Overviews

Processes
- Polymer Processing
  - Blow Molding
  - Injection Molding
  - Metal Injection Molding
  - Thermoforming
- Metal Casting
  - Centrifugal Casting
  - Die Casting
  - Investment Casting
  - Permanent Mold
  - Sand Casting
  - Shell Mold Casting
- Machining
  - Milling
  - Turning
  - Hole-making
  - Drill Size Chart
  - Tap Size Chart
- Sheet Metal Fabrication
  - Forming
  - Cutting with shear
  - Cutting without shear
  - Gauge Size Chart
- Additive Fabrication
  - SLA
  - FDM
  - SLS
  - DMLS
  - 3D Printing
  - Inkjet Printing
  - Jetted Photopolymer
  - LOM

Materials
- Metals
- Plastics

Case Studies
- Cost Analysis
- Part Redesign
- Product Development

Resources
- Curriculum Resources
- Glossary

Sheet Metal Forming

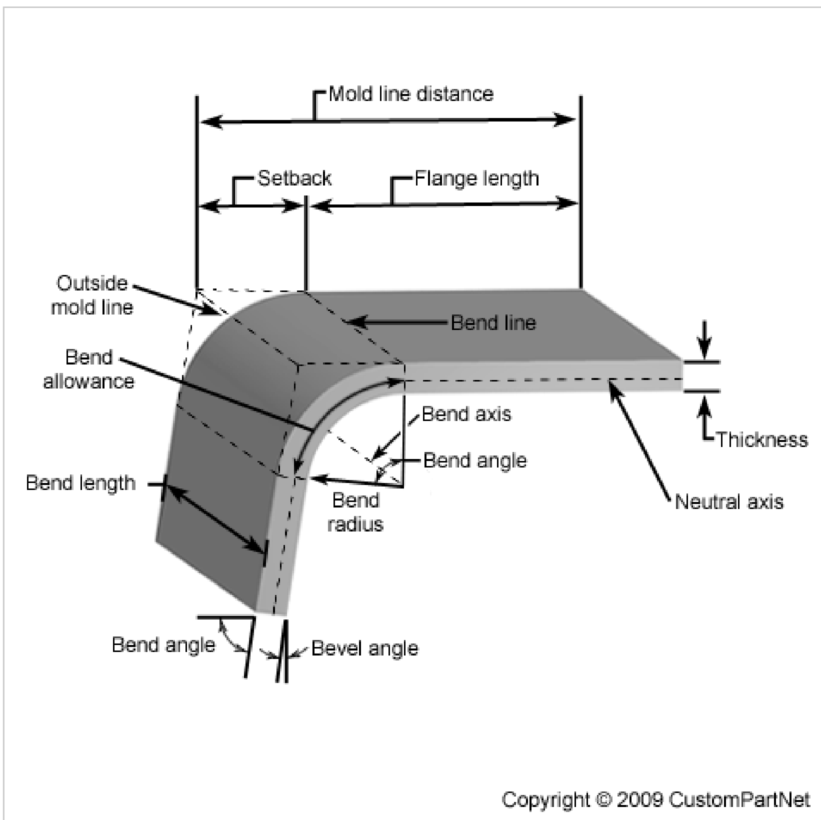
Sheet metal forming processes are those in which force is applied to a piece of sheet metal to modify its geometry rather than remove any material. The applied force stresses the metal beyond its yield strength, causing the material to plastically deform, but not to fail. By doing so, the sheet can be bent or stretched into a variety of complex shapes. Sheet metal forming processes include the following:

- Bending
- Roll forming
- Spinning
- Deep Drawing
- Stretch forming

Return to top

Bending

Bending is a metal forming process in which a force is applied to a piece of sheet metal, causing it to bend at an angle and form the desired shape. A bending operation causes deformation along one axis, but a sequence of several different operations can be performed to create a complex part. Bent parts can be quite small, such as a bracket, or up to 20 feet in length, such as a large enclosure or chassis. A bend can be characterized by several different parameters, shown in the image below.

