SUDAN UNVERSITY OF SCIENCE AND TECHNOLOGY

COLLEGE OF GRADUATE STUDIES

CAD, MODELING AND SIMULATION OF A PROGREESIVE DIE

(Case study)

تصميم ونمذجة ومحاكاة قالب مجمع باستخدام الحاسوب (حالة دراسية)

A thesis Submitted to the College of Graduate Studies in Partial Fulfillment of the Requirements for the Degree of M.Sc. in Mechanical Engineering (Production)

By: BADY IBRAHIM ALI TAHA

Supervisor: Dr. ALKHAWAD ALI ELFAKI

AUGUST 2017

الآية

بِسْمِ اللَّهِ الرَّحْمَنِ الرَّحِيمِ

﴿ قُلْ هُوَ اللَّهُ أَحَدٌ (1) اللَّهُ الصَّمَدُ (2) لَمْ يَلِدْ وَلَمْ
يُولَدْ (3) وَلَمْ يَكُنْ لَهُ كُفُوًا أَحَدٌ (4) ﴾

صدق الله العظيم

سورة الاخلاص

Dedication

To my Parent with much love

To my Brother and Sister

For their inspiration and encouragement with much gratitude an affections that I can't well put down here.

ACKNOWLEDGEMENTS

I would like to thank my Supervisor for his professional advice and assistance. Dr. ALKHAWAD ALI ELFAKI served as a mentor and directed me along the traveled path and he surrendered many valuable days revising my work.

My friends ALFADIL OMER and ABD ALLAH KAMAL .they always seemed to know when to encourage me to work on this research or do something interesting. Full thanks to my wife AFRAH OMER best flower in my life.

MY brothers and sisters were great friends, and for all in Al Madinah Al Munawarah Factory for Electric Lamps.

ABSTRACT

This research presented a new method for the mass production of a high quality, low cost, sheet metal products using a die, press and transferring mechanism. The research presented a definition of the tool (die), its design parts and the types which differ as per the products and operations concerned. The researcher selected the progressive die used for mass production, for more elaborated studies because it combines a number of die design and function in one frame. Al Madinah Al Munawarah Factory for Electric Lamps products formally produced by three individual stages. Anew one progressive die was designed for this product, using the well-known Solid work software dedicated for sheet metal products die and mould design.

المستخلص

قدم هذا البحث طريقة جديدة للانتاج الكمي لمنتجات الصفائح المعدنية ذات الجودة العالية والاقل تكلفة باستخدام القالب والمكبس كآلية تحريكية للقالب. قدم البحث تعريف لهذه الاداة (القالب) وكيفية تصميمه واجزاءه ووظائفه وانواعه التي تختلف باختلاف المنتجات والعمليات، ثم اختار الباحث نوع واحد من هذه الانواع وهو قالب الانتاج الكمي التقدمي المستمر (القالب المجمع) لدراستة بتوسع اكثر لانه يضم مجموعة من القوالب في طرق تصميمها ووظائفها في اطار واحد وتعرض البحث لامكانية الاستفادة القصوى من المادة الخام. أختير منتج من مصنع المدينة المنورة للمصابيح الكهربائية كان ينتج بواسطة ثلاث مراحل مفردة وصمم قالب واحد (القالب المجمع) لانتاج المنتج وتم ذلك باستخدام البرنامج الحاسوبى (Solid works) لتصميم القالب.

Table of Contents

List of Figures

Figure Name **Page No.**

List of Tables

Table Name **Page No.**

CHAPTER ONE

INTRODUCTION

1-1 General Introduction:

Sheet metal is metal formed by an industrial process into thin, flat pieces. It is one of the fundamental forms used in metalworking and it can be cut and bent into a variety of shapes. Countless everyday objects are constructed with sheet metal. Thicknesses can vary significantly; extremely thin thicknesses are considered foil or leaf, and pieces thicker than 6 mm (0.25 in) are considered plate. Sheet metal is available in flat pieces or coiled strips. The coils are formed by running a continuous sheet of metal through a roll slitter. The thickness of sheet metal is commonly specified by a traditional, non-linear measure known as its gauge. The larger the gauge number, the thinner the metal. Commonly used steel sheet metal ranges from 30 gauge to about 8 gauge. Gauge differs between ferrous (iron based) metals and nonferrous metals such as aluminum or copper; copper thickness, for example is measured in grams (and represents the thickness of 1 ounce of copper rolled out to an area of 1 square foot). There are many different metals that can be made into sheet metal, such as aluminum, brass, copper, steel, tin, nickel and titanium. For decorative uses, important sheet metals include silver, gold, and platinum (platinum sheet metal is also utilized as a catalyst.) Sheet metal is used for car bodies, airplane wings, medical tables, roofs for buildings (architecture) and many other applications. Sheet metal of iron and other materials with high magnetic permeability, also known as laminated steel cores, has applications in transformers and electric machines. Historically, an important use of sheet metal was in plate armor worn by cavalry, and sheet metal continues to have many decorative uses, including in horse tack.

Modern continuous rolling mills produce large quantities of thin sheet metal at low cost. A substantial fraction of all metals are produce as thin hot-rolled strip or cold-rolled sheet, This is then formed in secondary processes into automobiles, domestic appliances, building products, aircraft, food and drink cans and a host of other familiar products. Sheet metal parts have advantage that the material has a high elastic modulus and high yield strength so that the parts produced can be stiff and have a good strength –to-weight ratio. A large number of techniques are used to make sheet metal parts.

1-2 Problem statement:

The research problem involved the following:

a. **Complexity of part, produced using sheet metal working (with ref. to case study):**

Of the most important stages of the bulbs industry is the stage of the metal base industry, the complexity where it is in three basic stages and each stage needs special machine, the three-stages are bending, punching and shearing. Every stage need to focus and accuracy.

b. **Loss of time when using multiple stages for sheet metal working:**

when we use more stage or more machine we lost time .each process of items are to be produced on two machines (or stages) that is loss of time. Each processes can handle only one machine at a time and each item must be processed through one machine. The setup time plus work time for each item for each machine must be minimized. A simple decision rule is obtained in this research for the optimal scheduling of the production so that the total elapsed time is a minimum by using one machine and one stage.

c. **Accuracy lack of parts, made using multi stages sheet metal working:**

A product is produced in several stages, or is used in the production number of machines, it leads to a lack of accuracy as a result of the large number of processes and frequent effort of machines and workers so when you use a single-stage process in one single machine it leads to increase accuracy and reduce the effort and reduce time production, leading to reduce the production costs.

1-3 Problem importance

Using traditional proceeded for achieving part with a shape of U channel involve multi stage to accomplish each process separate, this traditional method when considering mass production become ineffective since increase the cost of production, because many stage of production increase the setup time required for preparation for the part of the next operation in addition to uncontrolled variable, that affect the quality of the final product. Using progress die mold for achieving same shape can eliminate many of the extra cost by reducing the multi-processing into single processing that can do pending and punching which can essential be reliable in resolving such problem.

1-4 Research objectives

The objectives of this work are:

a. To design a progressive die for (case study) mass production.
b. To model and simulate the production line of (case study) mass production.

1-5 Research methodology

This research depends on many theories of die design and sheet metal forming. Most of die theories are based on recognized experiments and testing on workshops, while other theories were depend on the sheet metal properties.

SOLID WORK is advanced computer software for sheet metal dies design was used to design the new tool. Metal part of fluorescent lamps(T-shape)is selected as case study.

1-6 Research layout:

In this section a brief description of the contents of the research chapters are presented as following:

Chapter one: give a generic description of sheet metals and states the problem of the research, objective, methodology and general search layouts. Chapter two: Cover the literature review of the sheet metal process, stamping, types of dies, and, previous studies. Chapter three: define the progressive die and discuss Steps of Progressives dies design and analysis. Chapter four: this chapter gives details of modeling the Progressive Tool and the software used. Chapter five: a conclusion presented in addition to recommendations.

CHAPTER TWO

Theoretical background and previous studies

2-1 Sheet metal forming process

Sheet metal forming is one of the most important manufacturing processes. This is particularly valid for the automotive industry where sheet metal forming has an even more important key position. The automotive industry is the leading sector in many countries and the main driving force behind the sheet-metal forming developments as well.

