SolidPractices: Sheet Metal

SOLIDWORKS® Standard

Last Update: 13 November 2019

Revision 1.0

3DEXPERIENCE®

Table of Contents

Revision History

Rev #	Date	Description
1.0	*Nov 2019*	*Revised for use by customers and reset as document version 1.0*

Note

All SolidPractices are written as guidelines. You are recommended to use these documents only after properly evaluating your requirements. Distribution of this document is limited to Dassault Systèmes SolidWorks employees, VARs and customers that are on active subscription. This document may not be posted on blogs or any internal or external forums without prior written authorization from Dassault Systèmes SolidWorks Corporation.

This document was updated using version SOLIDWORKS 2019 SP04. If you have questions or need assistance in understanding the content, please get in touch with your designated reseller.

1) Preface

This SolidPractice document walks you through many situations that you can encounter when using the SOLIDWORKS Sheet Metal functionality.

The document provides a general description of the most common sheet metal "problems" and misunderstandings, and compiles issues reported in over 3000 service requests from sheet metal customers.

Your Feedback Requested

We would like to hear your feedback and also suggestions for new topics. After reviewing this document, please take a few minutes to fill out a brief survey. Your feedback will help us create the content that directly addresses your challenges.

2) Best Approach To Modeling Sheet Metal Bodies

SOLIDWORKS has specific sheet metal features that allow the creation of sheet metal bodies very quickly. However, in some circumstances, when the design demands certain types of geometries, the user has the option to use non-sheet metal feature tools and then use the **Insert Bends** or “**Convert to Sheet Metal**” features.

When designing with sheet metal, it is important to think about the best approach to model or design a part. At times, it may appear quicker to use non-sheet metal features (boss-extrude, etc.), and then insert bends or convert to sheet metal. However, these options are problematic and the least flexible.

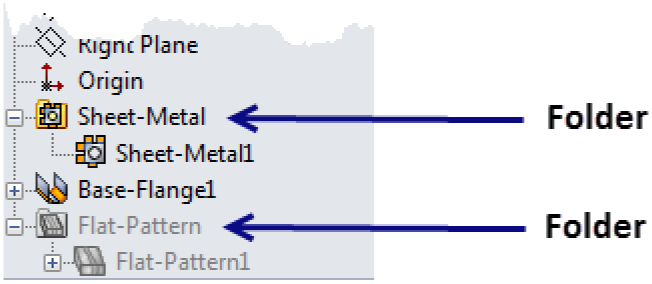
When designing sheet metal parts, the order preference for use of feature tools are as follows:

1. Use sheet metal features such as base-flanges, edge-flanges, miter flanges, etc.
2. Use the **Insert Bends** feature.
3. Use the **Convert to Sheet Metal** feature.

When using the **Insert Bends** or **Convert to Sheet Metal** features, apply the features as early as possible during the part design phase. When possible, insert bends or convert to sheet metal immediately after creating the first body feature.

3) Sheet Metal Architectural Changes In SOLIDWORKS 2013

Effective with the release of SOLIDWORKS version 2013, there is a revision to the architecture for the sheet metal functionality. With this release, the sheet metal FeatureManager® design tree appears as follows:



If you are using an older part template and the sheet metal FeatureManager tree does not look like this, you must recreate your part template.

4) Verification On Rebuild

Activating the **Verification on rebuild** option (Figure 4) and pressing ***Ctrl + Q*** to force regeneration of the model catches most feature problems immediately after creating a feature.

As a best practice, force regenerate the model after creating each feature.

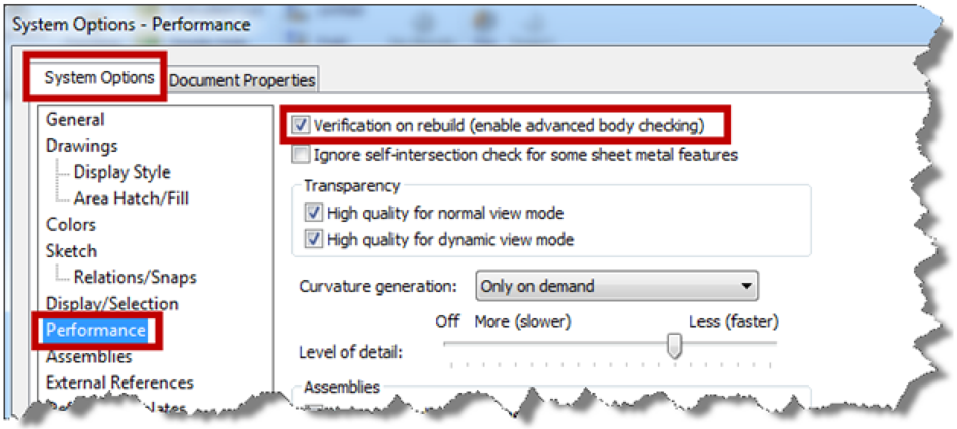


Figure 4

5) Cut-Extrude with Normal Cut Option

It is a recommendation to activate the **Normal Cut** option for most cut-extrude features that you apply to sheet metal bodies, even when this does not seem required needed. The only time that you must not use this option is when there is a specific reason for it. For example, if the material removed by a normal cut is more than expected, or if you need a specific beveled edge.

In addition, whenever possible, create sheet metal holes by using the **Cut-Extrude** feature. This is because the **Normal Cut** option is not available for the **Simple Hole** feature and the **Hole Wizard** feature.

The next image (Figure 5) depicts a cut-extrude without the normal cut option. Notice that apparently, the **Normal Cut** option is not required in this case. Inside the cutout, there is an edge-flange on both sides of the cut. Zooming in to one side of the edge flange (Figure 5a) reveals the appropriate reliefs. The edge flange on the other side (Figure 5b) does not have the proper reliefs. Figure 5c shows the flat pattern for the part. Notice the irregular area at the center of the part.

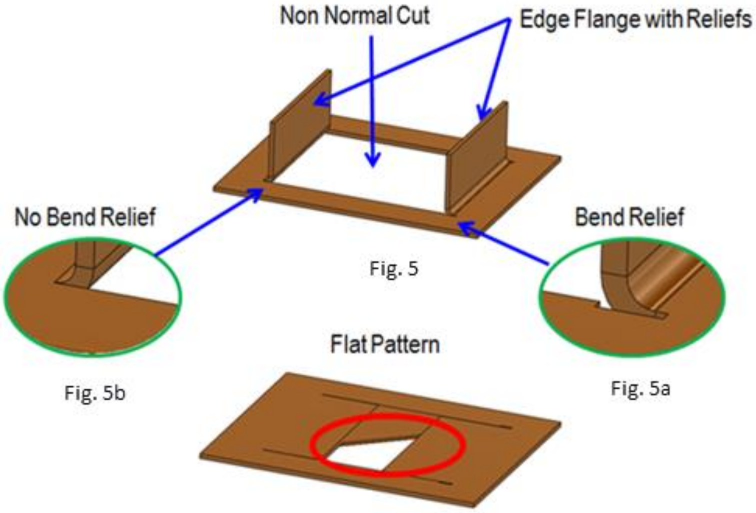


Fig. 5c

(Figure 5d) shows the same part. However, in this case, the normal cut option is active. Notice that the bend reliefs appear on both sides of the normal cut (Figure 5e & 5g). The flat pattern (Figure 5f) is now correct.

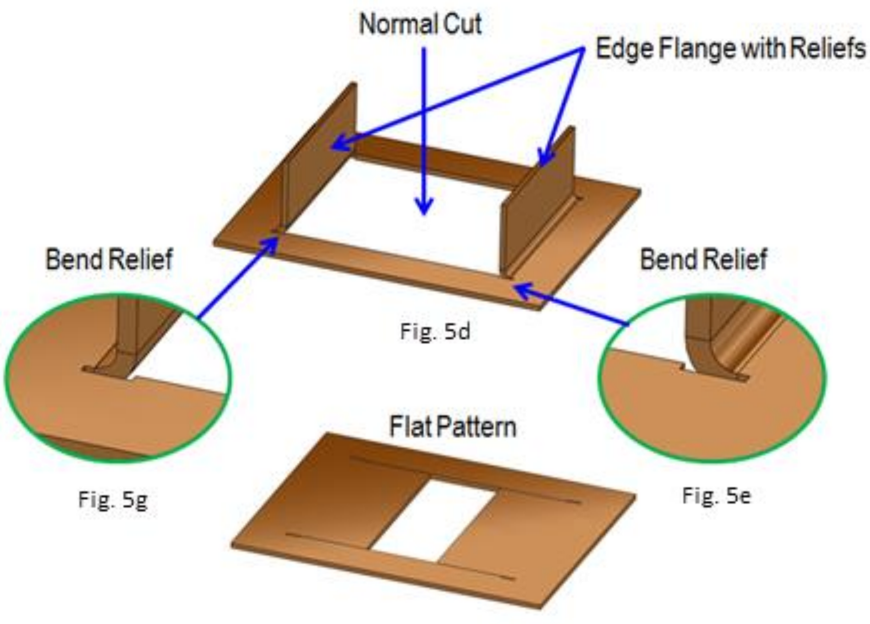


Fig. 5f

6) Zero Gap and Self-Intersection

One of the most common misunderstandings when working with sheet metal is the *zero gap* issue.

A zero gap means that two faces "fuse" to each other (Figure 6 & 6a). This is why SOLIDWORKS displays the message "*...the part self-intersects...*" This message only appears if the **Verification on rebuild** option is active.

The solution for this type of problem is to create a very small gap (0.001 mm) between the faces. This separation is typically negligible for any sheet metal part, but not for SOLIDWORKS.

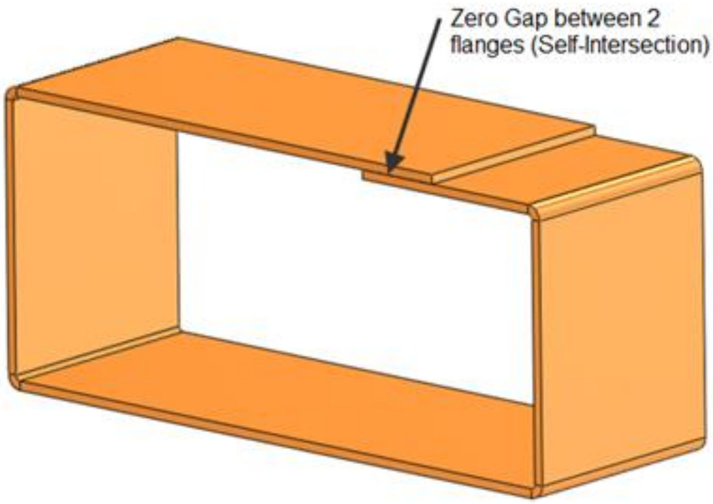


Figure 6

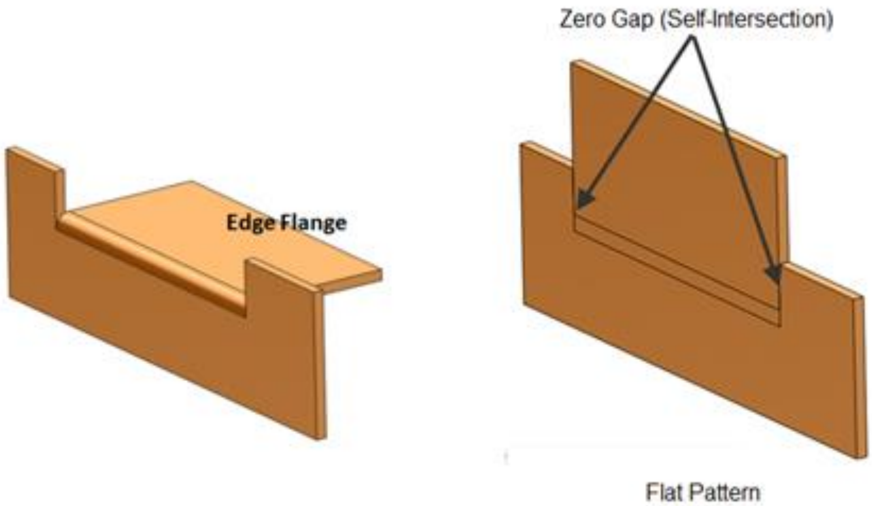


Figure 6a

7) Avoid Self-Intersection with Edge-Flange

As shown in (Figure 6a), one of the most typical situations for self-intersection is when applying an edge flange on a cut-extruded area. There are two basic approaches to avoid self-intersection. The first approach (Figure 7a) is to create a small gap at each side of the edge flange.

