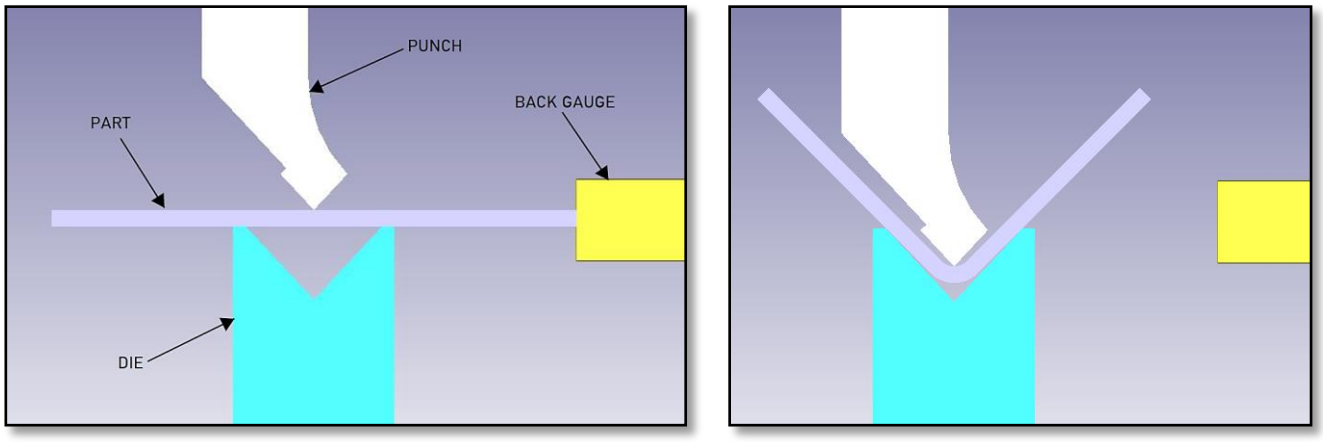
Sheet Metal Bending Guide

At GA Industries we use the air bending process to form flanges on sheet metal parts. Air bending involves placing the part on a V-die and against a back gauge to determine and control the flange length. A punch is then driven down into the part to a specific depth in the V-die. Different angles are achieved by driving the punch to different depths in the V-die, but never to the bottom of the die (hence the 'air' in air bending).



The inside radius (IR) of the bend is generally controlled by the size of the V-die opening. The smaller the die, the smaller the radius. There are several factors that also affect die selection, such as tonnage requirements based on part design.

Tonnage

Due to tooling limitations and machine capacity, there are a limited amount of die openings available for any given thickness.

The tonnage required for a specific bend is based on the following factors:

- Material type (e.g. Stainless steel requires more tonnage than aluminum)
- Material thickness (thicker material requires more tonnage)
- Length of bend (Longer bends require more tonnage)
- V-Die size (Bending in dies with a smaller V opening require more tonnage)

