fictiv

# SHEET METAL DESIGN AND MANUFACTURING GUIDE



# Sheet Metal: An Overview

### What is Considered Sheet Metal?

Sheet metal is metal that has been rolled into thin and flat pieces or sheets. The term 'sheet metal' refers to the raw material (thin sheets of metal), the process of metalworking or making parts from thin sheets of metal.

Standard sheet metal size is up to 3'x6'. Anything larger would be non-standard.

### Sheet Metal Thickness

At Fictiv, sheet metal refers to thicknesses less than 6.5mm but greater than 0.15 mm. Anything above 6.5mm is considered plate stock, and anything below is considered foil.

## Sheet Metal Materials

The ability of a metal alloy to be drawn into a sheet is related to its ductility, but more specifically, the relationship between major and minor strain.

Common sheet metal materials are aluminum, steel, copper, stainless steel, and brass. Here is a list of standard sheet metal materials at Fictiv.

www.fictiv.com/sheet-metal-service

This guide will cover the processes that are utilized to manufacture sheet metal components and the design guidelines you should know about when designing sheet metal parts, such as:

- Bend radius
- Bend-to-hole distance
- Bend to cut-out distance
- Hole-to-edge distance
- Slot to bend distance
- Minimum hole diameter
- Bend allowance
- Flat blank sizing

Proper sheet metal design is important because it will impact the lifespan of your parts, increase quality, and improve the overall performance of the components and assembly. From a design for manufacturability (DFM) standpoint, improper design features may result in the deformation of the part during the cut-and-bend operation.

# SHEET METAL FORMING

## Press Brake Forming

Press-brake forming is the process whereby metal is bent or formed into shapes. The process works when a die and a punch are used to bend a sheet of metal. Press brake forming is most often utilized to form parts from ferrous alloy sheets and plates. It is a smooth process and involves operator skill to do it correctly.

Press brake forming is great for both prototype and production runs.

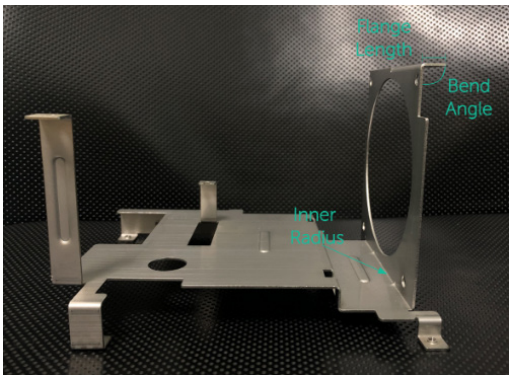
### Maximum Thickness for a Press Brake

The maximum sheet thickness is ideally less than 6.5mm, but thicknesses up to 25mm have been successfully formed with a press brake.

### Workpiece Dimensions

The length of the plate or sheet that can be bent is limited by the size of the press brake.

If the included angle of the bend is greater than 90°, if the bend radius is larger than stock thickness, or if the length of the bend is less than the bed length, a lower capacity press can be used.




### Types of Bends

Four types of bends can be made with a press brake machine:

1. **Minimum-radius**
   a) The smallest internal radius that can be formed without causing fracturing or creasing.
   b) For standard tooling, the minimum radius is 0.8-2X sheet thickness for sheet steel; for copper and aluminum alloys, the radius of the bend angle may be much smaller—as low as 0.25X sheet thickness.
   c) Smaller radii can be made with custom tooling

2. **Sharp**
   a) A bend with an internal radius smaller than the minimum recommended radius.
   b) Forms a crease or material fracture at the bend
   c) Not ideal for high-loading

3. **Perfect**
   a) A bend with a radius equal to or up to 25% more than the thickness of the sheet metal
   b) Desired bend for the highest strength

4. **Radius**
   a) A bend that is greater than 1.25X the sheet metal thickness

# Dies and Punches

### V-Bending Dies

V-bending dies/punches are the most common press-brake forming tools.

The die opening width is typically 8x the material thickness or more. The punch's nose radius should be greater than 1x the material thickness for low-carbon steel, increasing as the raw material becomes less formable.

V-bending die radius should be greater than the nose radius of the punch, typically by an amount equivalent to the raw material thickness. Here are the types of die/punches commonly utilized in press brake forming:

- 90° V-bending
- Radiused 90° bending
- Acute-angle bending
- Offset bending.
- Flattening, for three types of hems
- Combination bending and flattening.
- Gooseneck punch for multiple bends
- Special clearance punch for multiple bends



### Offset Dies

To bend a material of the same size and thickness with an offset bend die/press vs a 90-degree V-bend die/press requires 4X the force. So, offset bending should only be used for less than 3.2 mm thin sheet metal.