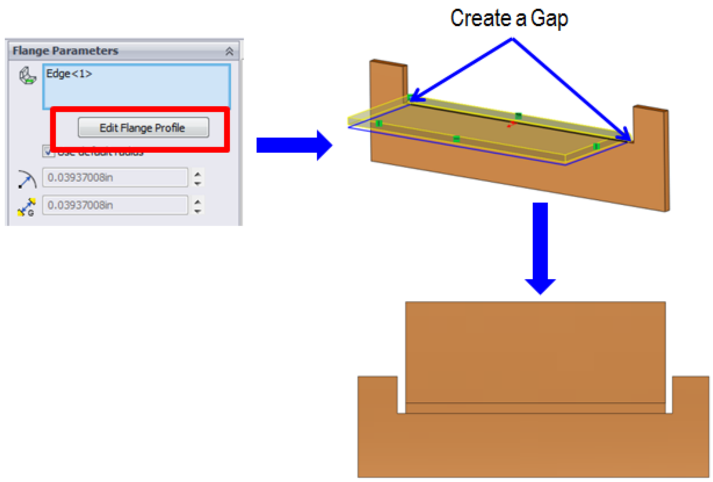


Figure 7a

The second approach takes advantage of the **Offset** option within the edge flange functionality. When active, this option automatically creates reliefs at the flange sides (Figure 7b)

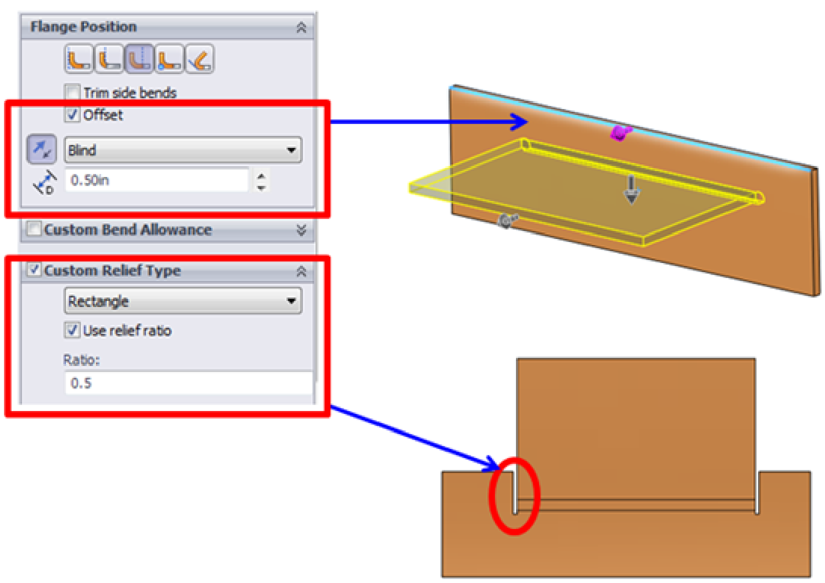


Figure 7b

8) Designing Sheet Metal From the Flat

When designing sheet metal parts from the flat, it is important that every feature you create have the proper reliefs, and that these reliefs belong to the feature you create. You must either create the bend reliefs before the bend operation (that is, create the feature), or create the reliefs with the feature.

For example, the following image (Figure 8) shows the part before applying an Edge-Flange feature to that edge.

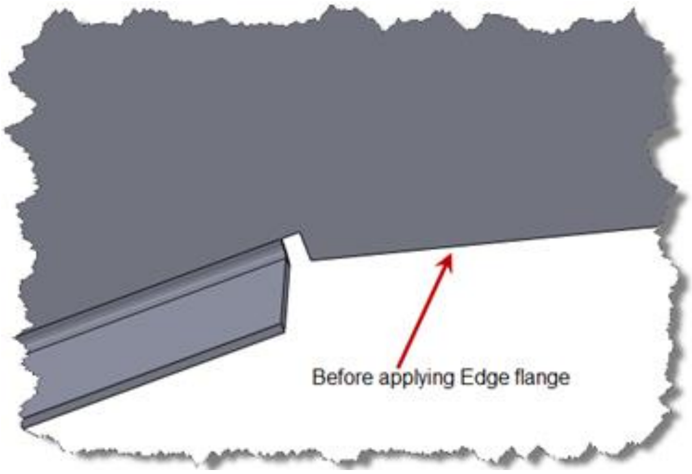


Figure 8

The next image (Figure 8a) shows the area after applying the Edge-Flange feature. Notice that the area has an improper relief.

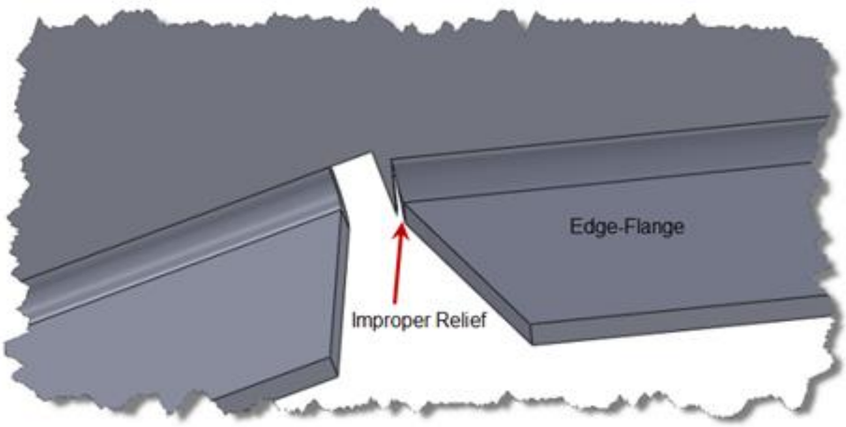


Figure 8a

(Figure 8b) shows the application of a manual relief by removing the “spiked” material using a cut-extrude in a downstream feature. However, SOLIDWORKS is a feature-based parametric modeler, which rebuilds from the top-down. Therefore, when the modeler rebuilds, because the Edge-Flange feature is above the Cut-Extrude feature, the user encounters a self-intersection problem.

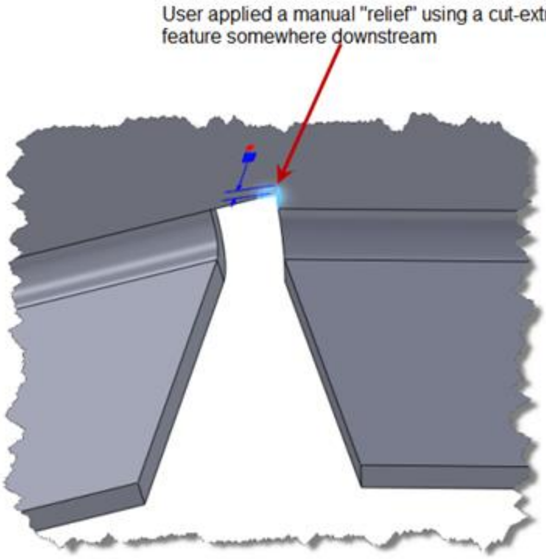


Figure 8b

Figure 8c shows the proper way to handle this problem. The first solution is to avoid the problem altogether, either by creating an Edge-Flange feature with the bend outside of the flange position (if possible), or by creating conditions for the edge flange that do not need reliefs.

The other option is to use the automatic Edge-Flange reliefs (Figure 8c) and then use the cut-extrude feature to remove the “spiked” piece.

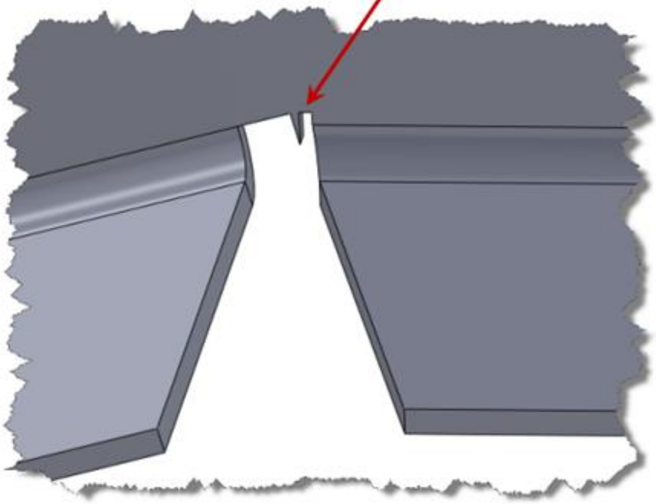
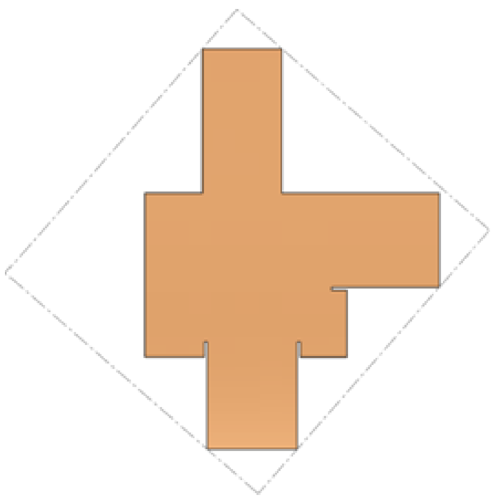

The proper way to handle this area, is by preventating this problem before the edge flange is applied, or to use the automatic edge-flange relief to the area and then cut-extrude the remaining piece

Figure 8c

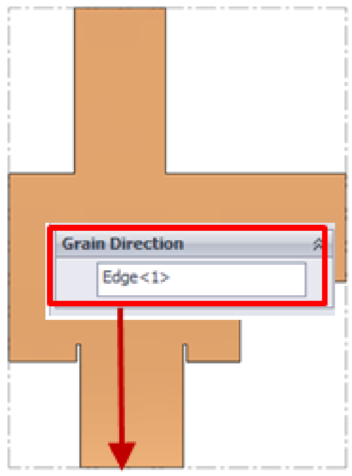
9) Bounding Box and Grain Direction

The bounding box algorithm finds the smallest area in which to enclose the flat pattern for the part. At times, the direction of the bounding box is not very effective (Figure 9). When this occurs, you can specify the bounding box in any direction by using a sketch or an edge in the **Grain Direction** option in the FeatureManager (Figure 9a).



Automatic Bounding Box
BB Area = 11.20 sq. inches

Figure 9



Manual Bounding Box
BB Area = 11.21 sq. inches

Figure 9a

10) Sheet Metal Mirror Features

The sheet metal **Mirror** feature simplifies modeling by using existing features (Figure 10). However, to ensure that the mirror feature produces the right results, you must understand how the mirror feature performs in the flat pattern.

After creating a mirror feature, it is a recommendation that you flatten the part and compare the mirror bend region with the parent counterpart (Figure 10a).

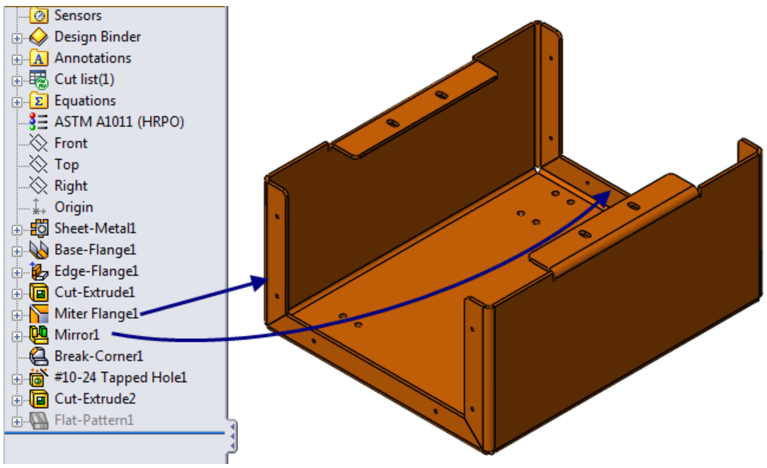


Figure 10

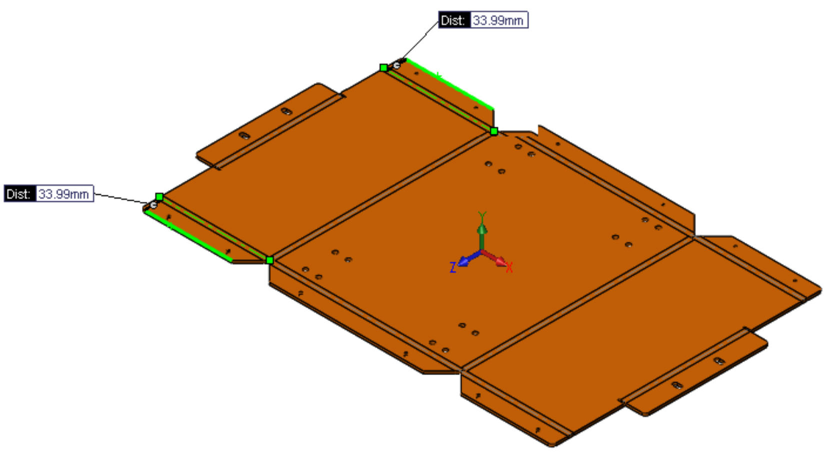


Figure 10a

11) Swept Flange: Two Different Flat Pattern Options

There are two options to create a flat pattern from a swept flange body (Figure 11):

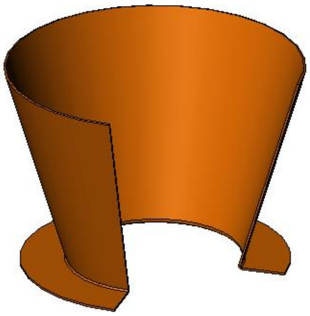


Figure 11

- The default flat pattern (Figure 11a)

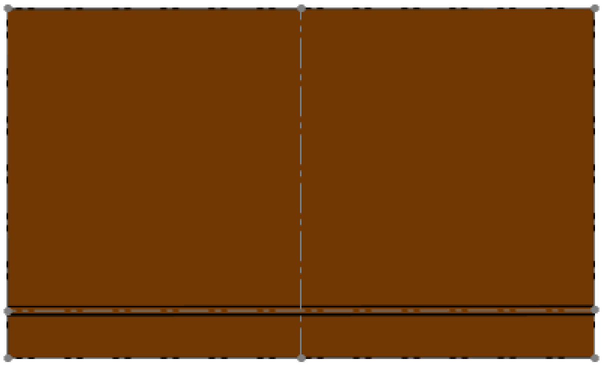


Figure 11a

- The flat pattern with **Cylindrical/Conical Bodies** (Figure 11b):

