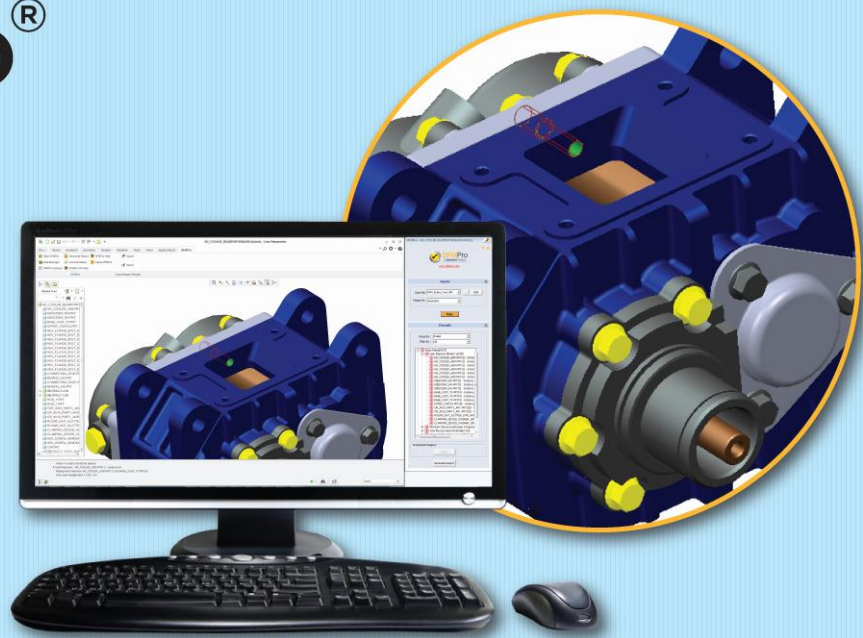
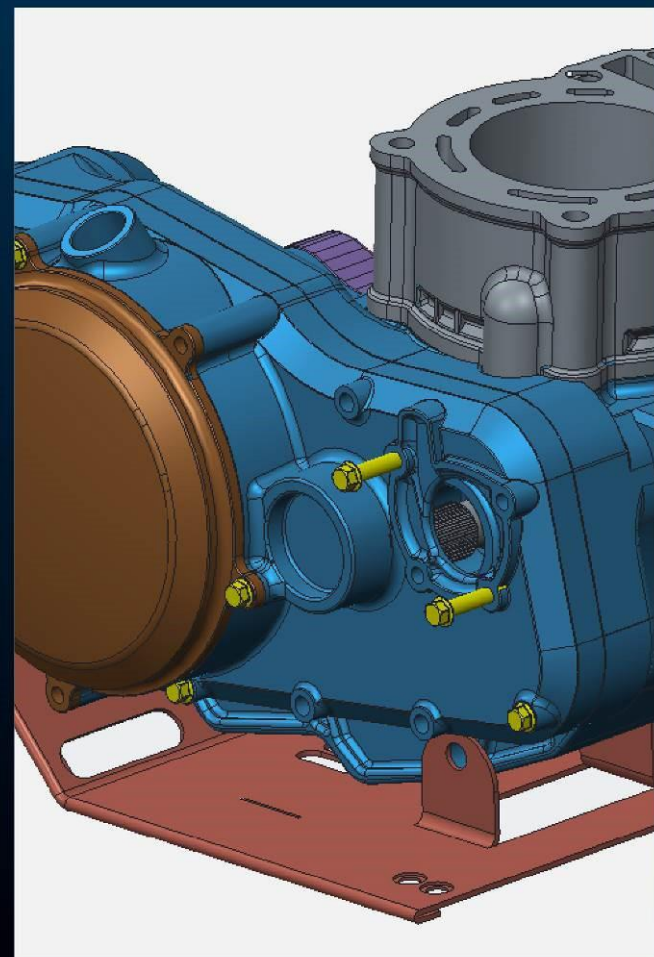
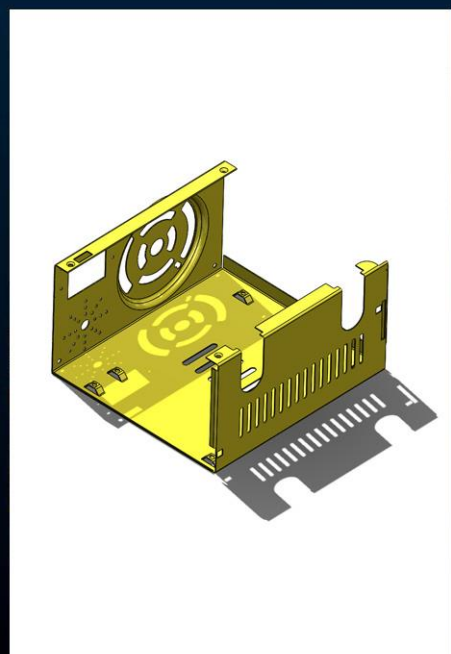
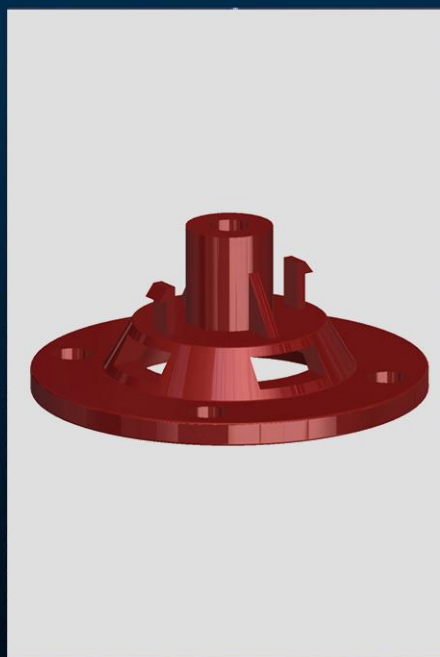
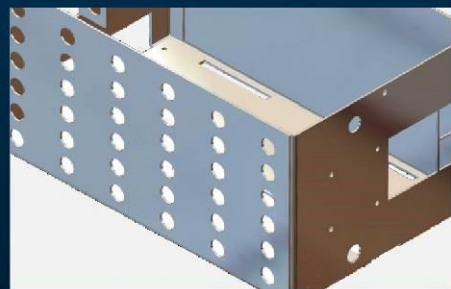




An HCL Technologies Product



A DEFINITIVE GUIDE TO DESIGN FOR MANUFACTURING SUCCESS



Copyright Notice

© HCL Technologies Ltd. All rights reserved.

No part of this document (whether in hardcopy or electronic form) may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise, to any third party without the written permission of HCL Technologies Limited. HCL Technologies Limited reserves the right to change the information contained in this document without prior notice.

The names or trademarks or registered trademarks used in this document are the sole property of the respective owners and are governed/ protected by the relevant trademark and copyright laws.

This document is provided by HCL Technologies Limited for informational purposes only, without representation or warranty of any kind, and HCL Technologies Limited shall not be liable for errors or omissions with respect to the document. The information contained herein is provided on an “AS-IS” basis and to the maximum extent permitted by applicable law, HCL Technologies Limited hereby disclaims all other warranties and conditions, either express, implied or statutory, including but not limited to, any (if any) implied warranties, duties or conditions of merchantability, of fitness for a particular purpose, of accuracy or completeness of responses, of results, of workmanlike effort, of lack of viruses, and of lack of negligence, all with regard to the document.

THERE IS NO WARRANTY OR CONDITION OF NON-INFRINGEMENT OF ANY INTELLECTUAL PROPERTY RIGHTS WITH REGARD TO THE DOCUMENT. IN NO EVENT WILL HCL TECHNOLOGIES LIMITED BE LIABLE TO ANY OTHER PARTY FOR LOST PROFITS, LOSS OF USE, LOSS OF DATA, OR ANY INCIDENTAL, CONSEQUENTIAL, DIRECT, INDIRECT, OR SPECIAL DAMAGES WHETHER UNDER CONTRACT, TORT, WARRANTY, OR OTHERWISE, ARISING IN ANY WAY OUT OF THIS DOCUMENT, WHETHER OR NOT SUCH PARTY HAD ADVANCE NOTICE OF THE POSSIBILITY OF SUCH DAMAGES.

Contents

<i>Form Feature Design Guidelines</i>	5
Minimum Distance from Dimple to Bend	6
Minimum Distance from Dimple to Cutout	7
Minimum Distance between Dimples	8
Minimum Distance from Dimple to Hole	9
Minimum Distance from Dimple to Part Edge	10
Maximum Embossment Depth	11
Gussets	12
Minimum Distance from Hole to Gusset.....	13
Minimum Distance from Extruded Hole to Part Edge.....	14
Minimum Distance between Extruded Holes	15
Minimum Distance from Extruded Hole to Bend.....	16
<i>Curl and Lance Design Guidelines</i>	17
Curl Radius.....	18
Minimum Distance between Curl and Hole	19
Minimum Depth of Lance.....	20
Minimum Distance from Bend to Lance	21
Minimum Distance from Hole to Lance	22
Minimum Spacing between Lances.....	23
<i>Hem Design Guidelines</i>	24
Open Hem	25
Rolled Hem	26
Tear Drop Hem	27
Rolled Hem to Hole Edge Distance.....	28
Closed Hem	29
Knife Edge.....	30
<i>Notch and Tab Parameters</i>	31
Notch Parameters	32
Notch to Bend Distance	33
Minimum Distance between Notches.....	34
Minimum Distance between Notch to Hole	35
Tab Parameters	36
Minimum Distance between Tabs	37
<i>Burring Hole Design Guidelines</i>	38
Burring Hole Height.....	39
Burring Hole Inner Diameter	40

Burring Hole Distance to Bends	41
Burring Hole Distance to Part Edge.....	42
Burring Hole Distance to Cutout	43
Burring Hole Spacing	44
General Design Guidelines	45
Preferred Sheet Sizes	46
Minimum Hole Diameter	47
Interference in Flat Pattern.....	48
Half-Shear Parameters	49
Hole To Part Edge Distance	50
Minimum Bend Radius	51
Bend Relief	52
Minimum Hole Diameter	53
Curls Feature	54
Hem Feature.....	55
Notch Feature	56
Bend Relief Parameters.....	57
Spacing Between Holes in Perforated Metal	58
Hole to Bend Distance in Perforated Metal.....	59
Offset Bend Parameters.....	60
Distance between Offset Bends and Extruded hole	61

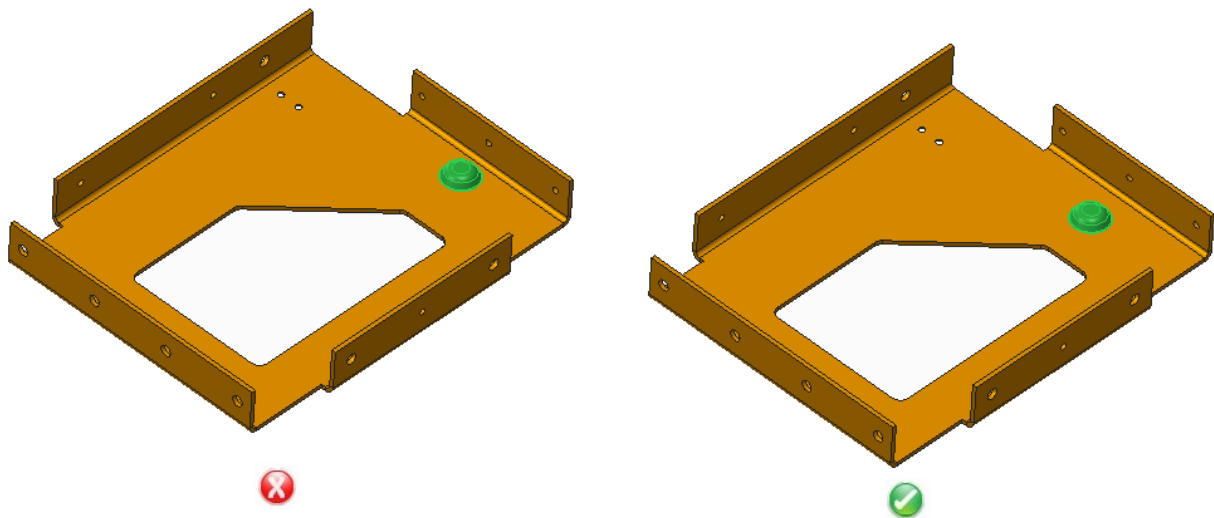
Sheet Metal Design Guidelines

Form Feature Design Guidelines

Minimum Distance from Dimple to Bend

A certain minimum distance must be maintained between dimple and the bend feature to avoid deformation and fracture of the metal.

It is recommended that the minimum distance between dimple to bend should be two times sheet thickness plus the inside radius of the dimple plus radius of the bend.

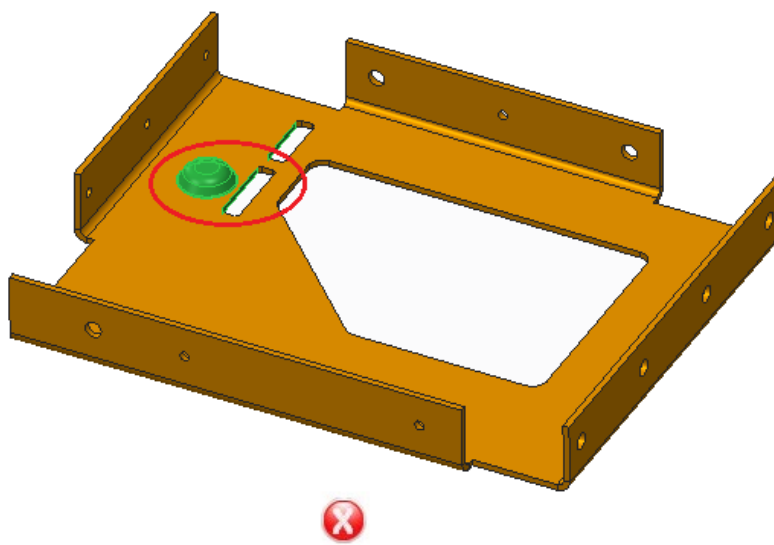


t = Sheet metal thickness

Minimum Distance from Dimple to Cutout

A minimum distance must be maintained between dimple and cutout edge to avoid deformation and fracture of the metal.

It is recommended that a minimum distance of four times the sheet thickness plus the inside radius of each dimple must be maintained between a dimple and cutout edge.