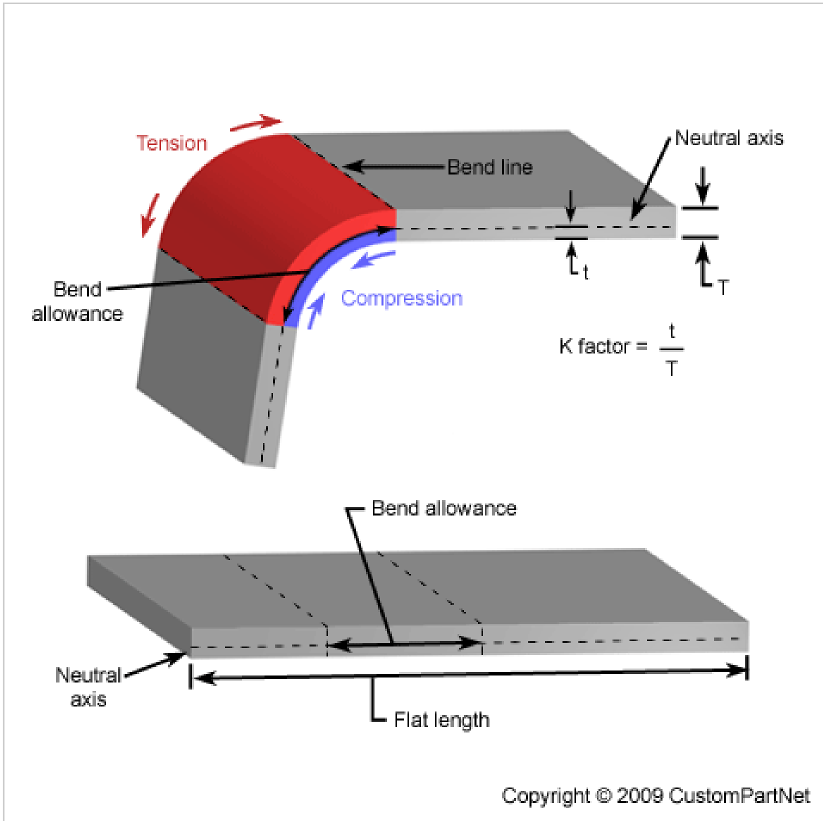


Bending Diagram

- *Bend line* - The straight line on the surface of the sheet, on either side of the bend, that defines the end of the level flange and the start of the bend.
- *Outside mold line* - The straight line where the outside surfaces of the two flanges would meet, were they to continue. This line defines the edge of a mold that would bound the bent sheet metal.
- *Flange length* - The length of either of the two flanges, extending from the edge of the sheet to the bend line.
- *Mold line distance* - The distance from either end of the sheet to the outside mold line.

- *Setback* - The distance from either bend line to the outside mold line. Also equal to the difference between the mold line distance and the flange length.
- *Bend axis* - The straight line that defines the center around which the sheet metal is bent.
- *Bend length* - The length of the bend, measured along the bend axis.
- *Bend radius* - The distance from the bend axis to the inside surface of the material, between the bend lines. Sometimes specified as the inside bend radius. The outside bend radius is equal to the inside bend radius plus the sheet thickness.
- Bend angle - The angle of the bend, measured between the bent flange and its original position, or as the included angle between perpendicular lines drawn from the bend lines.
- *Bevel angle* - The complimentary angle to the bend angle.

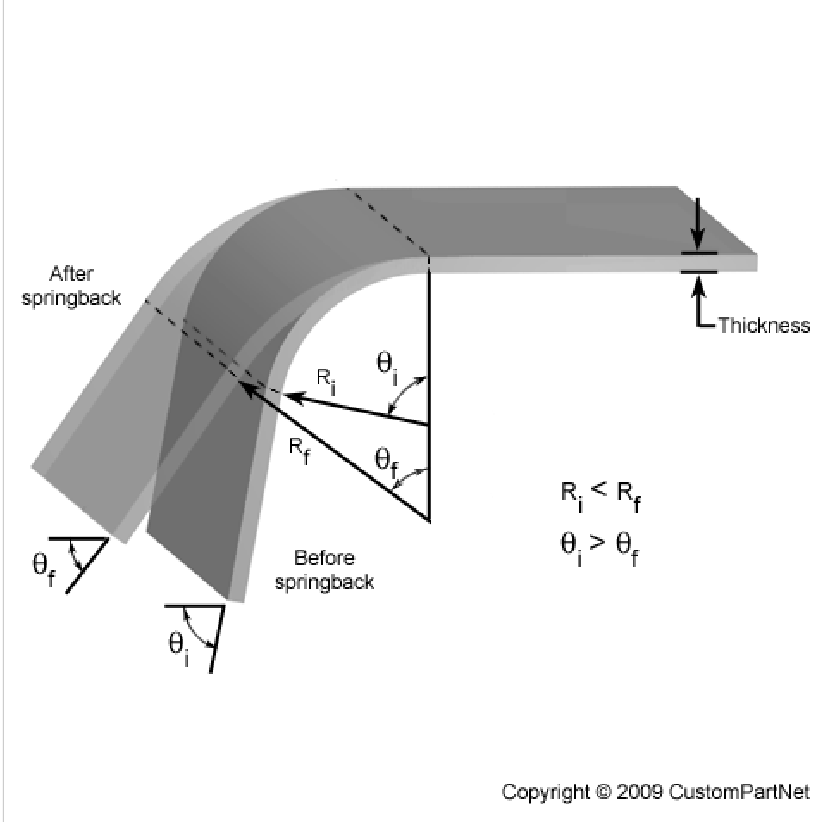
The act of bending results in both tension and compression in the sheet metal. The outside portion of the sheet will undergo tension and stretch to a greater length, while the inside portion experiences compression and shortens. The neutral axis is the boundary line inside the sheet metal, along which no tension or compression forces are present. As a result, the length of this axis remains constant. The changes in length to the outside and inside surfaces can be related to the original flat length by two parameters, the bend allowance and bend deduction, which are defined below.