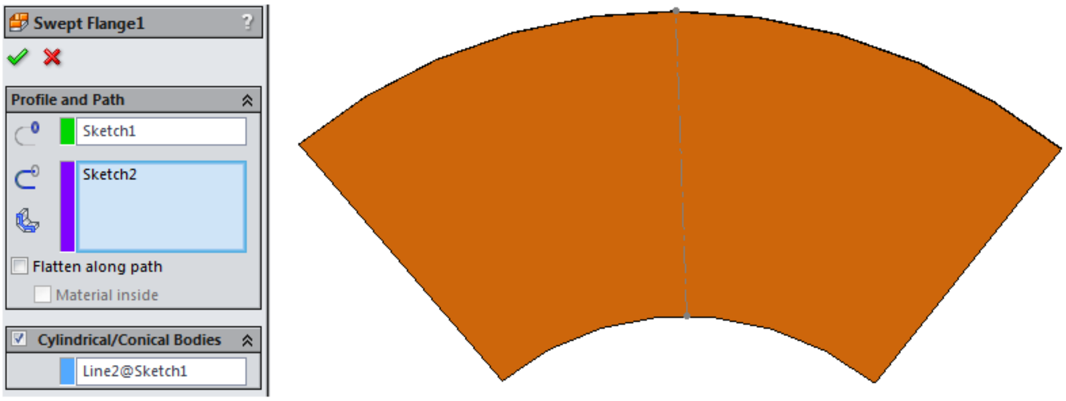


Figure 11b

12) Lofted Bend “Bent” Manufacturing Method

The **Bent** manufacturing method for lofted bends became available with the release of SOLIDWORKS 2014. This manufacturing method generates "true" bends and bend lines, and is the preferred method to create lofted bends if you require flat pattern and bend line accuracy.

Unlike the **Formed** manufacturing method, the **Bent** manufacturing method does not require sketched filleted corners (Figure 12).

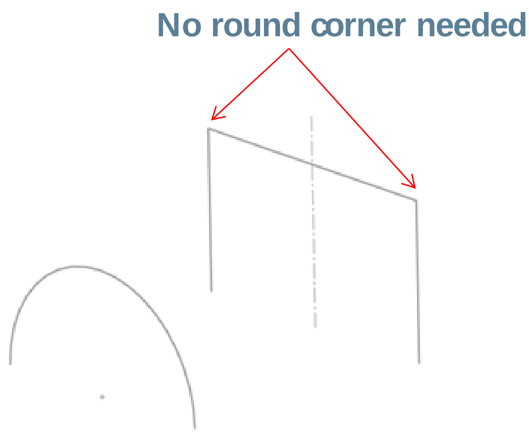


Figure 12

However, be aware that if bend lines merge to a point, the feature will break the corner automatically (Fig 12a).

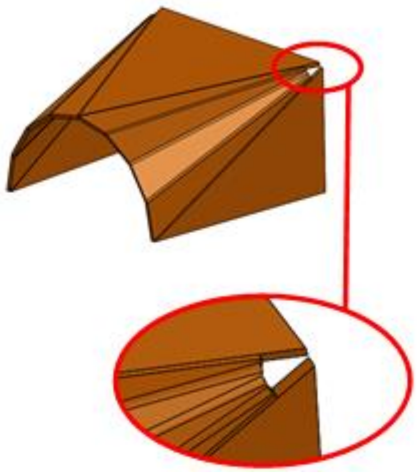


Figure 12a

When the **Refer to endpoint** option is active, the system tries to find a solution to the problem by automatically "rounding" the bend area (Figure 12b).

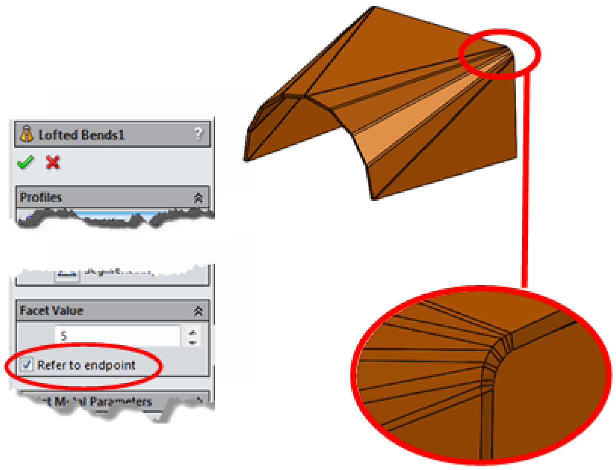


Figure 12b

The **Bent** manufacturing method for lofted bends is only available on parts with the new architecture. This is why you cannot reuse sketches from a lofted bend part created prior to the release of SOLIDWORKS 2013 to create this new feature.

13) Lofted Bend "Formed" Manufacturing Method

a) Bend lines

For lofted bend parts to display bend lines, the part must meet the following prerequisites:

- The planes of the two sketches are parallel (Figure 13a1).

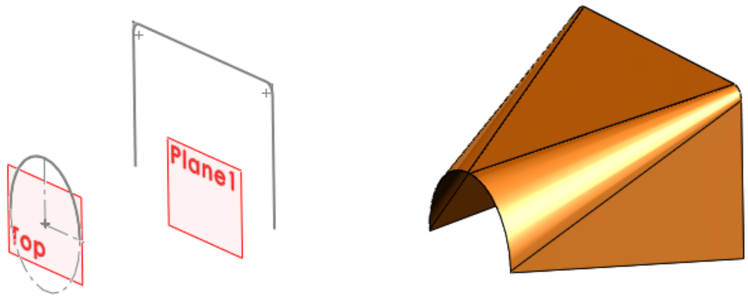


Figure 13a1

- The sketches have an equal number of linear and non-linear segments (Figure 13a2).

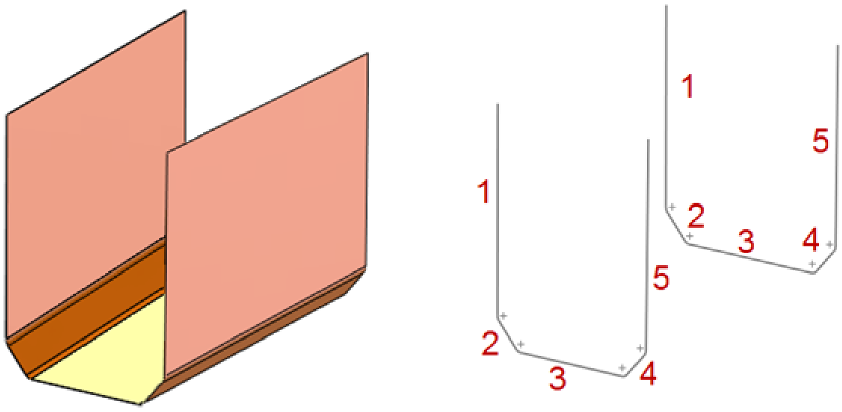


Figure 13a2

- For each linear sketch segment in the first sketch, there is a corresponding parallel sketch segment in the second sketch (Figure 13a3).

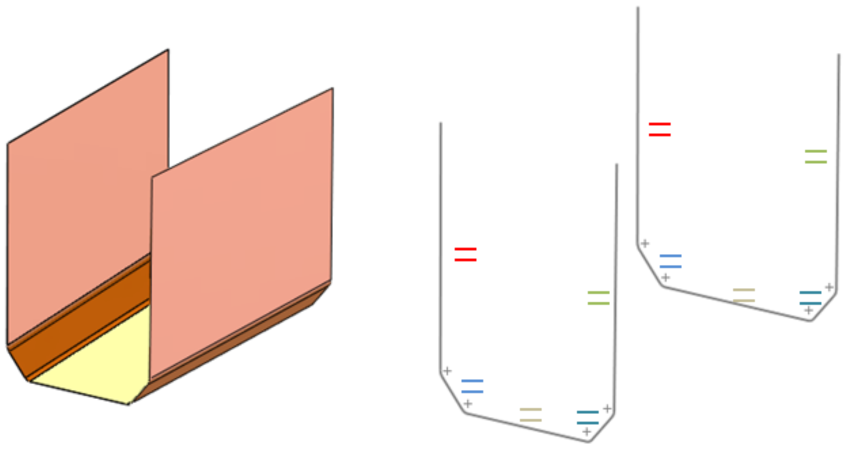


Figure 13a3

b) Application of lofted bend lines

If a lofted bend part complies with the requisites, there is a uniform distribution of bend lines across each bend region (Figure 13b).

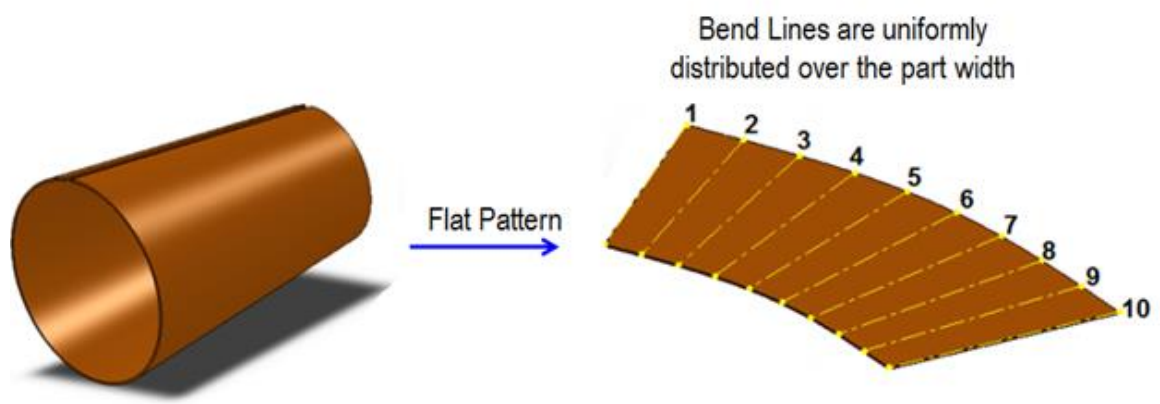


Figure 13b

14) Flat Pattern Drawing Views

When creating flat pattern drawing views, particularly of older parts, it is important to edit the flat pattern feature and ensure that you select the appropriate fixed face.

SOLIDWORKS only processes a true flat pattern drawing if the system creates this view automatically by using a flattened derived configuration. This is an important distinction. Some users create a derived configuration, unsuppress the flat pattern, and then create a frontal view of the part in this configuration. Although this drawing view is very similar to a flat pattern drawing view, the view is not a true flat pattern drawing.

Effective with the release of SOLIDWORKS 2013, the new architecture includes updates to the sheet metal bend line notes in flat pattern drawing views. The updates “repair” a number of issues from the earlier versions of SOLIDWORKS. If you encounter any issue with bend line notes in flat pattern drawing views, SOLIDWORKS Technical Support can repair these parts and drawing files and convert them to the new architecture.

15) Sketched Bend Feature and Bend Reliefs

When using the **Sketched Bend** feature, give extra consideration when folding a tab if the sketched bend sketch is close to the edge (Figure15)

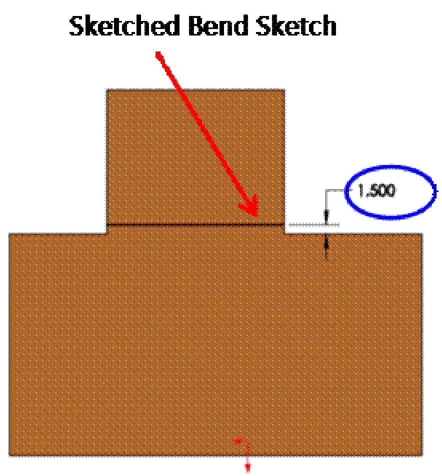


Figure 15

The Sketched Bend feature does not create automatic bend reliefs. This can be particularly troublesome when using the **Bend Centerline**, **Material Inside**, or **Material Outside** (Figure 15a) feature bend options.