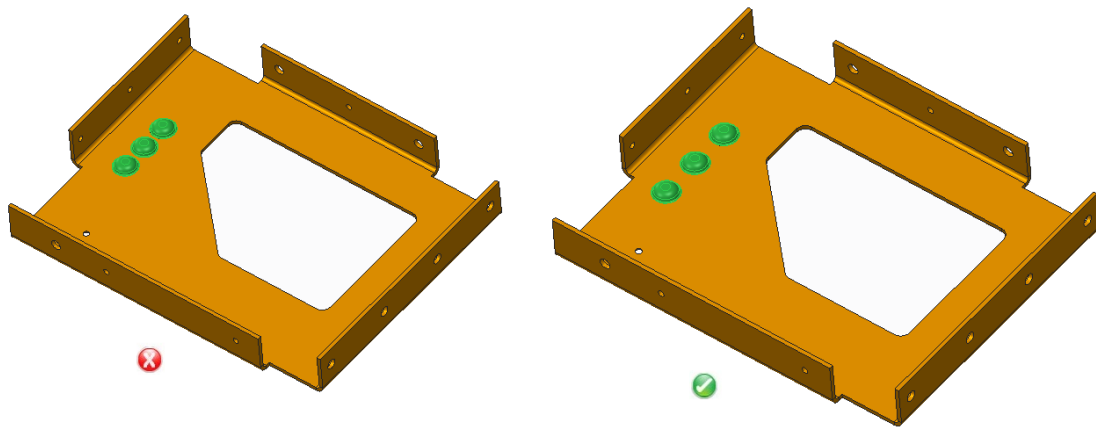
t
=

Sheet metal thickness

Minimum Distance between Dimples

Care need to be taken when placing formed features close to each other. If a station does not clear a form already placed in a part, the form will get flattened out.

It is recommended that the minimum distance between dimples should be four times sheet thickness plus radius of the dimple.

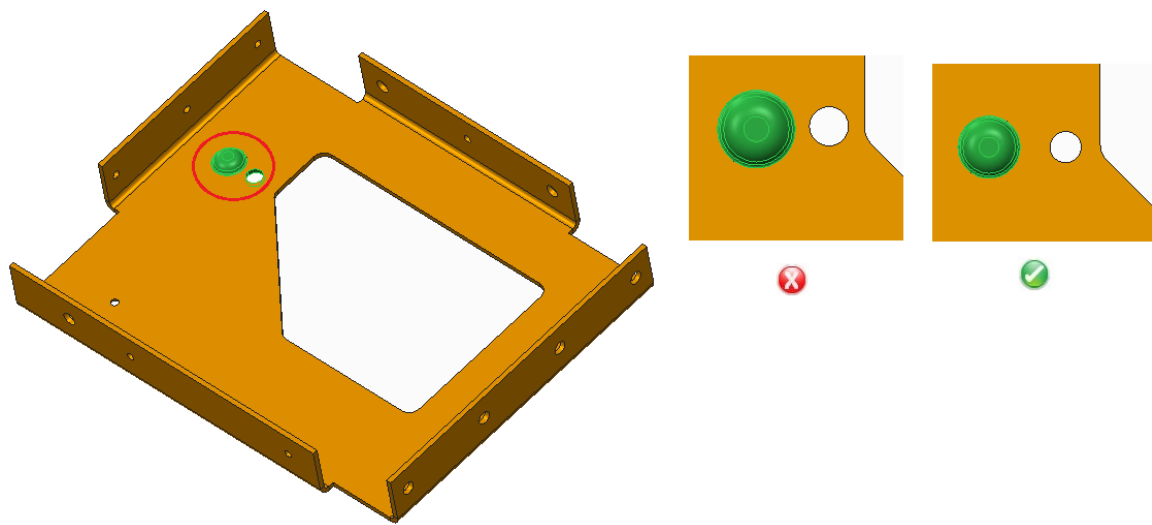


t = Sheet metal thickness

Minimum Distance from Dimple to Hole

To avoid deformation and fracture of the metal a certain minimum distance should be maintained between dimple and adjacent holes

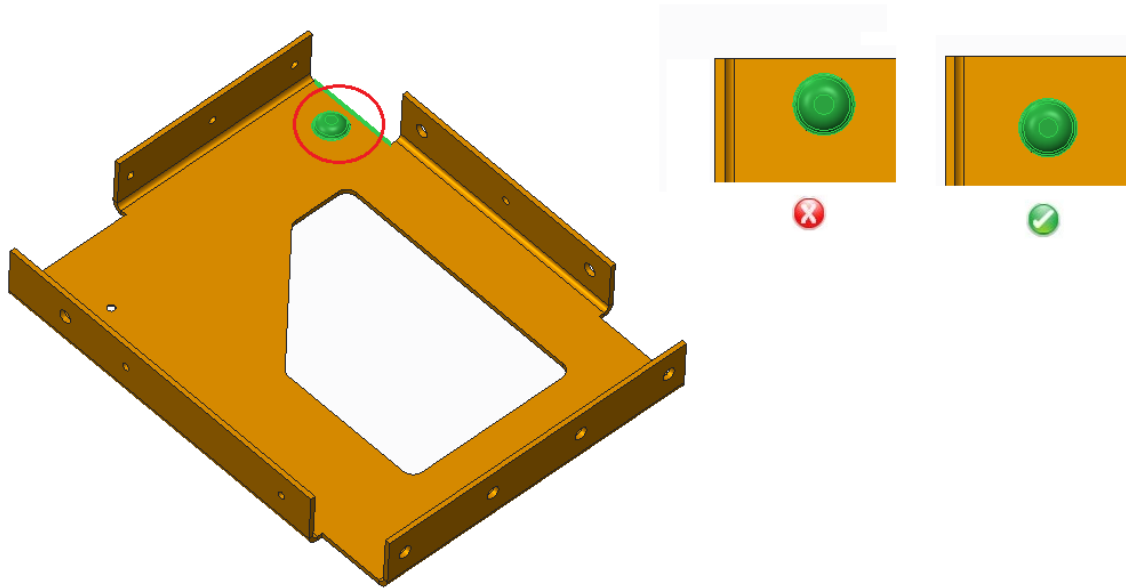
It is commonly recommended that the minimum distance between dimple and hole should be three times sheet thickness.



t = Sheet metal thickness

Minimum Distance from Dimple to Part Edge

It is recommended that the minimum distance between dimples to part edge should be four times material thickness plus radius of the dimple to avoid deformation and fracture of the metal.

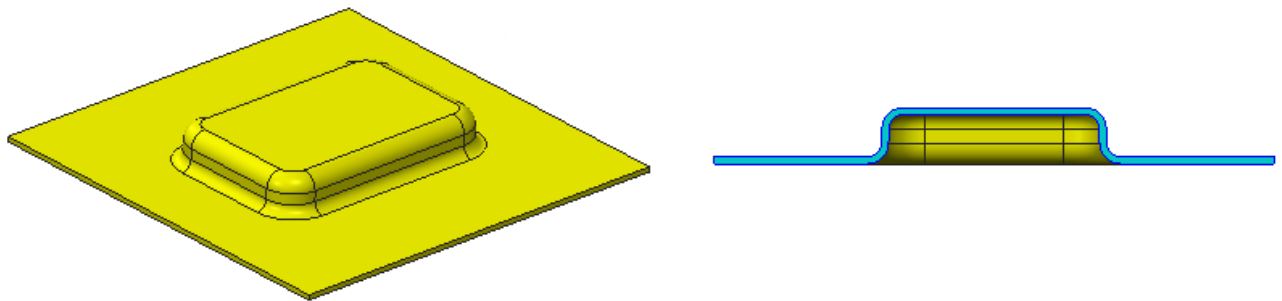


t = Sheet metal thickness

Maximum Embossment Depth

Embosses are small, shallow formed projections on the surface of stamped parts. During this operation, stretching is the main deformation mode resulting in high tension. Thereby the metal is subject to excessive thinning or fracturing. Consequently, the depth of the embossed feature is restricted by the material's thickness and ability to stretch in addition to the emboss geometry.

It is recommended that the maximum depth of embossment be less than or equal to three times material thickness.

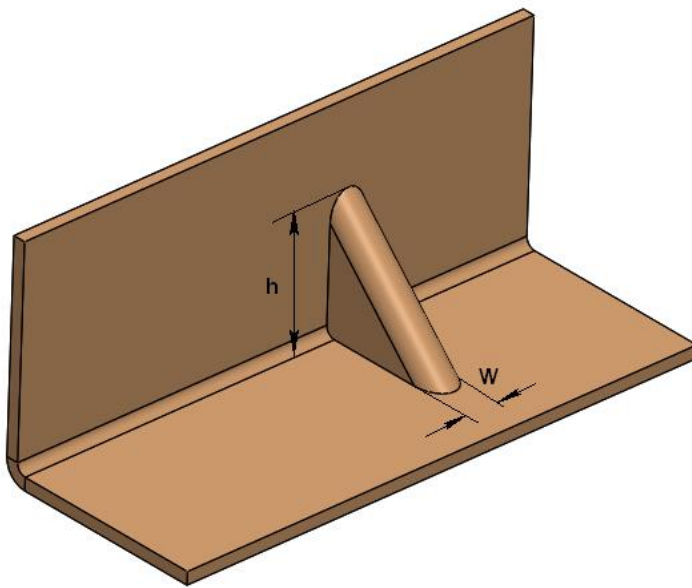


t = Sheet metal thickness

Gussets

Gussets are used to strengthen a flange without the need for secondary processes such as welding.

A general guideline suggests that the width and depth of gusset at an angle of 45 degrees is directly proportional to the radius and material thickness.



t = Sheet metal thickness

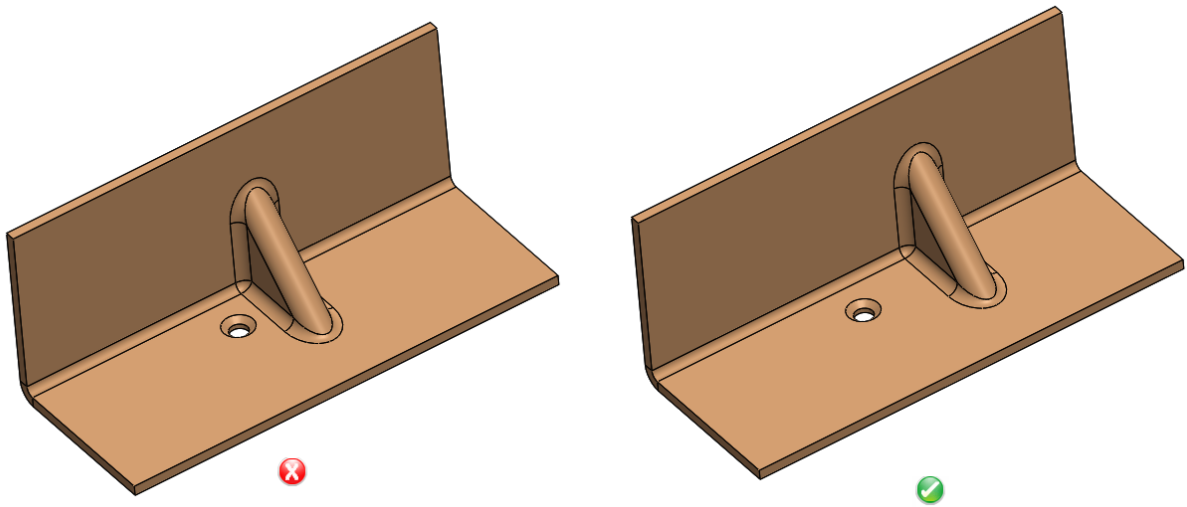
h = depth of gussets

W = width of gussets

Minimum Distance from Hole to Gusset

A certain distance must be maintained between a hole and gusset to avoid metal deformation and fracturing.

It is recommended that minimum distance between hole edge to gusset should be at least eight times the material thickness.

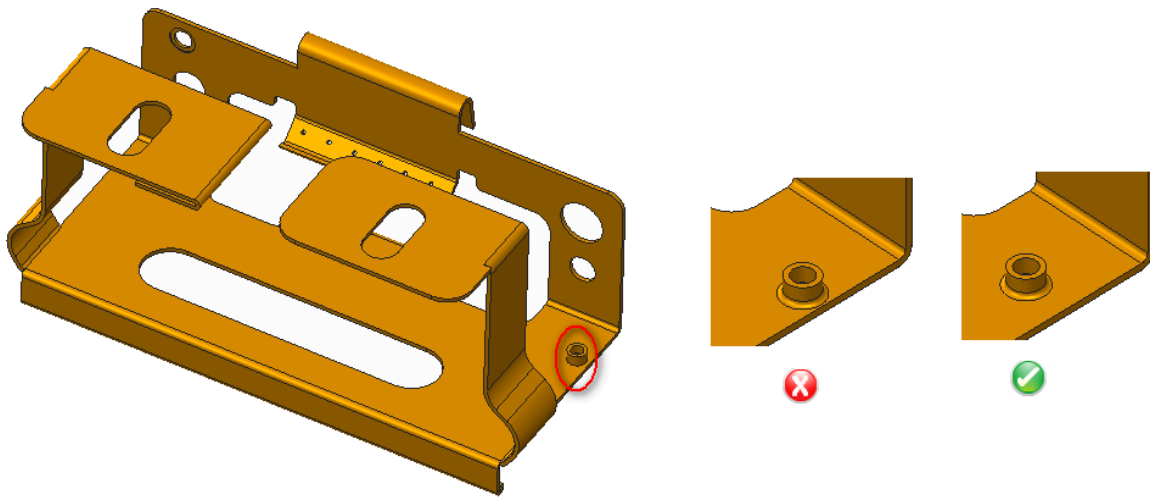


t = Sheet metal thickness

Minimum Distance from Extruded Hole to Part Edge

Extruding metal is one of the most extreme pressure applications in press working and generates lot of friction and heat. If an extruded hole is too close to the part edge, it can lead to deformation or tearing of the metal.

It is recommended that the minimum distance between the extruded holes to part edge should be at least three times the thickness of sheet.

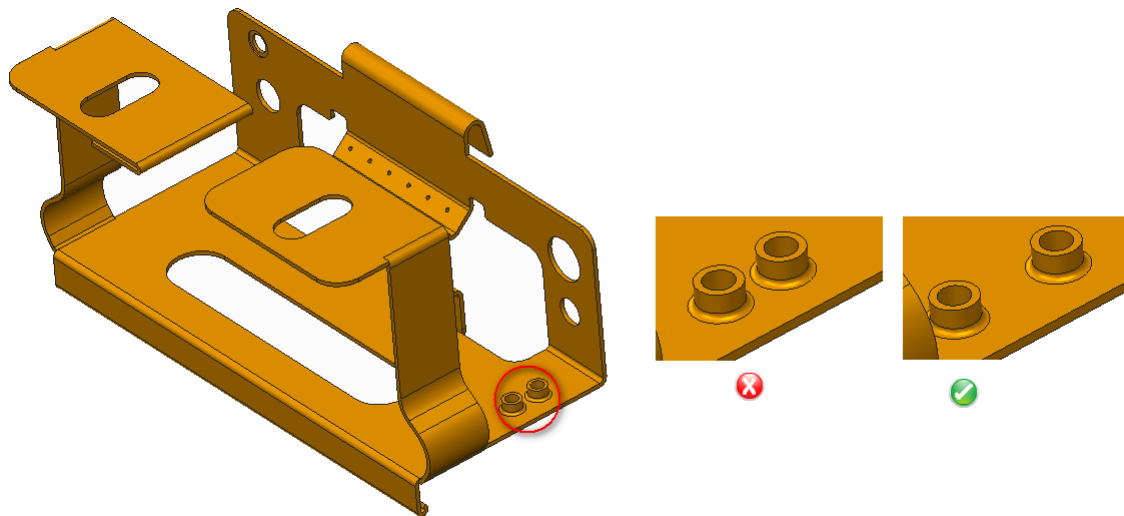


t = Sheet metal thickness

Minimum Distance between Extruded Holes

Certain distance should be maintained between two extruded holes in sheet metal designs. If extruded holes are too close it can lead to metal deformation.

