Extract of the book:
The Fascinating World of Sheet Metal

The Fascinating World of Sheet Metal

Publisher and Editor:
TRUMPF GmbH + Co.
Johann Maus Straße 2
71254 Ditzingen
Federal Republic of Germany

Authors:
Dr. Hubert Bitzel, Johanna Borcherdt, Jörg Müller, Frank Neidhart, Dr. Klaus Parey, Armin Rau, Sabine Riecke, Annegret Schmid, Gabriele Trentmann, Gerald Vorländer, Klaus Zimmermann.

Graphics:
Gisela Ostermann, Rainer Deuschle, Bernhard Walter

Translation:
unitext®
Technische Übersetzungen und Technische Dokumentationen GmbH & Co. KG, Berlin

Project coordination:
Annegret Schmid

Cover photo:
Udo Loster Fotostudio GmbH

Layout design:
Heike Hauß, Jürgen G. Rothfuß

Printing:
Rondo-Druck, Ebersbach-Roßwälden

The original edition was published in German by the Dr. Josef Raabe Verlags-GmbH, Stuttgart in 1996 and entitled: »Faszination Blech - Flexible Bearbeitung eines vielseitigen Werkstoffes«.

Bending – Sheet metal processing with corners and edges

Origami is the Japanese art of creating intricate shapes from flat pieces of paper without using any technical aids. In the Western world it is also difficult not to find the sight of folded flowers, swans or crabs a fascinating one. On the other hand, anyone who takes time to study a drink vending machine in a railway hall, the counters of a fast-food restaurant or the range hood in a kitchen will hardly show the same enthusiasm. Nevertheless, the manufacture of these metal components is based on a similar principle as Origami.

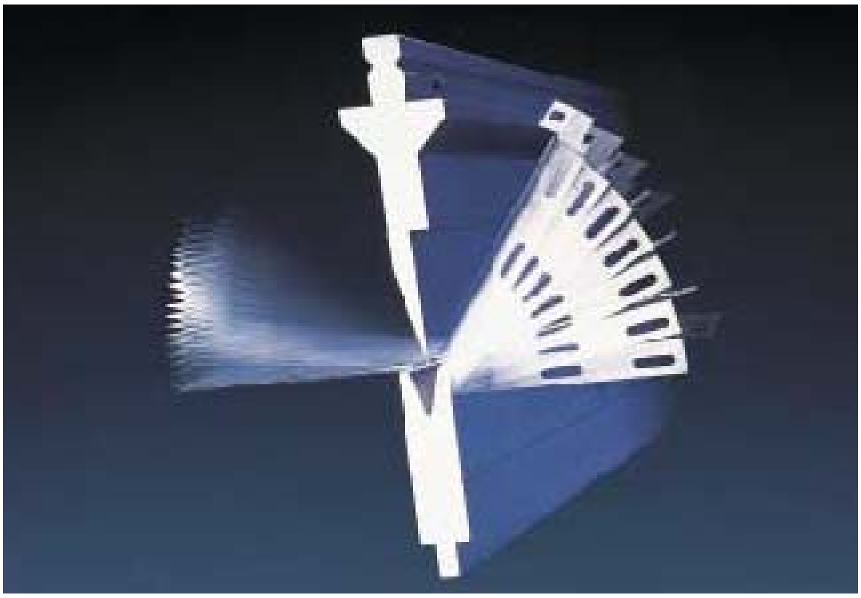
An art is hidden behind these rather everyday objects which textbooks simply refer to as "bending by forming", or as it is known in practice, bending or press brake bending. Although industrial sheet metal processing and the Japanese art of folding apply to different materials, they nonetheless share an overriding feature: the starting material for press brake bending and Origami is a flat section which is then formed into a three-dimensional shape.

Press brake bending: One principle – three processes

Press brake bending is a process in which a metal workpiece is bent along a straight line. A workpiece – usually a tailored sheet with a maximum thickness of 25 mm – is pressed into a die by a punch. At first sight, this appears to require little technical know-how in comparison to other processes. But, if you take a closer look, you will see that a lot of knowledge and experience went into making a precisely bent edge. The three most important procedures involved in the processing of sheet sections on a bending press are:

- free bending
- coining
- three-point bending

Press brake bending: A flexible way to fold sheet metal



Free bending: press brake bending with little force

Free bending or air bending is a process in which a punch bends a workpiece without pressing it to the bottom of the die. This process only requires a comparatively low press tonnage. Consequently, economically priced machines can be used.

It is this economical aspect that has made free bending the most widely used bending process. One disadvantage of this process is, however, that the finished parts produced can only be partially reproduced. Different angles for identical parts are caused by fluctuations in the material structure. This, in turn, leads to different resiliency properties.

Not all batches of a material possess exactly the same characteristics. The material constants such as material thickness and tensile strength which affect resiliency vary from one batch to the next. These can produce considerable angle errors even within the same batch. A difference in sheet thickness of 1/100 mm causes an angle difference between 7′ and 12′ when freely bent.