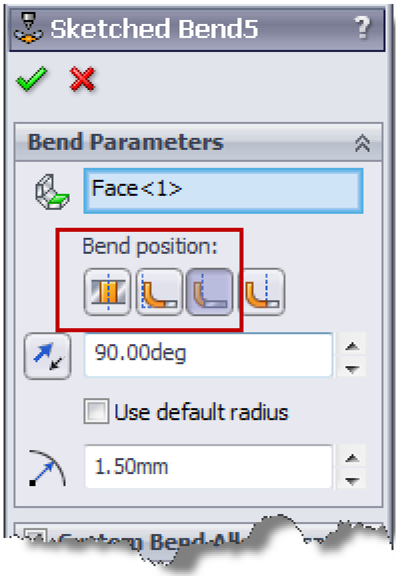


Figure 15a

If the Sketched Bend feature sketch line is close to the edge, and if the feature bend radius is large, there is a possibility for the feature to "bend" adjacent edges also. (Figure 15b)

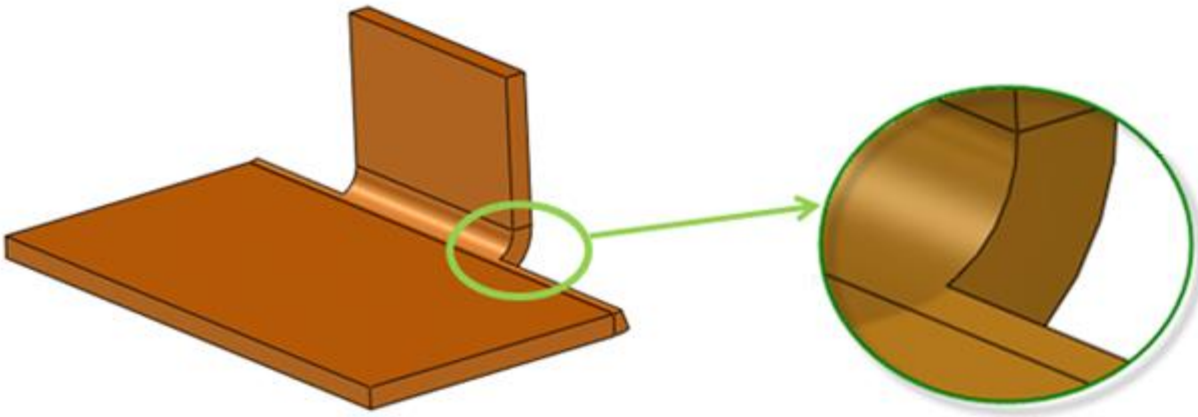


Figure 15b

In a worst-case scenario, this problem of "over bending" occurs when using the **Material Inside** option (Figure 15c). In such cases, when you create the sketched bend feature, the position of the *Sketched Bend Line* is at the outer face of the bend. Therefore, the *Sketched Bend Line Distance* must be equal to or greater than the *Bend Radius* plus the part thickness.

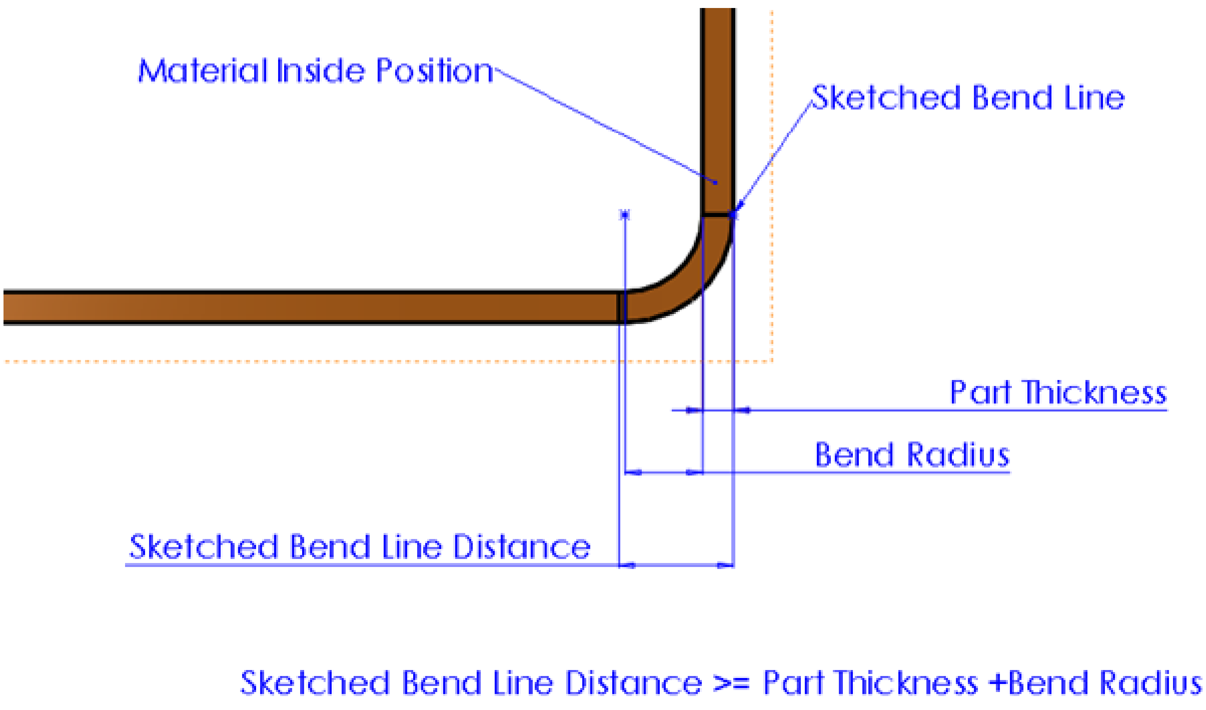


Figure 15c

16) Improve Flat Pattern Performance in Complex Parts with Feature Patterns

It is a best practice to activate the **Verification on rebuild** option. However, when this option is active, the system spends additional time checking for interferences. You can significantly improve the performance of the flattening operation by deactivating the **Verification on rebuild** option.

17) Alternative Way of Thinking About the Flat Pattern

Assume that you have a cylindrical part with many patterned cutouts (Figure 17), and that it will take a few minutes to create the flat pattern. When it is necessary to perform this operation multiple times, this can adversely affect a user's productivity.

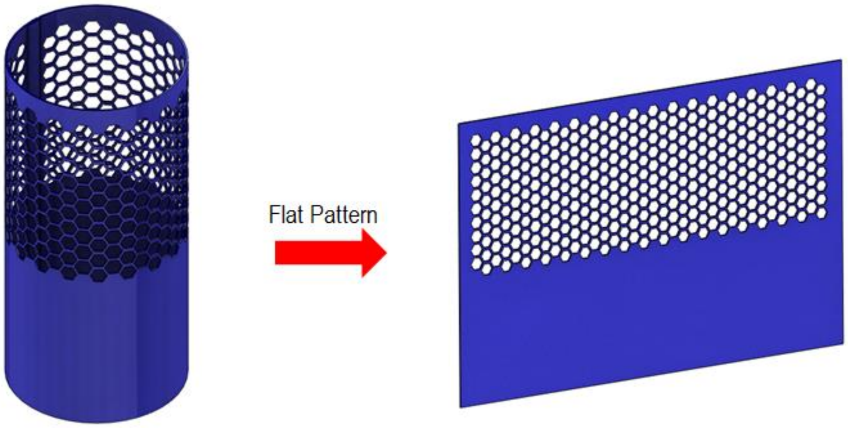


Figure 17

When considering this cylindrical part, determine if it is necessary to see the full extent of the pattern features.

An alternative way to think about this is "Where does the user really need to display the cutout pattern?"

If it is only necessary to see the cutout pattern detail in a flat pattern drawing view, then it is probably only necessary to display the pattern in that drawing view. Follow these steps:

1. Suppress the feature pattern in the folded state (Figure 17a).

Figure 17a

2. Create a flat pattern drawing view (Figure17b).

Figure 17b

This automatically creates a flattened derived configuration (Figure 17c).

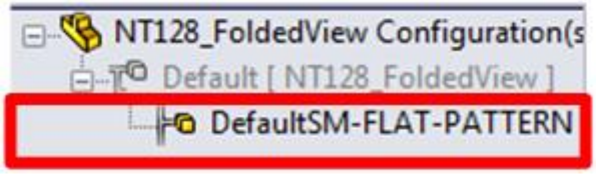


Figure 17c

3. Recreate the feature pattern in the derived configuration (Figure 17d).

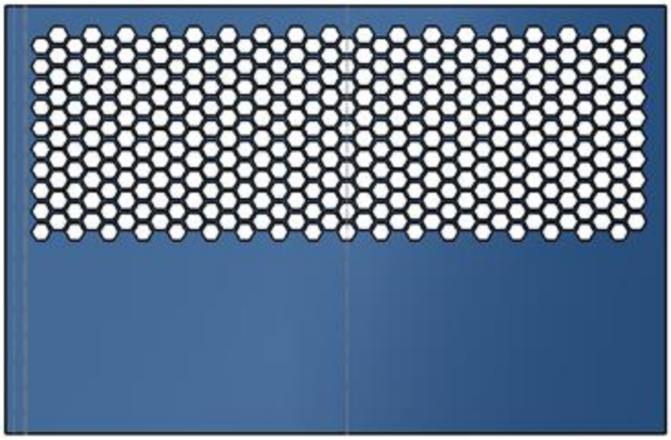


Figure 17d

After these steps, the following items appear in the FeatureManager tree. The feature pattern is unsuppressed in the default configuration while working in this state (Figure 17e).

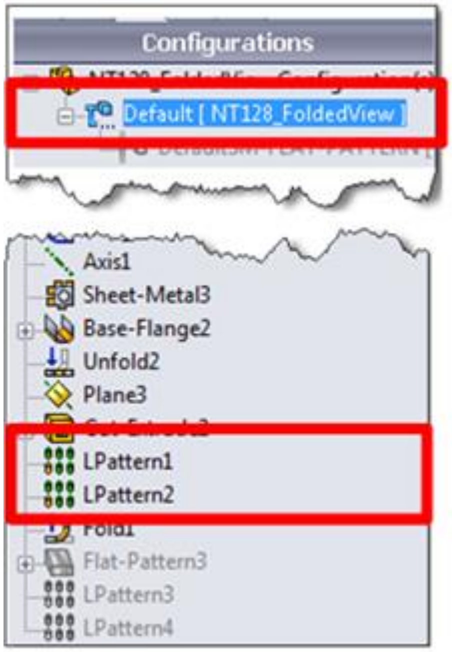


Figure 17e

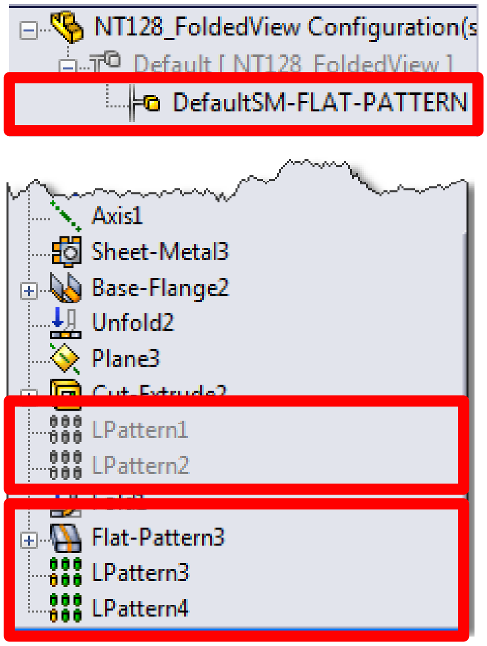


Figure 17f

When working in the flat pattern state, the software switches to the flat pattern derived configuration as shown in (Figure17f.)

In the folded state, the feature pattern is suppressed, and the newly created feature pattern in the derived configuration is unsuppressed.

18) Important Sheet Metal Options

a) The multibody sheet metal option

This option (Figure 18) does not apply to multibody sheet metal parts after SOLIDWORKS 2012. Effective with the release of SOLIDWORKS 2013, all sheet metal bodies display the available flat patterns.

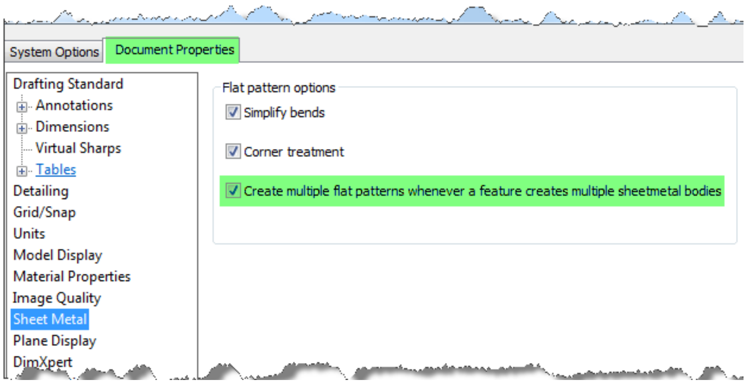


Figure 18

b) The 'Simplify bends' option

The **Simplify bends** option is only available for a flat pattern feature. By default, this option is active (Figure 18b), and as the name suggests, "linearizes" the bend region. This simplifies the cutting operation.

Because this option only exists in the flat pattern, when the part unfolds, the bend region is non-linear. This option is typically the cause of discrepancies in appearance between the flat pattern and the unfolded state.

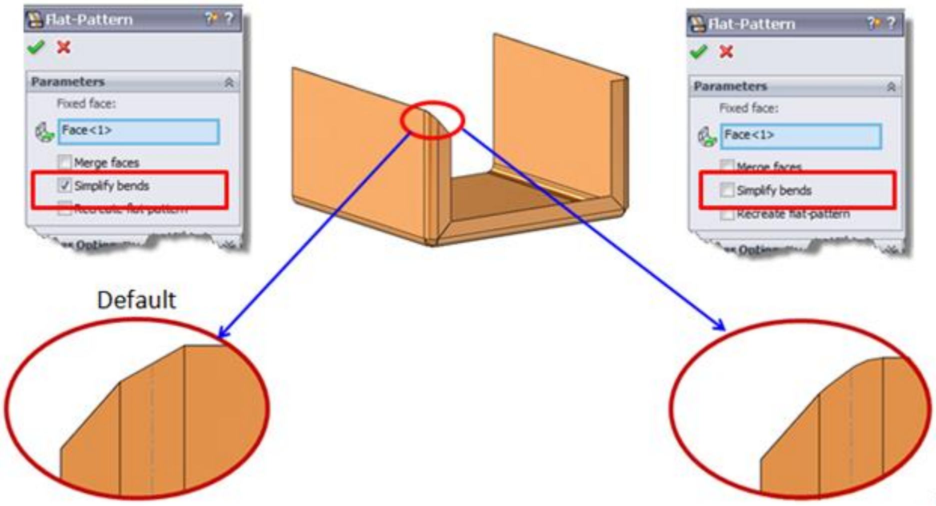


Fig 18b

c) The ‘Merge faces’ option

This option is active by default. That is, SOLIDWORKS applies the **Merge faces** option by default (Figure 18c). When this option is not active, SOLIDWORKS displays the bend regions in the flat pattern state.