The competition in car manufacturing is extremely strong leading to larger model variety and shorter model cycles. The increased competition also leads to a very intense development activity to increase productivity and to reduce costs [1].

Application of light-weight design principles is one of the most important trends to meet the above-mentioned requirements. Obviously, the new design concepts require new materials. The new materials often require new, innovative forming processes and new tooling concepts, as well.

The increased competition also requires shortening the lead times from the concept to final realization: to reduce the lead times, the application of various methods of Computer Aided Engineering (CAD/CAM/CAE and FEM techniques) is inevitable [2].

2-2 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 figure below.

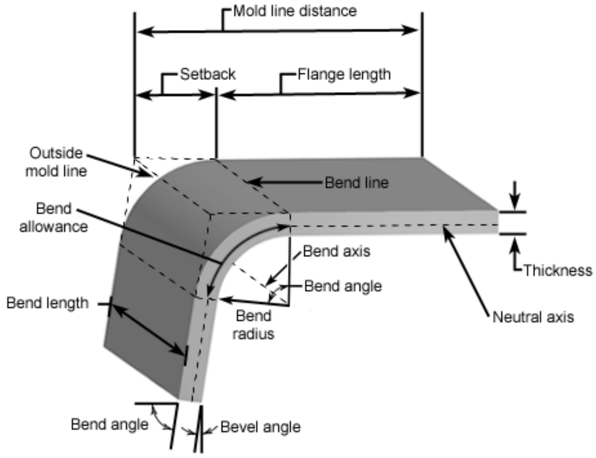


Fig. (2. 1) the bending generation parameters

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 [2]. 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 in figure [2-2] .

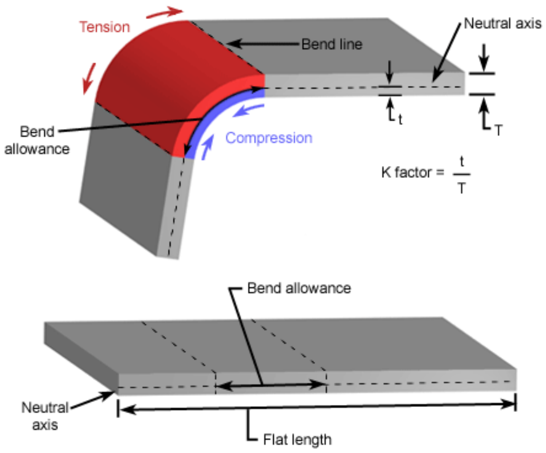


Fig. (2. 2) parameters of bend surfaces changes

a- 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 spring back 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. The amount of springback depends upon several factors, including the material, bending operation, and the initial bend angle and bend radius which are defined in figure (2-3).

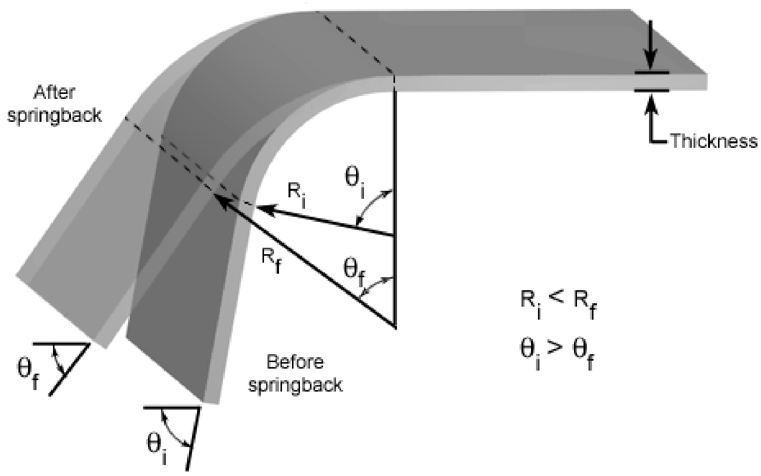


Fig. (2. 3) bend operation angles

Where:

θ_f final bend angle

θ_i initial bend angle

R_i initial radius

R_f final radius

b- Spring back

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 [2].

2-3 Roll forming

Roll forming 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 [2].

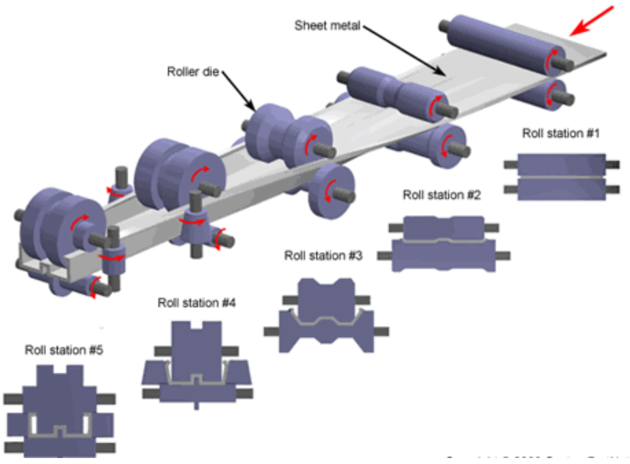


Fig. (2. 4) process on a roll forming line

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 Heat Ventilation and Air Condition applications (HVAC).

2.4 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 show figure (2-5).

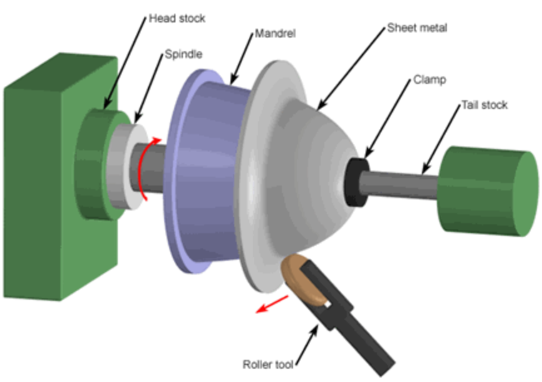


Fig. (2. 5) Spin forming

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.

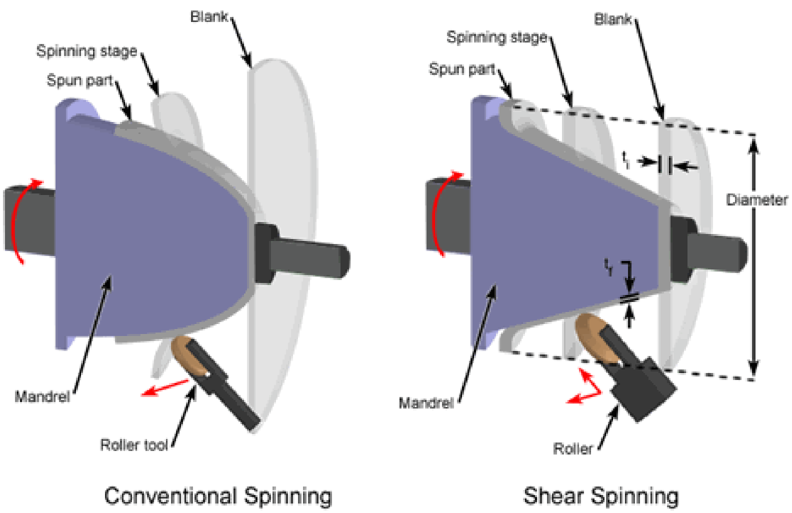


Fig. (2. 6) Conventional Spinning vs. Shear Spinning

2.5 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 [2].

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, show Fig. (2.7). 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.

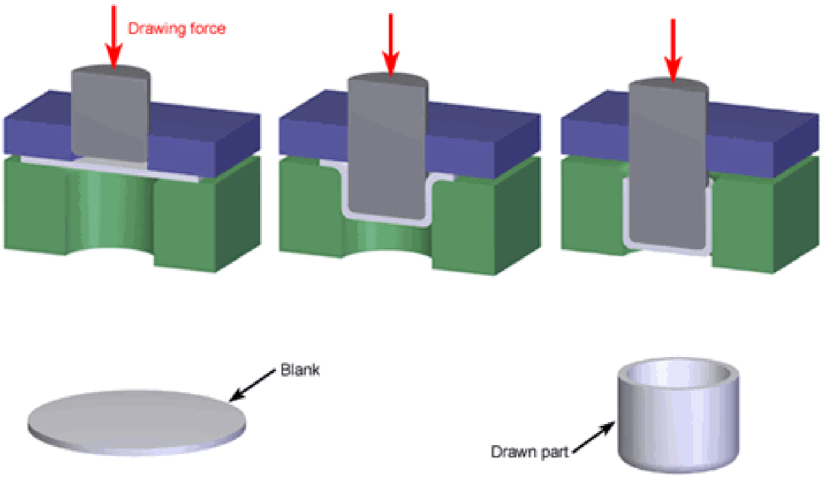


Fig. (2. 7) Deep Drawing Sequence

2.6 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.

