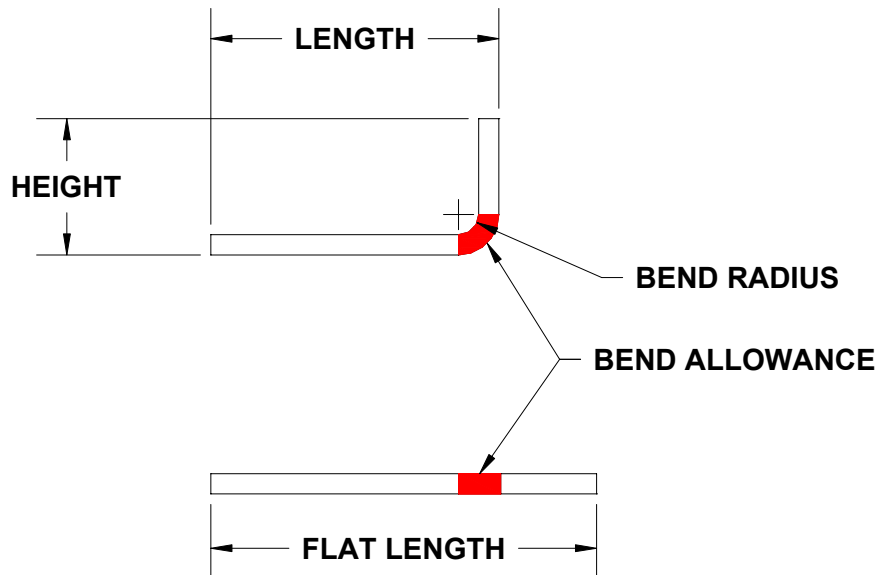


### 1. Introduction

Bend allowance is a term which describes how much material is needed between two panels to accommodate a given bend. Bend allowance, while being oftentimes tricky to determine for all cases, is fairly easy to predict and calculate for many standard circumstances. Determining bend allowance is commonly referred to as “Bend Development” or simply “Development”.



**If in doubt, make a test piece.**

Often bend allowances are calculated for a sheet metal part and used to make costly tooling or production parts that require a lot of labor to produce. A scrap tool or production run can be very costly, much more so than a test piece. So if you are ever not sure of your developed flat length, make a test piece (laser, turret or sheared piece) to confirm your development.

One of the easiest ways to make a test piece is to shear a piece to an exact length, and then form it using the exact process that will be used to create the part. After the part is formed, the part is measured and compared to the expected lengths and the bend allowance is adjusted as needed. Often times, when hard tools are produced, laser cut blanks are used to validate the forming tools and part development before the cutting tools are completed.

**No rule will apply to every case.**

While most bend developments can be predicted with ease and will develop correctly, there is no perfectly scientific method for predicting bend allowance due to the many factors like tooling conditions, actual vs. planned thickness, forming method and the given part tolerance. Many companies will develop their bend allowances based on standard formulas, standard forming practices and historical trial and error.

## Bend Allowance Overview

**Know the difference between a lazy bend and a bad development.**

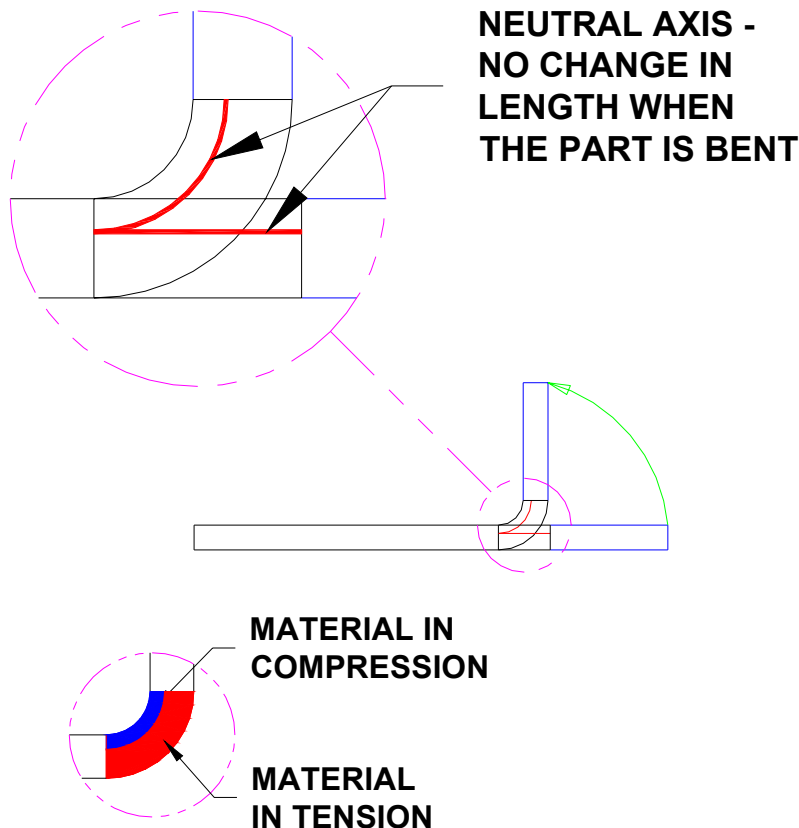
Often times a correctly developed part will be wrong due to poor forming, especially during the first run of the parts. Generally, it takes some time to validate tooling and make sure everything is tuned properly. It is not uncommon for a part with the correct development will have features that “act long” because of “lazy” forming. The most common cause of lazy forming is not bottoming the forming tools adequately.

## 2. General Principles

**The Neutral Axis does not change.**

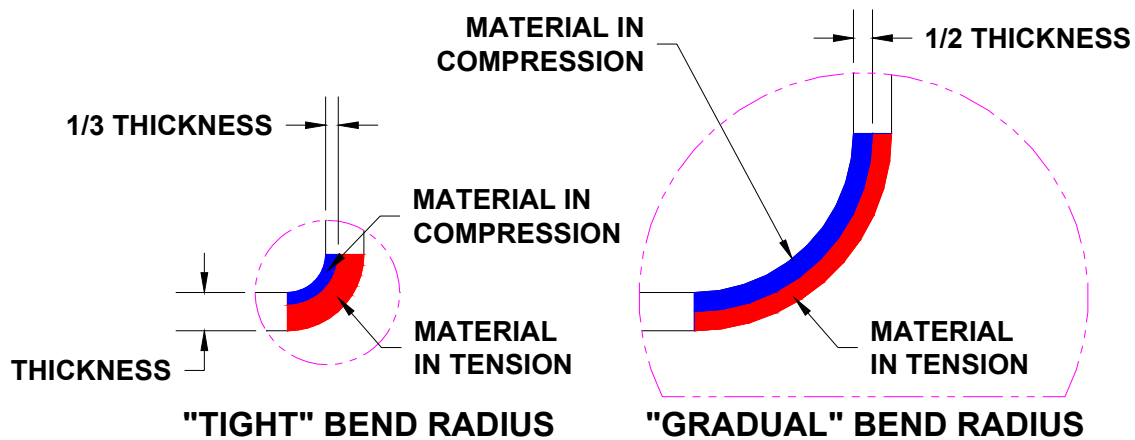
When developing a flat blank length, there is a length of the part that does not change. This length is called the neutral axis. Material on the inside of the neutral axis will compress, while material on the outside will stretch. Based on the material thickness, form radius and forming methods, the ratio of compression to tension in the part will change.

A part that is bent over a very sharp radius, when compared to the thickness, will stretch more on the outside, which means that the neutral axis will lie closer to the inside of the bend. A part that is gradually bent will have less outside stretch, which means that the neutral axis will lie closer to the center of the part.



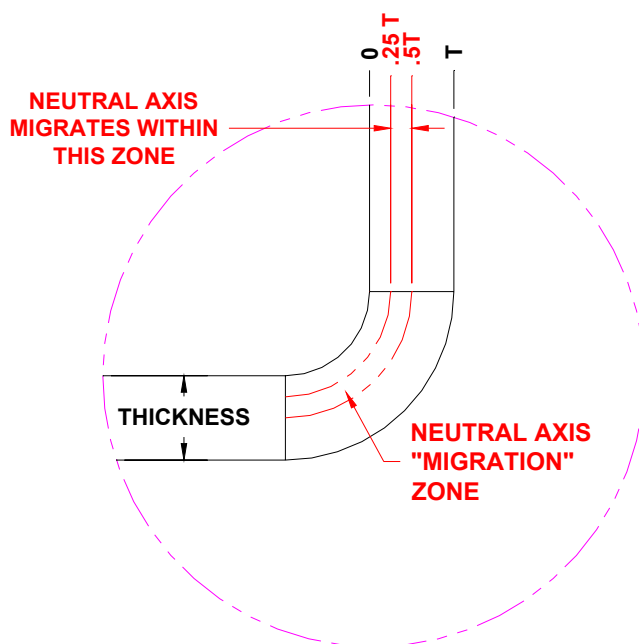
## Bend Allowance Overview

### Compression/Tension Ratio Depends Mostly On Geometry.



### K-factor – Effectively 50%T Max / .25%T Min

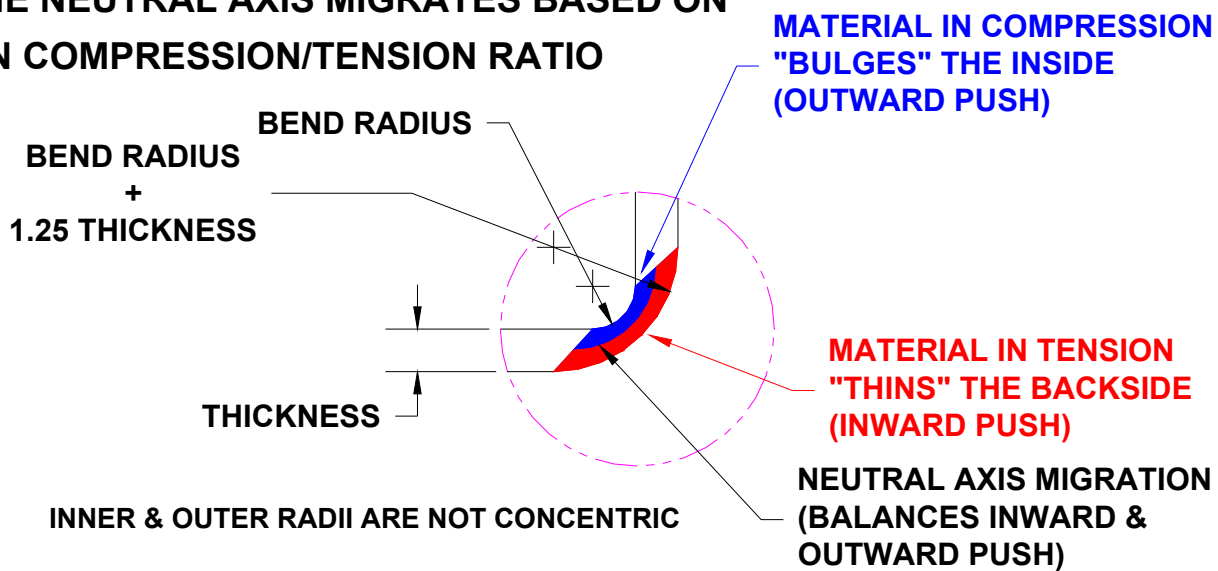
Where the neutral axis is situated in a bend is commonly called the “K-Factor” as it is signified as “K” in the development formulas. Since the inside compression can not exceed the outside tension, the k-factor can never exceed .50 in practical use. This means that the neutral axis cannot migrate past the midpoint of the material (i.e. towards the outside). A reasonable assumption is that the k-factor cannot be less than .25.



## Bend Allowance Overview

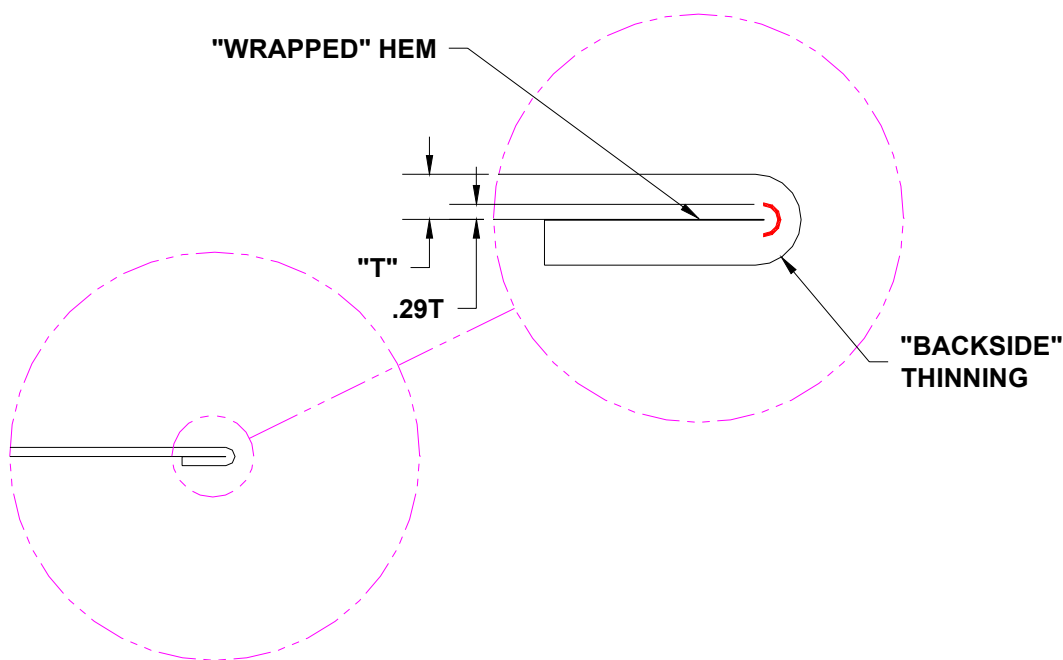
The neutral axis migrates based on the compression to tension relationship of the given bend.

### THE NEUTRAL AXIS MIGRATES BASED ON ON COMPRESSION/TENSION RATIO

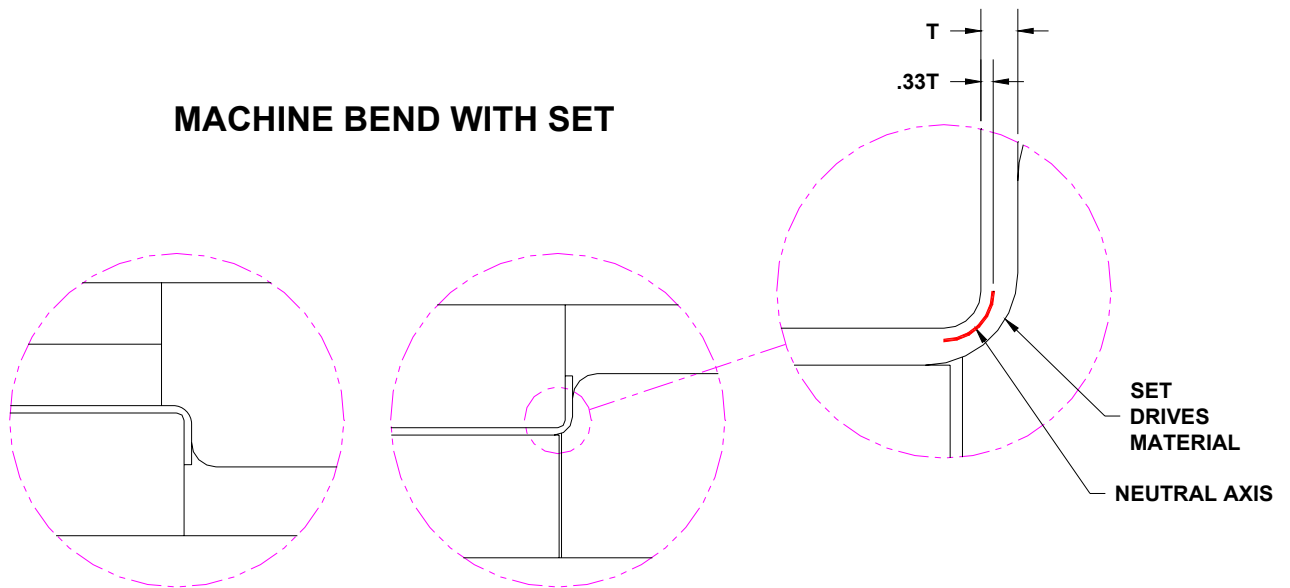


## 3. Different Bend Types & K-Factors

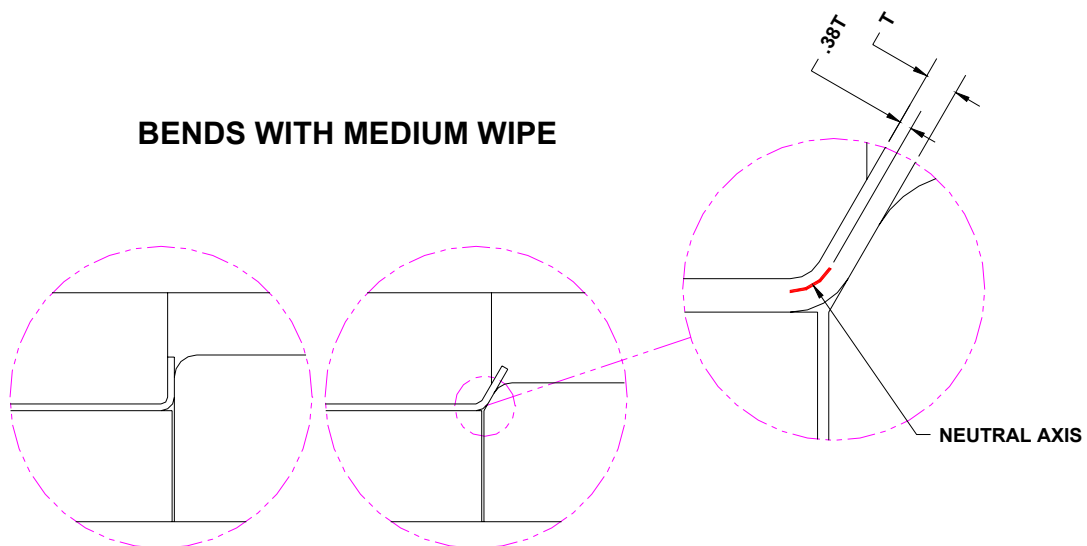
### Wrapped Hem (.29 k factor)



