TECHNICAL SOLUTIONS
GUIDE
PUNCH PRESS
ENGLISH

MATE PRECISION TECHNOLOGIES
RESPECT • SUPPORT • INSPIRE

Technical Solution Guide

PUNCHING AND FORMING SHEET METAL IN A PUNCH PRESS ENVIRONMENT

Headquartered in Anoka, Minnesota, in a 300,000 sq. ft. (28,000m2) state-of-the-art facility.

OVER FIVE DECADES OF EXCELLENCE

Founded in 1962, Mate is a world-class manufacturer of superior solutions for the metal cutting and metal forming industries. We manufacture workholding systems, CNC punch press tooling, and offer a complete line of press brake tooling and laser consumables. Mate products and services are available worldwide, fully supported by more than 80 dealers in every industrialized country.

PERSONAL, RESPECTFUL RELATIONSHIPS

Mate does business with people, not companies. Our connection to you is personal. Mate's team of manufacturing and metalworking professionals knows what you go through. We know what it's like to compete for that next job, manage deadlines or even need a rescue. With Mate you have a partner that respects your knowledge and is dedicated to helping you succeed.

YOUR GO-TO SOURCE

Serving our customers is at the core of who we are. In your plant or on the phone, we're up for whatever metalworking challenges you face. Your Mate representatives are experts who know from experience what happens on the shop floor and provide our legendary in-field support. They speak your language, fully capable of helping you improve processes and solve problems. Mate customer service is ready to assist with fast quotes, guiding your order on to our top-notch machinists and shipping pros.

GET INSPIRED!

With our vast knowledge and broad product range we inspire innovative thinking. Our customer's projects can be seen around the world: from unique building façades thought to be impossible to make, to a new way to add strength to thin material. The possibilities are endless, so think big, bold and beyond.

WE'VE GOT YOU COVERED

Dedicated to quality in every aspect of our business, Mate offers an extensive standard product line that can be delivered with same day or next day service. All Mate products are backed with our industry leading 100% customer satisfaction guarantee.

MATE'S MISSION AND PROMISE TO YOU:

Mate's mission is to personally **Respect, Support** and **Inspire** metalworking professionals around the world with high-quality products and services for factory productivity.

[Dimensions in Inches (mm)]

TABLE OF CONTENTS

[Dimensions in Inches (mm)]

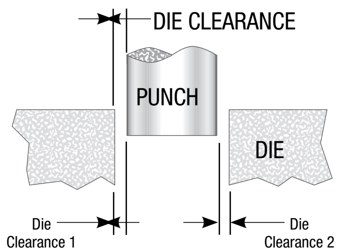
SUGGESTED FORMING APPLICATIONS BY INDUSTRY

	IF YOUR BUSINESS IS IN OR SERVES THIS INDUSTRY/ CONSIDER THESE POTENTIAL APPLICATIONS	INDUSTRY: AGRICULTURE OR HEAVY EQUIPMENT	APPLIANCES	ARCHITECTUAL	AUTOMOTIVE	AVIATION	ELECTRICAL ENCLOSURES	ELECTRONICS	HVAC	LIGHTING	OFFICE FURNITURE
FORMING APPLICATIONS	BEADING TOOL		•					•	•		•
	BRIDGE LANCE & FORM TOOLING					•		•			•
	CARD GUIDES							•			
	CLUSTER TOOLS	•	•	•	•	•		•		•	•
	COLD FORGED EMBOSS (LOGOS)										•
	CORNER LANCE (BALLAST RETAINER)									•	
	COUNTERSINK TOOLS	•	•			•		•			•
	EASYBEND™		•	•	•	•	•	•	•	•	•
	EASYMARK™	•		•	•	•	•	•	•		•
	EASYSNAP™		•	•	•	•	•	•	•	•	•
	EASYTAP™						•		•		
	ELECTRICAL KNOCKOUT TOOLS		•	•		•	•	•	•	•	
	EMBOSS		•	•		•	•				
	EXTRUSION TOOL							•	•		
	FISHTAIL/BI-TRAPEZOIDAL SHAPE	•	•			•		•	•	•	•
	FLANGE FORMING TOOLS		•			•			•	•	
	FULLY GUIDED TOOLING	•	•	•	•	•			•	•	•
	GROUND SYMBOL STAMP					•		•	•	•	
	HEAVY DUTY TOOLING	•		•							
	HEXLOCK™	•						•			•
	LARGE KNOCKOUT TOOLS						•				
	LOUVERS	•	•		•	•	•	•		•	
	MAXIMA™ COATING	•	•	•	•	•			•		•
	NITRIDE TREATMENT	•									
	NON-MARKING APPLICATIONS			•		•					
	PIERCE & EXTRUDE TOOLING								•		
	POWERMAX™								•		
	ROLLERBALL DEBURR™	•	•		•	•	•	•	•		•
	ROLLERBALL™		•		•	•	•	•	•	•	•
	SAFETY EDGE CURL		•			•		•	•		•
	SCREW THREAD EXTRUSION		•							•	•
	SHEETMARKER™		•	•		•			•		•
	SLUG FREE LIGHT™ DIES									•	•
	SNAPLOCK™	•	•	•	•		•	•		•	•
	SPECIAL FORMING TOOLS	•		•			•				
	SPECIAL SHAPES	•	•	•				•			
	SUPERMAX™ COATING	•	•	•	•	•			•		•
	THREADFORM/HYBRID THREADFORM		•			•				•	•
	ULTRA LIGHT™ SPRING PADS				•	•				•	•
	URETHANE PADS				•	•				•	
	V-LINE STAMP (LOGOS)	•	•		•						•

[Dimensions in Inches (mm)]

WHAT IS DIE CLEARANCE?

Die clearance is equal to the space between punch and die when the punch enters the die opening.

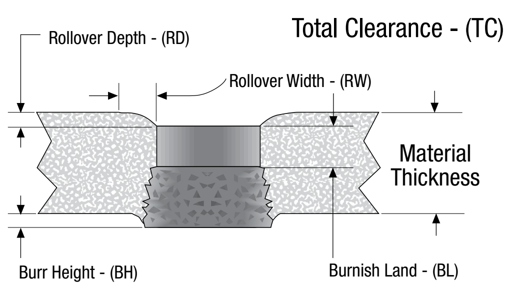


Total Die Clearance = Die Clearance on both sides of punch

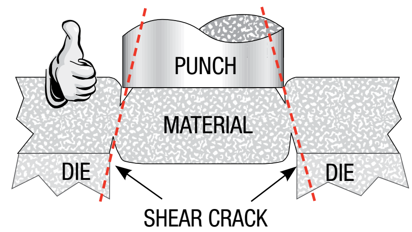
Total Die Clearance = Die Clearance 1 + Die Clearance 2

Regardless of sheet thickness, the recommended penetration of the punch into a Slug Free® die is 0.118(3.00).

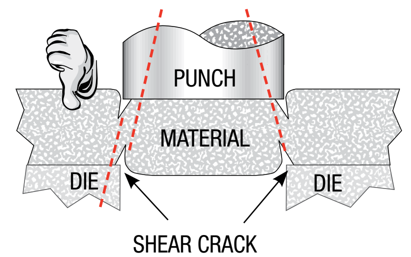
ANATOMY OF A PUNCHED HOLE



WHY USE PROPER DIE CLEARANCE?



PROPER CLEARANCE —
shear cracks join, balancing punching force, piece part quality, and tool life.



CLEARANCE TOO SMALL —
secondary shear cracks are created, raising punching force, and shortening tool life.

Blanking Tools are used to punch out a small part down the slug chute.		Piercing	Blanking
Material Type (Typical Shear Strength)	Material Thickness (T)	Total Die Clearance (% of T)	Total Die Clearance (% of T)
Aluminum 25,000 psi (0.172 kN/mm²)	Less than 0.098(2.50)	15%	15%
	0.098(2.50) to 0.197(5.00)	20%	15%
	Greater than 0.197(5.00)	25%	20%
Mild Steel 50,000 psi (0.344 kN/mm²)	Less than 0.118(3.00)	20%	15%
	0.118(3.00) to 0.237(6.00)	25%	20%
	Greater than 0.237(6.00)	30%	20%
Stainless Steel 75,000 psi (0.517 kN/mm²)	Less than 0.059(1.50)	20%	15%
	0.059(1.50) to 0.110(2.80)	25%	20%
	0.110(2.80)to 0.157(4.00)	30%	20%
	Greater than 0.157(4.00)	35%	25%

[Dimensions in Inches (mm)]

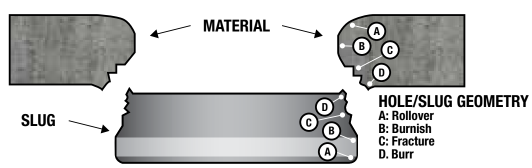
DIE CLEARANCE

WHAT YOUR SLUGS TELL YOU

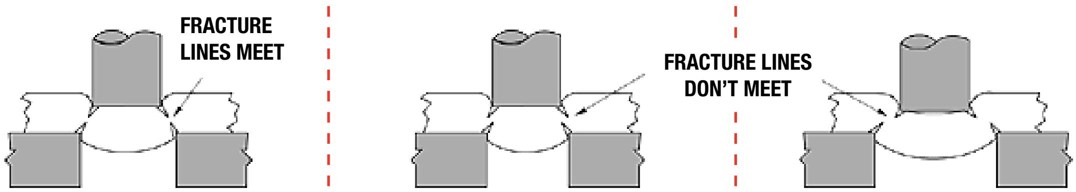
The slug is essentially a mirror image of the hole with the same parts in reverse order. By examining your slugs you can tell if the punch-to-die clearance is correct; if tool angularity is correct; or tooling is dull.

An **ideal slug** is created when the fracture planes coming from the top and bottom of the material have the same angle and form in alignment with each other. This keeps punching force to a minimum and forms a clean hole with few burrs. When clearance is proper, tool life is extended. **(pic 1)**

If the **clearance is too large,** the slug will show a rough fracture plane (C) and a small burnish zone (B). The larger the clearance, the greater the angle between the fracture plane (C) and the burnish zone (B). Excess clearance makes a hole with large rollover (A) and fracture (C) so that the profile is somewhat pointed with a thin burr (D). When clearance is too large, tool life is reduced. **(pic 2)**



If **clearance is too small**, the slug will show a fracture plane (C) with little angle, and a large burnish zone (B). Inadequate clearance makes a hole with small rollover (A) and steep fracture (C) so that the profile is more or less perpendicular to the surface of the material. When clearance is too small, tool life is reduced. **(pic 3)**



PROPER CLEARANCE (pic 1)

CLEARANCE TOO TIGHT (pic 2)

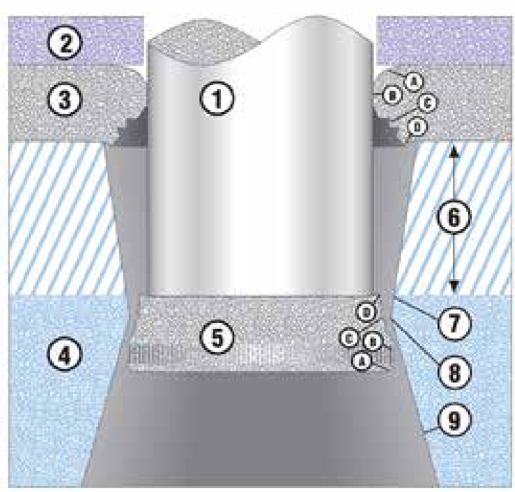
CLEARANCE TOO LARGE (pic 3)

ANATOMY OF A PUNCHED HOLE

1. Punch
2. Stripper
3. Material
4. Slug Free Die
5. Slug
6. Grind Life
7. Entry — Constricting taper
8. Pressure Point
9. Exit — Relief Taper

HOLE SLUG GEOMETRY

A. Rollover
B. Burnish
C. Fracture
D. Burr



[Dimensions in Inches (mm)]

DIE CLEARANCE

BENEFITS OF PROPER DIE CLEARANCE:

- Longer tool life
- Better stripping
- Smaller average burr height
- Smaller average burr thickness
- Cleaner, more uniform holes
- Little or no shavings
- Reduced galling
- Flatter work pieces
- More accurate hole locations
- Lowest force required to pierce the material
- Quieter punching

INSUFFICIENT DIE CLEARANCE:

- Galling
- Work piece shavings
- Shortened tool life
- Slow/erratic stripping
- Poor hole quality
- Excessive heat
- Warped sheets
- Smaller initial burr
- Larger, thicker burr
- Reduced rollover
- Reduced break-away area
- Reduced slug pulling
- Work hardened burrs

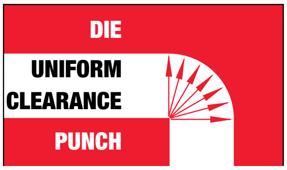
EXCESSIVE DIE CLEARANCE:

- Increased slug pulling
- Work piece shavings
- Poor hole quality
- Increased work piece distortion
- Increased burr
- Increased rollover
- Increased breakaway area
- Rounded slugs
- Work hardened burrs

"CLEARANCE CORNERS" IN DIES CONTROL CORNER BURRS

Including a radius in the corners of rectangular and square dies with clearance greater than 0.020(0.5) keeps clearance uniform around the corner of the punch.

If the die is sharp cornered too, then the distance between punch and die corners would be greater than side clearance, resulting in larger burrs. To get clearance corners always order dies as "punch size plus clearance."



[Ex. 1.000 + 0.037(25.4+0.9)].

WATCH THE VIDEO:

Watch an animation about punching holes with optimal die clearance:

https://youtu.be/85YRdhxbACI

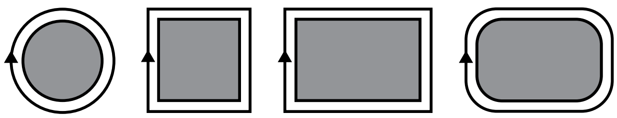
[Dimensions in Inches (mm)]

CALCULATING PUNCHING FORCE

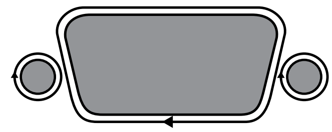
TONNAGE FORMULA

TONNAGE = Punch Perimeter x Material Thickness x Material Tonnage Value x Material Multipler

PUNCH PERIMETER

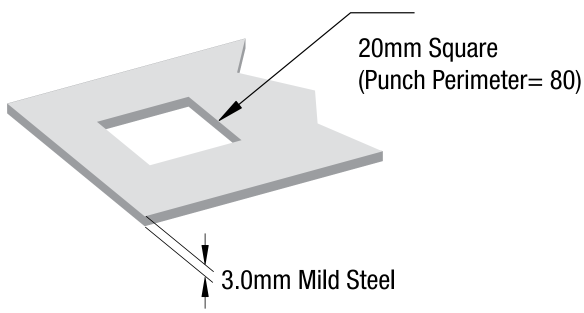


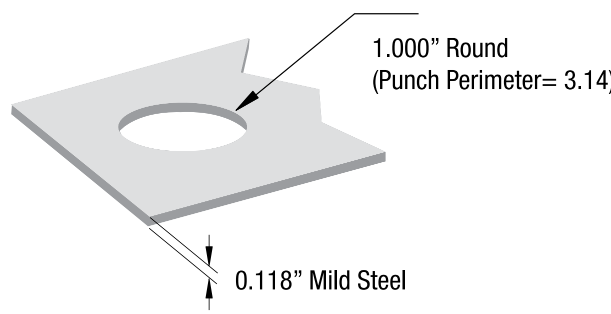
Perimeter is simply the linear distance around a punch of any shape. For a round punch, this would be the circumference.



For a cluster punch, the perimeter would be the sum of the linear distances of each of the punch components.

EXAMPLE OF TONNAGE CALCULATION





Metric Example:
Metric Tonnage for a 20mm square in 3.0mm Mild Steel
Tonnage = 80 x 3.0 x 0.0352 x 1.0 = 8.45 Metric Tons

Inch Example:
Imperial Tonnage for a 1.000" round in 0.118" Mild Steel
Tonnage = 3.14 x 0.118 x 25 x 1.0 = 9.27 Imperial Tons

MATERIAL MULTIPLIER

MATERIAL TYPE	MATERIAL MULTIPLIER
Aluminum (soft sheet)	**0.3**
Aluminum (1/2 hard)	**0.4**
Aluminum (full hard)	**0.5**
Copper (rolled)	**0.6**
Brass (soft sheet)	**0.6**
Brass (1/2 hard)	**0.7**
Mild Steel	**1.0**
Stainless Steel	**1.6**

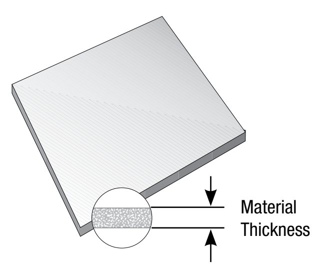
MATERIAL SHEAR STRENGTH — Material shear strength is a measure of maximum internal stress before a given material begins to shear. This property is determined by metallurgical science and expressed as a numerical factor. Popular materials have approximate shear strengths of:

MATERIAL :	SHEAR STRENGTH — psi/in²(kN/mm²):
Aluminum	25000(0.1724)
Brass	35000(0.2413)
Mild Steel	50000(0.3447)
Stainless	75000(0.517)

MATERIAL TONNAGE VALUE

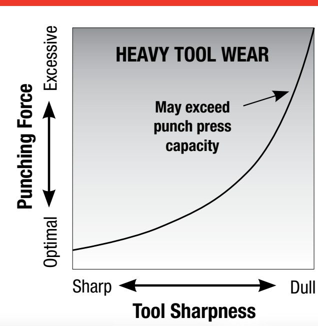
Metric (Metric Tons/mm²)	Inch (Imperial Tons/in²)
0.0352	25

MATERIAL THICKNESS



Material thickness is the amount the punch must penetrate the workpiece or sheet in making a hole. Generally the thicker the material the more difficult it is to punch.

PUNCHING FORCE CHANGES AS TOOLS DULL



[Dimensions in Inches (mm)]

CALCULATING PUNCHING FORCE

ABOUT PUNCH SHEAR

Punch "shear" is the geometry of the punch face. Shear helps reduce tonnage because the punch is not hitting with the full face on the material.

PUNCH WITHOUT SHEAR

FORMULA:
Punch perimeter in (inches)mm x material thickness in (inches)mm x material shear strength in lbs/in2(kN/mm2) =
Punching force in lbs(kN)

CONVERSION FACTORS:
To convert to Imperial Tons: divide lbs by 2000
To convert to Metric Tons: divide kN by 9.81

EXAMPLE OF PUNCHING FORCE PROBLEM –

Using 20.0mm square punch into 3.0mm mild steel:
Punch perimeter is 80.0mm,
Material thickness is 3.0mm,
Material shear strength is 0.3447 kN/mm2.
80.0mm x 3.0mm x 0.3447 kN/mm2 = 82.7 kN

PUNCH WITH SHEAR

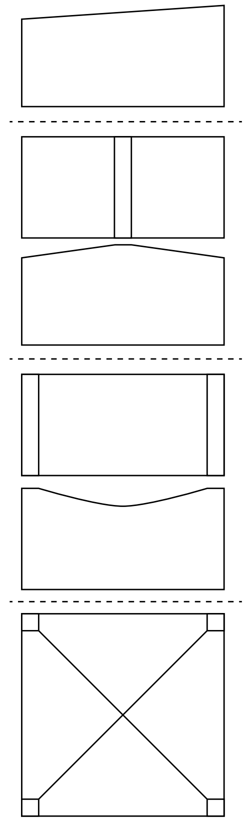
FORMULA:
Punch perimeter in inches(mm) x material thickness in inches(mm) x material shear strength in lbs/in2(kN/mm2) x SHEAR FACTOR (see chart below) = **Punching force in lbs(kN)**

CONVERSION FACTORS:
To convert to Imperial Tons: divide lbs by 2000
To convert to Metric Tons: divide kN by 9.81

PUNCHES WITH SHEAR – CONSIDERATION:
Punch shear tends to lessen punching force. The degree to which this happens is the SHEAR FACTOR. Shear factor does change as the punch becomes less sharp.

ADVANTAGES OF SHEAR

- Tonnage reduction
- Noise reduction
- Slug control
- Reduced shock loads
- Improved stripping



COMMON TYPES OF SHEAR
Rooftop shear is the best shear for minimizing tonnage in thicker materials.
Concave shear is an alternative shear for nibbling (without shear is recommended for nibbling).
One-way shear is best for minimizing tonnage when blanking.

SHEAR FACTORS FOR MATERIAL .050"(1.2MM) TO .250"(6.4MM) FOR PUNCHES WITH SHEAR

Material Thickness	.050"(1.2mm)	.060"(1.5mm)	.075"(1.9mm)	.105"(2.7mm)	.120"(3.0mm)
Shear Depth: .060"(1.5mm)	.50	.50	.58	.72	.75
Material Thickness	.135"(3.4mm)	.165"(4.2mm)	.190"(4.8mm)	.250"(6.4mm)	
Shear Depth: .060"(1.5mm)	.78	.83	.86	.90	

EXAMPLE: Formula for punching with shear (20.0mm punch)
80.0mm x 3.0mm x 0.3447 kN/mm2 x .75 = 62.0 kN

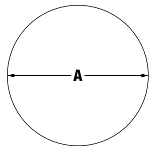
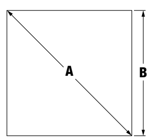
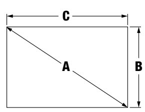
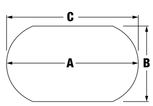
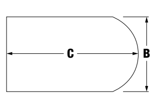
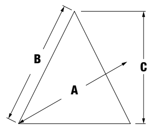
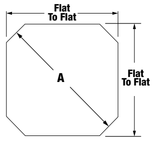
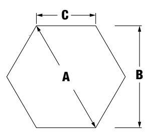
NOTE: Mate **does not recommend** using shear to bring punching force within press capacity because dull tool edges quickly raises punching force — resulting in exceeded press capacity.

[Dimensions in Inches (mm)]

SHEARS

CALCULATING PUNCHING FORCE

DIMENSION CHART FOR TONNAGE CALCULATION

Shape		"A" Dimension Dictates Station Size	"L" Dimension Outside Perimeter
Round		A = Diameter	L= 3.14 x A
Square		A = B x 1.414	L = 4 x B
Rectangle		A = $\sqrt{(B^2 + C^2)}$	L=2x(B+C)
Oval		A=C	L 2C + 1.14B
Rect / Oval		A= $\sqrt{(B^2 + C^2)}$	L = 2C + 1.57B
Equilateral Triangle		A= 1.334 x C	L = 3 x B
Quad D		A = Diameter	L (approx.) 3.14 x A
Hexagon		A = 1.155 x B	L = 3 x A

[Dimensions in Inches (mm)]

PUNCH & DIE MAINTENANCE

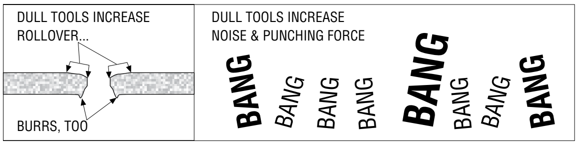
FACTORS AFFECTING TOOL WEAR	
HOLE SIZE	Small punches will wear faster than larger punches.
HOLE CONFIGURATION	Sharp corners will show wear much more quickly than straight or curved edges, particularly on punches. Narrow sections will wear faster than heavier sections.
SHEAR FACE ON PUNCH	The portion of the punch that strikes first does most of the work and will therefore, wear faster.
CLEARANCE	Proper clearance will yield longer tool life.
PUNCHING CONDITIONS	Reducing the hitting shock and holding the sheet flat allows the punch to cut clearly and will give better life to the punch.
STRIPPING CONDITIONS	Stripping the work piece from the punch evenly contributes to easy stripping.
TURRET ALIGNMENT	Mechanical damage of punches and dies is often misinterpreted as wear. Tight clearance on one side of a punch and die will accelerate wear at that point. It is critical to regularly check turret alignment in order to prevent problems such as unacceptable part quality and turret wear.
TOOL MATERIAL	To develop toughness and hardness required for long life, high speed steel (HSS) undergoes several heat treatments. The punches are double tempered to C62 Rockwell hardness. Hardened (59 Rockwell C) tool steel is the optimum material for dies. It correctly balances the need for maximum edge wear without breakage.
PUNCHING SPEED	High punching speeds can, under certain conditions, generate enough frictional heat to soften a punch. A softer punch will wear much faster.
LUBRICATION	A lubricant will increase tool life significantly.
WORK PIECE THICKNESS	Thicker material will cause faster punch wear.
WORK PIECE PROPERTIES	Physical and mechanical properties of the work piece will greatly affect tool life.
PUNCH/DIE WEAR	Punches will generally wear faster than dies. Dies are less affected by the factors described above than are punches.

PUNCH & DIE MAINTENANCE

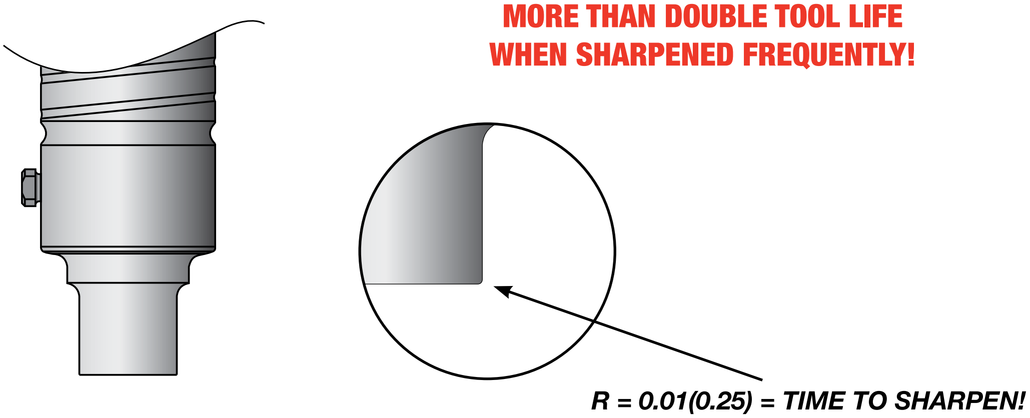
WHEN TO SHARPEN TOOLS

If a piece-part is starting to show too much roll over, the punch press is making more noise or it's working harder than it used to — perhaps a tool is dull.

Mate recommends sharpening tools when the edges are worn to 0.01 (0.25) radius. At this point, just a small amount of sharpening will "touch up" the cutting edge. Frequent touch up works better than waiting for the punch to become very dull. Tools last longer and cut cleaner with less punching force, and you get improved consistency in work quality. The maximum amount of sharpening depends on the thickness of material being punched, size of punch (length and width) and punch press station.

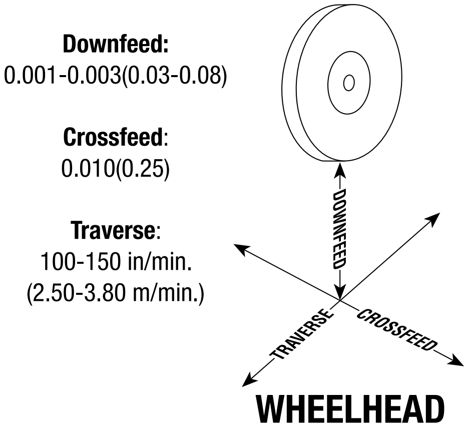


MORE THAN DOUBLE TOOL LIFE WHEN SHARPENED FREQUENTLY!



HOW TO SHARPEN TOOLS

1. To sharpen, clamp the punch squarely in a V-Block on the magnetic chuck of a surface grinder. Only 0.001 to 0.002(0.03 to 0.05) should be removed in one "pass". Repeat until the tool is sharp, normally 0.005-0.010(0.13-0.25)total.
2. Use a standard vitrified bond, aluminum oxide wheel: hardness range "D" to "J"; grain size 46 to 60. A "ROSE" wheel made especially for grinding high speed steel is a good choice but not mandatory.
3. Dress the wheel using a rigid single or multi-point diamond: downfeed 0.0002-0.0008 (0.005-0.020); crossfeed quickly 20-30 in/min (508-762 mm/min).
4. Apply coolant with as much force and as close to the tool and wheel as is practical. Use a good general purpose grinding coolant as recommended by the manufacturer.
5. Feeds and feed rates:
   a. Downfeed (wheelhead), 0.001-0.003 (0.03-0.08)
   b. Crossfeed (infeed), 0.005-0.010 (0.13-0.25); for nitride punches, 0.002-0.007(0.05-0.18)
   c. Traverse (sideways), 100-150 in/min (2,540-3,810 mm/min)
6. After sharpening, lightly stone the sharp cutting edges to remove any grinding burrs and leave a 0.001-0.002 (0.03-0.05) radius. This reduces the risk of chipping. Do not use a file of any kind.
7. Demagnetize the punch and spray on a light oil to prevent corrosion.



[Dimensions in Inches (mm)]

PUNCH & DIE MAINTENANCE

FIXING SHARPENING PROBLEMS

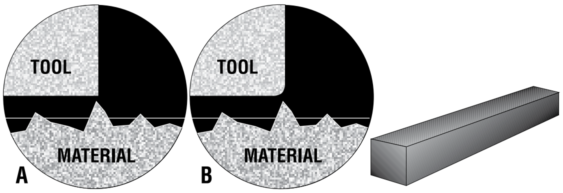
PROBLEM:	CAUSE:	CURE:
Discoloration** and/or surface cracks and/or "fish scale"	Insufficient coolant	Increase or redirect flow.
	Improper wheel	Use coarser grain, softer grade grinding wheel.
	Improper dress	Drop wheelhead 0.0002-0.0004 (0.005-0.010) and redress. Move crossfeed approx. 50 in/min. (1.25 m/min.)
Harsh cutting sound and/or poor surface finish	Excessive stock removal	Less downfeed; lower crossfeed rate
	Improper wheel	Use coarser grain, softer grade grinding wheel.
	Improper dress or glazed wheel	Redress wheel, break glaze on wheel surface

**Dark discoloration indicates damage not necessarily limited to the tool surface. Removal of the burned surface will not rectify damage. Mate recommends replacement of the tool if it is in this condition.