Show Fig. (2.8) 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.

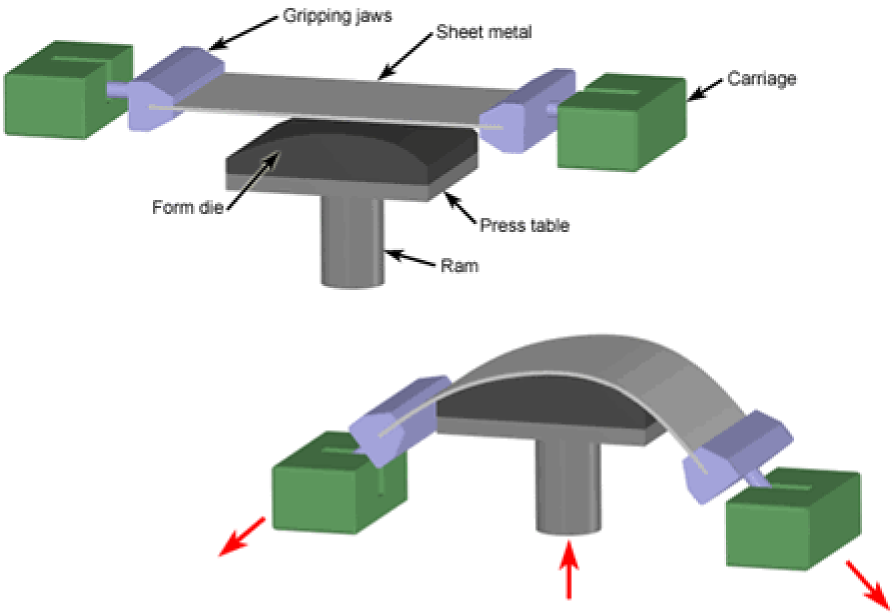


Fig. (2. 8) Stretch forming process

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.

2.7 Sheet Metal Stamping:

Stamping is the all operations done on sheet metal like pierceing (make holes with circular shape or other shapes in sheet metal) or blanking (same to piercing process but the cutting area is the product) or bending or embossing or drawing or extruding or curling to produce specific product. [1]

In other words stamping is the process of getting a product from sheet metals (raw material) by using tool (die).

2.7.1 Sheet Metal Stamping In Comparison With Other Metal Fabrication Processes:

In today's cost-conscious world, sheet-metal parts have already replaced many expensive cast, forged, and machined products. The reason is the relative cheapness of stamped products, mass production and greater control of their technical and esthetic parameters, for these reasons the world slowly turned away from heavy and complicated shapes, and replaced them with functional simple and logical forms, by the well design of sheet metal product can eliminate the need for rivet or other fastening process like welding also stamping can use to improve existing designs that are costly and labors intensive and improve function ability that cause decreasing in production cost.[1]

The tool used to get the products from sheet metal it is a die which it an ideal tool that can produce very much quantities of parts in short time with good quality. The dies have many shapes and construction up to the type of stamping processes, if the part is auto part the die has different shape and construction to electrical part.

No stamping design work can be considered optimum until in the judgment of the press working department or customs, it holds out the strong probability of achieving the following [2]:

1. A die or set of dies that combines maximum production and least maintenance with lowest feasible life cost.
2. Maximum utilization of the least expensive stamping material that will serve satisfactorily.
3. Most efficient press working practices.
4. Stamped product that consistently meets sales and service requirement of shape, dimensions, strength, finish, style and utility.

When evaluations a part for sheet metal production (stamping) the most restrictive aspect to be considered is the cost of the tooling, to build metal stamping tool (die) costly process involving many people, many modern machines and advance technology , for these reason the demand for this tool must first be economically justified.

The designer and presswork man must make good allocation for the resources between the times of delivery for big quantities and the type of the die (strip layout, speed) to build simple die or complex die is costly.

2.7.2 Behavior of sheet Metal Stamping Process:

To get the final products by stamping, forces must be applied on sheet metal by press, the press may be hydraulic or mechanical; the press applied force must be enough to cause deformation on the strip, until reach the wanted shape.

The metal stamping process can alter sheet-metal in many ways; Parts may be blanked, pierced, drawn, formed, or embossed, just to name a few basic operations. Each of these processes exerts its influence upon the structure of the material that of the part and that of the scrap.

Often, a congested piercing can cause stresses that will produce an increase in area measurements of that particular section, which is called bulging. Forming or drawing, on the other hand, can produce wrinkling, tearing, ironing, or undesirable folding of metal.[1]

2.7.3 Plasticity Theories:

In an attempt to control the material-related defect sheet metal, several theories on its plasticity developed. Plasticity of metal is the capacity to withstand the application of force, which-when excessive-may produce its deformation. When this force is small enough to fit within that material's yield strength limit, the deformation is only temporary, and after the release of the load, the material returns to its initial state. However, when the force exceeds the yield strength of that particular material, the resulting deformation of its crystal lattice remains permanent, and the part is permanently deformed, or perhaps, formed [1].

Stress force applied to the material can be categorized to fit into one of the three following groups:

- Linear influence of stress, during which the stress is applied along a single axis only, with the two other axes remaining stress-free
- Plane-type influence of stress, with stresses applied along any two axes, while the third axis remains stress-free
- Volumetric influence of stress, where stresses act along all three axes

Needless to say, the vast majority of metal-forming processes belong in the third, volumetric, group.

Plasticity theory is a mathematically oriented approach to evaluation of metal-altering (forming) processes. However, its mathematical equations, when used to solve actual problems, were often found quite inadequate and invalid. For that reason, a method called an elementary plasticity theory was developed. [1]

According to the elementary plasticity theory; various forces can be calculated without immediately considering the metallurgic properties of the material, and this way their behavior and their influence during metal's shape-altering (forming) process can be predicted. This theory, based on actual records of macroscopic observations of the deformation stage, deals with concrete data pertaining to particular qualities of the material in question, such as the stress-strain rate and yield criteria in tension and compression. A material's background data, such as metallurgical processes chosen for its manufacture, are mostly omitted.

During the metal-forming (or deforming) process, a displacement of metal material occurs, and its direction coincides with the direction of forces acting upon it. If we take a small particle of material and make it a representative of the whole mass, the displacement looks similar to that shown in Fig (2.10).

The amount of time such a displacement needs to occur is called Δt. During that period points A, B, C will change their location to A', B', and C', in congruence with the direction of forces acting upon them, and this movement will permanently alter the shape of that particle. We may generalize that if (at least) two points in any material change their relative location within the time interval t, that particle is exposed to the influences of deformation, or strain.

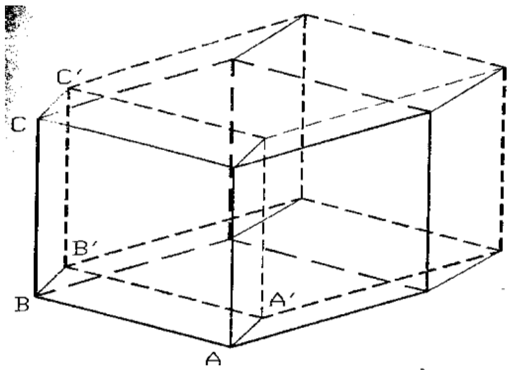


Fig. (2. 9) Displacement of particle in time Δt

The type of deformation most often considered in metal-forming processes is permanent deformation. The nonpermanent deformation, or elasticity, is usually neglected unless it falls into the category called spring-back.

2.8 Types of dies:

There are many operations can be done on sheet metal as known previously, up to variations of these operations the dies also vary in types and construction.