## Machine Bend with Set (.33 k factor)

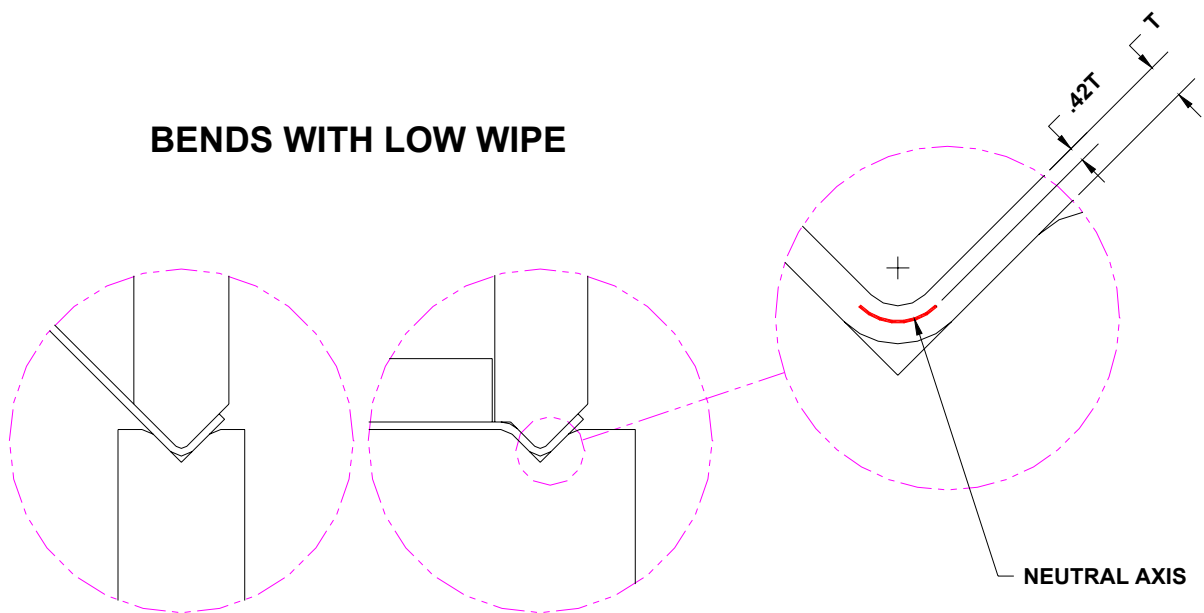


## Machine Bend With No Set (.38 k-factor)

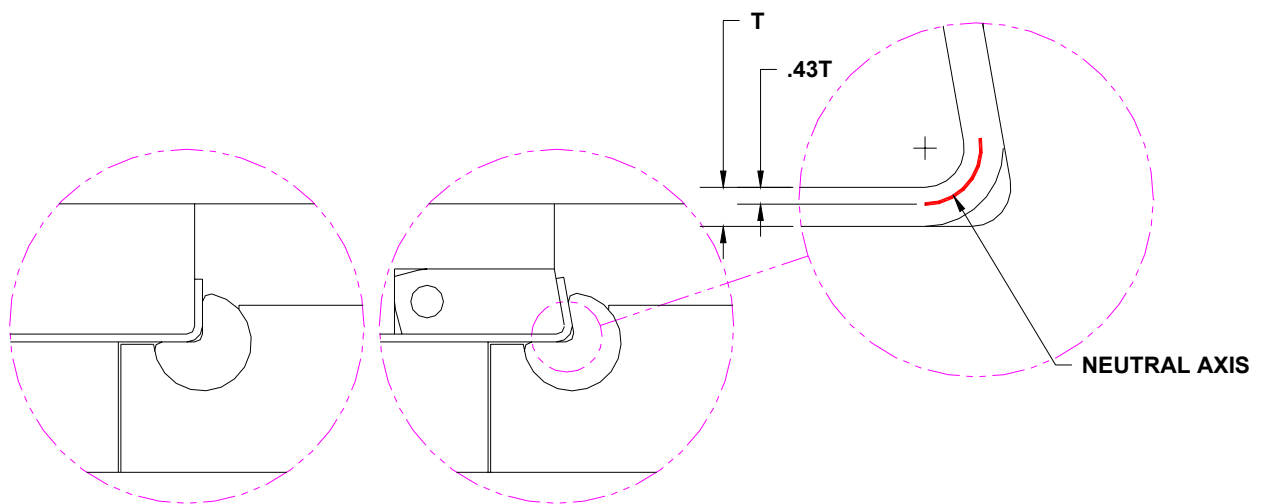


## Bend Allowance Overview

### V-Bend Or Brake Tool (.42 k-factor)



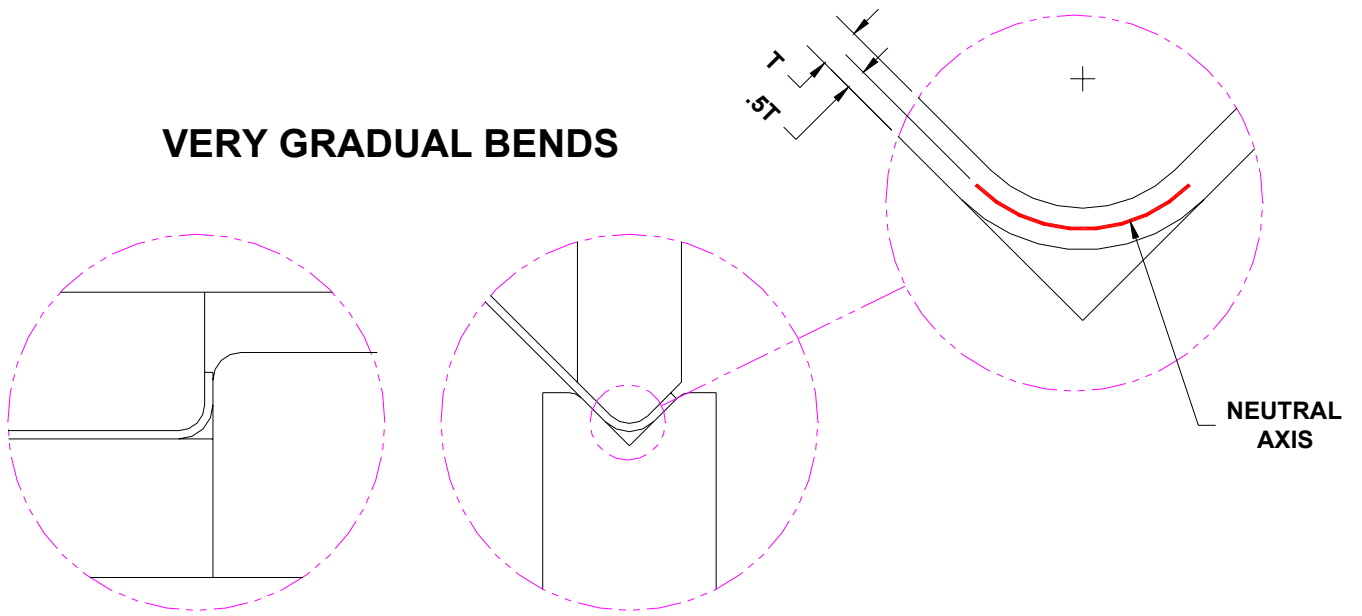
### Rotary Benders (.43 k-factor)



### Gradual Bends / Large Radii (.50 k-factor)

## Bend Allowance Overview

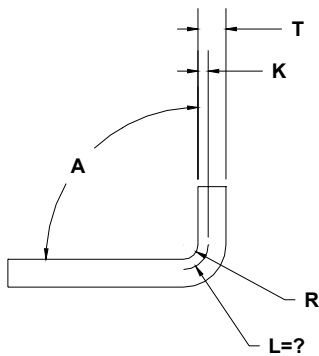
### VERY GRADUAL BENDS



### 4. Related Formulas

#### Radian Formula

When a developed length is calculated in radians, the equation is extremely simplified because the radian is the actual arc length, so no additional “translation” into angles is needed as in the “standard” formula below. In fact, the “standard” formula is the radian formula plus a “built in” angle conversion from radian measure to (base 360) degrees, shown in the “Common Formula”.



**BEND ALLOWANCE FORMULA  
(FOR ANGLE IN RADIANS)**

$$L = A (R + KT)$$

A = ANGLE (RADIANS)

R = BEND RADIUS

K = NEUTRAL AXIS OFFSET (K-FACTOR)

T = THICKNESS OF MATERIAL

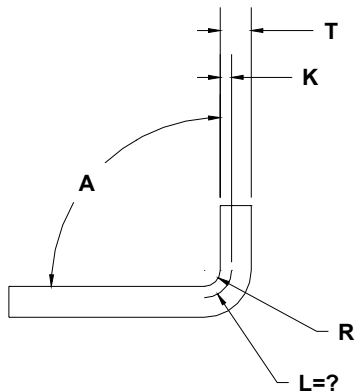
L = LENGTH OF BEND ALLOWANCE

#### Common Formula

Since it is more common to develop a part based on degrees instead of radians, the bend allowance formula commonly incorporates the degrees to radians conversion.

Recalling that 360 Degrees =  $2\pi$  Radians, then 1 Degree =  $2\pi$  Radians / 360

To convert the radian formula to work with degrees, we make the substitution  $2\pi/360$



$$L = 2 \pi A (R + KT) / 360$$

OR

$$L = A (R + KT) / 57.3$$

A = ANGLE (DEGREES)

R = BEND RADIUS

K = NEUTRAL AXIS OFFSET (K-FACTOR)

T = THICKNESS OF MATERIAL

L = LENGTH OF BEND ALLOWANCE

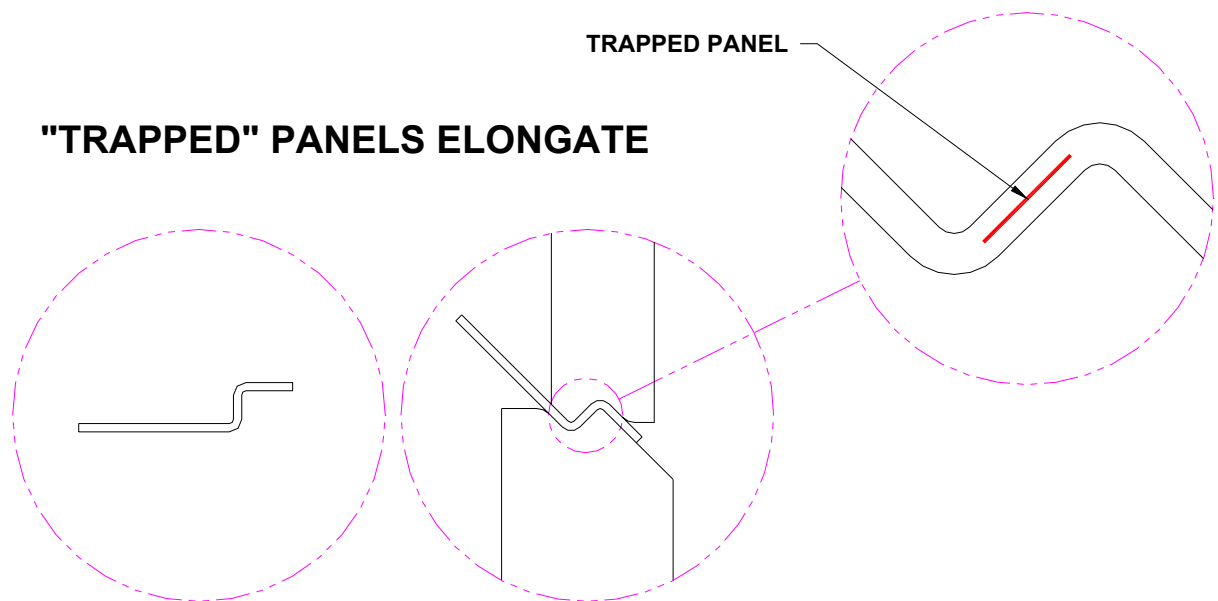
Pi = 3.14



## 5. Special Cases

### Single Hit Z-Bend

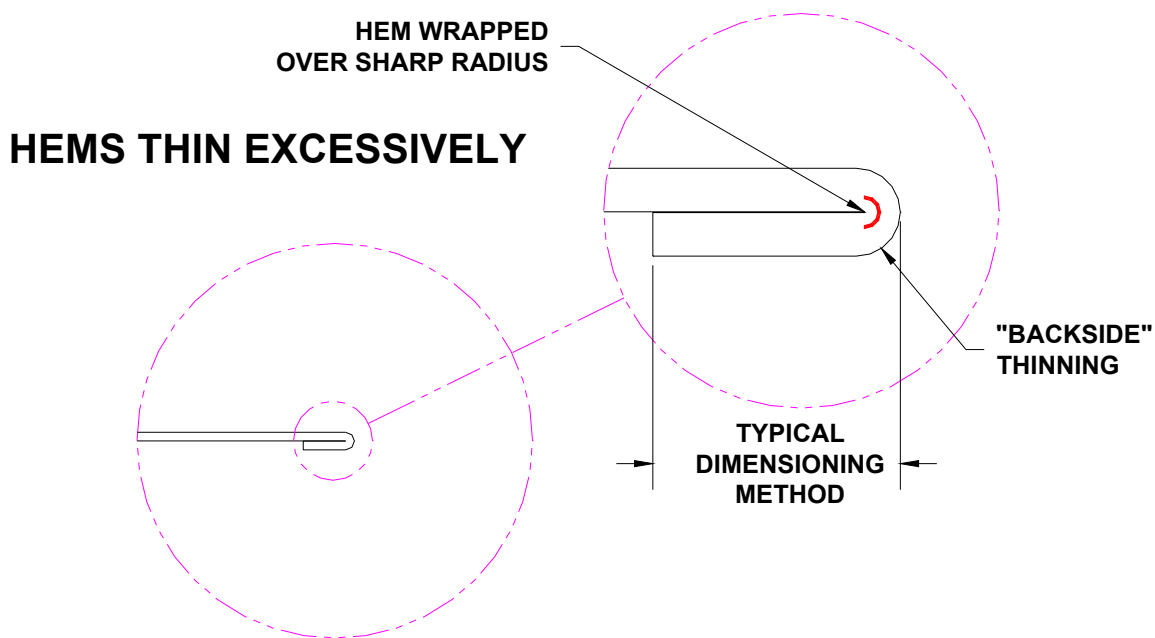
When a z-bend is hit in one hit, the middle panel will stretch more than expected. This is because the middle panel is trapped between two v-forms. A typical example might be on a .312 deep zee bend with .060 material, which might elongate .010”.



### Wrapped hems

Wrapped hem developments should be treated with caution. While they will generally develop with a .29 k-factor, they are at minimum made with a two hit process and subject to a bit more variation. If the part has a reasonable tolerance, then the risk is minimized. Often times, a wrapped hem is used as a safety edge or a cosmetic feature.

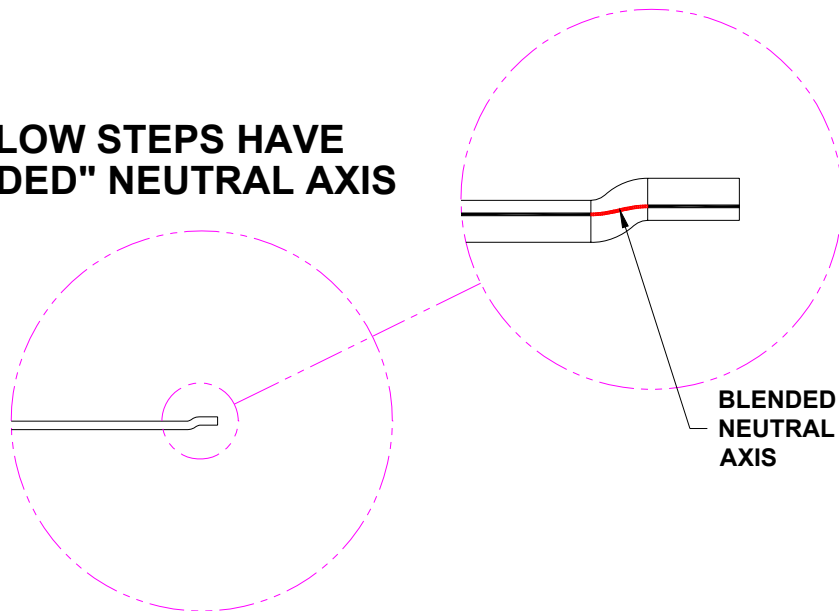
Additionally, a wrapped hem will see significant “backside” thinning which usually influences the leg length. Most parts are designed to a nominal outside thickness and not this backside “thinned out” thickness, so this must be accounted for in the developed length.



### Shallow z-bends

When a shallow bend is used, the neutral axis of one bend blends into another one and does not completely stay within the form arc. This often means that the developed length is only slightly longer than the flat length.

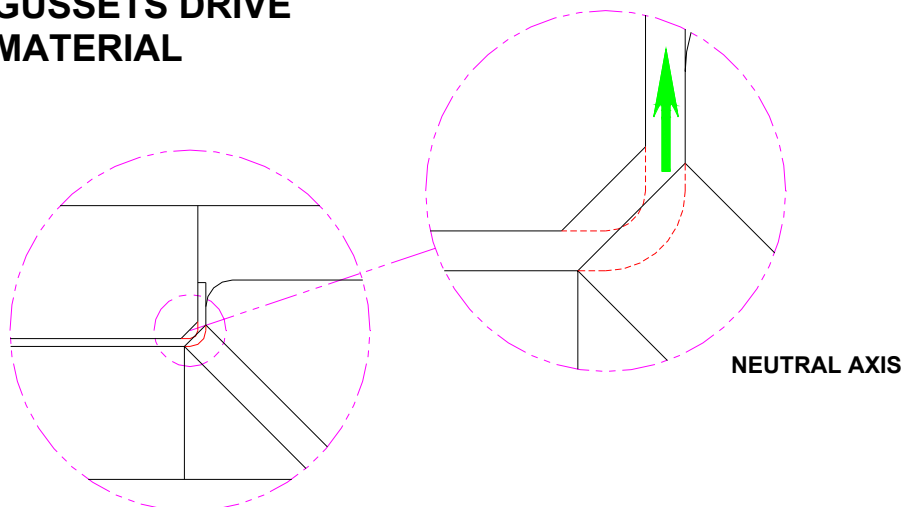
#### SHALLOW STEPS HAVE "BLENDED" NEUTRAL AXIS



### Gusseted Bends

When a part has a gusset in a formed flange, the gusseted area will generally form "high" as the gusset drives material beyond the expected development.

#### GUSSETS DRIVE MATERIAL



## 6. PDF Version

A printable PDF version of this information is available [here](#).

## 7. “Legalese”

All suggestions made here are for training purposes only.

The author is not liable for any loss relating to the misapplication or usage of this information.

You may freely use, reproduce and print this data as long as it remains intact in its original format.

The information presented here may be used for “paid” training purposes.

## 8. Document Info

Author: S. M. Adams

Last Revised On: 3/1/2005

Burrs on sheet metal parts are a reality that both designers and fabricators have to deal with. This paper describes some general principles about burrs and how to manage them.

When punching a sheet metal part there always a burr on the "part" and the "scrap" and the direction of these two is always opposed.

Part burr is generally down and the scrap burr is generally up.

For most all punching applications, the part's burr is down because the punch that creates the part is in the upper portion of the punching machine generally called the "ram" (punch press, turret press, etc).

This does not disqualify a burr from being up like in the case when the part is punched down through a die (matrix). There are also conditions where a part's burr is "reversed", for example when the majority of a part is punched in a progressive die (burrs down) and then the part is taken off the carrier strip by being punched thru the lower die. This is not generally a good practice for other reasons not covered here.

In most of the cases, burr direction is down and it is a good practice to indicate this condition and to show a flat blank with the burr down. This is sometimes referred to as "viewed thru the upper" (ram). Alternately, "Burr Direction" is opposed to another thing which is called "Punch Direction". When the term "Punch Direction" is used, it implies that the burr will be on the OPPOSITE side.

In general, part burrs are considered acceptable within 10% of the part thickness, hence, a part of .060" thickness will generally be allowed to have burrs of up to .006" high.

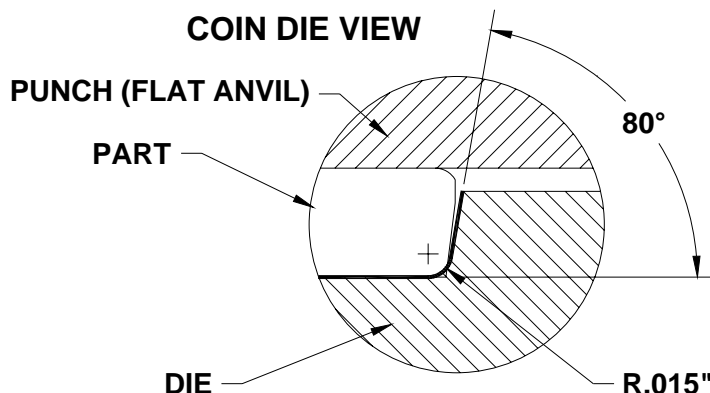
When designing parts for high production (i.e. tooled) it is a good practice to indicate "Burr Side" or alternately "Punch Direction". It is also a good practice to form burrs to the "inside" of a bend particularly the smaller it gets. Because the "sheared" portion of a cut sheet metal part has a high burnish, cracks are less likely to form on the outside of a bend (not to be confused with orange peeling). Burrs are also often formed to the inside of an enclosure for human safety reasons.

It is important for designers to consider the significance of burrs in a high production setting and the divergence of punched features from the ideal model. Generally, none of the cut edges on a sheet metal part is square to the faces of the part.

A cut edge has Roll-Over (about 5% of the thickness), Shear (about 30% of the thickness), Break (The remaining % of the thickness) and Burr, potentially up to 10% of material ([More Detail](#)).

It is generally not economically viable to have parts that are 100% burr free in any settings. Common strategies for burr management are:

- 1 - Know the burr liability and control the burr side by a callout based on the context of one's design. (Cheapest option)
- 2 - Secondary op tumbling (not generally bad for smaller parts and a good option for parts handled by humans). This may be done in the flat state or in the formed state.
- 3 - A sanding process known as "Time Saving" - this is where the parts are run between sanding belts - not too bad, but this must be done in the flat state. The edges are still a bit sharp, but have no protruding burrs. Another thing similar to this is called DQ sanding and gives a circular looking surface.
- 4 - Burr flattening within tooling. This is often accomplished by a hardened bottoming block in a progressive die or sometimes is a hidden extra of a forming process that "bottoms" on the part, but only in the area that is contacted in a tool. Again, this is a "no protrusion" item like the above and is not relatively costly (only needs tooling real estate) since it is part of the progressive stamping op.
- 5 - Coin Burr within tooling - This is where a part edge is actually coined. Generally, a contouring die is needed and this die has a .015 radius with a 10 degree "lead-in" and an upper anvil block that coins the burr. This is a true safety edge and will generally not cut people, but it is expensive to implement in tooling. If this type of feature is used, a straight edge will be less expensive than an irregular contour. A straight edge can be ground much like a conventional form block profile (i.e. it is a "cylindrical" body), but a contoured edge must generally be hard machined with carbide (blocks are hard A2 or similar tool steel - rc58-60) or conventional EDM'd. Alternately these are finished soft and the distortion engendered by heat treating is tolerated. In any case, implementing a coin burr process gets expensive fast. Additionally, it is common practice to implement a "balancing" coin to keep the part from pushing away during coining (inevitable but a balance keeps the coin crisp along a long edge for example).



6 - Electro polish - In this process the part is dipped in a conductive vat of stuff and electrically polished (kind of electrically eroded). The burrs generally die and the part gets a higher sheen.

7 - Human Deburring Labor - Be prepared to pay dearly . . .

In any case, that's a little ditty on burrs.