Offset depth > 6X raw material thickness for optimal bend stability.

### Radius Forming

Radius forming is done with a 90° die and a punch, each having a large radius, to create a bend with a radius equal to some multiplier of the sheet metal thickness. The punch bottoms out to where the inside radius of the sheet conforms to fit the punch radius over a curve.

An increase in tonnage decreases the internal radius of the bend because this brings the sheet metal closer to the punch nose. This also results in less springback.

*NOTE: The homogeneity of the bend depends significantly on the uniformity of the raw material thickness. Irregularities in thickness result in incomplete curvature of the bend radius.*

**Sharp bends**, or bends smaller than 90°, are often created with air bending. This type of bending means the press is set so that the sheet metal doesn't bottom out or fully reach the bottom die.

You should always use the minimum radius because the sharp bend is not desirable due to the impact of grain structure and also due to the less than perfect homogeneity typically seen in raw materials.

For the same die setup and tonnage, a softer material will bend to a smaller inside radius than a harder material (the softer material deflects more than the harder material).

# Eliminating Common Problems with Sheet Metal Bending

### K-Factor

The K-factor is an important concept to consider when designing parts for bending and is utilized to design the blanks or the flat sheets that will be cut and then bent. Most CAD software will have input for a K-factor, which means your design can be streamlined and simple. A good rule of thumb is to keep the K-factor between 0-0.5.

Here is the calculation for the K-factor:

$$K = \frac{180 * (\textit{Allowable Bend})}{\pi * (\textit{Bend Angle}) * T} - \frac{r}{T}$$

Lower K values can be found with smaller bend radii and softer materials.

## Springback

Springback is what occurs when a metal is bent. When the bending force which is distributed across the surface is not enough to overcome the material's modulus of elasticity, **elastic** deformation occurs, not **plastic** deformation. This elastic deformation impedes permanent deformation needed for a bend to remain. Interestingly, near the geometrical center of sheet metal, there is a zone with low stress that only experiences elastic deformation, even when under large bending forces. This means that this part is susceptible to springback.

As the stiffness of a material increases, the amount of springback you can expect increases. As a general rule of thumb, this is the maximum amount of springback that may occur (you should plan for this variation when designing sheet metal parts):

- Soft metals like copper and aluminum: 0.5°
- Steel: to 1°
- Stainless steel: 3°



Bend angle also plays a crucial role. The smaller the relative effect on the elastic area in the neutral zone, the smaller the springback. This is particularly true with small bend angles and small bend radii (a sharp tool).

In general, you can expect less springback for a sharper tool, smaller bend angles, and smaller bend radii.

Here are some practical applications:

Scenario One:

- Sheet thickness: 0.8 mm
- Bend radius: 1s
- Springback: 0.5-1°

Scenario Two:

- Sheet thickness at 0.8 mm
- Bend radius: 77s
- Springback: 30°

For flatness requirements, the deviation degree is something to consider. For example, if you have a sheet of 100 mm, the end of the sheet may deviate as much as 1.7 mm. This may have an impact on post-processing, such as robotic welding or assembly.

The good news is that compensating for springback in your design is relatively straightforward.

The expected springback may be calculated as follows:

$$D = \alpha \frac{R}{2.1 * t}$$

Where: D = springback angle expected

R = radius of the bend angle

t = sheet metal thickness

α = correction factor

The correction factor can be estimated as the following:

- α for cold rolled steel = 1
- α for copper = 0.5
- α for hot-rolled steel = 0.75
- α for stainless steel = 2.0

Utilization of dull tools and/or using a large punch angle can cause negative springback.

### Galling

Chips and flakes of material that stick to the press brake tool during bending may result in galling (adhesive wear due to impact with rough chips and flakes of material on the sheet metal), which is a greater concern with the bottom die. Galling may result in damage to the sheet metal surface and can be minimized by selecting the proper bend radius for the V-die.