KEEP PUNCHES SHARP LONGER

Sharpened Tool Edges Stay Sharp Longer If Edges Are Dressed

The microscopic irregularities in workpieces and tools attack each other with each impact. Protrusions and sharp corners get flattened and knocked off. With very sharp tool edges (A), microscopic flakes are knocked off as the tool wears. Each flake leaves a rough surface vulnerable to additional flaking. For this reason, we recommend lightly dressing the edge of freshly sharpened punches (B) with an oil stone (India Oil Stone ST029807). This removes the corner, which is most vulnerable to breaking off. Although the radius is tiny, it strengthens the tool edge by distributing stresses that cause flaking.



With a radius of just 0.001-0.002(0.03-0.05), the tool can still be considered very sharp, and it stays that way longer. This small radius is applied with only one light pass of the stone per edge. You can't see the radius, but it's there!

PUNCH & DIE MAINTENANCE

GRINDING CONSIDERATIONS

A grinding wheel's abrasive particles, in effect, are break-away "teeth". These teeth can be made from a variety of very hard, abrasion resistant materials, such as diamond, borozon and, most commonly, aluminum oxide.

The abrasive particles are embedded in a softer matrix material and meant to fracture loose from the matrix as cutting pressure becomes greater. Cutting pressure can increase from raising the feed rate or dulling of abrasive particles. Pressure causes surface particles to fracture or break free from the wheel matrix and expose new sharp edges, resulting in the wheel's sharpness.

In selecting a vitrified bond aluminum oxide wheel, we need only be concerned with two variables: hardness and coarseness of the wheel. Hardness refers to the bond strength of the matrix. Coarseness refers to the size and concentration of the abrasive particles (grit).

Generally speaking, harder materials require softer wheels; softer materials require harder wheels. Grinding a harder and/or more abrasive resistant material, such as hardened tool steel, dulls abrasive particles quickly. The wheel then needs increased feed forces. A softer wheel allows spent particles to break loose from the matrix more easily. The newly exposed sharp edges will cut rather than rub and tear at the workpiece. Less pressure is required and the wheel runs cooler.

Coarse wheels with large, widely spaced abrasive particles perform less cutting per revolution and allow greater "chip" clearance. The wheel stays cleaner. Friction is reduced.

Balancing hardness and coarseness results in a wheel that stays sharp and clean to optimize cutting action. It meets the grinding objective of removing material from the workpiece while expending a minimal amount of wheel energy. Wheel energy losses largely translate to workpiece heating. Workpiece heating, in turn, will result in softened and/or highly stressed tools which will not perform well. Hardened tool steels are particularly vulnerable.

It is generally desirable to use a softer "G" or "H" hardness wheel with a grit concentration/size of about forty-six.

A-2 and S-7 STEEL
Grinding Wheel Hardness: **G-J**
Grit: **46-60**

M-2 and M4PM™ STEEL
Grinding Wheel Hardness: **D-G**
Grit: **46-60**

MPM82 STEEL
Grinding Wheel Hardness: **G-J**
Grit: **46-60**

DURASTEEL STEEL
Grinding Wheel Hardness: **G-J**
Grit: **46-60**

[Dimensions in Inches (mm)]

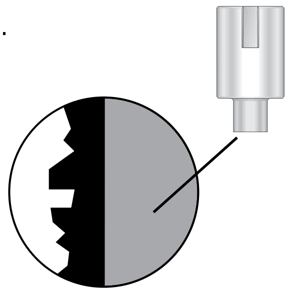
PUNCH & DIE MAINTENANCE

PUNCH BECOMES DULL TOO FAST

If punches become dull too fast, the clearance may be too tight. TOTAL clearance should be 20-25% of material thickness (not per side). In partial hitting (notching, nibbling, shearing), lateral forces may deflect the punch tip and tighten clearance on one side. Sometimes the punch tip may move far enough to shave the side of the die. This results in rapid deterioration of both punch and die.

PREVENT GALLING

Galling is an adhesion to the punch tip of metal being punched, caused by pressure and heat. The best technique for removing galling is to rub it off with a fine stone (ST029911). The rubbing should be done parallel to the direction of the punching motion. This will polish the surface then contacts the material, decreasing the chance of any future galling. Do not sandblast, belt sand or use other harsh abrasive methods; doing so creates a coarse surface finish making it easier for material to adhere to the tool.



If galling is a problem, order punches with 2° total back taper. Examine a Mate punch closely and you'll find that the punch tip is largest at the cutting edge. This is because we normally build in 1/4° total back taper (1/8° per side). This minute change in size facilitates stripping and material is much less likely to adhere to the punch. Grind life is not affected. The reduction in diameter is so small that the punch remains within normal tolerances for both hole size and die clearance throughout its life. Most likely you will not notice the back taper when examining your punches. If galling is especially troublesome, order a combination of 2° total back taper (1° per side) on the punch and 20% to 30% clearance for the die.

Mate Eliminator Lubrication Pads

Designed for thick turret punch assemblies, Mate Eliminator™ punch tip lubrication pads assist in keeping the punch tip lubricated during the punching process.

Studies have shown that properly lubricated punch tips help extend tool life and keep the punch from overheating. In many situations, lubrication helps eliminate unwanted galling during the punching process.

Mate Eliminator lubrication pads are easy to install, especially on Mate Ultra TEC® A and B stations. Simply use the punch and stripper to "punch" the hole into the foam. Saturate the pad with 46-68 ISO viscosity hydraulic oil and you're ready for gall-free punching.

- Made from polyether filter foam
- Available in A through E stations
- Compatible with all thick turret punch presses

PREVENT PUNCH OVERHEATING

A. Use a lubricant. This will decrease friction.
B. If lubricant is unacceptable or if slug pulling occurs: Use more than one punch of the same size in the sequence. By rotating the punches, there will be a longer time for each punch to cool down before it is used again.
C. Simply give the tool a rest. Plan the program so that the tool that is overheating alternates with different punches, or stop the press for awhile.
D. Use Mate Eliminator Lubrication Pads.

[Dimensions in Inches (mm)]

PUNCH & DIE MAINTENANCE

TOOL ALIGNMENT

If alignment of your punch press deteriorates to the point where tools dull too rapidly or workpiece quality is unacceptable, here are things you can check and fix:

A. Examine tool loading equipment for wear and damage. Adjust where possible. Replace where necessary. Clean and lubricate linkages.

B. Examine tool receptacles. Clean so that tools seat accurately and rigidly. Restore damaged or worn components. Check keys and keyways for proper clearance.

C. Keep your machine manual handy – refer to it regularly.

DIE MAINTENANCE

As with punches, keep dies clean and watch for wear. Use the same sharpening procedures — hold die on surface grinder's magnetic chuck; use same wheel and feed rates. Check die thickness after each sharpening and add shims as necessary.

COATINGS AND TREATMENTS

SUPERMAX™, MAXIMA™ COATINGS AND NITRIDE TREATMENT FOR PUNCH PRESS TOOLING

What is SuperMax™ Coating?

Mate SuperMax™ is a **proprietary** next generation coating specifically formulated for punch press tooling. A hard, wear resistant, and lubricious coating, SuperMax acts as a barrier between the punch and sheet metal to greatly improve stripping. In customer testing, SuperMax outperforms currently available premium coatings by 2 to 8 times, depending on the application.

Applied using the very latest nano-layer technology, SuperMax's harder, denser film greatly increases wear resistance and has a much lower friction coefficient of about 20%. Lower friction means less heat build-up, less galling and longer tool life. SuperMax is particularly good for adhesive wear tooling applications. The lubricity is also beneficial when punching sharp cornered shapes with a 90 degree or smaller angle.

SuperMax can be applied to M4PM™, M2, and Durasteel™.

What is Maxima™ Coating?

Maxima is Mate's premium tool steel coating formulated exclusively for punch press tooling applications. Hard, wear-resistant and lubricious, Maxima is a multilayer Zirconium Titanium Nitride coating. When applied to the surface of Mate's premium tool steel punches, Maxima acts as a barrier between the punch and the sheet metal. With its exceptional lubricity, Maxima greatly improves stripping by reducing the friction that occurs during this portion of the punching cycle. Since less friction means less heat build up, that means there's less galling and longer tool life. Maxima is particularly good for adhesive wear tooling applications. In real life tests, Maxima has increased tool life by a factor of 2 to 10 times, keeping tools in production longer with increased up time.

Maxima can be applied to M-2, M4PM™, and Durasteel™.

What is Nitride Treatment?

Nitride is a heat treatment feature for abrasive and adhesive wear environments when punching thin materials. It becomes an integral component of the structure of the punch material, extending tool life. Nitride treated punches are recommended for punching abrasive materials (e.g., fiberglass) or materials that cause galling (e.g., stainless steel, aluminum) and for high speed punching.

Nitride can be applied to M-2 and M4PM™ tool steel.

Coating	3000 & 5000 Series Aluminum	Galvanized Steel	Stainless Steel	Stainless Steel Under 14 ga.	Vinyl Coated Materials	Prepainted Materials Under 16 ga.	Cold Rolled Steel Under 12 ga.	Fiberglass
SuperMax™	•	•	•	•	•	•	•	•
Maxima™	•	•	•	•	•	•		
Nitride	•			•		•	•	•

Shape	Minimum punch size suitable for SuperMax™ Coating	Minimum punch size suitable for Maxima™ Coating	Minimum punch size suitable for Nitride Treatment	Minimum punch size suitable for Nitride when nibbling
Round	Minimum diameter = 0.098(2.50)	Minimum diameter = 0.098(2.50)	Minimum diameter = 0.158(4.01)	Minimum diameter = 0.500(12.70)
Rectangle	If length is > 0.250(6.35) The minimum width is 0.060(1.50) If length is < 0.250(6.35) The minimum width is 0.098(2.50)	If length is >z0.250(6.35) The minimum width is 0.060(1.50) If length is <0.250(6.35) The minimum width is 0.098(2.50)	Minimum width = 0.158(4.01)	Minimum width = 0.500(12.70)
Oval	If length is > 0.250(6.35) The minimum width is 0.060(1.50) If length is < 0.250(6.35) The minimum width is 0.098(2.50)	If length is >0.250(6.35) The minimum width is 0.060(1.50) If length is <0.250(6.35) The minimum width is 0.098(2.50)	Minimum width = 0.158(4.01)	Minimum width = 0.500(12.70)
Square	Minimum width = 0.098(2.50)	Minimum width = 0.098(2.50)	Minimum width = 0.158(4.01)	Minimum width = 0.500(12.70)
Others	Consult a Mate application specialist			

[Dimensions in Inches (mm)]

PUNCHING THICK MATERIAL

Punching Thick Material - over 0.157(4.00)

- Use sharp punches and dies - sharpen when cutting edge has a .003 - .005 (.07 - .13) radius proper sharpening is critical
- Clearance of 25 - 30% of material thickness (reference die clearance chart)
- Heavy duty back taper on punches
- Punch to material thickness of 1:1 (minimum)
- Minimum punch size of 0.250(6.40)
- 0.020(0.50) radius on all punch corners
- Inspect tools frequently for wear
- Lubricate the sheet, punch, guide
- Run machine on slow cycle
- Special care should be taken NOT to exceed press capacity (tonnage) when punching large shapes - for best results, use 80% of press capacity because tonnage increases as the tool becomes dull
- Bridge hitting is recommended - this will keep a balanced load on the punch
- Nibbling is NOT recommended - if you must nibble, use 70% minimum of punch length - Do NOT nibble with width of punch
- Tooling recommended - (114 Style, Turmpf Style and Ultra TEC® available)
- Use SuperMax™ or Maxima™ coated punches

Benefits of using a shear punch
- Reduced tonnage (only if shear depth is equal to material thickness)
- Noise reduction
- Slug control
- Reduce shock loads (machine doesn't work as hard)
- Improved stripping

Heavy Duty Tooling Advantages:

- 1º back taper (per side) on punches
- Heavy duty SLUG FREE® die design
- Heavy duty springs (Ultra TEC® canister)
- Rooftop shear - D station and larger (114), B station and larger (Ultra TEC), Trumpf
- Radius on all 90º corners to improve corner strength

[Dimensions in Inches (mm)]

PUNCHING THIN MATERIAL

Punching Thin Material — less than 0.020(0.50)

- Use sharp punches and dies
- Use appropriate die clearance
- Ensure proper tool alignment
- Use special point tolerance for punch and stripper
- Use proper die penetration
- Use guides in good condition
- Use Fully Guided Punches
- Demagnetize tooling after sharpening to help prevent slug pulling
- Use SuperMax™ or Maxima™ coated, or Nitride treated punches
- Use SLUG FREE® dies to reduce slug pulling (blanking may benefit from non-SLUG FREE dies)
- Avoid use of station adapters if possible
- Use shorter slitting tools to reduce angular tool wear
- Use SLUG FREE® Light dies

PUNCHING NON-METALLIC MATERIALS

Punching Non-Metallic Material

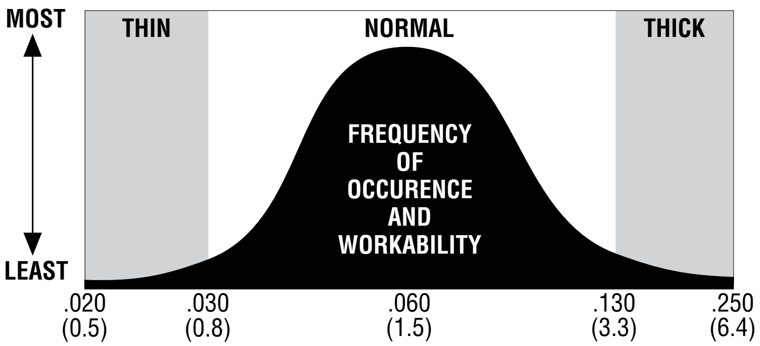
- Use sharp punches and dies
- Reduce die clearance by 5% - 8%
- Run the machine on slow cycle
- Lubricate hard plastic if possible
- Use SuperMax™ or Maxima™ coated punches
- If marking occurs use urethane pads or light spring packs
- Support thin material when possible

[Dimensions in Inches (mm)]

OTHER MATERIAL INFORMATION

What Constitutes "Normal" Sheet Metal?

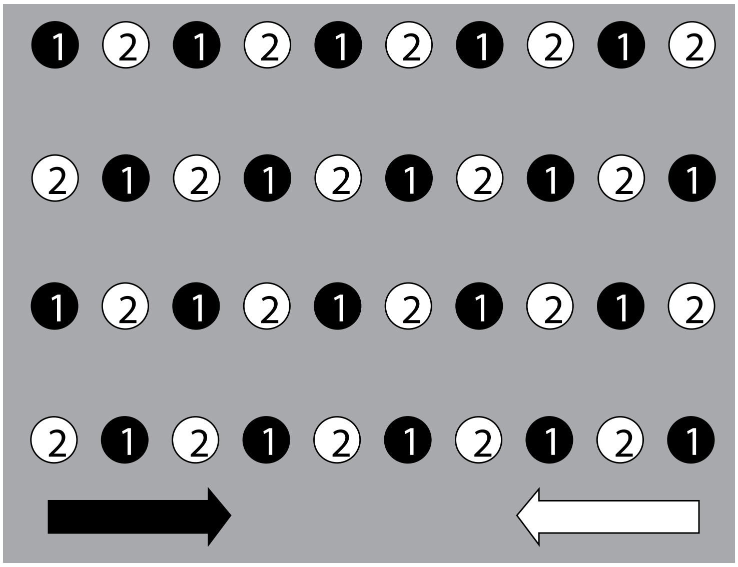
Thickness: 0.030-0.130(0.80-3.30)
Shear strength: 25,000-75,000 psi (0.172-0.157 kN/mm2)
Normal sheet metal will provide the most trouble-free operation and longest tool life. Material that is not in the normal range but still within the capacity of the punch press may require special tools, high lubrication, multiple hits and/or other procedures to produce a satisfactory job. Call Mate customer service for suggestions.



Combating Material Warpage

If you're punching a large number of holes in a sheet and the sheet does not stay flat, it could be caused by the cumulative effect of punching. Each time a hole is punched, material surrounding the hole is stretched downward, placing the top of the sheet in tension. The downward movement causes a corresponding compression at the bottom of the sheet. For a few holes, the effect is insignificant, but as the number of holes increases the tension and compression can multiply to the point where the sheet deforms.

One way to counteract this effect is to punch every other hole first and then come back and punch the remaining holes. This places the same amount of force on the sheet, but it disrupts tension/compression accumulation that occurs when punching operations follow one another in close succession and in the same direction. It also allows the first set of holes to absorb some of the distorting effect of the second set.



FULLY GUIDED TOOLING EXPLAINED

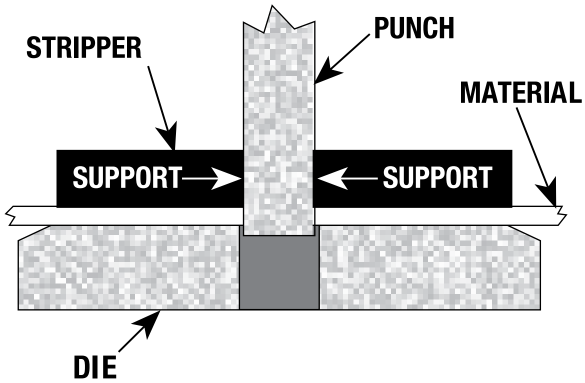
The punch guide assembly holds the stripper rigidly. The fully guided stripper has tight clearance and guides the tip of the punch to prevent lateral movement of the punch point to assure accurate hole punching and long tool life. The result is truly exceptional fully guided punching performance.

Fully Guided Tooling is ideal for:

- Slitting, and nibbling applications which induce lateral movement in the punch tip
- Applications with closely space holes
- Holes narrower than the material thickness

MATERIAL	CONVENTIONAL TOOLING	FULLY GUIDED
Aluminum	0.75 to 1	0.5 to 1
Mild Steel	1 to 1	0.75 to 1
Stainless Steel	2 to 1	1 to 1

Table shows the minimum recommended punch width as a ratio relative to the material thickness. Fully guided tooling system allows narrow punch widths to be punched.

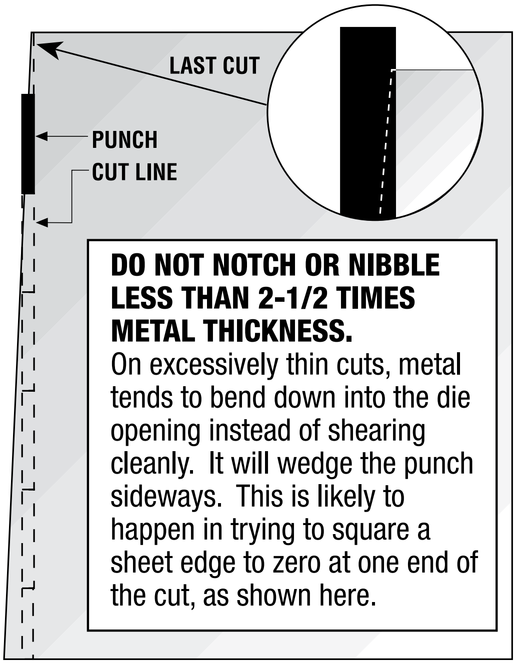


Narrow Punches Need Guiding

Punches narrower than material thickness are vulnerable to lateral forces which bend or break the tip. This results in tight punch-to-die clearance on the side toward which the punch is bending and the punch dulls quickly. If severely misused, the punch bends far enough to shave the die, damaging both tools. We recommend never nibbling a strip narrower than 2-1/2 times material thickness.

Even in normal operation, narrow punches benefit from support at the punch tip. Mate's fully guided assembly provides such support with close stripper-to-punch clearance. The stripper clamps material to the die during the entire working part of the stroke so that it can support the punch as near to the tip as physically possible.

At Mate, we recommend fully guided assemblies with full confidence for all applications using narrow punch widths. Quality of production consistently improves and tools last three or more times longer than without guiding.



DO NOT NOTCH OR NIBBLE LESS THAN 2-1/2 TIMES METAL THICKNESS.
On excessively thin cuts, metal tends to bend down into the die opening instead of shearing cleanly. It will wedge the punch sideways. This is likely to happen in trying to square a sheet edge to zero at one end of the cut, as shown here.

[Dimensions in Inches (mm)]

PUNCHING TECHNIQUES

A. BLANKING

Blanking is when the slug, normally the scrap part, becomes the saved or good part. The following recommendations will assist in making good quality blanks.

- Determine what blank dimensions are critical and notify Mate when ordering that the tools will be used for blanking. When blanking, the die size is the blank size. Punch dimensions are calculated from the die dimensions.
- Use only sharp punches and dies. This increases the straight or burnished portion of the blank to provide straighter walls on the required parts.
- Reduce the die clearance by 5%. This helps increase the burnish area and minimizes the dimensional difference between the top and bottom of the blank.
- Punches should be flat-faced.
- Use non slug free dies.
- Inspect tools frequently for wear. We recommend more frequent inspection of the tools, since tools will require sharpening more frequently when using reduced die clearances.

B. CORNER ROUNDING — FOUR-WAY RADIUS TOOL

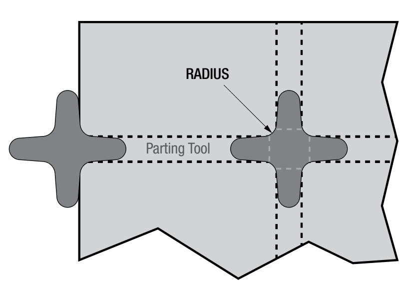
FOUR-WAY RADIUS TOOLS SPEED PROCESSES, MAXIMIZE SHEET RANGE

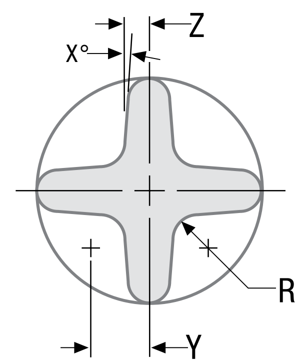
Parts with rounded edges are frequently used in sheet metal fabrication. Not only do they provide a more finished appearance, rounded edges also eliminate sharp, pointed corners that could cause injury during removal or damage internal components or wiring. Many times, fabricators use special tools that also require the use of an index station. Certain operations may require these, but there is an often overlooked tool that can improve process time, reduce costs and reduce machine wear: the four-way radius tool.

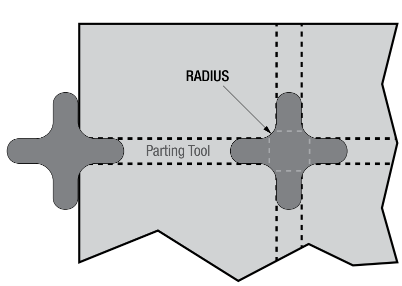
4-WAY RADIUS TOOL FEATURES:

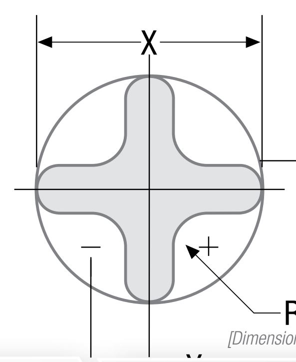
- Allows fabricators to get corner radii in parts with just one hit vs. nibbling or using a special tool
- Hits all four corners simultaneously
- Reduces or eliminates the need for an index station
- Can fit in a single station or a multi-tool
- Can be designed with micro-joint ends for shake-and-break parting applications

In a continuous parting application, the four-way radius tool uses one vertical hit between the parts to get the nest closer to the maximum range. The four-way radius tool can also help maximize sheet range and reduce machine wear. For example, if you have a sheet of 100 parts, you can punch the corners with 108 hits with the one tool vs. 400 hits with a special tool in an index station.







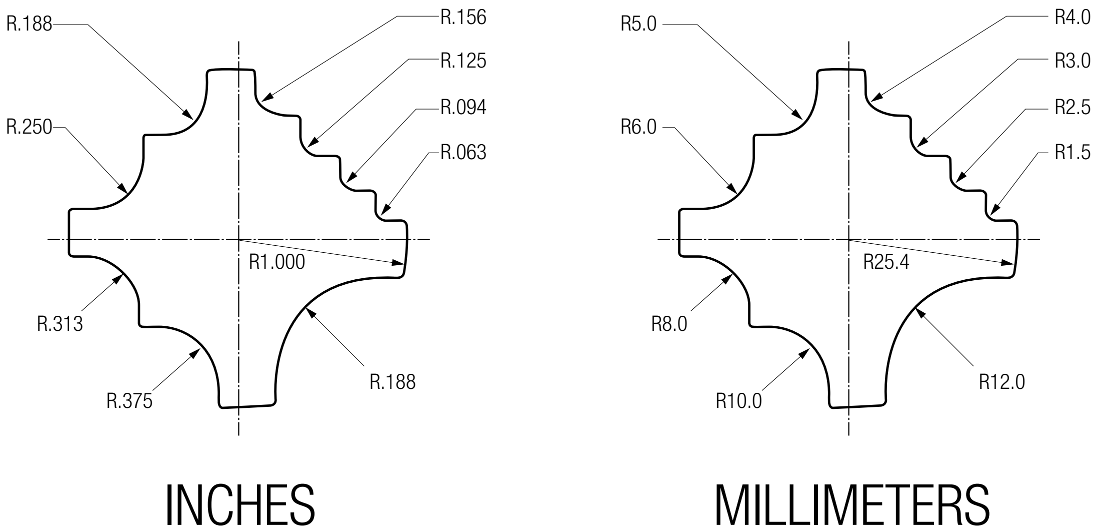


[Dimensions in Inches (mm)]

PUNCHING TECHNIQUES (CONT.)

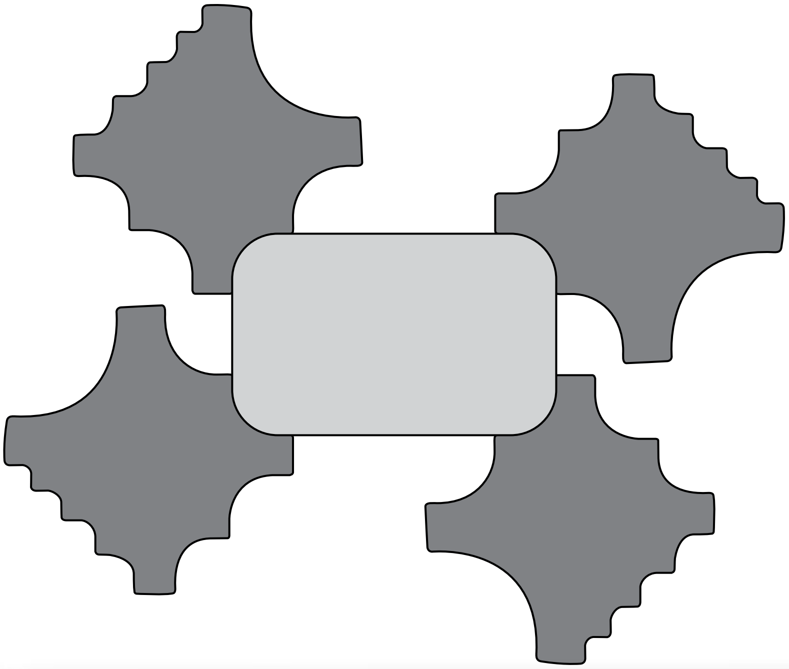
B. CORNER ROUNDING — 9-WAY CORNER ROUNDING

A single 9-Way Corner Rounding tool provides nine external popular radii from 1/2 to 1/16 inch. Auto Index programming selects and rotates the desired radius to round off all corners of a piece part. Fully Guided tooling is recommended because positive guiding action is important for support when only one side of the punch is used at a time.



Program coordinates furnished with each tool

When specifying custom radii, please submit a sketch drawn in the manner below. For adequate corner rounding performance, each radius should include at least 90o of arc (1/4 of a circle). For adequate tool strength, “Noses” between deep radii or groups of radii should be at least .188(4.7) wide as shown above. Any number of radii that will fit on the tools may be used - limiting factors are tool size and strength.



[Dimensions in Inches (mm)]

PUNCHING TECHNIQUES (CONT.)

C. HOLES, SMALL DIAMETER

When punching small diameter or narrow holes, check that tools are properly sharpened and maintained. The following recommendations are provided as guidelines to eliminate machine or tooling complications. In each situation, the user must consider the application, the machine, and the tooling before exceeding these recommendations.

Ratio of Punch to Material Thickness

NON-GUIDED TOOLING
(Ultra TEC® Tooling)

Material	Punch to Material Ratio
Aluminum	.75 to 1
Mild Steel	1 to 1
Stainless Steel	2 to 1

This means that if the material being punched is .078(2.0) thick aluminum, it is reasonable to punch a .059(1.5) diameter hole with the above listed styles of tooling. If the material being punched is .078(2.0) thick mild steel the smallest punch that is recommended is .078(2.0) diameter (or wide shape). If the material being punched is .078(2.0) stainless steel, the smallest punch recommended is .157(4.0) diameter (or wide shape).

FULLY GUIDED TOOLING
(MARATHON® & MARATHON PLUS™, Ultra TEC® Fully Guided Tooling)

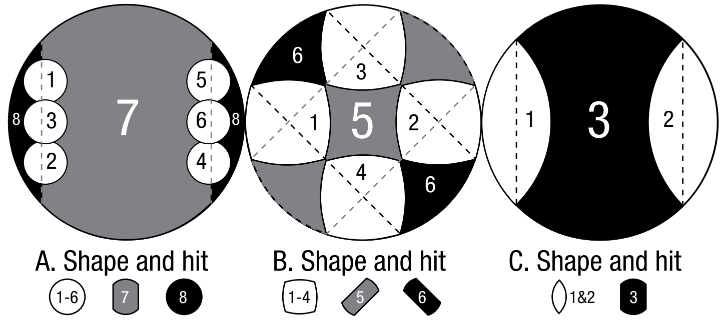
Material	Punch to Material Ratio
Aluminum	.5 to 1
Mild Steel	.75 to 1
Stainless Steel	1 to 1

This means that if the material being punched is .078(2.0) aluminum, it is possible to punch a .039(1.0) diameter hole using a Mate fully guided product. In mild steel that tool would need to be a minimum of .059(1.5), and in stainless a minimum of .078(2.0) diameter (or wide shape).