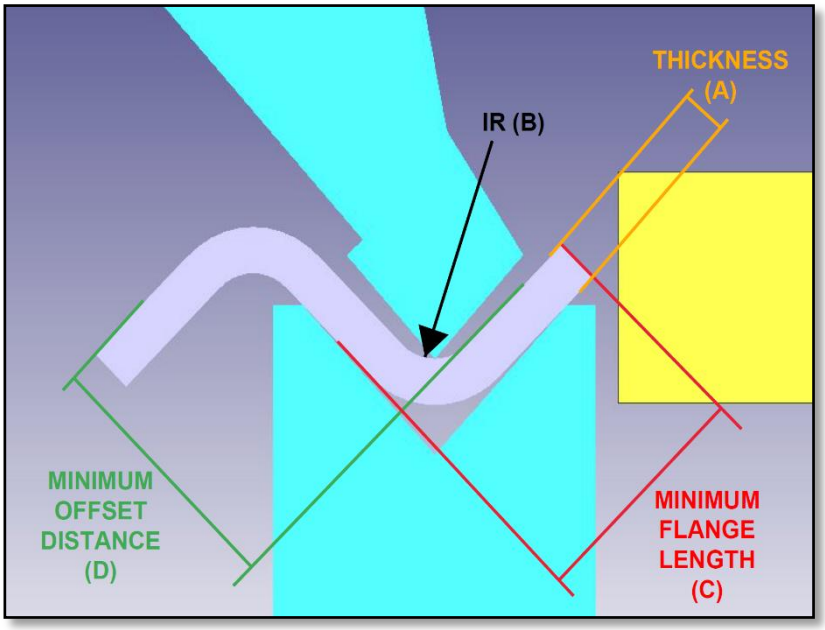
Sheet Metal Bending Guide – Bend Tables

To make designing parts easier, we have summarised our suggested IR for the sheet thicknesses we offer in the following tables. These tables give the necessary information to ensure that part can be fabricated.

Larger inside radii can be achieved for each of the thicknesses in the tables, but keep in mind that this constitutes a larger die which will increase minimum flange lengths. If a part requires a larger IR, please contact GA and we will be glad to assist in the design process.

The following terminology is used in tables 1 and 2:

Thickness (A): Nominal material thickness
IR (B): Inside radius
Maximum Length of Bend: Maximum distance of bend across the part
Minimum Flange Length (C): Minimum length of flange to bend without issues
Minimum Offset Distance (D): Minimum length for additional bends to avoid conflicts with the die
Overbend Max Angle: Maximum angle when bending past 90°
Overbend Max Length: Maximum length of bend when bending past 90°
Tolerance Per Flange: General bend tolerance (per individual bend)