### Bend Orientation vs Grain Direction

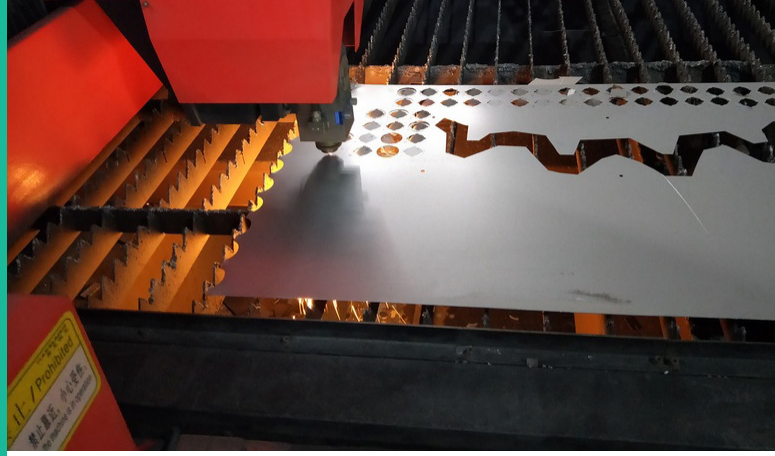
Anisotropic materials have a grain direction that impacts the bend direction. This is because anisotropic materials have varying material properties in varying directions or the material properties vary with varying crystallographic orientations.

For an anisotropic material, you always want your bend direction to run perpendicular to your grain direction. This traverse bending results in less cracking and a higher-strength bend.

For an anisotropic material, when your bend runs parallel to the grain direction, you get a weaker bend, and the outside bend radius is prone to cracking. If you must design a bend in the same direction as the grain, a larger bend radius will decrease the likelihood of cracking at the outer edge of the bend. At an atomic level, this is due to the formation of slip bands parallel to the grain direction.

Isotropic materials have homogeneity in their material properties, regardless of the direction. This means that the bending direction is not dependent on the orientation of the sheet metal.

# LASER CUTTING SHEET METAL



A laser beam is a column of high-intensity, single-wavelength light. For example, a CO2 laser only has light within the Infrared wavelength of the spectrum, so it's visible to the human eye. The beam of light is small but powerful, at only a diameter of approximately 0.75" as it goes from the laser resonator (generator) through the beam path. Beam benders (or mirrors) may bounce the laser beam around before it is focused on the sheet metal. Along with the laser beam, compressed gas, like oxygen or nitrogen, flows through the nozzle bore.

The ability of the laser beam to generate enough heat to cut through (or really melt) the material it impacts is due to beam focusing. This may be accomplished by using lenses or curved mirrors within the laser cutting head. The ideal laser beam is consistent and perfectly round, all of which create a lot of heat density at the point of impact on the sheet metal.

Whether or not the metal melts and is blown off or vaporizes, and removal is not needed, is dependent on the material being cut. Aluminum and stainless steel melt under the laser, and the molten metal is blown off the kerf with compressed gas. Mild steel vaporizes when cut with a laser.

The optimal metal chosen for laser cutting has a low reflectivity, a smaller thickness, and a low flammability. For example, copper and magnesium alloys aren't ideal for laser cutting. The thicker the sheet metal, the higher the risk for edge distortion.



### Maximum Thickness for a Laser Cutting

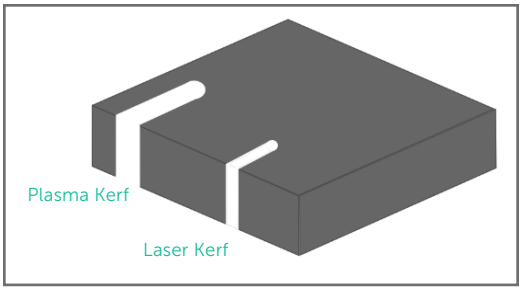
The maximum sheet thickness is based on the material properties and the laser power.

**Maximum Laser Cutting Thickness Chart at 500 Watts**

| Metal Alloy Family | Thickness (mm) | Thickness (in) |
|---|---|---|
| Aluminum | 2 | 0.08 |
| Stainless Steel | 3 | 0.12 |
| Brass | 2 | 0.08 |
| Carbon Steel | 6 | 0.24 |

### Workpiece Dimensions

The length of the plate or sheet that can be bent is limited by the size of the laser-cutting work table or platform where the raw material is laid. Fictiv has the ability to laser cut parts from raw sheet stock of up to 6mx2.5m in size.




### Kerf

Laser cutting starts with an initial pierce or hole in the material and then the laser follows a predetermined path to cut each part out. The kerf or cut width depends on the laser beam diameter but can be as small as a few thousandths of an inch. The entry hole or initial pierce is larger than the exit hole; it may be up to a diameter equivalent to 0.2x the sheet metal thickness.