D. HOLES, LARGE DIAMETER

Although tooling is available for round holes up to station maximums for the largest station, such holes can exceed press capacities, especially in high shear strength materials. Creating the hole with more than one hit may solve the problem. Using smaller tools to break long perimeters on large tools can cut tonnage by a half or more, without resorting to nibbling the entire periphery. The diagrams above use rounds, double D's, a quad radius and a biconvex radius. In all three, slugs fall away through the die, leaving no scrap on the punch press table.

HOW TO PUNCH LARGE HOLES WITHOUT EXCEEDING PRESS TONNAGE



A. Shape and hit B. Shape and hit C. Shape and hit

[Dimensions in Inches (mm)]

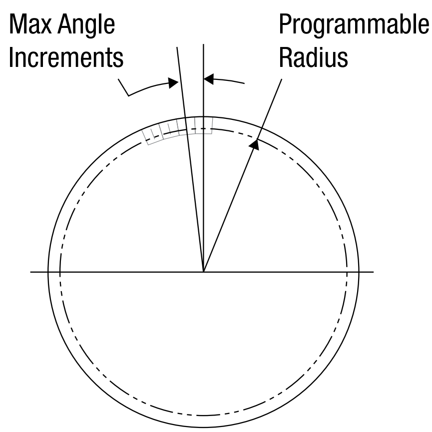
PUNCHING TECHNIQUES (CONT.)

E. NIBBLING

Quad radius tool for auto index stations

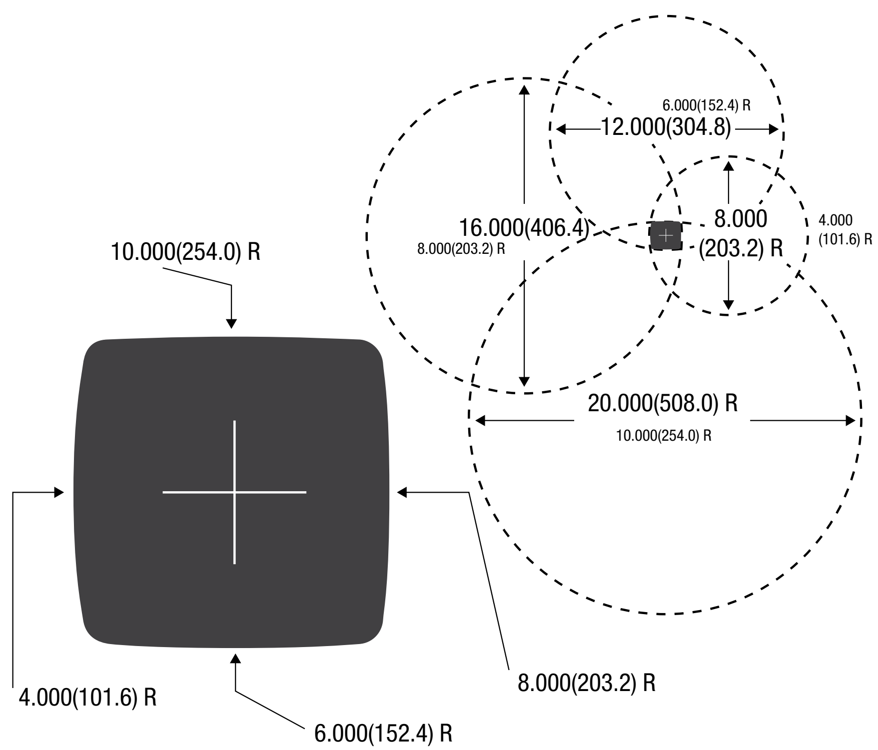
The quad radius tool nibbles large holes with smoother edges and with far fewer hits than using a round nibbling punch. In effect, the quad radius tool puts an 8, 12, 16 and 20-inch punch (or whatever hole size you need) into a single 2-inch station. It will nibble holes even larger, but as hole size exceeds a tool radius, scalloping begins to appear. See "program note" below for programming the tool into auto index stations. Fully guided tooling is recommended because the positive guiding action gives the punch the support it needs for nibbling and the stripper clamps the sheet securely to prevent lateral movement.

When ordering, specify machine model, station, maximum tool dimension, radii, thickness and type of material.



TOOL RADIUS	PROGRAMMABLE RADIUS	MAX ANG INCR	# HITS
10.000(254.0)	9.2679(235.40)	7.55°	48
8.000(203.2	7.2616(184.44)	9.56°	38
6.000(152.4)	5.2511(133.38)	12.56°	29
4.000(101.6	3.2299(82.04)	19.27°	19

Program Coordinates Furnished with Each Tool



[Dimensions in Inches (mm)]

PUNCHING TECHNIQUES (CONT.)

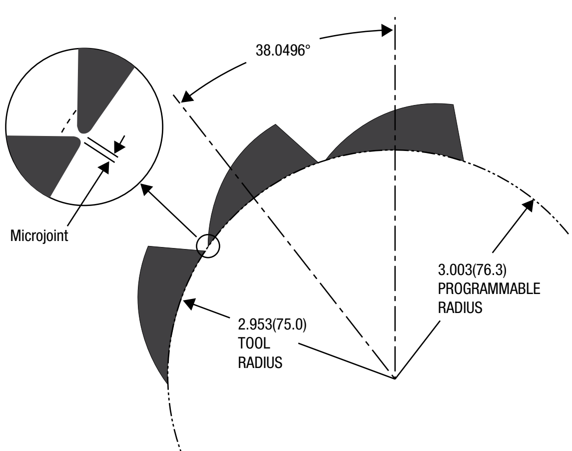
E. NIBBLING (CONT.)

Inside/outside radius

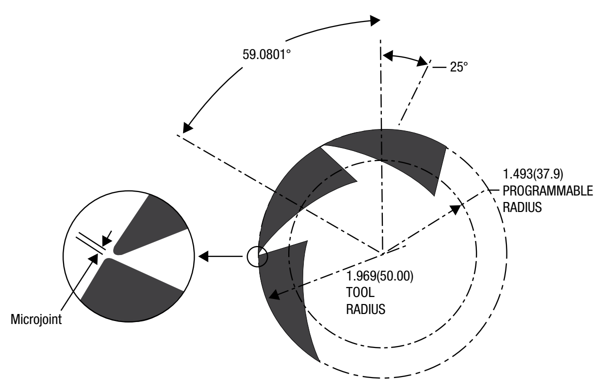
With this tool, you do not have to stop your machine to remove the slug for an oversize opening, or the blank if saving the slug. Small, precise tabs keep slugs and blanks intact while being punched, yet permit them to break away from each other easily off the machine. The precise tab is created by leaving a microjoint* between hits. The tool's large radii result in slugs or blanks with smoother edges produced with far fewer hits than using an ordinary radius punch for nibbling holes. One tool punches slugs or blanks of any size practical for its configuration with smoothest edges occurring when radius punched and tool radius coincide.

This tool is for use in auto index stations. Inside radius must be larger than outside radius. This tool can be programmed to punch holes with slugs or parts retained in the sheet, yet can be separated easily off the press.

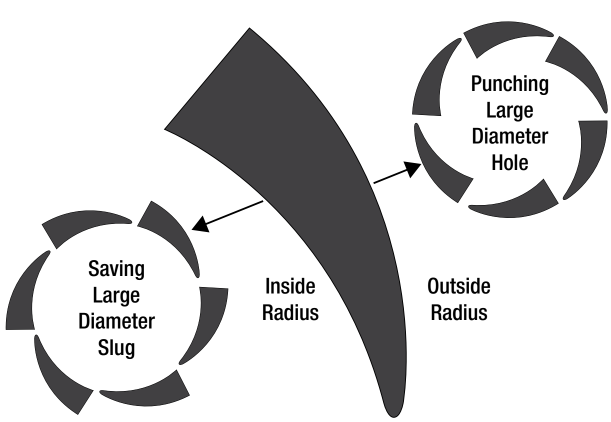
*Microjoint size depends upon the material type and thickness



SAVING LARGE DIAMETER SLUG



PUNCHING LARGE DIAMETER HOLE
CENTERLINE LAYOUT
Mate provides a centerline layout drawing for each tool to aid in prgramming



I/O RADIUS TOOL CUTS BOTH INSIDE AND OUTSIDE RADII

[Dimensions in Inches (mm)]

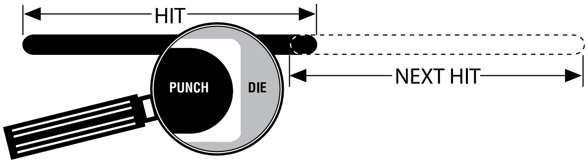
PUNCHING TECHNIQUES (CONT.)

F. SLITTING

The slitting process requires the tool to pierce material securely and accurately while overcoming various side loads. Parting a sheet includes an amount of punch overlap in each hit where sheet resistance is partially absent. This causes the punch to try and move towards the space where material is absent. The greater the area where material is absent, the greater the side load on the punch. In extreme cases where sheet thickness is thin, the material may even be folded into the die rather than fracturing and falling away. Any of these problems can reduce sheet quality.

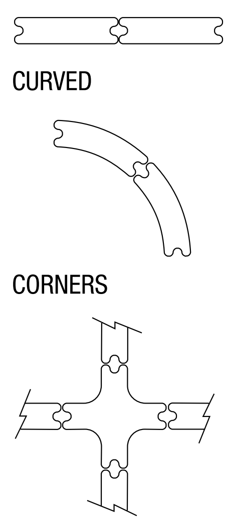
A Smooth Slitting Tip

To get rid of the small "teeth" left on edges by rectangular tools, it is a common practice to order oval punches with rectangular dies having radiused corners for slitting and parting. The radii blend into the next cut more smoothly even on older machines with play in the toolholder bores and workholders. Workpieces are less likely to cause cuts and scratches when being handled, need less finishing work later.



G. MICRO-JOINTS

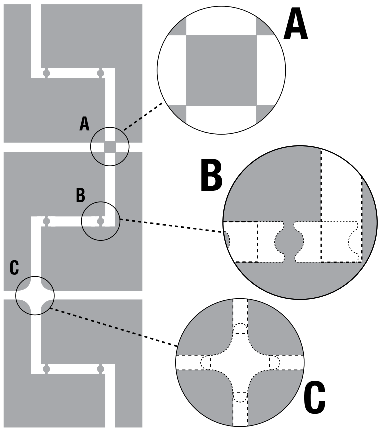
Shake-and-break is a popular name for this easy method of separating multiple parts from a sheet of material. The method is based on small, interconnecting tabs between the parts created by programming spacing of the shearing or slitting punch. These tabs keep the sheet and parts intact while being punched, yet easy to separate off the machine. A starting point for tabs can be .008(0.2) wide. The size can be varied to increase/decrease holding strength. Straight, curved, or corner shaped tools are available.



Applications

Scrap retention on the inside of parts, between parts, outside of parts, corners of parts. The most appropriate micro-joint technique is dependent upon the configuration of the product being fabricated. Rectangle shaped tools can be used to create micro-joints at outside corners. Bowtie or fishtail shaped tools can create micro-joints along a common parting line. Trapezoidal shaped tools can be used to create micro-joints when only one side of the tool corresponds with a part. There are other possible methods to create micro-joints depending upon the parts being fabricated, such as our Square EasySnap™ tool. Contact a Mate Precision Tooling Application Specialist to discuss your options.

Three Methods For Separating Parts Using Long, Narrow Rectangles



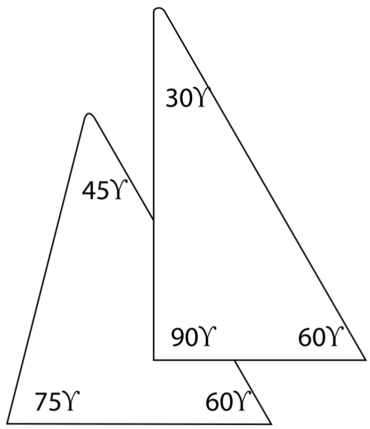
Shake-And-Break – By programming a small gap between hits at exterior corners (A), the corners remain connected to the sheet until removed from the press and shaken loose. This technique works where corners of four parts meet.
Tab Tool - By programming a larger gap adjacent to interior corners (B), a special tab tool can transform the gap into a 0.008(0.2) shake-and-break connection. Just one tangent or radial tool makes a tab at any corner without rotating when the corner is made by the shearing tool perpendicular to the tab tool.
4-way Corner Rounding Tool - If exterior corners don't need to remain connected (C), the 4-way corner rounding tool cuts and rounds all four corners in one hit. Tips are specially tapered to blend the corner radius into the sides – also available with shake-and-break tab tips.

[Dimensions in Inches (mm)]

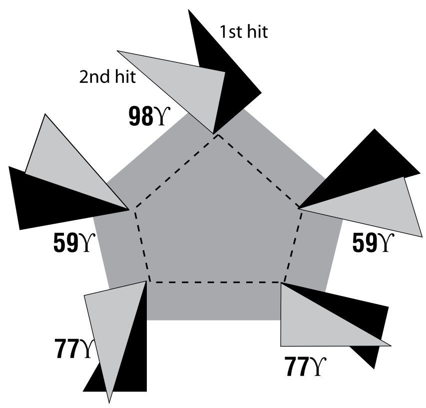
PUNCHING TECHNIQUES (CONT.)

H. NOTCHING

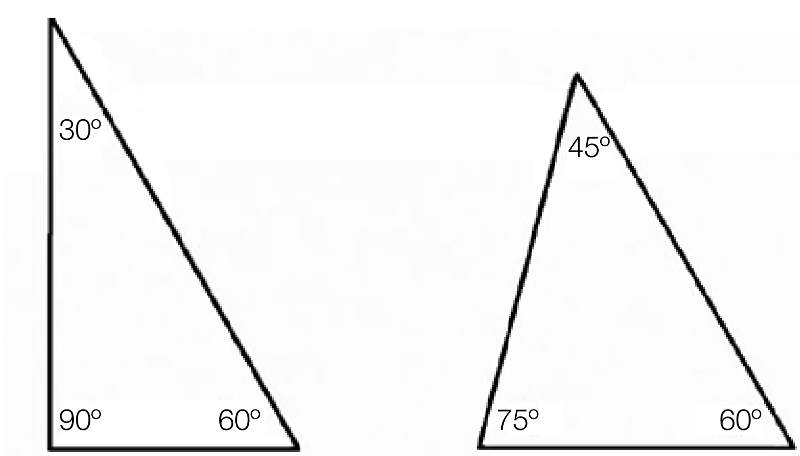
Notch a variety of angles with one set of 3-way corner notching tools. This is another application for an auto index station. A three-way corner notching tool can cut any angle larger than the smallest point by programming single or multiple hits. 15º is the smallest angle available.



TWO POPULAR CONFIGURATIONS…



The 3-Way Notching tool can include angles from 150º to 15º. Shown below are two popular arrangements. Fully Guided Tooling is recommended because positive guiding action supports the punch well in notching where only one corner is used at a time.

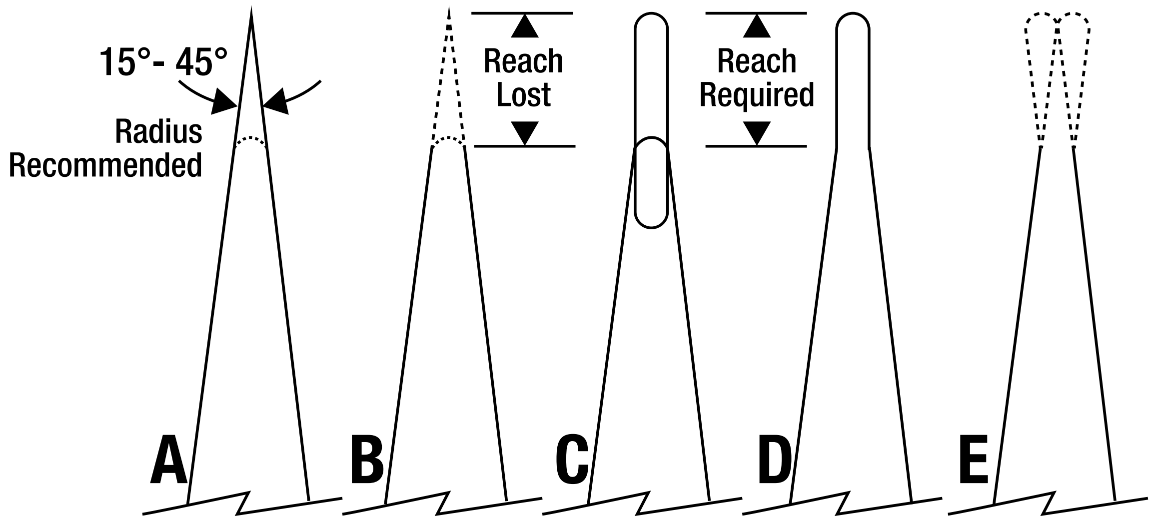


[Dimensions in Inches (mm)]

PUNCHING TECHNIQUES (CONT.)

H. NOTCHING (CONT.)

Min Radii: Aluminum, 1/4T; Mild Steel, 1/2T; Stainless, 1T (T=Thickness)
On acute angles (A), the sharp edge wears more quickly than larger angled corners. We recommend a minimum radius of .010(0.25) (B) on all angles under 45º to help overcome rapid wear. Since a radius shortens the sharp edge, it may be necessary to pre-punch acute notches to reach a part's bend line. Use a round or narrow oval hole at least one material thickness in diameter or width. We recommend the oval (C) in most cases because it provides relief where bends meet and forms a tighter joint than a round hole. It is also possible to order a 3-Way Notching Tool with a nose (D) that achieves the same purpose as pre-piercing. The nose can bisect the angle or continue from either side (E).



[Dimensions in Inches (mm)]

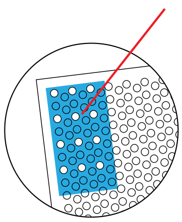
PUNCHING TECHNIQUES (CONT.)

I. CLUSTER PUNCHING

Cluster tooling is an ideal way to produce repeat holes or patterns in sheet metal. By increasing the number of holes per hit, clusters are efficient, reduce costs and can help reduce machine wear and tear. Many different punch designs and cluster areas are available, providing a wide variety of punching choices. Here are techniques you can use to ensure you get the results you require.

Punching Force Formula:
For cluster punching, the maximum recommended punching force **SHOULD NOT EXCEED 75% of press capacity.** The punching force increases as the tool becomes dull. Use the following formula to estimate the required punching force:

LINEAR LENGTH OF CUT X MATERIAL THICKNESS X SHEAR STRENGTH = PUNCHING FORCE IN TONS (METRIC TONS)
"Linear Length of Cut" = Hole Perimeter X Number of Punches in the Cluster
"Hole Perimeter," Round Hole = 3.14 X diameter
"Hole Perimeter," Shaped Hole = Sum of the lengths of the sides



EXAMPLE:
In the illustration above, the punch (represented by the blue rectangle) is a 12-hole round cluster, with the holes being .250(6,35) in diameter. The area of the cluster punch covers 48 holes, which is punched every fourth hole (12 holes, 4 times). The material is mild steel .060(1,52) thick.

12-ROUND HOLE CLUSTER:

	LINEAR LENGTH OF CUT										
	Hole Perimeter	**X**	**Number of Punches in Cluster**	**=**	**Linear Length of Cut**	**X**	**Material Thickness**	**X**	**Shear Strength**	**+**	**Punching Force Tons/ Metric Tons**
Inch	3.14 X .250 = .785	X	12	=	9.42	X	.060	X	25	=	14.1 tons
Metric	3.14 X 6,35 = 19,94	X	12	=	239,26	X	1,52	X	0,345	=	12.8 metric tons
If the holes were .250(6,35) square, instead of round, with all other factors being the same, the results would be: 12-Square Hole Cluster:											
Inch	4 X .250 = 1.00	X	12	=	12.00	X	.060	X	25	=	18.0 tons
Metric	4 X 6,35 = 25,40	X	12	=	304.80	X	1,52	X	0,345	=	16,3 metric tons

AVAILABLE IN ALL TOOLING STYLES AND STATION SIZES.

WATCH THE VIDEOS:
See how one Mate customer achieved over 4.1 million hits with a Maxima-treated cluster punch without sharpening or maintenance:
https://youtu.be/vkQHeZru6zE

View an animation of a Trumpf-style cluster:
https://youtu.be/I3NBrp0jqag

[Dimensions in Inches (mm)]

PUNCHING TECHNIQUES (CONT.)

I. CLUSTER PUNCHING (CONT.)

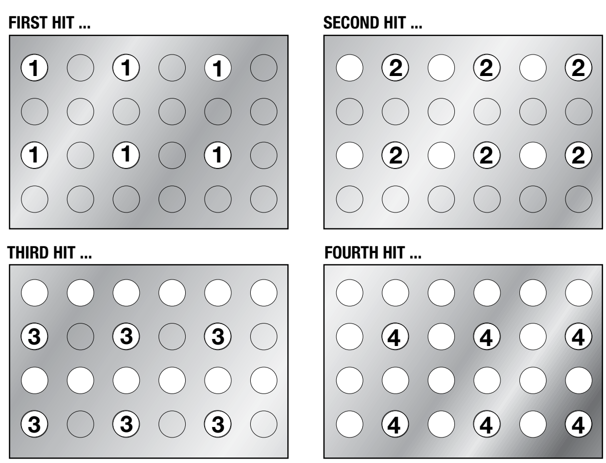
MINIMUM PUNCH SIZE:
When punching small diameter holes, check that the tools are properly sharpened and maintained. Use the following recommendations as guidelines to determine the smallest punch diameter to eliminate machine or tooling complications:

MATERIAL	PUNCH-TO-MATERIAL RATIO, STANDARD TOOLING	PUNCH-TO-MATERIAL RATIO, FULLY GUIDED TOOLING
Aluminum	.75 to 1	.5 to 1
Mild Steel	1 to 1	.75 to 1
Stainless Steel	2 to 1	1 to 1

This example applies the above ratios to materials that are .078(2,0) thick to get the smallest recommended diameter punch:

MATERIAL AND THICKNESS	SMALLEST PUNCH, STANDARD TOOLING	SMALLEST PUNCH, FULLY GUIDED TOOLING
Aluminum, .078(2,0)	.059(1,5) diameter	.039(1,0) diameter
Mild Steel, .078(2,0)	.078(2,0) diameter	.059(1,5) diameter
Stainless Steel, .078(2,0)	.157(4,0) diameter	.078(2,0) diameter

HOLE UNIFORMITY AND FLATTER SHEETS:
For greater hole uniformity and flatter sheets, spread the punches to avoid punching adjacent holes in the same hit. Repeat for the necessary number of times to complete the desired pattern.



USE FULLY-GUIDED CLUSTERS IN CHALLENGING APPLICATIONS:
In challenging applications or high production environments, fully-guided cluster punches are extremely helpful. The design suits small punches that benefit guiding at the tip to ensure accuracy. Fully-guided also works well for punch clusters with too few punches to provide a good punch-to-stripper guiding surface. Fully-guided cluster punches are ideal for heavy-duty service or long production runs.

SUPERMAX™ OR MAXIMA™ COATING FOR LONG PUNCH LIFE:
Mate's optional SuperMax™ and Maxima™ coatings imcrease the lubricity of the punch points, helping to resist wear and ensure cleanly punched holes. In fact, one Mate customer achieved over 4.1 million hits with a Mate cluster punch treated with Maxima without sharpening or maintenance!

LUBRICATE THE CLUSTER PUNCHES:
Use a good lubricant, such as vanishing oil, to reduce heat build up and prevent galling.
Use Mate Eliminator pads.

[Dimensions in Inches (mm)]

TROUBLESHOOTING QUICK REFERENCE

PROBLEM	POSSIBLE CAUSE	SOLUTION SUGGESTED
Excessive Burrs	Incorrect die clearance	Adjust to proper
	Differing material hardness although gage is the same	Adjust clearance
	Dull punches and dies	Sharpen tooling
	Slug pile-up or packing	Check dies and clearance
		Increase punch penetration
	Holder on station misaligned	Check alignment
Poor Hole Quality	Dull punches and dies	Sharpen tooling
	Improper clearance	Adjust to proper
	Die not properly seating	Check dies
	Holder or station out of alignment	Check alignment
	Punching thin material	Use guided tooling
Punch Breakage	Inadequate die clearance	Adjust to proper
	Crossed Shapes	Ensure tools are properly loaded in turret
	Size of punch less than one material thickness	Use guided tooling
Punch Does Not Strip	Dull punches or dies	Sharpen tooling
	Improper die clearance	Adjust to proper
	Difficult material	Adjust die clearance
	Weak spring	Replace spring
	Tool limitations exceeded	
	Galling	Lubricate tooling - Use Mate Eliminator pads (see page 17) Use SuperMax™, Maxima™ coating
Punch Galling	Dull punch	Keep tools sharp
	No lubrication	Lubricate work piece - Use Mate Eliminator pads (see page 17)
	High hit rate	Adjust
	No coating	Use SuperMax™, Maxima™ coating
	Inadequate die clearance	Increase die clearance
Punch Sticking in Work Piece	Dull punch and/or die	Sharpen tools
	Inadequate die clearance	Increase die clearance
	Galling on punch	Remove galling Use SuperMax™, Maxima™ coating
	Inadequate lubrication	Lubricate work piece - Use Mate Eliminator pads (see page 17)
	Weak spring	Increase stripping Replace stripping springs

[Dimensions in Inches (mm)]

TROUBLESHOOTING QUICK REFERENCE

PROBLEM	POSSIBLE CAUSE	SOLUTION SUGGESTED
Rapid Tool Wear	Inadequate die clearance	Increase die clearance
	Punch overheating	Lubricate tools
	Poor sharpening practices	See Punch & Die Maintenance
	Nibbling	Alter programming
	Poor stripping	Use SuperMax™, Maxima™ coating
	Poor tool alignment	Realign stations Level turret Replace tool holders
	Material being punched (for example, stainless)	Use SuperMax™, Maxima™ coating
Sheet Accuracy	Worn work holders	Adjust or replace Replace gripping surfaces
	Alignment problems	Realign table to press Inspect for worn turret bores Level turret
Slug Pulling	Magnetism in tools	Demagnetize
	Small diameter holes**	See below
	**The most common condition(s) for slug pulling are: round holes. 250 to .750(6.35 to 19) diameter in .039 to .078(1 to 2) thick material, with sharp tools, using optimum clearance, and minimum penetration on oiled material. The solutions suggested are to: •Maximize die penetration •Use Slug Free® dies •Slug ejectors	
Surface Cracks on Face of Punch	Tool improperly ground	Dress wheel and grind taking light cuts
Warpage of Work Piece	Dull tools	Sharpen punch and die (use coolant when sharpening)
	Improper clearance	Increase or decrease as necessary
	No lubrication	Lubricate sheet
	Poor stripping	Increase stripping
	Programming	Reprogram punching sequence Bridge hit large openings
Work Piece Marking	Excessive stripping force	Radius and polish die top Shim dies evenly
	Stripper plate small surface area	Self-adhesive urethane stripper pads
	Uneven die surface	Adjust stripping force if possible

[Dimensions in Inches (mm)]

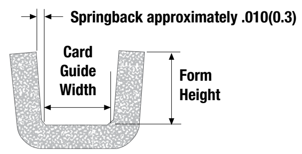
CARD GUIDE

This tool forms edges of pre-pierced slots into crisp, clean card guides for printed circuit boards. Open space between boards allows air to circulate to cool components. Smooth sidewalls with flared ends guide PC boards into their slots without marring. Sidewalls are steep and high enough to hold PC boards securely.

Card guides can be spaced as close as .402(10.2) center-to-center for excellent component density without losing ventilation. Forms mild steel, aluminum, stainless steel, etc.

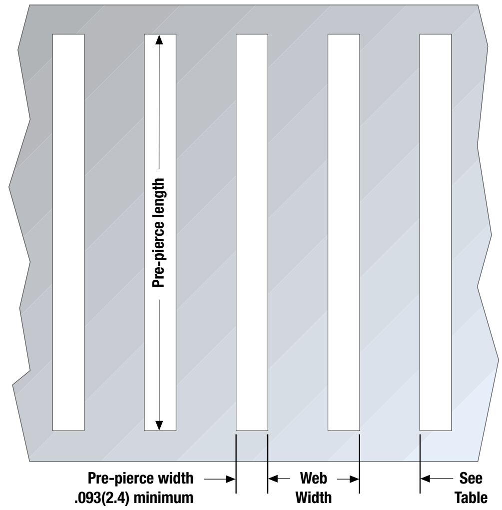
FORM HEIGHT:
Minimum .080(2.0)
Maximum .125(3.2)

MATERIAL THICKNESS:
Minimum .048(1.2)
Maximum .074(1.9)

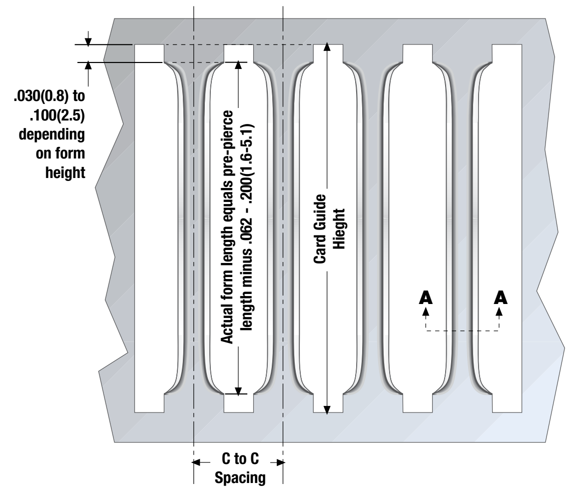


RECOMMENDED WEB WIDTH TABLE BASED ON .076(1.9) CARD GUIDE WIDTH		
MATERIAL THICKNESS	**FORM HEIGHT**	**WEB WIDTH**
(1.5)	.080(2.0)	.284(7.2)
	.090(2.3)	.304(7.7)
	.100(2.5)	.324(8.2)
	.125(3.2)	.374(9.5)
048(1.2)	.080(2.0)	.274(7.0)
	.090(2.3)	.294(7.5)
	.100(2.5)	.314(8.0)
	.125(3.2)	.364(9.2)

STEP 1, PRE-PIERCE THE MATERIAL



STEP 2, FORM THE CARD GUIDE



FORM HEIGHT CALCULATION NOTES:

- C to C Spacing - Web Width = Pre-Pierce Width
- Example for form height of .082(2.0) and material thickness of .046(1.2):
- From table recommended web width is .274(7.0).If C to C spacing is .700(17.8), then pre-pierce width is .700(17.8) - .274(7.0) = .426(10.8).
- It is recommended that the card guide tool is .062(1.6) shorter than the pre-pierce length for .031(0.8) clearance and relief on each end of form.