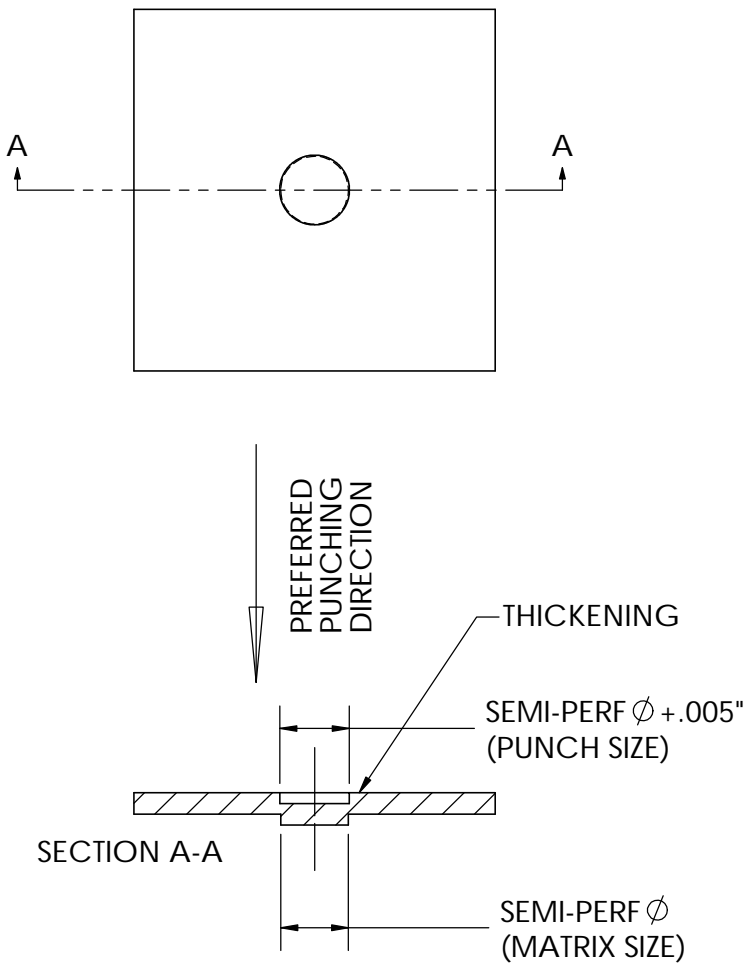
In a low production environment (laser), laser slag is often present where burrs would be, but it is usually remedied by Time Saving. In a low to medium (turret press) burrs are present and commonly there are "spike burrs" that come from line-to-line shearing.

Short answer - be aware that burrs are there and they can affect your design if uncontrolled.

# Semi-Perf

## SEMI PERF BASICS

- Heights are typically 1/2 material thickness, but higher heights are possible with stepped tooling.
- The punch is larger than the semi-perf to interrupt the shear line.  $+.005"/.15\text{mm}$  (total) is a common offset for the punch.
- Semi-Perfs higher than  $.025"$  ( $.6\text{mm}$ ) should be mechanically ejected from the die opening with a spring-pin.
- The backside of semi-perfs will have a thickened area due to the stock bulging around the rim.
- Semi-perfs are also called half-shear.
- These are used to locate one piece of sheet metal to another, usually through a mating hole or slot in another piece.
- The best direction to punch a semi-perf is down, but up semi-perfs are achievable with a hollow upper punch that acts as a matrix for the semi-perf. Up semi-perfs do not usually need to be ejected because the upper has a stripper or positive knockout.



## SEMI-PERFING

Description: This sheet outlines some common practices for semi-perfs.

TYPE  
T

©SheetMetalDesign.com

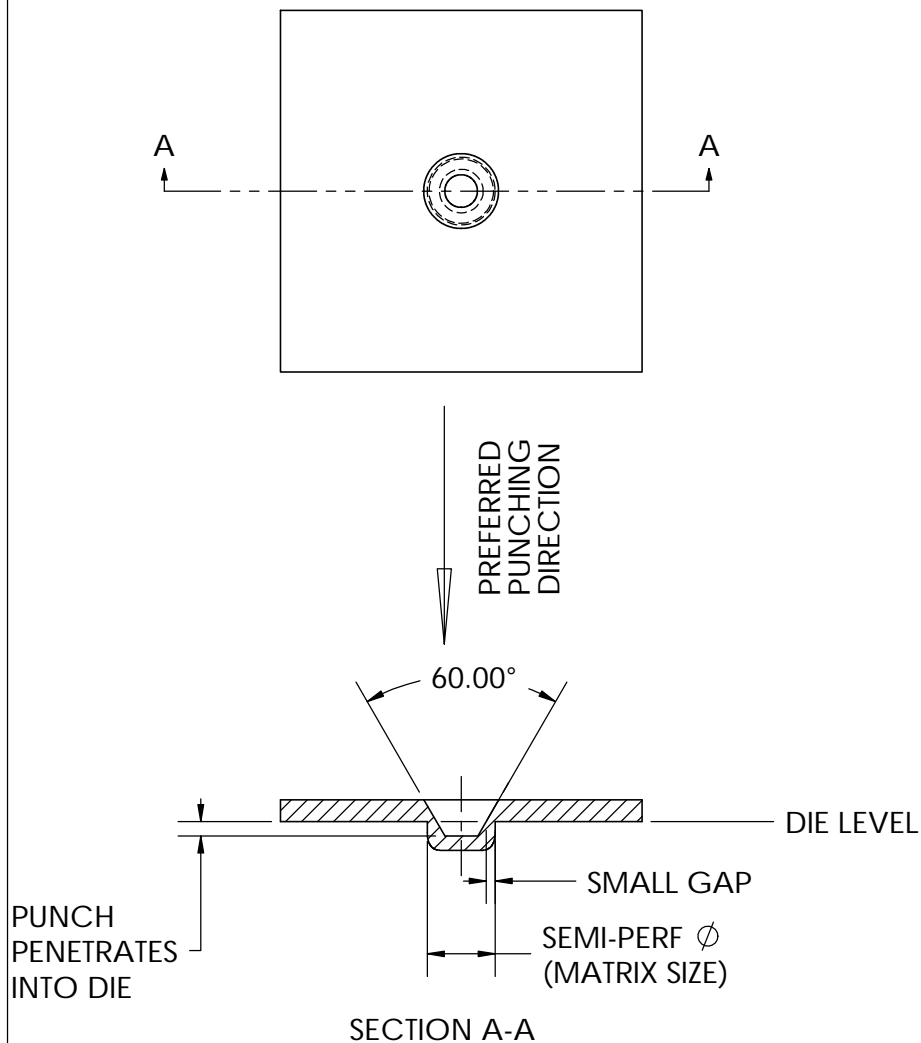
SHEET 1 OF 3



# Semi-Perf

## SEMI PERF - DOMED SEMI-PERF & TOOLING STRATEGIES

- Heights of up to a material thickness (for the dome) can be achieved. The active straight portion of the semi-perf is about the same as a standard semi-perf.
- The punch is angled for release from the workplace. This application is more of a swage than a perforation. Material flow is higher, so thickening on the punch side is more likely.
- The semi-perf punch actually penetrates the die to achieve a higher semi-perf height.
- These semi-perfs are easier to assemble as they provide a natural bullet nose.



### SEMI-PERFING

Description: This sheet shows a domed semi-perf that is assembly precess friendly an easy to attain.

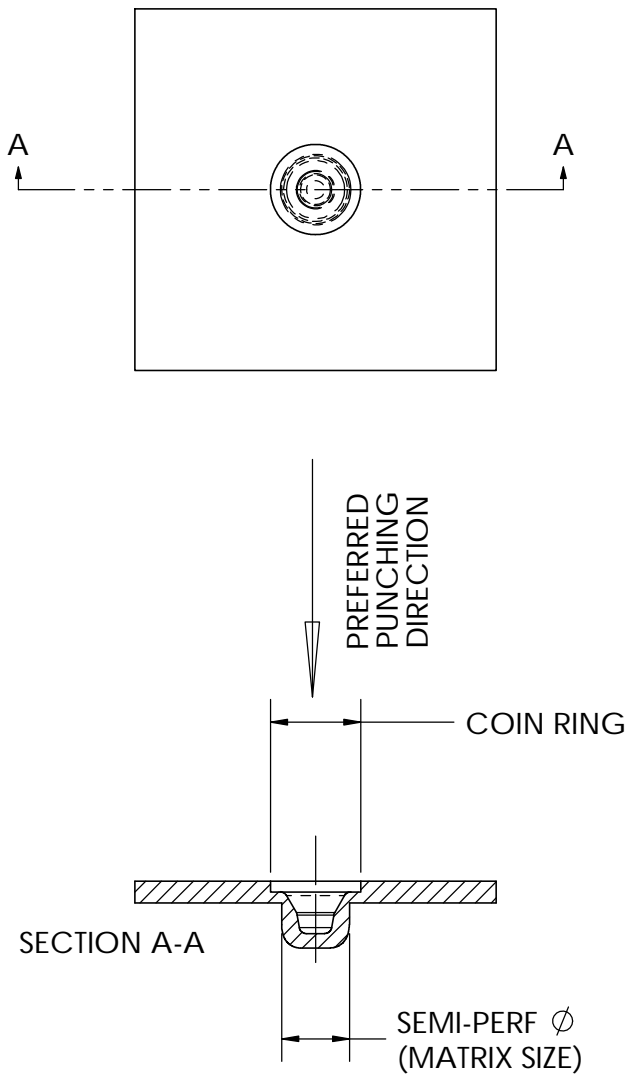
TYPE  
T

©SheetMetalDesign.com

SHEET 2 OF 3

# Semi-Perf

EXTREME HEIGHT SEMI-PERF



## SEMI-PERFING

Description: This sheet shows an extreme height semi-perf that can be attained with backside stock coining.

TYPE  
T

©SheetMetalDesign.com

SHEET 3 OF 3

## INTRODUCTION

Every sheet metal part starts as a flat sheet. There are many challenges in developing a flat blank for a part and considerable engineering time can be given to this task. This paper gives some suggestions and guidelines for effectively sharing CAD data to aid in the flat blank development process. It is often asked, "Should I give my suppliers a flat blank?" – Some will answer "no" and others will answer "yes", each with a compelling reason for their position. This paper will show the benefits of sharing a flat blank with a vendor and ways that CAD data might be shared without adding needless liability to the sharing process. We will also look at some of the liabilities when sharing and how to address those issues.

## BENEFITS OF SHARING DATA

When CAD data is shared, both the fabricator and the designer (buyer) can benefit greatly. Since a flat blank development must be created for any part to be made, some level of engineering must be done by the fabricator. To reduce the time needed for this process, having a model and even a flat blank supplied gives a distinct advantage.

In an ideal world, the fabricator and the part designer both work from a common data model and the CAD data supplied can correctly unfold to an accurate flat blank. Unfortunately, most fabricators and designers are not synchronized in such a way so it is difficult to realize this as a benefit. Designers rarely have the expertise needed to supply a completely viable flat blank. Many fabricators have an inherent distrust of customer supplied data.

Ultimately both parties pay for not having a good mechanism for sharing flat blank CAD data. In the worst case, the vendor is supplied with a part print only and is forced to recreate the model so as to attain a flat blank. As the part complexity grows, so does the time allotment for flat blank development. Vendors pass the development costs along to their customer resulting in a higher price. The time required for part development can also hinder delivery and translates to longer lead time.

When data is shared for the development of a flat blank, both the buyer and fabricator may benefit.

## RISKS OF SHARING DATA

There are a few classic arguments against sharing CAD data (particularly flat blank data) with a fabricator. The main argument against sharing a flat blank is that it is probably not exactly what the vendor needs. The issue is that the vendor will blindly cut the flat part to the supplied flat blank and may not be able to form the parts correctly. The vendor will not accept responsibility for the improper forming or development and will try to hold their customer responsible for "supplying a bad flat blank". In many cases, when a designer does share data with a fabricator and this bad outcome happens, the designer will simply avoid the problem in the future by not supplying the data. This of course is not an ideal situation, but one that many find themselves in.

Another common reason for not supplying data goes like this: The part print fully defines the part and the fabricator is “building to print”, so giving them data might make them ignore the print requirements and simply build to the CAD data. This is another common one and it sometimes happens when there is confusion between designer and fabricator as to what defines the part.

#### *HOW DOES SHARING HELP THE FABRICATOR?*

Sharing of CAD data is an immense help to the fabricator. When a model is shared, the fabricator has a head-start and can do a much better job of creating a correct program for a part. A flat blank (and a “fully formed model” as well), even if not perfectly developed can serve as a good starting point for a fabricator. Any fabricator worth their salt will be able to verify that a supplied flat blank is correct and adjust accordingly for their given process.

#### *HOW DOES SHARING A FLAT BLANK HELP THE ESTIMATOR / BUYER?*

An estimator (at a fabricator) can derive a great benefit from having a flat blank of a part supplied on a part print. A flat blank sheet can show the overall size of a part, required bends locations and the overall path-length for the part. From the path-length, the overall tonnage to blank the part can be easily established, giving the estimator a good idea of what press could handle the job (if it were tooled). The flat blank with bend centerlines might also help the estimator during a tooling package quote. Under these circumstances, the flat blank can be used as a “paper-doll” to help the estimator define bend sequences and determine the number of tools needed.

#### *HOW DOES SHARING HELP ME?*

Sharing CAD with your vendor gives them a head start and may translate directly into a reduction in your non-recurring engineering costs. Lead times will also generally improve as you strengthen your ties with your vendor and both sides benefit from the CAD sharing process.

#### *IT'S ALL ABOUT “RESPONSIBILITY”*

When making sheet metal parts, it should be very clear who is responsible for flat blank development. The fabricator should hold complete responsibility for developing an accurate flat blank suitable to their fabrication process in virtually every circumstance. Design engineers are typically not qualified to define all the parameters for a correct flat blank development and trusting them to define this is marginal at best. The fabricators know all of the special circumstances and conditions needed to define a correct flat blank for a formed part.

#### *WHAT CONTROLS PART PRODUCTION?*

There are three common scenarios. Each has its benefit and liabilities.

The most common method is “**Build to Print**”. This is a common method used where all of the part features are defined by a part print and there is no connection to a CAD model. All of the tolerances are clearly defined and all of the features are dimensioned.

This is probably the most common method of part definition and the “safest” as the part is fully defined in a single “universal” format (paper – requires no special technology to read). The liability of this method is simple – since paper cannot be used to directly define flat blank tool-path, engineering time must be allocated to create flat blank geometry in a CAD system. Everyone who builds the part must “recreate” it. Since the part is being reinterpreted, the likelihood of error is higher. This method can easily use traditional inspection techniques.

The second method is “**Build to Data**”. This is often used early in the development process to quickly establish a prototype without a lot of “front end” design engineering. This method is risky for the designer as it is difficult to document all the tolerances. The fabricators need to have a good working relationship with the designer and must have an understanding of critical features and the context of the parts. This method is possibly a good method for the fabricator, but does require that CAD tools are available. This method also assumes that the designer and the fabricator have a common CAD system or a neutral format that each can read and write (like DXF or IGES). This method will need to utilize “model” based inspection techniques, or presumes that the inspectors of the parts can chart a CAD model to validate a given part.

The third possibility is the “**Hybrid Model**” where some of the part features (usually those critical to function) are controlled by a “traditional” part print with dimensions and tolerances and other features are controlled by an associated CAD file. In this case, both the part print and the CAD model are needed to create the part. This is a good compromise between the “Build to Print” and “Build to Data” scenarios. In this case part validation requires CAD based interrogation of some sort (as described above).

#### *HOW CAN I DOCUMENT WHAT CONTROLS PRODUCTION?*

For “**Build to Print**” the absence of a note to use other means (i.e. CAD model) is sufficient to “legally assure” that the part print defines the part.

“PART TO BE FABRICATED ALL PART PRINT DIMENSIONS AND TOLERANCES.”

“123ABC\_REVA.DOC IS SUPPLIED FOR REFERENCE PURPOSES ONLY AND MAY BE USED FOR MANUFACTURING REFERENCE (PART PRINT CONTROLS PART FORM)”

For “**Build to Data**”, in the absence of other documentation, the purchase order might read:

“PART TO BE FABRICATED TO 123ABC\_REVA.IGS”

“ADDITIONAL TOLERANCES AND REQUIREMENTS ARE DEFINED IN 123ABC\_REVA.DOC”

For the “**Hybrid Model**” a part print with dimensions and tolerances is supplied and some note on the part print indicating:

"ALL UNDIMENSIONED FEATURES TO BE FABRICATED PER THE SUPPLIED FILE 123ABC\_REVA.IGS. FEATURES TO TAKE THE TITLEBLOCK TOLERANCE +/-0.010 UNLESS OTHERWISE SPECIFIED"

For any situation where a flat blank is supplied, it needs to be clearly noted that it is for reference only. If a layout of the flat is given, it is best to give it on it's own sheet and note that sheet as reference, for example:

"THE FLAT BLANK SHOWN IS SUPPLIED FOR REFERENCE PURPOSES ONLY. DEVELOPMENT RESPONSIBILITY RESTS WITH THE FABRICATOR. PART MUST CONFORM TO ALL DIMENSIONS AND TOLERANCES AS DEFINED ON THE PART PRINT. THE FLAT BLANK GEOMETRY WAS GENERATED FOR .060 THICK MATERIAL WITH .031 FORM RADIUS AND A .42 NEUTRAL AXIS.

### **CONCLUSION**

Since sheet metal parts cannot be created without a flat blank and in the vast majority of cases, today's 3D modelers can easily generate a flat blank, it is a great advantage for designers to share CAD data (formed models and flat blanks) with vendors. While there are some common pitfalls when CAD is shared, if a clear definition of the part acceptance criteria is established, both the vendor and the buyer can benefit. The vendor can focus on part fabrication and reduce their internal labor burden for programming and engineering. A buyer of parts can get quicker response time, more accurate parts and a less expensive part cost because of reduced engineering requirements.

## Introduction:

Outlined here are suggested practices for using the SolidWorks CAD software's sheet metal functionality for SolidWorks.

This information is supplied for training and may be redistributed and reused for training and instructional purposes, including posting on a company intranet and training (including paid) for reference usage.

---

## Parts:

### 1. Model A Formed Part For Most Parts.

Always model a completely formed part and unfold to get the flat blank. Don't make a flat and then try to bend it. This is absolutely impractical in 99% of all applications.

### 2. Define Sheet Metal Early In The Modeling Process.

Always define "sheet metal" immediately so that the "extrude to thickness" function is available. By doing this you will be able to "rollback" to the flat state periodically to confirm that you are making features that can be unfolded. Add all your new features in the "no bends" state (rolled back). Don't wait until the end to "insert bends" only to find out that your geometry is "unsuitable for unfolding". When inserting the sheet metal definition on a model with only a single panel, you will get the "*No Bends Found*" message. This is normal, as all your subsequent features will be placed directly after the sheet metal definition with the model rolled-back.

### 3. Name Your Panels.

When creating panel sketches, define them with descriptive names (example -----BottomPanel----, -----RightSidePanel-----, -----FrontFlanges---, etc). Always roll-back and add new features to the panel sketch if they are not needed as a pattern feature. For items that need a pattern in the future, add hole wizard holes that are sequentially after the panel in which they appear. Essentially, make good use of roll back to group all your features for a given panel together.

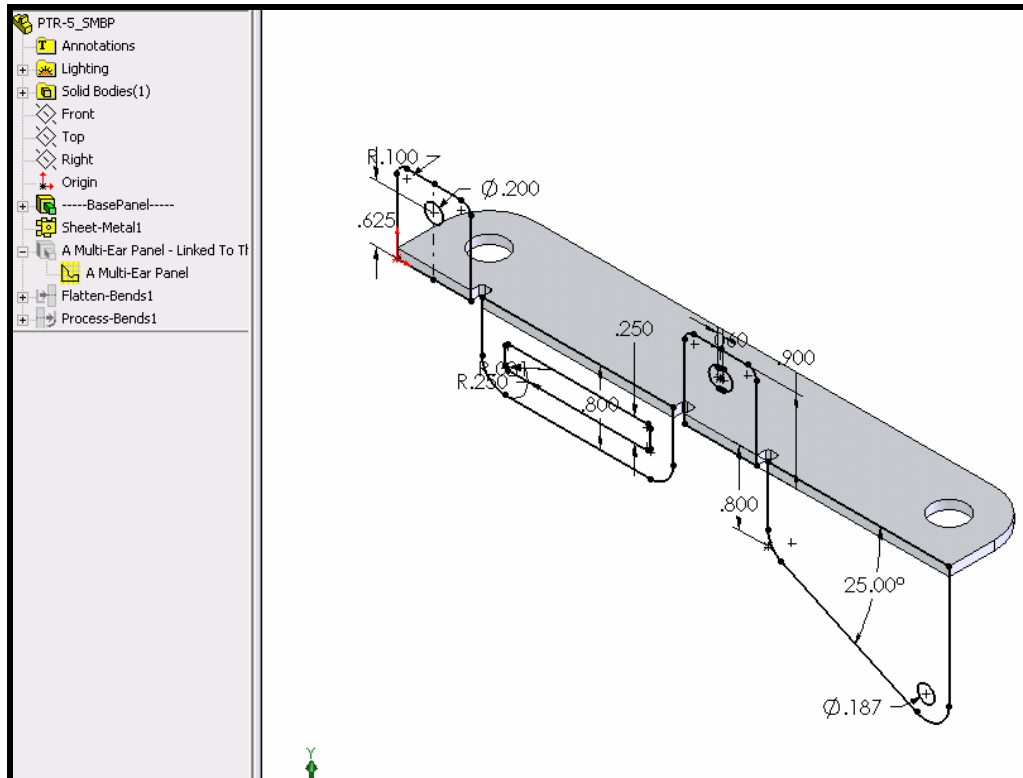
### 4. Let SolidWorks Create Your "Form Radii".