Resiliency

Resiliency is the tendency of the sheet to bend back into its original shape once pressure has been removed from the punch. This is caused by the fact that bending compacts the inner side of the sheet whereas the outer side is stretched. Freely bent angles are particularly resilient. For this reason, the degree of resiliency must be given due consideration when calculating the bending angle. In practical terms, this means that the sheet must be bent at an angle smaller than the angle desired. The more the sheet is bent, the less resiliency will be encountered. Thicker sheets are less resilient than thinner ones; stronger materials less than weaker materials. If one were to attempt to produce the same product from different materials without regard to resiliency, this would lead to angle errors of several degrees.

Coining: the variation with the highest precision

Coining differs from free bending in that the workpiece is pressed completely into the die by the punch until it comes to a stop. The bending angle can therefore not be chosen independent of the tool; in fact, the dimensions of the punch and die actually determine the angle. As a key fits into a lock, so does a punch into a die. Each angle requires its own tool set, which results in longer non-productive times for the machine, because the tools have to be exchanged more frequently than with free bending. The smaller the lot size, the more critical this problem becomes. In addition, coining requires more machine power. Press tonnage for coining is approx. three times greater than for free bending. As a result, the maximum sheet thickness which can be processed is considerably higher for free bending than for coining.

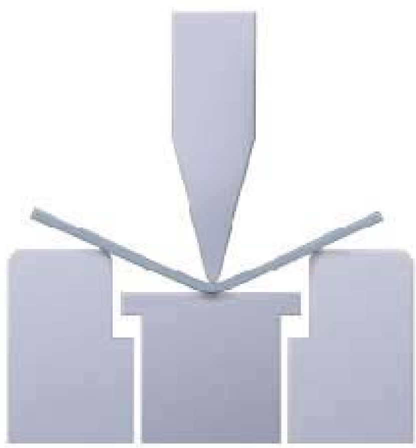
High precision is the feature which sets coining apart from free bending. There are very little angle fluctuations with coining due to the fact that pressure at the end of the punching stroke is high enough to minimize the effects of resiliency. In this way, fluctuations in the material structure are negated for the most part resulting in a maximum angle deviation of 15′ from the programmed angle. With free bending, deviations of up to 30′ are possible.

Three-point bending

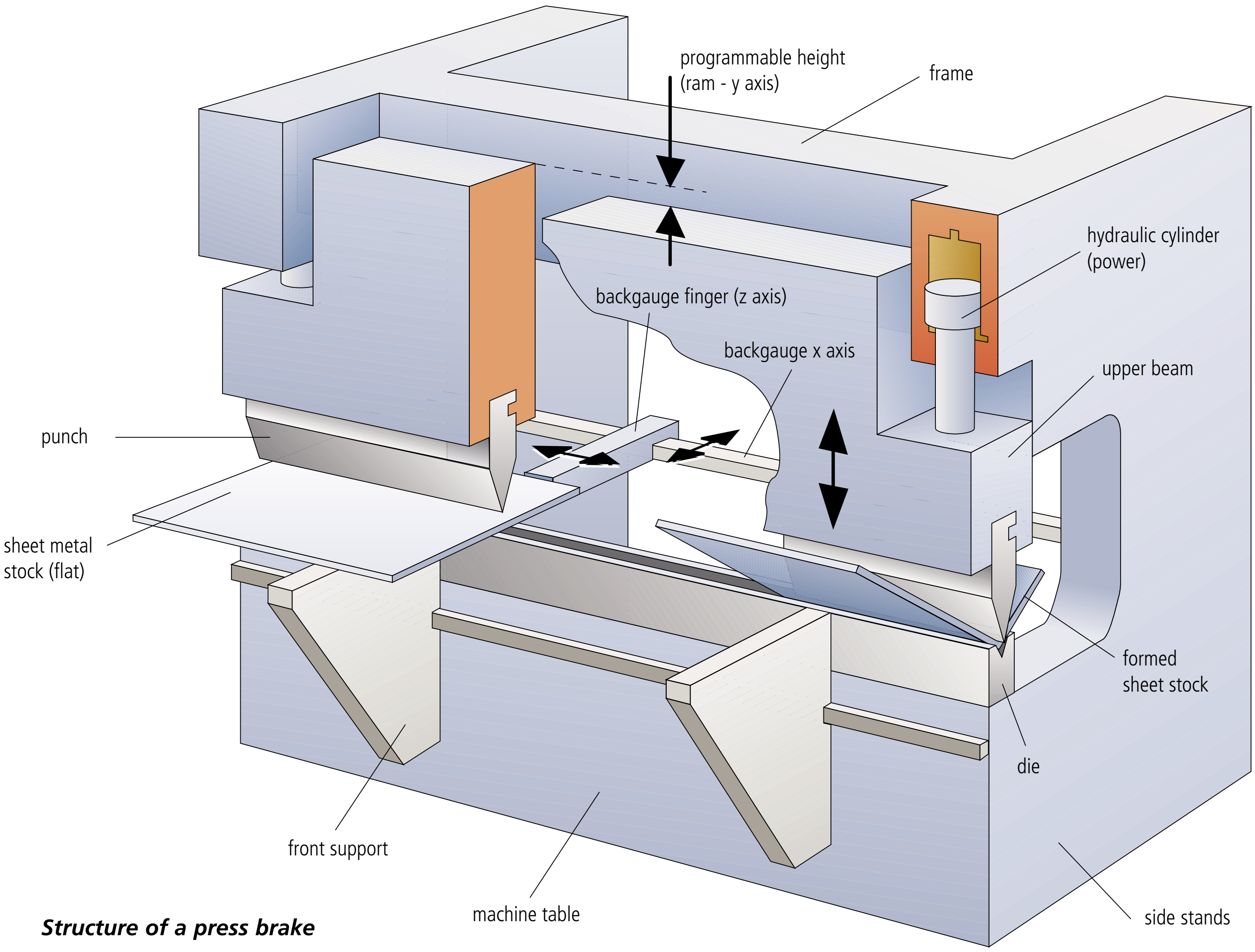
Less press tonnage and more flexible tools than coining on the one side and nearly the same angle precision on the other side: these are the arguments for three-point bending. These advantages are tempered by the considerable costs required for a complex control and tool technology. Three-point bending differentiates itself from other bending processes in particular by employing a special die base whose height can be adjusted by a servomotor. The bent sheet makes contact with both the outer edges of the die and the base between the edges – viewed laterally, there is