HELPFUL TIPS:

- Stations adjacent to this tool should not be used
- Do not pass workholders over step die
- For best forming results, use a good forming lubricant

WATCH THE VIDEO:
Watch an animation of a Card Guide

https://youtu.be/rkwZvimkElg

[Dimensions in Inches (mm)]

COUNTERSINK

Definition
A formed countersink is a coined form that displaces sheet metal to create an angled depression in the surface of the material to provide a seating location for a flat head screw, rivet, etc. This form can be done either to the top or the bottom of the sheet. A successful form is dependent upon the combination of material properties and countersink features.

Features
The features of a formed countersink are the form angle, major diameter, minor diameter, and depth. A formed countersink does not remove material as with a machined countersink but instead, displaces material. This plastic deformation of the material is generally opposite to the direction of the applied force, moving material back into the sheet and down into a pre-pierced hole. Countersinks typically cause the pre-pierced hole to close as the material is formed but it is possible a large countersink in thin material may enlarge the size of the pre-pierced hole.

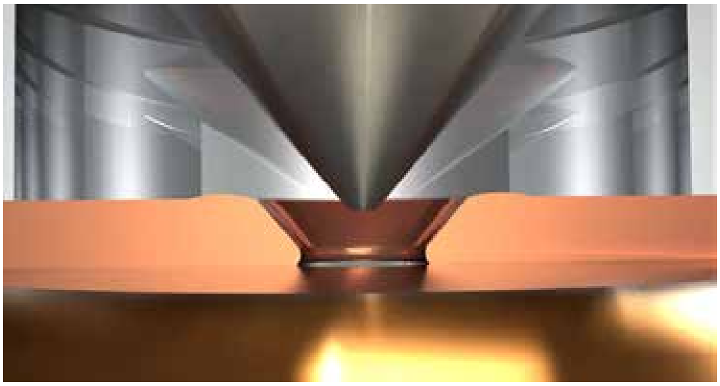
Process
Creating a formed countersink requires a specific volume of material to be moved in the sheet. The volume displaced is a function of the formed features which, in conjunction with the material being used, are variables used to determine an approximate pre-pierce size. The best countersinking tool will have a shoulder surface around the countersink point. The point and the surface are used to apply force to the sheet and displace material as required as the sheet is pressed against a blank die. The pre-pierce size is ultimately responsible for the finished minor diameter and depth. To decrease the minor diameter (and increase the depth) a smaller pre-pierce is needed. This smaller hole leaves more material for the countersink tool to displace and push into the hole, making the hole smaller. The countersink major diameter and angle are determined by the construction of the countersink tool. The force needed to create a formed countersink for a typical flat head screw will be less than 15 tons.

Summary
Not every countersink configuration can be formed into sheet metal because the material cannot be countersunk 100% of the thickness. Generally the most important aspects of a countersink are the angle and the major diameter. If the minor diameter is larger and the depth is less the formed countersink results will be totally functional. Successful formed countersinks result from a favorable combination of material and form features. Some swelling of the material around the major diameter is to be expected. Countersink tools are material thickness specific and the point may become damaged when running in thinner material because the point will go through the material and contact the die before the shoulder stops on top of the sheet. If problems are experienced with a formed countersink not producing the desired results check the following:

- Confirm the material thickness is correct for the tool design
- Check that the form is being sufficiently coined (witness mark around shoulder Diameter)
- Adjust pre-pierce size (typically smaller for more depth and larger for less depth)

[Dimensions in Inches (mm)]

UNIVERSAL STYLE COUNTERSINK



Mate's universal style countersink tool produces a highly burnished countersink surface free of burrs and chatter marks.

Common included angles for countersinking are 82° for inch style screws and rivets - 90° for metric. Other common angles are 100° and 120°. Your tool can be made to the angle you specify.

With varied penetration, a single tool can make countersinks for screws ranging from #4 to 1/2 (M3 to M14) in material thicknesses up to .250(6.35). Form depth may not exceed 60% material thickness with a universal style countersink. Making the through hole diameter larger may help overcome this limitation.

This type of forming tool does not remove material which is displaced rather than removed. Some pre-pierced holes will 'close in' while others will 'open up'. Smaller forms in thicker material will yield a smaller through hole than the pre-pierce diameter. Larger forms in thinner material will do the opposite.

Mild steel countersinks work well because of excellent ductility combined with excellent sheet strength. Stainless steel works well too, but requires more punching force to achieve the same result. Most aluminum will work, but not as well because of lower tensile strength. Don't exceed 60% material thickness and an excellent countersink can be obtained.

For deep countersink forming - greater than 60% material thickness, a dedicated style countersink tool can be made to accommodate a specific form in a specific material type and thickness. Your Mate representative can help you obtain the tools you need for an excellent countersinking result.

MATERIAL THICKNESS:
Maximum .250(6.4)

MAJOR DIAMETER:
Maximum .938(23.8)

HELPFUL TIPS:

- Countersink up to 60% thickness
- 82°, 90° and 100° standard angles fit most flathead screws and rivets.
- Any angle may be specified
- Universal style makes countersinks up to 60% material thickness with one set of tools in materials up to .250(6.4) thick.

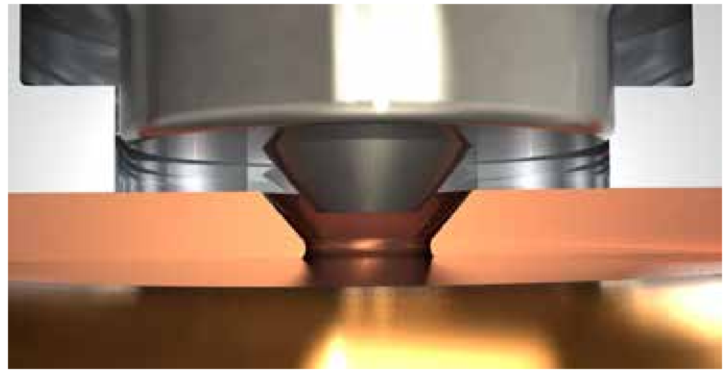
WATCH THE VIDEO:
Watch an animation of a Universal Countersink:

https://youtu.be/tWSdPKjGWio

[Dimensions in Inches (mm)]

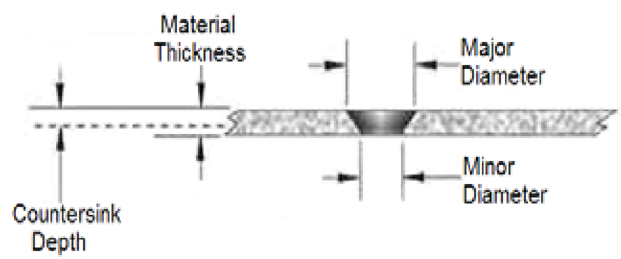
DEDICATED COUNTERSINK

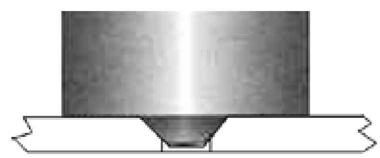
The dedicated countersink is designed with a large flat shoulder area that 'coins' the surrounding area of the countersink producing a form that has a machined appearance. For best performance, the insert should be designed for a specific screw or rivet size in a given material thickness.



Material Thickness Maximum: .250(6.4)

- Set punch holder to shut height of press minus material thickness.
- Countersink up to 85% thickness
- Pre-pierce diameter is midway between major and minor diameters (see pre-pierce calculator)
- 82°, 90° and 100° standard angles fit most flathead screws and rivets. Any angle may be specified





Dedicated style makes countersinks up to 85% material thickness in aluminum and keeps material flat even when countersinks are close together. 50% maximum depth is possible in stainless and cold rolled steel of .105(2.7) and thicker.

WATCH THE VIDEO:
Watch an animation of a Dedicated Countersink:

https://youtu.be/Pxf3xk81I9Y

[Dimensions in Inches (mm)]

COUNTERSINK

DEDICATED COUNTERSINK

TO CALCULATE THE COUNTERSINK DEPTH ENTER VALUES FOR A, B, C BELOW TO CALCULATE THE COUNTERSINK DEPTH % ENTER VALUES FOR A, B, C, T BELOW.

To Achieve Acceptable Form Results, The Following Rules Apply:

REGARDING MAJOR DIAMETER **A**

1) **A** must be <= .315(8,00) when **T** is >= .047(1,19) and < .060(1,52)
2) **A** must be <= .500(12,70) when **T** is >= 060 (1,52) and < .075(1,90)
3) **A** must be <= .656(16,66) when **T** is >= .075(1,90) and <= .236(6,00)

REGARDING DEPTH **D**

1) D can reach maximum of .85 T (85% material thickness) when:

a. **T** for Aluminum (ALUM) is >= .047(1,19)
b. **T** for Mild Steel (MS) is >= .047(1,19) and < .121(3.07)
c. **T** for Stainless Steel (SS) is >= .047(1,19) and < .077(1,96)

2) D can reach maximum of .60 T (60% material thickness) when:

a. **T** for Mild Sled is .> .121(3,07) and < .197(5,00)
b. **T** for Stainless Steel is >= .077(1,96) and < .118(300)

3) D. can reach maximum of .50 T (50% material thickness) when:

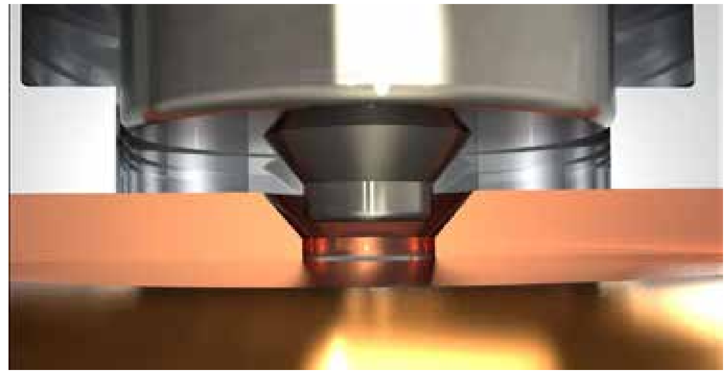
a. **T** for Mild Steel is >= .197(5,00)
b. **T** for Stainless Steel is >= .118(3,00)

PRE-PIERCE CALCULATOR FOR DEDICATED COUNTERSINK		
FEATURE		**VALUE**
A	MAJOR DIAMETER	
B	MINOR DIAMETER	
C	COUNTERSINK ANGLE IN DEGREES	
D	CALCULATED COUNTERSINK DEPTH	
T	MATERIAL THICKNESS	
S	MATERIAL TYPE (1=COPPER/BRASS, 2=ALUM. 3=MS/SS)	
E	CALCULATED PRE-PIERCE DIAMETER	
%	CALCULATED COUNTERINK DEPTH (% OF T)	
PRE-PIERCE TO MATERIAL THICKNESS RATIO (> 100%)		

[Dimensions in Inches (mm)]

PILOT NOSE COUNTERSINK

The pilot nose countersink is designed to precisely control the finished size of the minor diameter of the countersink form. A pilot goes through the pre-pierced hole and as the countersink form moves material it closes around the pilot diameter. This allows for countersinks that are close to 100% of the material thickness assuming the material can be formed as needed. If the finished minor countersink diameter size is not critical and the countersink depth is not close to 100% material thickness then the dedicated countersink would be the best choice.



Countersink up to 100% material thickness under the right circumstances
The pre-pierce diameter can be calculated using the Mate pre-pierce calculator
82°, 90° and 100° are common angles for most flathead screws and rivets. Any angle may be specified although shallow angles form much differently

Pilot nose dedicated style makes countersinks up to 100% material thickness in aluminum and thinner materials. 60-85% maximum depth is possible in thicker material and tougher materials like stainless and cold rolled steel.

WATCH THE VIDEO:
Watch an animation of a Pilot Nose Countersink:

https://youtu.be/m-yXfHSSVqk

STANDARD SCREW SIZES

SAE FLAT HEAD SCREWS								
Size	D	A1 Max	A1 Min	A2	H	.5H	1.3H	Material Range
#4	.112 (2.84)	.225 (5.72)	.207 (5.26)	.195 (4.95)	.067 (1.70)	.034 (.86)	.087 (2.21)	.047-.250 (1.19 - 6.35)
#5	.125 (3.17)	.252 (6.40)	.232 (5.89)	.220 (5.59)	.075 (1.91)	.038 (.97)	.098 (2.49)	.047-.250 (1.19 - 6.35)
#6	.138 (3.51)	.279 (7.09)	.257 (6.53)	.244 (6.20)	.083 (2.11)	.042 (1.07)	.108 (2.74)	.047-.250 (1.19 - 6.35)
#8	.164 (4.17)	.332 (8.43)	.308 (7.82)	.292 (7.42)	.100 (2.54)	.050 (1.27)	.130 (3.30)	.060-.250 (1.52 - 6.35)
#10	.190 (4.83)	.385 (9.78)	.359 (9.12)	.340 (8.64)	.116 (2.95)	.058 (1.47)	.151 (3.84)	.060-.250 (1.52 - 6.35)
1/4	.250 (6.35)	.507 (12.88)	.477 (12.13)	.452 (11.48)	.153 (3.89)	.077 (1.96)	.199 (5.05)	.075-.250 (1.91 - 6.35)
5/16	.313 (7.95)	.635 (16.13)	.600 (15.24)	.568 (14.43)	.191 (4.85)	.096 (2.44)	.248 (6.30)	.105-.250 (2.67 - 6.35)
3/8	.375 (9.53)	.762 (19.35)	.722 (18.34)	.685 (17.40)	.230 (5.84)	.115 (2.92)	.299 (7.59)	.220-.250 (5.59 - 6.35)
1/2	.500 (12.70)	.875 (22.23)	.831 (21.11)	.775 (19.69)	.223 (5.66)	.112 (2.84)	.290 (7.37)	.220-.250 (5.59 - 6.35)

METRIC FLAT HEAD SCREWS (DIN 7991)					
Size D	A1	A2	H MAX	H MIN	Material Range
M3	6	5.7	1.2	0.95	1.2 - 6
M4	8	7.64	1.8	1.55	1.5 - 6
M5	10	9.64	2.3	2.05	1.5 - 6
M6	12	11.57	2.5	2.25	1.5 - 6
M8	16	15.57	3.5	3.2	1.9 - 6
M10	20	19.48	4.4	4.1	3 - 6
M12	24	23.48	4.6	4.3	3 - 6
M14	27	26.48	4.8	4.5	3 - 6

[Dimensions in Inches (mm)]

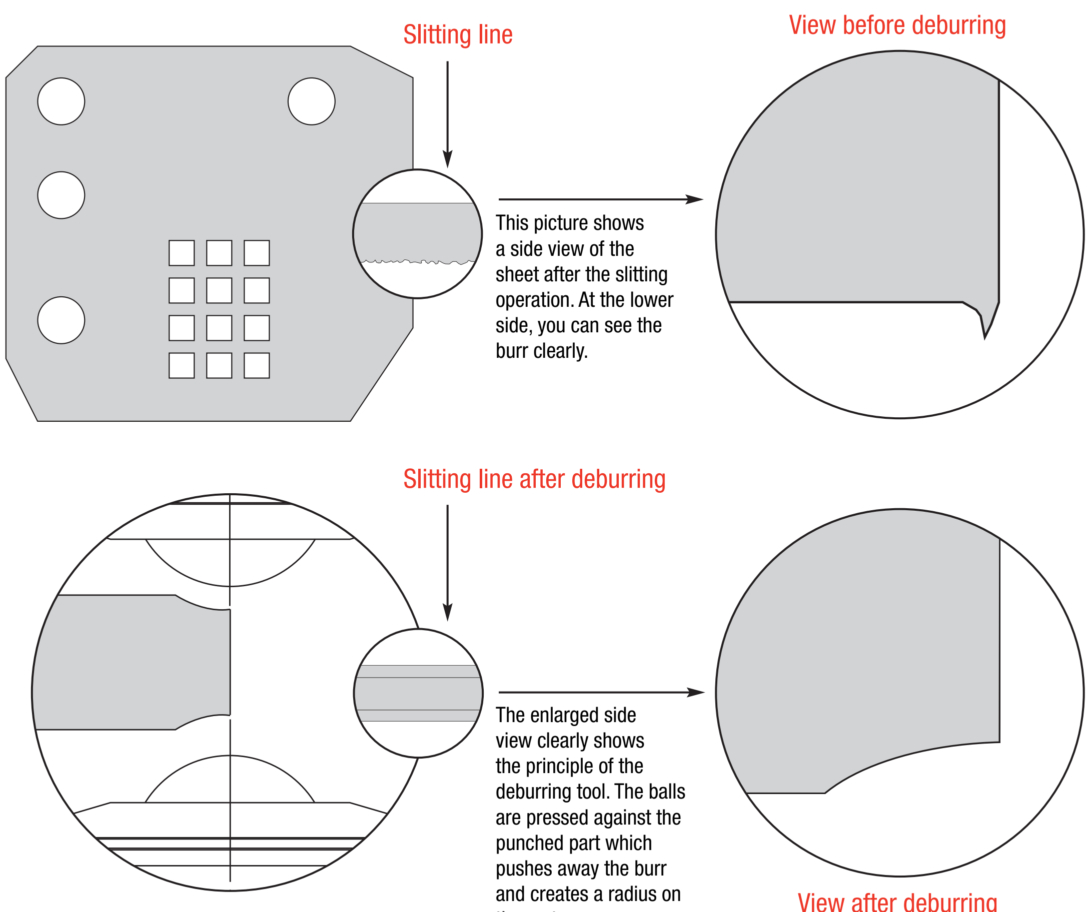
ROLLERBALL™ DEBURR

Punching processes frequently cause burrs on sheet metal parts. They are unavoidable. Removing them requires secondary deburring operations that are either performed manually or use specialized equipment. Now Mate helps you eliminate these costly secondary operations right at the turret with our Rollerball Deburr™ tool.

The Mate Rollerball Deburr tool is designed for Thick Turret, Murata Wiedemann, Turmpf and Thin Turret Strippit Style applications. This new tool takes advantage of Mate's Rollerball™ technology by using the extended programming capabilities of punch presses that can operate in the x and y axis with the ram down.

Rollerball Deburr pushes the burr away and creates a radius on the side of the part. Using a special ball in both the upper and lower part of the tool, every possible part contour can be processed - even small corners. Mate Rollerball Deburr can be used with materials of any thickness in mild steel, stainless steel and aluminum. Sold as a set, Rollerball Deburr comes complete with everything you need, including three springs and a reversible spacer, that allow you to adjust the tension appropriate to the material being punched.

Deburring sheets may reduce your OSHA recordables by reducing cuts from handling sheets with burrs.



[Dimensions in Inches (mm)]

STRAIGHTLINE DEBURR

AN ECONOMICAL WAY TO ELIMINATE BURRS AT THE TURRET

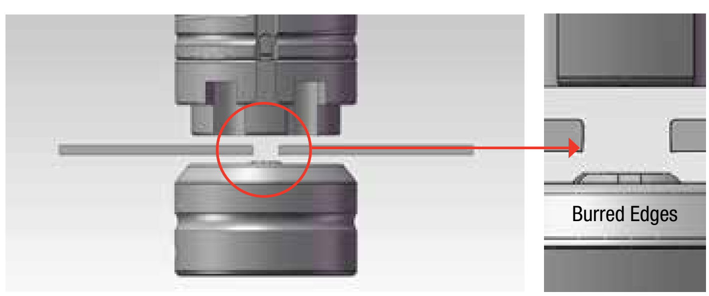
The punching process frequently causes burrs on sheet metal parts. Handling parts with these sharp edges can be hazardous for both operator and end customer. Removing burrs typically requires a secondary operation such as a straight line sander that adds time and cost to the fabricating operation. Further complicating matters are when the burrs are in tight corners, making removal even more time consuming. Mate can help you eliminate these secondary operations and improve safe material handling with a Straightline Deburr tool.

As its name implies, the Straightline Deburr tool is designed to economically deburr metal in a straight line. It's easy to set-up and program, and perfect for tight corners. The Straightline Deburr tool uses a raised area on the die that is angled at 20 degrees and coins the burr back into the material's fracture area. The 20 degree angle helps prevent sideways movement of the material or parts. Straightline Deburr is typically used in an Auto Index station or as two tools set at 90 degrees to each other.

While Straightline Deburr is not material thickness specific, it is parting tool specific. The tool should be designed to use with a specific width parting tool when deburring immediately adjacent parts. It can also be used to deburr a single side or the interior of a rectangular opening.

AVAILABLE TOOLING STYLES:
Available for all presses.

TIPS AND TECHNIQUES:
If overlapping, use the radius end for moving in a straight line and the straight end for inside corners.



Material before Straightline Deburr process

Material after Straightline Deburr process

[Dimensions in Inches (mm)]

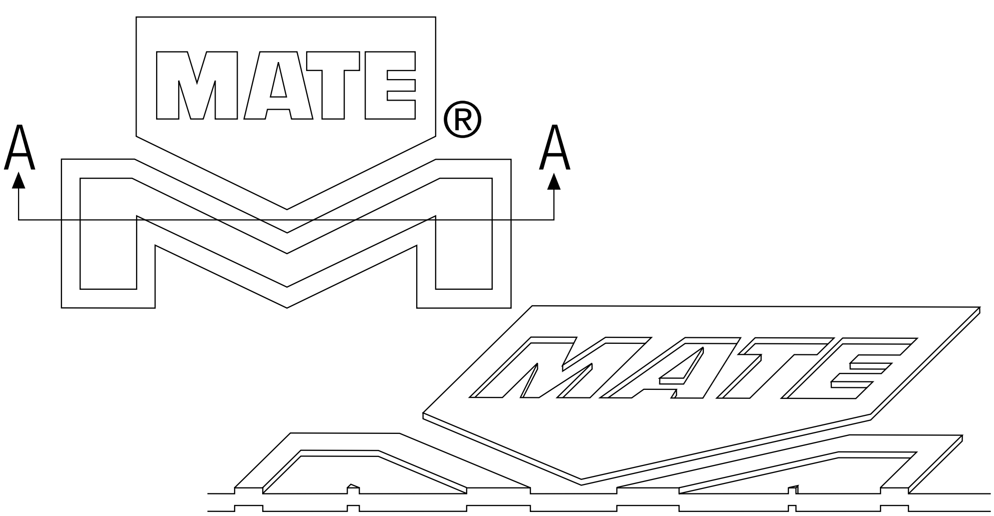
EMBOSS — COLD FORGED

Cold forged text or figures may be any size within a hypothetical circle up to the maximum emboss size of the station. When ordering, an accurate indication of each figure's size and shape is required. Text will be set in Standard Industrial Gothic. Other typefaces may be specified or artwork furnished, depending on the requirements of the subject matter.

MAXIMUM EMBOSS HEIGHT
.020(0.5)

MATERIAL THICKNESS:
Minimum .048(1.2)
Maximum .074(1.9)

HELPFUL TIPS:
- Stations adjacent to this tool should not be used
- Do not pass workholders over step die
- For best forming results, use a good forming lubricant
- F/G crossover punch and die holders available for select forming operations



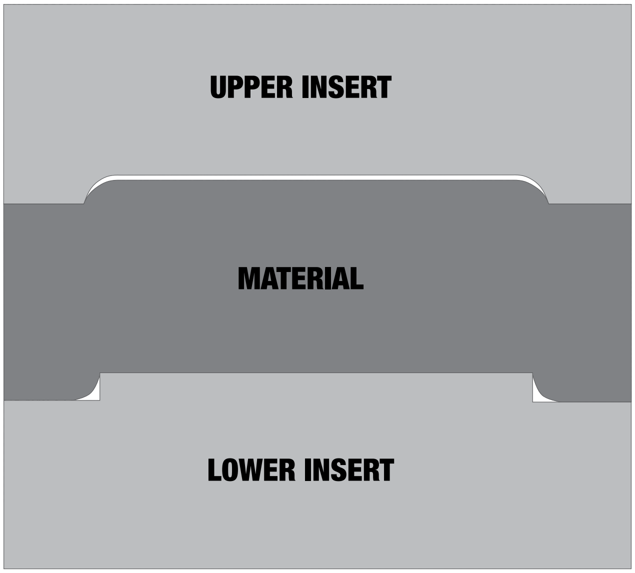
This cut-away of the Mate logo shows how a complex form is rendered by the cold forged embossing process.

[Dimensions in Inches (mm)]

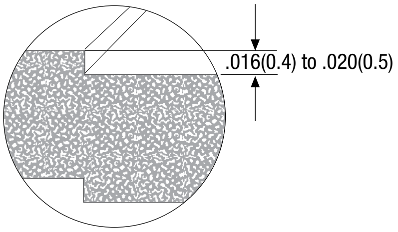
EMBOSS — COLD FORGED

A normal punch stroke shears metal and creates a slug. The enlarged view at left shows how material is forged upward by stopping the punch stroke before the metal fractures and shears off as a slug. This process can raise material up to 50% of sheet thickness depending on ductility without fracturing the material. Distinctly visible shapes and words can be formed.

Cold forged text and designs can be any size up to station maximum, see MAXIMUM FORMING LIMITS (DIAMETER) for this SPECIALS tool, page 107. When ordering, indicate each figure's size and shape accurately; or provide our engineering department a high quality line art drawing or CAD file.



For clear definition and readability, raise characters at least .016(0.4) to .020(0.5) above the surface.



TONNAGE CHART: MILD STEEL — STANDARD GOTHIC CHARACTERS			
	MATERIAL THICKNESS (T)	.018-.036(0.5-0.9)	.037-.125(0.9-3.2)
	EMBOSS HEIGHT	50%T MAXIMUM	.015-.020(0.4-0.5)
	CHARACTER HEIGHT	TONNAGE/CHARACTER	TONNAGE/CHARACTER
	.250(6.4)	1.4(1.3)	2.6(2.3)
	.375(9.5)	2.0(1.8)	3.9(3.5)
	.500(12.7)	2.7(2.4)	5.2(4.7)
	.625(15.9	3.4(3.1)	6.4(5.8)
	.750(19.1)	4.1(3.7)	7.7(6.9)
Multiply the figures above by: .6 for 5000 Series Aluminum; .7 for Brass and High Temper Aluminum; 1.0 for Bronze; 2.0 for Stainless Steel			

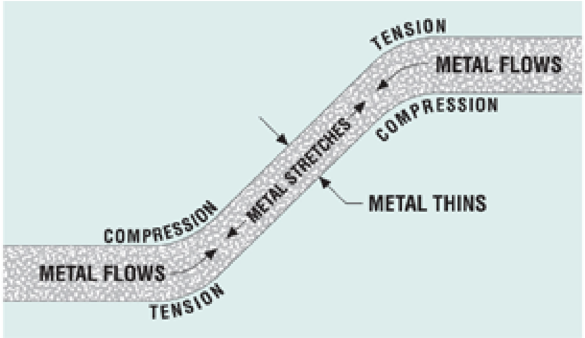
[Dimensions in Inches (mm)]

FORMED EMBOSS

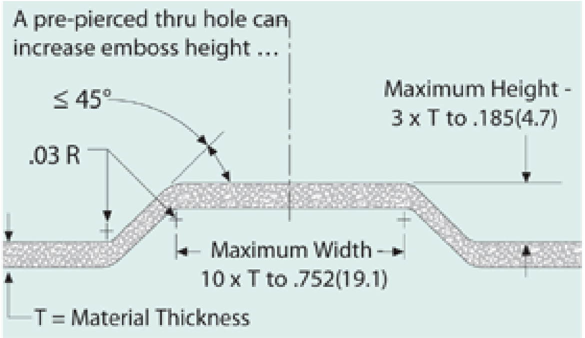
Definition
A formed emboss is created when the sheet metal is moved either above or below the surrounding sheet. A formed emboss requires material to flow and stretch while being changed into the desired shape. A successful form is dependent upon the combination of material properties and emboss features.

Features
Formed embosses are commonly used in sheet metal and vary greatly in purpose and function. Formed embosses are used to create cosmetically appealing surface treatments in architectural sheet metal panels. They are frequently used as mounting locations and stand-offs on electrical cabinets and as a sump to collect condensation inside of refrigeration units.

Process
The embossing process bends the material around the tool, compressing the inner surface and stretching the outer surface at the locations where the material changes direction. As the material flows over and around the tool material is also pulled into the formed emboss from the surrounding sheet. The amount of material being pulled into the form and the ductility of the material (its ability to be stretched) can affect the flatness of the surrounding sheet. As the form takes place, material that is within the formed emboss is stretched and thinned which creates stresses in the formed sheet that can exceed the strength of the material, resulting in failure. The most common areas for the failure to occur are at the transition points where a change in direction occurs. It follows that thicker material with high ductility will stretch more before failing than thin material with low ductility.



WHAT HAPPENS IN A FORMED EMBOSS (CROSS SECTION)



CONDITIONS FOR GOOD EMBOSSES WITH MINIMAL DISTORTION

FORMED EMBOSS

The form features also affect the success of the process. Embosses having shallow angles and large form radii will perform better than embosses with steep angles and sharp form radii. The example shown depicts emboss features that can reliably provide high quality, trouble free forms. Increasing the ratio of form height to material thickness, making the angle steeper, and/or decreasing the form radii all increase stresses in the formed material. Pre-punching a hole in the embossed area can decrease the forming stresses because it allows the material within the center of the form to be pulled outward into the formed emboss. The result is an increase to the size of the pre-punched hole resulting from the material being pulled into the form.