### Design Tips with Laser Cutting

Any hole diameter should be at least greater than one-half of the sheet metal thickness. For the highest quality holes, aim for a diameter greater than or equal to the sheet metal thickness.



The more you can connect lines or reduce the amount of time the machine has to start and stop, the greater reduction in cost per piece you will achieve.

When designing laser-cut parts for assembly, it's a good idea to create a tab and slot design for the alignment of parts during assembly or welding. This will speed up the process and help reduce costs.

Always put a radius on corners and edges, if possible. Sharp corners result in quality issues due to burning and formation of dross. For materials with lower melting points, the corners may burn off entirely.

### Advantages of Laser Cutting

Advantages of laser cutting are burr-free edges, better perpendicularity, small kerf, low heat affected zone, complex profile, tight tolerances, and high-speed cutting.

# Plasma Cutting Sheet Metal

While handheld plasma cutters are available, a CNC plasma cutter is the best option for cutting parts.

In plasma cutting, a gas is superheated with an electric arc and compressed to the 4th state of matter, plasma that flows through a torch, and the plasma jet cuts through the sheet metal. Superheated means up to 40,000 °F; the electric arc is about 5,000 VAC.

A precision CNC plasma cutter has a uniquely shaped shield cap and orifice diameter to reduce the plasma arc, resulting in a smaller kerf and low bevel angles, which means a much cleaner cut.

### What can a plasma cutter cut?

Compared to laser cutters, plasma cutters can cut through much thicker material, even in the range of 1-2 inches. It is ideal for steel alloys but is often utilized to cut aluminum alloys.

Plasma cutting is not ideal for thin, detailed parts that require tight tolerances or high precision. The kerf is much wider, and the cut edges are not as clean as laser-cut edges.

### Design Guidelines for Plasma Cut Parts

1. The hole diameter cut with a plasma cutter should be greater than or equal to the sheet metal thickness. A smaller hole will result in quality issues.
2. Spacing parts creatively and trying to connect them will reduce the amount of time the machine has to start and stop, resulting in a cost/piece reduction.
3. Since the kerf is larger with plasma cutting, you need to account for it during design. Subtract half the kerf size from the part's frame and then add the other half to the inner section.
4. Adding cutting nodes opposite each slot will reduce tension during cutting and lead to less distortion.
5. Utilize part nesting software to reduce waste as much as possible.
6. Parts must have a smooth radius on all sharp corners. The plasma kerf will not allow for sharp corners.

# CNC Turret Punch

A CNC Turret Punching Press or a CNC Turret Punch or a Punch Press essentially acts as a metal hole puncher. It is well-suited for high volumes because custom tooling is required per part geometry. The part-cutting mechanism is a punch that makes contact on the top surface of the sheet and a corresponding die of a negative impression that the sheet metal rests on top of. This results in a highly controlled shearing of material to the desired shape.

### Geometries Suitable for Punching

Punching is versatile and allows you to create a wide variety of shapes due to the endless combination of punch and die designs, and the intelligent processing mechanisms afforded by CNC punch presses.

Here are some shapes that are well-suited for punch presses:

1. Slots
2. Holes
3. Simple cutouts
4. Notches
5. Custom geometries
6. Louvers/perforations
7. Snap lines
8. Blank production (for subsequent ops)
9. Piercing

Here are some complex geometries that may cost a little more to produce with a punch press:

1. Lances
2. Tabs
3. Knockouts
4. Bridges
5. Embosses
6. Countersink holes
7. Hinges
8. Locking tabs
9. Ribs/Offsets

## Different Types of Punch Pressing

1. **Blanking**: this can be used to make blank parts for subsequent processing (like bending) or to create whole parts like washers; creating a hole and removing the unwanted material.
2. **Piercing**: used to make holes without material removal; best suited for fastener installation like rivets and screws.
3. **Slotting**: cutting a hole that is oblong, or a long and narrow slot
4. **Notching**: removal of material from the edge of a sheet of metal; often used to make tabs or notches.
5. **Embossing**: forming raised or recessed shapes or letters on the surface without material removal
6. **Lancing**: a process whereby a sheet is cut through part of its length and bent; good for making louvers and tabs.
7. **Perforating**: cutting of several tiny holes or shapes in a sheet; often used for drainage or decorative purposes.
8. **Stamping**: punch dies are utilized to stamp the parts; good for stamping part identification.



## Considerations for Punch Pressing

While punch pressing is typically a clean process that doesn't require much post-processing, burrs can be created at top-hole edges.