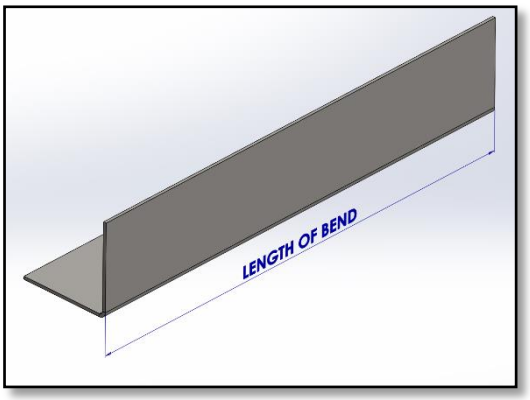


Table 1 – Stainless Steel Bending Guide

STAINLESS STEEL (304,316,430)							
UNITS: INCHES [MM]							
THICKNESS (A)	IR (B)	MAXIMUM LENGTH OF BEND	MINIMUM FLANGE LENGTH (C)	MINIMUM OFFSET DISTANCE (D)	OVERBEND MAX ANGLE	OVERBEND MAX LENGTH	TOLERENCE PER FLANGE
0.032 [0.8]	0.032 [0.8]	65.75 [1670]	0.175 [4.4]	0.340 [8.6]	45°	60 [1524]	±0.005
	0.055 [1.4]	120.00 [3048]	0.225 [5.7]	0.363 [9.2]	35°	33 [832]	
	0.078 [2.0]	75.00 [1905]	0.300 [7.6]	0.376 [9.5]	50°	70 [1778]	
0.036 [0.9]	0.036 [0.9]	65.75 [1670]	0.175 [4.4]	0.348 [8.8]	45°	60 [1524]	
	0.059 [1.5]	120.00 [3048]	0.225 [5.7]	0.371 [9.4]			
	0.070 [1.8]	75.00 [1905]	0.300 [7.6]	0.372 [9.4]	50°	70 [1778]	
0.048 [1.2]	0.048 [1.2]	120.00 [3048]	0.225 [5.7]	0.372 [9.4]	35°	33 [832]	
	0.100 [2.5]	75.00 [1905]	0.350 [8.9]	0.443 [11.3]			
	0.150 [3.8]	75.00 [1905]	0.400 [10.2]	0.533 [13.5]			
0.063 [1.6]	0.063 [1.6]	75.00 [1905]	0.300 [7.6]	0.392 [10.0]	50°	70 [1778]	±0.010
	0.078 [2.0]	120.00 [3048]	0.350 [8.9]	0.436 [11.1]			
	0.115 [2.9]	120.00 [3048]	0.550 [14.0]	0.611 [15.5]	55°	100 [2540]	
0.078 [2.0]	0.078 [2.0]	120.00 [3048]	0.335 [8.5]	0.451 [11.5]	35°	33 [826]	
	0.100 [2.5]	75.00 [1905]	0.400 [10.2]	0.513 [13.0]			
	0.125 [3.2]	120.00 [3048]	0.550 [14.0]	0.636 [16.2]	55°	100 [2540]	
0.105 [2.7]	0.100 [2.5]	75.00 [1905]	0.450 [11.4]	0.599 [15.2]			
	0.130 [3.3]	120.00 [3048]	0.550 [14.0]	0.668 [17.0]			
	0.150 [3.8]	120.00 [3048]	0.575 [14.6]	0.735 [18.7]			
0.125 [3.2]	0.125 [3.2]	75.00 [1905]	0.550 [14.0]	0.683 [17.3]	55°	80 [2032]	±0.015
	0.156 [4.0]	120.00 [3048]	0.575 [14.6]	0.761 [19.3]			
	0.172 [4.4]	120.00 [3048]	0.700 [17.8]	0.878 [22.3]			
0.135 [3.4]	0.135 [3.4]	100.00 [2540]	0.575 [14.6]	0.750 [19.1]			
	0.156 [4.0]	120.00 [3048]	0.700 [17.8]	0.872 [22.1]	50°	100 [2540]	
	0.188 [4.8]	120.00 [3048]	0.700 [17.8]	0.904 [22.9]			
0.188 [4.8]	0.188 [4.8]	65.50 [1664]	0.875 [22.2]	1.557 [39.5]			
	0.266 [6.8]	65.50 [1664]	0.875 [22.2]	1.635 [41.5]			
	0.313 [8.0]	115.00 [2921]	1.400 [35.6]	1.682 [42.7]			
0.250 [6.4]	0.250 [6.4]	63.50 [1613]	1.125 [28.6]	1.681 [42.7]			±0.025
	0.300 [7.6]	83.00 [2108]	1.400 [35.6]	1.731 [44.0]			
	0.406 [10.3]	74.00 [1880]	1.400 [35.6]	1.837 [46.7]			
0.313 [8.0]	0.438 [11.1]	32.75 [832]	1.800 [45.7]	2.326 [59.1]			±0.032
0.375 [9.5]	0.625 [15.9]	16.25 [413]	2.250 [57.2]	3.245 [82.4]			
	0.750 [19.1]	32.75 [832]	2.800 [71.1]	3.290 [83.6]			
	1.000 [25.4]	32.75 [832]	2.800 [71.1]	3.540 [89.9]			