Neutral Axis

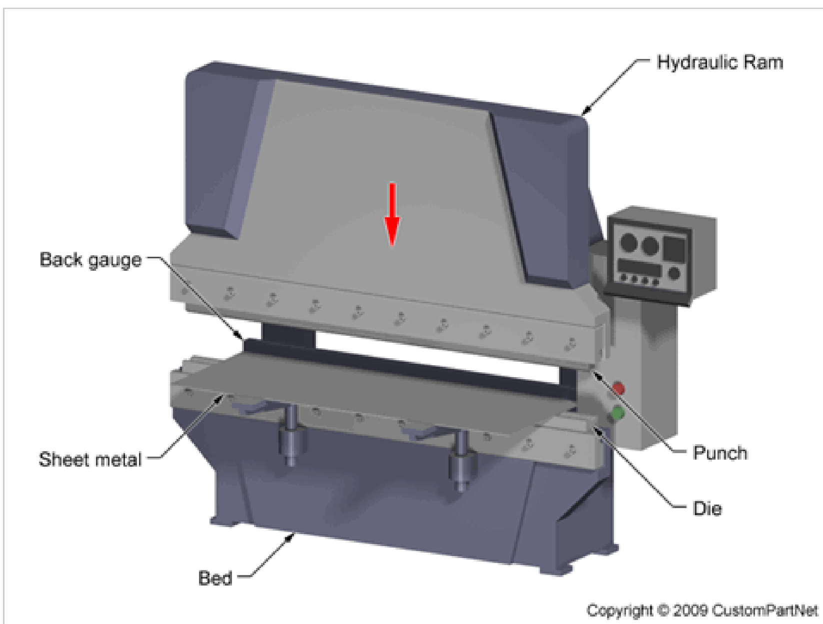
- *Neutral axis* - The location in the sheet that is neither stretched nor compressed, and therefore remains at a constant length.
- *K-factor* - The location of the neutral axis in the material, calculated as the ratio of the distance of the neutral axis (measured from the inside bend surface) to the material thickness. The K-factor is dependent upon several factors (material, bending operation, bend angle, etc.) and is typically greater than 0.25, but cannot exceed 0.50.
- *Bend allowance* - The length of the neutral axis between the bend lines, or in other words, the arc length of the bend. The bend allowance added to the flange lengths is equal to the total flat length.
- *Bend deduction* - Also called the bend compensation, the amount a piece of material has been stretched by bending. The value equals the difference between the mold line lengths and the total flat length.

When bending a piece of sheet metal, the residual stresses in the material will cause the sheet to springback slightly after the bending operation. Due to this elastic recovery, it is necessary to over-bend the sheet a precise amount to achieve the desired bend radius and bend angle. The final bend radius will be greater than initially formed and the final bend angle will be smaller. The ratio of the final bend angle to the initial bend angle is defined as the springback factor, K_S. The amount of springback depends upon several factors, including the material, bending operation, and the initial bend angle and bend radius.