Summary

Successful formed embosses result from a favorable combination of material and form features. Formed embosses generally require less tonnage than is needed to punching a hole of the same size. If problems are experienced with a formed emboss bursting or the surrounding sheet warping, improved results will be achieved by following these tips and techniques:

- Decreasing the form angle
- Decreasing the form height
- Increasing the form radii
- Pre-piercing a hole in the center of the emboss
- Using more ductile material
- Use forming lubricant on the sheet

EMBOSSING NOTES

Work with the metal

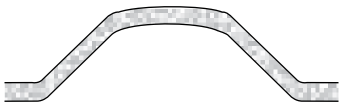
Gentle angle and generous radii perform best. When additional height and angle or less radius are required, the metal takes on added stress. A top area at least five times metal thickness helps to keep opposite sides of the emboss from stressing each other. A tall emboss may need a hole in the top for relief. A deep form may need polished metal and forming lubricant to slide over tools. A sharp angle may need to be made across the grain of the sheet to avoid cracking. If multiple embosses are required, space them at least .500(12.7) apart for tool clearance and stress distribution.

Emboss dimensions

Several dimensions are required to make a formed emboss. A normal circular emboss has diameter, height, angle of sidewall and radii of the sidewall bends. Diameter is the area inside the top plateau as measured between bend radii. For shapes other than round, specify length, width and any radii involved. Height is measured from the top surface of the sheet to the top of the plateau. With adequate sidewall angle and height, embossing forms can be any size or shape to maximum die forming limits.

Punching force

Generally, formed embosses require less punching force than a hole of the same size. Lubricant will help the metal flow over the tool surface as the emboss is formed to reduce punching force and produce a better emboss.



'DOMED' EFFECT IN EMBOSS*

Flatter Emboss Tops

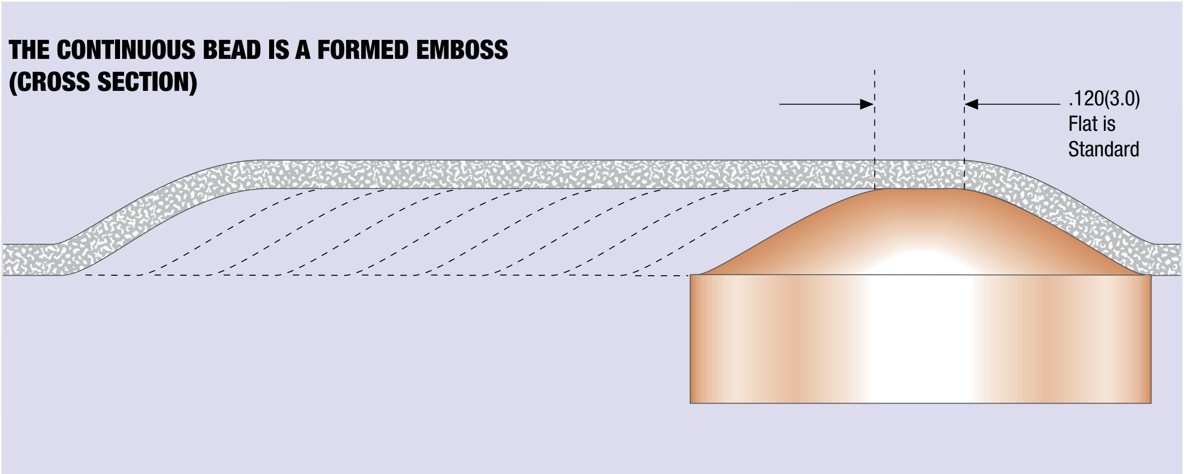
A combination of material properties and emboss proportions may cause the flat top of an emboss to arch into a domed shape after it is formed. If a domed emboss shape is unacceptable, pre-punching a hole in the top of the dome will remove much of the stress that is causing the material to arch. The form will remain much flatter across the top.

PRE-PIERCE FLATTENS TOP*

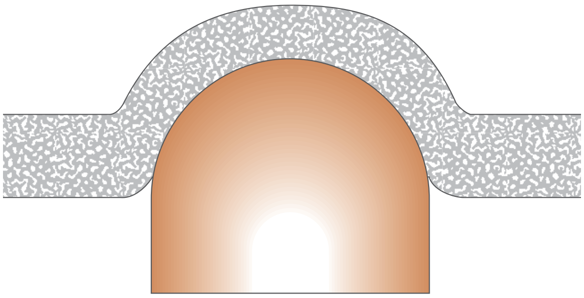
*Forms enlarged for effect

[Dimensions in Inches (mm)]

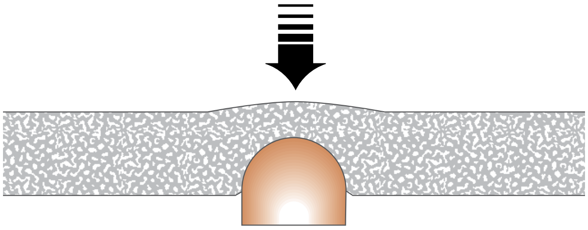
FORMED EMBOSS



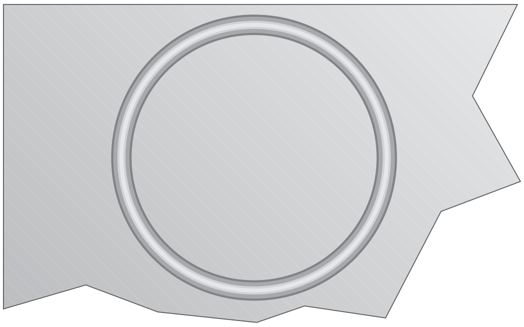
EXAMPLE 1



EXAMPLE 2



For satisfactory continuous embossing, radius of the lower unit must be at least 2 times material thickness (Example 1). A smaller radius tends to penetrate the material and hinder forming (Example 2).



AUTO INDEX stations give 360° angular freedom to segmented beading tools from making curved beads to turning down the edge of a circle.

[Dimensions in Inches (mm)]

PIERCE AND EXTRUDE

Definition

This type of extrusion is a flanged hole that is large enough to be created using a pierce and extrude process. This form is created to provide support, guide length, or soldering surface. The extruded hole is perpendicular to the sheet metal and must be formed upward to facilitate the proper evacuation of the slug created during the piercing step in the process.

Features

These extrusions are commonly used in for header plates in condensers and radiators. Copper tubing is run through the inside diameter of the form which is made to very close tolerance. The extruded hole provides additional support to thin walled copper tubing and does not have a sharp edge, like a punched hole. It also provides a large surface area for soldering contact when that is a required element in the manufacturing process.

Process

This type of extrusion, as with all extrusions, is created using a two-part process, a piercing step and a forming step. In this application both steps take place with a single hit from a pierce and extrude tool. The upper tool consists of a piercing punch that creates the proper size hole and a cavity into which the extrusion forms. The lower tool consists of a spring loaded assembly that contains a lower insert having a die opening for the pre-pierce and the O.D. of the insert creates the inside diameter of the extrusion.

The process starts with the upper tool moving downward until the piercing punch penetrates the material and punches a hole. The punched slug travels through the lower insert and exits the bottom of the lower assembly destined for the scrap bucket. As the tool continues downward the lower insert tool applies force to the bottom of the sheet and the radius on the insert enlarges the hole to the size of the desired extrusion. The extrusion is formed as the material turns into the “die” cavity from the force applied by the lower insert being pressed through the sheet. The form is completed as the radius on the lower insert clears the top of the extrusion. The upper piercing punch does not require stripping from the sheet because the material has been turned into the upper cavity and is no longer in contact with the piercing punch. The stripping forces required to remove the lower insert can be very high due to the friction from the flanged hole as it grips the punch. These stripping forces can be higher if the pre-pierced hole has a large burr because the burr will bite into the forming punch as it is pressed through the sheet. The stripping force will be decreased if a forming lubricant is applied to the sheet and/or a lubricating coating is added to the punch.

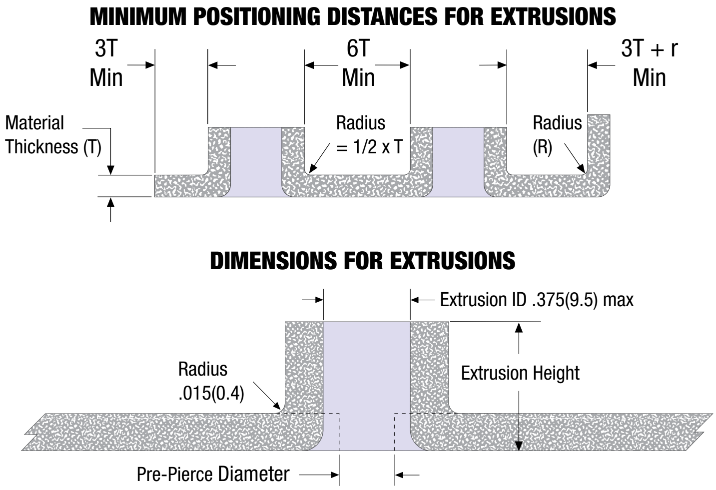
Summary

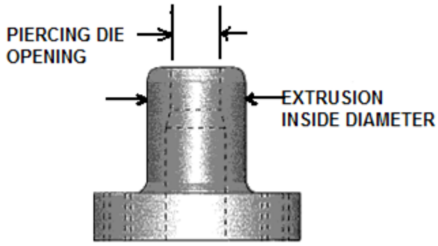
Successful extrusions result from a favorable combination of material and form features. The force required to create an extruded hole is slightly higher than the force required to punch a similar size round hole in the same material. Pierce and extrude applications are not recommended in material thicknesses greater than .075” (1,90) Mild steel. In thicker and tougher materials we recommend a two-step process where the pre-pierce and extrusion are performed with separate tools to prevent lower insert failure that may otherwise occur. The best results will be achieve by following these tips and techniques:

- Use sharp tools with proper die clearance for pierced hole (minimize burr)
- Use forming lubricant and coatings to improve stripping
- This tool is designed for a specific thickness of material

[Dimensions in Inches (mm)]

PIERCE AND EXTRUDE

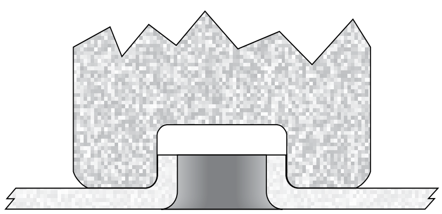




The recommended form height for a pierce and extrude application is dependent upon the lower insert which has an outside diameter for the extrusion and a hole through the center for the slug. The lower insert design becomes a thin walled cylinder that experiences tremendous forces from both piercing and forming. It is important the insert performs well for many cycles. Decreasing the form height requires a larger pre-pierce and consequently, a larger hole through the center of the lower insert. Some applications are best suited for a two-tool process to eliminate the weakness of the lower insert by pre-piercing a round hole with another tool. Extrusions can become deformed if they are placed too close to sheet edges, punched holes, or other forms including extrusions. Recommended minimum positioning distances between extrusions and other part features are shown below.

If Your Stainless Steel Extrusions Are Distorted

Apply a good forming lubricant to the material before making the extrusion. Not only will the material release from the die better, it slides over the die surface smoothly when being formed. This gives the material a better opportunity to distribute the forces of bending and stretching, preventing distortion in the formed wall and tearing at the root of the extrusion.



EXTRUSION AND PRE-PIERCE DIAMETERS

Pierce and Extrude Assemblies are made to the size your application requires. As an example of a typical application, here are sizes used in fabricating supports for refrigeration coils.

COIL TUBING DIAMTER	MATERIAL (MILD STEEL)	PIERCING PUNCH DIAMETER	EXTRUDE DIAMETER	PIERCING DIE I.D.	EXTRUSION HEIGHT
.392 (10.0)	.071 (1.8) .058 (1.5) .046 (1.2)	.248 (6.3) .248 (6.3) .248 (6.3)	.396 (10.1) .396 (10.1) .396 (10.1)	.262 (6.7) .260 (6.6) .258 (6.6)	.166 (4.8)
.530 (13.5)	.071 (1.8) .058 (1.5) .046 (1.2)	.373 (9.5) .373 (9.5) .373 (9.5)	.533 (13.5) .533 (13.5) .533 (13.5)	.387 (9.8) .385 (9.8) .383 (9.7)	.250 (6.4)
.658 (16.7)	.071 (1.8) .058 (1.5) .046 (1.2)	.502 (12.8) .502 (12.8) .502 (12.8)	.660 (16.8) .660 (16.8) .660 (16.8)	.516 (13.1) .514 (13.1) .512 (13.0)	.250 (6.4

[Dimensions in Inches (mm)]

TAPPING EXTRUSION

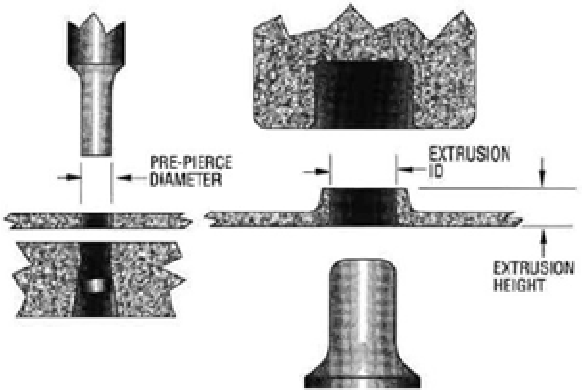
Definition

A tapping extrusion is a flanged hole created to provide increased threads of engagement for a tapped hole or for a thread cutting screw than would be available within the thickness of the sheet metal. The formed material can be extruded up or down and the inside diameter of the extrusion is perpendicular to the sheet metal.

Features

Screws used in extruded holes can develop up to twice the holding strength as when used in punched holes. The holding strength will be maximized if rolled thread taps and rolled thread screws are used. Extrusions can also be used with cut thread taps and thread cutting screws but the strength of the extrusion may be compromised as this process removes material from the wall thickness of the extruded hole.

Process



A tapping extrusion is created using a two-part process. First a round hole is pre-pierced at the location of the extrusion. The size of the pre-pierced hole is dependent upon the extrusion inside diameter, the desired form height, and the material being fabricated. The pre-pierced hole provides the correct amount of material to be formed by the extrusion tool. The second step is to use the tapping extrusion tool to form the material perpendicular to the sheet. The machine applies force to a "punch" that has a radius on the point and enlarges the hole to the size of the desired extrusion id. The extrusion is formed as the material turns into the "die" cavity as the punch continues through the sheet. When the punch has traveled far enough to finish creating the extruded hole the punch is removed from the inside of the extruded hole. The stripping forces required to remove the punch can be very high due to the friction from the flanged hole as it grips the punch. These stripping forces can be even higher if the pre-pierced hole has a large burr because the burr will bite into the forming punch as it is pressed through the sheet. The stripping force will be decreased if a forming lubricant is applied to the sheet and/or a lubricating coating is added to the punch.

The recommended maximum form height for a tapping extrusion is 2.5 times the material thickness from the bottom of the sheet to the top of the form. Exceeding this height can result in tearing at the top of the extrusion. Extrusions can become deformed if they are placed too close to sheet edges, punched holes, or other forms including extrusions. Recommended minimum positioning distances between extrusions and other part features are shown on the previous page.

Summary

Successful tapping extrusions result from a favorable combination of material and form features. The force required to create an extruded hole is slightly higher than the force required to punch a similar size round hole in the same material. Tapping extrusions are not recommended in material thicknesses greater than .105" (2,67) Mild steel. In thicker materials the most threads for tapping will be achieved by punching and shaving a hole. The best results with tapping extrusion will be achieved by following these tips and techniques:

- Use sharp tools with proper die clearance for pre-pierced hole (minimize burr)
- Use forming lubricant and coatings to improve stripping
- Keep the maximum form height (bottom to top) to 2.5 times the material thickness

[Dimensions in Inches (mm)]

TAPPING EXTRUSION

Extrusion notes

Prior to making an extrusion, the workpiece is pierced with a standard punch and die. This pre-pierced hole permits the metal to flow uniformly in the extrusion process. To develop a normal extrusion two-and-a-half times as high as material thickness, the diameter of the pre-pierced hole will be about 65% of the diameter of the finished extrusion.

Screws in extruded holes develop nearly twice the holding power as in non-extruded holes. Maximum strength is obtained with thread rolling screws. Taps or thread cutting screws remove some of the extrusion wall, making it weaker. Extrusions in material thinner than 20% of screw diameter do not develop holding power in proportion to screw strength. Material thicker than screw diameter will develop enough holding power to break the screw even in holes that are not extruded.

Extrusions can become distorted if placed too close to edges, bends, or other extrusions. Minimum distance between the edge of an extrusion and the inside edge of a bend should be three times material thickness plus the inside radius of bend, but not less than .032(0.8) plus inside bend radius. Minimum distance between extrusions and metal edge should be at least three times material thickness. Minimum distance between extruded holes should not be less than six times material thickness.

EXTRUSION AND PRE-PIERCE DIAMETERS

TAP OR SCREW SIZE	EXTRUSION I.D.		PRE-PIERCE DIAMETER		MAXIMUM MATERIAL THICKNESS
	CUT THREAD	ROLLED THREAD	CUT THREAD	ROLLED THREAD	
	STANDARD SIZES				
1/4-20	.089(2.3)	.100(2.5)	.050(1.3)	.060(1.5)	.048(1.2)
#5-40	.100(2.5)	.112(2.8)	.060(1.5)	.072(1.8)	.060(1.5)
#6-32	.107(2.7)	.120(3.0)	.070(1.8)	.076(1.9)	.075(1.9)
#8-32	.136(3.5)	.150(3.8)	.085(2.2)	.093(2.4)	.075(1.9)
#10-24	.150(3.8)	.167(4.2)	.090(2.3)	.100(2.5)	.090(2.3)
#10-32	.159(4.0)	.174(4.4)	.095(2.4)	.104(2.6)	.090(2.3)
#12-24	.173(4.4)	.194(4.9)	.104(2.6)	.116(2.9)	.090(2.3)
#4-40	.201(5.1)	.219(5.6)	.121(3.1)	.131(3.3)	.105(2.7)
1/4-28	.218(5.5)	.235(6.0)	.131(3.3)	.141(3.6)	.105(2.7)
5/16-18	.257(6.5)	.275(7.0)	.154(3.9)	.165(4.2)	.105(2.7)
5/16-24	.272(6.9)	.288(7.3)	.163(4.1)	.172(4.4)	.105(2.7)
3/8-16	.312(7.9)	.343(8.7)	.187(4.8)	.206(5.2)	.105(2.7)
3/8-24	.332(8.4)	.343(8.7)	.199(5.1)	.206(5.2)	.105(2.7)
	METRIC SIZES				
M3	.098(2.5)	.108(2.7)	.059(1.5)	.065(1.6)	.060(1.5)
M4	.130(3.3)	.146(3.7)	.078(2.0)	.088(2.2)	.075(1.9)
M5	.165(4.2)	.183(4.6)	.099(2.5)	.110(2.8)	.090(2.3)
M6	.197(5.0)	.216(5.5)	.118(3.0)	.130(3.3)	.105(2.7)
M8	.266(6.8)	.293(7.4)	.160(4.1)	.176(4.5)	.105(2.7)
M10	.334(8.5)	.369(9.4)	.200(5.1)	.221(5.6)	.105(2.7)

[Dimensions in Inches (mm)]

JOINING MATERIALS: HINGE TOOL

Hinges are a more complex form that may be processed in a punch press. By using a hinge tool on the punch press, fabricators can form many different types, lengths and diameters of knuckles. Fabricated metal enclosures and similar cabinetry that require hinges can be fabricated with integral hinge knuckles. This eliminates secondary operations to attach hinge components and can eliminate the need for separate hinges, fasteners, spot welds or assembly operations. Forming hinges in a punch press could eliminate the cost of specialized press brake or stamping tooling necessary to create hinge forms. Forms created in a punch press can also ensure accuracy over more manual secondary operations.

PROCESS

Forming a successful hinge in a punch press is a process that typically involves two forming tools—Forming Tool 1 and Forming Tool 2 (also referred to as the "knuckle tool" — and three forming strokes. These forming tools are needed in addition to standard punching tools used to create the tabs for forming.

- After the tabs have been punched in the sheet to be formed, Forming Tool #1 is used to make the first 2 forming strokes. This first form produces the leading edge of the tab that will slide around the interior of the second tool during the final forming stroke. The first form is placed in all of the tabs to be formed in a part or a sheet of nested parts.
- The second forming stroke is added to all tabs to be formed. This forming stroke bends the tab up to a designed angle, typically between 75 and 88 degrees. The stroke depth for these first two forming strokes is identical. The spring loaded lower assembly, or die, pushes the sheet off the lower insert before the sheet advances to the next forming location.
- The final forming stroke uses Forming Tool #2, or the "knuckle tool." As the knuckle tool lowers down towards the sheet metal tab, the raised tab enters the upper tool, making contact near the front (or lower) edge of the tool. As the tool continues to descend, the tab is forced to slide around the perimeter of the upper tool, curling around to form a hinge. This tool also includes a spring loaded lower assembly to lift the material off the lower insert before advancing to the next forming location.

MATERIAL AND OTHER RESTRICTIONS:

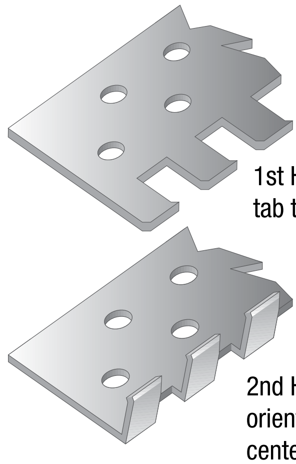
- Hinges can be formed in mild steel, aluminum and stainless steel
- Thicknesses can range from 0.030"(0,80 mm) to 0.060"(1,50 mm)* incorporating pin diameters from 0.062"(1,60 mm) to 0.188"(4,77 mm)**:**
- Material thickness restrictions are determined by pin and knuckle diameter.
- Hinge will be designed for a pin clearance of .001-.006(0.03-0.15) for the inside diameter.
- Maximum Knuckle diameter stated for material thickness in cold rolled steel.
- .050(1.3) maximum material thickness for stainless steel.

* Thickness of material can possibly exceed stated range - consult with a Mate Application Specialist for more information.

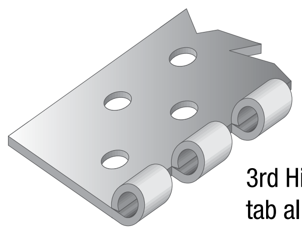
[Dimensions in Inches (mm)]

JOINING MATERIALS: HINGE TOOL (CONT.)

TOOL #1



TOOL #2



TYPICAL PIECE PART LAYOUT

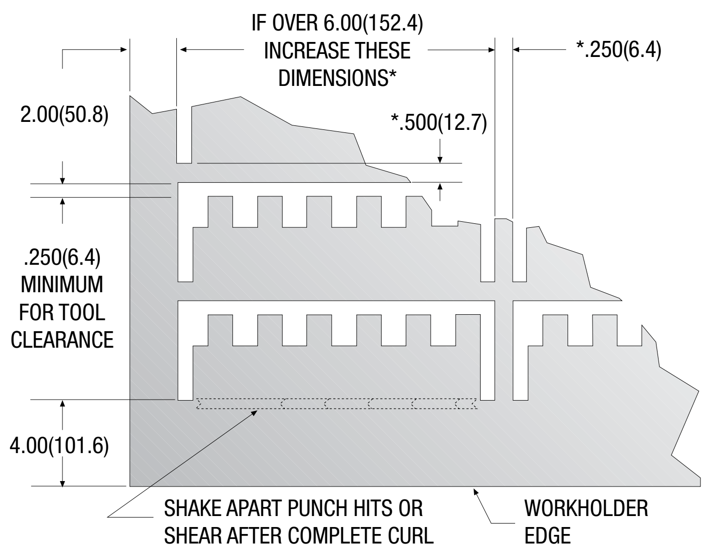
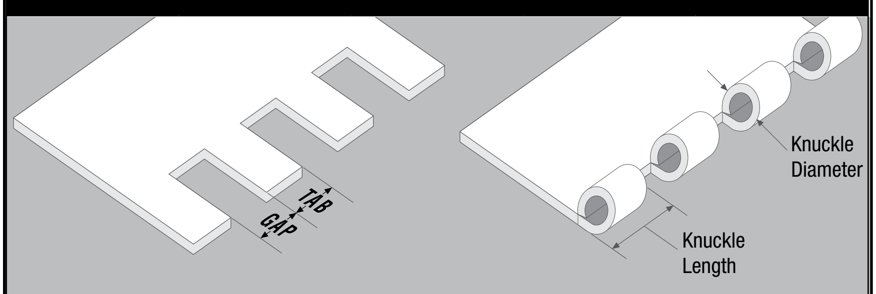


CHART FOR DETERMINING KNUCKLE DIAMETER FOR GIVEN PIN DIAMETER
KNUCKLE DIAMETER TOLERANCE — .005(0,13) + .010(0,25)



Material Thickness	Knuckle Diameter for .062(1.6) Pin Diameter	Knuckle Diameter for .094(2.4) Pin Diameter	Knuckle Diameter for .125(3.2) Pin Diameter	Knuckle Diameter for.156(4.0) Pin Diameter
.030(0.8)	.122(3.1)	.154(3.9)	.185(4.7)	.216(5.5)
.036(0.9)	.134(3.4)	.166(4.2)	.197(5.0)	.228(5.8)
.048(1.2)	.144(3.7)	.190(4.8)	.221(5.6)	.252(6.4)
.060(1.5)	.182(4.6)	.214(5.4)	.245(6.2)	.276(7.0)

[Dimensions in Inches (mm)]

JOINING MATERIALS: HEXLOCK™ TOOL

Mate HexLock™ is a forming assembly engineered to provide a reliable and secure method of retaining common threaded fasteners in sheet metal.

Mate HexLock is an inverted spring loaded triple lance and form assembly. The geometry of the form securely retains hexagon headed fasteners prior to assembly. The robust tool construction includes: highly wear-resistance steel cutting edges for extended service life, positive upper and lower stripper mechanisms for reliable operation and fully interchangeable replacement parts for maximum convenience.

Mate HexLock is fully compatible with DIN933 hexagon head bolts and DIN934 hexagon nuts in M5, M6 and M8 thread sizes. Benefits of Mate HexLock include:

Versatility
Mate HexLock accepts a hexagon nut or the head of a hexagon bolt. A nut allows items to be assembled with a bolt from the opposite side of the component. Alternatively the head of a bolt can be installed, with the thread protruding through the sheet metal.

Reliability
Mate HexLock uses the CNC precision of the press to assure accurate and reliable fastener location.

Security
The fastener is securely gripped between the forms to prevent rotation. The head of the fastener is gripped more tightly as torque is applied to the fastener from the opposite side of the component.

Economy
Conventional hexagon fasteners are less expensive and more readily available than self-clinching fasteners or weld-nuts. They are easily installed without the need for costly secondary operations or special installation equipment.

Mate HexLock is available to suit metric or inch threads and many popular tool styles. Please contact our Customer Service Engineers to discuss your next application.

WATCH THE VIDEO:
Watch an animation of a HexLock:

https://youtu.be/oy4QipCtrpk

[Dimensions in Inches (mm)]

JOINING MATERIALS: THREAD FORM

Screw holding threads are pierced and formed in one operation with this assembly. This is a very good option to fasten two pieces of material together without the use of nuts or locking washers.

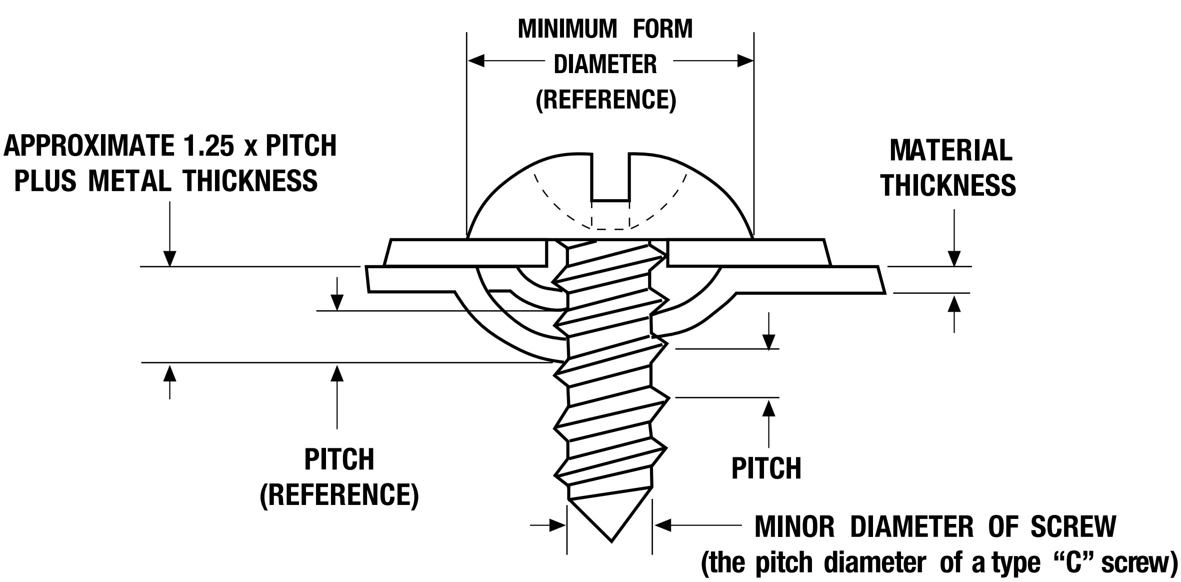
When ordering thread forming tools:

- Specify screw type and size
- Material type and thickness

Material thickness:
Minimum .048(1.2)
Maximum .074(1.9)

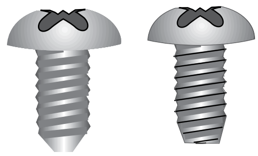
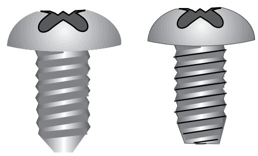
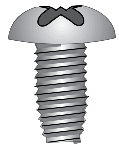
Tips and Techniques:

- Set punch holder to shut height of press minus material thickness.
- Stations adjacent to this tool should not be used
- Remove fixed machine stripper on machines so equipped if clearance is needed for forming height and/or step die installation
- Do not pass workholders over step die
- For best forming results, use a good forming lubricant



[Dimensions in Inches (mm)]

JOINING MATERIALS: THREAD FORM

	Thread Size	Material Thickness	Pitch (Reference)	Maximum Minor Diameter of Screw	Diameter
	TYPE “A” SCREWS - DIMENSIONS IN INCHES				
A 	6x18	.020-.030	.0556	.102	.276
	8x15	.025-.040	.0667	.123	.306
	10x12	.030-.045	.0833	.133	.352
	12x11	.035-.054	.0909	.162	.406
	14x10	.038-.060	.1000	.185	.442
	20x90	.050-.060	.1111	.234	.556
	TYPE “B” AND “AB” SCREWS - DIMENSIONS IN INCHES				
AB B 	6x20	.020-.030	.0500	.104	.208
	8x18	.020-.030	.0566	.122	.296
	10x16	.020-.030	.0625	.141	.374
	12x14	.030-.040	.0714	.164	.400
	1/4x14	.030-.040	.0714	.192	.400
	5/16x12	.030-.040	.0833	.244	.552
	TYPE “C” SCREWS (MACHINE SCREW) - DIMENSIONS IN INCHES				
C 	6x32	.010-.020	.0313	.118	.290
	8x32	.010-.020	.0313	.144	.324
	10x24	.015-.025	.0417	.163	.370
	10x32	.010-.020	.0313	.170	.348
	1/4x20	.020-.030	.0500	.218	.478
	5/16x12	.020-.030	.0556	.276	.490
	METRIC SHEET METAL SCREWS - DIMENSIONS IN MILLIMETERS				
	2.9x1	.38-.63	1.10	2.18	5.6
	3.5x1.3	.50-.70	1.30	2.64	6.9
	3.9x1.4	.50-.80	1.40	2.92	7.5
	4.2x1.4	.50-.80	1.40	3.10	8.2
	4.8x1.6	.60-1.0	1.60	3.58	9.5
	5.5x1.8	.75-1.0	1.80	4.17	10.8
	6.3x1.8	.75-1.0	1.80	4.88	12.5
	8.0x2.1	.80-1.2	2.10	6.20	13.9
	METRIC MACHINE SCREWS - DIMENSIONS IN MILLIMETERS				
	3.5x0.6	.20-.38	0.60	3.09	7.0
	4.0x0.7	.25-.45	0.70	3.52	8.0
	5.0x0.8	.25-.50	0.80	4.46	10.0
	6.0x1.0	.38-.63	1.00	5.32	12.0
	8.0x1.25	.50-.75	1.25	7.16	16.0

[Dimensions in Inches (mm)]

JOINING MATERIALS: HYBRID THREAD FORM

In many industries, there's a need to join two pieces of material using a threaded machine screw. If the thread pitch is greater than the material thickness, then a conventional threadform tool is a great solution. But what if the screw thread to join the two pieces requires a material thickness greater then the pitch of the screw?

