toolbar. This is a collection of tools designed to make in-context assembly design easier.

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

3. Click **Translate Component** ⚙️ from the 3DQuickPress Misc. Tools toolbar.
4. The GuideLifter assembly is automatically brought in as the component to translate. 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-UDC1.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.

2. Click **Separate Component** 🍃 and input the file name QPTD-UDC 1A.SLDPR. Click **Save**.

3. Right-click the upper guide and select **Edit Part** 🍃.

4. Cut the corner with the **4mm x 2.5 mm** shape shown below.

![Diagram](image)

5. Exit edit part mode.

6. **Save** and **Close** the part.

**Note:** The original part is not modified even though the part in the assembly 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 Materials automation just to name a few. The 3DQuickPress Edit File Properties tool can also be customized to predefined 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 Exist> – 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 10, 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 \( \text{} \) from the 3DQuickPress Misc. Tools toolbar 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 the components 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 3DQuickPress Misc. Tools toolbar 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

CREATE A DIE SET

- Introduction
- Create Die Set
- Design with Layout Sketches
- Add Standard Components
- Interference and Clearance Design
- File Property Automation
- Cut Die Set Holes
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 snapshot of the current strip layout and all 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 case studies.

Case Study 5 – 1: Initiate Die Set Creation (Horizontal Design).

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

2. Click Create Dieset from the 3DQuickPress toolbar to create the die set assembly.

3. Set the die set Templates to 00.

4. Click to finish the command.

   a. Note: 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: Initiate die set Creation (Vertical Design)

1. Open the file named T5 Strip Layout-T5 Part.sldasm. File Location: Desktop\Training\Dieset Creation\T5 Strip Layout-T5 Part_Vertical\.
2. Click **Insert UDC** from the 3DQP Punch Design CommandManager tab to insert the die set assembly.

3. Select the **Die Sets** group and **00** die set template.

4. Set the link type to **None** and select the center point of the **die set size** sketch in **DSS**.

Click ✓ to finish the command. 3DQuickPress will automatically insert the components of the die based on the PRL template.
Adding standard components can be done using many techniques with SolidWorks and 3DQuickPress. This technique is a 3DQuickPress method known as a Layout Sketch 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**

Insert a new sketch in the UPPER die set assembly to locate new screws.

1. Right-click U-T5 Strip Layout-01.sldasm and select Open Assembly.
2. Select the Front Plane from the feature tree. From the 3DQuickPress Sketch Tools toolbar select Layout sketch.
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 and close the Point Tools dialog box.
7. Click Component Layout from the 3DQuickPress toolbar.
8. Under Part Select select Catalog as the Library Source. Press the Select button and browse to C:\Program Files\3DQuickPress\Standard Components\Misumi\Socket Head Screw. Select CBxx-xxx.sldprt.
10. Under Location Reference 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. Set the distance from reference location to **2mm**.
   (Note: You may need to click the Flip button if the screw preview shows the screws coming out of the plate instead of going into it.)

13. Click ✔️ to finish the command and the socket head screws will be added to the assembly.

**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 3DQuickPress Sketch Tools toolbar.

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** and close Point Tools dialog box.

8. **Save and Close** the T5Strip Layout-(DSS) part.

9. Click **Component Layout** 🌐 from 3DQuickPress Misc. Tools tool bar.

10. Under Part Select select **Catalog** as the Library Source. Press the **Select** button and browse to C:\Program Files\3DQuickPress\Standard Components\Misumi\Guide Pin. Select **SGPHxx-xxx.sldprt**.

11. Under Part Configuration select **SGPH 13-080**.

12. Under Location Reference select the face indicated below.


14. Set the distance from reference location to **0mm**.

15. Press ✓ to finish the command.

16. **Save and Close** the U-T5 Strip Layout-01 assembly window.

---

**Insert Guide Pins 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 **Component Layout** 🌐 from 3DQuickPress toolbar.

4. Under Part Selection select **Catalog** as the Library Source. Press the **Select** button and browse to C:\Program Files\3DQuickPress\Standard Components\Misumi\Guide Pin Bush. Select **SGBDxx-xx.sldprt**.
6. Under Location Reference select the face indicated below.

![Select Reference Component Face](image1)

7. Under Layout Sketch select the Guide Pins sketch from the feature tree under T5Strip Layout-(DSS).
8. Set the distance from reference location to 0mm.
9. Press ✓ to finish the command. The guide pins are added to the assembly.
10. Save and 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.
3. Click Component Layout 🤝 from 3DQuickPress toolbar.
4. Under Part Select select Catalog as the Library Source. Press the Select button and browse to C:\Program Files\3DQuickPress\Standard Components\Misumi\Guide Pin Bush. Select SGBDxx-xx.sldprt.
6. Under Location Reference select the face indicated below.

![Select Reference Component Face](image2)
7. **Under Layout Sketch** select the **Guide Pins** sketch from the feature tree under **T5Strip Layout-(DSS)**.

8. Set the **distance from reference location** to **0mm**

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 3DQuickPress Misc. Tools toolbar.

2. Select the die plate (M-T5 Strip Layout) and the stripper plate (L-T5 Strip Layout) so they are listed in the Targets selection box. Select from the fly-out Feature Manager design tree the Strip Layout part called SL-T5 Strip Layout-01 while the Tools selection box is active.

3. Click ✔ to finish the command.
4. Select **Yes** to the warning message to open the folder.

![Warning dialog]

5. Double-click on the links in the folder 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.

6. Go to the **DP-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**.

7. In the **T5 Strip Layout-01** assembly, **Clearance Tools** in the **DieSet Design CommandManager** tab.

8. Select the top face of the **DS-T5 Strip Layout-01** part file.

9. Select the interference bodies into the **Clearance entities**.

10. Set the **Geometry** to **Rectangular**.

11. Set the **Fillet Radius** to **3mm**.

12. Set the **W** to **200mm** and **L** to **3mm**, and **Depth** to **10mm**.

13. Click ✓ to finish the command.

Tip: 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 remain in the file for reference 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. Change to the top view of the Die and select the lower left corner for the Datum point.