Free bending

Coining



Three-point bending



Structure of a press brake

contact at three points. Since the height of the base can be adjusted to an accuracy of 1/100 mm, the bending angle can be set by adjusting this height. The lower the die bottom, the smaller the bending angle.

A spring system in the die base makes it possible to bend precisely with a relatively low amount of press tonnage. As soon as the sheet is pressed into the die, the base begins to submerge into a hydropillow. If the pressure on the pillow exceeds a specified value, the control moves the punch back slightly towards the top. This method affords high reproductibilty of the bending process, because fluctuations in the material thickness are compensated for. Moreover, neither the system nor individual tools can be overloaded.

Whether three-point bending should be considered a process of its own or simply a bending tool with which the bending angle can be corrected, is a matter of personal preference. An exact bending angle cannot only be achieved by a controlled three-point bending process, but can be an automated bending angle measurement.

Machine and work processes

Bottom bending machines are press brakes which in a vertical motion of the beam presses the punch into the lower tool of a die. Normally, the machine frame of this kind of bending press is made of steel. The machine illustrated at the top has a frame consisting of two C shaped side stands which are connected at the top by a crossbeam and at the bottom by a table. The lower tool adapter is located on the table, and the punch holder is mounted on the beam. The beam is powered by two hydraulic cylinders both of which are integrated in the upper sections of the two stands. The hydraulic pump and the compensator reservoir are located at the bottom of the crossbeam, invisible from the front. In order for the workpiece to be bent along the entire bending edge at the same angle, both hydraulic cylinders must be synchronized exactly. The control is finally mounted to a swivel arm. In this way the operator always has the screen and the control panel within reach.

The bending length of most of the serial production machines ranges from 1 to 4 m. The bending speed is between approx. 1 and 15 mm/s. In comparison to the times required for placing, positioning and removing the workpiece, the time needed by the

Press brake

machine to bend the material is insignificant. That means that on the one hand positioning aids, tool handling and ergonomic aspects – in short: the user-friendliness – play a big role. On the other hand, the extent to which the production time for a product depends on the level of training and motivation of the personnel becomes apparent.

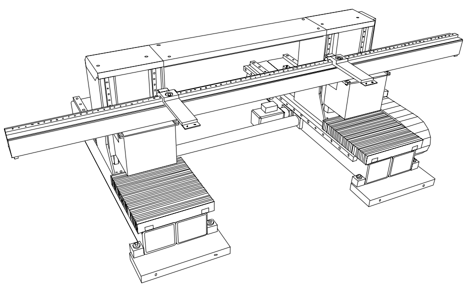
Positioning workpieces

During positioning, the workpiece rests on supports which can be moved laterally making it easier for the operator to place heavy and protruding pieces horizontally onto the machine. The pieces are aligned with the help of a backgauge. The workpiece is pushed far enough into the machine until it makes contact with the back stop. Depending on the precision, price, and equipment, the back stop can be positioned manually or controlled by a program. At any rate, exact positioning requires the workpiece not only be aligned on one side. For this reason, press brakes are usually equipped with two finger stops. Another requirement allowing exact positioning is a section which has been precisely processed. If, for example, the edge facing the stop is not completely flat, or if the edge was cut at an angle, this will lead to deviations in the programmed bending line. For most applications, such deviations can be tolerated up to 0.2 mm. These inaccuracies can, however, add up once a workpiece has to be aligned at edges or corners which have already been processed.