2.8.1-Cut off Dies:

The basic operation of a cut off die consists in severing strips into short lengths to produce blanks. The line of cut may be straight or curved, and holes or notches or both may applied in previous operations. Cutoff dies are used for producing blanks having straight, parallel sides because they are less expensive to build than other dies. In operation, see fig (2-11) the material strip A is registered against stop block B. Descent of the upper die causes the cut off punch C to separate the blank from the strip. Stop block B also guides the punch while cutting occurs to prevent deflection and excessive wear on guide posts and bushings. A conventional solid stripper is employed.[3]

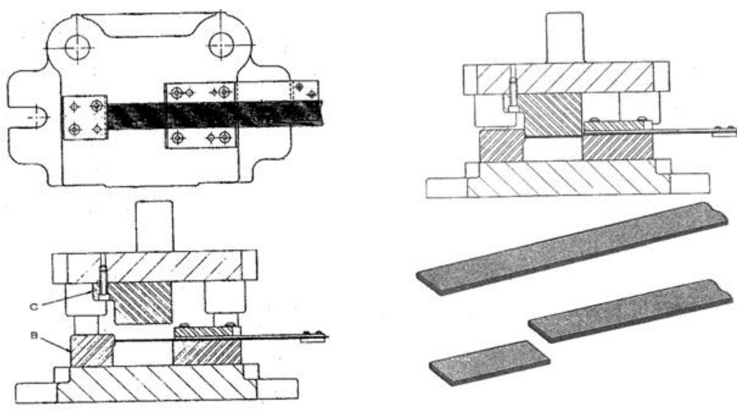


Fig. (2. 10) Cut off dies

2.8.2 Blanking Dies:

A blanking dies produce a blank by cutting the entire periphery in one simultaneous operation. Three advantages are realized when a part is blanked:

1- Accuracy. The edges of blanked parts are accurate in relation to each other.
2- Appearance. The burnished edge of each blank extends around its entire periphery on the same side.

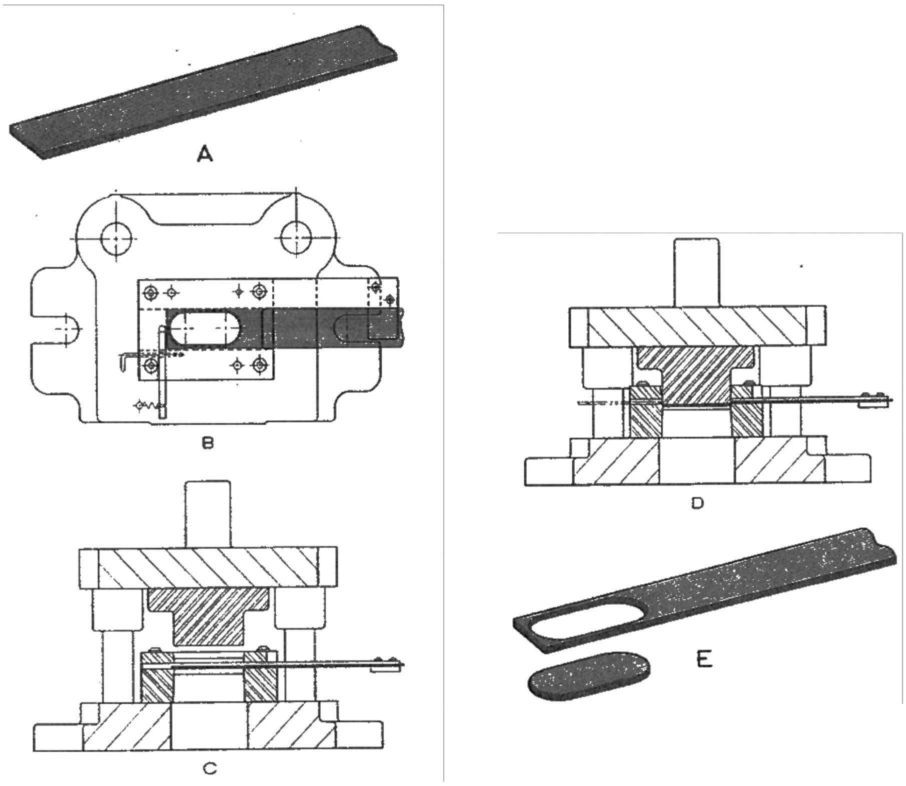


Fig. (2. 11) Blanking die

3- Flatness. Blanked parts are flat because of the even compression of material between punch and die cutting edges.

In fig (2-12) the inset at A shows a material strip ready to be run through a blanking die. At B is shown the top view of the die with punches removed. The section view at C shows the die in the opening position with the upper punch raised to allow advance of the strip against the automatically stop. At D the die is shown closed with a blanked punch out of the strip.

Blanking dies may produce plain blanks as shown in inset E but more frequently holes are pierced at one station and the part is then blanked out at the second station. Such dies are called (pierce and blank) dies.

A- A material strip ready to be run through ablnking die

B- top view of the die with punches removed

C- Die in opening position with the upper punch

D- The die closed with a blanked punch out of the strip

E- Blanking die may produce plain blanks

2.8.3 Compound dies:

In a compound die, seen in figure (2.13) holes are pierced at the same station as the part is blanked, instead of at a previous station as is done in a pierce and blank die. The result is greater accuracy in the blank. Whatever accuracy is “built in” the die will be duplicated in every blank produced by it. [3]

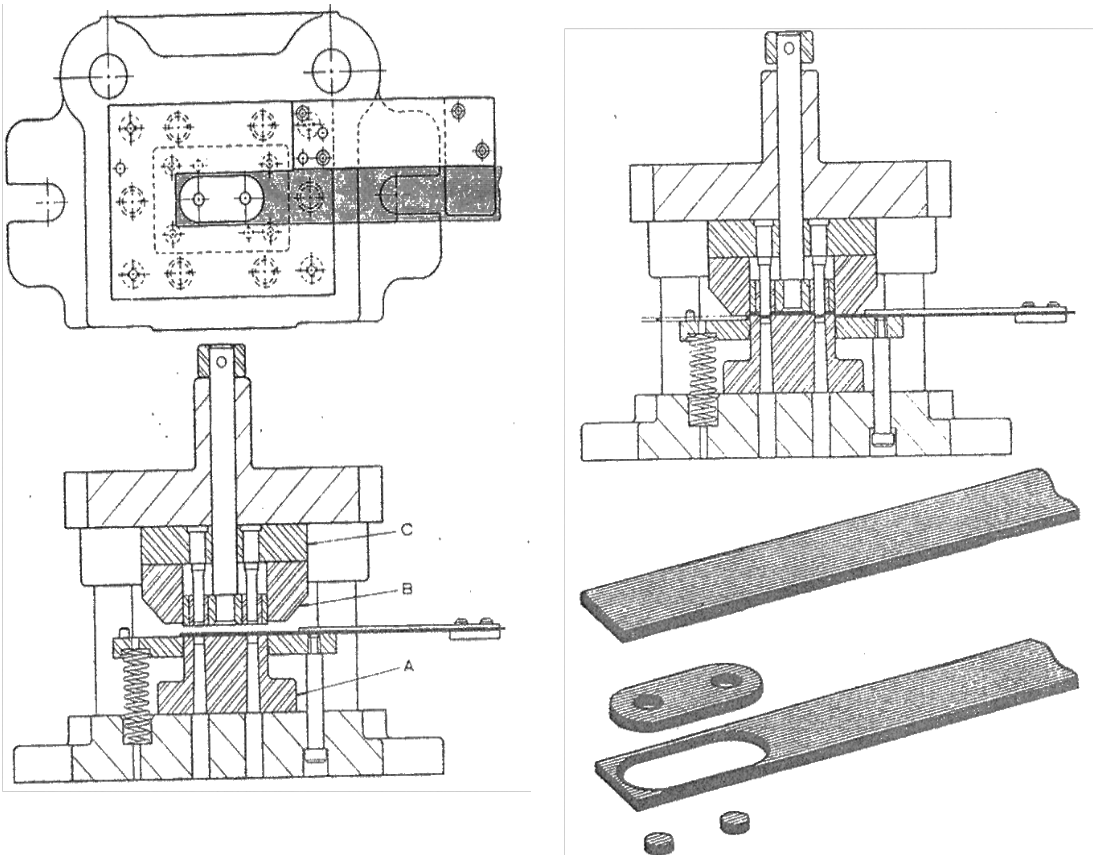


Fig. (2. 12) Compound dies

2.8.4 Trimming Dies:

Trimming dies shown in figure (2.14) cut away portions of formed or- drawn work pieces that have become wavy and irregular. This condition occurs because of uneven flow of metal during forming operations. Trimming removes this unwanted portion to produce square edges and accurate contours. The illustration in Fig. (2.13), shows a flanged shell after the drawing operation.