2. Click ✔️ to finish the command.

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: These file properties will not update automatically if a part(s) changes in any way. The Component Properties Update function needs to be re-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
Create Die Set Holes (Clearance Body Automation)

Some holes and clearances are created for certain components. Upon insertion, these components are not automatically clearance 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, components that 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.

Currently the only automatic globe function to create all cavities is the following; the enhancement for the Cut Extrude is almost complete for batch mode


2. Check options PD holes and Cut by Inserts. Ensure that the Feature Scope option is set to Auto-select

3. Click ✓ to finish the command. The die set Holes will be cut automatically.
Note: All options available for using die set holes are described below:

- **PD holes** – All components (and hidden hole bodies) from the Punch Design assemblies are computed to create the holes in die set plates and inserts.
- **No ext. ref.** – Breaks associativity between the cavities’ features cutting bodies. The performance of the system will benefit if this is checked. However, if a change occurs, the die set holes will not update. They will need to be deleted and recreated unless manual clearances are created. Typically this is unchecked.
- **Cut by Std. Comp.** – Standard components (typically purchased components) will be cut by any cavities that run thru them. Typically this is unchecked.
- **Cut by Inserts** – Allows inserts to be cut in the punch and/or die set design. Typically this is checked.
- **DS holes** – In rare instances the user may need to select the DS for cutting. Typically this is unchecked.
- **Through all** – SHCS Automatic hole creation for Socket Head Cap Screws when intersecting any die set plate(s). This is a legacy (discontinued) option from version 1 and is no longer functioning unless the user has version 1 data. On newer files, the **Component Opening** command should be used to create standard components holes going forward.
- **Drill through only** – SHCS Automatic hole creation for Socket Head Cap Screws when intersecting any die set plate(s). This is a legacy (discontinued) option from version 1 and is no longer functioning unless the user has version 1 data. On newer files, the **Component Opening** command should be used to create standard components holes going forward.
- **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.
Chapter 6
DETAIL PARTS

- Smart Hole Table
- Section Line Tools
- Smart Hole Callout
- Edge Annotation
- Advanced Ordinate Dimension
- Apply Manufacturing 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.slddrw`. 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 move it onto the drawing paper using the arrow keys on the keyboard. Repeat this process for the second half of the table.

8. Below shows some of the result of the hole table created.
**Add EDM Sketch Holes to the Hole Table**

1. Select Smart Hole Table 📊 from the 3DQuickPress Drawing and Machining toolbar.

2. Select the lower left corner of the die plate as the origin.


4. Enable the Sketches and Edges option, and select Sketch44 from the graphics area.

5. Select the EDM hole sketch “Sketch 44” into the Sketches and Edges area.

6. Click ✅ to proceed.

7. Change the Tag Prefix to P.

8. Click ✅ to finish the command.

9. Move the new table onto the drawing.

10. Save the drawing.
Create 3DQuickPress Section Line

1. Select 3DQuickPress Section Line from the 3DQuickPress Drawing 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 (right click on any segment and pick Select Chain), and click Section View from the view layout toolbar.

6. Place the section view below the current drawing view.
Create 3DQuickPress Smart Hole Callout

1. Select **Smart Hole Callout** from the 3DQuickPress Drawing toolbar.

2. Select the two edges indicated below.

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

Create Edge Annotation

1. Select **3DQuickPress Mark Edge** from the 3DQuickPress Drawing 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 3DQuickPress Ordinate Dimension**

1. Select **3DQuickPress Ordinate Dimension** 📏 from the 3DQuickPress Drawing toolbar.

2. Select the vertical and horizontal side edges as the Dimension References.

3. For Selected Entities, pick in the **Select edges to insert ordinate dimensions** box to make it active. Select edges from some features.

![Image of 3DQuickPress Ordinate Dimension interface](image)

Note: The face of this part could have been selected to gather ordinate dimensions for all of the edges. However, this creates numerous dimensions that require cleanup. For the time restraints in this class, just a few edges are chosen instead.

4. Select **Four Side** for Dimension Placement.

5. Enable **Use Mfg. Info**.

6. Click ✔️ to finish the command.

7. **Save** and **Close** the drawing.
Apply Manufacturing Color Information 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\n
2. Right-click **Drawing View 1** and select **Open Part**.


4. Pick in the orange square next to **WEDM1** to make it active.

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. Continue applying manufacturing methods to the part as indicated below. Use **Set color for selected face** for Mill1 and JG1. Use **Set color for selected feature** for WEDM1, CB, and Drill1.

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

10. **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\n
2. **Pre-select** all balloons by box selecting over the entire area.

3. **Click** 3DQuickPress, Drawing & Machining, Sort Balloons.

4. **Select the BOM template**, browse to Program Files\3DQuickPress\3DQuickPress Templates\BOM Template & Click OK.

5. **Click Yes** to create and place the BOM.
6. The balloons are now in a Clockwise sequence after the 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 quantity 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** and **Close** the drawing.
Chapter 7
DESIGN CHANGE

- Change Operations in a Strip Layout
- My Blank Method for Easy 2D Changes
- Modify the Original 3D Part and Update 3DQuickPress data
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\Design Changes\

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.
5. **Rebuild** the file. The part now has rebuild errors due to deleting items that were referenced in other features. Select **Stop and Repair** on the message box to fix the error.

![SolidWorks 2010 dialog box](image)

6. A dialog box pops up to show us that the Mirror1 feature has the error. **Close** this box.

![What's Wrong dialog box](image)

7. Using the SolidWorks Feature Manager design tree, right-click on Mirror1 and select **Edit Feature**. The mirror feature was looking for the notching feature that we previously deleted. Select ✓ to finish the command and confirm that the notching body is no longer needed in the mirror.