Table 2 – Aluminum Bending Guide

ALUMINUM (5052)							
UNITS: INCHES [MM]							
THICKNESS (A)	IR (B)	MAXIMUM LENGTH OF BEND	MINIMUM FLANGE LENGTH (C)	MINIMUM OFFSET DISTANCE (D)	OVERBEND MAX ANGLE	OVERBEND MAX LENGTH	TOLERENCE PER FLANGE
0.032 [0.8]	0.030 [0.8]	120.0 [3048]	0.225 [5.7]	0.338 [8.6]	35°	32.75 [832]	±0.005
0.040 [1.0]	0.030 [0.8]	120.0 [3048]	0.225 [5.7]	0.346 [8.8]	35°	32.75 [832]	
	0.035 [0.9]	75.0 [1905]	0.300 [7.6]	0.341 [8.7]	50°	70.00 [1778]	
	0.094 [2.4]	75.0 [1905]	0.400 [10.2]	0.469 [11.9]			
0.050 [1.3]	0.030 [0.8]	65.5 [1664]	0.175 [4.4]	0.356 [9.0]	45°	60.00 [1524]	
	0.032 [0.8]	120.0 [3048]	0.225 [5.7]	0.358 [9.1]			
	0.050 [1.3]	120.0 [3048]	0.300 [7.6]	0.366 [9.3]	50°	100.00 [2540]	
0.060 [1.5]	0.030 [0.8]	120.0 [3048]	0.225 [5.7]	0.366 [9.3]	35°	32.75 [832]	±0.010
	0.060 [1.5]	75.0 [1905]	0.225 [5.7]	0.396 [10.0]			
	0.080 [2.0]	75.0 [1905]	0.300 [7.6]	0.406 [10.3]			
0.080 [2.0]	0.031 [0.8]	120.0 [3048]	0.225 [5.7]	0.387 [9.8]			
	0.047 [1.2]	75.0 [1905]	0.350 [8.9]	0.422 [10.7]	35°	32.50 [826]	
	0.063 [1.6]	75.0 [1905]	0.350 [8.9]	0.438 [11.1]			
	0.125 [3.2]	120.0 [3048]	0.550 [14.0]	0.638 [16.2]			
0.090 [2.3]	0.065 [1.7]	75.0 [1905]	0.350 [8.9]	0.450 [11.4]	35°	32.50 [826]	
	0.090 [2.3]	75.0 [1905]	0.400 [10.2]	0.515 [13.1]			
0.100 [2.5]	0.063 [1.6]	75.0 [1905]	0.350 [8.9]	0.458 [11.6]			
	0.090 [2.3]	75.0 [1905]	0.400 [10.2]	0.525 [13.3]			
	0.095 [2.4]	120.0 [3048]	0.550 [14.0]	0.628 [16.0]	55°	100.00 [2540]	
0.125 [3.2]	0.109 [2.8]	75.0 [1905]	0.450 [11.4]	0.628 [15.9]			±0.015
	0.125 [3.2]	120.0 [3048]	0.550 [14.0]	0.683 [17.3]			
	0.250 [6.4]	80.0 [2032]	0.575 [14.6]	0.855 [21.7]			
0.157 [4.0]	0.125 [3.2]	120.0 [3048]	0.550 [14.0]	0.715 [18.2]			
0.188 [4.8]	0.090 [2.3]	75.0 [1905]	0.700 [17.8]	0.859 [21.8]			
	0.125 [3.2]	65.5 [1664]	0.875 [22.2]	1.494 [37.9]			
	0.157 [4.0]	95.0 [2413]	1.125 [28.6]	1.526 [38.8]			
	0.250 [6.4]	80.0 [2032]	1.125 [28.6]	1.619 [41.1]			
0.250 [6.4]	0.195 [5.0]	65.5 [1664]	0.875 [22.2]	1.626 [41.3]			±0.025
	0.250 [6.4]	65.5 [1664]	0.875 [22.2]	1.681 [42.7]			

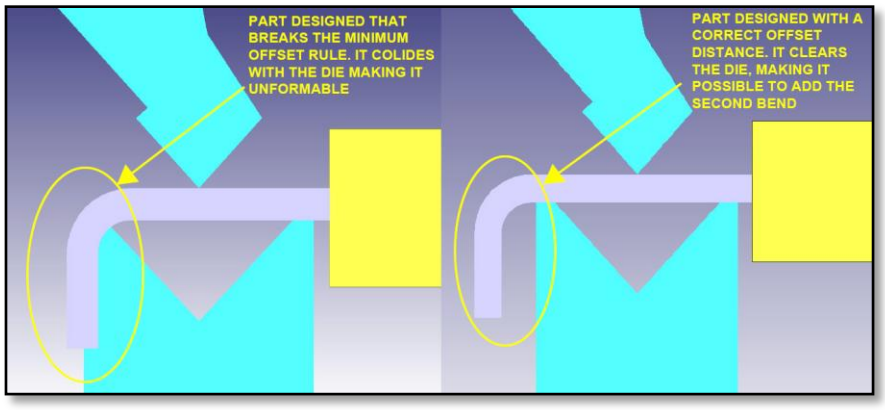
Sheet Metal Bending Guide – Design and Limitations

Minimum Flange Length

The whole flange must be able to reach both sides of the die to be bent. Breaking this rule will cause deformation and unbent sections, which in turn will affect the consistency and accuracy of the part. Cut-outs (holes, slots, etc..) on the flange must also be positioned at least the minimum flange length away from the bend or they will also deform. See page 10 for more information.