This is an important option especially when working with the Bend Allowance and Bend Deduction, etc., because it allows you to check the bend region size and determine if there is any problem with these regions.

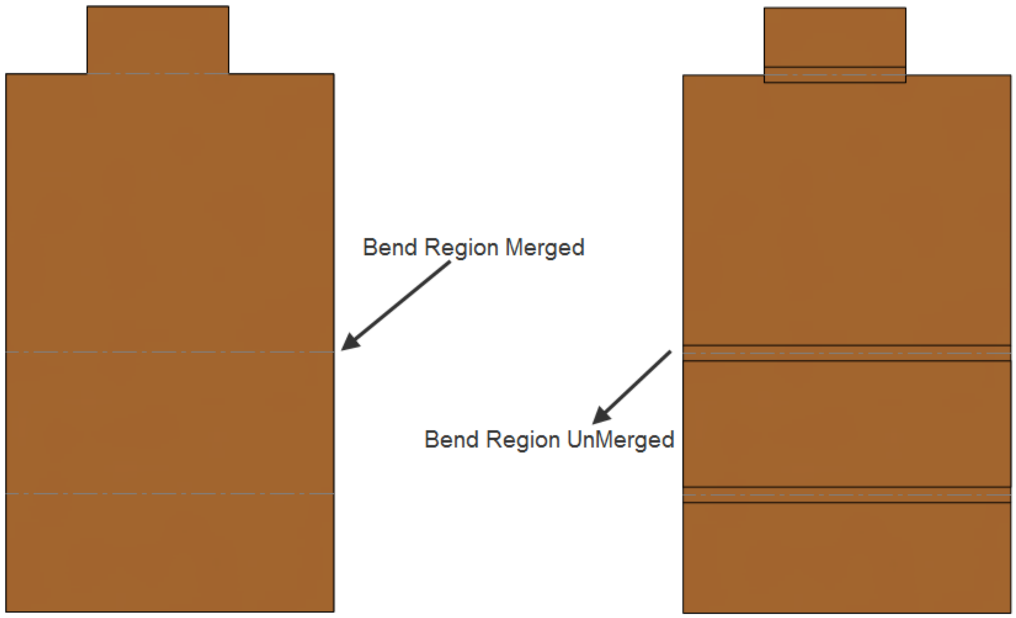


Figure 18c

19) Bend Allowance in SOLIDWORKS

a) Round bends and bend allowance

It is very important to be careful when using the 'Bend Allowance' feature with large round bends. When working directly with Bend Allowance, remember that when the part is in the flat pattern state, all of the bend's regions have the Bend Allowance width value. Therefore, any large round bend seems "odd" or "wrong" (Figure 19a).

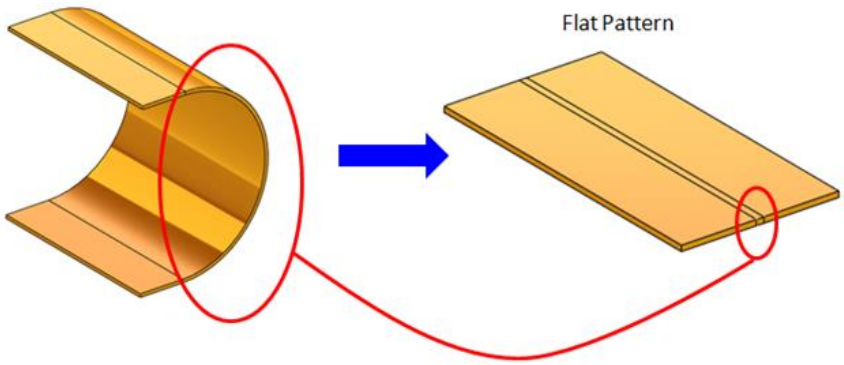


Figure 19a

The solution for this problem is to edit the round bend, and manually enter the arc length of the round bend minus the bend deduction.

b) Bend allowance control option

You can control the sheet metal Bend Allowance options (bend allowance, bend deduction, and K-factor) from different areas of the user interface (UI). In the default installation, the top-level Sheet Metal features control all of these options for lower level sheet metal bodies (Figure 19b).

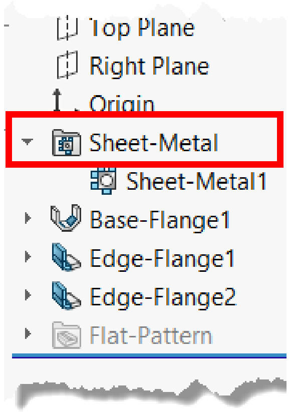


Figure 19b

In SOLIDWORKS 2013 and later, the software explicitly displays the body or lower level sheet metal feature. This allows explicit control of the bend allowance for the body (Figure 19b1).

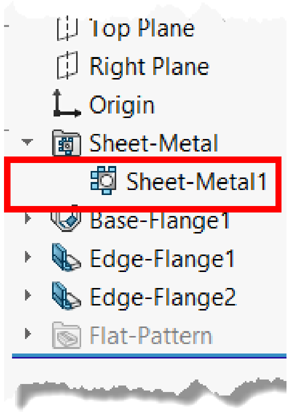


Figure 19b1

However, it is possible in SOLIDWORKS to narrow the bend allowance control at the feature level (Figure 19b2). When a Bend Allowance option is controlled at the feature level, editing the feature definition displays the **Custom Bend Allowance** group as active (checked). If a Bend Allowance option is controlled at the feature level, and all bends belonging to the feature are not specifically controlled, the feature then controls all of the bends.

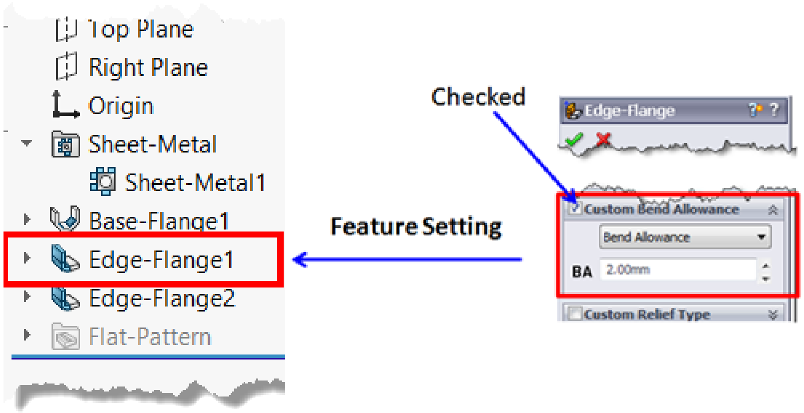


Figure 19b2

The narrowest control is at the **EdgeBend** level (Figure 19b3).

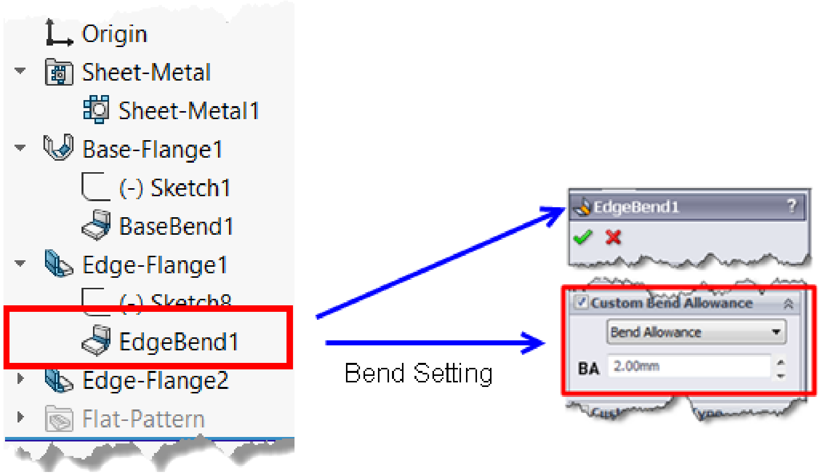


Figure 19b3

c) Flatten length calculation using bend deduction

SOLIDWORKS always calculates the bend deduction from the virtual point created by two adjacent flanges. The following example demonstrates the steps necessary to calculate the flatten length by using a bend deduction.

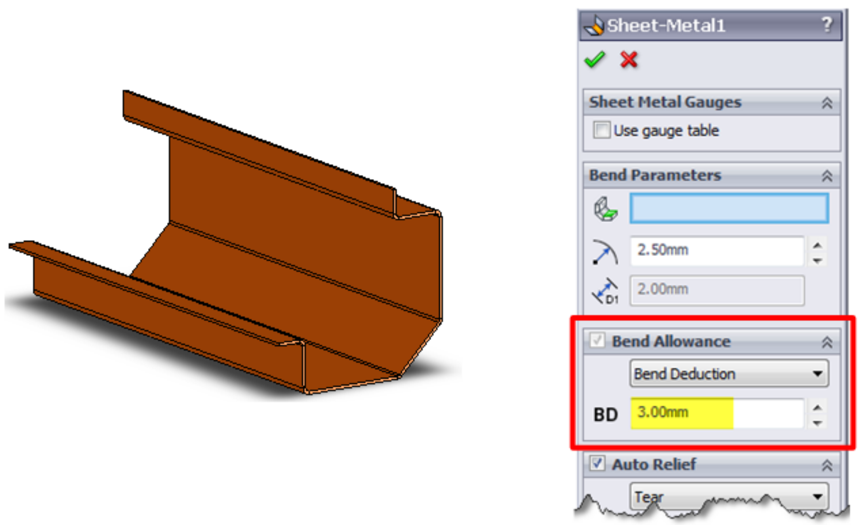


Figure 19c

The image on the left side of Figure 19c shows a simple sheet metal part. The image on the right side shows the sheet metal parameters that the software uses to calculate the flatten length. The most important value for this exercise is a *3.00 mm* bend deduction.

The first thing to consider is the sum of the sheet metal sides (Figure 19c1).

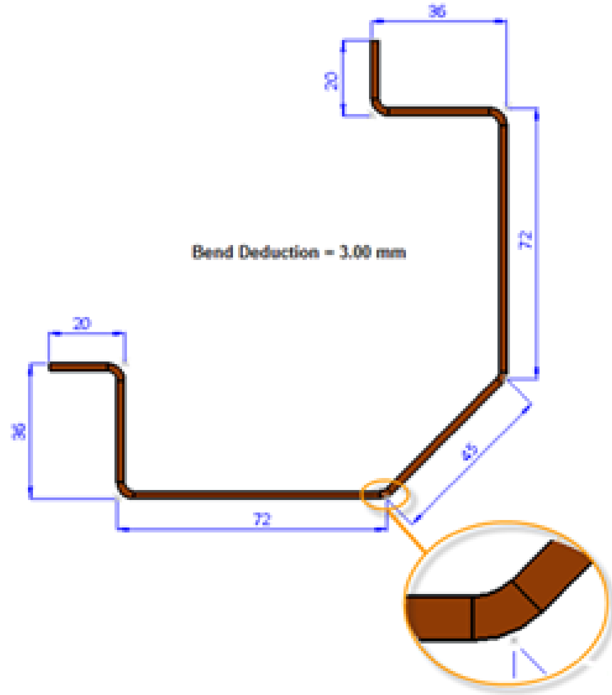


Figure 19c1

Calculations:

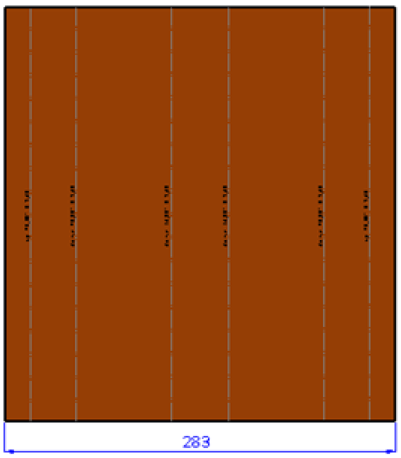
Flatten Length = Sum of all sides – (Bend Deduction x number of bends)

The Sum of all Sides = 20 + 36 + 72 + 45 + 45 + 72 + 36 + 20 = 301 mm

Flatten Length = 301 mm – (3.00 mm x 6) = 283 mm

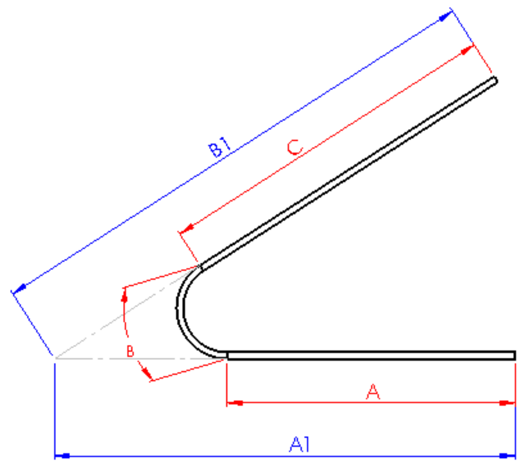
After determining the flatten length, compare this value with the flatten length obtained from the flat pattern. The image in Figure 19c2 shows that the flat length from the mathematic calculation is the same as the flat length obtained from the part.