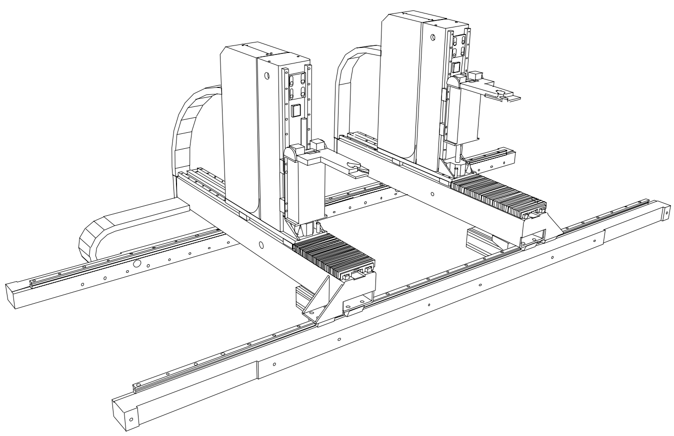
If the finger stops can only be moved together – excluding lateral movements –, the workpiece can only be aligned on a single surface or edge which runs parallel to the bending edge. If, on the other hand, the finger stops can be moved independently of each other in any direction, it is possible to align the workpiece on various surfaces or edges. In this case, edges can be aligned which are not parallel to the bending edge. This method of alignment, however, requires six independently programmable NC axes.

The maximum degree to which the workpiece can be inserted into the bending press depends on the edge clearance of the machine and the bending radius. "Edge clearance" in this context refers to the space behind the tools in which the workpiece can move during processing without colliding with a machine component. The range of workpieces that can be processed is limited by the edge clearance of the press brake.

Backgauge systems



Movable finger stops only laterally independent of each other

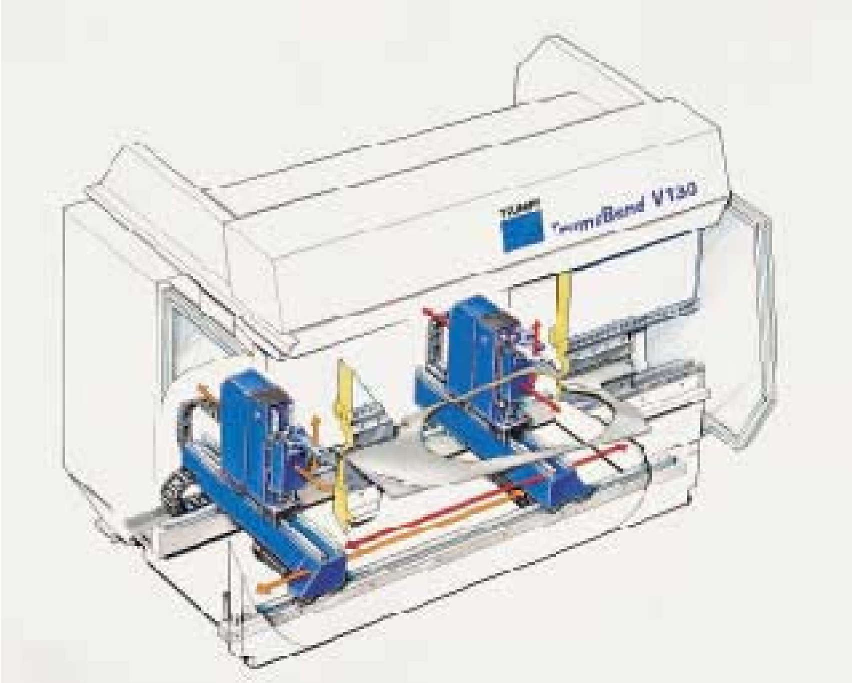


Movable finger stops totally independent of each other

Edge clearance of press brake

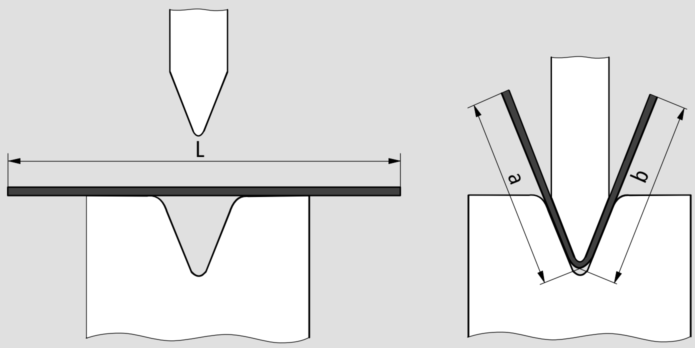
Bottom bending machines do not only have a stripper that holds the workpiece after the operator has positioned it. To prevent slipping, the workpiece bent upwards during the bending process must be guided. The operator can perform this manually with light, easy-to-handle sheets. Heavy workpieces that protrude quite far require either a second operator or a bending aid. The bending aid, a pivotable plate, is mounted at the front of the table. While the workpiece is being bent upwards, the leg piece protruding from the machine is supported by the plate. If the bending aid is no longer required, it can be moved away to the side like a support.

If a bending aid is not available, the workpiece is guided manually during processing.