For tight tolerances, a CNC punch press is almost always necessary. While up to 1" of sheet metal can be punched, the potential for edge distortion may increase as sheet thickness increases.

Punch pressing is often a good manufacturing method for simple geometries and high volumes because the tooling requires little maintenance, and the process produces little waste.

# SHEET METAL DESIGN FOR MANUFACTUR-ABILITY

## General Sheet Metal Design Guidelines

*Note: These are all general guidelines. Actual recommendations depend on sheet metal material/grade and often whether bending is longitudinal or transverse.*

### A Note on Tolerances for Sheet Metal

All manufacturing tolerances for all processes must be added together. These are known as uncontrolled tolerances, and they add up. For example, add all tolerances for laser-cutting, bending, and welding to get the overall tolerance. At the very minimum, most sheet metal parts require two machines: one to cut and one to bend.

Another important note is that sheet metal manufacturing tolerances will be larger than those achieved with CNC machining. This is a less accurate but equally important manufacturing method.

A profile cutting tolerance results from the minor positional variance from cutting sheet metal. At a minimum, you can expect this tolerance to be +/- 0.005”. You can also

expect this to be the additional tolerance minimum for each bend made on the part. You can expect a tolerance of about 1° for each bend.

### Bend Design Constraints

#### Bend Direction

Remember to run bend lines perpendicular to grain direction for anisotropic materials.

Bend allowance should be compensated for when designing sheet metal parts.

#### Bend Allowance

Bend allowance is equivalent to the length of the arc of the neutral axis of the bent area. Bend allowance is the length of extra material that should be cut to allow for bending without disrupting the part dimensions. This provides you with the dimensions you need for your flat pattern. It is the amount of material that is additionally needed for the curvature of the bend.



Bend allowance can be calculated as:

$$\text{Bend Allowance} = \frac{\theta * \pi}{180} * (r + K * SMT)$$

Where: $\theta$ = *bend angle in radians or degrees*

r = inside radius of bend

K = K-factor (see press brake section for equation)

SMT= sheet metal thickness

The flat blank length you need may be calculated as the following:

$$L_{flat\ blank} = L_{first\ leg} + L_{second\ leg} + BA$$

Where: L = length

BA = bend allowance

### Bend Height

Bend height is the term used to define the height or length of the flange made after a bend is made. The bend height should be greater than or equal to 2x the sheet metal thickness (SMT) plus the radius of the bend (internal):

$$\text{Bend Height} \geq 2 * \text{SMT} + R$$

Where: SMT = sheet metal thickness

R = internal radius of bend

### Avoiding Bent Flange Interference

Any time two walls are bent and occupy two sides of a corner, there is a chance of overlap or interference. Adding an offset will help to reduce the likelihood of interference. An enlarged offset will dramatically reduce the impact of inference on sheet metal bending to a corner, but a good target minimum is an offset of 0.2 mm. Keep in mind that welding a larger offset gap is more difficult.



### Bend Reliefs

Designing bend reliefs is a good practice to prevent the failure of bent parts at areas of high-stress concentrations. A bend relief should be placed to separate material on either side of a bend by removing material from the sheet metal on both sides of the bend.

Bend relief can also be used to prevent tearing when a bend is close to the edge of a sheet.

The bend relief depth should be larger than the radius of the bend and the width larger than the thickness of the sheet metal.

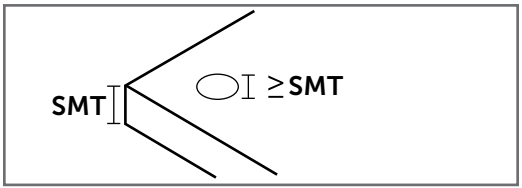
Bend reliefs can take many shapes and sizes, such as circular holes, rectangular slits, or half-slots.

### Clearance between Slots and Bends

Slots should be greater than the bend radius plus 4x the sheet metal thickness (SMT) away from the bend.