Let the modeler create the "form radius" fillets between panels. Model your panels and features with sharp corners and let the system create

the inner and outer radii for the model when you insert your sheet metal definition (or “roll forward” your sheet metal definition).

## 5. Use “Complex” Panels To Keep Your Tree Short.

When using panel sketches, be aware that you can extrude multiple closed contours to define many co-planar formed tabs that are not contiguous. Use a single extrude feature to define many panels that are coplanar.



## 6. Add detail to your edge flanges by using “edit flange”.

When using “edge flange” functionality, if more detail is needed in the flange, always “edit flange profile” to add detail instead of cutting away. Also, if the flange is already created, edit the sketch that created the flange to add more detail.

## 7. Control Your Bend Reliefs By Modeling Them.

Always model your own relief clearances, as this gives you maximum control over the form, size and location. If you decide to let the system generate your relief features, be prepared to compromise a little on the form, size and location.



## 8. How To “Globally” Redefine Your Material Thickness.

When needing to re-define the thickness of a part, double-click the sheet metal definition, revealing two model dimensions on screen (thickness & default radius). Edit the thickness and rebuild the model to update to the new thickness. Note: If your attempt to redefine your material thickness and the model “errors-out”, the panels defined may not have been extruded using “link to thickness”. To clear the error, re-edit your features using “link to thickness”.

## 9. Design With Small Gaps When Needed.

Design with small gaps (.001) between panels that will touch in reality. Doing so will allow the software to unfold the model. Don’t design one flange exactly coincident into another when it is in the folded up state, as the program will not be able to unfold the model. Of course this does not apply to any flange being taken off of its supporting base –these will always touch.

## 10. Minimize Your Reference To Bend Lines.

Don’t reference bend lines or the endpoints of bend radii when dimensioning any new feature, as this lends to possible instability of location when unfolded. Form Tools are particularly susceptible to this problem.

## 11. Use “Split Line” To Allow Two-Stage Unfolding.

For formed profiles that need to be bent in multiple tooling stages, split the form radius into two segments and constrain the break point, yielding two concentric bends that can be independently formed to simulate your tooling steps. This is helpful with multi hit bends, like for an acute bend that requires two hits or a curl operation that is hit in multiple hits. If you are on an older version of SolidWorks, use small “micro” lines between the arcs.

## 12. Use An Unfold To Make “Non-Oblique” Cuts.

Always add an “unfold” feature when adding a cutout to a flat blank, but only when the cut required cannot be put in any other way. I.e. when a cut needs to traverse bend lines at an angle in the flat, or a shaped cut needs to be put into a large radius.

## 13. Add A Fold After An Unfold.

Always add a “fold” feature after adding an “unfold” if a cut was made, or other feature added to return the model to its “finished” state.

#### 14. Understand The Limitation Of “Edge-Breaks”.

Avoid the edge-break command as it only does external edges, and not internal ones. Use the fillet or chamfer feature instead as you can control both inner & outer edges. As another alternative, add the fillets to the panel sketches if possible as this keeps the feature tree “tidy”.

#### 15. Shallow “S-bends” – Avoid Thin Extrude Features Here.

When modeling shallow “S-bends” that conform to the pattern line-arc-arc-line, always model the profile as a closed loop to assure more robust unfolding. Don’t use a thin extrude when there is an arc to arc condition with a shallow form, as bend deduction is less robust in this situation.

#### 16. Choose “Common” Form Radii For Your Defaults.

Use (inner) bend radii that are compatible with common manufacturing practices. Always specify inner radius as the outside radius is not easy to control. Keep the bend radius common throughout the part, as this allows you to keep all your relief clearances common. This also helps the manufacturer minimize tooling requirements. These sizes are suggested as commonly available:

Common Imperial Sizes	Radius Size
Often what is used for “sharp” corners	.008
Most Common	.016
Most Common	.031
Most Common	.062
	.125
	.250

#### 17. Add Configurations To Help Your Manufacturing Process.

When adding features such as extrusions that require a pre-hole, add the pre hole in a configuration and then model your extrusion “over” the pre-hole in a subsequent configuration. Doing this allows you to show both configurations for tooling.

#### 18. Use Numbered Configs For Progressive Ops.

When laying out a progressive stamping operation, create the model with numbered configurations (1 –flat blank, 2 –first form, 3 - second form, 4 –cam-pierce, etc.) that show the work to be performed at each station (whether in progressive die or hand fed). If the operations are performed in one tool, make an assembly with a component pattern. The component pattern's "repeat" would match the advance of the tool. Adjust each instance of the pattern to the configuration showing the work done at each station. Thus this can serve as a guide for your strip development activities.

## 19. Understand The Limitation Of "Form Tools".

Avoid the use of form tools as the geometry created by them is not entirely editable. In cases when you decide to use form tools, be aware that SolidWorks will let you model impossible features. I.e. Single sided louvers without any material "pull back" from the ideal shear line, lances tabs will have same size punch and die openings, which are actually different based on the punch/die clearance. Essentially form tools will not account for cutting clearance in many cases without some "extra" work. If form tools are used, be aware of the editing limitation.

## 20. Use Your Model To Create A Draw Tool.

When a draw tool is needed for a multifaceted part for which you have geometry whether imported or native to SolidWorks, the cavity function can be used to pattern an upper & lower punch & die, the same way a mold cavity is created.

## 21. Reduce or minimize "normal cuts".

Limit the use of normal cuts. While normal cuts are set as the default once a part becomes sheet metal, realize that this type of cut is not commonly needed for most sheet features.

## 22. Miter Flanges May Also Contain Arcs.

When using miter flanges, realize that the profile need not be a single line and that the miter need not be perpendicular to the panel that is taken off of. That is miter flanges may be complex profiles and need not always be 90 degree bends. Miters may only contain lines in 2001+.

## 23. Set Your Relief Ratio To A Reasonable Number.

When using the SolidWorks automatic relief ratio, use a factor of at least 1.0 instead of the default .500, as the default number is not

manufacturing friendly in any way. If possible, set this as high as 2. If you are comfortable with the registry, set these default values there.

## 24. Consider your “K-factor”.

When taking the default bend “k-factor”, if you intend to use your models for any manufacturing, set this to less than the (sometimes) default .500, as this will not unfold accurately for most bending scenarios. Manually override this setting in SW or if possible, set the registry to .42 or .33 based on your most likely tooling method. The table below shows some common k-factors, but always develop and test your particular material and forming conditions if you are unsure:

Bend Type	Common “K” factors
Large Radius Bend (radius 4x+ larger than stock)	.50
Rotary Bender (like “Ready bender”)	.43
Vee Bend	.42
90 Deg. Wipe - No Set Radius	.38
90 Deg. Wipe Plus Set Radius	.33

## 25. Legacy Installs Have Limitations.

Understand the limitations in 2000/2001+ when inserting base flanges. Parts created with this method cannot unfold swept profiles (i.e. miter profiles cannot contain arcs, nor will “wall” flanges unfold because they are inserted “post” bends). Individual bend parameters (k factor) are not configurable per bend, but inherited globally. 2003 remedies all these issues.

---

## Assemblies:

## 26. Use Real “PEM” Models.

Do create an assembly with all your parts and PEM nuts in them. Use real PEM models from a library, toolbox or the PEM website

[www.pemnet.com](http://www.pemnet.com).

Don’t use fake PEMs or part features to simulate PEMs. If you use “fake” PEMs then use configs to make the proper hole without the hardware.

## 27. Define Semi Perfs In The “Attached” Part.

Always define semi-perfs in the parts that they are in and have then drive the size and position of the corresponding in-context “hole” features in the receiving part (typically a hole & slot).

## 28. Use Patterns For Hardware.

Always use patterns for multiple PEM and fastener features so that you may insert a single PEM into the “primary” hole and use a component pattern in the model. Always name these patterns so you can easily identify the parts.

## 29. Use Assembly Overlays To Show Part Progression.

For tooling layouts in particular, to show a part in its pre-tooled and post-tooled state, create an assembly of your sheet metal part and superimpose an instance of your part upon itself. Create 1 assembly configuration corresponding to each sequential per part state (see PRT-18, configure instance 1 to “2 –first form” and instance 2 to “3 - second form”). Then create a drawing of the assembly showing instance 1 and use an alternate position view to show instance 2. In this way, you can get a superimposed view of 2 states of the part. This is critical as all tooling receives a part in a given state and outputs it in another state. This is a work around for the fact that alternate positions cannot be made for parts.

---

## Drawings:

## 30. Use The “System Flat Blank”

When initially inserting your flat blank into a drawing, always take the system generated “flat pattern”, as this flat blank “joins and solidifies” the panels into one panel with continuous profiles. For CNC processes, this eliminates the “line-line-line” condition that may hinder turret programming (in particular).

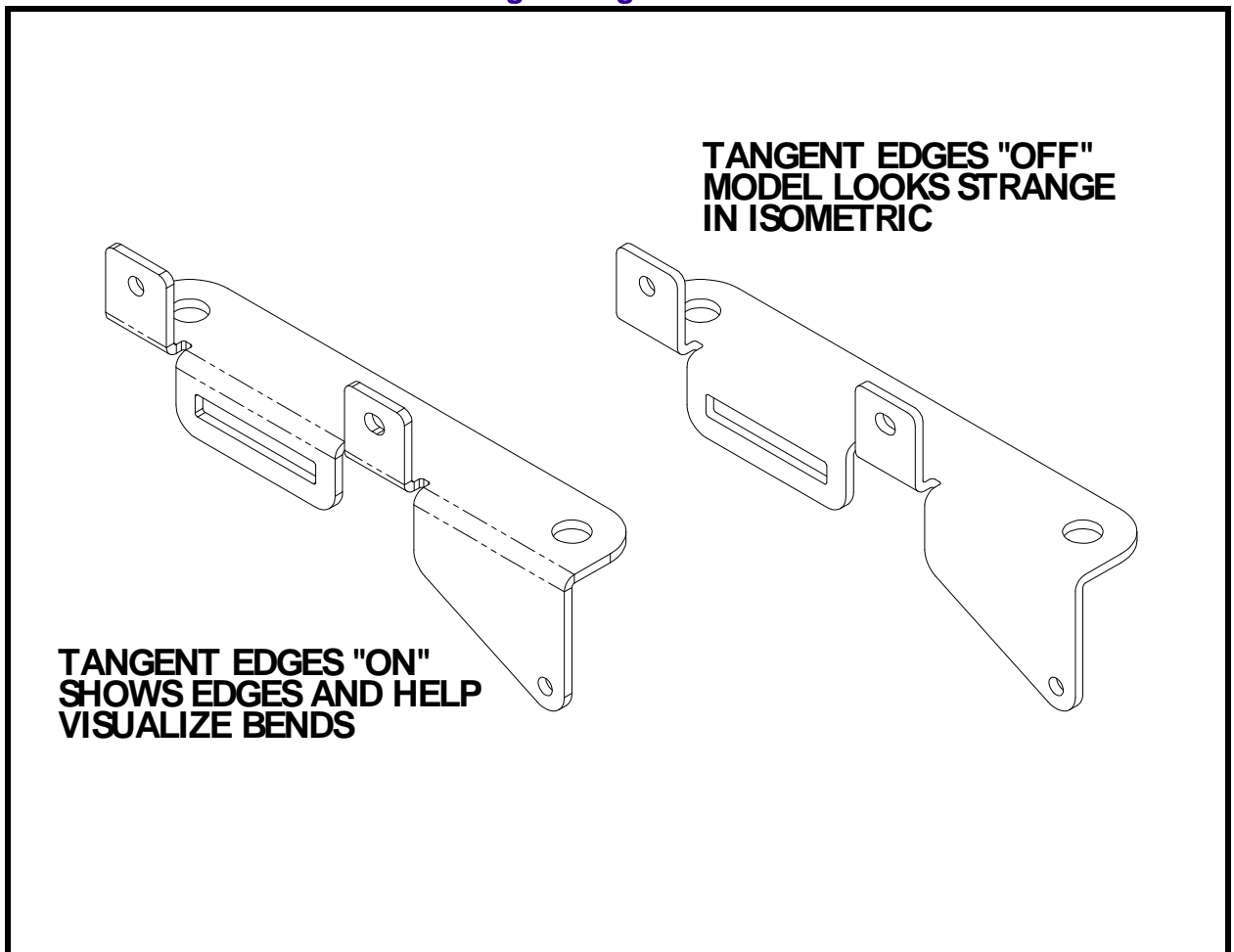
## 31. Provide A Reference Flat For Quoters & Fabricators

Always provide a reference flat blank with any design for estimation and manufacturing purposes. Use the “measure” command to chart out the

full “perimeter” of the part (simply touch the main surface) and include a note indicating this length. This can be very useful in determining laser time, cutting tonnage and wire-cutting time for hard tooling. Always require your vendor to be responsible for bend development and flat pattern accuracy and note this in the layout.

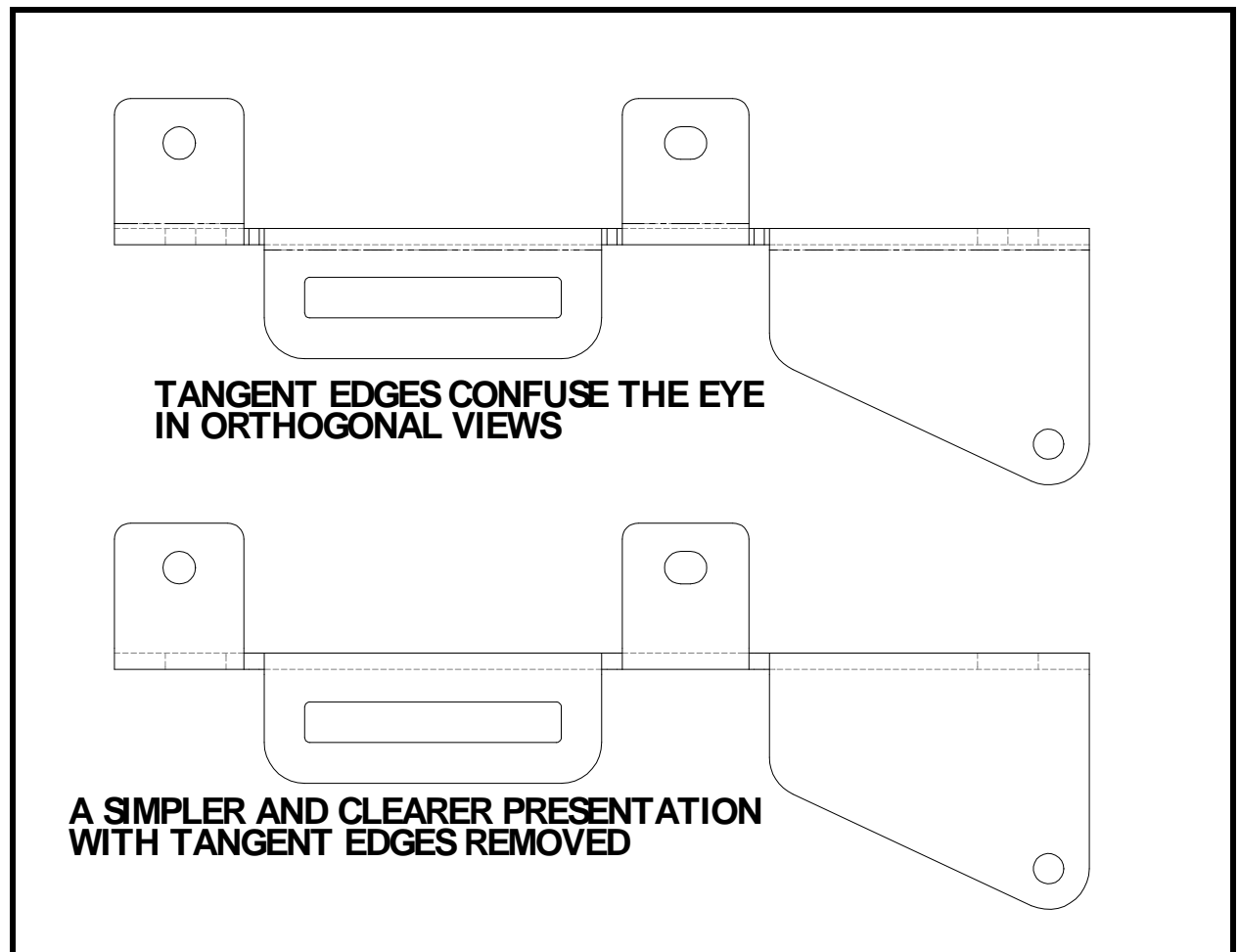
### 32. Turn On “Tangent Edges” In ISO Views

When inserting an isometric (or other 3D view) of a sheet metal part, always turn on “Tangent Edges with Font” or “Tangent Edges Visible”. Never leave the view with the tangent edges turned off.



### 33. Turn Off “Tangent Edges” In Ortho Views

Generally turn off “Tangent Edges With Font” or “Tangent Edges Visible” in an orthogonal view, as this often confuses the eye with the stock thickness. Ignore this rule when you need to show clarity.



---

## Imported Data & Feature Works:

### 34. Sharpen & “Strip” Radii

When working with an imported sheet metal part, to gain control over the inner form radii, use feature works to “recognize” all the inner and outer (form) radii and then delete them from the model. After this is accomplished, insert the sheet metal definition into the model that now has dead sharp corners and you will be able to define the exact form radius. Use this method to overcome the “locked” state of form radii on imported models.

### 35. Recognize Panels With FeatureWorks

For an in depth treatment, see [Using Imported Data](#).

For imported sheet metal parts, to gain control over the panel sketches, use feature works to strip the radii as above and then manually “recognize” each panel starting with the outermost panels and working your way “inward” to the panel that shares the most bend lines with other panels. Once this is done, add sheet metal and edit the sketches as needed.

### 36. Verify Recognition With “Quick” Unfold

After initial insertion of an imported model, insert a sheet metal definition into the part to verify its unfoldability. If it unfolds without issue, the part will be “receptive” to full feature works recognition.

### 37. Get Data Closest To Your Kernel

Get data that is closest to you kernel as possible. I.e. get parasolids before IGES if possible.

### 38. Use A Native To “Dumb” Overlay Assembly To Confirm Your Part

When you are recreating a native SW model (if you are a vendor) from a customer supplied imported model, define an assembly and superimpose your model upon theirs (with mating). This allows you to make a shape comparison of both models. Ideally, you would superimpose your model upon theirs immediately after defining the first main panel. Doing this allows you to check for errors feature-by-feature as you create your model. Additionally, you can now derive edges from their model to develop your panels. Also this may be the only available method to derive geometry from the supplied model in the case of a “hybrid” (i.e. paper print + model control the part geometry).

---

### “Legalese”

All suggestions made here are for training purposes only.

The author is not liable for any loss relating to the misapplication or usage of this information.

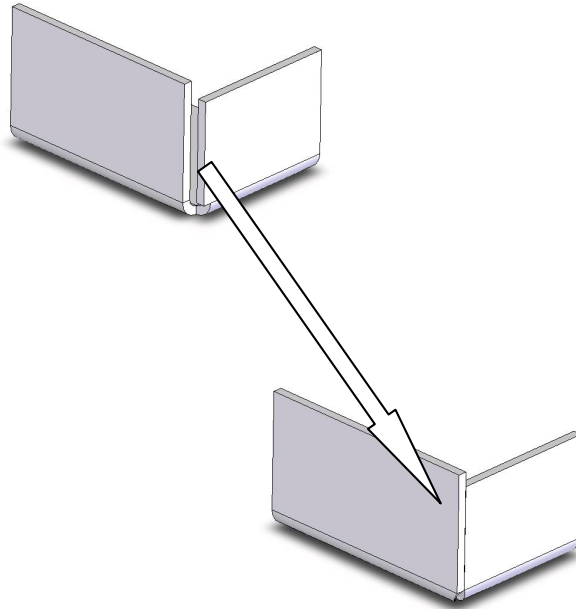
You may freely use, reproduce and print this data as long as it remains intact in its original format.



The information presented here may be used for “paid” training purposes.

### 1. Introduction:

This is a “how-to” which explains the different scenarios for the SolidWorks “closed corner” sheet metal feature. It explains some of the limitations and situations. Also, some alternatives to the closed corner and work-arounds are suggested. Essentially a “closed corner” closes the gap in the corner of a sheet metal part.



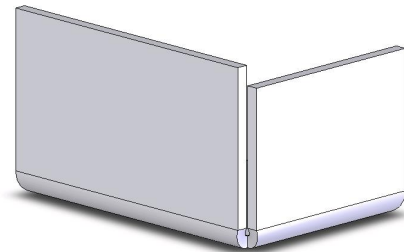
**Open Corner Becomes Closed**

## **2. Basic Use and Definitions**

### **Closed Corner Types**

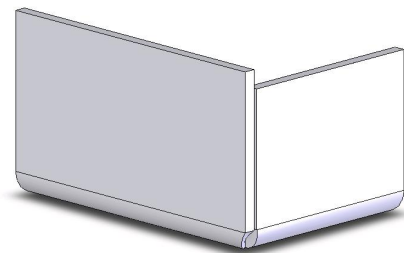
A sheet metal closed corner can extend one or two corner faces as needed to create three possible corner options:

#### **Type1 – Inner Corner:**



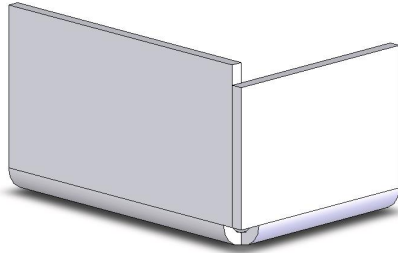
**Inner Corner to Inner Corner.** In this case, both panels are extended to meet in a common corner. This might be useful when using a corner weld on the outside and a minimal envelope is needed for the part.