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

9. **Delete** all the sketch entities.

10. Establish cutting clearance for the blanking punch. Select the upper blank and **Offset Sketch Entities** a distance of **0.07mm** to the inside.

![Offset Entities settings](image)

11. Click ✓ to finish the offset command.

12. **Exit Sketch**.
13. **Rebuild** the file to ensure no errors were caused by the change.

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

15. Create a blanking punch for the lower blank. Select the lower blank and **Offset Sketch Entities** a distance of 0.07mm to the inside.

16. Click ✔️ to finish offset command.  **Exit Sketch** ✅.

17. Rename Notching 1 to **Blank1** and Notching 4 to **Blank2**.

18. Right-click **Blank 1**, select **Apply To**, set to station 5. Repeat for Blank 2.

19. 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.

20. Click ✔️ to finish the command.

21. 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.

22. Click ✔️ to finish the command.
23. Right-click Xform Part 1, select Apply To, set the station to 5. Repeat for Xform Part 2.

24. 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 the original part named Tutorial-Align.sldprt location: Desktop\Training\Design Changes\. If the part is already open, use Ctrl + Tab to activate that window.

2. Modify the original part. Sketch an 8mm diameter circle on the midpoint of the rectangular opening in the part. Extrude cut, Through All.

3. Right-click Sheet Metal Object from the 3DQuickPress feature tree, and select Rebuild Unfolding Model.
4. Switch to the Strip Layout window (Blankoff.sldprt). Right-click on Strip Layout at the top of the 3DQuickPress feature tree and select Rebuild Strip Layout Model.

5. Activate any piercing geometry that may have changed, using right-click, **Activate Here**.

6. **Save** the part.
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.sldprt*. 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 listed beneath the My Blanks feature.
3. Hide the 3DQuickPress model by toggling off Show Unfolding Model from the 3DQuickPress Unfolding toolbar.
   a. **Modifying the My Blank**
   b. In this example two areas of the My Blank will be modified.
4. 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.
5. Second, fillets will be added to the corners of the tabs.
6. Right-click My Blank 2 and select Edit Sketch.
7. Hold control and select both segments of the circle.
9. Select **Bounding Box** ️ from the 3DQuickPress Sketch Tools toolbar. Under Geometry select **Circle**, and under Enlarge\Shrink select By **Offset Distance** and set the value to **0.00mm**. Click ✅ to finish the command.

10. Select the original spline segments (look for cursor feedback ⬤ to ensure you are selecting the spline), and check the box **For Construction** in the feature tree.

11. Select the new sketch circle and add a Fixed ✖️ relationship to lock it in place.

12. 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.

13. Click ✅ to finish the command.

14. **Close** the sketch ✐ from the confirmation corner.

15. **Show** the 3DQuickPress model by selecting **Show Unfolded Model** ️ from the 3DQuickPress Unfolding toolbar to see the updates made to the My Blank surface.

16. Right-click **My Blank 2** from the 3DQuickPress feature tree and select **Hide Reference Surface**.

17. **Save** and **close** the part.
MODIFY ORIGINAL 3D PART AND UPDATE 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.sldasm. File location: Desktop\Training\Design Changes\0108-Front_Plate\

2. Switch to the Front_Plate.SLDPRT window using Ctrl + Tab.

3. Click Toggle Unfolding Display 📐 from the 3DQuickPress Unfolding toolbar to display the solid model of the part.

4. Right-click on the face below and Insert Sketch 🖼.

5. Complete the sketch as shown below.
6. Click **Insert, Cut, Extrude** and set the end condition to **Through All**. Click ✔️ to finish the command.

7. 

8. Click **Toggle Unfolding Display** from the 3DQuickPress Unfolding toolbar to display the sheet metal part.

9. Right-click **Sheet Metal Object** from the 3DQuickPress feature tree and select **Rebuild Unfolded Model**.

10. Switch to the **Front_Plate Strip Layout.SLDPRT** window using Ctrl + Tab.

11. Right click **Strip Layout** from the 3DQuickPress feature tree and select **Rebuild Strip Layout Model**.

12. **Drag and drop** **Piercing 27** and **Piercing 28** from Unprocessed Features to **Station 2** to activate these features.

13. Switch to 0108-Front_Plate.SLDASM.

14. Click **Yes** when asked to rebuild the punch design.
16. **Right-click** 0108-Piercing 24.1<1> *(found under the 0108-UC assembly)* in the FeatureManager design tree and select **Hide** 📑. Then, right-click the red cutting punch surface for Piercing 27 and select **Create Punch**.

17. Right-click the red cutting punch surface for **Piercing 28** and select **Create Punch**.

18. **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
- Copy / Rename for Multiple Strips

Still In Development
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:

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.

<table>
<thead>
<tr>
<th>Section</th>
<th>Restricted IDs</th>
<th>File Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Naming Convention(DSS)</td>
<td>85, 77, 76</td>
<td>3DQP ComponentName</td>
</tr>
<tr>
<td>Naming Convention(Components)</td>
<td>All IDs are free to change</td>
<td>3DQP ComponentName</td>
</tr>
<tr>
<td>Process</td>
<td>All IDs are free to change</td>
<td>3DQP Operation</td>
</tr>
<tr>
<td>Part Type</td>
<td>DieSetStructure,SLDPRT&lt;4&gt;, 22</td>
<td>3DQP DocType</td>
</tr>
<tr>
<td>Assembly Type</td>
<td>33, 4, 8SU, 77M, 76L, 3, 85, 77, 76, 12, 11, 14</td>
<td>3DQP DocType</td>
</tr>
</tbody>
</table>
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

```
xml

<QuickPressDataDatabase>
  <Naming_Convention>
    <DSS>
      #comment System reserved
      <Name>
        <ID>85</ID>
        <My_Name>Punch</My_Name>
      </Name>
      <Name>
        <ID>77</ID>
        <My_Name>Stripper</My_Name>
      </Name>
      <Name>
        <ID>76</ID>
        <My_Name>Die</My_Name>
      </Name>
      <Name>
        <ID>3DQPComponentName</ID>
        <My_Name>Rail</My_Name>
      </Name>
      <Name>
        <ID>3DQPComponentName</ID>
        <My_Name>Rail</My_Name>
      </Name>
      <Name>
        <ID>System reserved</ID>
        <My_Name>3DQPComponentName</My_Name>
      </Name>
    </DSS>
  </Naming_Convention>
</QuickPressDataDatabase>
```