It is recommended that the minimum distance between two extruded holes should be six times the thickness of sheet metal.

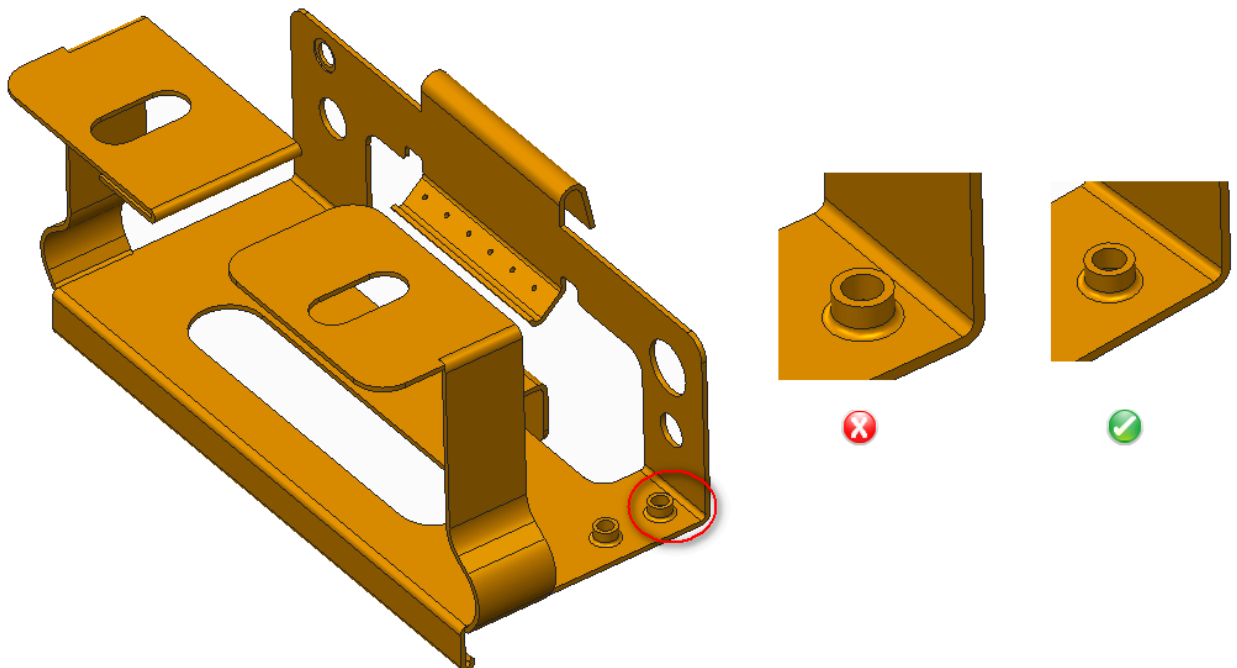


t = Sheet metal thickness

Minimum Distance from Extruded Hole to Bend

A certain distance must be maintained between extruded hole to bend to avoid metal deformation and fracturing.

It is recommended that the minimum distance between extruded hole and bend should be three times the thickness of sheet metal plus bend radius.



t = Sheet metal thickness

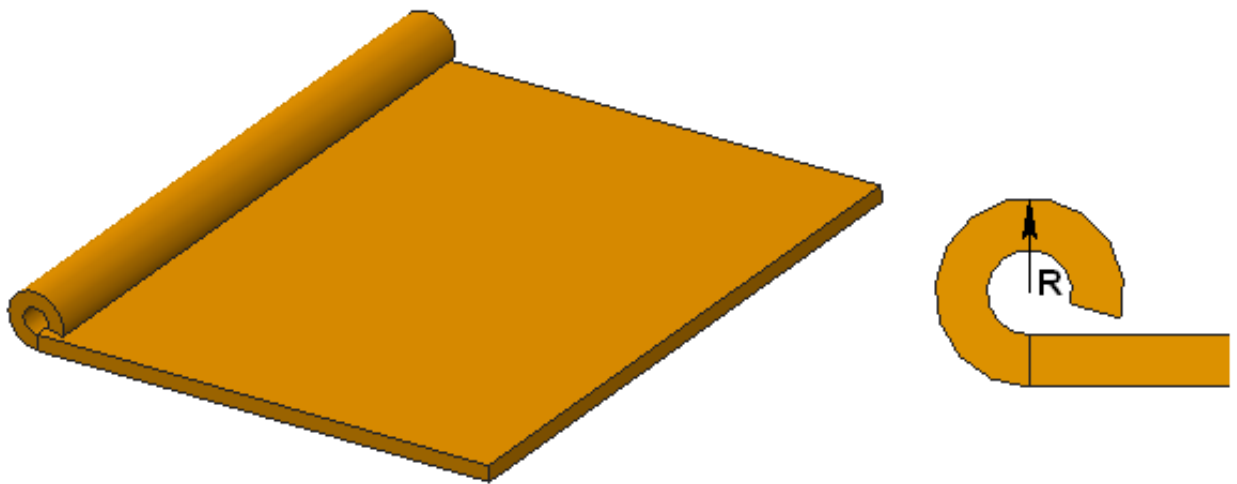
Curl and Lance Design Guidelines

Curl Radius

Curling is the process of forming the sheet metal flange into a rolled shape. Curling strengthens the edges and provides smoothness to the surface. It is commonly used as a means of joining two components.

Curls are often added to avoid sharp edges and make parts safer for handling and use.

It is recommended that the outside radius of curl should be minimum 2X of the material thickness.

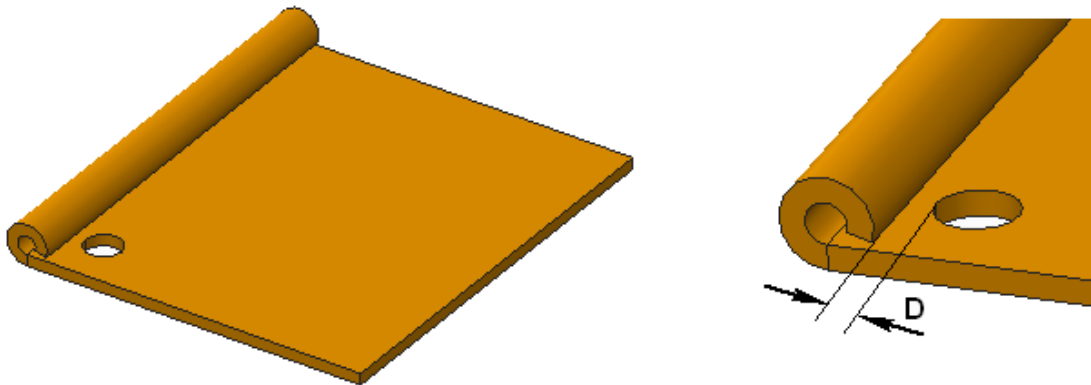


t = Sheet metal thickness

R = Outside radius

Minimum Distance between Curl and Hole

It is recommended that the minimum distance between a curl and the edge of a hole should be sum of curl radius and material thickness.



t = Sheet metal thickness

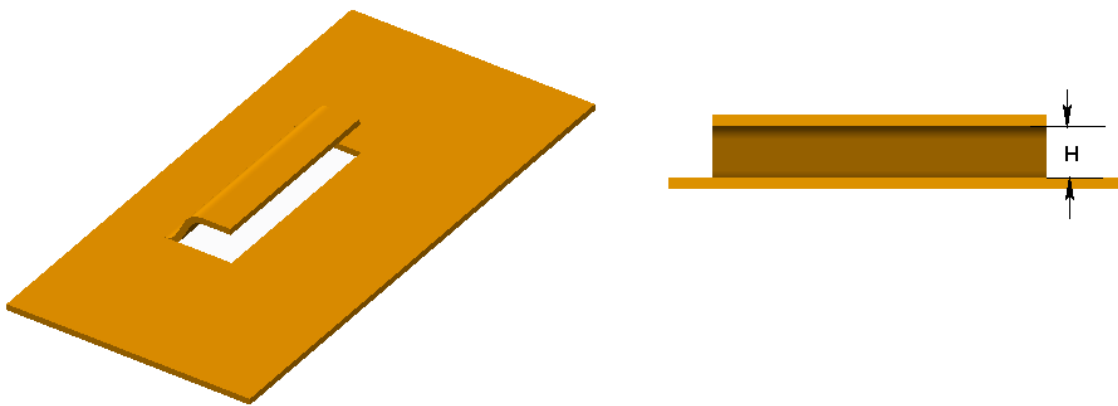
D = Distance

Minimum Depth of Lance

Lancing is a piercing operation in which the work piece is sheared and bent with strike of a die. In this process there is no material removal however it only modifies the geometry. Lancing can be used to make partial contours and free up material for other operations.

Lancing is used to make tabs, vents and louvers.

It is recommended that the minimum depth of lance should be 2X the material thickness.



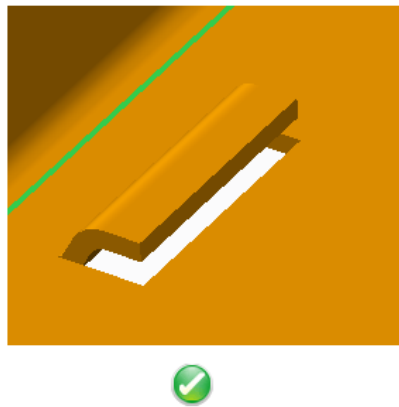
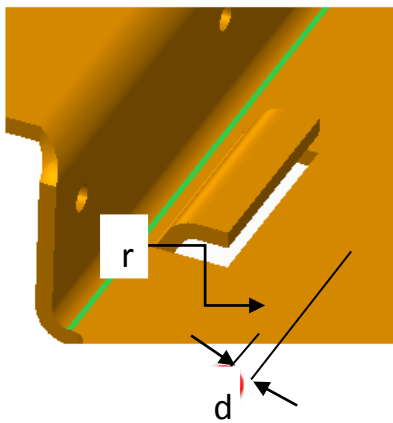
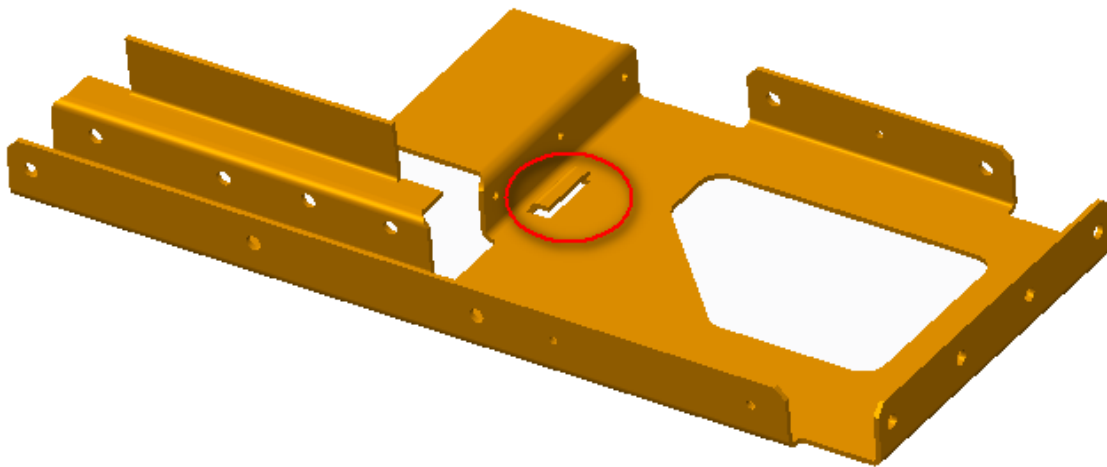
t = Sheet metal thickness

H = Depth of lance

Minimum Distance from Bend to Lance

During lancing operation a sufficient degree of clearance should be given around the lance feature and bend.

It is recommended that minimum distance between lance and bend should be $3X$ the material thickness plus bend radius.

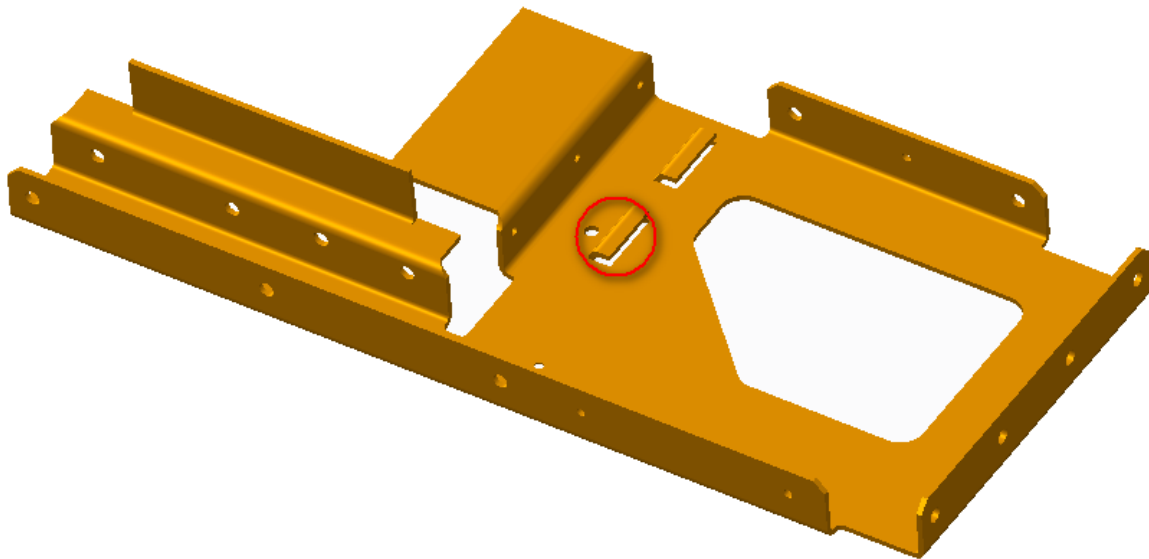


t = Sheet metal thickness
 r = Inside bend radius

Minimum Distance from Hole to Lance

During lancing operation we need to maintain sufficient degree of clearance around the lance feature.

It is recommended that the distance between lance and hole should be 3X the material thickness.