#### **Type2 - Outer Corner:**



**Full Overlap to outer surface.** In this case, one panel is extended to the outside of the other.

### Type3 - Inner Plus Overlap:



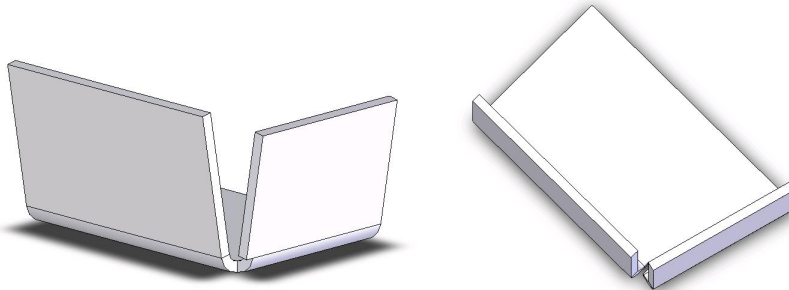
Extension to inner surface with outer overlap. In this case, one panel is extended to the inside of the other and the outer one is extended to the outside of the other. The simple case shown here was modeled with “traditional modeling” to show the third option, which correctly previewed, but would not resolve as previewed using closed corner. Generally this type of corner is hard to attain with the standard closed corner feature, but can be easily modeled.

See [Section 4](#) below on alternatives and work arounds.

## 3. Conditions / Failure Modes

### Required Conditions For Closed Corner:

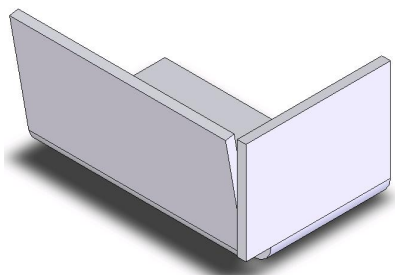
They only work for 90 degree corners with 90 degree flanges (except with “inward” offsetting).



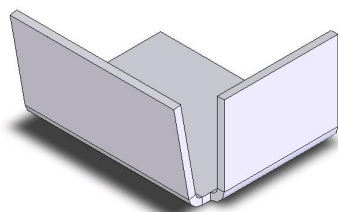
These fail:

An exception to the 90 degree rule is when a flange is offset into the part. Then an “inner to inner (Type1)” closed corner will work, but will only influence the second flange and not trim the first.

## SolidWorks – Closed Corner



**This works:**



**This Does Not:**

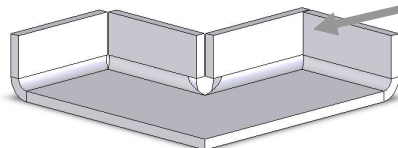
### Use With Edge Flanges

Closed corners are used in conjunction with edge flanges (i.e. not bends that come from the “insert-bends” method).

### Use With Miters

They work well on external miter flange profiles, but with the corner relief conditions of the miter (typically splined relief). Also works on internal miter corners, but with the flat blank intersection error that is inherent with miters.

This miter allowed closed corners, but could not unfold one corner:

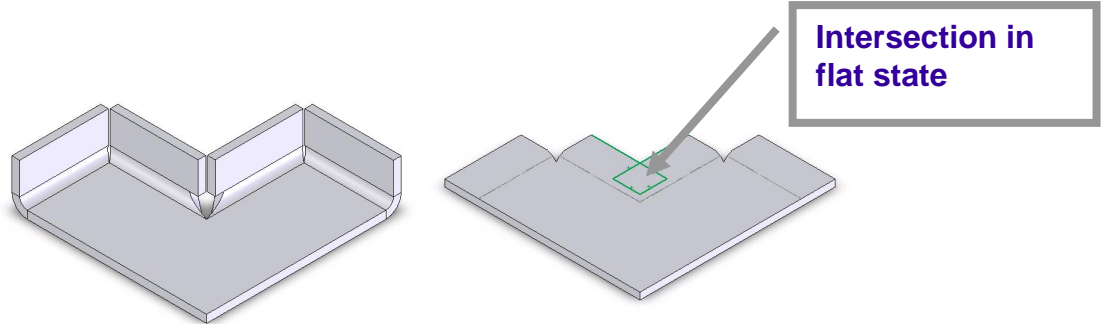


**Problem Area  
For Unfold**

A “Type1” inside-inside closed corner was substituted for the problem area shown above and the part would then unfold.

## SolidWorks – Closed Corner

Note the splined relief (bend waist) in the formed & flat part as well as the intersection in the flat:

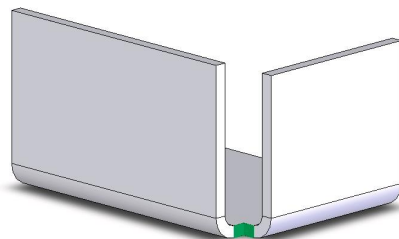


Note: It is very common for the modeler to preview a state of a closed corner which it will not be able to create. This is the “down side” of using the closed corner. While it shows the preview, it will often times not create the feature.

### Common Failure Modes:

#### Modeled Pre-Relief

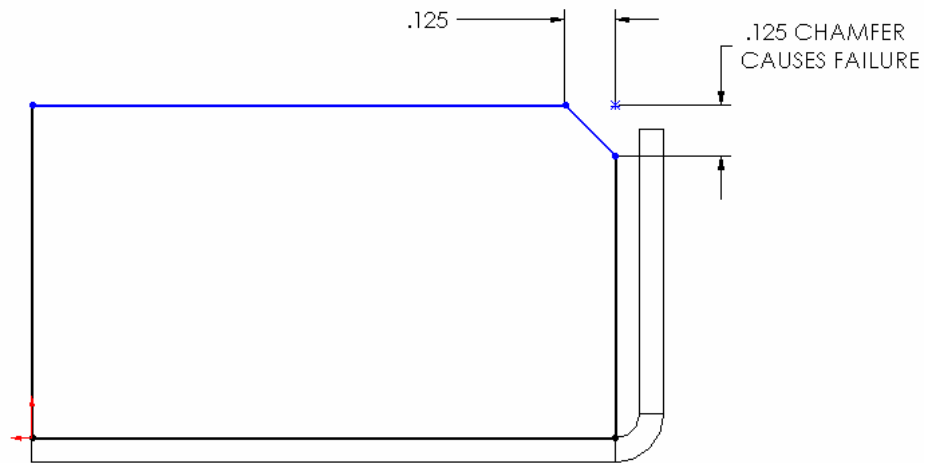
When the given corner has been pre-relieved, or does not originate from a single point. This is similar to the “excessive offset” problem, but will fail even if the pre-modeled relief is less than the bend radius. The feature depends on the original edge flanges and relief being taken from an orthogonal point as the meeting point of three planes (bottom and two side flanges)



#### Flange Sketch “Near Edge” Modified

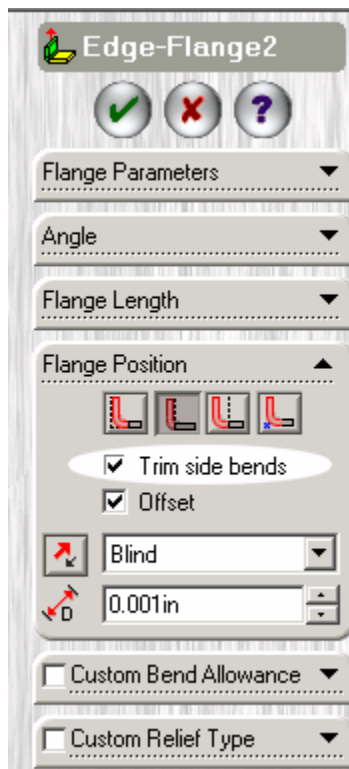
When a radius or chamfer has been added to the top corner of the edge-flange sketch or the sketch has an interruption in the edge near the corner.

## SolidWorks – Closed Corner

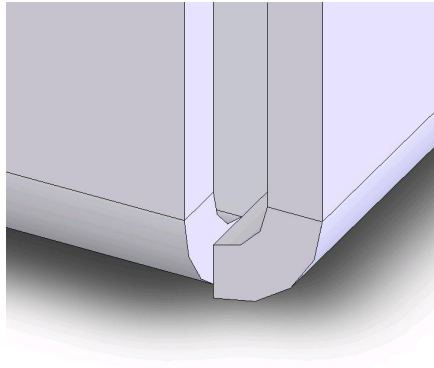


### Trim Side Bends = Off

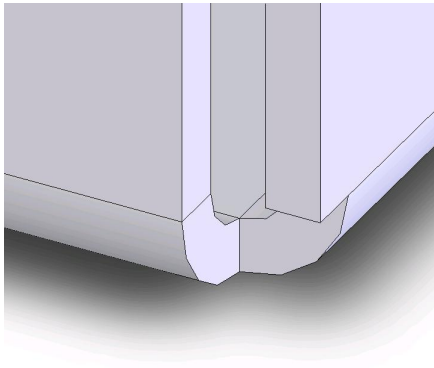
When the “Trim Side Bends” option is off, closed corners will not work. As an added note, “Trim Side Bends” only seems to work for 90 degree corners in any case.



## SolidWorks – Closed Corner



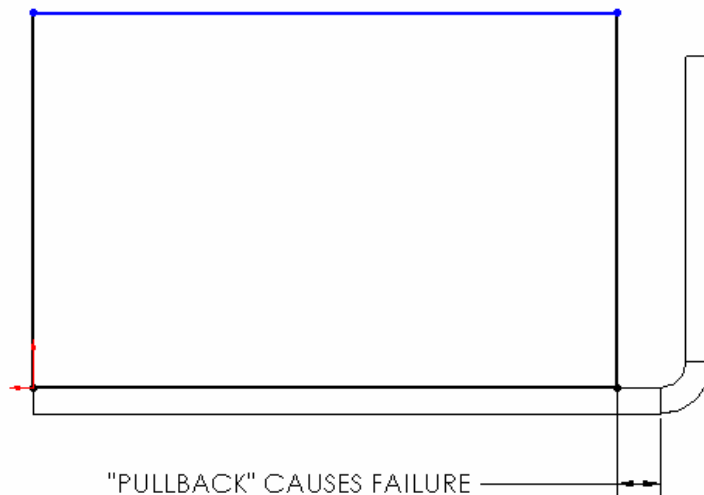
With “Trim Side Bends” Off



With “Trim Side Bends” On

### Flange Sketch Pull Back

When the edge of the face to be extended has been pulled back from the corner (even .001), the closed corner will fail (pulled back or moved away from the corner via dimension).

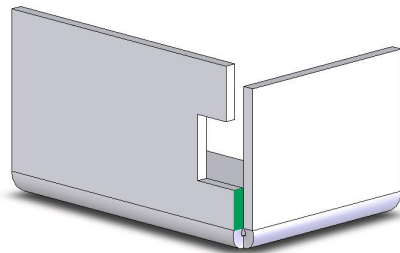
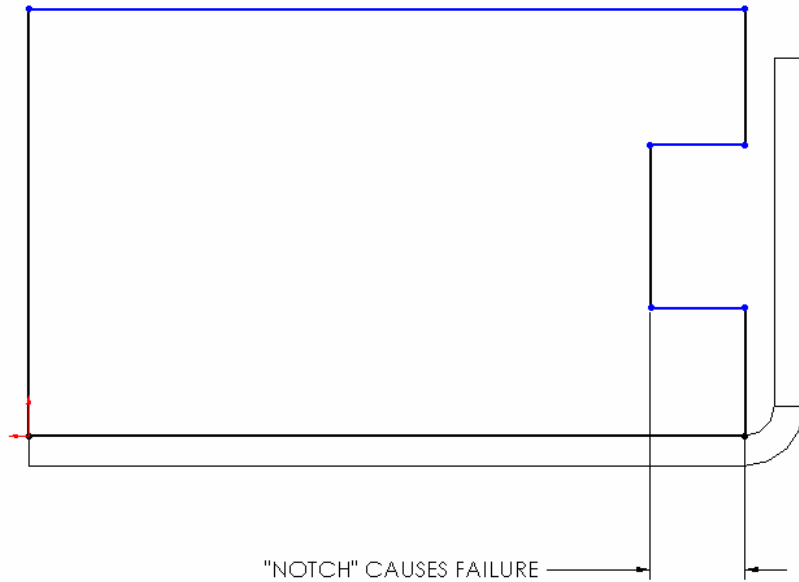




## SolidWorks – Closed Corner

### Sketch Modification On Near Edge

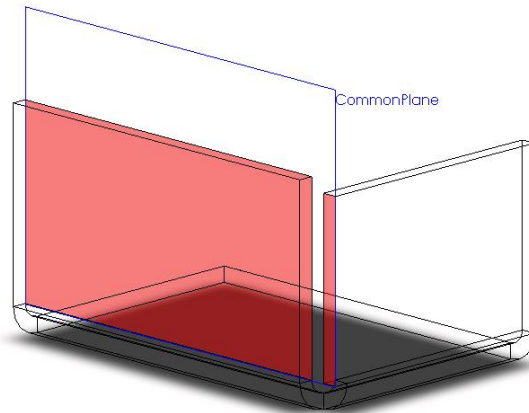
When the Sketch has been notched, only the face directly adjacent to the corner can be extended. (I.e. Extension is not multi region).



Only the bottom edge can be extended in this case.

### Perfect Planarity

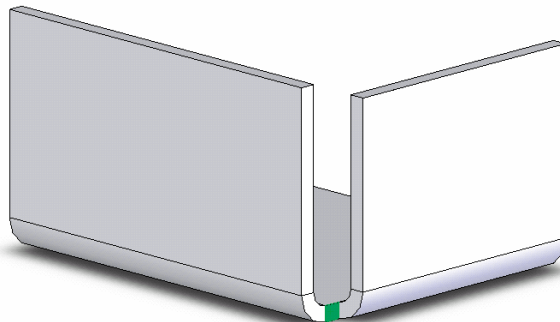
When “Perfect Planarity” exists between the inner face of an edge-flange and the “extended to” edge-flange, the closed corner cannot reach the far-side. Try adding a .001 micro-gap offset to the “second” edge-flange feature.



Perfect planarity is when the inner edge flange and the outer flange edges (red) are perfectly co-planar. This condition limits the ability of the closed corner to reach the outside of the part. This is a variant of the “zero thickness geometry” limitation in the modeler.

### Excessive Offset in “Second” Edge Flange

When the “second” edge flange has been offset too far to the outside, a closed corner is not attainable. The closed corner will fail when the offset exceeds the inner bend radius of the edge flange.



The green region shows the offset amount.

## 4. Alternatives / Work Arounds

Note: Both of these alternatives fail under perfect planarity conditions, as do most sheet metal features.

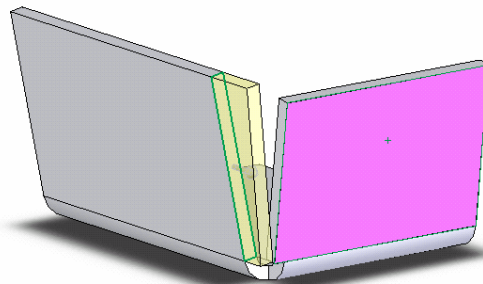
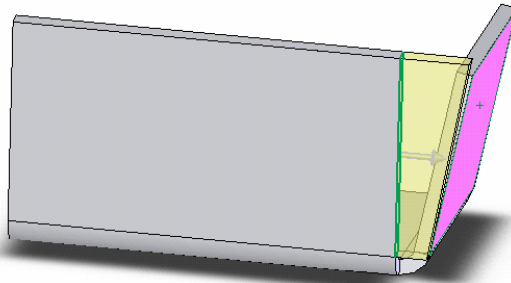
### Traditional Extrude

This method also allows shortened extensions (I.e. Mid-material) and multiple profiles.

The basic method is shown below on a corner with two 80 degree edge flanges. This works as well with a 90 degree bend or an alternate angle.

The nice part about this method is that it will work for many angles and is more robust when the angles are changed.

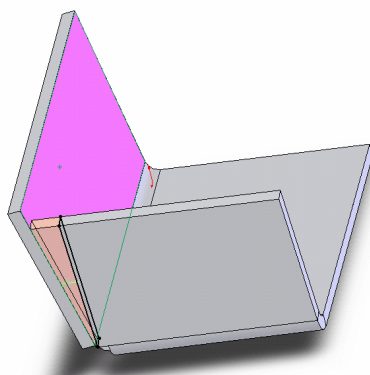
First, extrude the end face “Up to Surface”



(Alternate view)

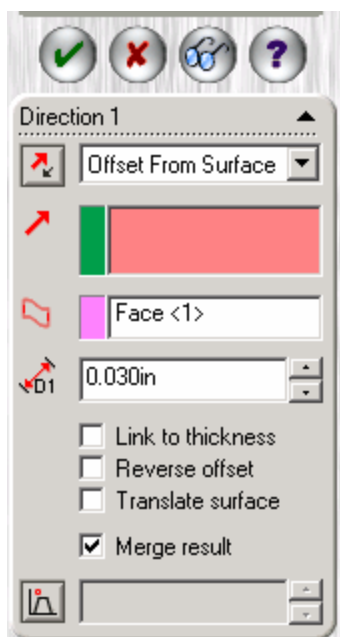
Then, Extrude the second end face “offset from surface” by .001 or whatever makes sense for your given manufacturing process.

## SolidWorks – Closed Corner

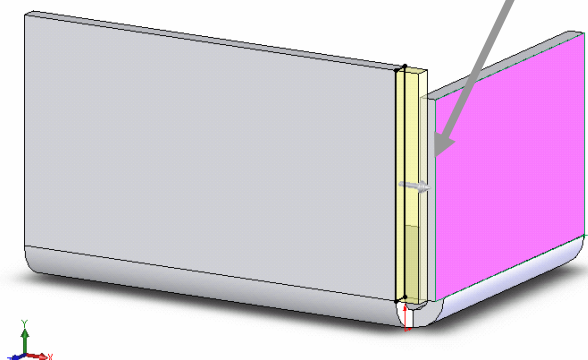


For the method above, the desired corner style determines which surfaces to extend to. For example, if a “type 1” corner is needed, the inside surfaces would be selected instead.

Additionally, this is a great method for closed corners with a “mid-material” requirement (for example, when an outside welded corner bead or tooling clearance is needed). In this case, “Offset from Surface” can be used and made 1/2 material thickness, or whatever is needed.



Weld Ledge or Clearance

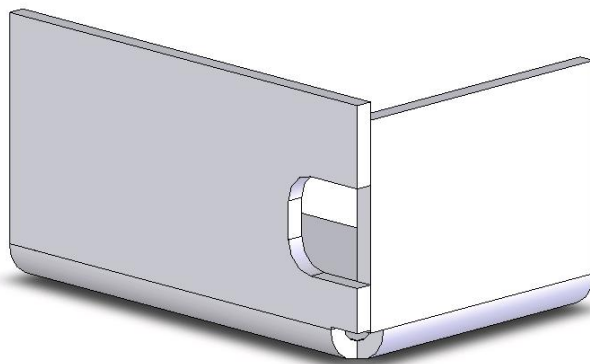
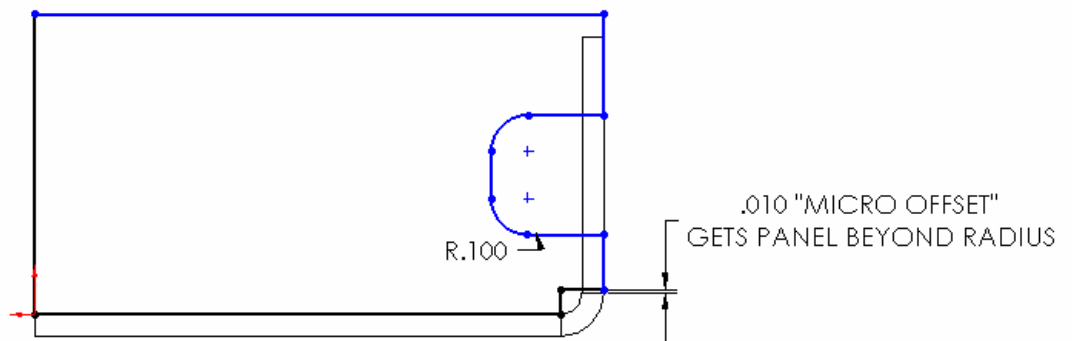


### Edge Flange Sketch Detail

The “sketch method” is probably the most elegant and easy to use. It simply consists of making the sketch for the second flange have more detail and creating the overlap as the required application.

Often the required closed corner can be attained by editing only one sketch. The closed corner may also have interruptions in the panel as shown below, or may be a simple solid panel, as needed.

Additionally a sketch modification can be combined with the “traditional” extrude described above.



### **PDF Version**

A printable PDF version of this information is available [here](#).

### **“Legalese”**

All suggestions made here are for training purposes only.