3. The files in the DSS are restricted ID's.

4. Expand the Component folder.

```
<Naming_Convention>
  <DSS>
    <Components>
      #comment 3DQPComponentName
      <Name>
        <ID>Special case for standard part, use isfastener</ID>
        <My_Name>Special case for standard part, use isfastener</My_Name>
      </Name>
    </Components>
  </DSS>
</Naming_Convention>
```

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

5. Expand the plates folder

```
<Plates>
  <Name>
    <ID>UPParallel</ID>
    <My_Name>UPParallel</My_Name>
  </Name>
  <Name>
    <ID>Upper Parallel</ID>
    <My_Name>Upper Parallel</My_Name>
  </Name>
  <Name>
    <ID>Punch Backup</ID>
    <My_Name>Punch Backup</My_Name>
  </Name>
  <Name>
    <ID>Punch Backup</ID>
    <My_Name>Punch Backup</My_Name>
  </Name>
  <Name>
    <ID>Punch Holder</ID>
    <My_Name>Punch Holder</My_Name>
  </Name>
  <Name>
    <ID>Punch Holder</ID>
    <My_Name>Punch Holder</My_Name>
  </Name>
  <Name>
    <ID>Stripper Backup</ID>
    <My_Name>Stripper Backup</My_Name>
  </Name>
  <Name>
    <ID>Stripper Backup</ID>
    <My_Name>Stripper Backup</My_Name>
  </Name>
</Plates>
```
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.

**Add a New Plate to the Group.**

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**
Launching the COT Manager

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 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 1\textsuperscript{st} 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 multiple details after a design has been completed, the detail number will be entered into the box, if searching for multiple details add a comma between them, by selecting search, 3DQPress will isolate the detail.

![Find Tab](image)

Utilities Tool Bar

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

![3DQP Tools](image)

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**.

**3 different processes are handled**

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. If a property has been already set, features will highlight inside the Set Component Information.

3. If nothing highlights 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.

![Component Name](image)

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 saved.

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 2\textsuperscript{nd} die set design into the new Main Assembly.
4. Select the refresh icon on the COT Utilities bar.

5. After refreshing the 2\textsuperscript{nd} die assembly will be setup in the COT Manager.
Chapter 9
CONCURRENT DESIGN ENVIRONMENT

- Set up a Concurrent Design
- Understand the Concurrent Design
- Add Designers to a project
- Remove Designers from a project
- Combine Designers
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 The Environment

This case study sets up and manages a concurrent design process. The project is starting 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 3DQuickPress 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 Straight
  - 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

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

   
   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 \( \rightarrow \) 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 \( \rightarrow \) 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 \( \checkmark \) to finish the command.

Deleting Users

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

2. Select the sub-punch design that will be deleted.

3. Click \( \checkmark \) to finish the command.

Combining Users (Sub Designs)


2. Select the two sub-punch designs that will be combined.

3. Click \( \checkmark \) to finish the command.
Chapter 10
ADMINISTRATION AND CUSTOMIZATION

- Adjust File Behaviors
- General Environment Setup
- Introduction to PRLs
- Create PRLs
- Create Die Sets PRLs
CONFIGURE 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 3DQuickPress 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 3DQuickPress 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 3DQuickPress Misc. Tools 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 3DQuickPress Misc. Tools 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

<3DQuickPress installation 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 10 – 1: How to customize SolidWorks File Properties inside 3DQuickPress

Add a Group for Specific Customer Related Properties

1. Click **New** and start a new part file using any template.

2. Click **File Properties** from the 3DQuickPress Misc. Tools toolbar to launch File Properties Editor.

3. Click **Edit** button

4. Add the following lines in Note Pad.
   - #Customer Info.
   - Customer Part No., text,
   - Customer name, text,
   - Description, text,

5. Save and exit notepad.

6. Click **Exit** button to exit File Properties Editor.

7. Click **File Properties** from the 3DQuickPress Misc. Tools toolbar to launch File Properties Editor re-initializing the group file.

8. Click the dropdown under Grouping and Customer Info will now be listed.
SET UP COLOR INFORMATION FOR UNFOLD FEATURES

Feature Color Information

3DQuickPress can color code the features inside the model when the unfolding process begins. These can be customized to your specifications, the color coding of part faces is extremely helpful when the unfolded part is used inside the strip layout.

Where to find it: **3DQuickPress, Options, Color Setting**

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

Colors of unfolding sheet metal object top and bottom faces should not be change because it follows the same color as setting top and bottom faces inside user define.
Matching the colors of the unfolded and the strip layout will help with consistency.
**SET UP PAINT MANUFACTURING INFORMATION**

*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.
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\3DQuickPress_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.

<table>
<thead>
<tr>
<th>R</th>
<th>G</th>
<th>B</th>
<th>Name</th>
<th>Description</th>
<th>Precision</th>
<th>Precision(In.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>#128</td>
<td>128</td>
<td>255</td>
<td>Drill1</td>
<td>Pecking Drill</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>#255</td>
<td>128</td>
<td>255</td>
<td>CB</td>
<td>CounterBore</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>#128</td>
<td>64</td>
<td>64</td>
<td>Mill1</td>
<td>Pocket Mill</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>#255</td>
<td>128</td>
<td>0</td>
<td>WEDM1</td>
<td>WireEDM 2 cut</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>#255</td>
<td>255</td>
<td>255</td>
<td>JG1</td>
<td>Jig Grind</td>
<td>3</td>
<td>5</td>
</tr>
<tr>
<td>#228</td>
<td>168</td>
<td>27</td>
<td>EDM</td>
<td>EDM</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>#235</td>
<td>219</td>
<td>163</td>
<td>Reaming</td>
<td>Ream</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>#0</td>
<td>255</td>
<td>255</td>
<td>Grinding</td>
<td>Grinding</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>#255</td>
<td>43</td>
<td>43</td>
<td>Tapping</td>
<td>Tapping</td>
<td>1</td>
<td>3</td>
</tr>
</tbody>
</table>
Case Study 10 – 2 : Setting up Mfg. Color Info.