Minimum Offset Distance

Depending on the size of the die used, you are limited on the length of offset available. PICTURES HAS SPELLING MISTAKE 'COLLIDES'

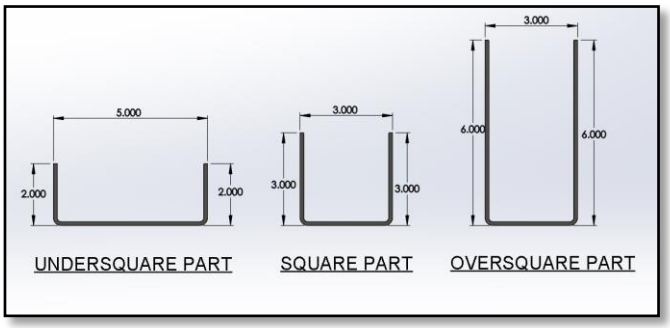


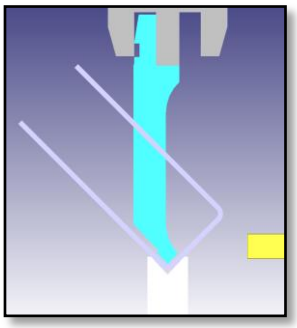
Bend Tolerancing

Bending sheet metal causes a controlled and uncontrolled side when it comes to tolerancing. Details about this can be found on pages ___ The tolerances provided in the tables above indicate the tightest tolerance range *per flange* for each thickness of material.

Oversquare Parts

Return flanges are limited based on the punch tooling size and shape. In general, you do not want to design square & oversquare parts. The image below shows how an oversquare part collides with the punch.





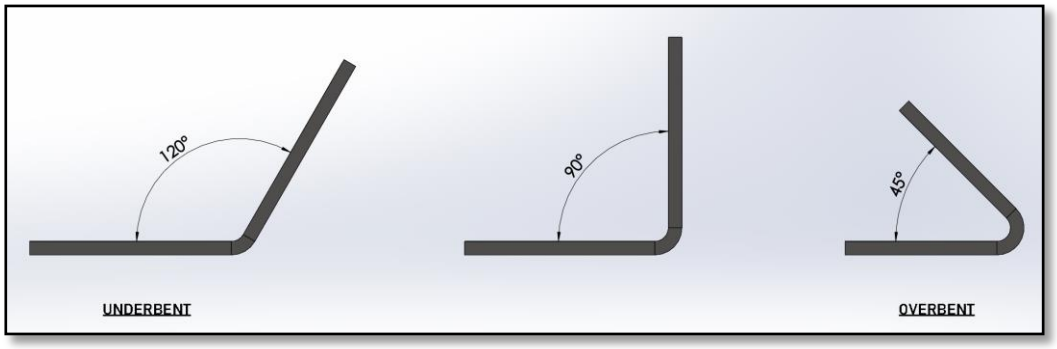
We carry gooseneck punches that allow for certain oversquare profiles. Generally, most oversquare profiles under 3" by 3" are achievable.

Sheet Metal Bending Guide – Bending Features

GA Industries offer multiple types of bending features, including;

OVERBENDING

Overbending is when a bend goes past 90°. Overbending is more restrictive in the tooling options available which will limit the IRs available. Also, keep in mind the minimum flange length for overbends is larger because the punch needs to go deeper into the die to achieve the required angle. See the tables on pages ____ for the available options for overbending.