The author is not liable for any loss relating to the misapplication or usage of this information.

You may freely use, reproduce and print this data as long as it remains intact in its original format.

The information presented here may be used for “paid” training purposes.

## **1. Introduction: Tips for SolidWorks Flat Blank Features & Using Fold/Unfold**

The SolidWorks fold/unfold is a very powerful sheet metal feature. With Fold/Unfold any given model flange (or set of flanges) may be unfolded to the flat state and then re-bent back to the finished state. Features, like cuts & extrudes, may be added to the model when it is in the flattened state.

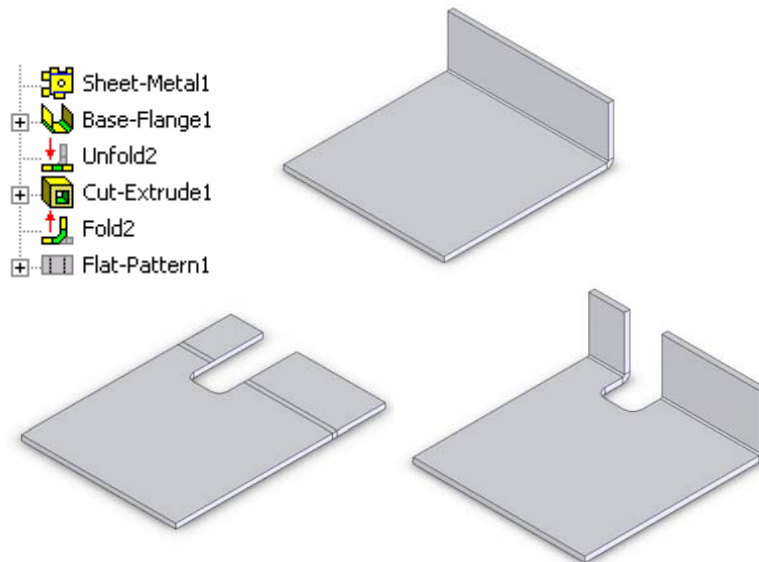
## **2. Nest Features in Fold/Unfold**

When creating features in the unfolded state, always “nest” them between a fold & unfold feature. This allows you to make your flat feature and then return your model to the finished state.

## **3. Many “Flat Features” Are Attainable By Conventional Methods.**

Determine the most suitable place for an unfolded feature – often times a flat feature is not necessary. One common example of a less than perfect use is shown below.

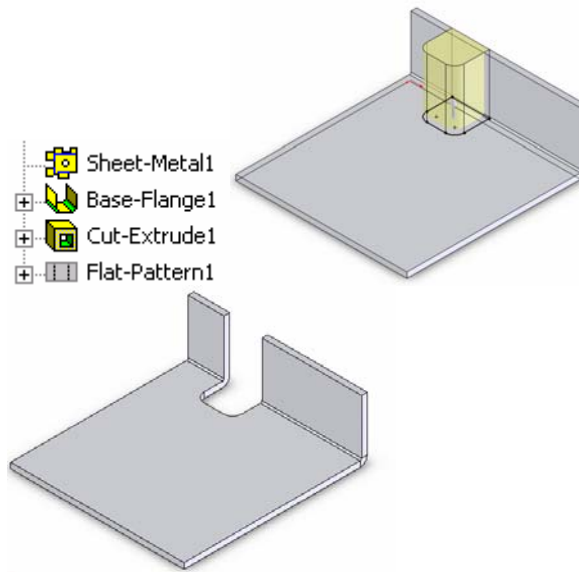
This sequence (unfold-cut-fold) makes the least sense and makes for a larger feature tree and the need to unfold & refold the geometry to rebuild the features, which translated to longer rebuild times.



**Features 8, Solids 1, Surfaces 0: Total rebuild time in seconds: 0.39**

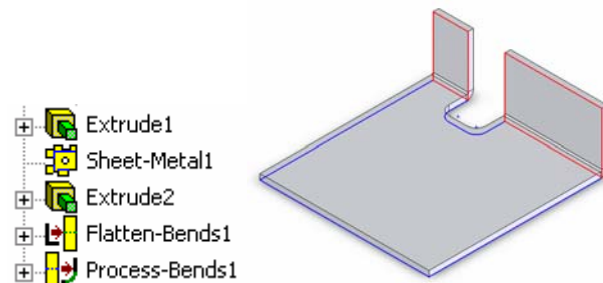
This “traditional cut” on base flange makes more sense and rebuilds faster:

## SolidWorks – Flat Features



Features 6, Solids 1, Surfaces 0: Total rebuild time in seconds: 0.14

Or even better, an “insert-bends” model with two sketches (red & blue).



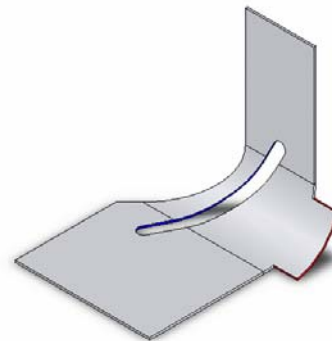
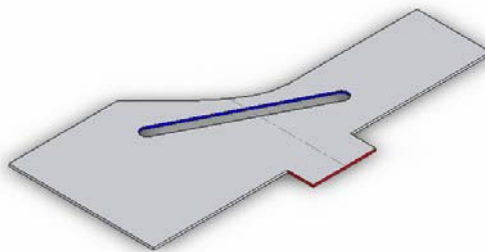
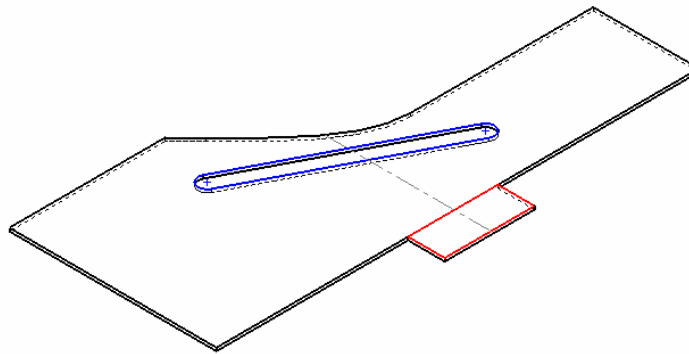
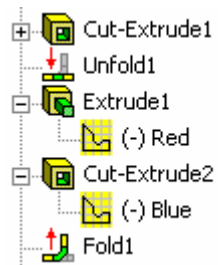
Features 7, Solids 1, Surfaces 0: Total rebuild time in seconds: 0.75

### 4. When to use “Flat Feature” geometry

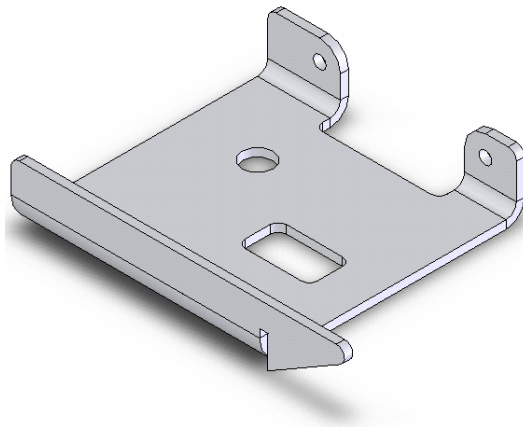
A flat cut is needed when the cut crosses a bend-line at an angle, or a circular cut crosses a bend line. When a flat cut does not cross a bend-line at an angle, a flat cut feature is not necessary.



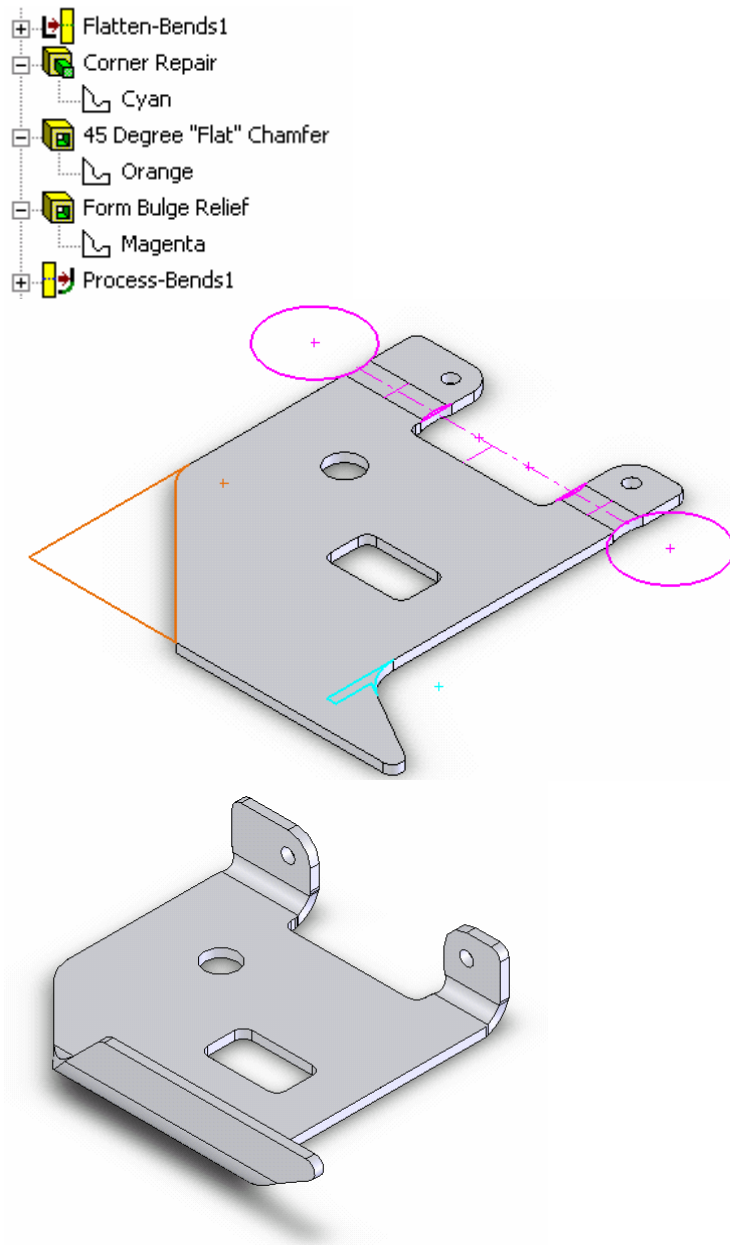
## SolidWorks – Flat Features



A flat extrude is needed when automatic corner clearances need to be filled. This method works with the “insert-bends” method and the features are put in between the “Flatten Bends” and “Process Bends” features. Sample Model [Here](#).



## SolidWorks – Flat Features



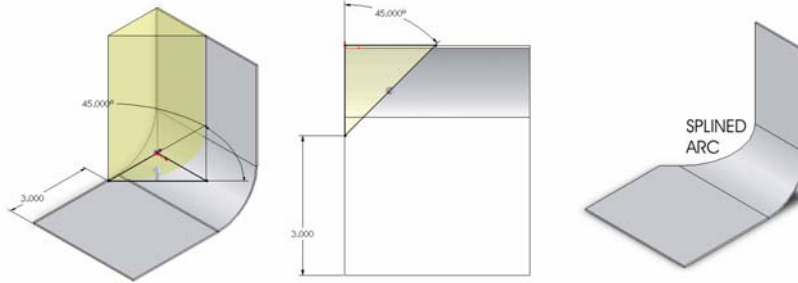
### **5. Simplify bends only simplifies single segment linear cuts**

When “simplify bends” is toggled on, the simplification of the resulting spline is “chordal”. Additionally, multiple segment linear cuts and curved cuts will not be simplified. The interaction is most apparent when a cut is made in a large radius.

A cut crossing a bend radius at a non-orthogonal angle created a splined arc when flattened:

## SolidWorks – Flat Features

These simple types of cuts will simplify:



"CHORDAL"  
SIMPLIFICATION

SPLINED  
SEGMENT

Flattened Part -- >

### 6. In context use of flat features is limited

Flat-state cuts are not generally able to have viable “in-context” relationships defined, as they show the model in a “non-finished” state.

### 7. Unsuppressing the flat-pattern feature to flatten the model

In many cases, unsuppressing the flat-pattern feature will flatten the model.

### 8. Direct suppression of bends

Often times, when the flat blank is unsuppressed, individual bends may be suppressed to return them to the folded state. The advantage of this is that the bends that are unfolded will have merged faces, which make the output geometry cleaner.

### 9. Fold/Unfold + Configurations – Extremely helpful for manufacturing processes

When a part will go thru many bending operations, configurations with fold and unfold states are very helpful in documenting the manufacturing process. Additionally, a step by step process for the part is absolutely essential for tooling design. As formed parts migrate through the manufacturing process, they go from flat sheet to finished part. SolidWorks can very nicely document this process.

## 10. PDF Version

A printable PDF version of this information is available [here](#).

## 11. “Legalese”

All suggestions made here are for training purposes only.

The author is not liable for any loss relating to the misapplication or usage of this information.

You may freely use, reproduce and print this data as long as it remains intact in its original format.

The information presented here may be used for “paid” training purposes.

## 12. Document Info

Author: S. M. Adams

Last Revised On: March 8, 2005

Status: Working Draft - Under Development

# Using Imported Data With SolidWorks Sheet Metal Functionality

## 1. Simple Flattening Of an Imported File

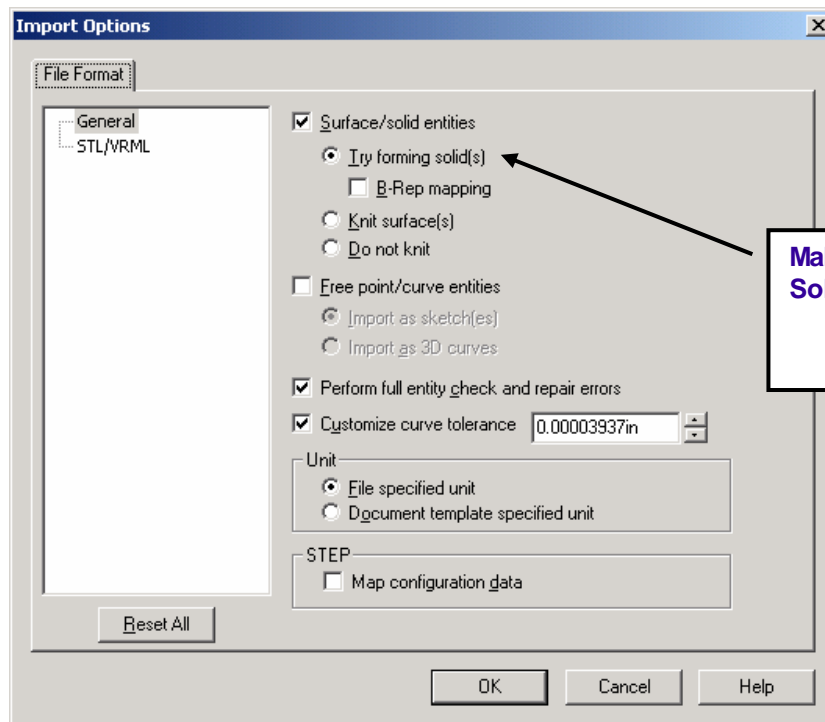
### Imported Models Will Unfold

Often times you might have a model available to flatten which you did not create and has been sent to you as a neutral format. You might need to get a flat-blank layout to make a laser or turret program from. Many fabricators find themselves in this situation, especially during the pre-production phase of a project. You can save considerable time and effort when you use your customer's data to create a flat blank.

## 2. Import Your File

### The Model Must Be “Water-Tight”

Using the “open” function, import the file using the appropriate filter. Once the geometry has been imported, you want SolidWorks to try to form a solid. If your file has surface data that is “water tight” then the file should import and form a solid without any trouble.



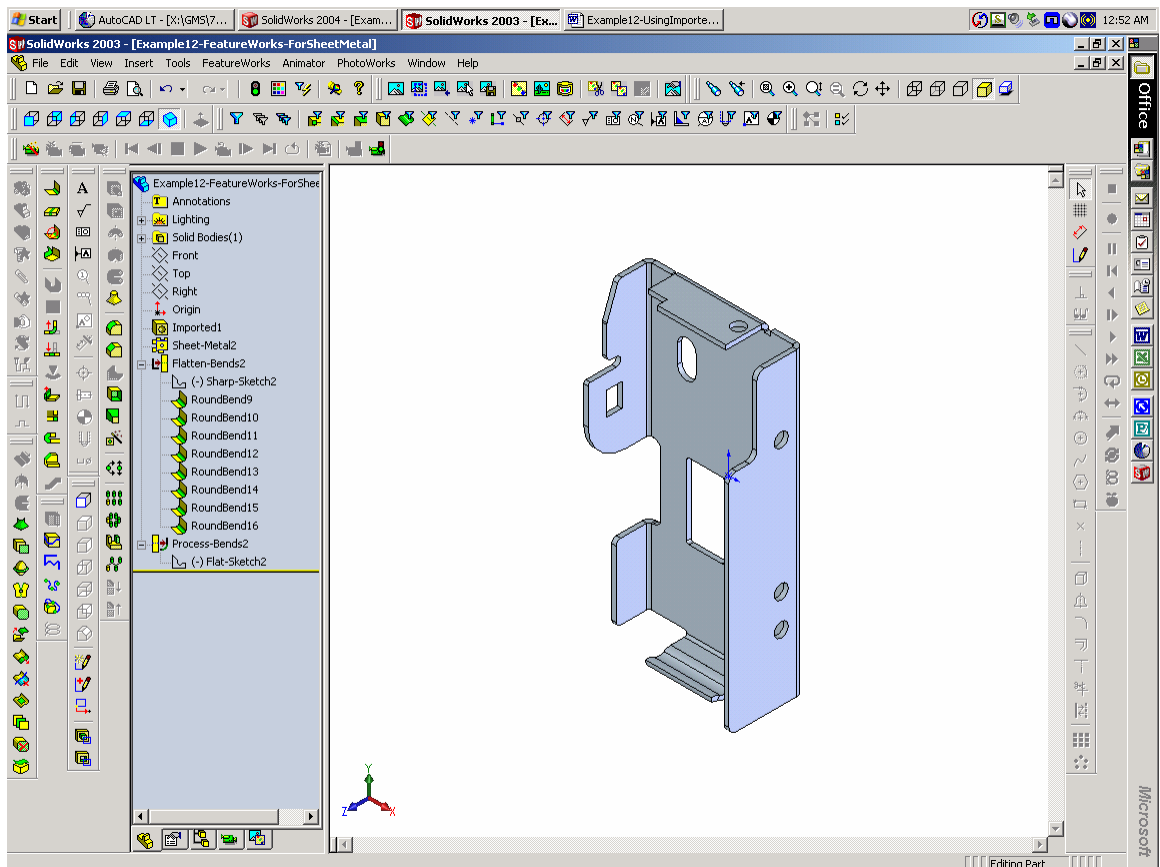
# Using Imported Data With SolidWorks Sheet Metal Functionality

## 3. Insert Sheet Metal

### Insert Sheet Metal After Import.

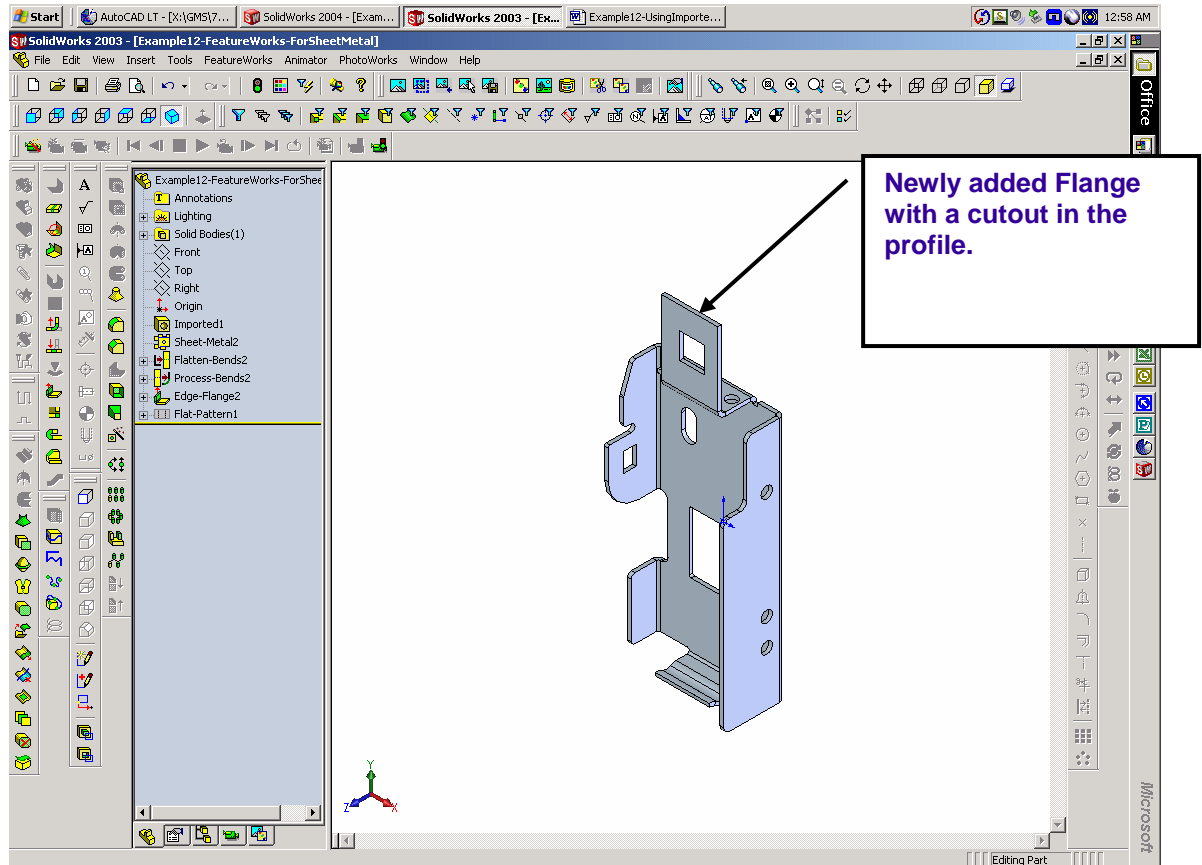
Once your model is imported and SolidWorks has successfully made a solid, insert a sheet metal definition into your model (Insert→Sheet Metal→Bends). The imported model shown below has had a sheet metal definition added.