Other fastening methods, such as installing a self clinching fastener, using a tapping extrusion or self tapping screw, add expensive secondary operations or special hardware. Is there a cost-effective way to address the issue, especially in a highly competitive industry?

Mate can solve this problem with the Hybrid Threadform forming tool. Unlike a conventional threadform tool, the hybrid threadform tool is designed to thin the material in the center of the form, and create the threadform helix in just one operation.

At the heart of the tool is an upper and lower insert that is machined with a profile that precisely matches the thread helix of the screw thread. The geometry of this part is modeled electronically by a Mate Applications Specialist using our advanced CAD software, creating a three-dimensional solid model for your specific requirements.

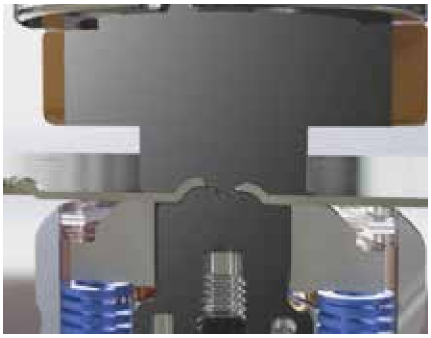


Figure 1: Cross sectional view of the sheet metal, produced using the hybrid threadform tool. It shows the material has been thinned as the threadform helix is created so the material fits between the pitch of the screw thread.

The Hybrid Threadform forming tool provides the following benefits:

- Eliminates secondary operations
- Eliminates tapping operations
- Reduces debris in the machine caused by tapping
- Reduces component cost by eliminating any special fasteners

AVAILABLE TOOLING STYLES:

- Thick Turret
- Thin Turret
- Trumpf Style
- Murata Wiedemann
- Salvagnini

STATION SIZES:

- Thick Turret B-Station and up
- Trumpf Style, Size 2 and up
- Murata Wiedemann, D-Station and up
- Salvagnini, C-Station and up
- Thin Turret, B-Station and up

Note: Smaller sizes may be possible. Contact your Mate Applications Specialist to determine viability.

WATCH THE VIDEO:

Watch an animation of a Hybrid Thread Form:

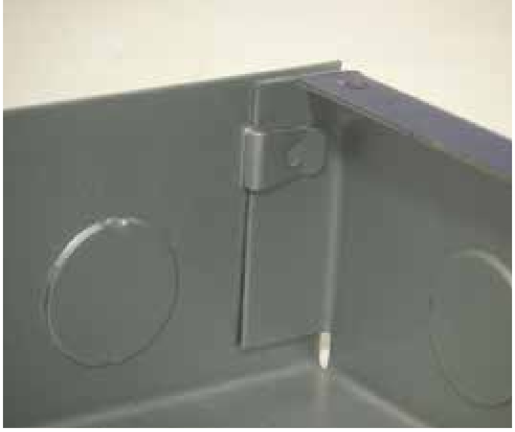
https://youtu.be/78OvkYrRN4w

MATERIAL RESTRICTIONS:

Contact a Mate Sales Engineer or Application Specialist to determine viability.

[Dimensions in Inches (mm)]

JOINING MATERIALS: SNAPLOCK™



Definition

A snaplock™ lance and form is a form inside a form, created when sheet metal is cut and formed both up and down simultaneously. The design of the snaplock lance and form provides a self-locking, spring-loaded metal tab that snaps securely into a pre-punched hole. A successful snaplock lance and form is dependent upon a favorable combination of material properties and form features.

Features

The snaplock lance and form can be used to replace conventional fasteners and eliminates secondary processes such as spot welding. This form allows you to create fabricated assemblies efficiently and cost effectively with CNC precision. The snaplock lance and form can be customized to the customer's application requirements.

Process

The snaplock process consists of a snaplock lance and form which has a small button within the larger form. This button is semi-sheared to provide a positive mechanical lock when engaged in the hole of the receiving part. The snaplock lance and form tool uses a single tool to cut and form both features. The snaplock lance and form can be configured to receive a material that is a different thickness that that being lanced. Dissimilar material types can be joined such as stainless steel and aluminum, which cannot be joined together by spot welding. The functional requirements of the sheet metal assembly will help determine the best form size and the number of forms needed per joined area. A sheet metal product development effort will be needed to determine the ideal combination. The best results are achieved when this type of form is made upward in the sheet as downward forms have exposed edges that can easily catch in the press and become damaged in processing.

Thick material and tough abrasive material are more challenging to form and will accelerate the wear on cutting edges. In these circumstances a snaplock lance and form may not be the best tool for the job. Instead, the snaplock fastening principle can be achieved using a two-step process. Pre-piercing around the area then "embossing" the form into the desired shape may help achieve difficult applications in thick abrasive material.

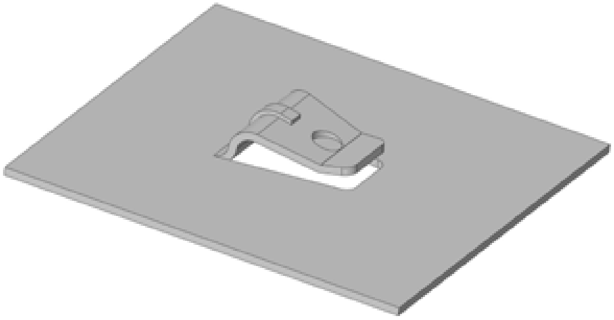
Summary

Successful snaplock lance and forms result from a favorable combination of material and form features. These applications generally require less tonnage than is needed for punching a hole having a perimeter equal to the cutting length of the lance and form shape. If problems are experienced with a snaplock lance and form sticking, creating a burr, or becoming damaged in processing, improved results will be achieved by following these tips and techniques:

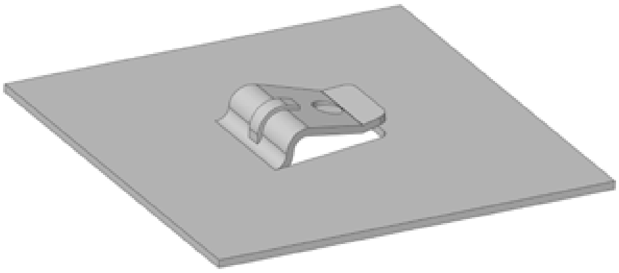
- Use forming lubricant on the sheet
- Replace the cutting components when they become dull
- Confirm the tool was designed for the material type and thickness being formed
- Form up whenever possible

[Dimensions in Inches (mm)]

JOINING MATERIALS: SNAPLOCK™ WITH REINFORCED TAB



Fabricators use fasteners (screws, rivets) or welds to join parts. These secondary operations and materials add cost to the job. Joining parts doesn't necessarily require fasteners or welds. Mate recommends using a SnapLock™ tool to lower manufacturing costs and speed delivery time. SnapLock eliminates expensive secondary operations such as spot welding, riveting or fastening with threaded hardware. With SnapLock, fabricated assemblies can be created efficiently and effectively with CNC precision.



SnapLock works great in thicker — 16 gauge (1,50mm) or greater. For thinner materials, Mate's SnapLock™ with Reinforced Tab is for just such a purpose. The SnapLock with Reinforced Tab allows fabricators to join parts made from thinner—16 gauge (1,50mm) or thinner—materials, even if they are of dissimilar type and/or thickness. The reinforced tab incorporates a stiffening rib for greater integrity. In fact, a stiffening rib may be added to all lance and form applications for use with thinner materials

SnapLock is a Lance and Form assembly that provides a self-locking, spring loaded tab that snaps securely into a pre-punched hole. When engaged with a hole in the receiving tab, the small button provides a positive mechanical lock. It can also be removed, if needed. Benefits include:

- Precision joint placement with CNC accuracy
- Assembly done in press brake
- Parts attached by hand in assembly
- Form placement can be inside or outside
- Eliminates or minimizes fasteners
- Joins dissimilar material types, such as stainless steel and aluminum
- Joins dissimilar material thicknesses
- Eliminates expensive secondary operations
- Fabricate and assemble pre-painted material

[Dimensions in Inches (mm)]

KNOCKOUT

Definition

A knockout is a feature in sheet metal that is created by punching a slug free of the sheet but keeping that slug attached to the sheet with small un-cut areas called tabs. A knockout allows for variation in the use and function of a fabricated product as it provides the opportunity to create a hole in the part at a later time by removing, or knocking out, the retained slug. A successful knockout is dependent upon a favorable combination of material properties, tab sizes, and tab location.

Features

Knockouts are common in sheet metal fabrication and provide a variety of options for features and function of the fabricated product. Most frequently knockouts are used in electrical enclosures as an access point to bring wires into the enclosure. A very important feature of a knockout is the tabs, which keep the knockout attached to the sheet. The tab size, quantity, and location are critical to determining the ease or difficulty with which the knockout is removed. The ease of knockout removal is subjective and determined by the functional requirements of the product manufacturer or field installation personnel who will be removing the knockout. There is a fine line between a "fall out" and a "beat out" and that is determined by the tab specifications. Knockouts can be formed up or down with the direction determined by the finished product requirements and manufacturing method.

Process

The knockout punching process uses a single tool to create a slug and the tabs. It is common for the slug to be displaced slightly more than one material thickness to ensure it has been cut free from the sheet. The tabs are stretched and weakened when the slug is displaced. The ductility of the material (its ability to be stretched) and the tab size, quantity, and location can affect the success of the desired result. Small tabs in thicker material may not be able to stretch enough to keep the slug attached to the sheet, resulting in the tab failing and the knockout breaking free. Precise, consistent stroke control in the press is very important for producing high quality, consistent knockouts. Variations in the stroke of the press and changes in the material being punched will result in knockouts that are inconsistent in height and tab strength.

It is common for knockouts to be pressed back into the sheet (planished) to create a closed feature. Although this flattening process will not press a knockout completely flat back into the sheet it does prevent dust intrusion into the enclosure. Planishing knockouts introduces stresses into the sheet that may result in a slight bowing of the knockout and/or the surrounding sheet.

Summary

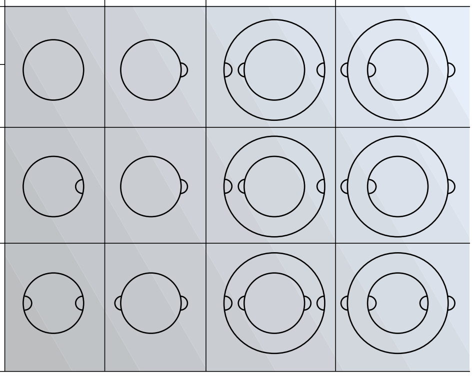
Successful knockouts result from a favorable combination of material properties and tab features. The force required to produce a knockout is essentially the same as that required for punching a hole. If problems are experienced with a knockout sticking, creating a burr, tabs failing, difficulty removing the knockout, or the form becoming damaged in processing, improved results will be achieved by following these tips and techniques:

- Use forming lubricant on the sheet
- Confirm the tool was designed for the material thickness being formed
- Sharpen or replace the cutting components when they become dull
- Review tab sizes, locations, and quantities
- Confirm the form height is correct (1 to 1.1 x the material thickness)
- Form up whenever possible

[Dimensions in Inches (mm)]

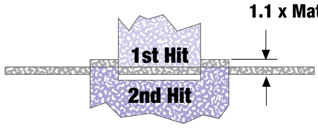
KNOCKOUT

INDUSTRY TAB LOCATIONS INDUSTRY TAB LOCATIONS WILL BE FURNISHED UNLESS OTHERWISE SPECIFIED				
ACTUAL DIAMETER	SINGLE KO LOWER	SINGLE KO UPPER	DOUBLE KO LOWER	DOUBLE KO UPPER
*0 -.249 (0 - 6.3)				
.250 -.313 (6.4 - 8.0)				
.314 -1.374 (8.0 - 34.9)				
> 1.375 (34.9)				



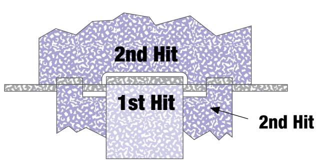
* Tab needed on lower K.O. if material is thicker than .063(1.6).

SPLITTING PUNCHING FORCE WITH TWO HITS ...



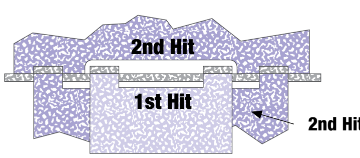
DOUBLES

If punching force is over capacity, make first hit single K.O. down, second hit single K.O. up with relief.



TRIPLES

Make first hit single K.O. up, second hit double K.O. up with relief.



QUADS

Make first hit double K.O. up, second hit double K.O. up with relief.

THICKNESS VARIATION

A knockout tool assembly accommodates a ± .016(0.4) range in material thickness. Beyond ± .016(0.4), penetration is affected and knockout performance deteriorates.

PLANISHING

The planishing punch pushes knockout back to 75% material thickness, leaving 25% still raised. Further planishing makes the knockout difficult to remove without distorting the sheet. Satisfactory knockouts can be produced without planishing.

CUSTOM KNOCKOUTS

Custom knockouts are available to suit your application; rectangular knockouts for circuit boxes, oval knockouts for locking cylinders, small knockouts for screw holes or slots, etc.

[Dimensions in Inches (mm)]

KNOCKOUT

PUNCHING FORCE CHART
SINGLE KNOCKOUT, MILD STEEL

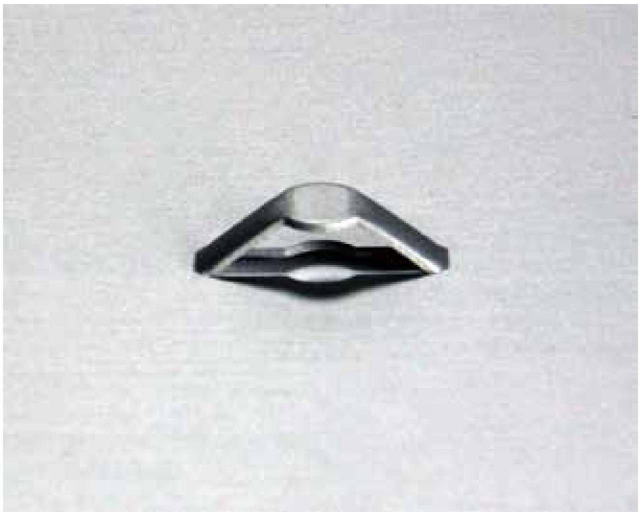
REF CONDUIT SIZE IN INCHES	ACTUAL DIAMETER	MATERIAL THICKNESS					
		20 Ga. .035(0.9)	18 Ga.. .047(1.2)	16 Ga. .060(1.5)	14 Ga. .075(1.9)	12 Ga. .105(2.7)	11 Ga. .120(3.0)
	.375 (9.5)	1.0 tons 8.9 kN	1.4 tons 12.3 kN	1.8 tons 15.7 kN	2.2 tons 19.7 kN	3.1 tons 27.5 kN	3.5 tons 3l.4 kN
	.500 (12.7)	1.4 tons 12.3 kN	1.8 tons 16.4 kN	2.4 tons 21.0 kN	2.9 tons 26.2 kN	4.1 tons 36.7 kN	4.7 tons 41.9 kN
3/8	.687 (17.5)	1.9 tons 16.8 kN	2.5 tons 22.6 kN	3.2 tons 28.8 kN	4.0 tons 36.0 kN	5.7 tons 50.4 kN	6.5 tons 57.6 kN
	.750 (19.0)	2.1 tons 18.3 kN	2.8 tons 24.6 kN	3.5 tons 31.4 kN	4.4 tons 39.3 kN	6.2 tons 55.0 kN	7.1 tons 62.9 kN
1/2	.875 (22.2)	2.4 tons 21.4 kN	3.2 tons 28.7 kN	4.1 tons 36.7 kN	5.2 tons 45.9 kN	7.2 tons 64.2 kN	8.2 tons 73.4 kN
	1.000 (25.4)	2.7 tons 24.5 kN	3.7 tons 32.8 kN	4.7 tons 41.9 kN	5.9 tons 52.4 kN	8.2 tons 73.4 kN	9.4 tons 83.8 kN
3/4	1.125 (28.6)	3.1 tons 27.5 kN	4.2 tons 36.9 kN	5.3 tons 47.2 kN	6.6 tons 59.0 kN	9.3 tons 82.5 kN	10.6 tons 94.3 kN
	1.250 (31.8)	3.4 tons 30.6 kN	4.6 tons 41.0 kN	5.9 tons 52.4 kN	7.4 tons 65.5 kN	10.3 tons 91.7 kN	11.8 tons 104.8 kN
1	1.375 (34.9)	3.8 tons 33.6 kN	5.1 tons 45.2 kN	6.5 tons 57.6 kN	8.1 tons 72.1 kN	11.3 tons 100.9 kN	13.0 tons 115.3 kN
	1.500 (38.1)	4.1 tons 36.7 kN	5.5 tons 49.3 kN	7.1 tons 62.9 kN	8.8 tons 78.6 kN	12.4 tons 110.0 kN	14.1 tons 125.8 kN
	1.625 (41.3)	4.5 tons 39.7 kN	6.0 tons 53.4 kN	7.7 tons 68.1 kN	9.6 tons 85.2 kN	13.4 tons 119.2 kN	15.3 tons 136.2 kN

WATCH THE VIDEO:
Watch an animation of a Double Knockout:

https://youtu.be/c7N08FSKMZ8

[Dimensions in Inches (mm)]

LANCE AND FORM



Definition

A lance and form is created when sheet metal is cut (lanced) and formed simultaneously. Lance and form applications cover a wide range of function and appearance but all are cut and formed into the desired shape with the same tool. A successful lance and form is dependent upon a favorable combination of material properties and form features.

Features

Lance and forms are common in sheet metal fabrication and vary greatly in features and function. They can be used to create simple to complex mechanical connections points for other components, as positioning locators. Some lance and form types can be used to replace threaded fasteners or as mounting devices for assemblies. There are two general types of lance and forms, an open-end form that is cut on three sides (clip lance) and a closed end form that is cut on one or two sides (thread form, bridge lance).

Process

The lance and form process uses a single tool to cut and form material into the desired shape. The configuration of the lance and form may require the material to be stretched a great amount during the cutting and forming. The form shape and the ductility of the material (it's ability to be stretched) can affect the success of the desired result. Small bend radii and steep form angles can be more challenging to produce. It may also be necessary to provide draft angles on a clip lance and form to facilitate stripping the material from the tool. It is most common for this type of form to be made upward in the sheet as downward forms can easily catch in the press and become damaged in processing.

Thick material, tough abrasive material, and small delicate form features may not be able to be produced with a lance and form tool. There are times when the form and material requirements may make it necessary to create the desired form by pre-piercing around the area to be formed then "embossing" the form into the desired shape. This two-step process could be used in place of cutting and forming with the same tool.

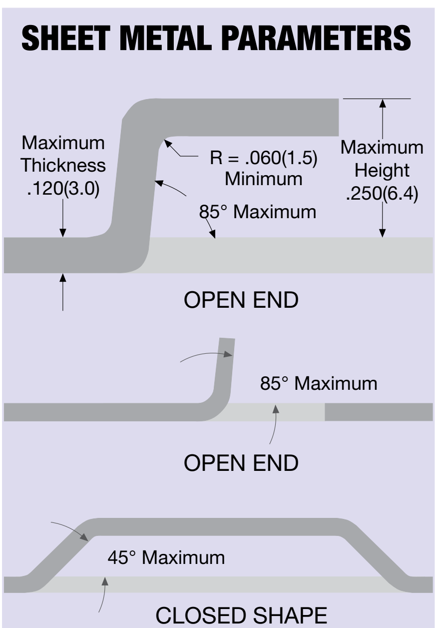
[Dimensions in Inches (mm)]

LANCE AND FORM

Summary

Successful lance and forms result from a favorable combination of material and form features. These applications generally require less tonnage than is needed for punching a hole having a perimeter equal to the cutting length of the lance and form shape. If problems are experienced with a lance and form sticking, creating a burr, or becoming damaged in processing, improved results will be achieved by following these tips and techniques:

- Use forming lubricant on the sheet
- Replace the cutting components when they become dull
- Confirm the tool was designed for the material type and thickness being formed
- Increasing the form radii
- Decrease the form height
- Use a more ductile material
- Form up whenever possible



LANCE AND FORM NOTES

For forms with sharp angles or curves, a ductile material such as aluminum or mild steel of medium thickness normally produces the best results.

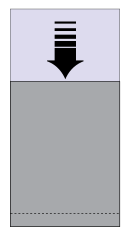
Lance and form assemblies can be any size and shape. Forming generally requires less press power than punching a hole of the same size.

When ordering a lance and form assembly, send drawings of the part showing all pertinent configurations and elevations.

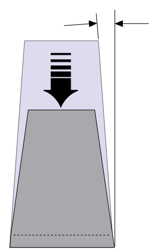
For certain closed forms a draft angle is recommended. Also specify material type and thickness, and model of press the assembly is designed for.

DRAFT ANGLE

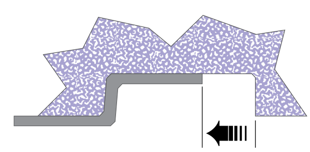
- Improves form quality
- Improves piece part quality
- Improves stripping



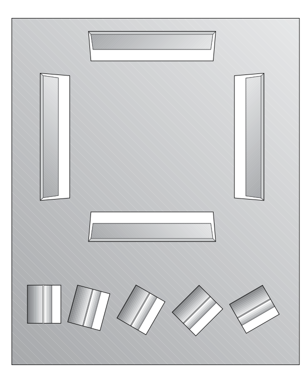
TOP VIEW
WITHOUT DRAFT ANGLE
FORM BINDS IN UPPER
DIE CAVITY



DRAFT ANGLE
5° Minimum
TOP VIEW : WITH DRAFT ANGLE, SHOWING
EFFECT AS FRONT EDGE OF TAB MOVES
BACK IN DIE



SIDE VIEW
FRONT EDGE OF TAB MOVES BACK IN DIE AS IT
IS FORMED



AUTO INDEX stations give 360° angular freedom to lance and form tools from forming all sides of a pocket with just one tool to angling a stop tab with another.

[Dimensions in Inches (mm)]

BRIDGE LANCE AND FORM

Definition

A bridge lance and form is created a strap of material is cut and formed up from the sheet, having the appearance of a bridge across an opening. Bridge lance and form applications cover a wide range of function and appearance but all are cut and formed into the desired shape with the same tool. A successful bridge lance and form is dependent upon a favorable combination of material properties and form features.

Features

Bridge lance and forms are common in sheet metal fabrication and vary greatly in features and function. They are frequently used to create a fastening point to zip tie wires into an enclosure. They can also be used to create simple to complex mechanical connections points for other components, as positioning locators, or two bridge lance and forms closely spaced function as a guide within an assembly. There are two general types of bridge lance and forms, a flat top and a radius top.

Process

The lance and form process uses a single tool to cut and form material into the desired shape. The configuration of a bridge lance and form may require the material to be stretched a great amount during the cutting and forming. The form shape and the ductility of the material (its ability to be stretched) can affect the success of the desired result. Narrow width bridges, small bend radii, tall form height, and steep form angles can be more challenging to produce. Those features cause the bridge of metal to stretch and thin and it may result in a weakened form or fracturing of the bridge from the sheet. It is most common for this type of form to be made upward in the sheet as downward forms can easily catch in the press and become damaged in processing.

Thick material, tough abrasive material, and small delicate form features may not be able to be produced with a bridge lance and form tool. There are times when the form and material requirements may make it necessary to create the desired form by pre-piercing along the sides then "embossing" the form into the desired shape. This two-step process could be used in place of cutting and forming with the same tool.

Summary

Successful bridge lance and forms result from a favorable combination of material and form features. These applications generally require less tonnage than is needed for punching a hole having a perimeter equal to the cutting length of the two sides of the bridge lance and form. If problems are experienced with a bridge lance and form sticking, creating a burr, or becoming damaged in processing, improved results will be achieved by following these tips and techniques:

- Use forming lubricant on the sheet
- Replace the cutting components when they become dull
- Confirm the tool was designed for the material type and thickness being formed
- Increasing the width of the bridge
- Increasing the form radii
- Decrease the form height
- Use a more ductile material
- Form up whenever possible

[Dimensions in Inches (mm)]

DOUBLE BRIDGE LANCE AND FORM

Definition

A double bridge lance and form is created when two straps of material are cut and formed up from the sheet, each having the appearance of a bridge across an opening. Double bridge lance and form applications can vary in size and appearance but they are cut and formed into the desired shape with the same tool. A successful double bridge lance and form is dependent upon a favorable combination of material properties and form features.

Features

Double bridge lance and forms are common in sheet metal fabrication and are typically used to create a guide within an enclosure. They can also be used to create simple to complex mechanical connections points for other components, or as positioning locators. There are two general types of double bridge lance and forms, a flat top and a radius top.

Process

The double lance and form process uses a single tool to cut and form material into the desired shape. The configuration of a double bridge lance and form may require the material to be stretched a great amount during the cutting and forming. The form shape and the ductility of the material (its ability to be stretched) can affect the success of the desired result. Narrow width bridges, narrow spaces between bridges, small bend radii, tall form height, and steep form angles can be more challenging to produce. Those features cause the bridge of metal to stretch and thin and it may result in a weakened form or fracturing of the bridge from the sheet. Those features can also create narrow, weak sections within a tool that may diminish the service life of the tool. It is most common for this type of form to be made upward in the sheet as downward forms can easily catch in the press and become damaged in processing.

Thick material, tough abrasive material, and small delicate form features may not be able to be produced with a double bridge lance and form tool. There are times when the form and material requirements may make it necessary to create the desired form by pre-piercing along the sides then “embossing” the two bridges into the desired shape. This two-step process could be used in place of cutting and forming with the same tool.

Summary

Successful double bridge lance and forms result from a favorable combination of material and form features. These applications generally require less tonnage than is needed for punching a hole having a perimeter equal to the cutting length of the four sides of the double bridge lance and form. If problems are experienced with a double bridge lance and form sticking, creating a burr, or becoming damaged in processing, improved results will be achieved by following these tips and techniques:

- Use forming lubricant on the sheet
- Replace the cutting components when they become dull
- Confirm the tool was designed for the material type and thickness being formed
- Increasing the width of the bridge
- Increase the space between bridges
- Increasing the form radii
- Decrease the form height
- Use a more ductile material
- Form up whenever possible

WATCH THE VIDEO:
Watch an animation of a Double Bridge:

https://youtu.be/rkwZvimkElg

[Dimensions in Inches (mm)]

CLOSED END LOUVER

Definition

A closed end louver is a lance and form that is created when sheet metal is cut (lanced) and formed simultaneously. Closed end louvers are cut on one side and the material adjacent to the cut edge is raised to create an opening in the sheet. Louvers can vary in width and length and may have an angled or radius form profile.

Features

Closed end louvers are a common feature used in sheet metal products as a way to provide for the movement of air into or out of an enclosure for cooling or moisture control purposes. Louvers can be grouped close together to provide a large open area in a specific region of the sheet. The open area of a louver determines the airflow and is dependent upon the open height and length of the form. There are two general types of closed end louvers, one with a straight back form profile and another with a curved back form profile. It is possible to create a straight back closed end louver with a progressive hit tool that “nibbles” a louver opening with multiple hits, effectively expanding the capability of the press. The aesthetic appearance of a continuous louver is diminished from a single hit louver.