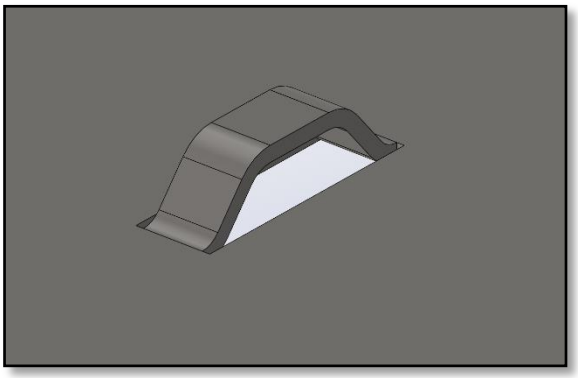


HEMS

We offer hems on parts up to 0.063" thick stainless (thicker stainless requires tonnage that is over machine capacities) and 0.048" aluminum (thicker aluminum cracks). Hems are excellent for avoiding sharp edges when necessary, such as wire protection or to increase durability due to the thicker edge. On thicker parts we recommend edge rounding instead. Keep in mind hems are not easily controlled and should not be given a tight tolerance, a ±0.025" tolerance is suggested.

BRIDGE LANCES

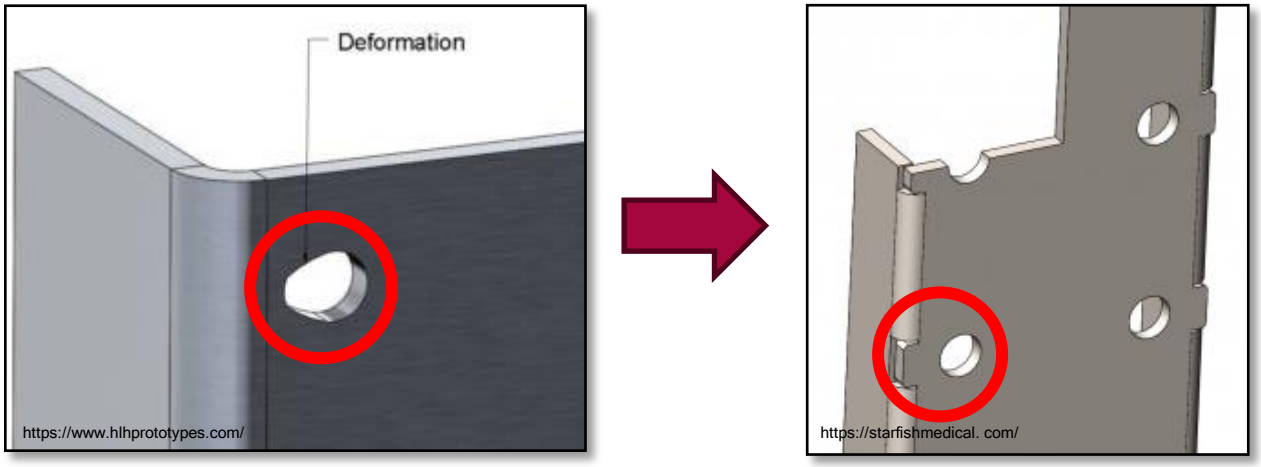
At GA we can add lance features to your sheet metal products. These features can be used as locators, card guides, dividers, ventilation, and wire tie downs. They do require custom tooling that we design and make in house. The limitations and details change from one application to the next. If you need a specific lance feature on your parts, please feel free to contact us and we will be glad to assist with achieving your design.



Sheet Metal Bending Guide – Design and Limiations Cont'd.

Features too close to bends

Holes/Slots and other features that are in close proximity to a bend may become distorted. Splitting the bend (as shown in right image below) will avoid this issue – if a smooth radius is required, the split section can be welded and finished to match the bend radius. Other options include reaming out holes after forming, changing the radius or feature location, etc.



Drawing Tolerance Rounding

As per example below, the rounding of the drawing already takes the part out of spec as the actual model is designed to 7.005". If we fabricated to exactly 7.005", we would already be out of tolerance based on the drawing (with -.00" lower tolerance). Please keep in mind rounding errors when using tolerances; symmetrical tolerancing is preferred. More details on the mating component will assist with passing quality control.