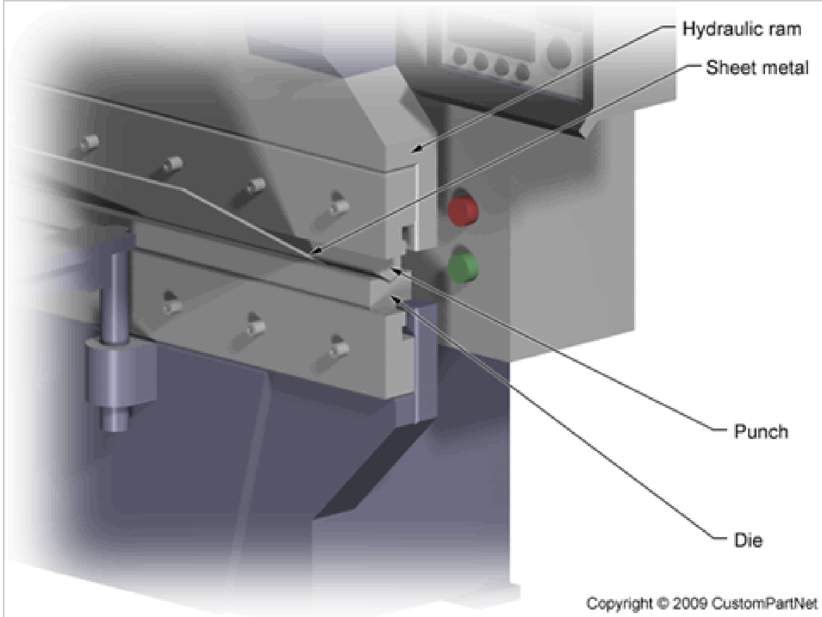


Springback

Bending is typically performed on a machine called a press brake, which can be manually or automatically operated. For this reason, the bending process is sometimes referred to as press brake forming. Press brakes are available in a range of sizes (commonly 20-200 tons) in order to best suit the given application. A press brake contains an upper tool called the punch and a lower tool called the die, between which the sheet metal is located. The sheet is carefully positioned over the die and held in place by the back gauge while the punch lowers and forces the sheet to bend. In an automatic machine, the punch is forced into the sheet under the power of a hydraulic ram. The bend angle achieved is determined by the depth to which the punch forces the sheet into the die. This depth is precisely controlled to achieve the desired bend. Standard tooling is often used for the punch and die, allowing a low initial cost and suitability for low volume production. Custom tooling can be used for specialized bending operations but will add to the cost. The tooling material is chosen based upon the production quantity, sheet metal material, and degree of bending. Naturally, a stronger tool is required to endure larger quantities, harder sheet metal, and severe bending operations. In order of increasing strength, some common tooling materials include hardwood, low carbon steel, tool steel, and carbide steel.

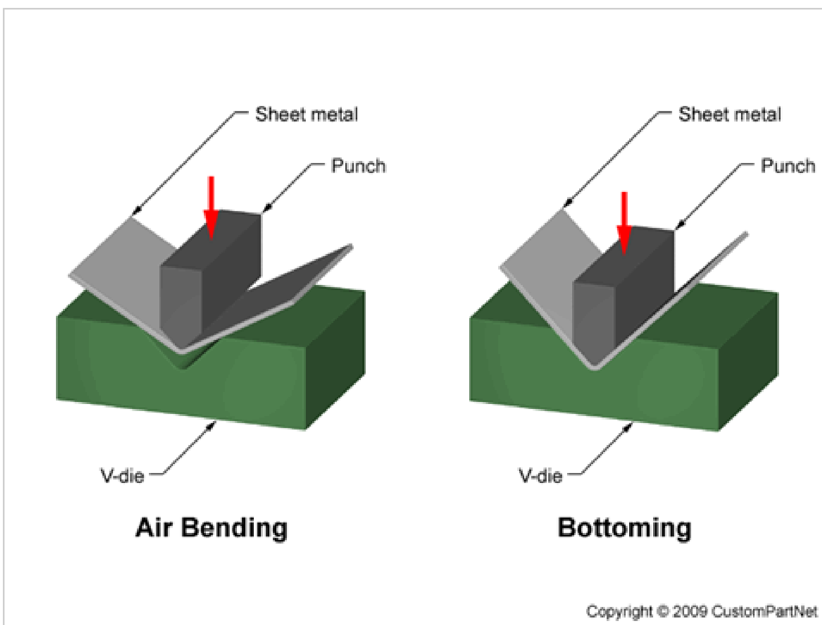


Press Brake (Open)



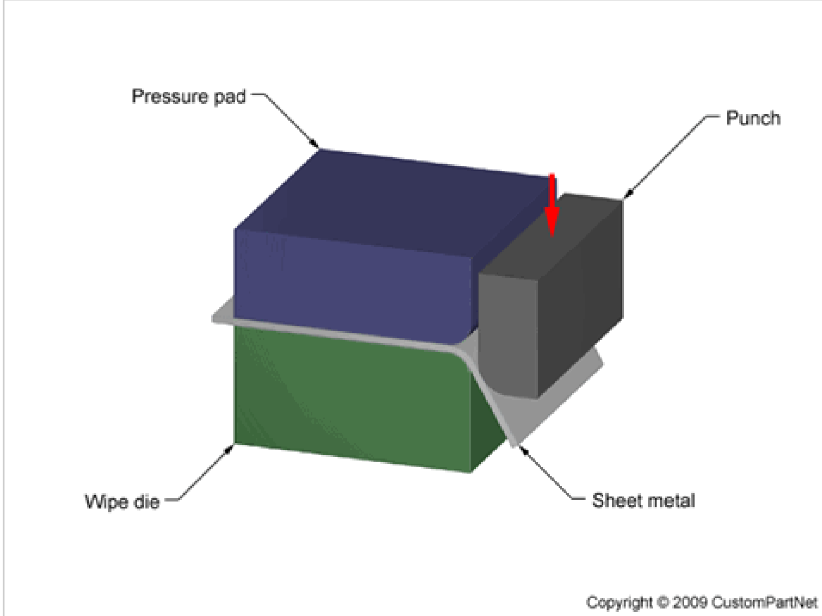
Press Brake (Closed)