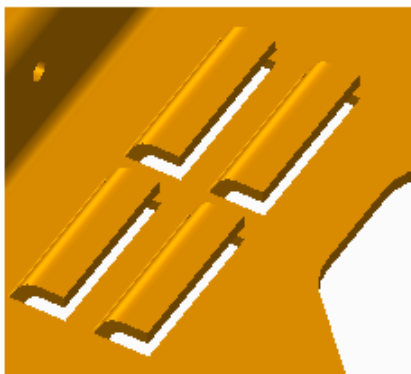
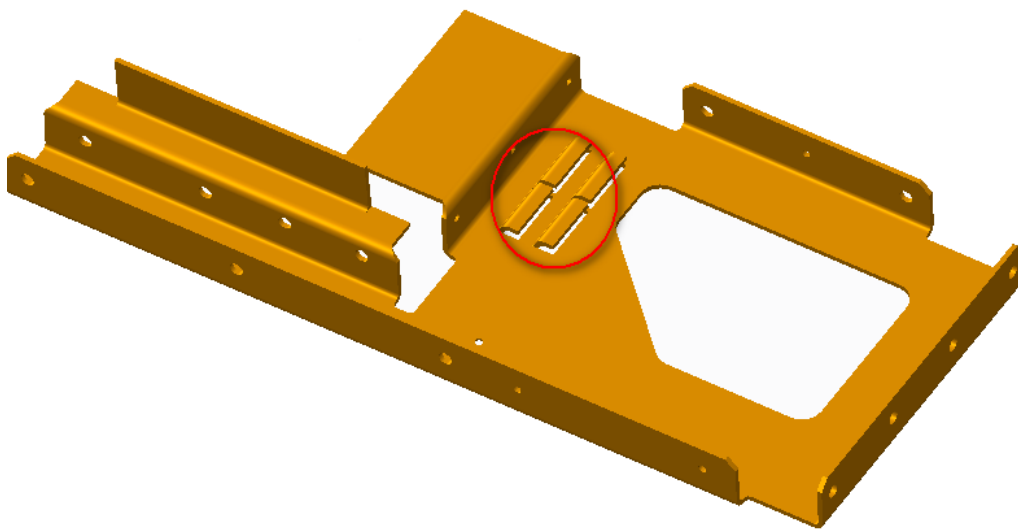


t = Sheet metal thickness

Minimum Spacing between Lances

During lancing operation sufficient degree of clearance should be maintained around the lance feature as the punch and die will need some degree of clearance around the feature in order to hold down the work piece during operation. If another lance is placed inside this working envelope it will be crushed by the punch and die, potentially damaging the work piece and tools.

It is recommended to maintain sufficient clearance between two lance features by considering die and punch clearance allowance.



t = Sheet metal thickness

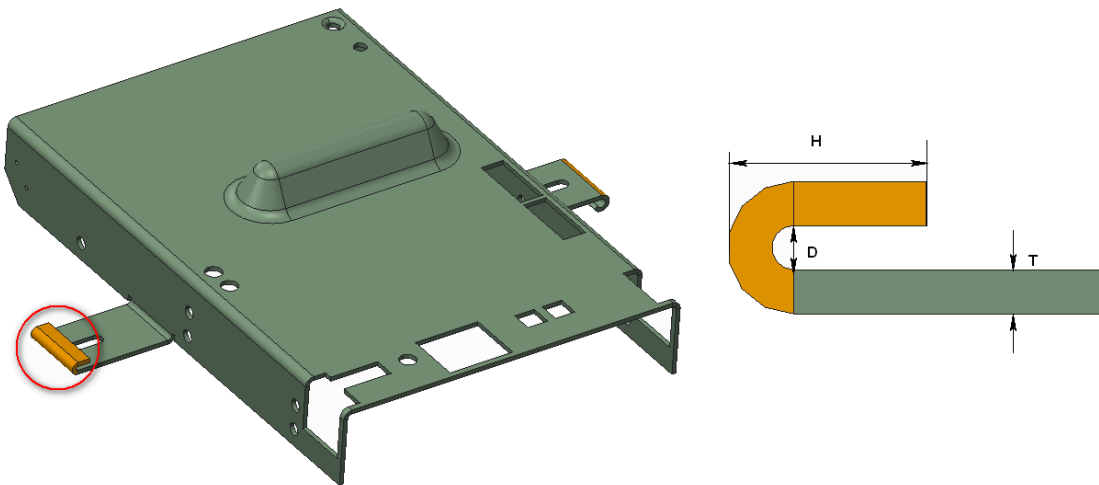
Hem Design Guidelines

Open Hem

A Hem has a full round feature and a return flange. In open hem bend angle is equal to 180 degrees.

It is recommended that the ratio of the open hem radius to the sheet metal thickness should be greater than or equal 0.5.

Also, the ratio of return flange height to the sheet metal thickness should be greater than or equal to 4.



Where,

H = Return flange height

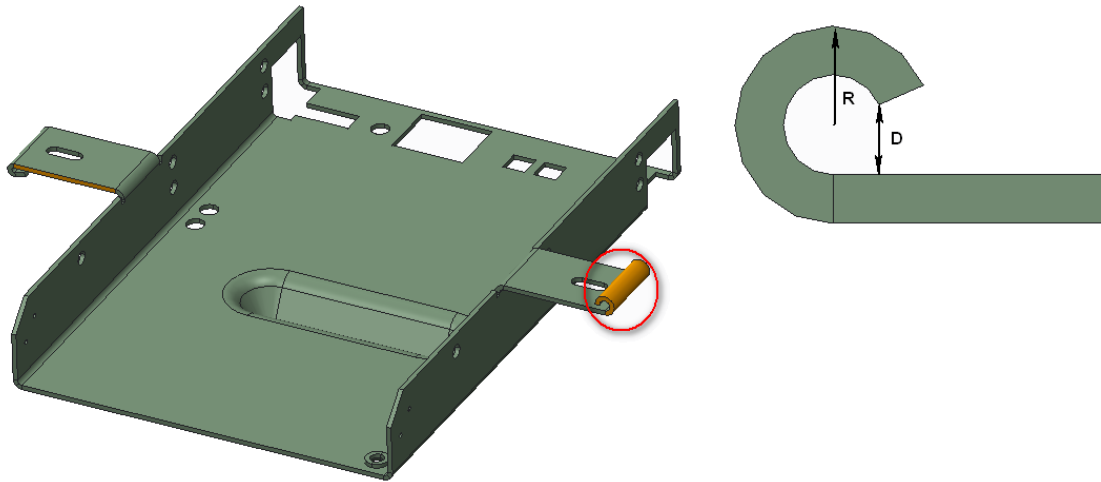
D = Open hem diameter

T = Sheet metal thickness

Rolled Hem

It is recommended that the ratio of the external radius of a rolled hem to material thickness should be greater than or equal to 2.

Also, the ratio of the rolled hem opening to material thickness should be greater than or equal to 1.



R = Rolled hem's external radius

D = Rolled hem's opening

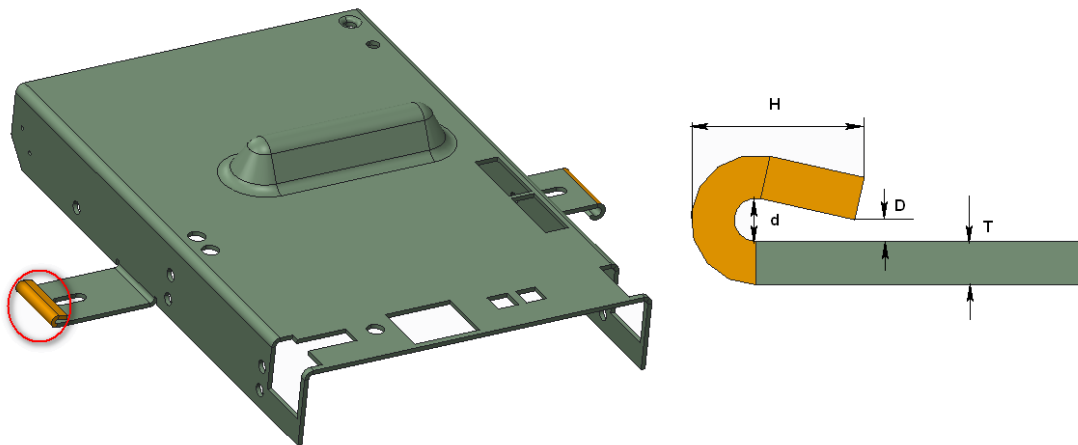
Tear Drop Hem

In tear drop hem the bend angle is greater than 180 degrees. It is recommended to consider the following guidelines while designing a tear drop hem -

The ratio of the radius of a tear drop hem to the sheet metal thickness should be greater than or equal to 0.5

The ratio of the return flange height to the sheet metal thickness should be greater than or equal to 4.0

The ratio of the hem opening (spacing between the hem edge and the part) to the sheet metal thickness should be greater than or equal to 0.25



Where,

d = Diameter of a tear drop hem.

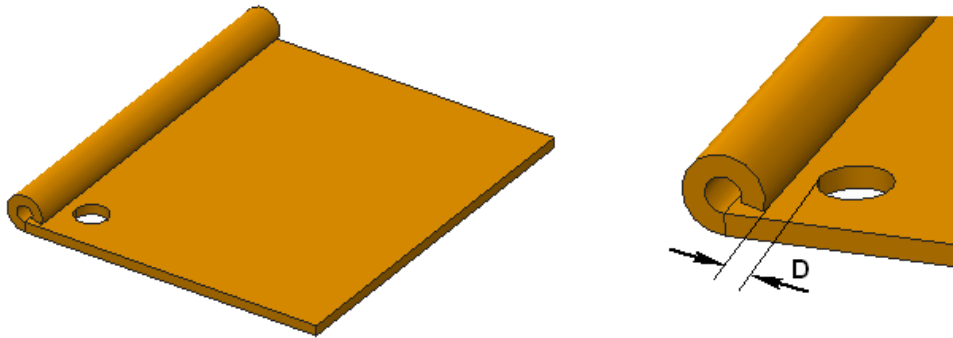
H = Return flange height of the tear drop hem.

D = Hem Opening

T = Sheet Metal Thickness

Rolled Hem to Hole Edge Distance

The ratio of the distance between the rolled hem and the edge of a hole to the sheet thickness should be greater than or equal to 1.



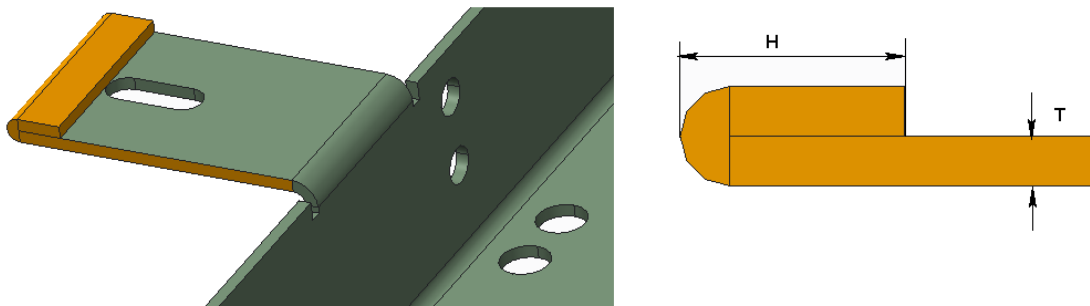
T = Sheet metal thickness

D = Distance

Closed Hem

Hems without any inside radius are called as closed hem. Closed hems are not recommended if the hem geometry is to be painted or if the material used is SST or Aluminum.

In closed hems, return flange length from outside the bend should be equal to or greater than four times the material thickness.



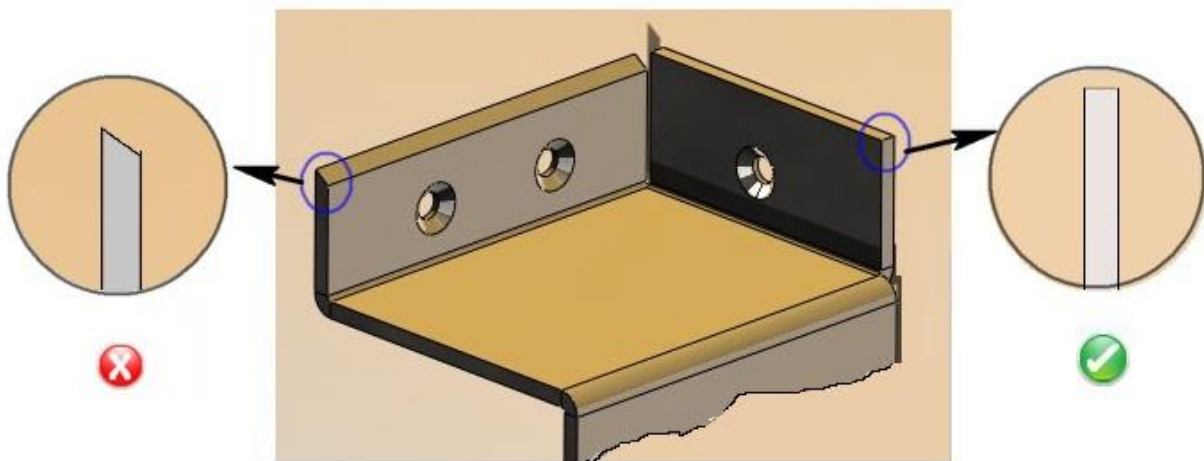
H = Return flange height of the tear drop hem

T = Sheet Metal Thickness

Knife Edge

Knife edge features on sheet metal parts should be avoided for ease of manufacturing. To achieve knife edges on sheet metal parts, additional operations may be required which increases the cost of the product. Knife edge conditions can also lead to fatigue failures in certain conditions.

It is recommended to avoid knife edge conditions as it can either increase the manufacturing cost and complexity or lead to fatigue failures.



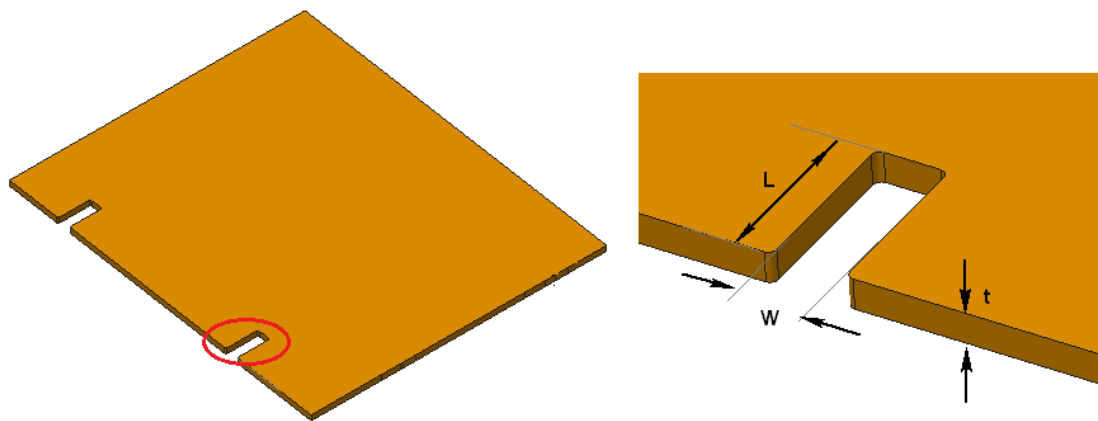
Notch and Tab Parameters

Notch Parameters

Notching is a shearing operation in which a section is removed the outside edge of metal strip or part. Notching is typically manual and low-cost process performed with a small range of standard punches.