Process

The closed end louver process uses a single tool to cut and form material into the desired shape. As the louver is formed upward, the cut edge moves up and back from the sheet. This simplifies the upper tool design, eliminating the need for positive stripping. The louver shape and the ductility of the material (its ability to be stretched) can affect the success of the desired result. The spacing between louvers should provide a flat area of at least 3 material thicknesses from the back of one louver to the front of the next. This will ensure a flat finished part with high quality forms. It is most common for this type of form to be made upward in the sheet as downward forms have a raw edge that can easily catch in the press and become damaged in processing.

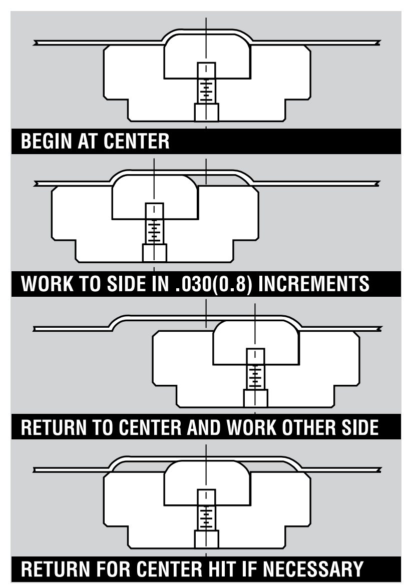
Tall, steep louvers in thick or abrasive material can be more challenging to produce. The form features and material requirements may make it necessary to create the desired form by pre-piercing along the louver opening then “embossing” the form into the desired shape. This two-step process could be used in place of cutting and forming with the same tool.

Summary

Successful closed end louvers result from a favorable combination of material, form features, and form spacing. These applications generally require tonnage equivalent to punching a hole having a perimeter equal to the open cut length of the louver. If problems are experienced with a closed end louver sticking, creating a burr, or becoming damaged in processing, improved results will be achieved by following these tips and techniques:

- Use forming lubricant on the sheet
- Replace the cutting components when they become dull
- Confirm the tool was designed for the material type and thickness being formed
- Decrease the form height
- Use a more ductile material
- Form up whenever possible

WATCH THE VIDEO:
Watch an animation of a Closed End Louver:



Start Continuous Louvers In The Center

Continuous louver tools are now designed to produce smooth-edged, level-topped louvers when recommended procedures are followed. Start in the center and form to one side and then the other in 0.030(0.8) increments. If needed, complete the process by rehitting the center for ultimate flatness.

[Dimensions in Inches (mm)]

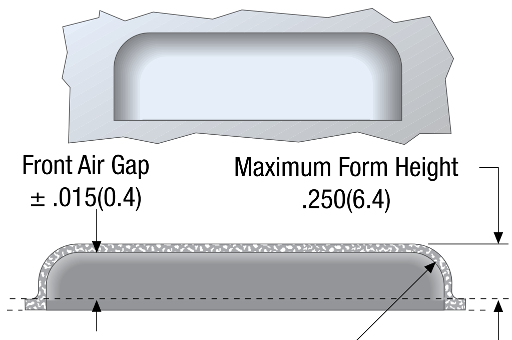
CLOSED END LOUVER

LOUVER FORMING NOTES

Closed end louvers

- High strength configuration
- No exposed corners
- External placement

Because of their smooth edges and round enclosed corners, closed end louvers serve best on exposed surfaces of panels.

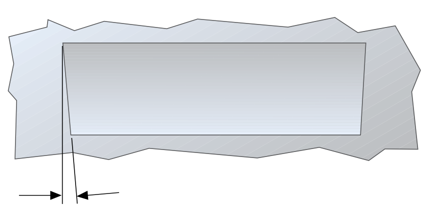


.500(12.7) radius recommended for stainless and cold rolled steel under .060(1.5 mm) to prevent corners from tearing.

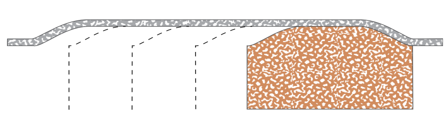
Open end louvers

- Maximum air flow
- Internal placement

Area of front gap includes the end angles for additional air flow. The exposed ends are best contained inside panels and enclosures.

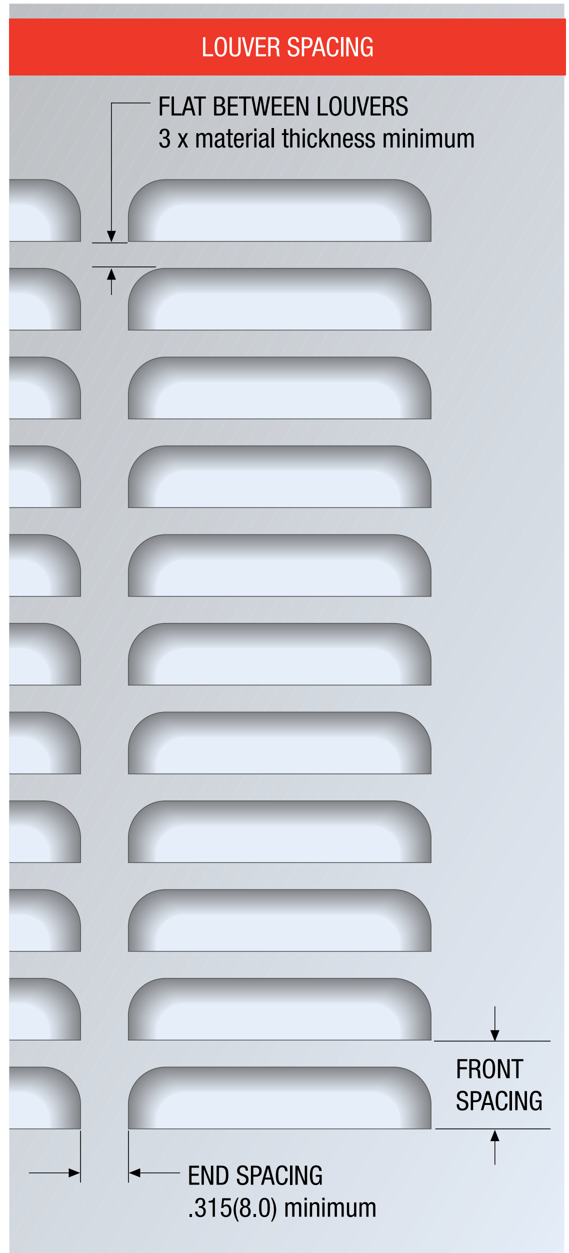


10° standard, 5° minimum. During forming, the shorter front edge rises and the longer back edge bends to allow the sides to strip easily.

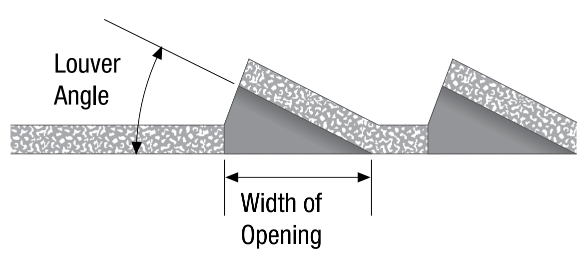


Continuous louvers

- Prototypes
- Short production runs
- Extra long louvers



SPECIFY LOUVER ANGLE OR WIDTH OF OPENING



Continuous louvers are made by forming a closed end louver and then gradually advancing the tool along one axis to expand the opening. The upper cavity is open on the ends to allow for unlimited length.

[Dimensions in Inches (mm)]

PARTS MARKING: EASYMARK™

EFFICIENT COMPONENT MARKING & IDENTIFICATION WITH EASYMARK™

THE PROBLEM:
Are your downstream processes clearly communicated? Do you want to improve them? When manufacturing processes frequently take place in multiple locations, clear communications are key to ensuring the accuracy of the sequential next process. To minimize error, components should be marked before they move on to the next step for easy identification. Typically, this involves some sort of traditional marking. While effective, traditional marking can remove material or damage the piece part.

THE MATE SOLUTION:
Determined to find a better component marking solution, Mate Precision Tooling and Finn-Power embarked on a development project for a turret-based part marking system. The result was the EasyMark™ 5-in-1 tool that provides users with five part marking capabilities using the same A-Station guide and canister:

1. InkMarker™:
Write on sheet metal using a special marker that does not dry out for two weeks or more. With InkMarker, the part is not harmed and the mark does not rub off unless required. What's more, the specially designed market will not dry out for more than two weeks with the cap removed!

2. Sheetmarker™:
Scribe or etch sheet metal surfaces using one of two (120 or 150 degree) diamond inserts for more permanent marking.

3. Film/Tape Cutter:
Cut protective film with a brass insert to expose localized areas without damaging the underlying material.

4. Centerpoint Down:
Uses a carbide insert in conjunction with machine depth control.

5. High-Speed Dot-Matrix Marking:
Perform dot matrix marking using the above carbide insert. Control the depth of downward stroke to vary the marking. Dot-matrix markings may be seen after painting operations.

Improved communication would be realized with the use of the visual aids EasyMark creates. Using any one or all of its capabilities could reduce downstream operations and/or paperwork right at the turret. EasyMark can create both permanent (scribe, dot matrix) and non-permanent (marker) markings. Possible reduction in scrap could be achieved through proper identification of components and or steps.

AVAILABLE TOOLING STYLES:
Thick Turret
Murata Wiedemann 114 style.

[Dimensions in Inches (mm)]

PARTS MARKING: EASYMARK™ (CONT.)

WITH EASYMARK, THE POSSIBILITIES FOR WHAT YOU CAN MARK ARE ALMOST ENDLESS:

- Job numbers
- Revision numbers
- Identify label or gasket locations to obtain consistent results without fixtures
- Part numbers
- Serial numbers
- Weld lengths, types and locations with symbols, if desired
- Date codes
- Ground screw locations and symbols
- Simple process instructions about hardware, bending, painting, etc.
- Use in conjunction with written instructions
- Bend line locations, distance and direction
- Produce creative signs or designs (“Welcome Visitors” with graphics…)
- Material removal area identification
- Warnings about special situations
- Insert hardware size, location and direction (e.g., “M4-up”)

EXAMPLES OF HOW ACTUAL CUSTOMERS ARE USING EASYMARK:

Marking all of their bend lines because they don’t rely on the back gauge when they use two people on the press brake. Scribing a line to shear parts down after they go through the press brake so they don’t have to use back up material on the flange.

Marks all of parts with lot or job numbers for traceability if the same part is being used on different assembly lines.
Used intentionally on powder coated parts because the marker will bleed through the paint, making sure the parts can be indentified when they come off the paint line.
Mark lines that help during plasma cutting.

AVAILABLE TOOLING STYLES:
Thick Turret
Murata Wiedemann 114 style.

STATION SIZES:
Thick Turret A-Station only, no index station required

REQUIREMENTS:

- Need a clean, oil free surface to use the marker
- Turret press needs the proper software to use EasyMark
- Most presses that can control the ram location precisely will work. Mechanical machines are not suited for using EasyMark
- The ram must be able to be set at a specific location downwards while the sheet is being moved
- Machines capable of using a wheel, scribe or similar application

Tonnage Restrictions:
None known

KIT INCLUDES:

- Special A-Station guide and canister
- Roller ball die
- Marker pen
- Blank die
- 2 diamond inserts
- All tools necessary to assemble EasyMark into any of its configurations
- 1 brass insert

WATCH THE VIDEO:
Watch an animation of an EasyMark in use:

https://youtu.be/ycASu1ZkaAg

[Dimensions in Inches (mm)]

PARTS MARKING: SHEETMARKER™

SHEETMARKER IS DESIGNED TO MAKE ETCHINGS AND GROOVES IN A VARIETY OF MATERIALS.

MARK OR ETCH SHEET METAL DURING PUNCH PRESS OPERATIONS

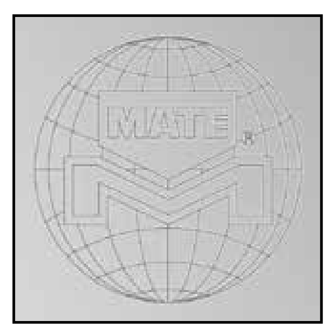
- Produce a wide of results…from very light etching to deep grooves
- Combine inserts and spring pressures to produce desired results
- Remove protective plastic with optional brass insert
- For all material types and thicknesses

PRODUCT FEATURES:

- Standard with two diamond inserts for etching or marking: 120° or 150°
- Optional brass insert…cut protective film on delicate materials
- Three color-coded springs:
  Red (Heavy Duty), Green (Medium Duty), Zinc (Light Duty)
- Storage case to protect tool when not in use
- Protective film cut using Sheetmarker with optional brass insert

KIT INCLUDES:

- Three springs with different loading pressures
- One retainer
- Two diamond inserts with different point angles
- One roller die
- Instructions
- Hardware is included for assembly and disassembly



WATCH THE VIDEO:
Watch an animation of a SheetMarker in use:

https://youtu.be/ycASu1ZkaAg

[Dimensions in Inches (mm)]

PARTS BENDING: EASYBEND™

Fabricating jobs often require the bending of parts. Typically, fabricators use press brakes for these operations. Sometimes, using a press brake seems like overkill, especially on smaller parts, and adds a secondary operation and cost to a job. Bending parts by hand could help lower manufacturing costs and speed time to delivery.

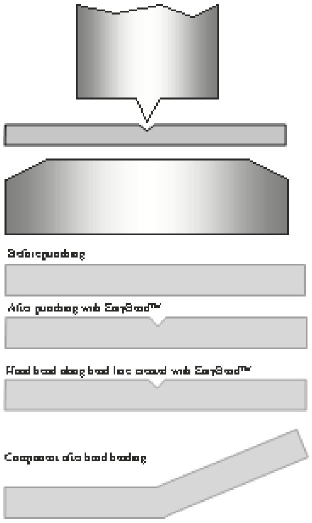
Figure 1

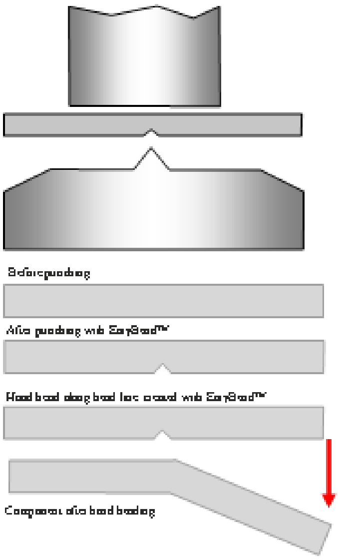
EasyBend™ is a tool that creates bend lines to make hand bending operations easy and convenient. EasyBend is ideal for intricate assemblies where conventional press brake forming techniques are inconvenient.

EasyBend employs a linear V-line to create a stencil in the sheet metal. The tool penetrates the sheet metal, creating a crisp bend line, enabling it to be bent by hand. The angle of the stencil point is related to the angle of the desired form, which must be specified when ordered. The actual depth of penetration is dependent on the ductility and thickness of the work piece. The continuous nature of the design allows the length of the bend-line to be a maximum suggested length of 12 inches (300mm).

Figure 2

Figure 3





EasyBend may be ordered to allow bending up. To bend up, the upper scores the top of the sheet (Figure 2). To bend down, the lower scores the bottom of the sheet (Figure 3.)

The benefits of using EasyBend include:

- Precision bends done by hand
- Eliminates bending in a press brake
- Bends can be up or down
- Perfect for smaller parts or lighter gauge materials
- Eliminates expensive secondary operations
- Fabricate and bend pre-painted material
- Parts can remain flat for transport or shipping saving space and allowing parts to be bent manually before assembly

EasyBend opens up a lot of possibilities. One such idea is to use it to make tabs that can lock two parts together (Figure 4), which can be particularly useful in field installations or repair applications in various industries. You can also use EasyBend to perform multiple and decorative bends. Figure 1 shows a display in Mate's Customer Solution Center that used EasyBend and other forming applications such as louvers.

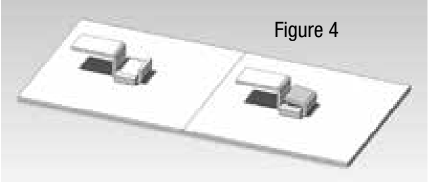


Figure 4

AVAILABLE IN ALL TOOLING STYLES

MATERIAL RESTRICTIONS:

- Material thickness less than 16 gauge (1,50mm) (Consult with a Mate Application Specialist for thicker applications)
- Maximum 12 inches (300mm) length recommended
- Bends greater than 90 degrees not recommended
- Design variations are available for thicker materials

[Dimensions in Inches (mm)]

PARTS BENDING: VARIBEND™

One of the most common sheet metal fabricating operations is bending. As fabricators look to streamline operations to reduce costs and increase efficiency, bending usually tops the list because it typically requires a secondary operation on the press brake. Yet the opportunities to rationalize bending operations have been limited. Mate offers one solution — EasyBend™ — for smaller piece parts where using a press brake seems like overkill. With VariBend™, Mate expands the possibilities of making fabrication more efficient by more effective use of the punch press.

Mate's VariBend is a flexible, multi-purpose tool that allows forming of sheet metal tabs in a punch press environment, eliminating some secondary operations normally reserved for a press brake. Flexible, because it can be used for a range of material thicknesses, VariBend is also multi-purpose, because it can be used to form tabs at any angle up to (and in some cases exceeding) 90°.

HOW IT WORKS
The upper forming tool is lowered on to the sheet until bending begins. Then, the flange is over bent to account for the material springback (left illustration below). As the upper returns, the desired bend is formed (right illustration below).

Although not required, the VariBend is well suited for machines with upforming capabilities and especially machines with stroke control. Upforming allows the highest form possible for the tool design. Since the die sits lower than a forming die in a normal forming operation, the sheet lies flatter. This increases the accuracy of the formed tab while reducing sheet marking on the bottom side of the sheet. Using VariBend in machines with stroke control assures easy setup and accuracy when forming tabs in a punch press.

VariBend can also be used in machines without upforming capabilities or stroke control features. Formed tab height may be restricted, depending on the height of the turret gap of the user's machine. In thick turret environments, Mate Ultraform® holders may be used for length adjustment in machines without stroke control. Formed angles can be modified by adjusting the length of the Ultraform holder with the VariBend tools installed.

OTHER INFORMATION:

- VariBend can be programmed as a normal forming tool (regular forming or as an upforming tool).
- To obtain bends of 90°, a punched tab must have a minimum length of 0.243" (6,17mm) from the end of the tab to the tool centerline in order for the rotating die to initiate the forming process. A shorter length tab will result in forms less than 90°.
- With varibend, it is easy to adjust the angle of the form. Generally speaking, to change the formed angle by 1°, adjust the stroke length or tool length by 0.005" (0,13mm).
- VariBend will produce form heights of approximately 0.300"–0.700" (7,60mm –17,75mm). With caution and reduced speeds, slightly shorter forms can be achieved. The maximum height is determined by the design and height of the upper tool and the space available in the machine.
- When possible, all forms should be processed as close to the end of the program as possible to prevent damage to forms from the normal punching process.

[Dimensions in Inches (mm)]

PARTS BENDING: VARIBEND™ (CONT.)

AVAILABLE TOOLING STYLES AND STATION SIZES

- Thick Turret, B through E Stations
- Trumpf Style, Size 2
- Murata Wiedemann 114 Style, D through L Stations
- Thin Turret, 1¼" and 3½" Stations
- Nova, D through G Stations

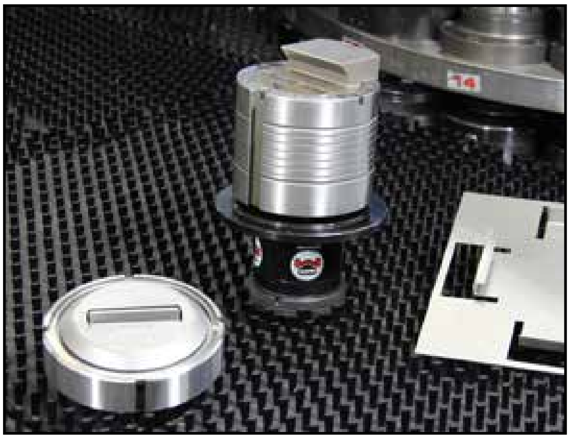
MATERIAL AND OTHER RESTRICTIONS:

Minimum material thickness:		0.020" (0,51mm)
Maximum material thickness:		
	Mild Steel:	0.098" (2,49mm)
	Aluminum:	0.098" (2,49mm)
	Stainless:	0.079" (2,01mm)

WATCH THE VIDEO:

Watch an animation of VariBend:

https://youtu.be/0HRbVf5lwjw



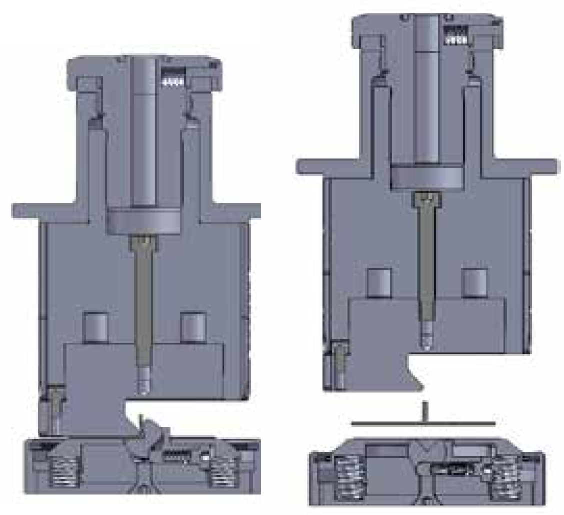


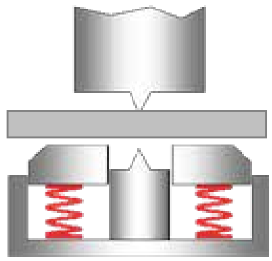
Illustration of VariBend, thick turret style, in forming position (left) and after upstroke (right). 90° angle shown.

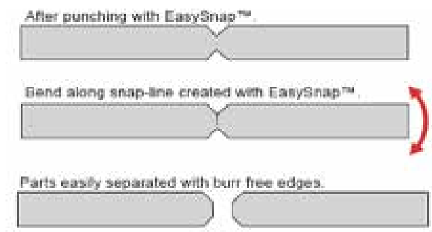
[Dimensions in Inches (mm)]

PARTS REMOVAL: EASYSNAP™

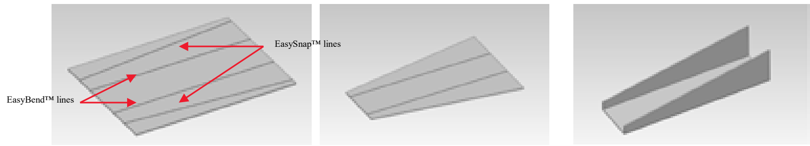
USING EASYSNAP™ FOR QUICK, CLEAN PARTS SEPARATION

Fabricators use a variety of methods to separate multiple parts from a sheet of material. A frequently used method places small tabs between parts by programming the spacing of a slitting punch. While the tabs keep the parts intact during punching, they can be difficult to remove and often leave rough or sharp edges that usually require a secondary operation to remove.





Using a Mate EasySnap™ tool solves these issues because it allows you to build in way to snap parts out of sheet metal without using a slitting or punching tool. EasySnap allows fabricators to simply snap punched components out of sheet metal by hand, making life much easier. Since EasySnap leaves a clean, smooth edge, there is no need for secondary operations.

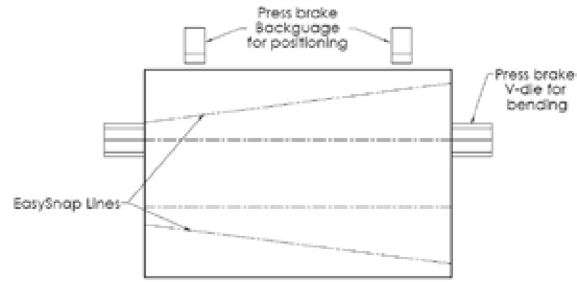


a

EasySnap uses a V-line stencil machined onto the face of the upper and lower tools. As the tools penetrate the sheet, they create a line of weakness (snap-line) in both surfaces of the sheet metal. The sheet metal can then be snapped apart by bending the material along the snap-line. The actual depth of penetration and force required to snap the part is dependent on the ductility and thickness of the material being punched. It also keeps parts removal easy since the parts simply snap off the sheet.

HOW CUSTOMERS USE EASYSNAP:

- Temporary paint hook. Run the part through the paint line, then simply snap off the unneeded part. Eliminates scratching, extra hooks and is easier to run.
- Leave a blank in for a window part that can be separated at a later step in the manufacturing process.
- Breaking down the skeleton into smaller parts for easier, neater disposal.
- Press brake back stop locator for smaller parts. Simply snap off at the end of the press brake operation. It's safer, faster, and more accurate…eliminates "eyeballing". (Right).
- Easier processing of smaller parts.
- Clean, smooth edge after separating parts.



AVAILABLE IN ALL TOOLING STYLES AND STATION SIZES

MATERIAL AND OTHER RESTRICTIONS:

- Maximum material thickness: up to .060"(1,52mm)
  (Consult with a Mate Application Specialist for thicker applications)
- Minimum material thickness: .024"(0,61mm)
- The continuous nature of the design allows a maximum suggested snap-line length of 12 inches (300mm).

WATCH THE VIDEO:

Watch an animation of EasySnap:

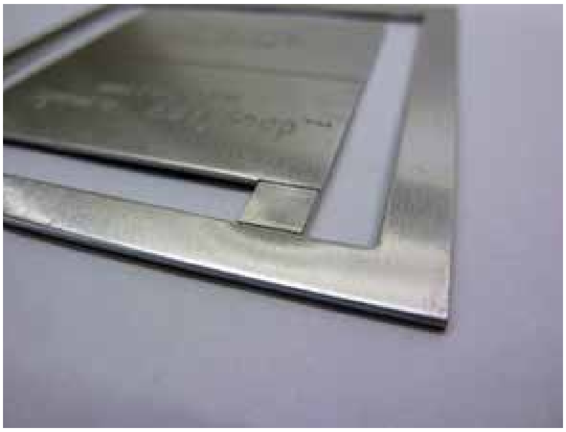
https://youtu.be/b5zu_Az4x10

TIPS FOR SUCCESS:

- Tool will NOT work as desired in materials thicker for which it was designed.
- In Trumpf presses, classify the tool as a tool type 13 (Embossing).
- Setup information sheets are provided with the tool; these are to be used only as a guideline. Customer will be advised to start with lesser values, and adjust accordingly to their material, machine and other variables.

[Dimensions in Inches (mm)]

PARTS REMOVAL: SQUARE EASYSNAP™



Micro-joints (or "shake and break") are an easy method fabricators use to separate multiple parts from a sheet of material. The method places small, interconnecting tabs between parts by programming spacing of the slitting punch. The tabs keep the sheet and parts intact during punching, but make it easy to separate off the press. There are times, such as nesting of really small parts, parts too big to be blanked or corners, when a micro joint is not possible. In these situations, fabricators often use a wire joint. While this works, wire joints leave spurs on the edge, which, unless removed, could interfere with other downstream operations such as bending.

Square EasySnap™ solves these issues because it leaves a smooth edge and makes parts removal fast and simple.

Like EasySnap™, Square EasySnap is a scrapless part retention system that allows fabricators to simply snap punched components out of sheet metal. Square EasySnap features a V-line stencil machined onto the face of the upper and lower tools.

As the tools penetrate the sheet, they create a line of weakness (snap-line) in both surfaces of the sheet metal. The sheet metal can then be snapped apart by bending the material along the snap-line. The actual depth of penetration and force required to snap the part is dependent on the ductility and thickness of the material being punched. It also keeps parts removal easy since the parts simply snap off the sheet.

WHERE TO USE SQUARE EASYSNAP:

- Nesting very small parts in a large sheet.
- Sheets where the parts are too big to be blanked to make sure they will not fall away during the punching process.
- Putting the Square EasySnap away from the corner so there are only 2 sides of the tabs holding the sheet make it easier to break along one line.
- Rounded part corners where a corner micro-joint is not possible.
- Scribing a line to shear parts down after they go through the press brake eliminating the need for back up material on the flange.

AVAILABLE TOOLING STYLES AND STATION SIZES

- Thick Turret B-Station
- Thin Turret, B-Station
- Murata Wiedemann 114/112, D-Station
- Trumpf Size 2
- Salvagnini, 33mm

MATERIAL RESTRICTIONS:

Maximum material thickness: up to .060"(1,52mm) (Consult with a Mate Application Specialist for thicker applications)
Minimum material thickness: .024"(0,61mm)

TIPS FOR SUCCESS:

- Leave square tabs slightly smaller than the width of the parting tool used.
- Tab size may be reduced for thicker materials, if necessary.
- Tab location will vary per customer's design and application.
- During removal, apply leverage to the part leaving the tab in the skeleton. This will prevent the necessity to remove the tab later from the good part.
- Tool may work for a material thinner than for which it was designed, but the desired coining effect will be diminished.
- Tool will NOT work as desired in materials thicker for which it was designed.
- In Trumpf presses, classify the tool as a tool type 13 (Embossing).
- Setup information sheets are provided with the tool; these are to be used only as a guideline. Customer will be advised to start with lesser values, and adjust accordingly to their material, machine and other variables.

[Dimensions in Inches (mm)]

PARTS REMOVAL: SQUARE SHEARBUTTON

Micro-joints (or "shake and break") are a traditional method fabricators use to separate multiple parts from a sheet of material. In thicker materials, the micro-joints are not always strong enough to hold the part while punching. If the size of the micro-joint is increased, this tends to leave an even larger burr on the part.

To review, the traditional micro-joint method places small, interconnecting tabs between parts by programming the spacing of the slitting punch to leave material unpunched. However, micro-joints leave burrs on the edge, which, unless removed, could interfere with other downstream operations such as bending. Micro-joints may also cause serious injury if not removed.

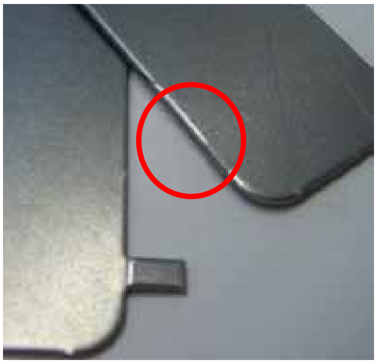
A Square ShearButton™ tool is a part retention system for thicker materials. Available in both form up and form down, Square ShearButton allows fabricators to snap punched components out of sheet metal. It is quite similar to a round half shear, but square. A benefit of the form down is that once the tab is removed, the edge closely matches the quality of a punched edge.