**Customize Color Codes**

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

![Color Settings Window](image)

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.

![Confirmation Dialog](image)

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.

When setting Mfg information to a sketch, follow the below process:

**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).sldprt. File location: C:\Program Files\3DQuickPress\Standard Components\Danly\Guide Posts\GPost

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

3. Select **Hole** sketch in the FeatureManager design tree, and select **WEDM1** in the Mfg Information dialog box. Click **Set color for selected sketch** to assign color for the sketch.

4. **Save** and **Close** the file.

**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.sldprt File location: C:\Program Files\3DQuickPress\PRL\PunchDesignTemplate\Metric\User Defined\Inserts\INS1\n

3. Select the **Hole Body** in the feature tree and select **WEDM1** in the Mfg Information dialog box. 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 manufacturing color information 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.

![Image of 3DQuickPress interface showing Mfg Color Tool]

**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 PRL toolbar, QP Misc. Tools, Mfg Color Tool.

![Image of 3DQuickPress toolbar highlighting 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 3DQuickPress 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:

<table>
<thead>
<tr>
<th>Show folders for:</th>
<th>Folders:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Library</td>
<td>C:\Program Files\3DQuickPress\PRL\Library Features</td>
</tr>
<tr>
<td>Document Templates</td>
<td>C:\Program Files\3DQuickPress\3DQuickPress Templates</td>
</tr>
</tbody>
</table>
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 can be set outside of SolidWorks by using the following process. You can find this under Programs, 3DQuickPress, 3DQuickTools, Utilities, File Properties

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Type</th>
<th>Value</th>
<th>Delete</th>
</tr>
</thead>
<tbody>
<tr>
<td>UC/MC/LC</td>
<td>Text</td>
<td>UC</td>
<td></td>
</tr>
<tr>
<td>UpdateFlag</td>
<td>Yes or no</td>
<td>Yes</td>
<td></td>
</tr>
<tr>
<td>DP</td>
<td>Yes or no</td>
<td>Yes</td>
<td></td>
</tr>
<tr>
<td>INS</td>
<td>Yes or no</td>
<td>Yes</td>
<td></td>
</tr>
<tr>
<td>F-SH</td>
<td>Yes or no</td>
<td>Yes</td>
<td></td>
</tr>
<tr>
<td>INS</td>
<td>Text</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**UC/MC/LC** - You can control where Punch PRLs will be inserted using this property if using a Horizontal assembly. 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) – This property is for extra small PRL’s with a retaining heel.

**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) – inserts in the lower die block.

**XSL** (Yes|No) – This property is usually set to a PRL that will form or coin material, when the PRL is inserted this property will automatically export the selected current snap shot of surface faces of the strip layout to part bodies in the PRL. This functionality minimizes unnecessary assembly associativity to reduce rebuild times in SolidWorks.

**3DQuickPress 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.
Dimension’s Names Used by PRL’s

Any dimension can be rename to follow information leveraged by 3DQuickPress, a list of those names are as follows.

1. Strip Layout
   - Progression
   - Stock Width

2. Cutting Punch Sketch Name
   - PH HoleSketch – C6 of the DSS
   - SBP HoleSketch – C5 of the DSS
   - SP HoleSketch – C4 of the DSS
   - DP HoleSketch – C3 of the DSS
   - DS HoleSketch – C2 of the DSS
   - LDS HoleSketch – C1 of the DSS

3. Adding <A(0.xx )> as shown to the end of any sketch supports updating land and taper angle of the Modify DSS.

4. Adding <T(0.xx )> as shown to the end of any sketch supports Taper that will not update to the Modify DSS.

5. PRL’s Dimensions.
   - OverallX
   - OverallY
   - ShankDia
   - Lifter Level
   - Stripper Stroke
   - Lower Parallel
   - Custom dimensions using Param.def

   - OverallX-X
   - OverallY-Y
Case Study 10 – 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. DieSet Structure allows the user to select specific Die Set Structure for the PRL that is supported to be inserted.
5. Specify the Functional Group for this PRL. Select **Existing** and select **Rect. Blank Punch** and then click >> Button.

![Functional Group Selection](image)

6. 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.

![Part Selection](image)
7. 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:\ProgramFiles\3DQuickPress\PRL\PunchDesignTemplate\Metric\User Defined\Rect.Blank Punch\.

**Create the Initial Geometry**

1. Select the reference plane named: **Punch Length-DieSetStructure**.

2. From the SolidWorks Menu, select: **Insert, Reference Geometry, Plane**. Check the **Offset distance** option.

3. Check the direction of the offset to assure it is in the direction of the punch.

4. Set the distance of the offset to **0.00mm** and click ✔ to finish the command.

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

![Insert Die Set Structure (DSS) icon and screen shot of PRL Wizard]

6. Right-click HL plane and Insert Sketch.

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

8. 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 Retaining Geometry

1. Show Base Sketch in the feature tree by expanding the feature Boss-Extrude1. 
   Note: You may also want to hide the extruded body and work with the sketch. Open the PRL Wizard for ease of setting Sketch Planes to suit.