A trimming die is required to trim the irregular edge of the flange. The shell is placed over a locating plug B and descent of the upper die causes the scrap ring to be cut from around the flange. After trimming, the shell is carried up in the upper die and a positive knockout ejects it near the top of the stroke. The scrap rings are forced down around the lower trimming punch until they are split in two by scrap cutters C applied at the front and back of the die. The scrap pieces fall to the sides, away from the operation of the press.

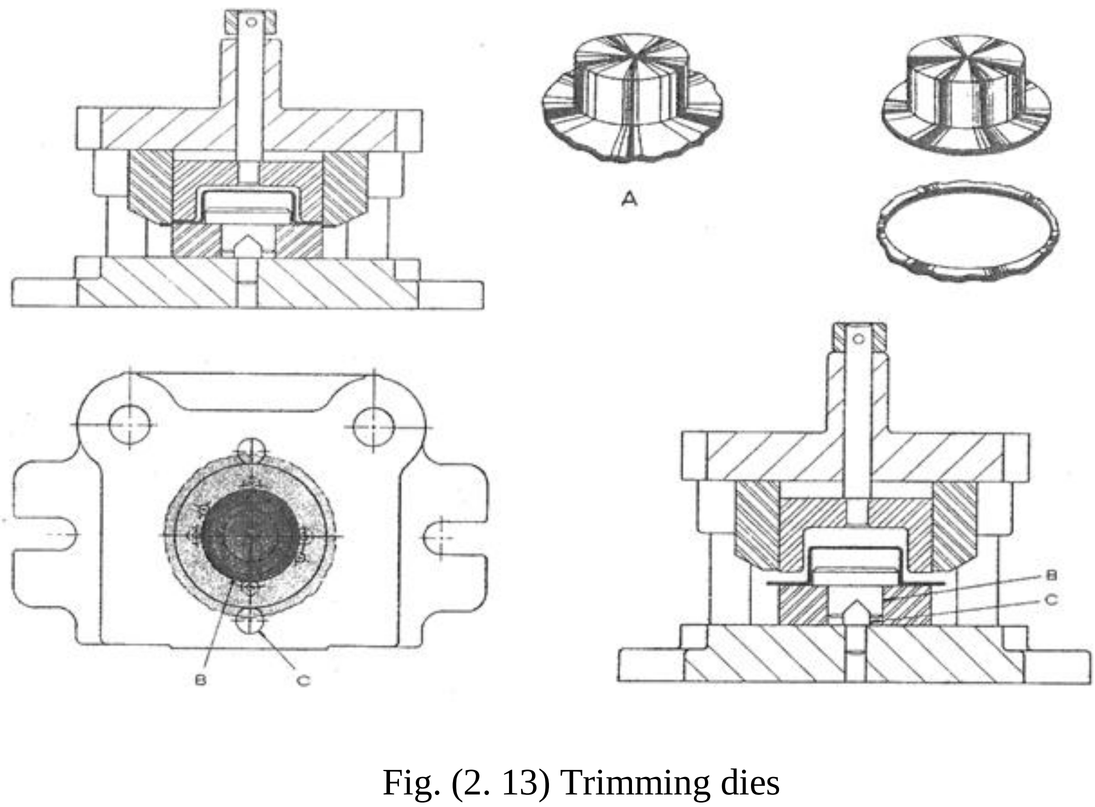


Fig. (2. 13) Trimming dies

2.8.5 Piercing Dies:

Piercing dies pierce holes in stampings. There are two principal reasons for piercing holes in a separate operation instead of combining piercing with other operations which are:

1. When a subsequent bending, forming, or drawing operation would distort the previously pierced hole or holes.
2. When the edge of the pierced hole is too close to the edge of the blank for adequate strength in the die section. This occurs in compound and combination dies in which piercing and blanking are done simultaneously.

In the inset in Fig (2.14) is shown a flanged shell requiring four holes to be pierced in the flange. If the holes were pierced before the drawing operation they would become distorted because of the blank holder pressure applied to the flange in the drawing process.[3]

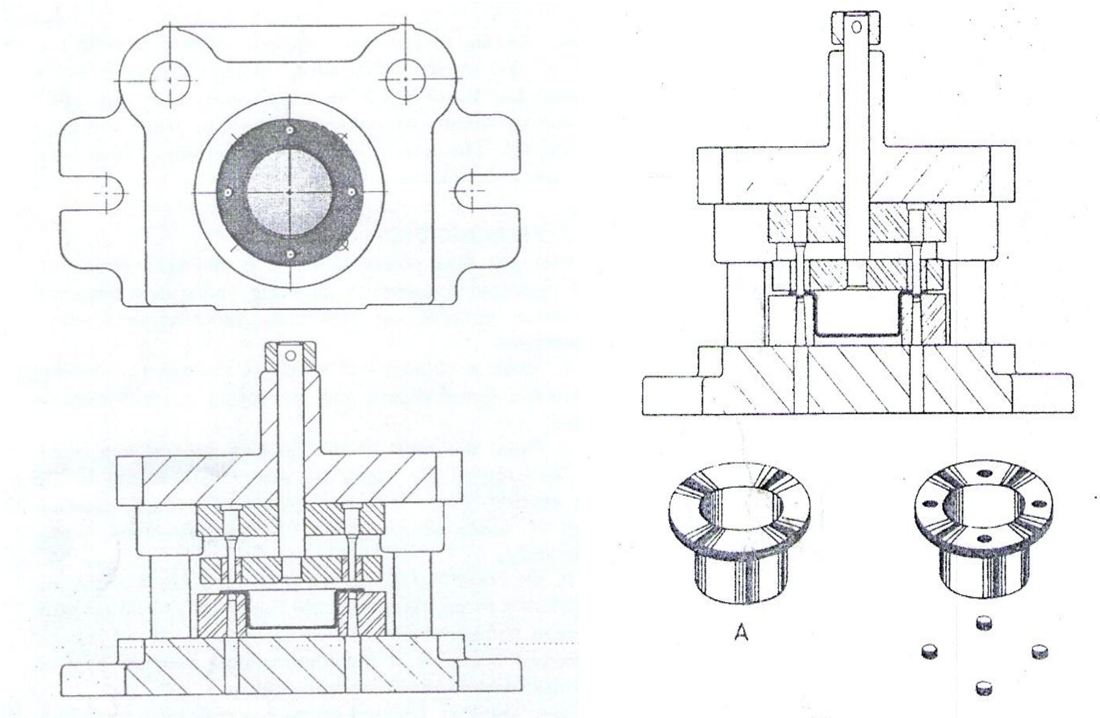


Fig. (2. 14) Piercing dies

The shell is located in an accurately ground hole in the die block. Piercing punches are retained in a punch plate fastened to the punch holder, and a knockout effects stripping after the holes have been pierced.

2.8.6 Side Cam Dies:

Side cams shown in Figure (2.16) transform vertical motion from the press ram into horizontal or angular motion and they make possible many ingenious operations, in the illustration at A, a flanged shell requires two holes pierced in its side. The shell is placed over die block B of the die. Descent of the upper die causes pressure pad C to seat the shell firmly over the block. Further descent causes side cams D to move the punch - carrying slides E for piercing the holes. Spring strippers F strip the shell from around the piercing punches as they are withdrawn.