While using a press brake and standard die sets, there are still a variety of techniques that can be used to bend the sheet. The most common method is known as V-bending, in which the punch and die are "V" shaped. The punch pushes the sheet into the "V" shaped groove in the V-die, causing it to bend. If the punch does not force the sheet to the bottom of the die cavity, leaving space or air underneath, it is called "air bending". As a result, the V-groove must have a sharper angle than the angle being formed in the sheet. If the punch forces the sheet to the bottom of the die cavity, it is called "bottoming". This technique allows for more control over the angle because there is less springback. However, a higher tonnage press is required. In both techniques, the width of the "V" shaped groove, or die opening, is typically 6 to 18 times the sheet thickness. This value is referred to as the die ratio and is equal to the die opening divided by the sheet thickness.



V Bending

In addition to V-bending, another common bending method is wipe bending, sometimes called edge bending. Wipe bending requires the sheet to be held against the wipe die by a pressure pad. The punch then presses against the edge of the sheet that extends beyond the die and pad. The sheet will bend against the radius of the edge of the wipe die.



Wipe Bending

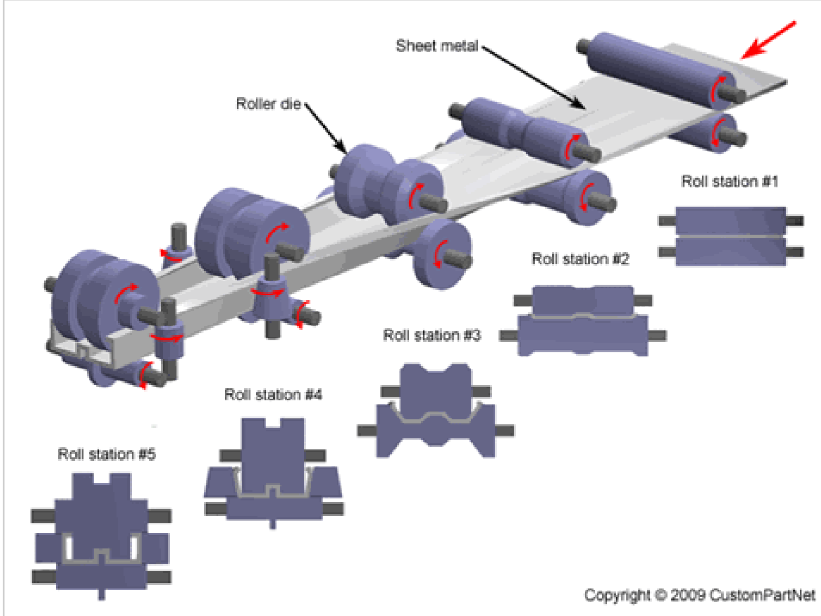
Design rules

- Bend location - A bend should be located where enough material is present, and preferably with straight edges, for the sheet to be secured without slipping. The width of this flange should be equal to at least 4 times the sheet thickness plus the bend radius.
- Bend radius
  - Use a single bend radius for all bends to eliminate additional tooling or setups
  - Inside bend radius should equal at least the sheet thickness
- Bend direction - Bending hard metals parallel to the rolling direction of the sheet may lead to fracture. Bending perpendicular to the rolling direction is recommended.
- Any features, such as holes or slots, located too close to a bend may be distorted. The distance of such features from the bend should be equal to at least 3 times the sheet thickness plus the bending radius.
- In the case of manual bending, if the design allows, a slot can be cut along the bend line to reduce the manual force required.

Return to top

Roll forming

Roll forming, sometimes spelled rollforming, is a metal forming process in which sheet metal is progressively shaped through a series of bending operations. The process is performed on a roll forming line in which the sheet metal stock is fed through a series of roll stations. Each station has a roller, referred to as a roller die, positioned on both sides of the sheet. The shape and size of the roller die may be unique to that station, or several identical roller dies may be used in different positions. The roller dies may be above and below the sheet, along the sides, at an angle, etc. As the sheet is forced through the roller dies in each roll station, it plastically deforms and bends. Each roll station performs one stage in the complete bending of the sheet to form the desired part. The roller dies are lubricated to reduce friction between the die and the sheet, thus reducing the tool wear. Also, lubricant can allow for a higher production rate, which will also depend on the material thickness, number of roll stations, and radius of each bend. The roll forming line can also include other sheet metal fabrication operations before or after the roll forming, such as punching or shearing.