2. Right-click Base Sketch, and select the eye glass icon.

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

6. Check Show all dimensions.

7. Select dimension 10 and click OverallX.

8. Select dimension 8 and click OverallY.

9. Under the Container property, select UC from the drop-down menu and click the Set button.

10. Under the Update Flag property, select Yes from the dropdown menu and click the Set button.

11. Click the Report button to check that all parameters have been set. Click OK.
12. Click the ✓ to finish this operation.

13. 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.


15. Select Edit Current Model and then click the >> button.
16. Select **Insert PRL Param.def file**\[2\], 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.

17. **Save** and **close** the text file, and **Exit** the PRL Wizard.

18. Select **3DQuickPress, Punch Design, Define PRL**, repeat steps 5 thru 11 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 7 & 8 but the newly added dimensions.

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

20. Right-click on the drawing and go to **Properties**. Set the projection to **Third Angle** and **1:1** scale.
On the View Layout toolbar, select **Model Views**, select R01 from the open documents window, then click the **Next** button. Check the option **Create multiple views**, then select both the **Right** and the **Bottom** views.

![Image of View Layout toolbar](image)

21. Click **Insert**, **Block** and insert the XYPos. File location: C:\training\annotations\XYPos.sldblk block to the drawing.

22. **Save** the drawing as R01.sldrw under the same folder as the part.

23. Return to the R01 part, set the part configuration back to Default.

24. **Save** all files.
Case Study 10 – 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**

2. Click Create New PRL and then click >.
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, Rect. Blank Punch and select R01. Check Open for Edit and then click >.
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 Boss-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 the 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 >>.

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.
CUT PUNCH PRLS

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 3DQuickPress

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.

<table>
<thead>
<tr>
<th>Folder</th>
<th>Description</th>
<th>Preview</th>
<th>Controlled by</th>
</tr>
</thead>
<tbody>
<tr>
<td>C0</td>
<td>Straight</td>
<td><img src="#" alt="Image" /></td>
<td>Nothing</td>
</tr>
<tr>
<td>C1</td>
<td>Single Flange</td>
<td><img src="#" alt="Image" /></td>
<td>KR</td>
</tr>
<tr>
<td>C2</td>
<td>Keeper Slot type</td>
<td><img src="#" alt="Image" /></td>
<td>KR</td>
</tr>
<tr>
<td>C3</td>
<td>Keeper Slot style with Key</td>
<td><img src="#" alt="Image" /></td>
<td>KR</td>
</tr>
<tr>
<td>C4</td>
<td>Full Flange type</td>
<td><img src="#" alt="Image" /></td>
<td>Double KR</td>
</tr>
</tbody>
</table>
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 10 – 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.
3. Click Create New PRL and then click >.
4. Click Piercing Punch, Inch, Input PRL name = C2 Square then click >.
5. Select Cutting Punch and then click >.
6. Click Copy From select C2 and check Open for Edit and then click >.
7. Click the 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

9. Right-click Cut-Extrude1 found under the C2 part 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. Exit the sketch.

10. Right-click Key Layout sketch and select Edit Sketch 📖. Delete the Midpoint11 relation as show in the 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.
11. Exit **Edit Component** mode.

12. Right-click on the feature **Extrude1** feature in the **FKey** component, select **Edit Feature**. Change the Depth of the extrude to be **0.05mm**. Exit **Edit Component** mode.

13. **Rebuild** the file and then **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.

**Create a New Style Cutting Punch C2 Round Keeper**

1. Select the cutting punch that is the closest to the new style. For this case study we will use C2 Square.
3. Click Create New PRL and then click >>.
4. Click Piercing Punch, Inch, Input PRL name = C2 Round then click >> button.
5. Select Cutting Punch and then click >>.
6. Click Copy From select C2 Square and check Open for Edit and then click >>.
7. Click Exit.
8. Because the slot is already updated, we only have to replace the keeper with a round one. **Delete** the FKey component, and the mates that have errors.

![Sketch](image)

10. **Extrude** the sketch, **Blind**, .05 in.
11. **Save** the new round keeper as RKey to C:\Program Files\3DQuickPress\Standard Components\Accessories.
12. **Insert** the RKey file into the C2Round assembly. **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. Expand the C3 part in the FeatureManager design tree.
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 screws 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.
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 >.
3. Click Piercing Punch, mm, Input PRL name = C9 and then click >.
4. Click Cutting Punch and then click >>.
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.

![PRL Wizard](image)

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

![Rollback Menu](image)

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.

![Parallelogram](image)

9. Click **Exit Sketch** from the confirmation corner.
10. Rename the sketch to \textit{XY} or select the sketch in the feature tree and click the \textbf{XY button} on PRL Wizard.

11. \textbf{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 \textbf{Edit Feature}.

14. \textbf{Uncheck} Direction 2 and click \checkmark to finish the command.

15. Expand Solid Bodies folder, Right-click on Punch and select \textbf{Hide}. Right-click on DP Hole and select \textbf{Show}.

16. Right-click on DP RefPlane and click \textbf{Insert Sketch}.

17. Select Face A and click \textbf{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, and select the Die Plate-DieSetStructure reference plane. Check Selected Bodies, and select the DP Hole body.

![Image](https://via.placeholder.com/150)

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 rollback bar and select Roll to End.

22. Expand the LDS folder, expand the LDS Hole feature and right-click LDS HoleSketch and select Edit Sketch 📌.


![Image](https://via.placeholder.com/150)

24. Select the 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.

![Offset Entities](image)

26. Click ✅ to finish the offset command.

27. Add a **sketch fillet** of **1.5mm** to all 4 corners.

![Fillet](image)

28. **Exit** the sketch.

**Modify the Punch with Shear to Reduce Cutting Force**

29. Click **Insert, Reference Geometry, Plane** and select the line entity from sketch **XY** for First Reference and the endpoint of the line for Second Reference.
30. Click ✔️ to finish the command.
31. Rename the plane to Shear.
32. Click on plane Shear and click Insert Sketch.
33. Set the view normal to the Shear plane by clicking Normal To.
34. Select Line and click on A, B, C to create 2 lines.
35. Select Point and click on D to create a sketch point.