A workpiece can be aligned on two different edges in a backgauge system with six NC axes which are totally independent of each other.

Shortened leg pieces



L = a – Δ x + b, whereby L = elongated length and Δ x = shortening factor

The material distribution of a workpiece is altered when it is plastically reshaped. A designer must take this effect into consideration when he calculates the developed view of a product. In the drawing the legs measured on the outside of the bent workpiece are longer than in the developed view. To calculate the elongated lengths of a bent workpiece in the drawing, a compensating factor, or shortening factor, must be subtracted from the sum of the leg lengths. The shortening factor can be calculated using standardized formulas. These formulas can, however, only provide approximate values, because even though they take factors such as type of material, sheet thickness, the inner radius and the bending angle into account, they do not do so for the tools, the process (free bending, coining, three-point bending) or the machine. Angles smaller than 65° can even have a negative compensating factor, i.e. in reality an elongating factor. Users requiring exact edges have no other choice but to come up with their own compensating factors by conducting several tests for each machine and recording the results in tables or diagrams. This approach assumes that there is an active flow of information between the participating parties. The designer must be informed of the empirically-calculated compensating factors, and the personnel at the machine must know the tools and the processes that the designer has planned for each processing step.

Press tonnage

The maximum press tonnage of a production machine lies between approx. 500 and 8000 kN. In order to free press a 2 mm thick sheet with a tensile strength of 400 N/mm^2 and bending radius of 2.6 mm into a 16 mm wide lower tool, a press tonnage of approx. 120 kN/m is required.

The press tonnage required for bending a sheet is taken from tables, which indicate the pressure per meter. It depends on various factors: first of all, the thickness and the tensile strength of the material must be taken into account as well as the inner radius of the fold. Normally, the programmer has no control over these factors. What he can do, however, is change the required press tonnage by selecting the process and tools. The lower tool width in particular which is inversely proportional to the press tonnage allows you to vary the press tonnage easily.

You cannot, however, decrease the press tonnage to any amount by selecting wider lower tools. This is due to the fact that the minimum length for the legs increases when the lower tool width increases and hence only folds with long lengths can be carried out. The minimum leg length is the length of the shortest leg possible. If a leg falls below the minimum leg length, it will be pulled into the die during processing. Irregularities in the angles are the result.

Programming

Press brakes can be programmed at the machine control. Program systems are also available for press brakes which make it possible to create NC programs remotely, as with almost all numerically-controlled machines. The market offers a wide range of software solutions – from simple 2D programs to entire software packages with 3D simulations in color. Powerful controls and programming systems are able to calculate the ideal sequence of the individual bending steps from 2D drawings of the bending profile or from 3D drawings of the product. In addition, a collision view is provided. In other words, each processing phase is tested to determine if the workpiece will collide with the tool or with the machine. When all calculations have been performed, the system generates the bending plan and the NC program which can be used to shape the section into the desired form automatically.

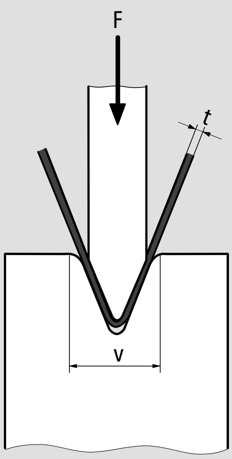
The bending plan is one of the most important sources of information for the machine operator. It provides, for example, the sequence of the bending operations, the assignment of tools to a bending edge or the bending direction. The bending plan is also used as a setup plan. That means, it provides the operator with information as to which tools have to be inserted into which particular tool positions at what particular time.

During processing, the operator is led through the control by a graphical user interface. The operator sees the current form of the workpiece on the screen at every bending step and the position which the workpiece must be placed in for the next processing step. Before inserting the first workpiece, the operator must always carry out a test run. For this purpose, he moves the punch and die away from each other to the side, so that they will not make contact with each other during a work stroke. He then starts the NC program and checks if he has set up the proper tools or if a collision is possible. Once he has excluded the possibilty of a collision, he then moves all the tools to their proper location, inserts the first workpiece and restarts the NC program. If he does not have an automatic measuring system at his disposal, he must interrupt processing after every fold and check the angle and dimensions. If he detects deviations, the control provides him with the possibility of entering corrected values and repeating the fold.

Graphic user interface of a control

Determining the press tonnage

Machine-specific tables are used to determine press tonnage. To take full advantage of all possibilties which go beyond simply reading the values, the programmer must be familiar with the following relationships:



$F = k \times 1/V$
Doubling the lower tool width (V) reduces the required press tonnage by 1/2.

$F = k \times t^2$
Doubling the thickness (t) increases the press tonnage by a multiple of four.

$F = k \times l$
The bending length (l) is proportional to the press tonnage.

$F = k \times \sigma b$
The tensile strength (σb) of the material is proportional to the press tonnage.