Figure 19c2



d) Bend deduction for angles less than 90 degrees between faces

Use caution when using bend deduction with bend angles of less than 90 degrees between flanges. SOLIDWORKS uses the following method (Figure 19d) to calculate the flatten length. Be aware that there may be a difference between the flatten length you expect and the flatten length that SOLIDWORKS calculates.



Expected Flatten Length = A + B + C - Bend Deduction

SolidWorks Flatten Length = A1 + B1 - Bend Deduction

Figure 19d

20) Sheet Metal Form Tool

Minimum radius of curvature

When designing a forming tool, it is very important to consider the sheet metal thickness on which the tool is used. If possible, ensure that the form tool minimum radius of curvature is below that thickness.

To understand this concept, it is important to understand how form tools work when inserted into a sheet metal part.

As the form tool inserts into the part, the part is “shelled” and gradually thickens to its original thickness.

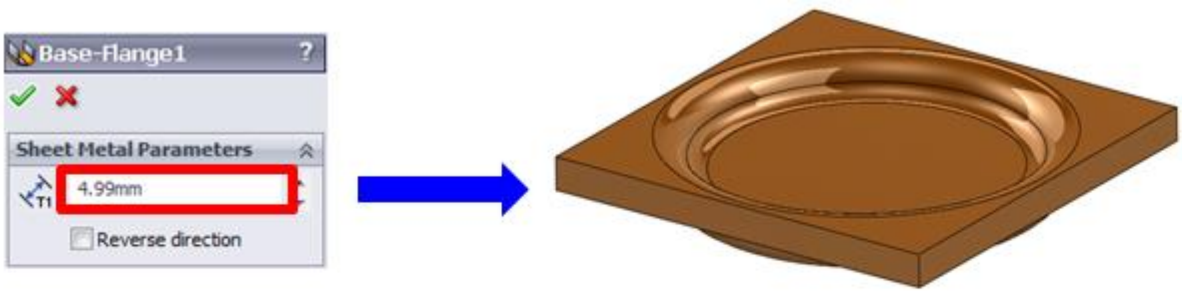


Figure 20a

The form tool works without any problem when inserted in a 4.99 mm thick sheet metal part. However, when the thickness reaches 5.00 mm, the form tool fails (Figure 20b).

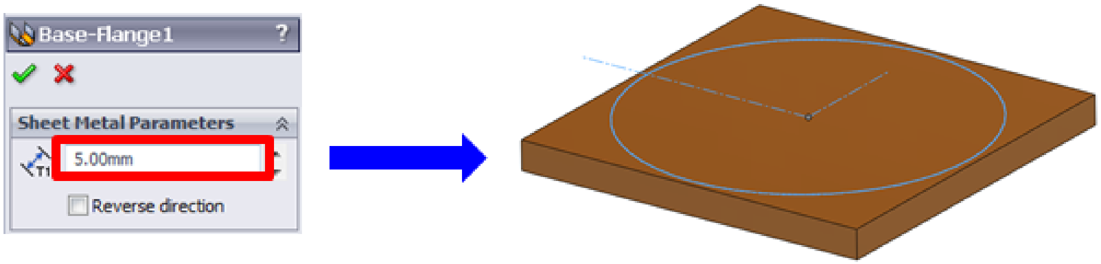


Figure 20b

When this happens, the following message appears (Figure 20c):

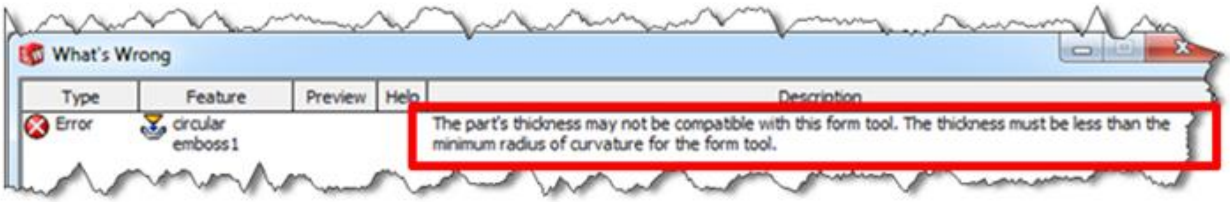


Figure 20c

Checking the form tool reveals that the minimum radius of curvature is 3 mm (Figure 20d).

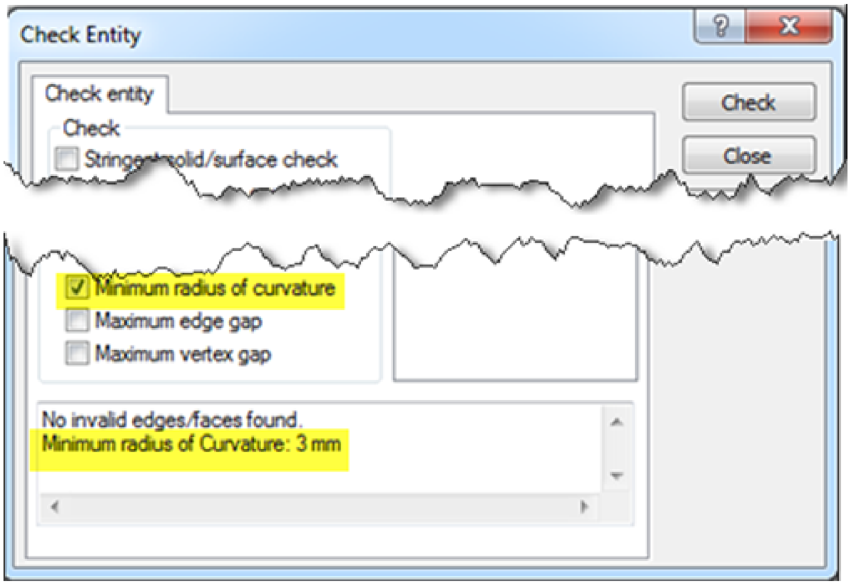


Figure 20d

The 3 mm fillet (Figure 20e) is the feature that is generating the minimum radius of curvature.

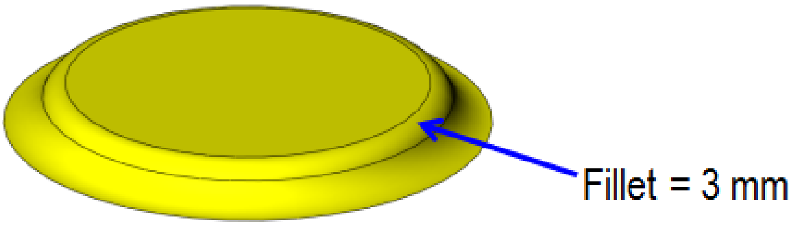


Figure 20e

However, as shown in Figure 20f, this is a cross section of the part at the thickness limit. Notice that the form tool minimum radius of curvature (MRC) of 3 mm is not a factor in the form tool failure.

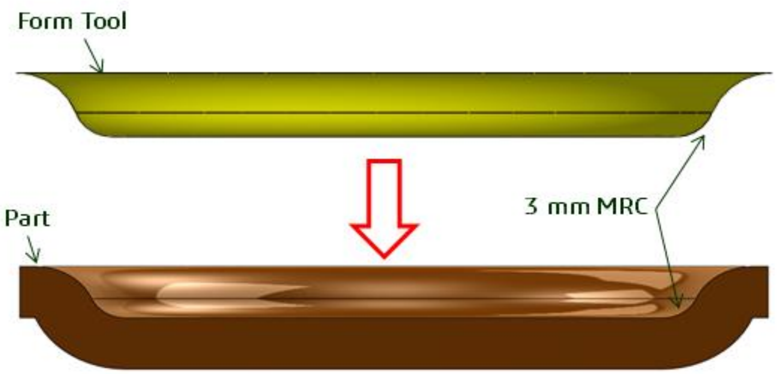


Figure 20f

The next image (Figure 20g) shows the area where the failure is happening. Here, when the part reaches a thickness of 4.999 mm, the radius shown is ***0.001 mm***.

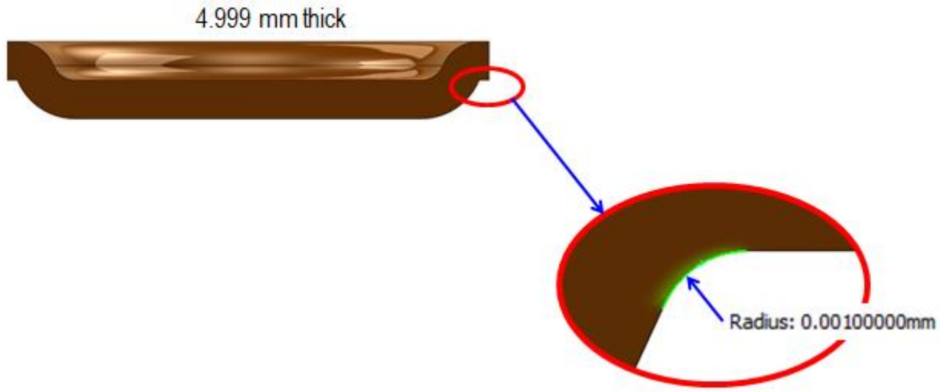


Figure 20g

21) Bend Tables

a) Why to use bend tables

In the default installation, SOLIDWORKS assigns a single Bend Allowances value for each part. It does so regardless of the bend angle and bend radius. Users can manually change bend allowances per body, per feature, and per bend. This gives users finer control over those bends. However, this is a very time consuming manual process and errors are common.

b) How to use SOLIDWORKS bend tables

SOLIDWORKS provides default bend allowances, bend deductions, and K-factor bend tables. If your company uses specifics bend tables, the best way to adapt those tables to SOLIDWORKS is to copy the data from your tables to one of the SOLIDWORKS default tables. This preserves the table format and makes the process of inserting bend tables less prone to errors.

Avoid using bend tables that use the old text file format. These tables are not embedded in the file, and they are very prone to errors. Use these tables only if you do not have access to Microsoft Excel®.

If you work with K-factors, it is important to use the K-factor bend table (Figure 21b).

Type: K-Factor
Material: Soft Copper and Soft Brass

Angle	Radius / Thickness									
	1.00	2.00	3.00	4.00	5.00	6.00	7.00	8.00	9.00	10.00
15										
30										
45										
60										
75										
90										
120										
150										
180										

Figure 21b

c) Bend table and bend angles other than 90 degrees

When using a bend table and designing parts with bend angles that differ from 90 degrees, it is preferable to use multiple thickness bend tables (Figure 21c). These types of tables offer the flexibility of using multiple thicknesses, bend radii, and bend angles.

Thickness:	1/64							
Angle	Radius							
	1/32	3/64	1/16	3/32	1/8	5/32	3/16	7/32	1/4
15									
45									
60									
90									
120									
135									
180									

Thickness:	1/32								
Angle	Radius								
	1/32	3/64	1/16	3/32	1/8	5/32	3/16	7/32	1/4
15									
45									
60									
90									
120									
135									
180									

Figure 21c

d) Rules for bend tables and gauge tables

- **Bend table bend radius columns should have the same values (Figure 21d).**

This rule is only applicable for bend tables.