Square ShearButton is a punch and die system. One of the components is a spring loaded set that allows partial punch penetration and allows the material to strip off from the tool set. When the tools penetrate the sheet, they move material in the shape of the punched tab (up or down) to a point before the material fractures. This leaves a tab that is strong enough to hold the part in place, but will still break off easily for removal. The actual depth of penetration and force required to snap the part is dependent on the ductility and thickness of the material being punched and the width of the shear button. It also keeps parts removal easy since the you simply snap the parts off the sheet.

Square ShearButton may be used with a variety of material types:

- Stainless Steel
- Aluminum
- Cold Roll Steel
- and More

Depths and heights may be adjusted to suit the user's application.



[Dimensions in Inches (mm)]

PARTS REMOVAL: SQUARE SHEARBUTTON (CONT.)

WHEN TO USE SQUARE SHEARBUTTON:

- Nesting parts in a large sheet (photo above right).
- Very large or heavy parts with minimal micro-joints. These tabs are much stronger than the traditional micro-joint, so fewer are needed.
- Rounded part corners where a corner micro-joint is not possible.

AVAILABLE IN ALL TOOLING STYLES AND STATION SIZES

MATERIAL RESTRICTIONS:
Works best on materials between 0.06”(1,52mm) and .12”(3,04mm)

TIPS FOR SUCCESS:

- Square ShearButton tools may be ordered the same width as a parting or slitting tool.
- For standard Square ShearButton strength, program to penetrate the material by 50% of the thickness.
- Square ShearButton tab sizes or widths may vary, depending on the user’s preference and application. Example: a width of .118”(3,00mm) using a .197”(5,00mm) square shear button.
- In a nesting application, use Square ShearButton on one edge of the part for single part removal or between parts.
- Use Square ShearButton as the last tooling operation if at all possible
- If this is not possible, punch both sides adjacent to the tabs before forming with the Square ShearButton. This will help prevent the forms from being smashed with tools used after the Square ShearButton is formed.
- To prevent leaving burrs behind after part removal, use sharp corner parting or slitting tools up to the Square ShearButton tab. This will help ensure the tabs break evenly.
- If edge quality is not your primary concern, try using a trapezoid shaped punch for slitting. A small burr will result on the part from the radius corner of the punch. Using this method will leave a smaller section on the part than the skeleton. This ensures all tabs break away from the part, not the skeleton.
- In Trumpf-style presses, select Tool Type 14 (forming).

Square ShearButton (form down) after removal from sheet shows smooth edge quality.

[Dimensions in Inches (mm)]

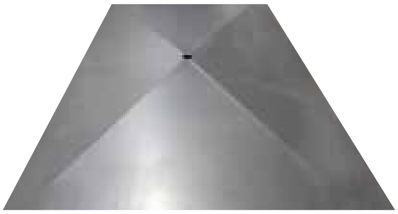
ROLLERBALL™

ROLLERBALL™

Whether it's to add strength to lighter gauge material, create an emboss to provide additional clearance for a feature or hardware, or add visual or cosmetic enhancements, Mate's Rollerball is a highly versatile tool. Some of these applications allow the fabricators to use thinner material or eliminate secondary operations, resulting in material and labor cost savings, and reduced production time. The Mate Rollerball is designed to be used in machines that can hold the ram down while simultaneously moving the sheet around in the X and/or Y axis. When used in machines with accurate stroke control, a simple adjustment to the machines stroke length can create a deeper or shorter form. In other thick turret machines, using the Mate Rollerball in a Mate Ultraform holder offers easy adjustability; simply change the length of the holder with just a few clicks of the tool.

STANDARD CONFIGURATION

Let's start with the standard beading configuration. With a standard beading configuration, strengthening ribs and cross brakes can be added to sheet metal while still in the punch press. Let's face it, creating cross brakes in a press brake is time consuming and not highly accurate. Why not use your punch press to add these features, eliminating the need for the press brake? Since cross brakes typically can't be produced using a back gauge on a press brake, additional time is required to lay out the bend lines and, more importantly, hit these lines accurately when forming the cross brakes. Using the Mate Rollerball in the punch press eliminates the time consumed at the press brake, and increases the accuracy of the location of these features.



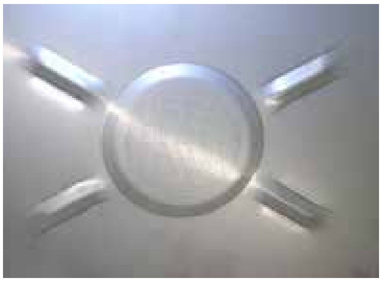
FORMED RIBS

Formed ribs can add structural integrity to sheet metal, thus allowing parts to be fabricated using thinner material than called for in the original design. Thinner material equals cost savings in most cases. A formed rib courtesy of the Mate Rollerball can also replace a continuous rib formed by a standard forming tool. A standard forming tool might require tens to hundreds of hits and minutes to produce a strengthening rib over the length of a sheet. Using the Mate Rollerball may reduce those to just a few seconds and with a much better looking end result.

When used in an auto index station, ribs can be added for cosmetic appearance as well. With a little bit of programming experience, logos or artistic impressions may be added to a sheet metal part, adding to the visual enhancement of the final product.

OFFSET FORMS

The Mate Rollerball also comes with a shortened insert to create offset forms. Whether it's for clearance required for additional features or existing hardware, perhaps a 3D appearance of the final product, or a subtle change in material levels, the offset feature of the Mate Rollerball can help. The offset configuration consists of one standard insert in the upper tool, and one shortened insert. This creates an offset or multi-level form. Depending on the location and direction of travel of the material, the result can be a formed upward or a formed downward offset.



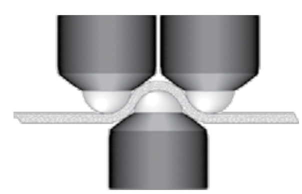
ROLLERBALL™ (CONT.)

With some creative programming, the offset configuration can also be used to create flared window openings, or flared part edges. This setup can typically achieve bends between 25-32 degrees, depending on the flange width and material thickness. Many companies that produce products with windows have used the Mate Rollerball offset configuration to create flared window openings that add strength to a part with a cutout, and incorporate a feature that allows gaskets or trim work to be added around the opening. Running this type of offset form along a part edge can also add significant strength to the final product; hinged doors or enclosures are a prime benefactor of this application.

Take a look at typical duct work for heating and air conditioning systems that use a lot of the stiffening and cross brake features. Many of these features can be created in the punch press using one or both of the Mate Rollerball configurations.

So, whether you have a great new idea or an old idea listed above, let the Mate Rollerball help cut your material and labor costs, reduce your production times and improve your accuracy, all while your parts are still in the punch press.

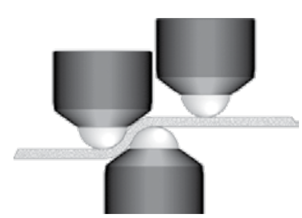
Set punch holder to shut height of press minus material thickness.



Rollerball inserts in position for bead forming

MATERIAL THICKNESS:
Minimum .020(0.5)
Maximum.105(2.7)

MINIMUM FORM RADIUS:
.433(11.0)



Rollerball inserts in position for offset

WATCH THE VIDEO:
Watch an animation of RollerBall in use:

https://youtu.be/PqvtZTHaYHw

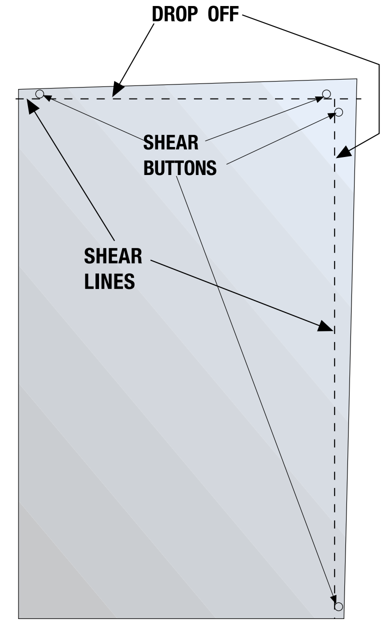
[Dimensions in Inches (mm)]

SHEARBUTTON

The shearbutton is a special purpose tool for placing locating tabs in sheet metal for further fabrication such as shearing and spot welding. The die insert can be provided with a slotted tip so that buttons can be put into thin material with less chance of falling out.

Two punch inserts are included with each .200(5.1) assembly. 1 - slotted tip for material up to .090(2.3) thick; 2 - solid tip for materials up to .188(4.8) thick that do not benefit from a tab.

- Stations adjacent to this tool should not be used
- Remove fixed machine stripper on machines so equipped if clearance is needed for forming height and/or step die installation
- Do not pass workholders over step die
- For best forming results, use a good forming lubricant



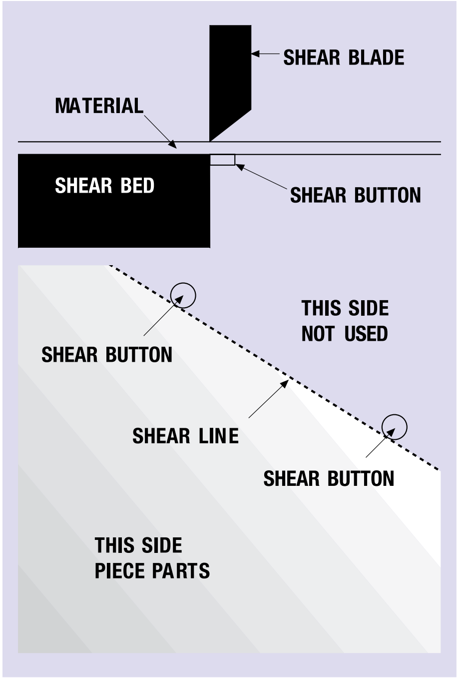
Using Shearbuttons to square large sheets

CONVENTIONAL USE OF SHEARBUTTONS

- Using shearbuttons to square large sheets in shear after punching
- Shearbuttons in lower sheet fit into matching holes in sheet plate
- Shearbuttons used as weld projections for precision placement of plates to be welded together

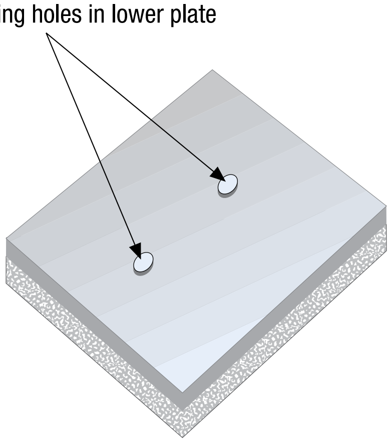
1. Slotted tip for material up to .090(2.3) thick puts tabs on shearbutton to keep it attached to the sheet; this is the standard setup insert installed on the .200 and (5.0) assemblies.

2. Solid tip for heavy duty use in materials up to .188(4.8) thick that do not benefit from a tab. For custom assemblies, one shearbutton insert will be supplied slotted or not depending on material thickness to be punched.



Conventional use of Shearbuttons

Shearbuttons in lower plate fit into matching holes in lower plate



Shearbutton used as weld projections for precision placement of plates to be welded together

[Dimensions in Inches (mm)]

STAMPING: ALPHA-NUMERIC

STAMPING - ALPHA-NUMERIC
Replaceable character inserts make this an economical marking tool since the same tool can be used for any number of messages. Type is available in standard sizes from .094(2.4) to .250(6.4) height in the Standard Industrial Gothic/sans serif typeface generally used for stamping metal.

MAXIMUM LINE LENGTH

D	1.000(25.4)
E	1.500(38.1)
F	2.000(50.8)
G	2.500(63.5)
H	3.125(79.4)
J	3.750(95.3)
K	4.500(114.3)
L	5.000(127.0)

HELPFUL TIPS:
- Stations adjacent to this tool should not be used when stamping the bottom of the sheet
- Do not pass workholders over step die

[Dimensions in Inches (mm)]

STAMPING: ALPHA-NUMERIC

CHARACTER HEIGHT

3/32" (2.4mm) CHARACTER HEIGHT

ABCDEFGHIJKLMNOPQRSTUVWXYZ0123456789&/.-()

1.000(25.4)

1/8" (3.2mm) CHARACTER HEIGHT

ABCDEFGHIJKLMNOPQRSTUVWXYZ0123456789&/.-()

1.000(25.4)

3/16" (4.8mm) CHARACTER HEIGHT

ABCDEFGHIJKL PQRSTUVWXYZ0123456789&/.-()

1.000(25.4)

ABCDEFGHIJKLM XYZ012345678&/.-()

1.000(25.4) 1/4" (6.4mm) CHARACTER HEIGHT

STAMPING DEPTH AND FORCE

CHARACTER HEIGHT		V-LINE DEPTH	FORCE IN TONS(kN) PER CHARACTER		
FRACTIONAL INCHES	DECIMAL EQUIVALENTS		Aluminum	Mild Steel	Stainless Steel
3/32	.094(2.4)	.004(0.1)	.12(1.1)	.32(2.8)	1.0(8.9)
1/8	.125(3.2)	.005(0.1)	.18(1.6)	.50(4.5)	1.50(13.4)
3/16	.188(4.8)	.008(0.2)	.40(3.6)	1.20(10.7)	3.40(30.3)
1/4	.250(6.4)	.010(0.3)	.70(6.2)	1.90(16.9)	5.50(49.0)

CHARACTER COUNT

STATION			3/32 .094(2.4)	1/8 .125(3.2)	3/16 .188(4.8)	1/4 .250(6.4)
B	DOWN		8	6	4	3
C	DOWN		16	12	8	6
D	DOWN		24	18	12	9
E	DOWN		32	24	16	12
B		UP	8	6	4	3
C		UP	24	18	12	9
D		UP	40	30	20	15
E		UP	48	36	24	18

[Dimensions in Inches (mm)]

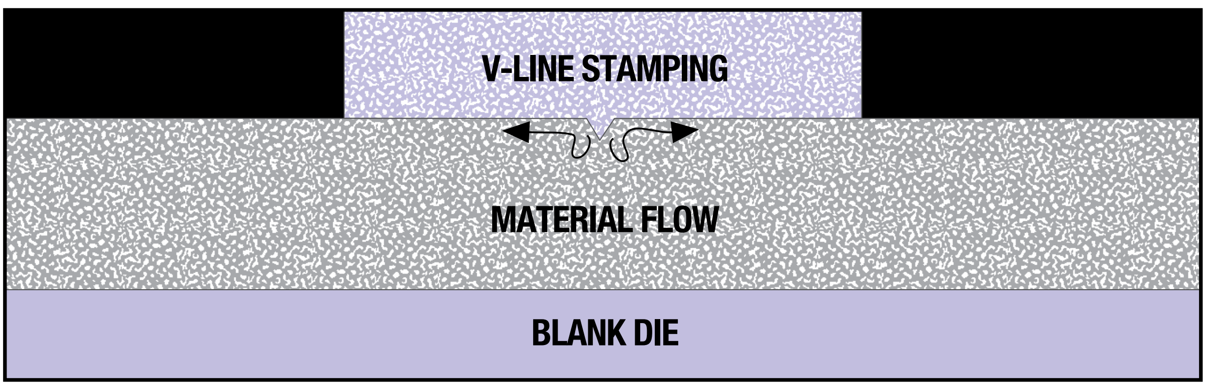
STAMPING: V-LINE

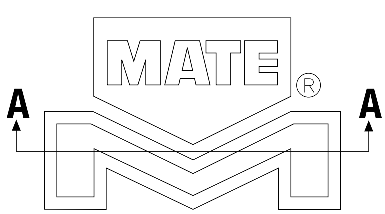
Custom designed logos or inscriptions may be any size within a hypothetical circle up to the maximum stamping size of the station or mulitple tools for larger applications. Submit a CAD drawing or camera ready art of logo and indicate finished size and angle.

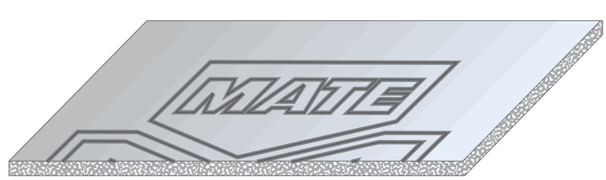
V-Line stamping renders the image with a thin, sharp line stamped into the surface. It requires relatively small force. Large complicated images are possible.

HELPFUL TIPS:

- Stations adjacent to this tool should not be used when stampling the bottom of the sheet
- Do not pass workholders over lower assembly

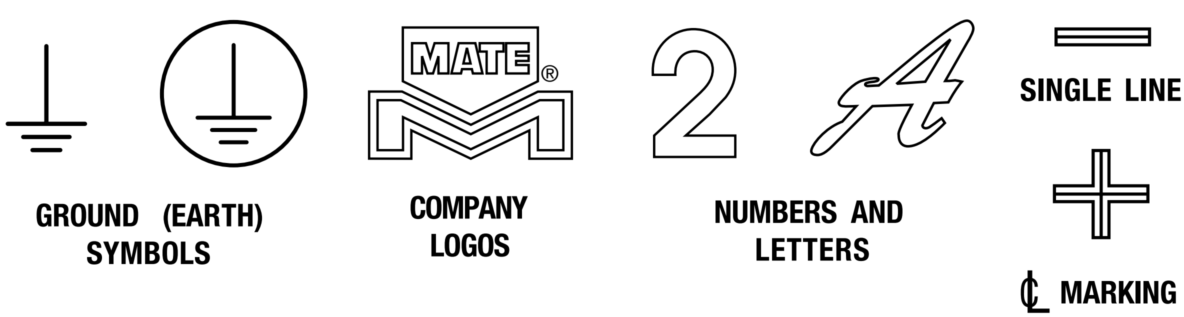






V-LINE STAMPING

A thin, sharp line reproduces the outline of the shape for an etched appearance. Low punching force.



Within press capability, figures may be any size up to station maximum. When ordering, an accurate indication of each figure's size, shape and depth below surface are required. Typefaces may be specified or artwork may be furnished, depending on the requirements of the subject matter.

[Dimensions in Inches (mm)]

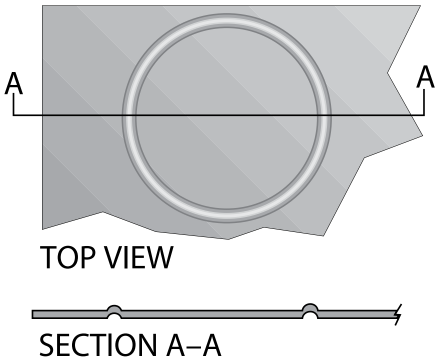
OTHER TIPS & TECHNIQUES

Tearing Or Splitting Of Overstressed Metal Forms

Reduce these by placing the form on the sheet with the grain running perpendicular to the form. Corners of high louvers, high extrusions, complex lance-and-forms, and card guides are typically vulnerable to this effect. Liberal application of a forming lubricant is also recommended to let the metal slide more freely over the forming surface of the tool, especially in stainless steel.

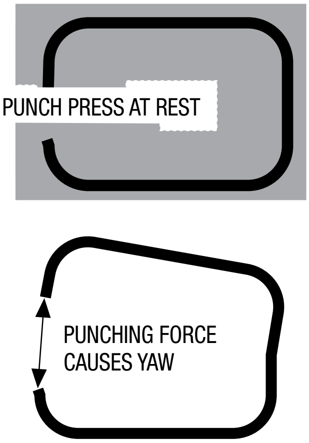
Form Raised Beads In Any Configuration

With Mate's beaded emboss tool, you can raise an embossed bead in virtually any configuration that fits on the sheet. This tool forms in 0.030(0.8) increments up to a height of 0.250(6.4) in materials 0.075(1.9) and thinner. It can be used for forming straight lines or curved lines.



Consistent Form Height Requires Stroke Control

Consistent, precise forming requires analysis of punch press stroke dynamics. When the upper unit meets material, several tons of force come into play. In applying this force, the frame of the punch press tends to move slightly in the opposite direction and "yaw" in a manner that increases shut height. As the punch pierces the material, punching force reduces sharply and the frame springs back toward its original position. This causes the upper unit to lunge deeper into the lower unit.



The lunging motion typically occurs before the forming operation has taken place. If the motion is not controlled, forming is performed by spring back of the press frame and there is very little control over accuracy of depth.

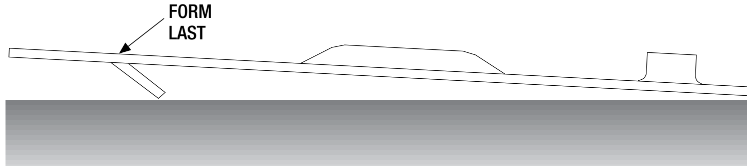
To counteract this process, Mate's Dyna-Form Stroke Control™ positions the forming operation to become complete with upper unit "bottomed" in the lower unit. Spring back does not affect the depth. One piece part is exactly the same as the next. An additional advantage of bottoming is to coin the forms, giving them a crisp, well-defined appearance. Dyna-Form Stroke Control is designed into all Mate special assemblies that include forming of material.

[Dimensions in Inches (mm)]

OTHER TIPS & TECHNIQUES

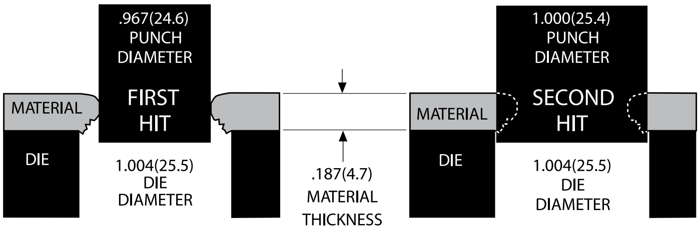
Form-Down Last

When using forming tools, form-down operations are generally avoided because they take up so much vertical room and any additional operations tend to flatten them out or bend the sheet. They can also drop into dies, get caught and pull out of work holders. However, if a form-down operation is the only solution for a particular piece part, make it the last operation on the sheet.



Shaving Makes Straight-Walled Holes Without Drilling

When you need a smooth, straight-walled hole for a shaft bearing or other use, shaving can save the time and trouble of performing a second operation on another machine. To do this, you need to punch the hole twice. First, use a punch with a punch-to-die Total Clearance equal to 20% material thickness. Second, use a larger punch exactly the same size as the finished hole. The die(s) used for both punches should be 0.004(0.10) larger than the second punch.



The second hit will shave the sides of the hole, removing most of the rollover and fracture effects caused in the first step, and enlarge the burnished area. This operation works best on mild steel and other materials ductile enough to shave.

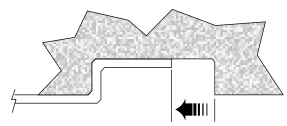
An easy way to order tools for shaving is to use the finished hole size as a reference. Order PUNCH #2 to the finished hole size, the DIE(S) to the same size as punch #2 + 0.004(0.10) Total Clearance, and for PUNCH #1 subtract 20% material thickness from the die size including clearance.

[Dimensions in Inches (mm)]

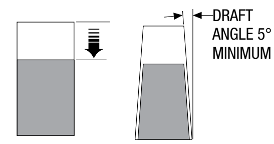
OTHER TIPS & TECHNIQUES

Tabs Designed With "Draft Angle" Won't Bind In Forming Dies

Tabs and louvers will bind in dies if they are the same width as the die opening. Designing a slight angle (draft angle) into the tab narrows the tip. Then the tip can move back into a wider space in the die as the tab is formed, leaving clearance on both sides so that the tab strips freely. Draft angle is normally 5°.



SIDE VIEW: FRONT EDGE OF TAB MOVES BACK IN DIE AS IT IS FORMED.



TOP VIEW: WITH AND WITHOUT DRAFT ANGLE, SHOWING EFFECT AS FRONT EDGE OF TAB MOVES BACK IN DIE.

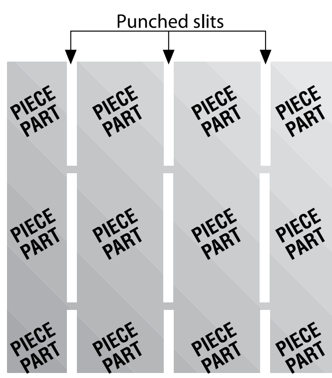
Precision, High Speed Separation Of Piece-Parts Combines Punch Press And Shear Operation With Automatic Gauging System

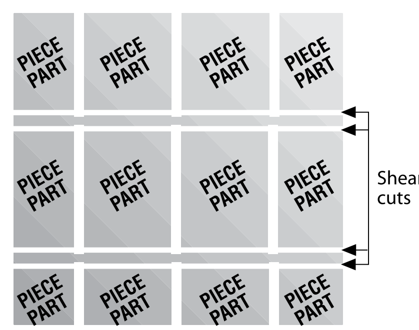
'Y" AXIS CUT ON THE PUNCH PRESS

Programming your CNC punch press to slit between piece-parts using a guided shearing assembly, prepares the Y axis for separation of the piece-parts. Enough material is left between parts to hold the sheet intact.

'X" AXIS CUT ON THE SHEAR

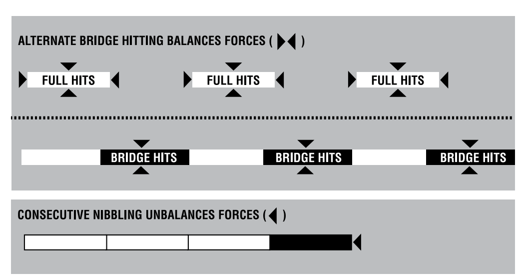
Specially gauged shear-cuts along the X axis intersect punched slits so that one pass through the shear separates piece parts completely.





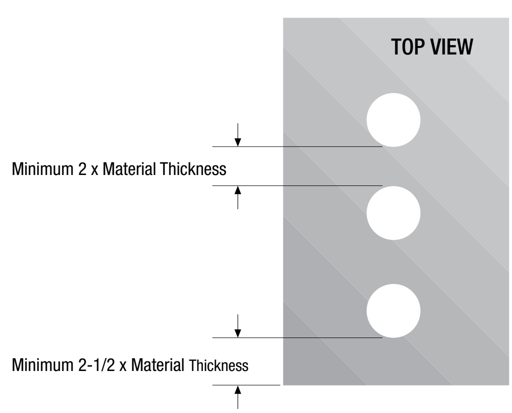
Bridge Hitting Reduces Tool Wear

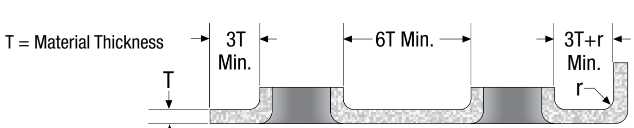
By alternating hits when performing shearing/slitting operations, forces upon the tool remain balanced from side to side and end to end. As a result, the punch operates square to the material and die. Over time you will notice a difference in the reduced frequency of sharpening and generally longer tool service. This practice is called "bridge" hitting because the full hits leave a "bridge" of material between them that is removed by the bridge hits.



Recommended Minimum Distances Between Holes, Forms And Edges Of Sheets

If holes and forms are placed any closer to each other or to edges of sheets than shown below, they will distort each other and the material because material flows when it is punched or formed.





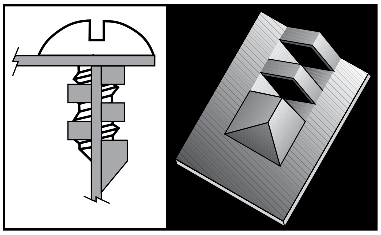
SIDE VIEW

[Dimensions in Inches (mm)]

OTHER TIPS & TECHNIQUES

For Access Panels That Fasten With Screws

Screw-pockets in the edge of one panel, coupled with screw holes in a joining panel, make a quick job out of preparing a removable panel on the punch press. No nuts needed. The screw pocket is formed in one hit by a lance and form tool assembly available in sizes to accommodate economical standard screws.

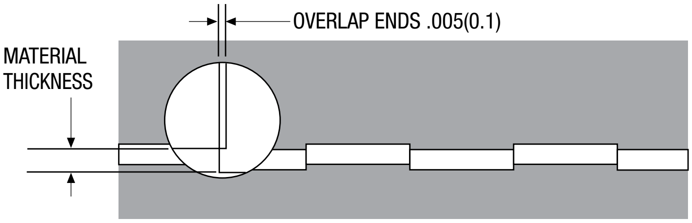


Making Accurate Butt Joints In Metal Is Easy

If You Cut Both Tab And Slot In The Same Hit

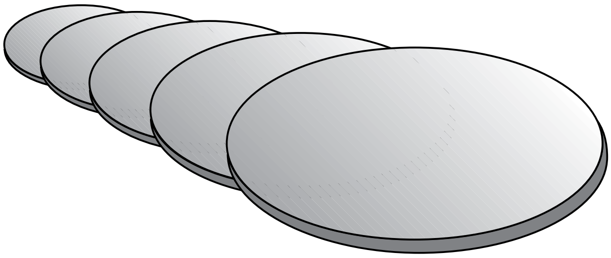
This practice cuts a modified dovetail joint out of the metal that leaves both pieces flush with each other on the sides and on the ends.

The trick is to position the pieces to be joined next to each other on the sheet so that the mating edges can be parted with a single row of hits. Then program the hits to overlap 0.005(0.10) end to end and position alternate hits to be offset one material thickness side to side.



Punch Painted Surfaces Without Marring

Use Mate's urethane stripper pads. These self-stick pads can be applied to Ultra®, Marathon® and Euromac® multi-tool stripper faces. Simply peel off the backing to expose an adhesive surface ready to stick. The pad covers the entire bottom with the hole being punched right along with work material when the punch is cycled. Pads come in sizes to fit stations 1/2" A through 4-1/2" E.



[Dimensions in Inches (mm)]

[Dimensions in Inches (mm)]

MATE PRECISION TECHNOLOGIES GLOBAL COVERAGE

WORLDWIDE HEADQUARTERS:
1295 Lund Boulevard, Anoka, Minnesota 55303 USA
Tel +1.763.421.0230 mate.com

orders@mate.com

LIT00682 Rev E 2021

Prices subject to change without notice