36. Click Add Relation,
- Select endpoints A & C and click vertical
- Select point D and endpoint B and click Horizontal
- Select the vertical line in Sketch XY and point D and click Midpoint
- Select endpoint A & F and click Horizontal
- Select endpoint C & G and click Horizontal

37. Click Smart Dimension, add a 1mm dimension between endpoint B and the Die face shift plane. Also add an angle dimension between line AB and the Die face shift plane.

38. Click Insert, Cut, Extrude. Flip the side to cut.

39. Click ✔️ to finish the command.
40. Save and Close the file.
Case Study 10 – 6: Creating a Cutting Punch

Creating the Cut Extrude Geometry inside PRL’s

For ease of setup open the PRL wizard and edit Current Model, the PRL wizard will open. Slide it out of the way until needed. The offsets that are added is only a reference, as the Prl is inserted the offsets will follow the DSS inside the PD, and DS.

1. Right-click and select the HL plane, and Insert Sketch.

2. Select the Base Sketch of the model and offset by .005mm for a Slip Fit Punch offset and exit sketch.

3. Rename Sketch to PH HoleSketch so as the DSS is updated the offset will follow.

4. From your Keyboard hold the CRTL KEY and select both the PH HoleSketch and the Punch Holder Plane of the DSS.

5. By selecting the on the PRL Wizard, 3DQuickPress will add a reference plane at the location of the plane selected from the DSS offset .000 The new plane added adds the Sketch Name to the plane as shown below.

![PH HoleSketch Plane](image)

Note: The Sketch name on the plane MUST match for the Cut Extrude to operate properly.

6. Continue adding new Sketches by selecting the Punch Holder DSS plane.

7. Right-click Insert Sketch, select the Base Sketch of the model and offset by .05mm and exit sketch.

8. Rename Sketch to SBP HoleSketch so as the DSS is updated the offset will follow.

9. From your Keyboard hold the CRTL KEY and select both the SBP HoleSketch and the Stripper Plane of the DSS.
10. By selecting the on the PRL Wizard, 3DQuickPress will add a reference plane at the location of the plane selected from the DSS offset .000 The new plane added adds the Sketch Name to the plane as shown below.

![Image of Sketches]

11. Continue adding new Sketches by selecting the Stripper DSS plane.

12. Right-click Insert Sketch , select the Base Sketch of the model and offset by .010mm and exit sketch.

13. Rename Sketch to SP HoleSketch so as the DSS is updated the offset will follow.

14. From your Keyboard hold the CRTL KEY and select both the SP HoleSketch and the Mat'l Thickness Plane of the DSS.

15. By selecting the on the PRL Wizard, 3DQuickPress will add a reference plane at the location of the plane selected from the DSS offset .000 The new plane added adds the Sketch Name to the plane as shown below.

![Image of Sketches]

16. Continue adding new Sketches by selecting the Z-Datum DSS plane.

17. Right-click Insert Sketch , select the Base Sketch of the model and offset by .005mm and exit sketch.
18. Rename Sketch to **DP HoleSketch<A0.25>** so as the DSS is updated the offset will follow.
19. The Added information <A0.25> to the DP Sketch Name will manage the Taper of the DSS for the Die Yoke.

20. From your Keyboard hold the CRTL KEY and select both the **DP HoleSketch<A0.25>** and the Die Plate Plane of the DSS.

21. By selecting the **** on the PRL Wizard, 3DQuickPress will add a reference plane at the location of the plane selected from the DSS offset .000 The new plane added adds the Sketch Name to the plane as shown below.

![Image of Sketches]

22. Continue adding new Sketches by selecting the Die Plate DSS plane.

23. Right-click **Insert Sketch** , select the Base Sketch of the model and offset by .010mm and exit sketch.

24. Rename Sketch to **DS HoleSketch** so as the DSS is updated the offset will follow.

25. From your Keyboard hold the CRTL KEY and select both the **DS HoleSketch** and the Die Backup Plane of the DSS.

26. By selecting the **** on the PRL Wizard, 3DQuickPress will add a reference plane at the location of the plane selected from the DSS offset .000 The new plane added adds the Sketch Name to the plane as shown below.

![Image of Sketches]
27. Continue adding new Sketches by selecting the Die Backup DSS plane.

28. Right-click **Insert Sketch**, select the Base Sketch of the model and offset by .015mm and exit sketch.

29. Rename Sketch to **LDS HoleSketch** so as the DSS is updated the offset will follow.

30. From your Keyboard hold the CRTL KEY and select both the **LDS HoleSketch** and the **Lower Die Set Plane** of the DSS.

31. By selecting the on the PRL Wizard, 3DQuickPress will add a reference plane at the location of the plane selected from the DSS offset .000 The new plane added adds the Sketch Name to the plane as shown below.

32. Hide all Sketches and Sketch Planes, best practice is to add a folder to contain all Sketches and Planes. Rename to Clearance Sketches.

**Case Study 10 – 7: Create Die Set PRL**

Topics covered:

- Create Hole features
- Create Hole groups
- 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 hole series to pass through multiple plates.

Insert component to the die set structure based on the hole series.

**Create Hole Features for Guide Pin SGOH 16mm with Bushing**

First we need to copy the C:\Program File\3DQuickPress\PRL\Library Features\Hole Series folder contents to your current PRL directory. This directory should be located under C:\Program Files\3DQuickPress\Training\MyPRL\Library features\Hole Series. If there are duplicates, copy over them.

1. Click **File, New** to create a new part document. Use a metric template.

2. Select **Front Plane** and **Insert Sketch**.
3. Click **Corner Rectangle** and create a **40mm x 40mm** rectangle. The lower left corner of the rectangle should be coincident to the origin.

4. Click **Insert, Feature, Boss/Base, Extrude** and extrude it in **Reverse** direction with a **blind** depth of **30mm**.