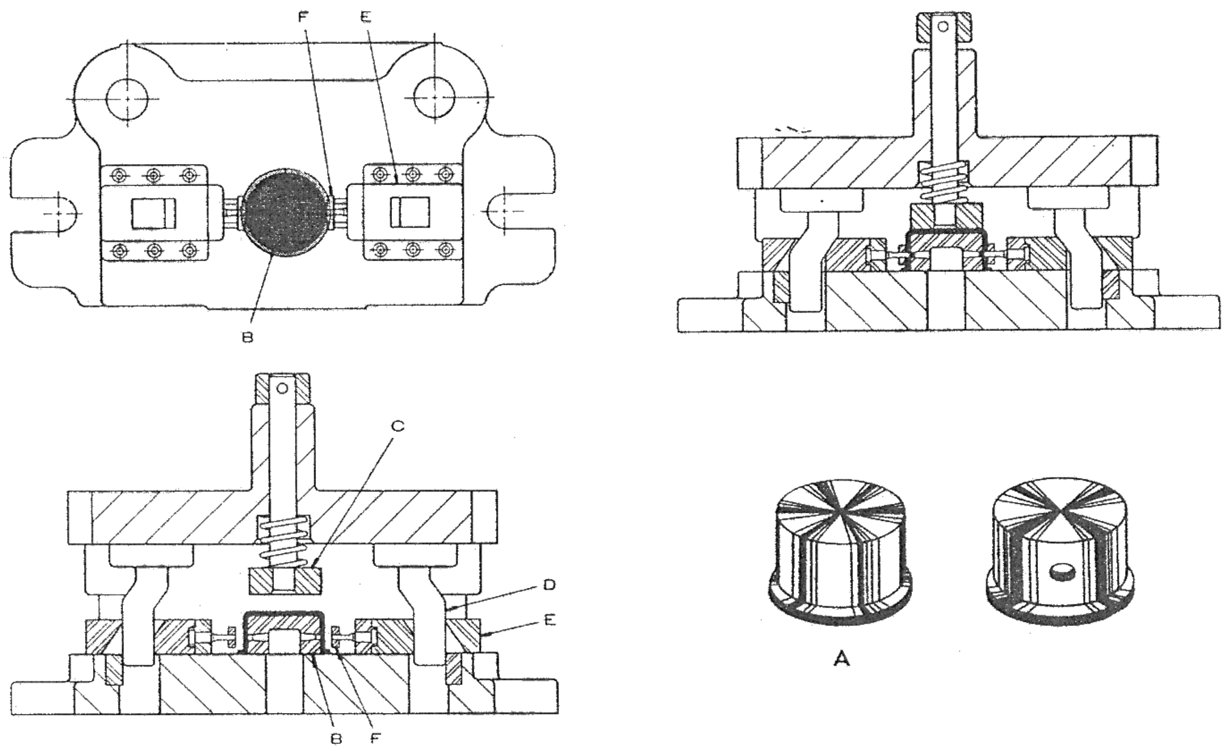


Fig. (2. 15) Side cam dies

2.8.7 Bending Dies:

A bending die fig (2.17) deforms portions of flat blanks to some angular position. The line of bend is straight along its entire length, as differentiated from a forming die which produces work pieces having a curved line of bend. In the illustration, a flat blank is to be given a double bend to form a U shape. The blank is inserted in gages A fastened on bending blocks B. The bending blocks, in turn, are fastened to the die holder. Upon descent of the upper die, the bending punch C grips the blank between its lower face and pressure pad D. Pins E extend to the pressure attachment of the press. Shedder F strips the work piece from the punch.

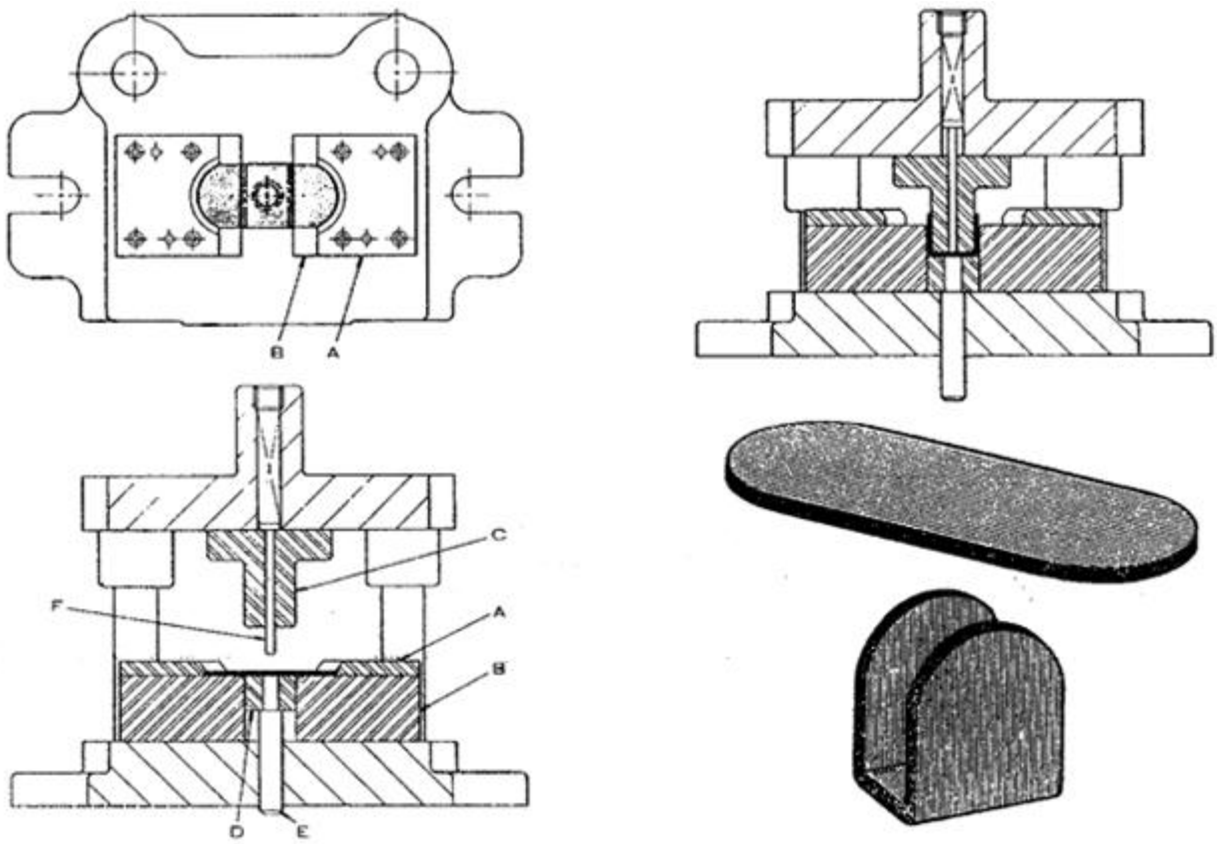


Fig. (2. 16) Bending dies

2.8.8 Forming Dies

The operation of forming is similar to bending except that the line of bend is curved instead of straight and plastic deformation in the material is more severe. In the illustration in fig (2.18) the flat blank at A is to be formed into a part having a curved contour. The blank is positioned in nest B composed of two plates mounted on pressure pad C. When the ram descends, the blank is gripped between the bottoms of forming blocks D and the surface of pressure pad C.