Bend Table

Unit:	Millimeters					
Type:	Bend Allowance					
Material:	Soft Copper and Soft Brass					

Thickness:	1					
Angle	Radius					
	0.50	1.00	2.00	3.00	4.00	5.00
15	0.6	1	1.4	1.8	2.2	2.6
45	0.7	1.1	1.5	1.9	2.3	2.7
90	0.8	1.2	1.6	2	2.4	2.8
135	0.9	1.3	1.7	2.1	2.5	2.9
180	1	1.4	1.8	2.2	2.6	3

Thickness:	2					
Angle	Radius					
	0.50	1.00	2.00	3.00	4.00	5.00
15	0.6	1.1	1.6	2.1	2.6	3.1
45	0.85	1.35	1.85	2.35	2.85	3.35
90	0.95	1.45	1.95	2.45	2.95	3.45
135	1.05	1.55	2.05	2.55	3.05	3.55
180	1.15	1.65	[illegible]	2.65	3.15	[illegible]

Figure 21d

- **Bend radius must be in ascending order (Figure 21d2)**

Type:	Steel Gauge Table	
Process	Chads K-factor Gauge Table	
Bend Type:	K-Factor	
Unit:	inches	
Material:	Steel and Alum	
Gauge No.	20 Gauge Steel	
Thickness:	0.036	
Angle	0.036	0.048
1	0.4500	0.4557
90	0.4500	0.4557
180	0.4500	0.4557

Ascending Order

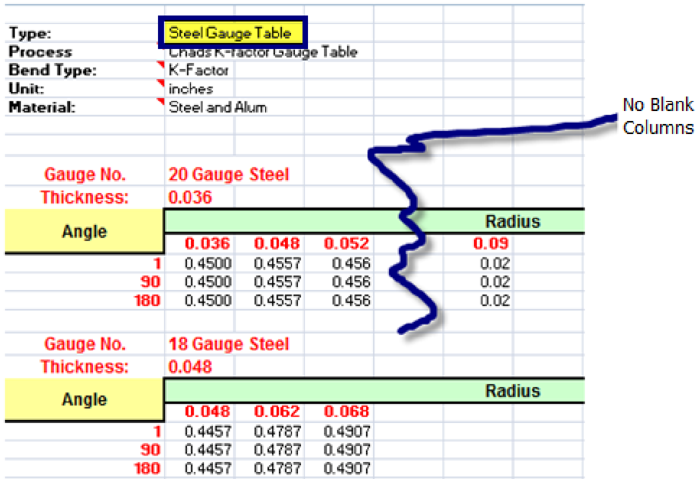
Figure 21d2

- **Material thickness must be in ascending order from top to bottom (Figure 21d3)**

Type:	Steel Gauge Table				
Process	Chads K-factor Gauge Table				
Bend Type:	K-Factor				
Unit:	inches				
Material:	Steel and Alum				
Gauge No.	20 Gauge Steel				
Thickness:	0.036				
Angle					Radius
	0.036	0.048	0.052		0.09
1	0.4500	0.4557	0.456		0.02
90	0.4500	0.4557	0.456		0.02
180	0.4500	0.4557	0.456		0.02
Gauge No.	18 Gauge Steel				
Thickness:	0.048				
Angle					Radius
	0.048	0.062	0.068		
1	0.4457	0.4787	0.4907		
90	0.4457	0.4787	0.4907		
180	0.4457	0.4787	0.4907		

Figure 21d3

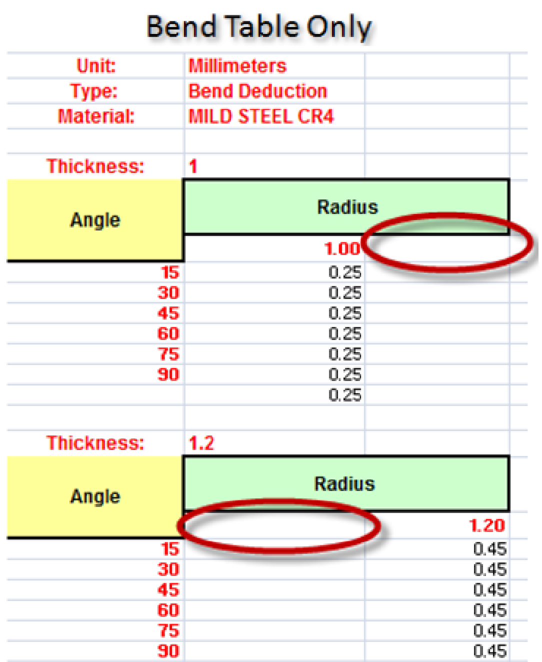
- **Gauge tables cannot contain blank columns (Figure 21d4)**



Type:	Steel Gauge Table				
Process	Chads K-Factor Gauge Table				
Bend Type:	K-Factor				
Unit:	inches				
Material:	Steel and Alum				
Gauge No.	20 Gauge Steel				
Thickness:	0.036				
Angle				Radius	
	0.036	0.048	0.052		0.09
1	0.4500	0.4557	0.456		0.02
90	0.4500	0.4557	0.456		0.02
180	0.4500	0.4557	0.456		0.02
Gauge No.	18 Gauge Steel				
Thickness:	0.048				
Angle				Radius	
	0.048	0.062	0.068		
1	0.4457	0.4787	0.4907		
90	0.4457	0.4787	0.4907		
180	0.4457	0.4787	0.4907		

Figure 21d4

- **Bend radius column values cannot be in blank (Figure 21d5)**



Unit:	Millimeters	
Type:	Bend Deduction	
Material:	MILD STEEL CR4	
Thickness:	1	
Angle	Radius	
	1.00	
15	0.25	
30	0.25	
45	0.25	
60	0.25	
75	0.25	
90	0.25	
	0.25	
Thickness:	1.2	
Angle	Radius	
		1.20
15		0.45
30		0.45
45		0.45
60		0.45
75		0.45
90		0.45

Figure 21d5

e) Handling bend table common problems

About 90% of all bend table problems occur for two reasons:

- The bend table is using a non-valid format.
- The thickness of the part falls outside of the bend table parameters.

The first problem relates to the breaking of bend table rules discussed in section 21d.

The second problem relates to how the part parameter values "fit" within the bend table parameters. Two of the most common issues in this area include:

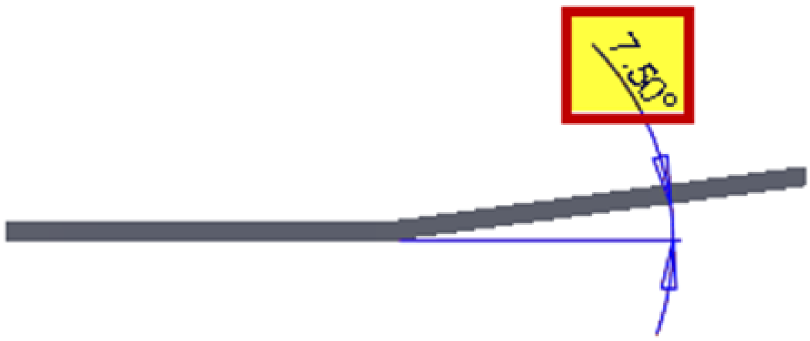


Figure 21e

1) When inserting a bend table (Figure 21e1) into the part (Figure 21e), the following message appears: *"The angle of this table fell outside the bend table…"*

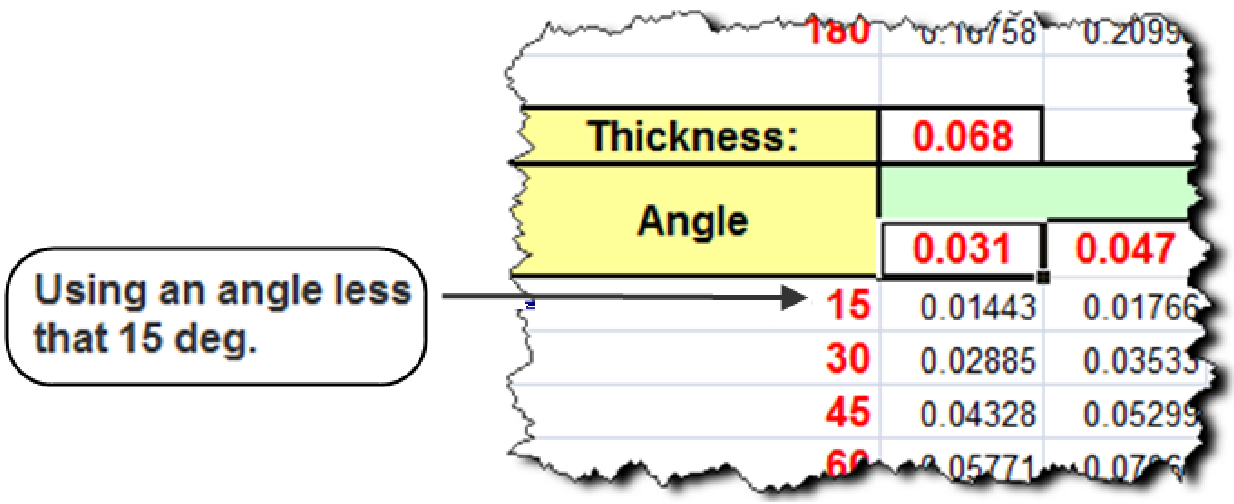


Figure 21e1

The problem here is that the part has a 7.50˚ bend angle. The minimum bend angle for the bend table is 15˚. This means that the 7.50˚ bend angle falls "outside" of the bend table parameters. Therefore, the table cannot interpolate without an angle value less than 7.5 deg. and 15 deg.

2) The second problem relates to the precision of the table and part.

The images in Figure21e2 depict a typical case where the part precision is two decimal places, but the table precision is three decimal places.

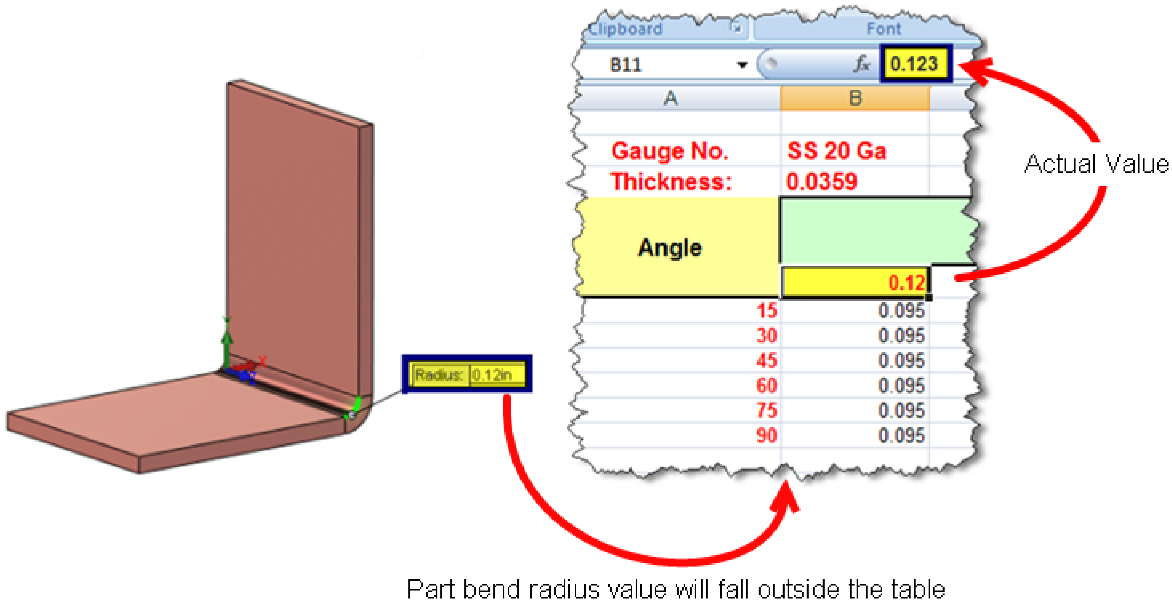


Figure 21e2

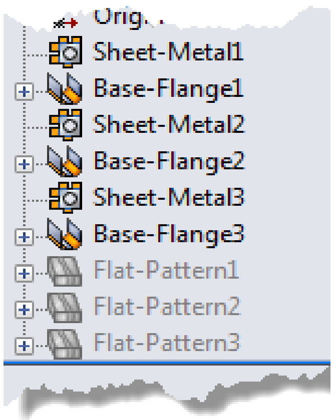
Although the bend radius value in the table appears to be 0.12, the actual value is a three place decimal value of 0.123 in. This value appears in the Excel formula bar. Therefore, the part bend radius value with an actual precision of 2 decimal places falls outside of the table, because there is no value between X <= 0.12 & 0.123.

22) Recreate Templates that Produce Old Sheet Metal Architecture Parts

As mentioned in section 3, SOLIDWORKS 2013 introduced a new architecture for sheet metal functionality. This change fixes a wide variety of problems. Therefore, it is very important to use the latest SOLIDWORKS versions to create new parts.

It is a common practice to reuse parts that were created in a pre-2013 version of SOLIDWORKS, as a template for new parts. However, doing so produces a sheet metal part with the old architecture. This is why it is important to ensure that all "new" sheet metal parts have the new architecture.

For example, imagine that you create a part from an existing part template. Afterward, the FeatureManager tree looks like the following image:



In this image, the FeatureManager tree shows that the **Sheet Metal** and **Flat Pattern** features are not contained within folders. This means that the part you are creating has the old architecture. In this case, you must recreate a new part template from scratch. If the part template has geometry that is necessary in a new template, then you must recreate the geometry in the new template.

One of the benefits of working with the new sheet metal architecture is that all of the new sheet metal features developed since the release of SOLIDWORKS 2013 work only with the new architecture.

23) Normal Cut with the ’Optimize Geometry’ Option

Only use the **Optimize Geometry** option if the normal cut does not produce the result you want.

24) Always Remove "Self-Intersection"

Section 7 recommends avoiding self-intersection. Self-intersection only displays when the **Verification on rebuild** option is active.

If you encounter self-intersection after creating a specific feature, it is important to remove the self-intersection before recreating the feature. This means that you must create the conditions to avoid self-intersection prior to creating the feature.

25) Flattened Mass

A sheet metal part retains the same weight regardless of how you fold or unfold the part (the flat pattern).

However, in SOLIDWORKS, the surface area of a part might increase or decrease depending on the bend allowance. Therefore, a sheet metal body could have two weights. One weight in the folded state and a different weight in the flat pattern. The regular mass displays the weight of the part in the folded state. The flattened mass is the weight in the flat pattern state.

26) Calculating the K-Factor for Swept Flanges

There are improvements in SOLIDWORKS to the calculation of the K-factor for swept flanges. However, there are some circumstances where SOLIDWORKS cannot calculate the K-factor. For more information, see the topic “K-Factor Bend Allowances in Swept Flanges” in the SOLIDWORKS Online Help.