Tools

A tool set for a press brake consists of a punch and a die, known as the lower tool. The hardness of a tool depends on the steel used and the way the tools were heat-treated. Work zones, i.e. areas in which the tool makes contact with the workpiece, are normally hardened. Laser-hardened work zones made of chromium molybdenum steel attain a hardness of up to 60 HRC. As they are subject to heavy loads, tools are frequently a source of angle deviations and drag marks on the workpiece. If the punch and dies are not handled carefully, or cleaned and checked regularly, contamination or wear can permanently impair the bending precision.

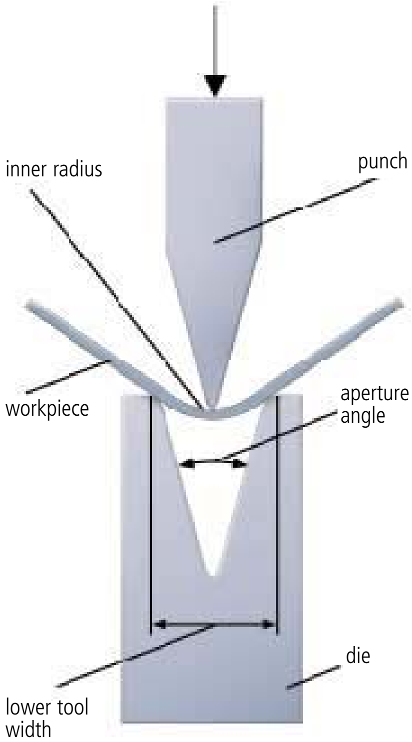
Segmented tools

The demands placed on the flexibility of a press brake today, call for the use of segmented tools. Segmented tools attain the required processing length by allowing individual tools to be set up next to each creating a continuous tool. Due to the relatively low weight, a single person can handle segmented tools by himself without any special aids. Some manufacturers even offer segmented punches which center themselves and can be unlocked by pushing a button. These tools do not have to be pushed out from the sides. The operator can simply remove them from the front. Flexibility in handling and reducing setup times are not the only advantages of segmented tools. Since segmented tools can be produced which are nearly perfectly parallel, they bend with a higher precision than single tools.

Furthermore, single tools limit the range of pieces which can be processed. Bending lines which extend into an already bent edge are, just like a formed area within a workpiece, only possible with tools which have the same or a shorter length as the bending line. To process such pieces, the operator sets up the machine with several tool stations, i.e. with several segmented tools. Since the resetting time for a press brake lies between 2 and 5 minutes depending on the number of tool stations, the programmer attempts to create the NC program so that the machine can be set up with all tool stations before processing begins.

Punch and die

Every bending tool has a form which remains constant over its entire length. The central distinguishing feature of a tool is its cross section. The most important characteristics of a punch cross section are, for one, the overhang, i.e. the form with which the punch deviates to the front or to the rear from the vertical line and, secondly, the angle at the tip. Free bending allows you to bend different angles with a single punch. For example, perpendicular deformations are possible using punches with angles between 30° and 88°. The form of an overhang required by a punch is determined by the form of



The tool set of a press brake

A press brake being set up with segmented, self-centering punches.

Press brake bending with several tool stations

U-shaped fold

the workpiece. In order to bend a workpiece into a U form in a V-shaped die, a punch overhang is required whose shape depends on the flange length and the width of the opening.

The handling of the workpiece is also influenced by the tool selected. The tool clearance – the largest possible distance between the punch point and the top of the die – should be large enough to allow the workpiece to be inserted into the machine and removed at the front of the machine at any time during processing. The height of the die and punch are to be selected accordingly. Due to the restrictions imposed by their shape, some workpieces have to be pulled out of the machine laterally. The operator will want to avoid this method if possible, because not only is it rather awkward and time-consuming, but it also reduces the productive time at the machine. Sometimes altering the bending sequence or using another punch with a different cross section is sufficient to allow the workpiece to be removed from the front side of the machine.

Frequently, edges can only be processed with tools which do not exceed the length of the bending line.