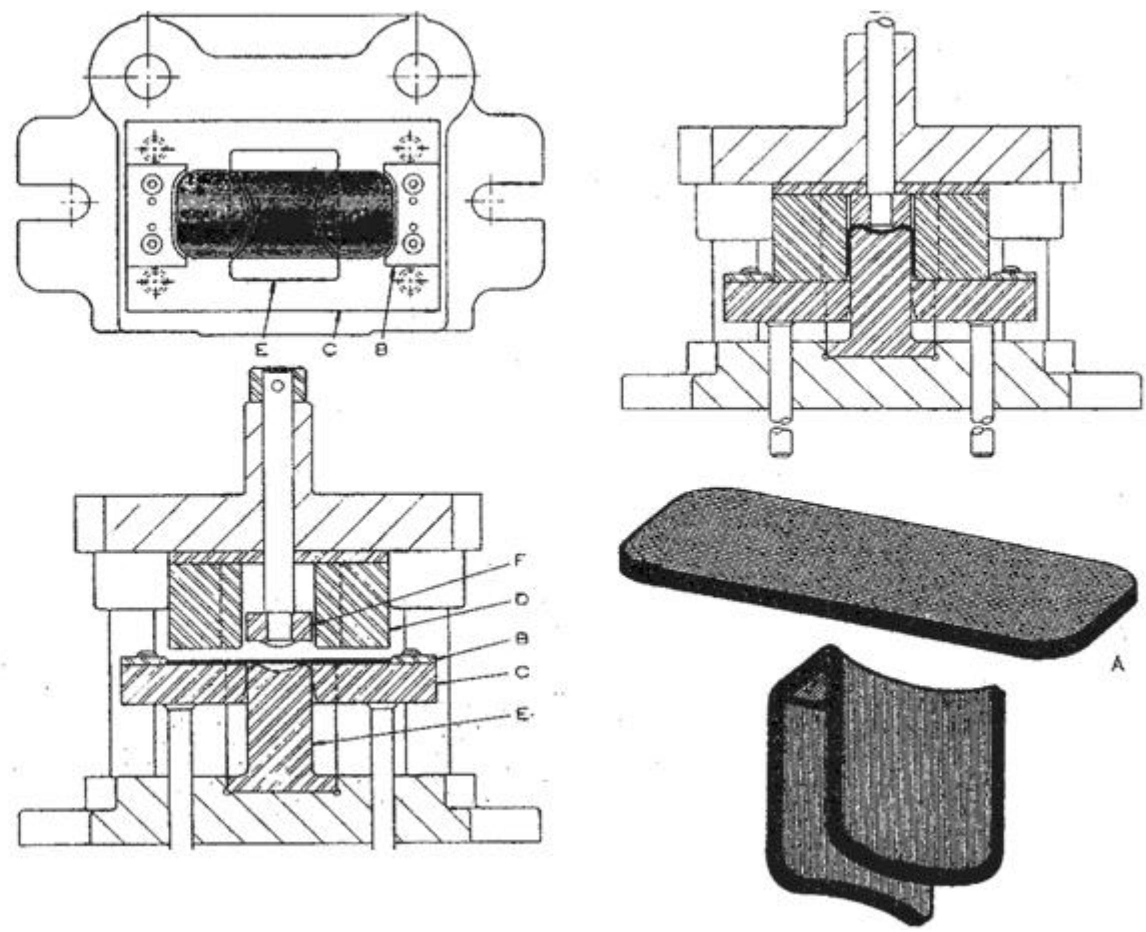


Fig. (2. 17) Forming dies

Further descent causes the sides of the blank to be formed to the curved shape of forming blocks D and forming punch E. At the bottom of the stroke, knockout block F applies the final form. It bottoms against a hardened spacer fastened to the punch holder thus “setting” the form. When the die ascends, the part is carried up within form blocks D. Near the top of the stroke it is ejected by knockout F.

2.8.9 Drawing Dies:

The operation of drawing is similar to forming, although usually there is more severe plastic deformation in the material. The difference between the two occurs in the extent of closure in the form. Consider a drawn cup such as a metal drinking cup. The material extends all around the sides and therefore the part is said to have been drawn. In a formed part the material does not extend completely around to surround a space, even though the formed contour may be quite intricate. In the illustration at fig (2.19) A, a flat disk is to be drawn into a cup. The blank is placed on pressure pad B of the drawing die and it is located by four spring-loaded pins C. Descent of the upper die causes the blank to be gripped securely between the surface of pressure pad B and the lower surface of draw ring D. Further descent of the ram causes the blank to be drawn over punch E until it has assumed the cup shape shown in the closed view at the right. Pressure pins F extend to the pressure attachment of the press.

The amount of pressure must be adjusted carefully. Excessive pressure would cause the bottom of the cup to be punched out. Insufficient pressure would allow wrinkles to form. With the proper amount of pressure, a smooth, wrinkle-free cup is produced. Drawing is extensively used for producing stampings ranging from tiny cups and ferrules to large shells for pressure vessels, ships, aircraft, and missiles.

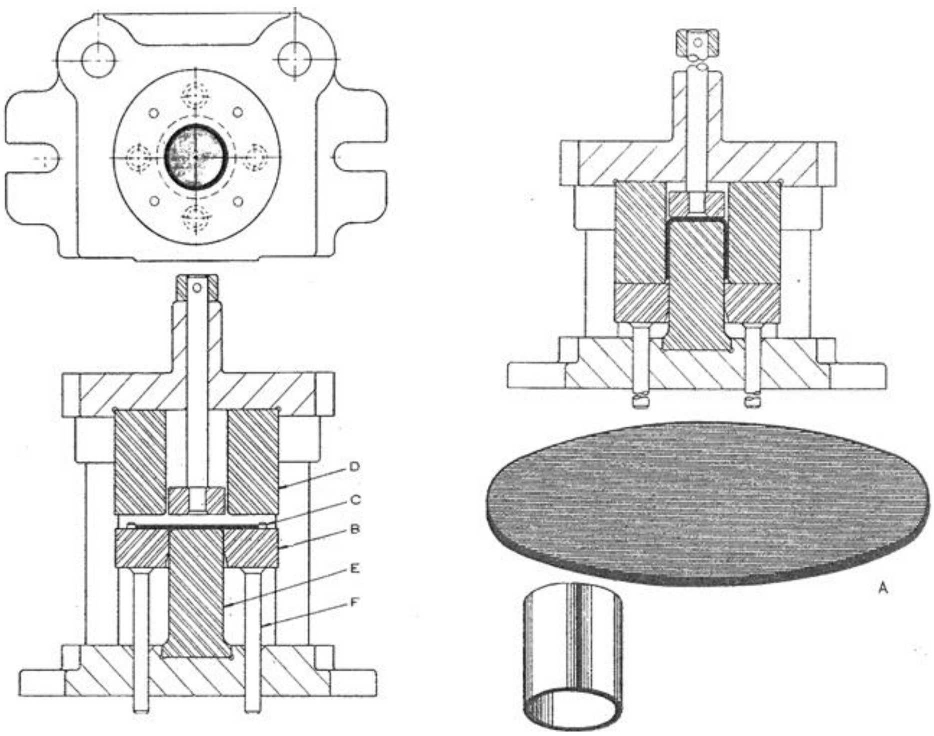


Fig. (2. 18) Drawing dies

2.8.10 Curling Dies:

A curling die forms the material at the edge of a work piece into a circular shape or hollow ring. Flat blanks may be curled; a common application is a hinge formed of two plates each of which is curled at one side for engagement of the hinge pin. More often, curling is applied to edges of the open ends of cups and shells to provide stiffness and smooth, rounded edges. Most pans used for cooking and baking foods are curled.

In the illustration at figure (2.20), a drawn shell shown at A is to be curled. The shell is placed in the curling die where it rests on knockout pad B. Descent of the upper die causes the knockout pad to be pushed down until it bottoms on the die holder. Further descent causes curling punch C to cur1 the edge of the shell. Near the bottom of the stroke, the lip of the material contacts an angular surface machined in curling ring D to complete the curl. When the punch goes up, the knockout raises the shell for easy removal.

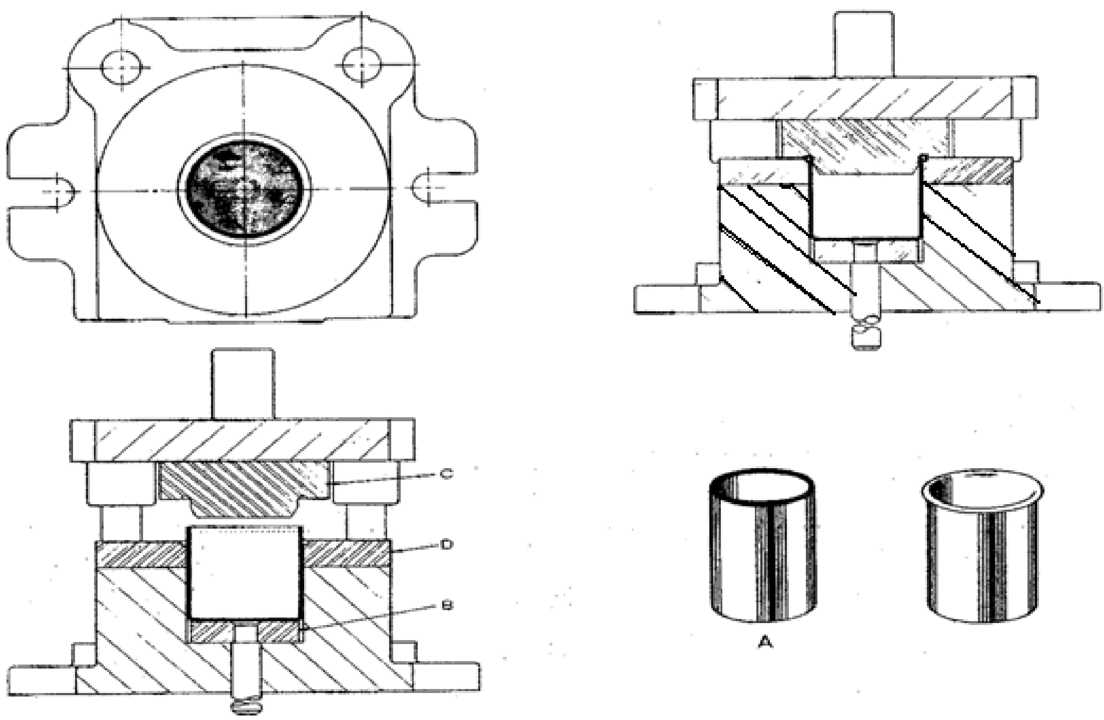


Fig. (2. 19) Curling dies

2.8.11 Extruding Dies:

The function of all the dies discussed so far is to perform work on sheet material — to cut sheet material into blanks, to perform further operations upon the blanks, or to perform operations on work pieces bend, formed, or drawn from the blanks. We come now to interesting classes of dies that perform secondary operations on small thick blanks called slugs. In these dies the slugs are severely deformed to make parts having no resemblance to the slugs from which they were made.