When a model can be unfolded immediately without any other trouble, it usually indicates that it can be fully “recognized” with FeatureWorks (covered later).



New features may also be added using the sheet metal functionality as normal, but all the geometry that came from the imported body is “frozen”.

## Using Imported Data With SolidWorks Sheet Metal Functionality



### 4. Unfold

#### Many Imported Models Will Unfold “Automatically”.

The unfolding and flat-blank operations may be used as normal. As a caution, make sure that you have checked your “k-factor” or bend allowance to confirm that you have set it to the correct factor – many systems are set for a default .50 k-factor which is generally too large for most bends where the radius is under 2xThickness.

#### Gaining Control Over the Inside Form Radius By “Sharpening Corners”

As a fabricator, you will often receive models that do not have the correct inner form radius. Often times the designer of the part may not be aware of what radius is commonly used, hence you sometimes need to adjust the inner form radii in a model. To fix this, you can sharpen and regenerate the corners (shown below) or you can “fake” the k-factor to make the part unfold correctly (obviously not preferred).

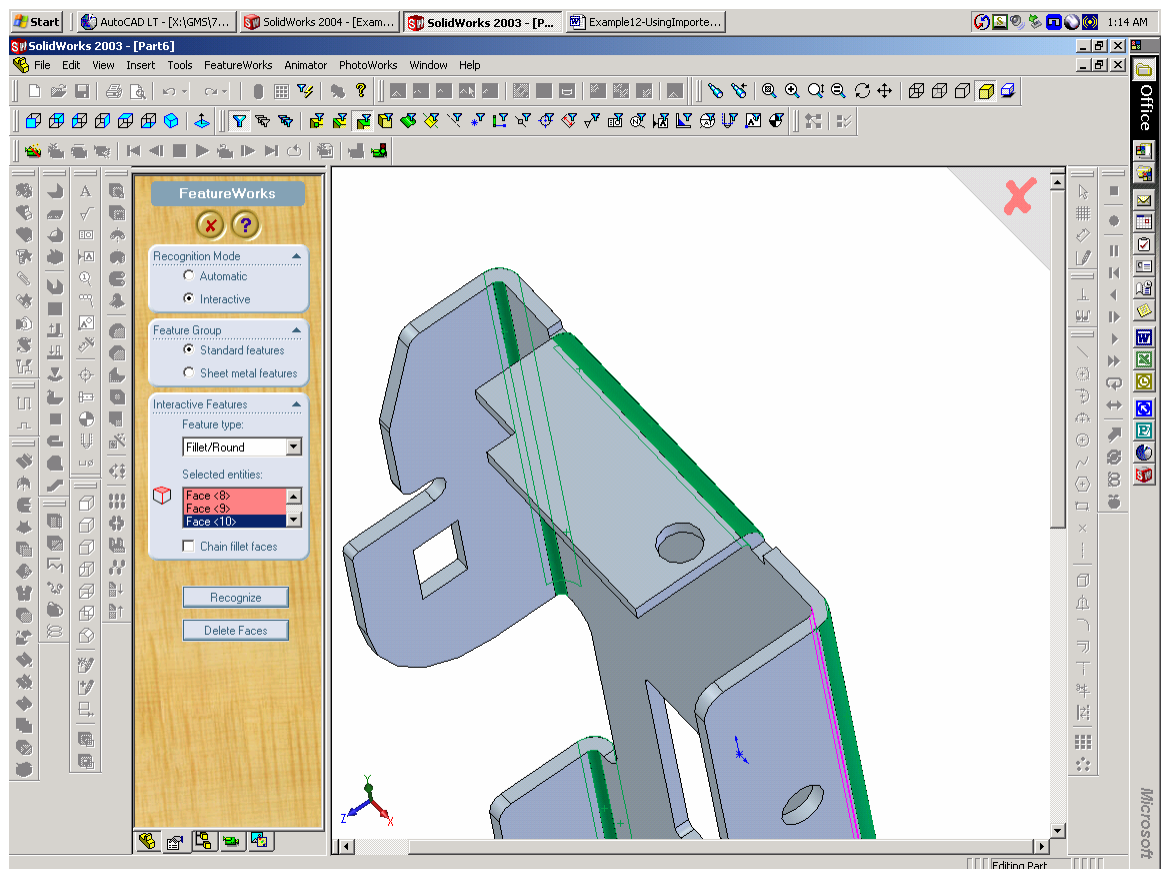
## Using Imported Data With SolidWorks Sheet Metal Functionality

### 5. Using FeatureWorks to “Sharpen” Corners

#### Stripping The Corner Radii Makes Recognition Easier.

If you imported the geometry from a neutral format, the inner form radii are frozen (not changeable), even after a sheet metal definition is successfully added to the model, the radii will not be changeable as they are “pre-existing” features. One way to overcome this is to “sharpen” all the inner and outer radii on the formed features, and let the sheet metal feature (inserted afterwards) re-create the form radii, ideally to the size that you need.

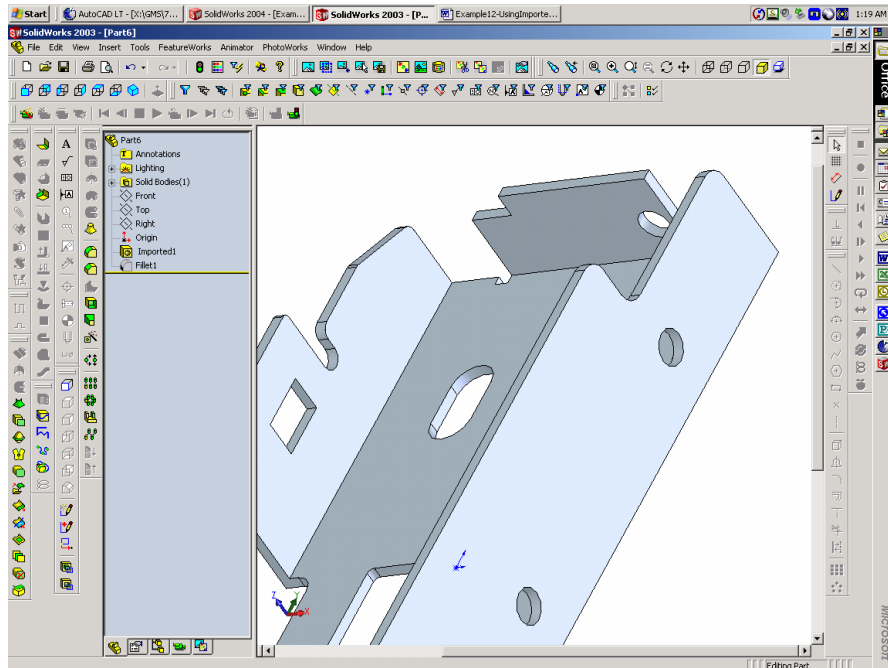
Below is our sample model with the radii “recognized” with feature works. The best results are obtained when using “interactive” recognition of “standard features”(Fillet/Round) to recognize and then delete the fillets.



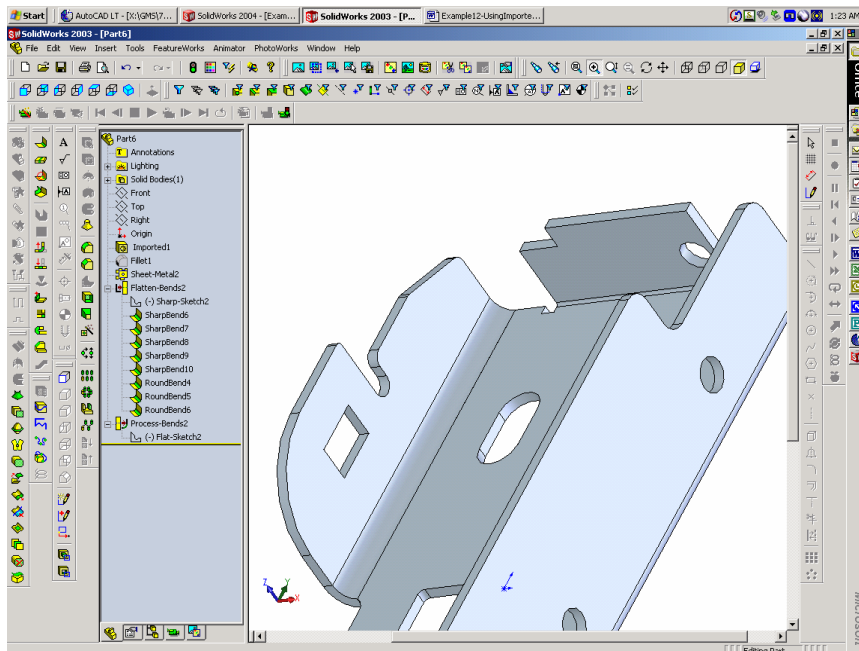
Once the fillets have been mapped, they can be deleted or suppressed. A sheet metal definition can now be inserted and the form radii of the model will be created by the sheet metal feature. The main advantage is that the radii may now be altered by editing them individually or through the global sheet metal definition.



## Using Imported Data With SolidWorks Sheet Metal Functionality



In the model shown below, we have made a dramatically bigger inner form radius on one of the bends and made the others smaller than the original model. This shows how the radii can now be individually manipulated and changed.



## Using Imported Data With SolidWorks Sheet Metal Functionality

### 6. Making the Whole Part Parametric With FeatureWorks – Recognizing Panels

Recognize And Strip The Sharp Corners.

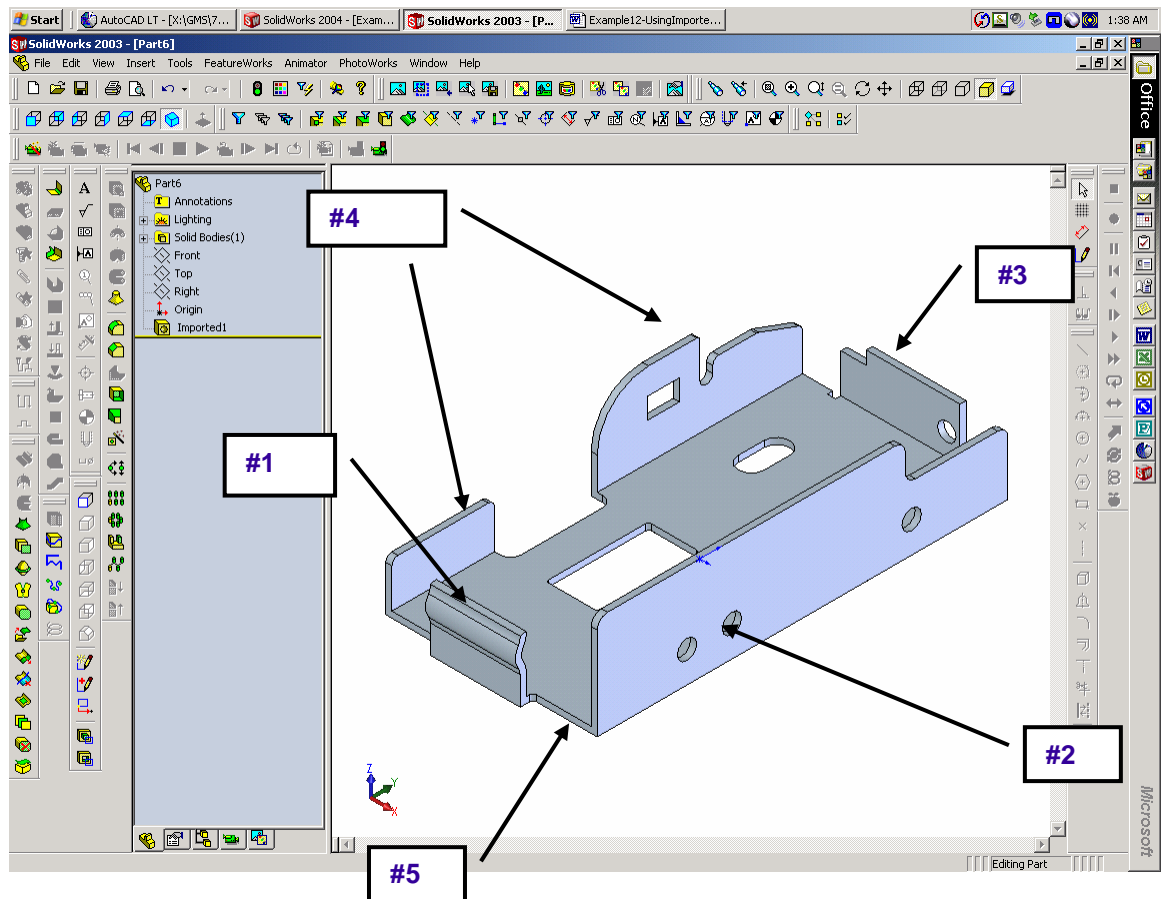
Shown below is a model that has had the corners “sharpened”. Get your model to this state. The model has 5 basic extrusion features.

We will recognize them in the order shown.  
Panel #5 will be recognized last as this can be thought of as the primary panel because it joins all the other panels together.

Analyze The Model For The Best Recognition Sequence.

By recognizing this panel last, it will be the first feature in the feature tree, as FeatureWorks maps feature in a “*first-recognized-last-mapped*” fashion.

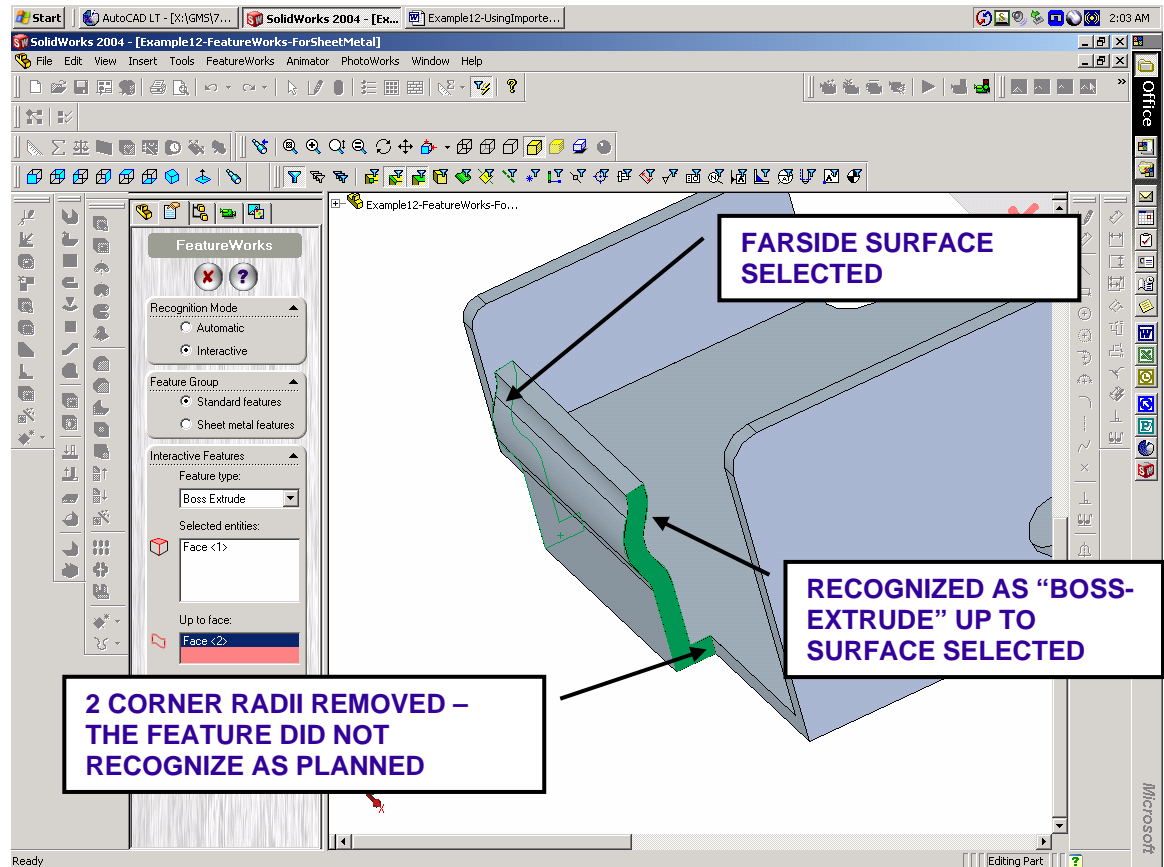
Our example uses the model: [Example1 - PanelMethod](#)



## Using Imported Data With SolidWorks Sheet Metal Functionality

### Recognize Certain Features First.

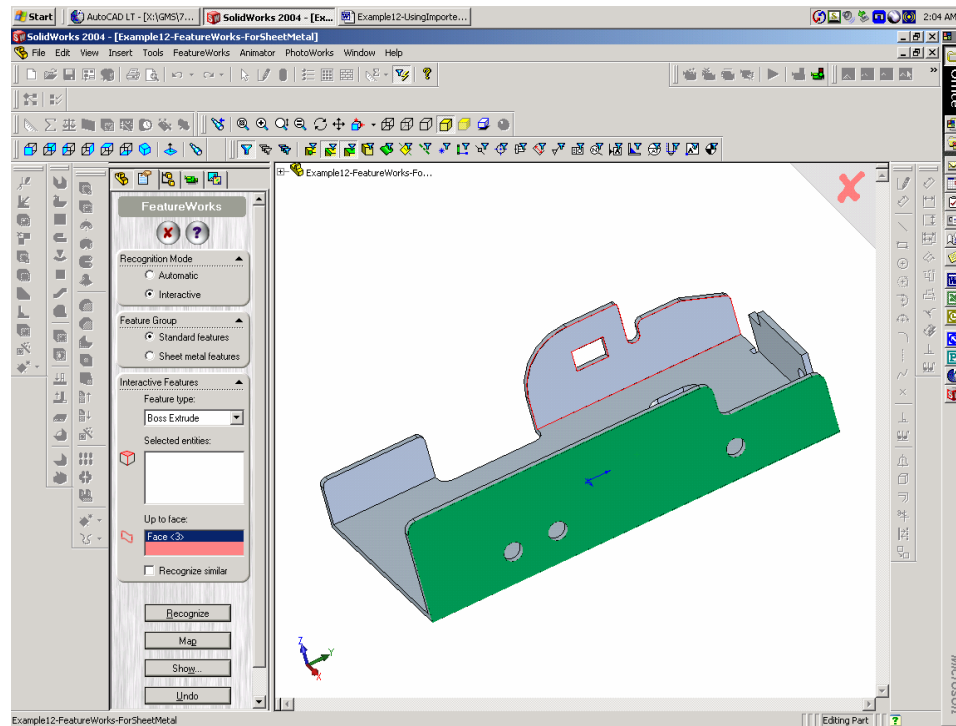
We start by recognizing the end profile by using boss-extrude, selecting the nearside and “up to face” on the far side.



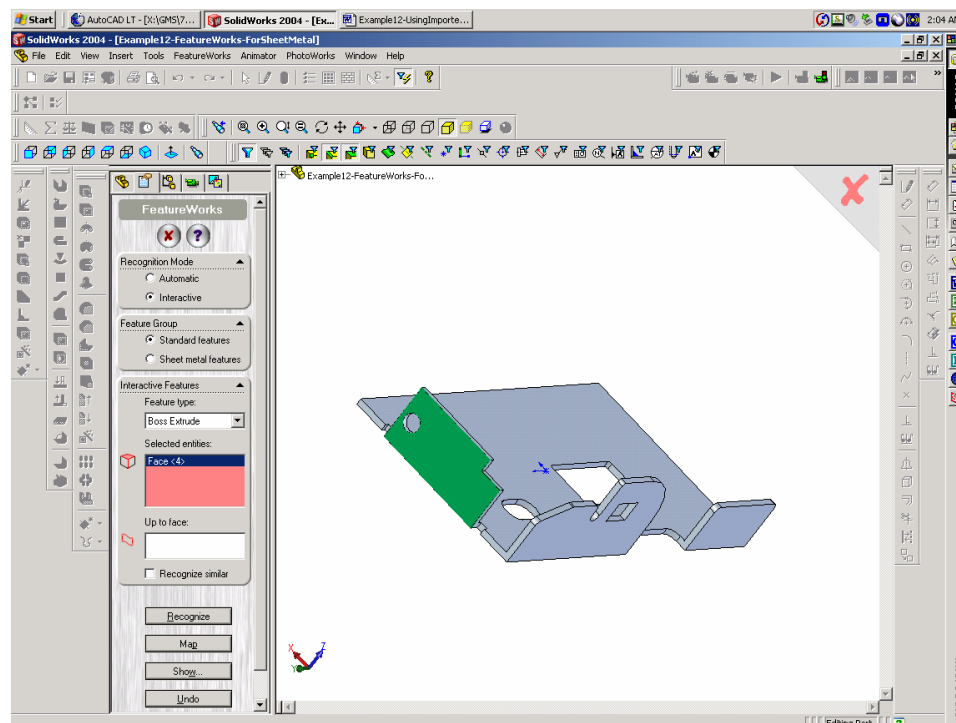
## Using Imported Data With SolidWorks Sheet Metal Functionality

Next, recognize the next panel using “boss-extrude”.