Appropriate notch parameters results in use of standard punch sizes

- Notch Width (W) should not be narrower than $1.5 \times t$
- Notch Length (L) should be maximum up to $5 \times t$
- Recommended corner radius for notches should be $0.5 \times t$



t = Sheet Metal Thickness

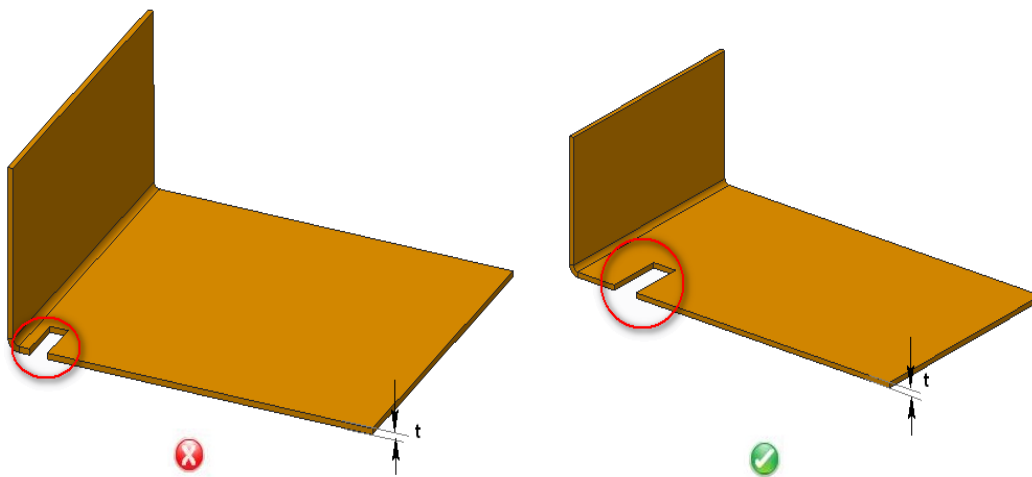
L = Length/ Depth of Notch

W = Width of Notch

Notch to Bend Distance

In case distance between notches to bend is very small then distortion of sheet metal may take place. To avoid such condition notch should be placed at appropriate distance from bend with respect to sheet thickness.

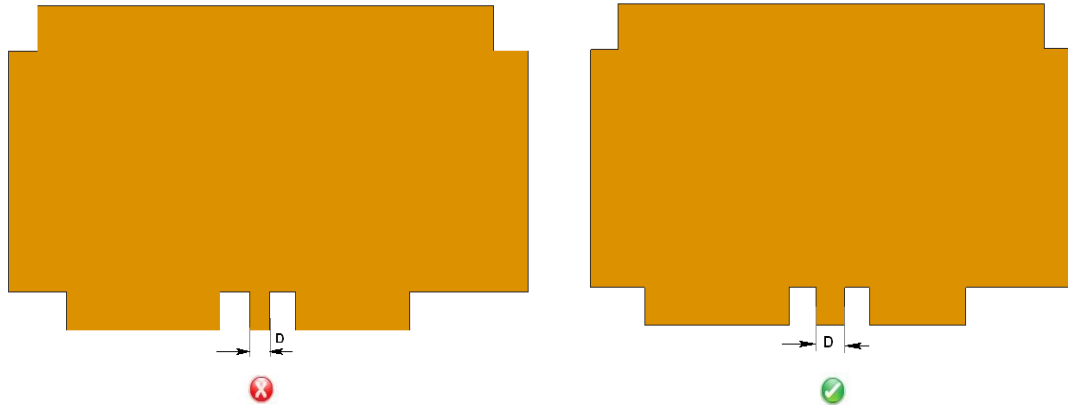
It is recommended that minimum distance from notch to bend should be 3 times the material thickness plus inside bend radius.



t = Sheet Metal Thickness

Minimum Distance between Notches

It is recommended that minimum distance between two Notches should be two times the material thickness.

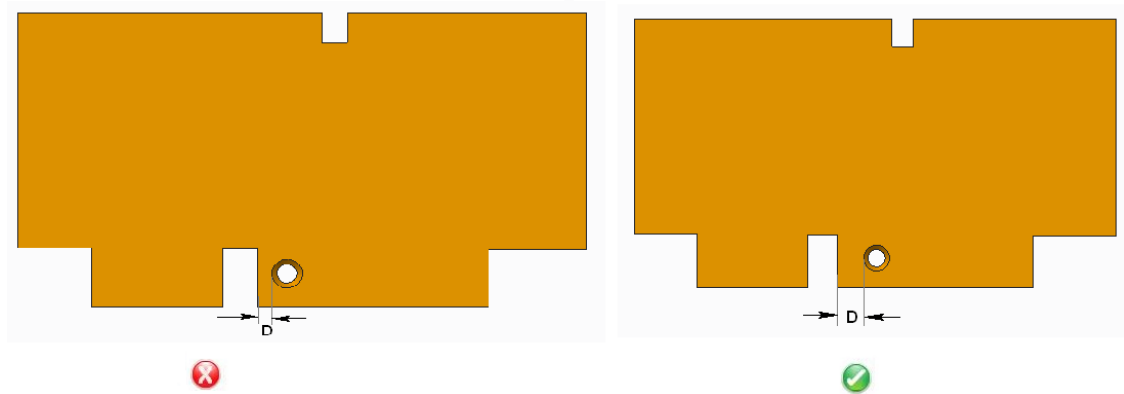


t = Sheet Metal Thickness

D = Distance between Notches

Minimum Distance between Notch to Hole

It is recommended that minimum distance between Notch and Hole should be 1.2 times the material thickness.



t = Sheet Metal Thickness

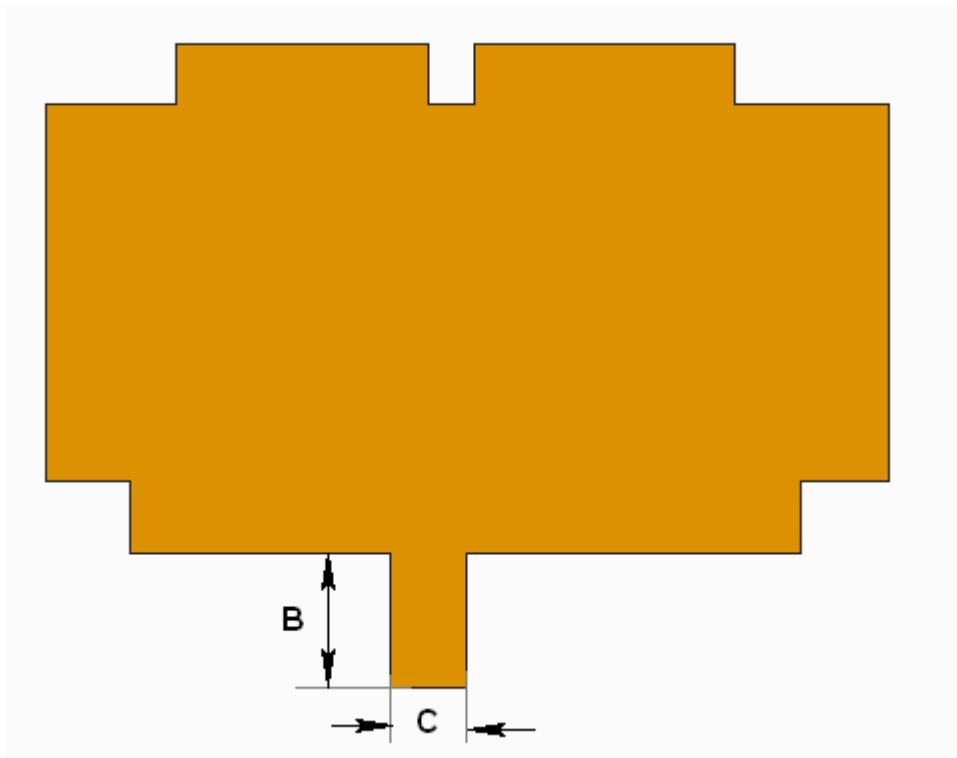
D = Distance between Notch and Hole

Tab Parameters

A protrusion from an edge of the sheet metal part is called a tab.

It is recommended that Tab Width [C] should be greater than two times the sheet thickness.

It is recommended that Tab Depth [B] should be less than or equal to five times the tab width.



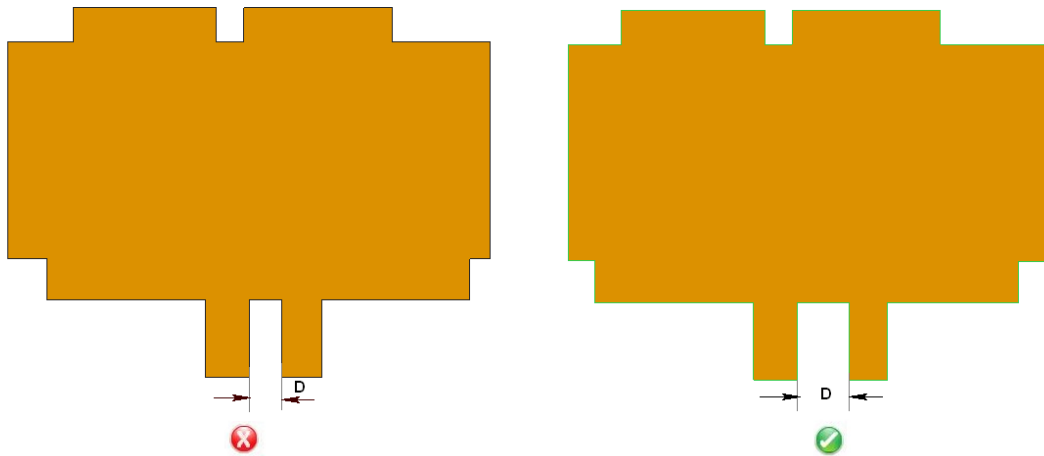
t = Sheet Metal Thickness

B = Tab Depth

C = Tab Width

Minimum Distance between Tabs

It is recommended that minimum distance between tabs should be 1.5 times the sheet metal thickness.



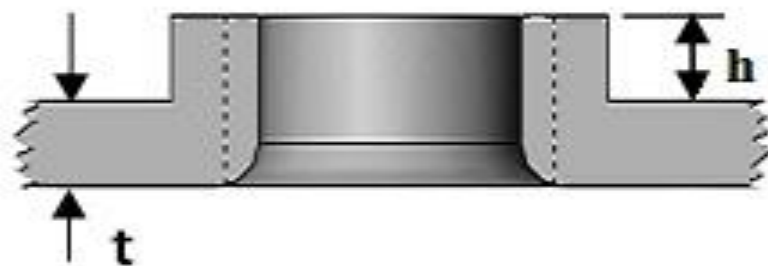
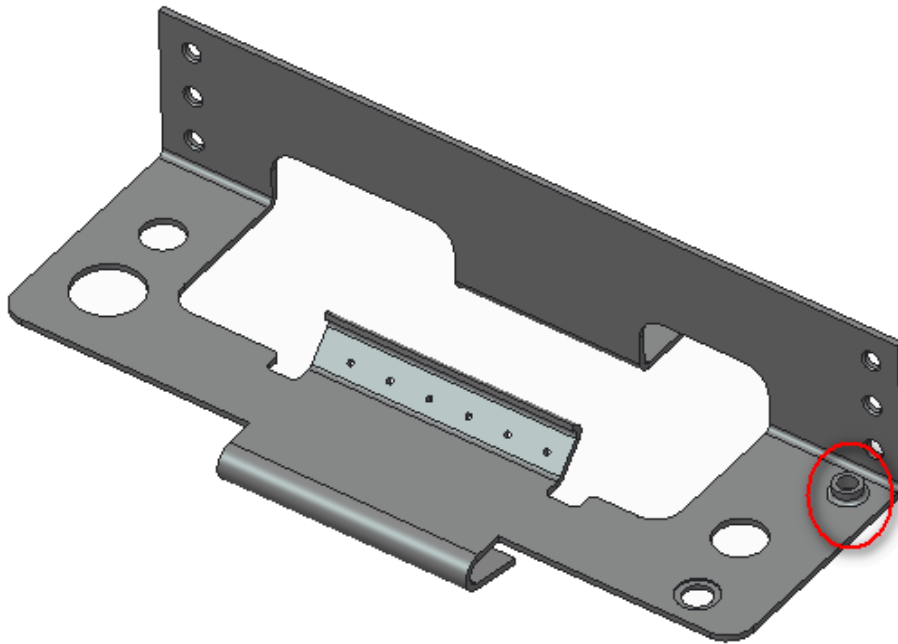
t = Sheet Metal Thickness

D = Tab Distance

Burring Hole Design Guidelines

Burring Hole Height

Sufficient burring holes height is needed to accommodate the stress during loading. It is recommended that the minimum burring height should be at least 2 times the sheet metal thickness.



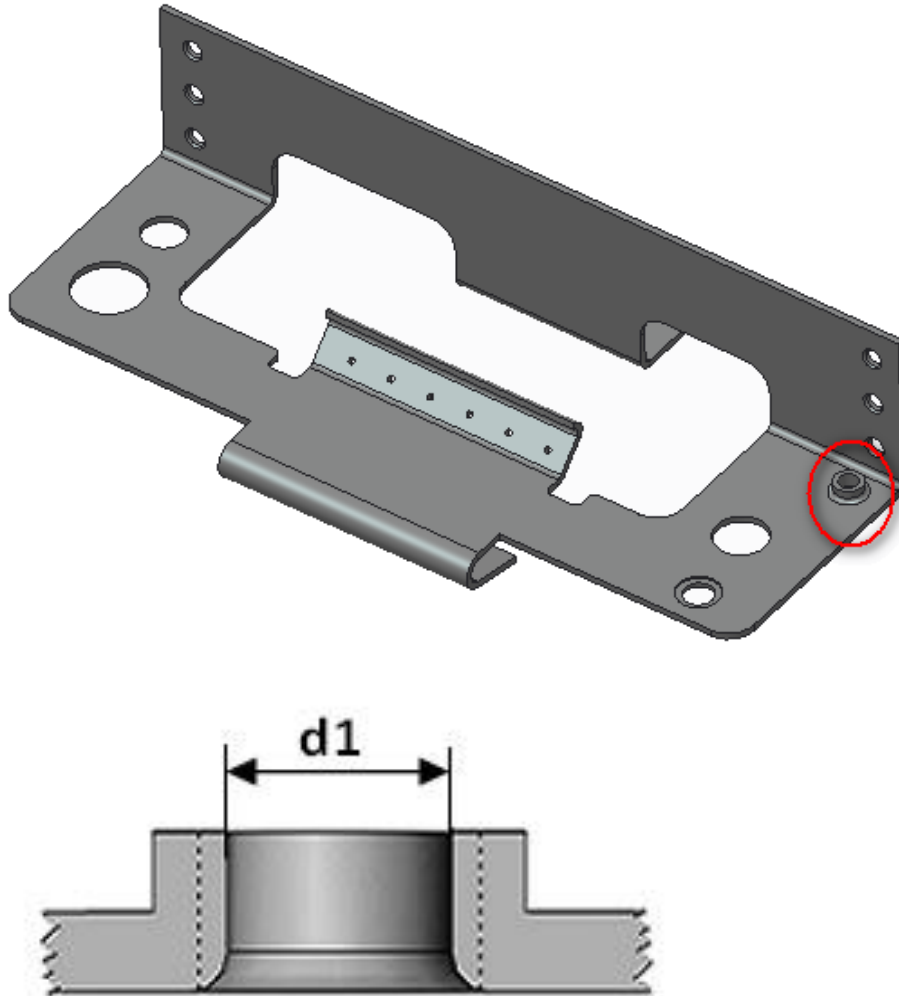
Where,

h = Burring hole height

t = Sheet metal thickness

Burring Hole Inner Diameter

Burring hole inner diameter should be sufficient enough to accommodate the pins, bolts etc.