Roll Forming Line

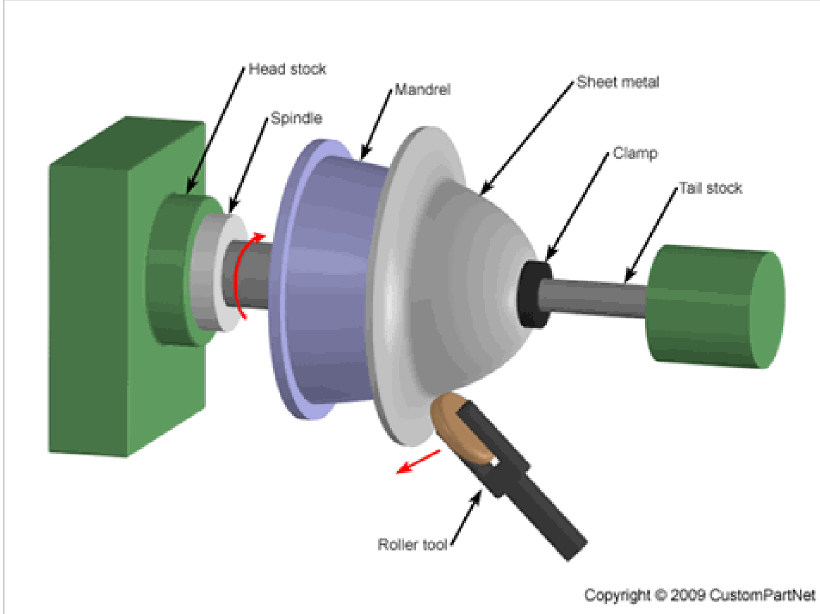
The roll forming process can be used to form a sheet into a wide variety of cross-section profiles. An open profile is most common, but a closed tube-like shape can be created as well. Because the final form is achieved through a series of bends, the part does not require a uniform or symmetric cross-section along its length. Roll forming is used to create very long sheet metal parts with typical widths of 1-20 inches and thicknesses of 0.004-0.125 inches. However wider and thicker sheets can be formed, some up to 5 ft. wide and 0.25 inches thick. The roll forming process is capable of producing parts with tolerances as tight as ±0.005 inches. Typical roll formed parts include panels, tracks, shelving, etc. These parts are commonly used in industrial and commercial buildings for roofing, lighting, storage units, and HVAC applications.

Return to top

Spinning

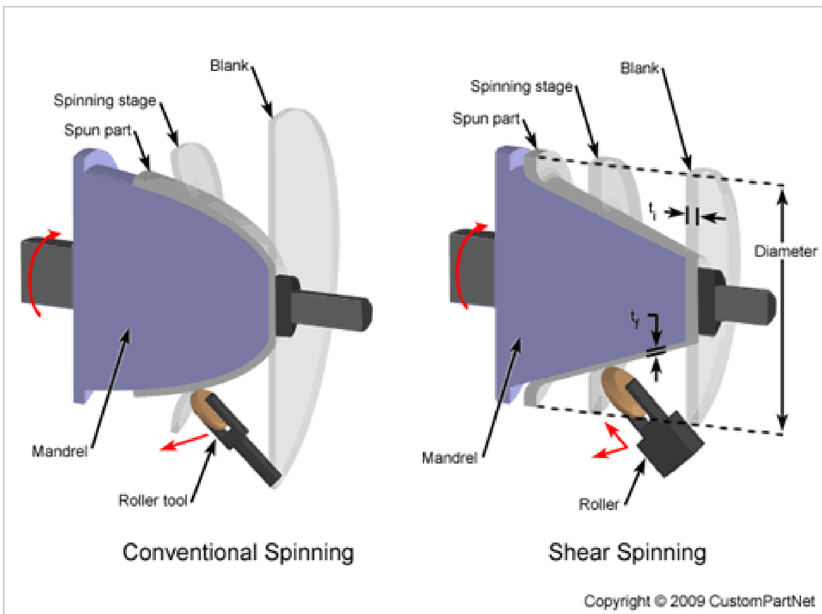
Spinning, sometimes called spin forming, is a metal forming process used to form cylindrical parts by rotating a piece of sheet metal while forces are applied to one side. A sheet metal disc is rotated at high speeds while rollers press the sheet against a tool, called a mandrel, to form the shape of the desired part. Spun metal parts have a rotationally symmetric, hollow shape, such as a cylinder, cone, or hemisphere. Examples include cookware, hubcaps, satellite dishes, rocket nose cones, and musical instruments.

Spinning is typically performed on a manual or CNC lathe and requires a blank, mandrel, and roller tool. The blank is the disc-shaped piece of sheet metal that is pre-cut from sheet stock and will be formed into the part. The mandrel is a solid form of the internal shape of the part, against which the blank will be pressed. For more complex parts, such as those with reentrant surfaces, multi-piece mandrels can be used. Because the mandrel does not experience much wear in this process, it can be made from wood or plastic. However, high volume production typically utilizes a metal mandrel. The mandrel and blank are clamped together and secured between the headstock and tailstock of the lathe to be rotated at high speeds by the spindle. While the blank and mandrel rotate, force is applied to the sheet by a tool, causing the sheet to bend and form around the mandrel. The tool may make several passes to complete the shaping of the sheet. This tool is usually a roller wheel attached to a lever. Rollers are available in different diameters and thicknesses and are usually made from steel or brass. The rollers are inexpensive and experience little wear allowing for low volume production of parts.