5. Click ✓ to finish the command.

6. Select the face below and click **Insert, Feature, Hole Wizard** ...

7. Go to Legacy Hole

8. Hole Type = Counterbored
9. End Condition Type = Through All
10. Diameter = 16mm
11. C-Bore Diameter = 19mm
12. C-Bore Depth = 5mm

13. Rename the hole feature to **GPCB**.

14. Select **GPCB** and click **File, Save as**, set the **Save as Type** to **LibFeatPart** and Input **SGOH-CB** for file name. Save in the hole series directory under the current PRL directory, Library features, Holes series.

15. **Rename** the configuration to **GP16mm**.
16. **Save** the part.

17. Click **File, Save As**, set the **Save as Type** to **Lib Feat Part** and input file name **SGBH-CB**.

18. **Rename** feature **GPCB** to **Bush CB**.

19. Right-click **Bush CB** and select **Edit Feature**. Change the dimension to:

20. Diameter = 20mm
21. C-Bore Diameter = 22.5mm
22. C-Bore Depth = 5mm

23. **Rename** the configuration to **Bush16mm**.

24. 

25. **Save** the part.

26. Click **File, Save As**, set the **Save as Type** to **Lib Feat Part** and input file name **SGOH-H**.

27. **Rename** **Bush CB** to **GP Hole**

28. Right-click **GP Hole** and select **Edit Feature**. Change Type to **Simple**, Diameter = 17mm.

29. **Rename** the configuration to **GP16mm**.

30. 

31. **Save** the part.

32. **Close** all open files.
Case Study 10 – 9: Creating a Die Set from System Template

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

Case Study 10 – 10: 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\DieSetAssemblyTemplate\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 the part in place.
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 the part in place.
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 FeatureManager design tree for the **U** assembly.
2. In the **U** assembly, expand the FeatureManager design tree for the **DieSetStructure**.
3. From the assembly, expand the FeatureManager design tree for the Sketches part.

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.


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.

![Diagram showing the creation of points and dimensions.]

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.

![Diagram showing the constraint at Point A.]

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

![Dimensioned circle in the 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.


3. Click Next, and select die plates by clicking on the plates in the graphic screen.

4. Select the Die Plate from the Hole Series parameter, and select the Library Feature and Configuration under Hole Details. Repeat for each plate.
   - U-1/PH-1, SGOH-CB, GP16mm, Top
   - M-1/SB-1, SGOH-H, GP16mm, Top
   - M-1/ST-1, SGBH-CB, Bush 16mm, Top
   - L-1/DP-1, SGBH-CB, Bush 16mm, Bottom
   - L-1/DS-1, SGOH-H, GP16mm, Top
   - L-1/LDS-1, SGOH-H, GP16mm, Top
Tip: These settings can be saved for future use. The file is located in C:/Program Files/3DQuickPress/PRL/Library features/HoleSeries/*.dat. Multiple group files can be saved.

**Save the Currently Used Hole Group for Reuse:**

5. Enter Group File Name as MyHoleGroup and Group Description as Guide Pin 16mm. Then click Add.

6. Click to finish the command.

The MyHoleGroup.dat can be viewed by opening the file in excel. The file is located in My PRL, Library Features, HoleSeries, MyHoleGroup.dat. This file can be edited manually and saved as different variations.

---

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
</tr>
</thead>
<tbody>
<tr>
<td>Group Name</td>
<td>Library Feature File</td>
<td>Configuration</td>
<td>Top/Bottom</td>
<td>Sequence</td>
</tr>
<tr>
<td>Guide Pin16mm</td>
<td>SGHO-CB</td>
<td>GP16mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Guide Pin10mm</td>
<td>SGHO-H</td>
<td>GP10mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Guide Pin16mm</td>
<td>SGBH-CB</td>
<td>Bush16mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Guide Pin16mm</td>
<td>SGBH-CB</td>
<td>Bush16mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Guide Pin16mm</td>
<td>SGHO-H</td>
<td>GP16mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Guide Pin16mm</td>
<td>SGHO-H</td>
<td>GP16mm</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**Add a Guide Pin and Automatically Pattern to Holes**

1. Open the assembly.

2. Right-click Guide Pins sketch in the Sketches part and select show.

3. Select Insert, Component, Existing Part/Assembly.
4. **Browse** for the part in C:\Program Files\3DquickPress\Standard Components\Misumi\Guide Pin\SGOHxx-xxx.SLDPRF, select configuration SGPN 16-080.

5. Drag the Guide Pin close to the first hole (Indicated by the sketch circle in Guide Pin sketch). Click to place the part.

6. Hold CTRL and select UDS and PS parts in the feature tree. Right click and select *Hide Component*.

7. Click **Insert, Mate** to add Concentric and Coincident mates to position the Guide Pin to the first hole.

8. Click **Insert, Component Pattern, Feature Driven**, select SGOHxx-xxx as the Component to Pattern and select Sketch-Pattern1 from the PH part for the Driving Feature.

9. Click ✔️ to finish the command.

10. **Repeat** this process to add Guide Bushings to the Stripper and Die plates in L and M subassemblies. Refer to the diagram below for part number and configuration information.
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).
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.

![Sketch with dimensions](image)

8. Exit 🚪 the sketch.

![Insert Hole Series dialog](image)

10. Click Next 🚪 and select the three plates under the Hole Series parameter.

12. Click ✓ to finish the command.

13. Insert a Socket Head screw into the first hole (Indicated by the sketch circle in the SHCS sketch). Select CBxx-xxx from file location C:\ 3DquickPress\Standard Components\Misumi\Socket head Screw\CBxx-xxx.SLDPRT, select configuration CB10-050.

14. Mate the pin to the 1st hole using a coincident and a concentric mate.

15. Click Insert, Component Pattern, Feature Driven, select CBxx-xxx as the Component to Pattern and select Sketch-Pattern2 from the DP part for the Driving Feature.

16. Save and Close the part.