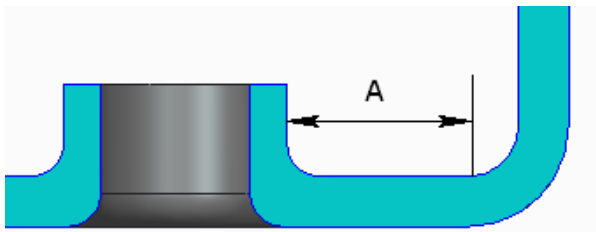
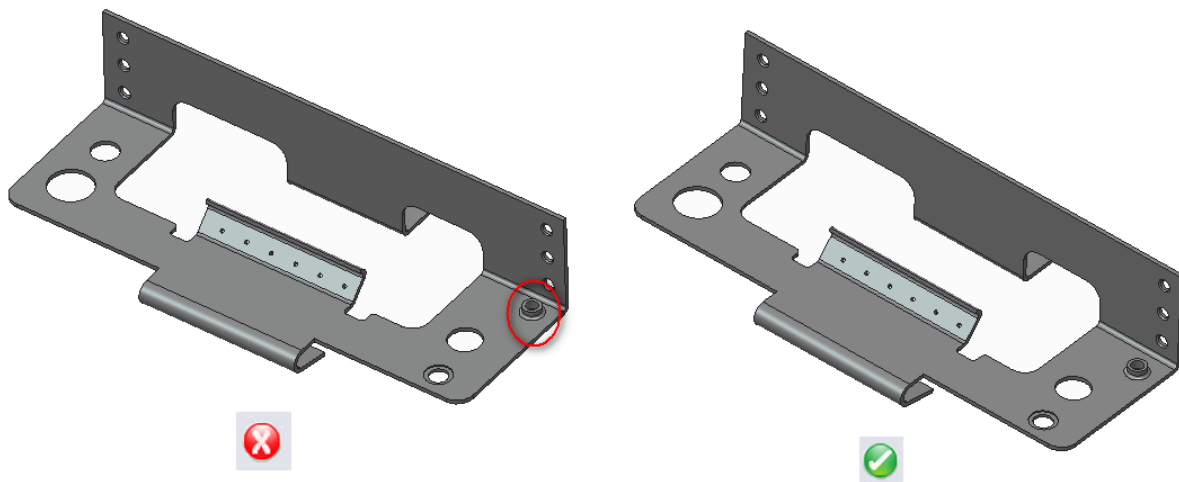
Where,

d1 = Burring hole inner diameter

Burring Hole Distance to Bends

Burring holes that are too close to the bend will distort during the bending process. Distortion will be minimal if the distance between the edge of the hole to the beginning of the inside bend radius is at least four times the material thickness.

It is recommended that the minimum distance between the burring hole edge and the bend should be at least 4 times the sheet thickness.



Where,

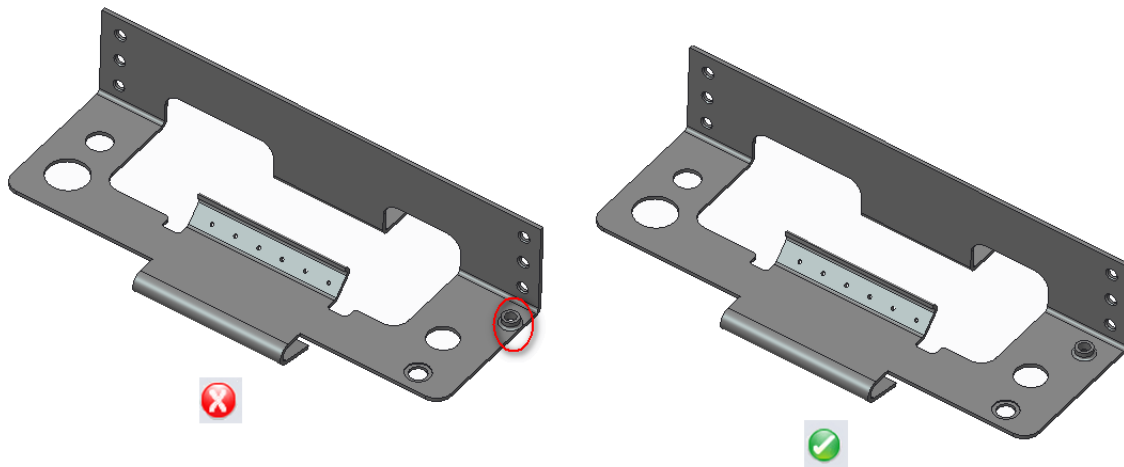
A = Minimum distance from a burring hole edge and bend

t = Sheet metal thickness

Burring Hole Distance to Part Edge

If the distance from a burring hole edge and the part edge falls below a minimum value, then it can lead to distortion or breaking of the sheet. If a burring hole is too close to an edge, the edge can distort, forming a bulge.

It is recommended that the minimum distance from a burring hole edge and the part edge should be at least 4 times the sheet thickness.



Where,

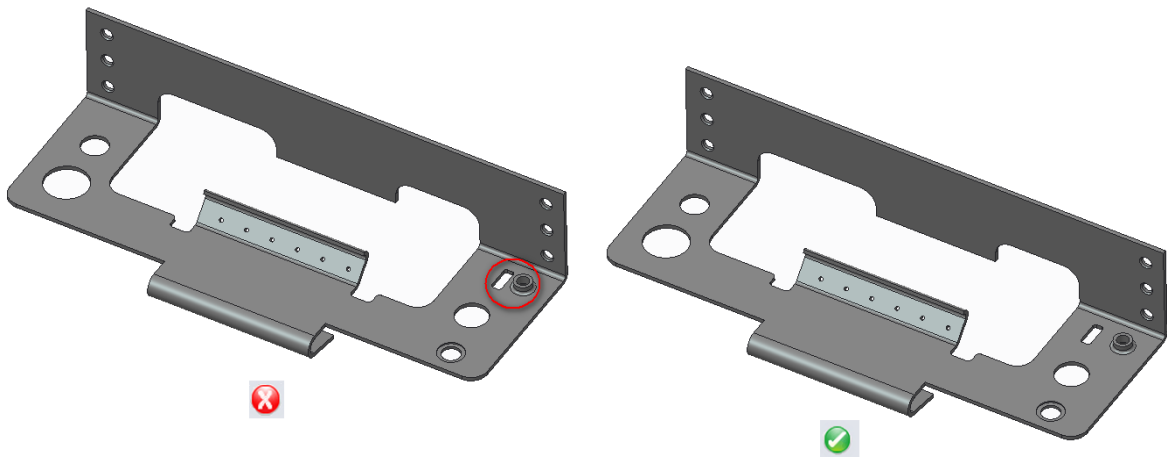
B = Minimum distance from a burring hole edge and the part edge

t = Sheet metal thickness

Burring Hole Distance to Cutout

If the distance from a burring hole edge and the cutout edge falls below a minimum value, there is a chance of distortion or cracking of the sheet in the area under consideration. If a burring hole is too close to an edge, the edge can distort, forming a bulge.

It is recommended that the minimum distance from a burring hole edge and the cutout edge should be at least 4 times the sheet thickness.



Where,

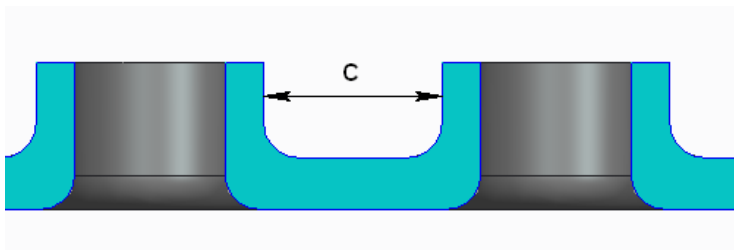
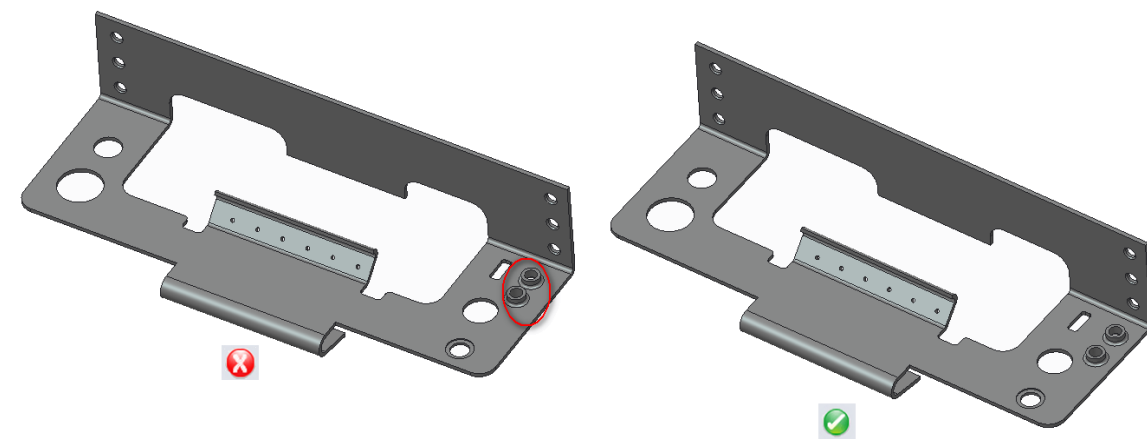
D = Minimum distance from a burring hole edge and cutout edge

t = Sheet metal thickness

Burring Hole Spacing

Sufficient, space should be mentioned between the two adjacent burring holes to avoid distortion or else harder materials may crack during fabrication.

It is recommended that the minimum distance between the burring hole edge and other burring hole edge should be greater than or equal to 4 times the sheet metal thickness.



Where,

C = Burring hole spacing

t = Sheet metal thickness

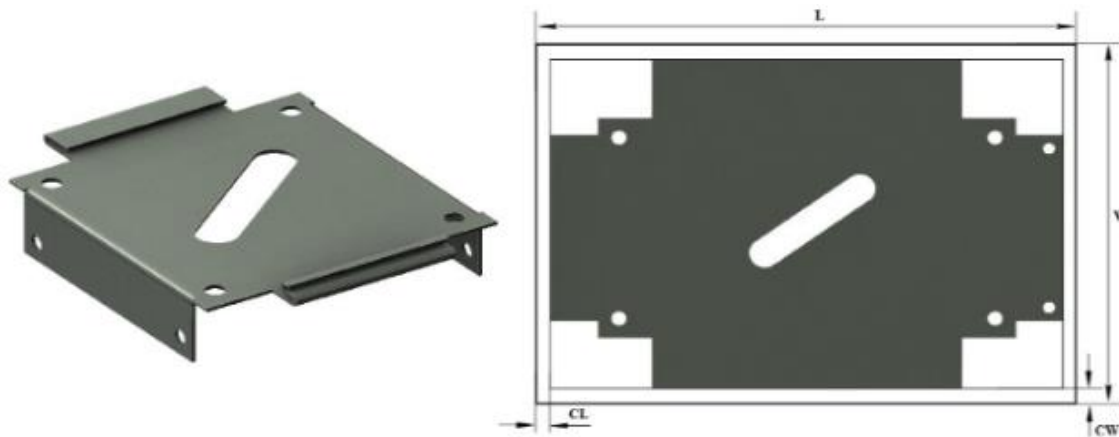
General Design Guidelines

Preferred Sheet Sizes

Length and width of a flattened sheet metal part should be as per the standard sheet sizes that are available in the inventory.

Each organization has its standard or preferred sheet sizes which are in accordance with the machine's capability and setup. At the initial stage of the design, designer should be aware of unfolded sheet size and availability of corresponding appropriate sheet sizes in the inventory. This will help designer to take decision related to inventory management. Indirectly it helps to reduce inventory of non-standard sheet sizes and cost of manufacturing.

Length and width of the flattened sheet metal part should be within the preferred sheet sizes used for manufacturing, depending on the material and sheet thickness.



L = Sheet metal length

W = Sheet metal width

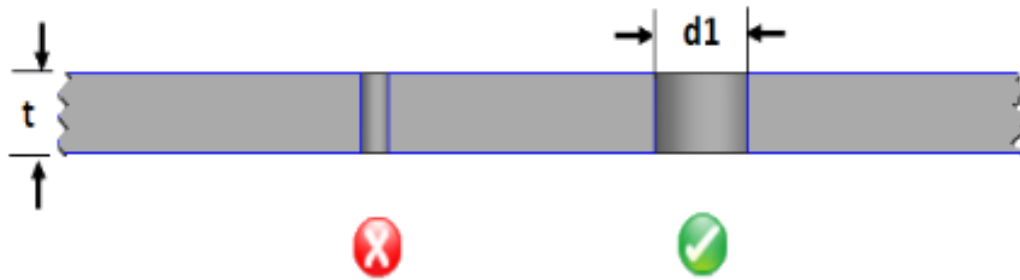
CL = Trimming clearance on length side

CW = Trimming clearance on width side

Note: For nested components, additional considerations will be required.

Minimum Hole Diameter

If the punch diameter becomes too small to bear the shear force required to punch the hole over a small area, it may lead to failure.