Spinning Lathe

There are two distinct spinning methods, referred to as conventional spinning and shear spinning. In conventional spinning, the roller tool pushes against the blank until it conforms to the contour of the mandrel. The resulting spun part will have a diameter smaller than the blank, but will maintain a constant thickness. In shear spinning, the roller not only bends the blank against the mandrel, it also applies a downward force while it moves, stretching the material over the mandrel. By doing so, the outer diameter of the spun part will remain equal to the original blank diameter, but the thickness of the part walls will be thinner.



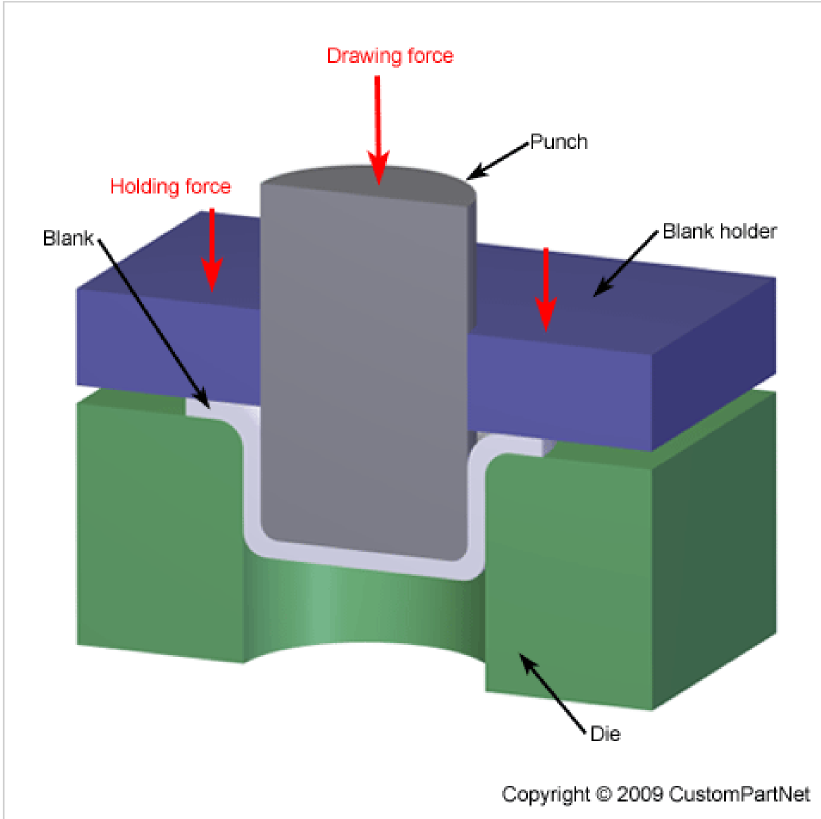
Conventional Spinning vs. Shear Spinning