### Recognize the Panels

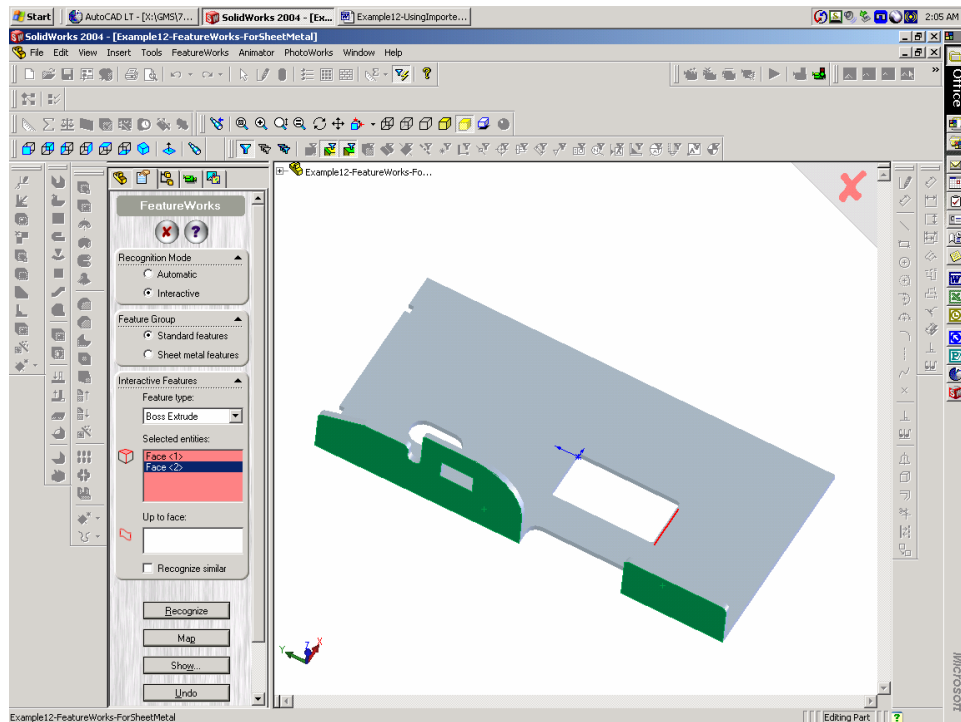


### Next, recognize the next panel

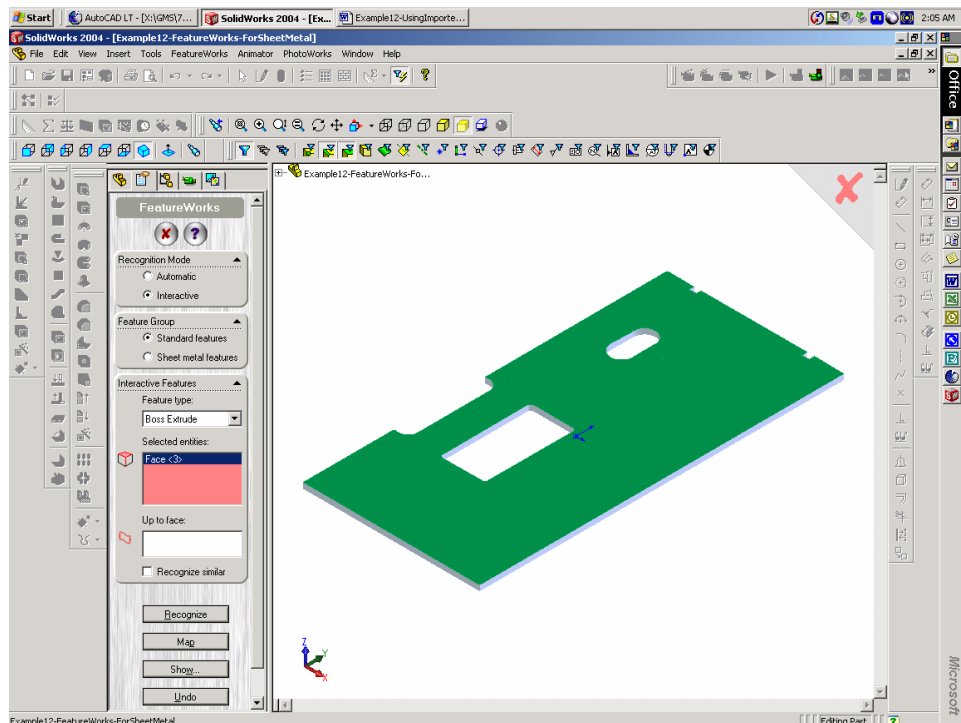


## Using Imported Data With SolidWorks Sheet Metal Functionality

Next, recognize the next panel. Note that the panels can be “multi-profile” as shown below. Since the sheet metal definition will be added later using the model-to-sheet metal, these will unfold nicely.



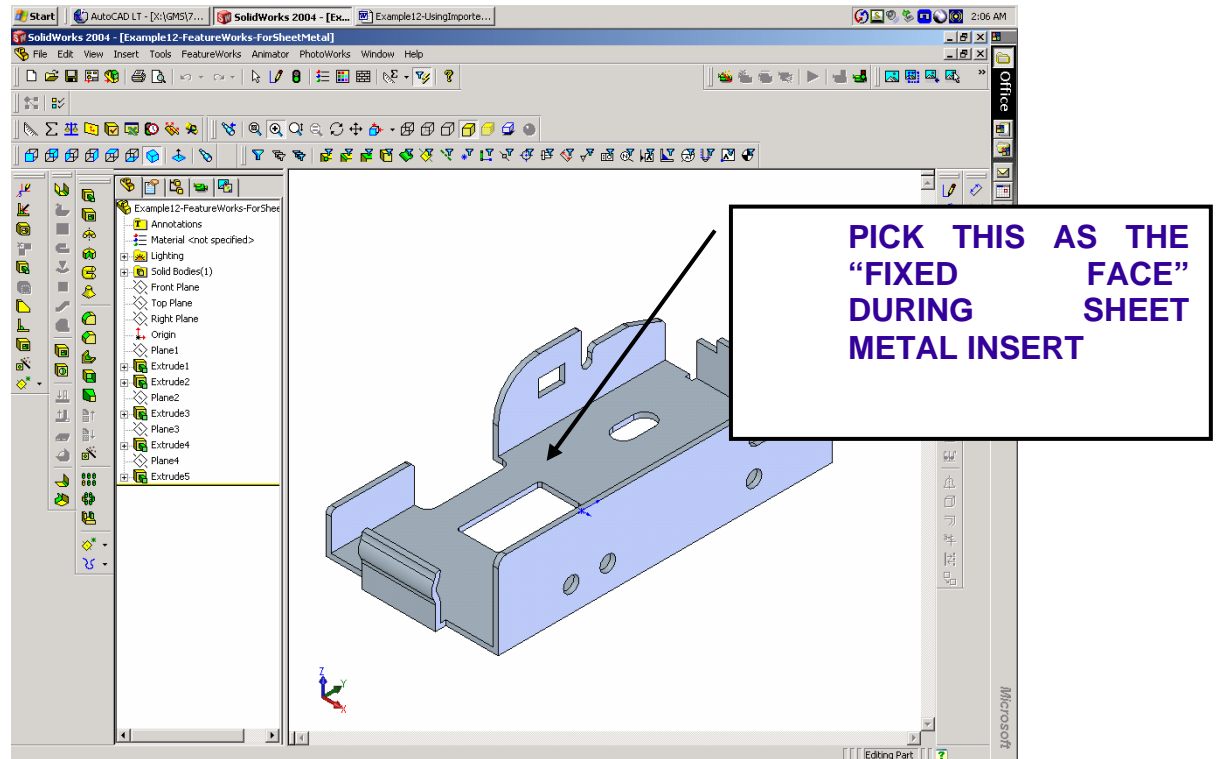
Last, recognize the final panel.



# Using Imported Data With SolidWorks Sheet Metal Functionality

## Map The Features

Next, map your features. The parametric model below will be created.

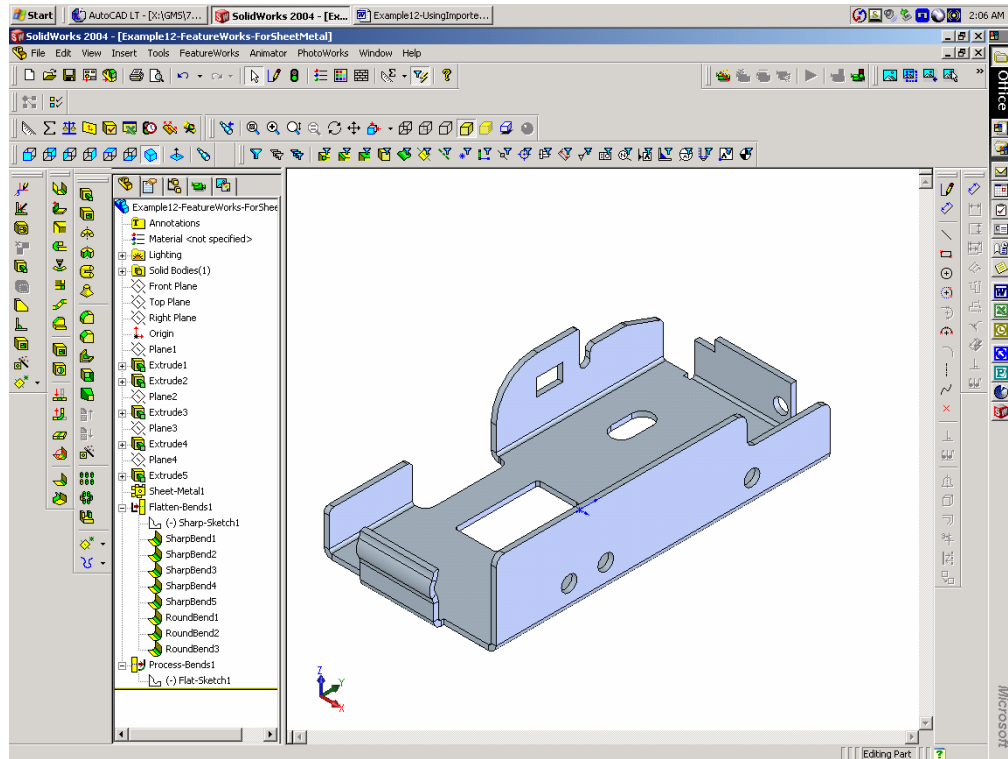


## Insert Sheet Metal

Next, insert a sheet metal definition. Insert→Features→Sheet Metal→Bends

Your part is now fully parametric sheet metal.

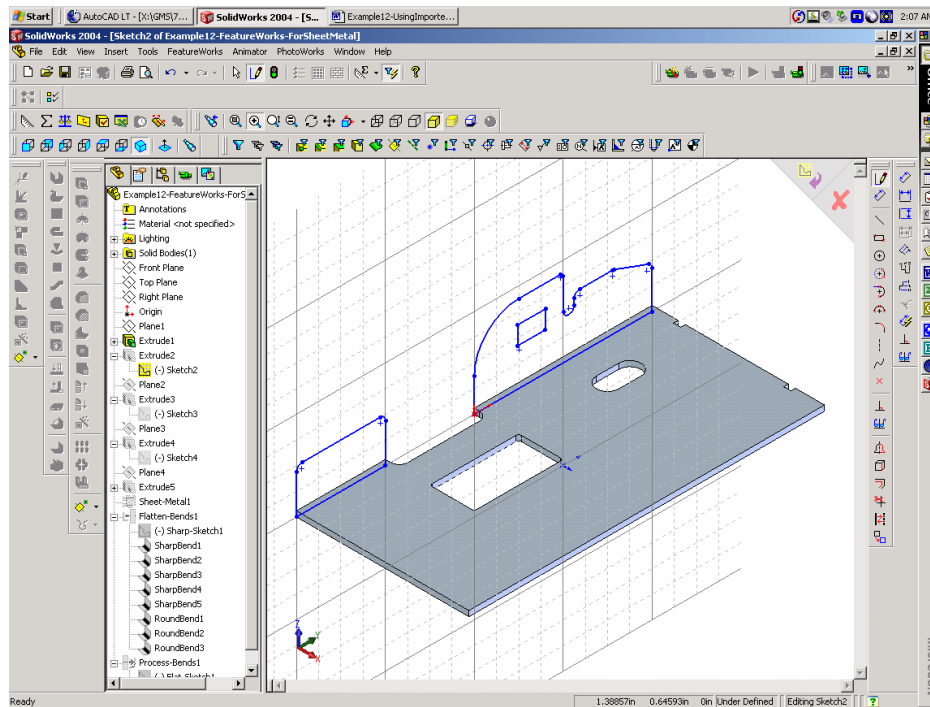
## Using Imported Data With SolidWorks Sheet Metal Functionality



## Using Imported Data With SolidWorks Sheet Metal Functionality

Unfortunately, your panel profiles are not fully constrained. To lock the profiles, you will need to edit the sketch. The easiest way to “lock” the profile is to select all the entities and use the “fix” relationship.

Alternately, you can use the “constrain all” sketch function to create all the relationships that are apparent to the system.

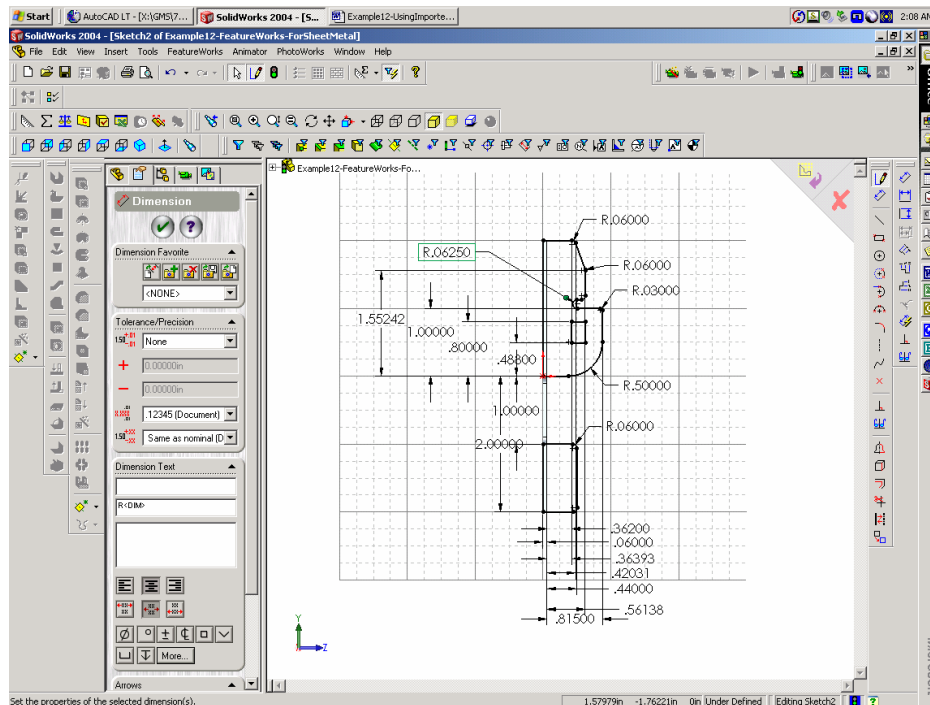




# Using Imported Data With SolidWorks Sheet Metal Functionality

## Auto Dimension

Also, you may use auto dimension to constrain the sketch. How the sketches are constrained depends on how much control you need over the part. This concludes our example.



## 7. PDF Version

A printable PDF version of this information is available [here](#).

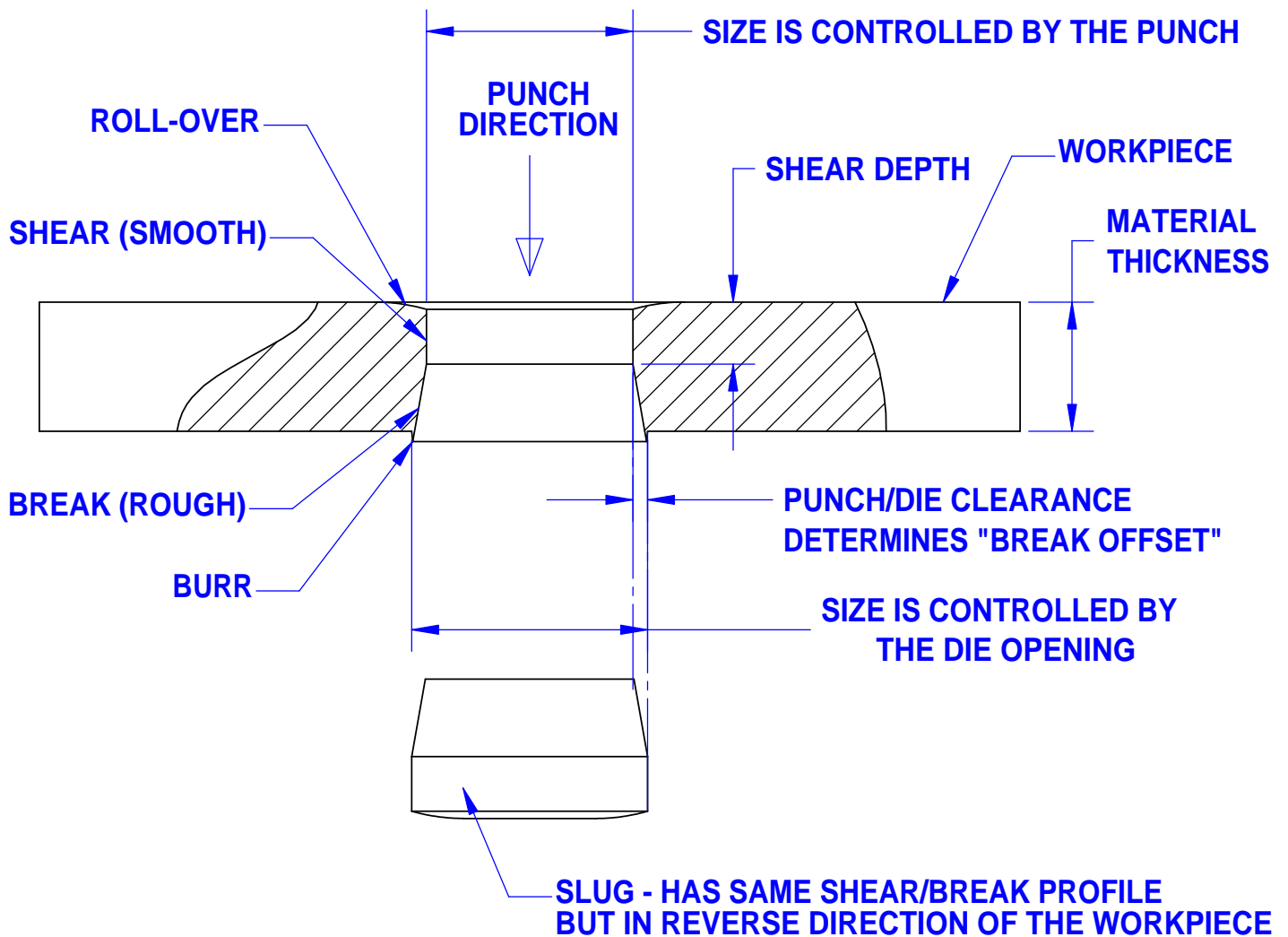
## 8. “Legalese”

All suggestions made here are for training purposes only.

The author is not liable for any loss relating to the misapplication or usage of this information.

You may freely use, reproduce and print this data as long as it remains intact in its original format.

The information presented here may be used for “paid” training purposes.

**KEY CONCEPTS & IDEAS:**

- ROLL-OVER IS THE RESULT OF PART DEFORMATION BEFORE THE PART IS SHEARED.
- SHEAR DEPTH IS TYPICALLY 20% TO 50% OF THE MATERIAL THICKNESS, DEPENDING ON THE MATERIAL HARDNESS AND THE PUNCH/DIE CLEARANCE.
- BURRS ARE TYPICALLY ALLOWED TO BE 10% (MAX) OF STOCK THICKNESS. FOR EXAMPLE, A .060" THICK PART CAN HAVE A .006" BURR.
- PUNCH/DIE CLEARANCE IS TYPICALLY 5%-15% (PER SIDE) OF MATERIAL THICKNESS DEPENDING ON MATERIAL STRENGTH AND THE DESIRED SHEAR DEPTH.