Where,

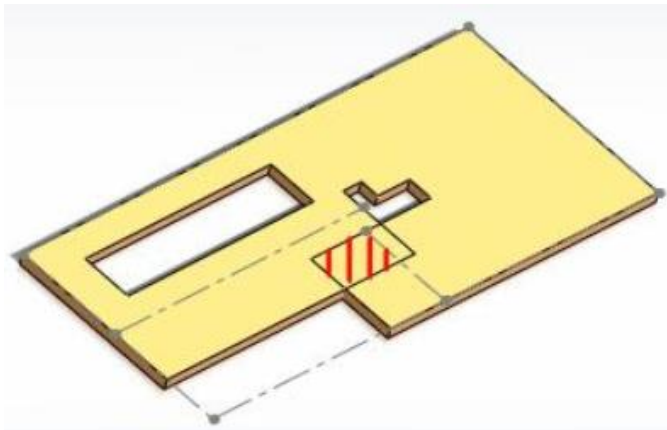
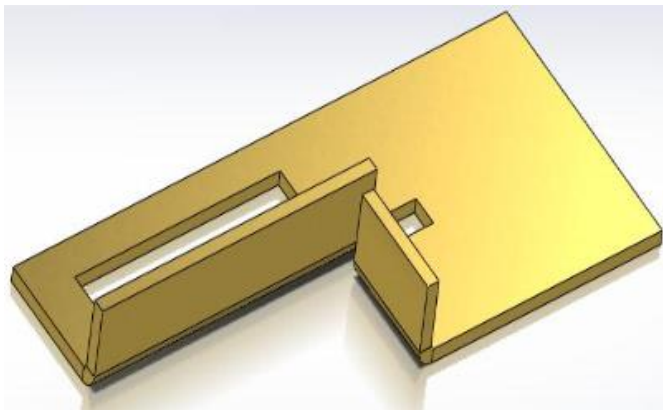
$d1$ = Hole diameter

t = Sheet metal thickness

Interference in Flat Pattern

It is not feasible to manufacture a sheet metal part when its design involves interference during unfolded state of sheet metal.

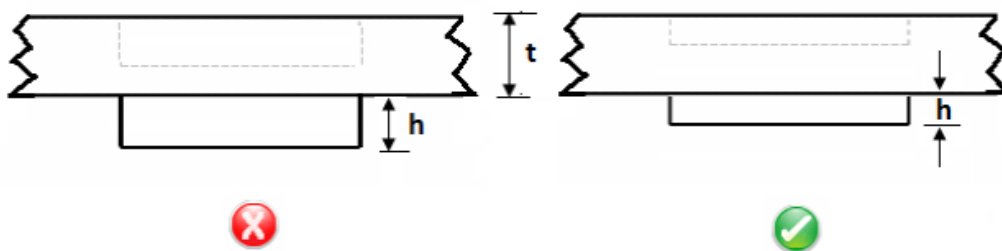
Interference in flat pattern should be avoided for ease of manufacturing as there should not be any interference in sheet metal component design when sheet metal part is unfolded.



Half-Shear Parameters

A Semi-shear or half-shear is a blind hole that is punched on the sheet metal surface. The ratio of half-shear depth to sheet metal thickness should not be too large to avoid deformation and fracture of the metal.

It is recommended that a maximum ratio of half-shear depth to sheet metal thickness should be less than or equal to 0.6 times the sheet metal thickness.

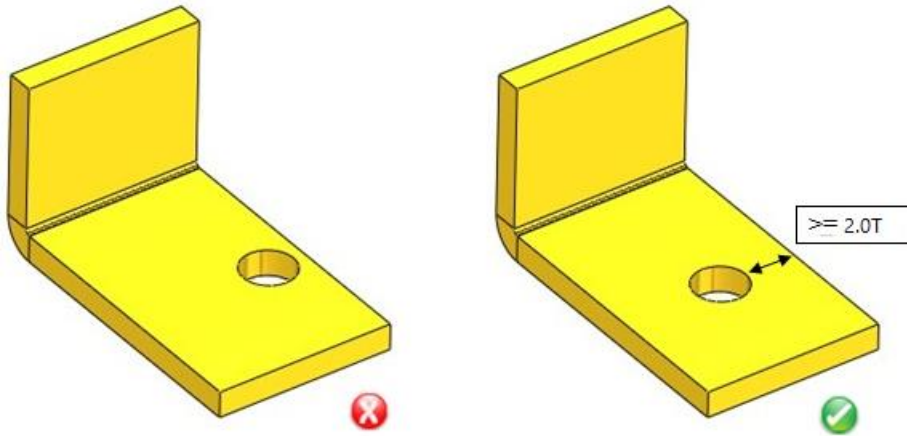


t = Sheet metal thickness

h = Half-shear depth

Hole To Part Edge Distance

To prevent distortion or tearing, hole should be sufficiently away from the part edge. It is recommended that minimum distance from a hole to edge of a part should be at least 2 times sheet thickness

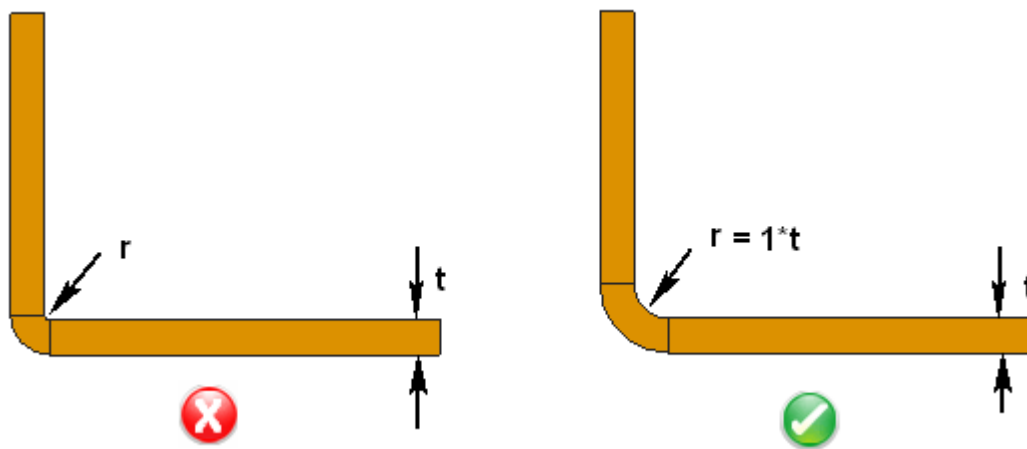


T= Sheet metal thickness

Minimum Bend Radius

Minimum bend radius is a function of the ductility and thickness of the material being worked. The minimum bend radii requirements can vary depending on applications and material. For aerospace and space applications, the values may be higher. When the radius is less than recommended, this can cause material flow problems in soft material and fracturing in hard material. Localized necking or fracture may also occur in such cases.

It is recommended that minimum inner bend radius should be at least $1 \times$ material thickness.



r = Inside Bend radius

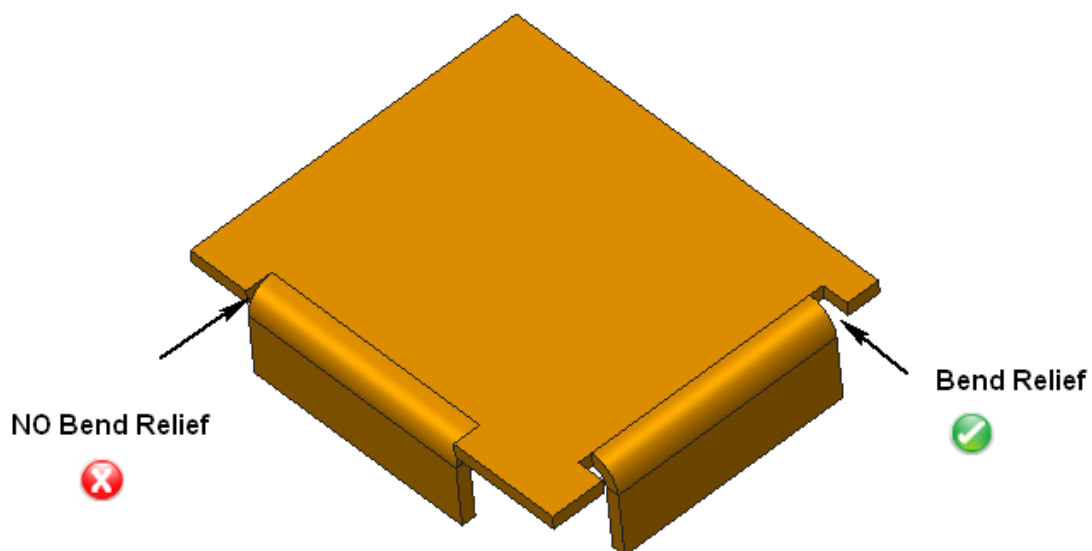
t = Sheet metal thickness

Bend Relief

Bend relief is the notch that needs to be created for sheet metal bending. Bend relief helps control the sheet metal material behavior and prevents unwanted deformation.

The flange which does not have relief will result in a greater amount of distortion or tearing of the adjacent material.

It is recommended that the depth of bend relief should be greater than or equal to the inside bend radius of the bend and also width of the bend relief should be equal to or greater than the sheet metal thickness.

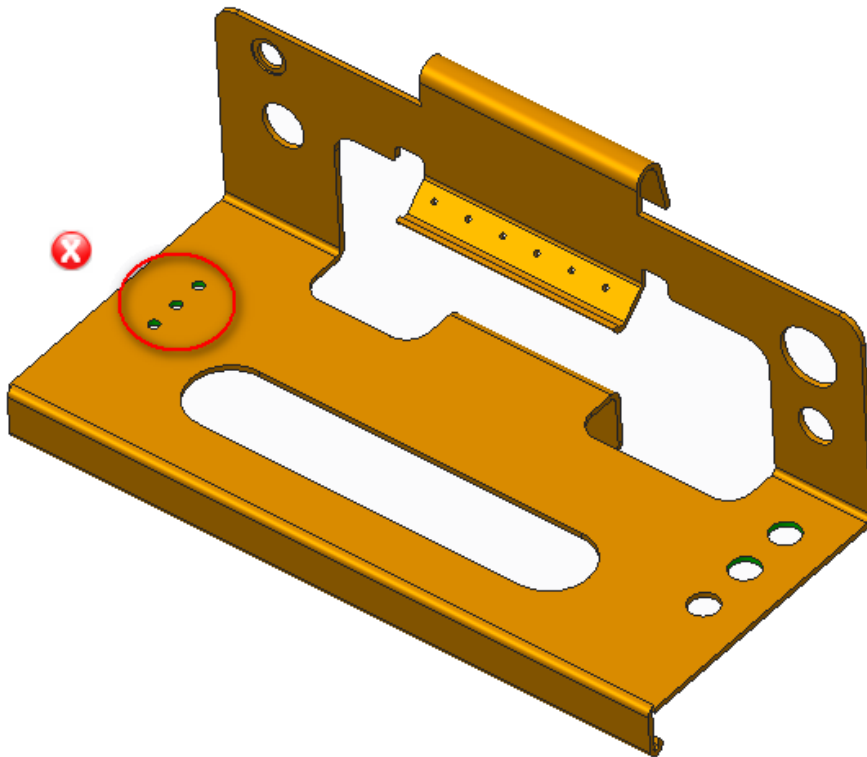


Minimum Hole Diameter

The diameter of the hole in sheet metal part should not be very small, small holes are created by piercing operation and for manufacture small holes, small sizes punches are required.

Small hole size in sheet metal requires smaller size punching tool which may leads to break during the operation.

It is recommended that the diameter of the hole should be equal or more than the thickness of the sheet metal.



A = Minimum Hole Diameter

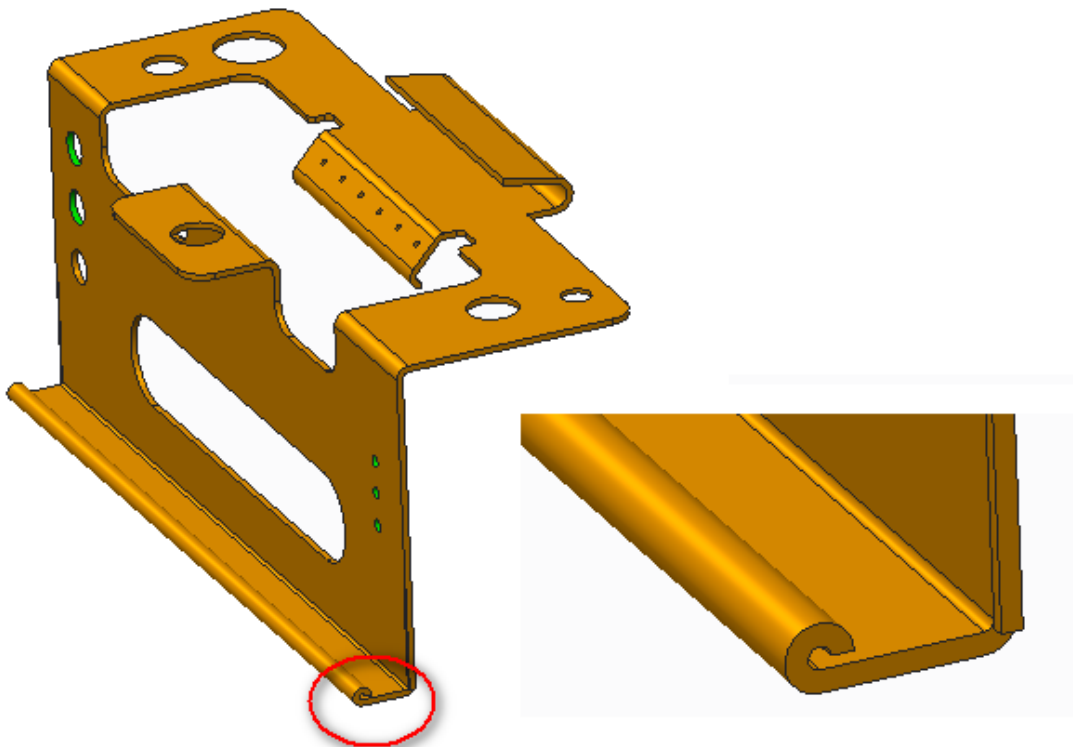
t = Sheet metal thickness

Curls Feature

Curling sheet metal is the process of adding a hollow, circular roll to the edge of the sheet. The curled edge provides strength to the edge and makes it safe for handling. Curls are most often used to remove a sharp untreated edge and make it safe for handling.

It is recommended that:

- The outside radius of a curl should not be smaller than 2 times the material thickness.
- A size of the hole should be at least the radius of the curl plus material thickness from the curl feature.
- A bend should be at least the radius of the curl plus 6 times the material thickness from the curl feature.