Return to top

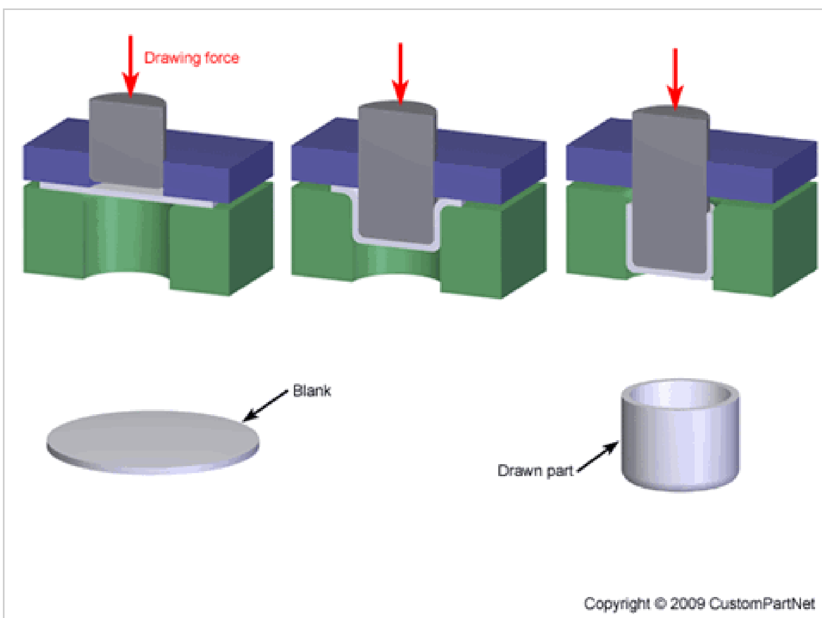
Deep Drawing

Deep drawing is a metal forming process in which sheet metal is stretched into the desired part shape. A tool pushes downward on the sheet metal, forcing it into a die cavity in the shape of the desired part. The tensile forces applied to the sheet cause it to plastically deform into a cup-shaped part. Deep drawn parts are characterized by a depth equal to more than half of the diameter of the part. These parts can have a variety of cross sections with straight, tapered, or even curved walls, but cylindrical or rectangular parts are most common. Deep drawing is most effective with ductile metals, such as aluminum, brass, copper, and mild steel. Examples of parts formed with deep drawing include automotive bodies and fuel tanks, cans, cups, kitchen sinks, and pots and pans.

The deep drawing process requires a blank, blank holder, punch, and die. The blank is a piece of sheet metal, typically a disc or rectangle, which is pre-cut from stock material and will be formed into the part. The blank is clamped down by the blank holder over the die, which has a cavity in the external shape of the part. A tool called a punch moves downward into the blank and draws, or stretches, the material into the die cavity. The movement of the punch is usually hydraulically powered to apply enough force to the blank. Both the die and punch experience wear from the forces applied to the sheet metal and are therefore made from tool steel or carbon steel. The process of drawing the part sometimes occurs in a series of operations, called draw reductions. In each step, a punch forces the part into a different die, stretching the part to a greater depth each time. After a part is completely drawn, the punch and blank holder can be raised and the part removed from the die. The portion of the sheet metal that was clamped under the blank holder may form a flange around the part that can be trimmed off.



Deep Drawing

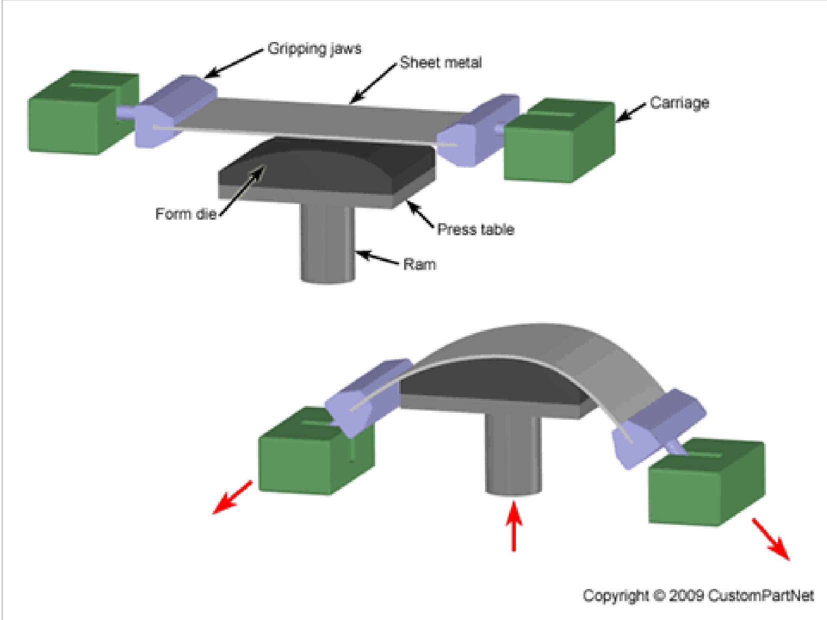


Deep Drawing Sequence

Return to top

Stretch Forming

Stretch forming is a metal forming process in which a piece of sheet metal is stretched and bent simultaneously over a die in order to form large contoured parts. Stretch forming is performed on a stretch press, in which a piece of sheet metal is securely gripped along its edges by gripping jaws. The gripping jaws are each attached to a carriage that is pulled by pneumatic or hydraulic force to stretch the sheet. The tooling used in this process is a stretch form block, called a form die, which is a solid contoured piece against which the sheet metal will be pressed. The most common stretch presses are oriented vertically, in which the form die rests on a press table that can be raised into the sheet by a hydraulic ram. As the form die is driven into the sheet, which is gripped tightly at its edges, the tensile forces increase and the sheet plastically deforms into a new shape. Horizontal stretch presses mount the form die sideways on a stationary press table, while the gripping jaws pull the sheet horizontally around the form die.



Stretch Forming

Stretch formed parts are typically large and possess large radius bends. The shapes that can be produced vary from a simple curved surface to complex non-uniform cross sections. Stretch forming is capable of shaping parts with very high accuracy and smooth surfaces. Ductile materials are preferable, the most commonly used being aluminum, steel, and titanium. Typical stretch formed parts are large curved panels such as door panels in cars or wing panels on aircraft. Other stretch formed parts can be found in window frames and enclosures.

[Return to top](#)