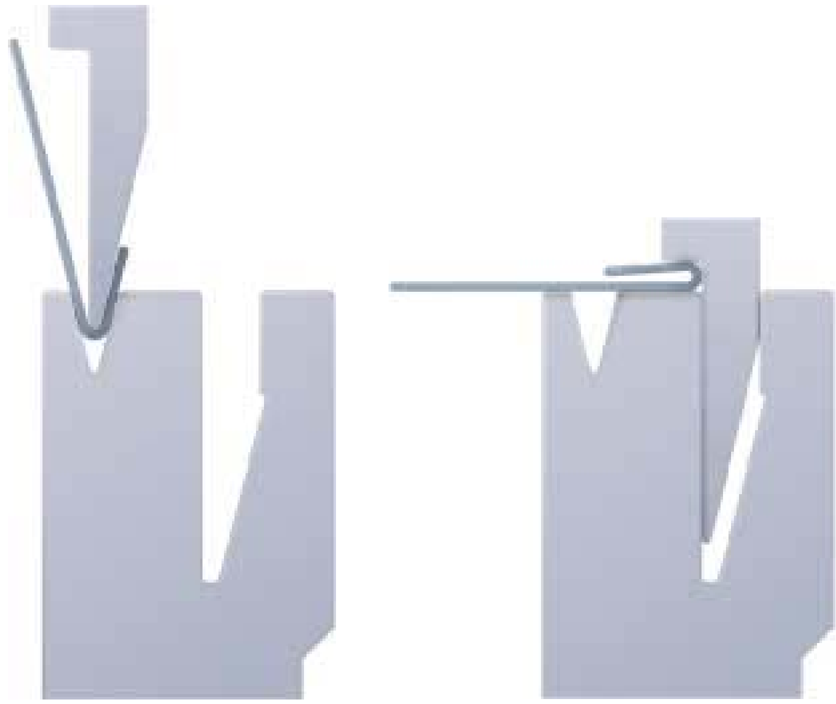
Special tools

Standard tools allow you to handle a wide spectrum of pieces. In addition to these, most manufacturers also offer special tools which are constructed according to customer specifications. Special tools provide the operator with a field of applications which would otherwise not be possible or would require an excessive amount of time. A simple example is folding sheets which require two work steps on a press brake: folding and pressing.

Despite their many advantages, careful consideration should be given to the use of special tools, since they are more expensive and their acquisition more difficult.

Automatic angle correction

The demands placed on bending precision has increased continuously in recent years. This is due to the geometric forms of the workpieces becoming increasingly complex. Here, inaccuracies can add up very quickly during processing. As discussed earlier, particularly fluctuations in the material structure only allow approximate bending of the angle. If greater accuracy is required, the angle must be measured and corrected during processing. The operator can choose between various automatic systems, for this purpose, ranging from systems with optical laser analysis to systems with contact sensors located on the sides of the sheet.



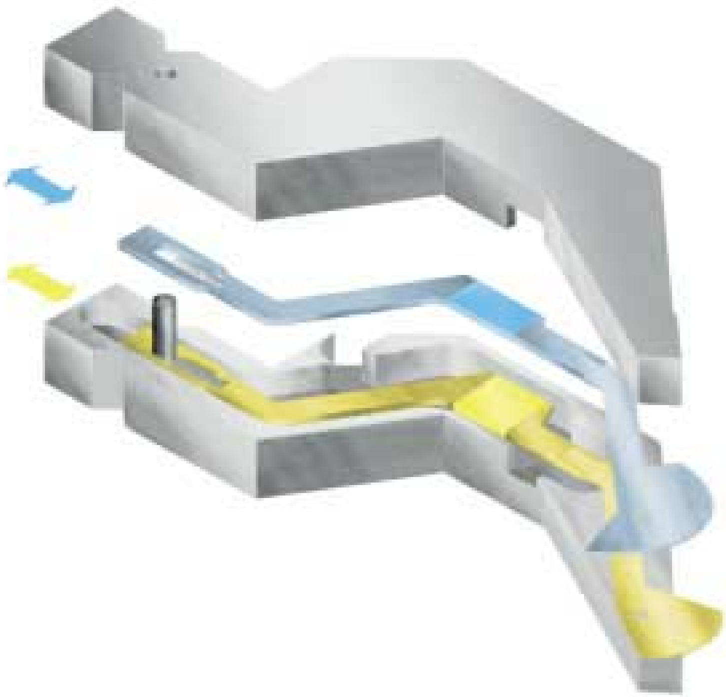
A double die and a special punch solve the problem of folding and pressing without having to change tools.

This punch with overhang can be unlocked by pressing a button.

Minimum bending radius

A radius smaller than the minimum bendng radius causes microcracking or creasing:

Every designer knows that a sheet section cannot be bent with any arbitrarily small inner radius, because a radius which is too small leads to microcracking on the outer side or creasing on the inner side. The smallest possible radius is determined by the material. Generally, materials which can be easily formed can be bent at a smaller radius than brittle materials such as mild steel or magnesium alloys. As a rule of thumb, you can assume that the minimum bending radius must be larger than the sheet thickness. The smallest inner radius can be achieved when the bending edge is at a right angle and preferably vertical to the rolling direction and when the bending step is carried out slowly.



Angle sensor assembly integrated into the punch

Integrating a contact sensor assembly into the punch is a particularly sophisticated solution. Two suspended discs of various size project over the punch tip. After bending, the punch moves to the top with an accuracy of 1/100 mm until the workpiece springs back and both discs lie only on the inner sides of the bent angle. The control calculates the size of the angle from the difference in height between both discs, corrects the press tonnage and presses the punch again into the bending edge.

This process reduces the material consumption. In addition, it considerably reduces the times needed for setting up the NC program, because manual measurement is not required and the NC program is automatically adapted. Particularly against the backdrop of lot sizes becoming smaller, reducing setup times is becoming increasingly important.