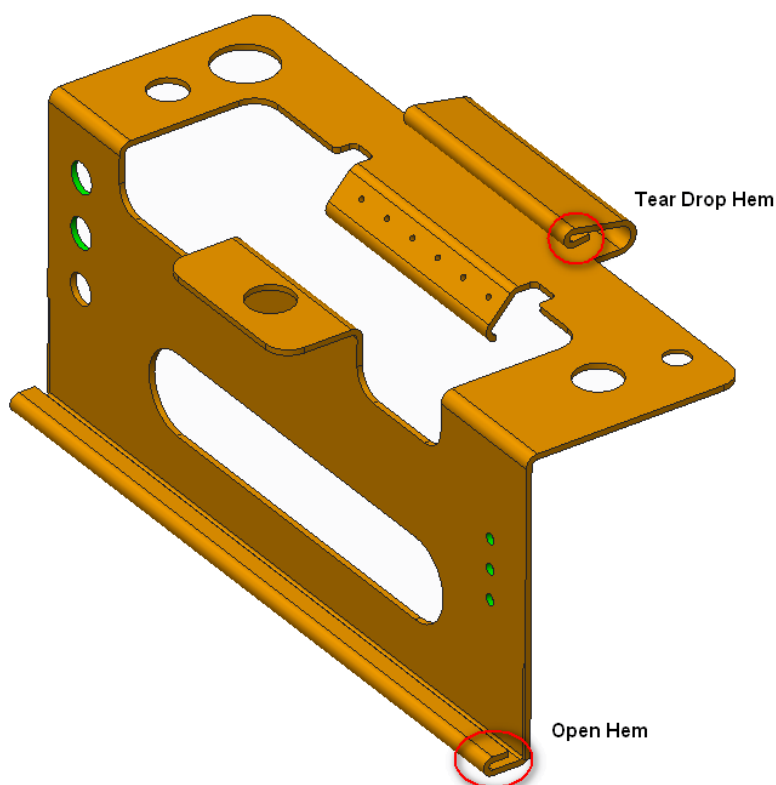
Hem Feature

Hemming is nothing but to fold the metal back on itself. In Sheet Metal Hems are used to create folds in sheet metal in order to stiffen edges and create an edge safe to touch.

Hems are most often used to remove a sharp untreated edge and make it safe for handling. Hems are commonly used to hide imperfections and provide a generally safer edge to handle. a combination of two hems can create strong, tight joints with little or minimal fastening. Hems can even be used to strategically double the thickness of metal in areas of a part which may require extra support.

It is recommended that:

- For Tear drop hems, the inside diameter should be equal to the material thickness.
- For Open hem the bend will lose its roundness when the inside diameter is greater than the sheet metal thickness.
- For bends, the minimum distance between the inside edge of the bend and the outside of the hem should be 5 times material thickness plus bend radius plus hem radius.



Notch Feature

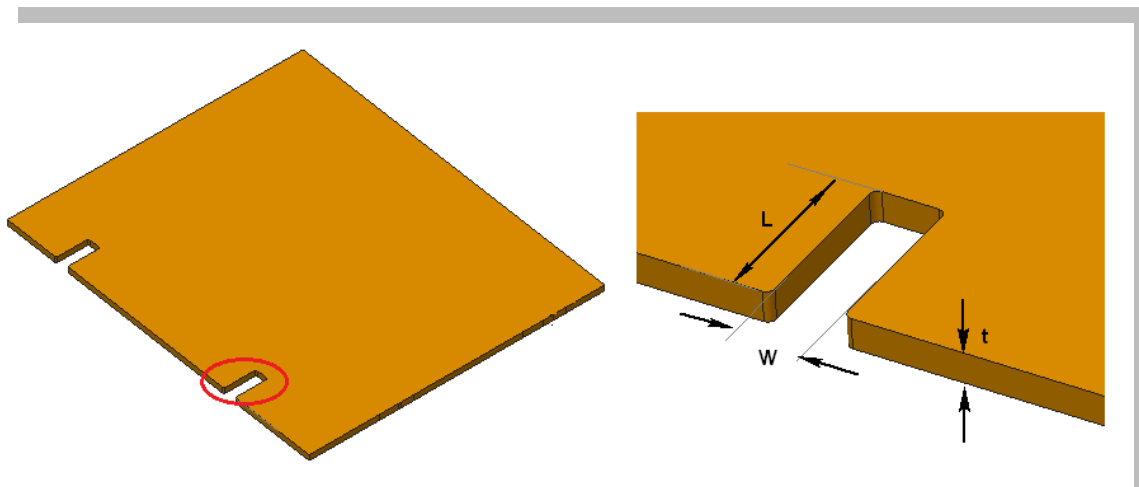
Notching is a shearing operation that removes a section from the outer edge of the metal strip or part. In case, distance between the notches to bend is very small then distortion of sheet metal may take place.

To avoid such condition notch should be placed at appropriate distance from bend with respect to sheet thickness.

Notching is a low-cost process, particularly for its low tooling costs with a small range of standard punches.

Recommendations for Notch Feature:

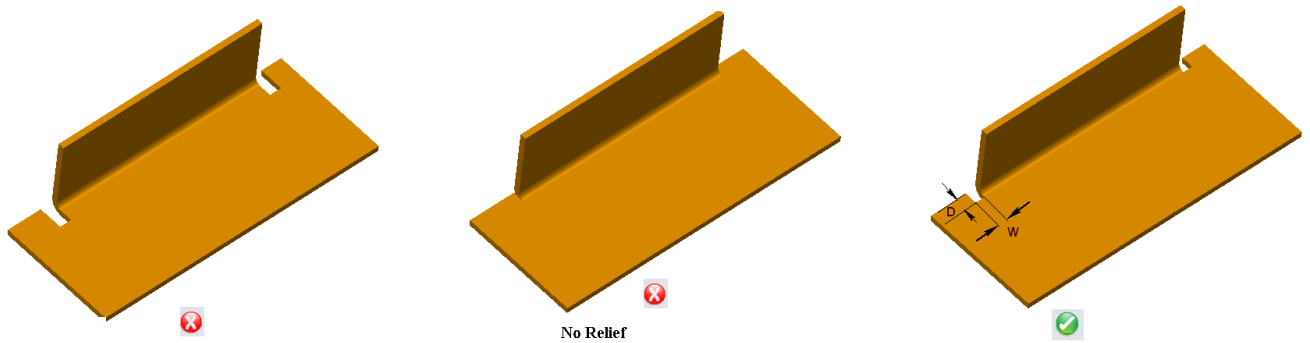
- Notch Width should not be narrower than $1.5 * t$
- Length of notches can be up to $5 * t$
- Recommended corner radius for notches should be $0.5 * t$



Bend Relief Parameters

When a bend is made close to an edge, the sheet metal may tear unless bend relief is given. Bend relief helps in controlling the sheet metal material behavior and prevents unwanted deformation when a bend does not extend the entire length of a part, or a form is close to other sheet metal features.

It is recommended to use rectangular relief. The minimum depth (D) of a bend relief should be equal to the material thickness plus the radius of a bend. The minimum width (W) of a bend relief should be equal to 1.5 times the material thickness



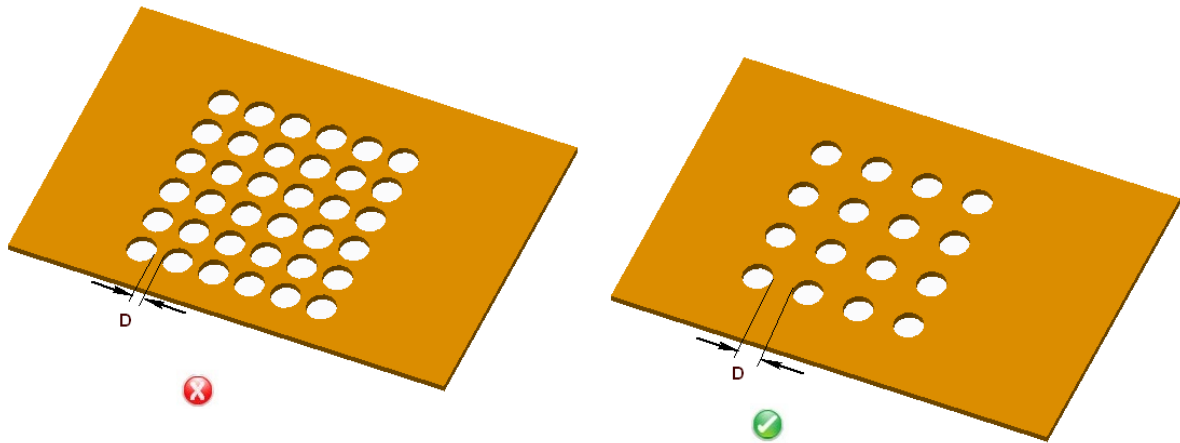
D = Depth of Bend Relief

W = Width of Bend Relief

Spacing Between Holes in Perforated Metal

Perforated metal is nothing but a pattern of holes on a sheet metal surface. It has endless applications in enclosures, partitions, sign panels, guards and screens as it offer several benefits like larger open areas, higher strength-to-weight ratio, good aesthetic appeal and different hole sizes.

It is recommended that minimum distance between the holes in a perforated metal should be equal to 1.2 times the material thickness.



D = Distance between holes

Hole to Bend Distance in Perforated Metal

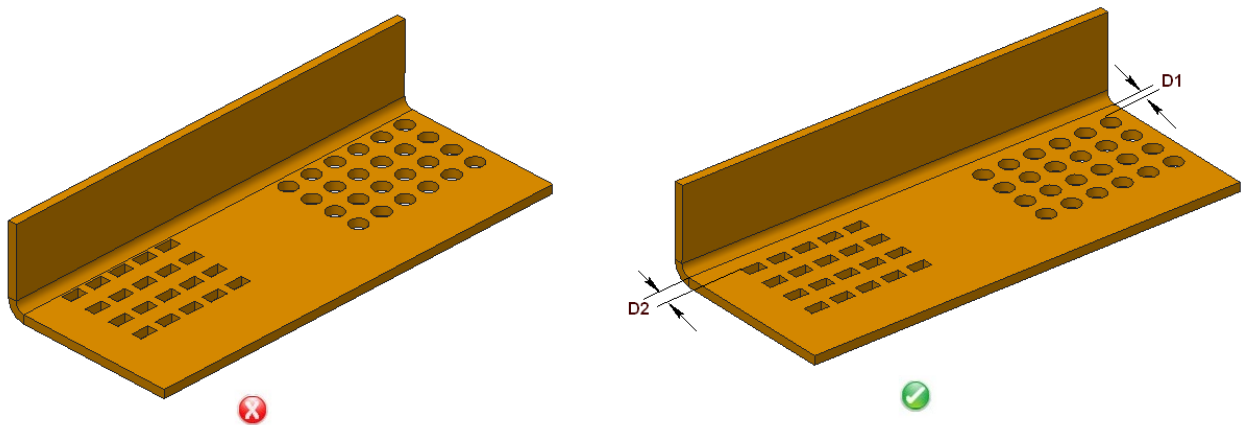
When a bend is made too close to the hole, the hole may become deformed. The minimum distance recommended for circular holes and rectangular holes is as following

For Circular Holes:

It is recommended that the minimum distance from the edge of a circular hole to a bend should be 3 times the material thickness

For Rectangular Holes:

It is recommended that the minimum distance from the edge of a rectangular hole to a bend should be 3.5 times the material thickness.



D1 = Distance Between Hole and Bend

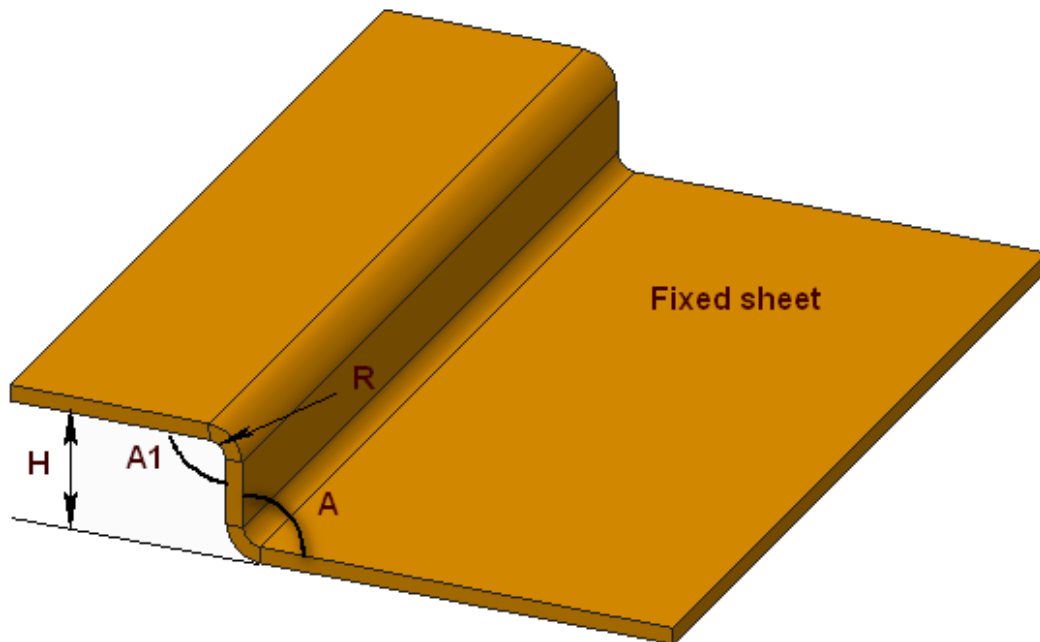
D2 = Distance Between Rectangular Holes and Bend

Offset Bend Parameters

An offset bend, also called as joggle is made up of two very short bends formed simultaneously. It is used to joggle other sheet metal surface or overlap of a sheet.

It is recommended that:

- The bends angles (A1) should be obtuse angles or right angles.
- The maximum height (H) of an offset bend should be five times of the material thickness.



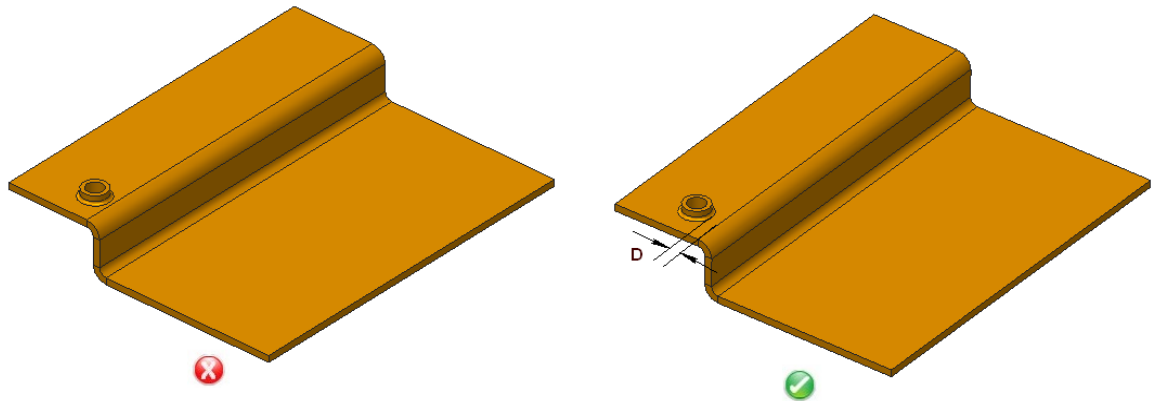
H = Offset Height

R = Bend Radius

A1 = Offset Bend Angle

Distance between Offset Bends and Extruded hole

The minimum recommended distance from the bend's inside surface to the major diameter of extrusion should be equal to 2.5 times the material thickness.



D1 = Distance between offset Bend to extruded Hole