$$\textit{slot dist to bend} = r_{bend} + 4 * \text{SMT}$$

## Minimum Hole Diameter




*Non-laser cut hole minimum diameter*

A sheet metal hole's diameter should be greater than or equal to the sheet metal thickness (SMT) for plasma cutting, stamping, and blanking:

$$Hole\theta_{\,non\ laser\ cut} \geq SMT$$

The hole diameter can be smaller for laser cutting because of the tapered nature due to the difference in the entry and exit diameter:

$$Hole\theta_{\,laser\ cut} \geq 0.2 * SMT$$



## Minimum Bend-to-Hole Distance

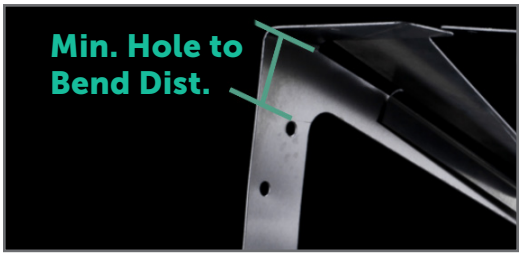



*Image depicts the minimum distance a hole must be from a bend to prevent hole distortion.*

Keeping holes away from bends is necessary to prevent hole deformation. Adding holes after bending is costly because it requires another machining process.

The following formula can be used to determine the minimum distance from the outside of a hole (with a diameter less than 25mm) to a bend:

$$D = 2 * SMT + r$$

Where: D = distance
SMT = sheet metal thickness
r = radius of the hole

OR a conservative calculation:

$$Dist\ hole\ to\ bend \geq 3 * SMT$$

### Distance from Hole to Sheet Metal Edge

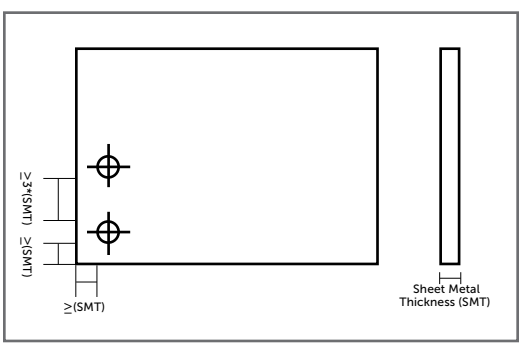
Keeping holes away from the edge of sheets of metal is also critical to ensure the integrity of the holes.

The minimum distance for laser-cut holes should be 1X sheet metal thickness (SMT).

$$Dist\ hole\ to\ SM\ Edge_{Laser\ cut} = SMT$$

For Stamping or Blanking, the distance should be larger:

$$Dist\ hole\ to\ SM\ Edge_{Stamp/Blank} = 2 * SMT$$




*Non-laser cut holes min distances.*



### Distance from Hole to Hole

Holes that are too close will result in sheet metal deformation between the holes, risking meeting tolerances and assembly requirements.

For plasma cutting, stamping, or blanking, holes should be no closer than 6x the sheet metal thickness (SMT)*

$$Dist\ hole\ to\ hole_{non\ laser\ cut} \geq 6 * SMT$$

*The distance from one countersink hole to another should be no less than 8x the sheet metal thickness.

For laser cutting, holes should be no closer than 2X the sheet metal thickness (SMT):

$$Dist\ hole\ to\ hole_{laser\ cut} \geq 2 * SMT$$

### Min Distance Between Parts — Laser Cutting

The minimum distance between parts when laser cutting should be 2-4X the kerf width. Ask your manufacturer for the kerf width, or if you don't have access to that information, a spacing of 2 mm is a good bet.

*part distance ≥ 2 * kerf width*

### Fillets

All sheet metal corners should be rounded or filleted to reduce handling injuries. A fillet is the less costly option and should be around 0.5x sheet metal thickness.



# About Fictiv

**Custom Manufactured Parts Done Right.**

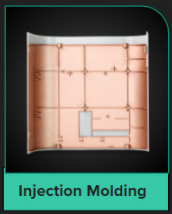
At Fictiv, we're not just simplifying sourcing; we're revolutionizing it. We provide you with an agile, meticulously vetted, and expertly managed global supply chain, accompanied by comprehensive engineering and program management support that adapts seamlessly to your evolving needs at every development stage.

Whether it's for prototyping, design iterations, mass production, or continuous improvement, our expansive global network of over 250 world-class manufacturing centers ensures that your requirements are met precisely when you need them.

Let Fictiv be your trusted partner in turning visionary ideas into tangible realities faster and more efficiently than you've ever imagined.

**Further Resources:**

Press Brake Technology: A Guide to Precision Sheet Metal (published by the Society of Manufacturing Engineers)



Injection Molding



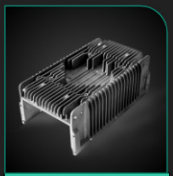
CNC Machining



3D Printing



Sheet Metal



Die Casting



Compression Molding



Urethane Casting

# fictiv

Let’s build something great, together.

fictiv.com