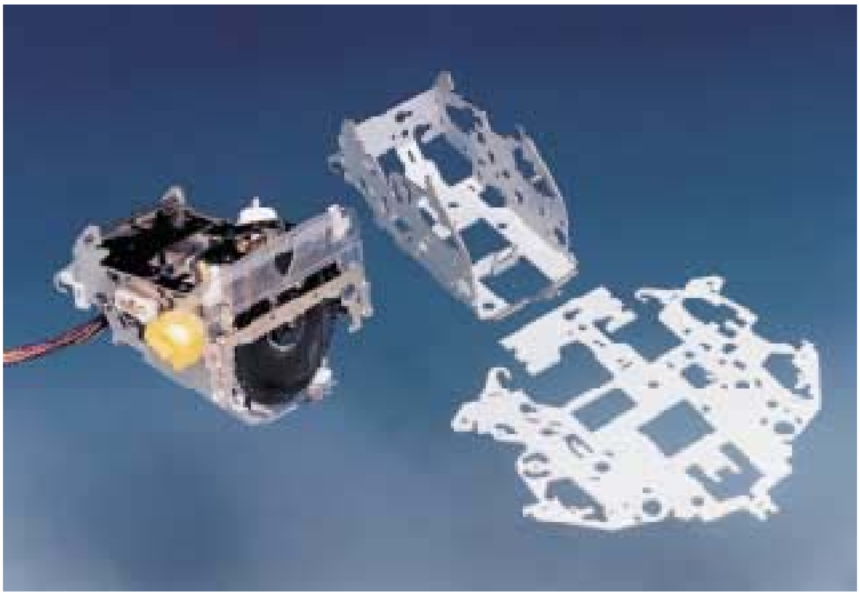
Carriers for loading and unloading a punching machine

Trends

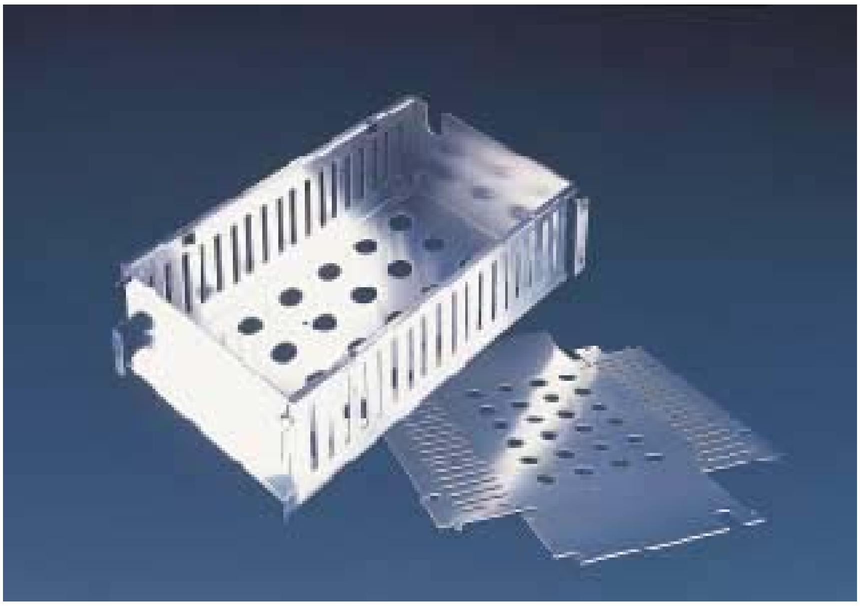
Technological standards and piece complexity are both increasing in flexible sheet metal processing. Driven by the ideal of a "near net shape", or in other words, removing a piece from the press brake whose form is as near as possible to the finished product, we are witnessing two developments which are taking place concurrently. First of all, attempts are underway to create as many small formed areas as possible during flat processing – for example, with a punching machine. Secondly, new tools provide designers with the possibility of creating products in which ideally all welding joints can be replaced by bending processes. As welding joints become increasingly superfluous, production time will decrease.

A further innovative advancement can be expected from the introduction of open control systems and new program techniques which support the process chain from the initial CAD design to the final production step by means of a universal concept. In this context the question as to the role of automation in the future is frequently asked. As far as bending is concerned, the same principle holds true as for all flexible sheet metal processes. Now, after the initial CIM euphoria has subsided, and empty factories have shown themselves to be inflexible special solutions due to the slave-like chaining of the individual components, manufacturing cells are becoming the solution of choice. In these manufacturing cells, a technician no longer performs various, independent tasks. He instead is responsible for organizing a single task in its entirety. A robot system has the task of assisting the technician and relieving him of tool handling tasks as much as possible.

At the core of this kind of system on a press brake is a gripper which is able to grasp the workpiece, place it, position it, and rotate it in any direction. The gripper's motions are monitored by optical and contact sensors. The requirements for such a system are, on the one hand, support from robot control and sensor assembly and, secondly, the ability to link it with a production and planning system, i.e. integrating the automated manufacturing cell into the sheet metal processing chain. The use of robots for press brakes has been hindered by the high investment costs and the inability of these robots to guarantee the high accuracy tolerances required. Nevertheless, automation concepts in bending will prevail as they have in flat processing because they promise an increase in productivity to an as yet unattained high level.



Printing unit chassis of a typewriter



Cabinet for electrolytic capacitors