27) Normal Cut 'Optimize Geometry' Option

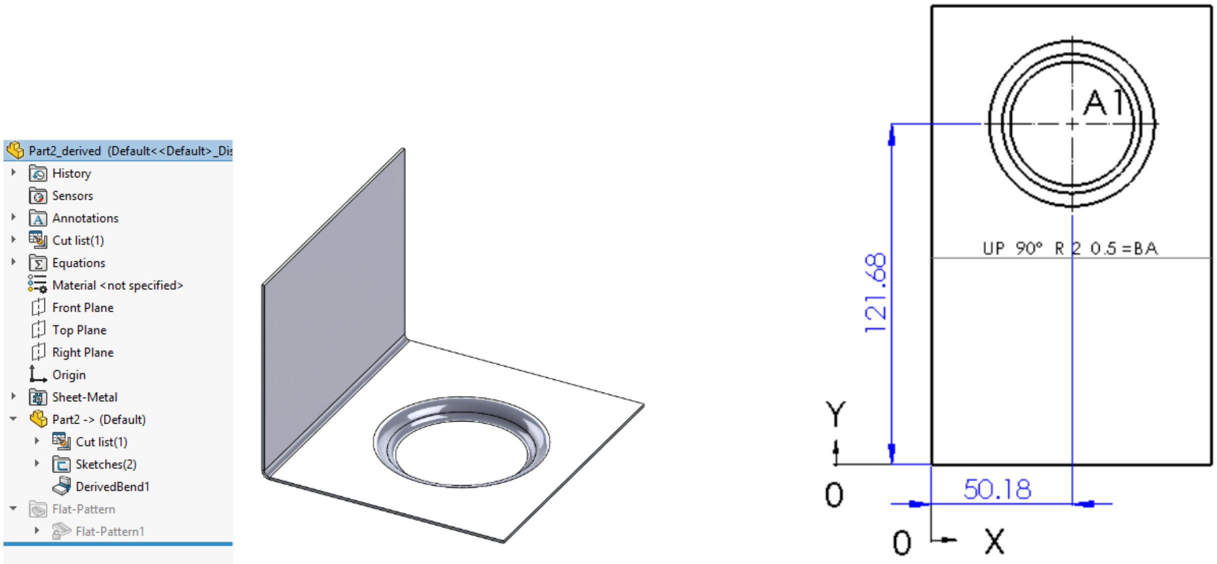
This option is available because in some circumstances, the normal cut could produce incorrect results. Use this option only when the normal cut fails or produces incorrect results.

28) Tab and Slot Feature in Assembly Components

When creating a tab and slot features in the context of an assembly, SOLIDWORKS creates the tab feature in one component and the slot feature in another component. Because the tab and slot features are dependent on each other to function correctly, it is important to maintain the in-context relationship between them. Breaking external references between the components produces a failure of the feature.

29) Add Punch Table to Drawing Views of Derived Sheet Metal Parts

SOLIDWORKS imports form tool sketches from a seed part to a derived (mirrored or inserted) part. SOLIDWORKS imports these sketches when the **Sheet metal information** option is active.



TAG	X LOCATION	Y LOCATION	ANGLE(X AXIS)	QUANTITY
A1	50.18	121.68	90°	1

Figure 29

30) Turn On or Turn Off Override Default Parameters

When you create the first sheet metal body (such as a Base Flange feature), the feature parameters (thickness, bend allowance, and relief) pass to the part. From that point on, all other sheet metal body parameters that you create in that part are by default, controlled by the part. That is, by the upper-level sheet metal feature. For example, changing the upper-level sheet metal feature thickness changes the thickness of all bodies in the part. In SOLIDWORKS 2017, there is a new **Override default parameters** document property for **New Sheet Metal Bodies**.

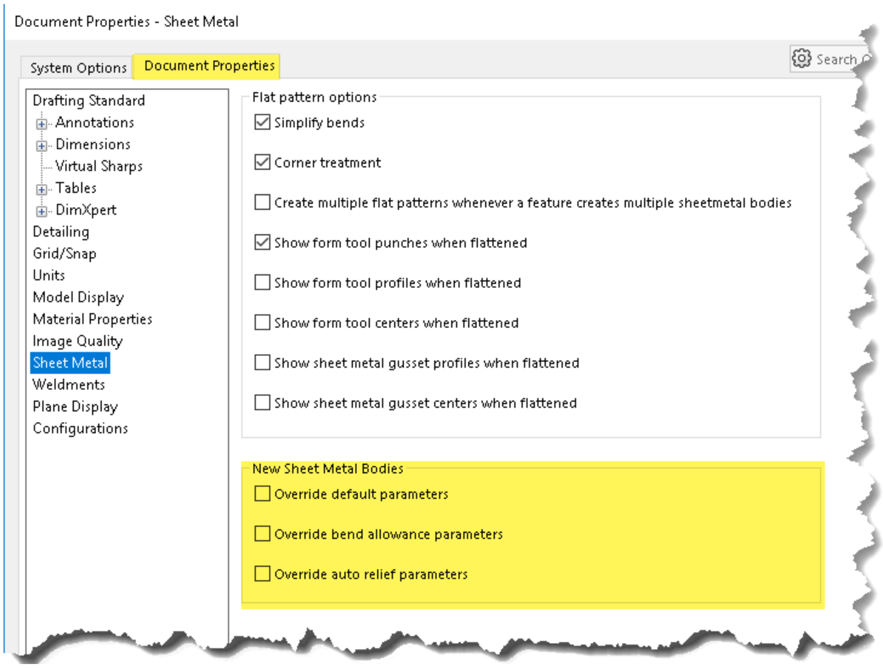


Figure 30

As mentioned, the first body you create in the part will receive the sheet metal parameters from the part. When you activate any of these new options, the respective "Override …" option is also active for the second body in the part. This means that this body and all successive bodies you create in this part will have independent parameters from the part parameters.

31) 3 Bend Corner Reliefs

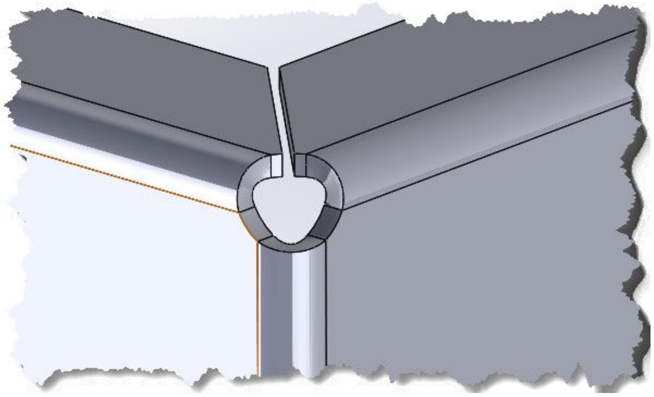
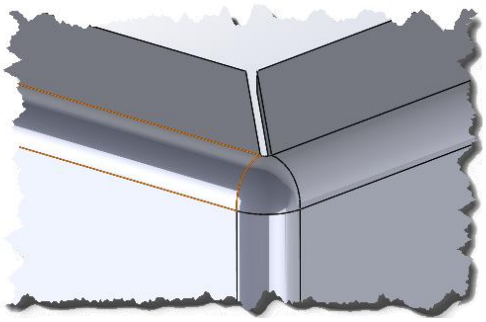
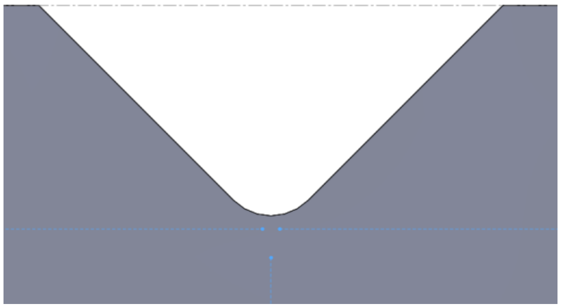


Figure 31

3 Bend Corner Relief –Full Round Corner

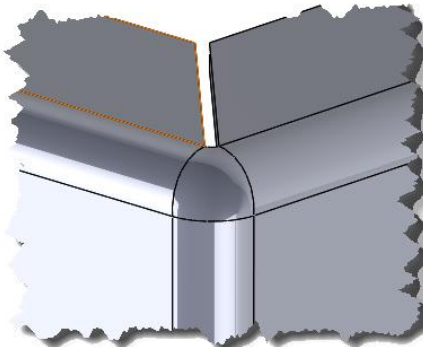
Full Round Flat Pattern

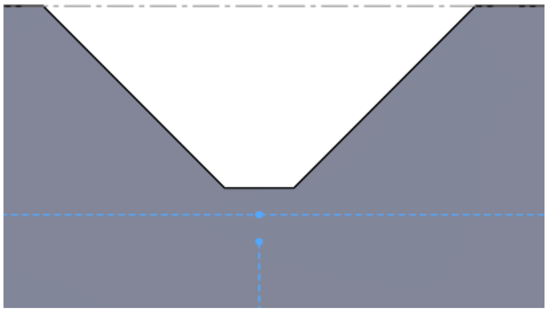




3 Bend Corner Relief – Suitcase

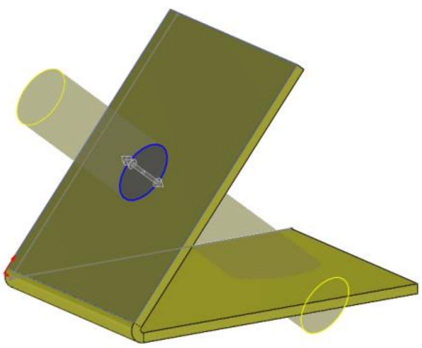
Suitcase Flat Pattern

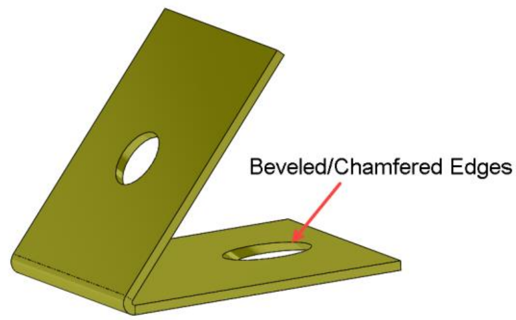




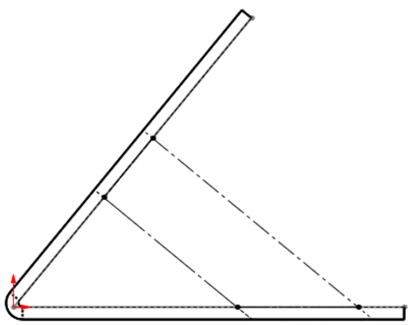
32) Normal Cut Standalone Feature

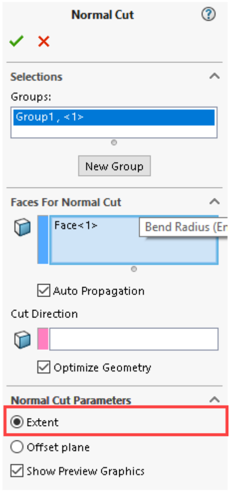
Section 5 discusses use of the **Normal Cut** option when creating a Cut-Extrude feature in a sheet metal body. However, there are times when other types of cuts are necessary when the normal cut option is not available.

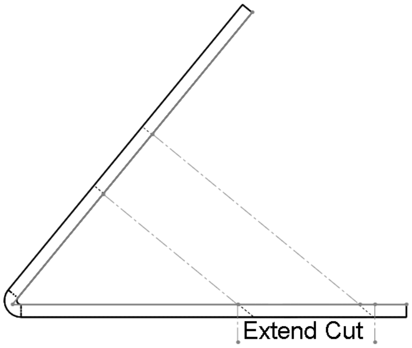


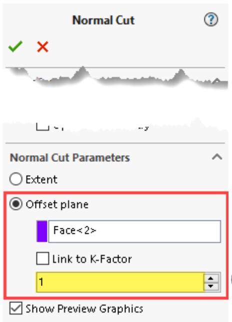


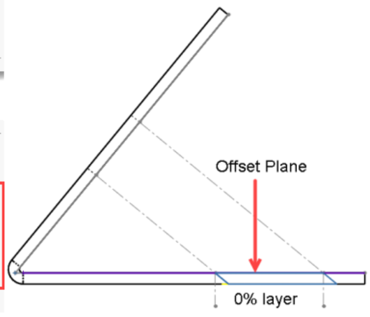
The **Faces for Normal Cut** option requires chamfered or beveled faces.

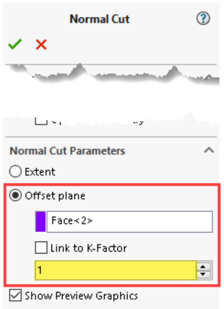












33) Linking Materials and Sheet Metal Parameters

The use of this feature requires creation of the needed material in the **Custom Material** folder. Upon opening the part template, you first must insert the material from the **Custom Material** folder then, proceed to create the sketch and part. If you have specific materials for sheet metal parts, you can save each of those materials in a part template.

You can use this area to work with sheet metal tables. However, most importantly, you can create your own **Thickness Range** table. This area allows you to define the part **Bend Allowance** based on the thickness of the part. You can create this table in place, or you can create the table in Excel and then import it in SOLIDWORKS.

Thickness Range

Add Remove Export Import

	From	<	Thickness	<=	To	Unit	Bend Allowance	Value
1	0.000000	<	Thickness	<=	0.062500	inches	K-Factor	0.425000
2	0.062500	<	Thickness	<=	0.125000	inches	K-Factor	0.500000
3	0.125000	<	Thickness	<=	0.187500	inches	K-Factor	0.575000
4	0.187500	<	Thickness	<=	0.250000	inches	K-Factor	0.650000
5	0.250000	<	Thickness	<=	0.312500	inches	K-Factor	0.725000

Figure 33