The first class is called extruding dies. In this type of die each slug is partly confined in a cavity and extremely high pressure is applied by a punch to cause the material in the slug to extrude or squirt out, much like toothpaste is extruded when the tube is squeezed. In the illustration (fig 2.21) the slug A is to be extruded into a thin-walled shell having a conical closed end. The slug is placed in die block B, backed up by a hardened plate C. The bottom of the cavity in the die block is formed by the end of knockout rod D.

When the press ram descends extruding punch E first squeezes the material in the slug until it assumes the shape of the die cavity and of the, working end of the extruding punch. Continued descent causes the material to extrude upward between the wall of the punch and the wall of the die cavity. The amount of clearance between the two determines the thickness of the wall of the extruded shell. The extruding punch is retained in punch plate F and because of the high pressure involved; it is packed up by backing plate G.

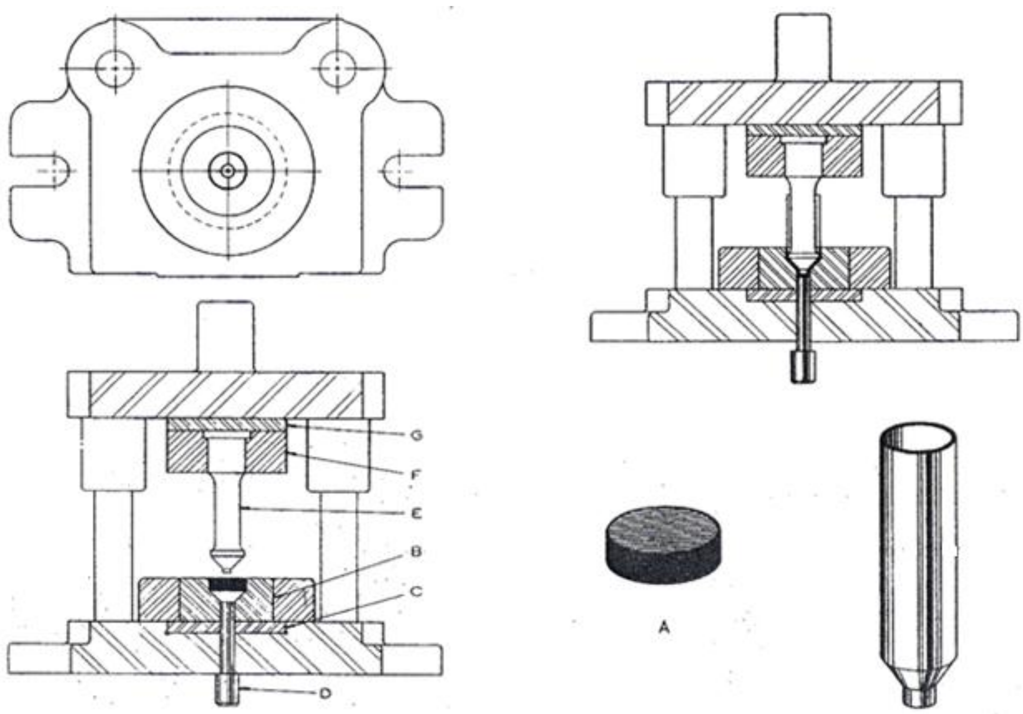


Fig. (2. 20) Extruding dies

2.8.12 Progressive Dies:

In a progressive die see fig (2.21) the strip is moved in stages from station to station. Different operations are performed on it at each station except at idle ones applied to provide room for components. A complete work- piece is removed from the strip at the final station. All of the operations described previously may be performed in progressive dies, For example, a single die of this type may do piercing at the first station, trimming at the second station, bending at the third, forming at the fourth etc. A progressive die may thus be considered as a series of different dies placed side by side with the strip passing through each successively. This analogy has some merit although it does not give a true picture of the extremely close interrelation ship between the various stations.

In the illustration in Fig (2.22), a pierced, trimmed and bent part is to be produced complete in a simple progressive die. At the first station the strip is notched and pierced and at the second station the blank is cut off and bent.

You should easily recognize all of the elements in this die-the die block, piercing punch, trimming punch, knockout, stop block, and all the others.[2]

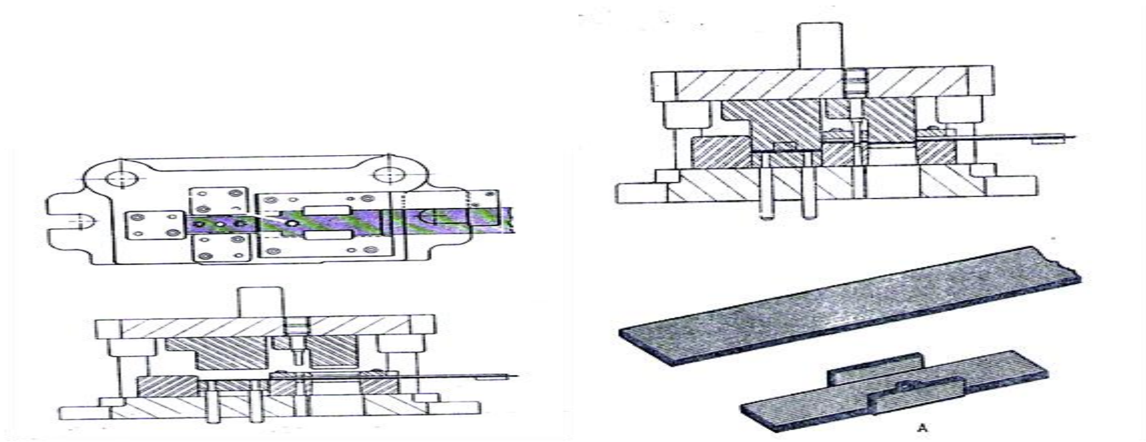


Fig. (2. 21) Progressive dies

2-9 Previous studies

Many studies conducted considering the progressive die design and analysis for many fields and applications one of the most important studies are the one by (Osman, 2010) in which he identify a new method for the production of sheet metal products by means of dies, especially (Progressive Die), also he identify parts of the Die and how to design, functions and relations between each other. This achieved by using the design software (VISI15) [5].

Esche et al 1996 developed knowledge-based systems to determine required forming stage in deep drawing of sheet metal components .knowledge for these systems is derived from plasticity theory, experimental results, and the empirical knowledge of field engineers. This approach has showed some success. However, it cannot consider the process conditions that are not already stored in the knowledge base [6].

Kim et al, 1998, 2000, proposed a roll-back method to predict the optimum initial blank shape. The method considering the final deformed shape and the target contour shape into account.

The MEL system which offers advantages in improving the design process is described. Their system consists of ten steps. Each step can be used independently of the others. This method enables us to design a new die, and, in addition, changes and corrections of part of an existing die can be done easily. In our system, a product shape is input hierarchically using elements such as points, lines, and circles, closed figures consisting of these elements, patterns consisting of these closed figures, and the product consisting of these patterns. We introduce a concept of the basic pattern which corresponds to the shape of a standard punch. There are several of these. These patterns can be input efficiently using the macro commands for the pattern in the hierarchy, hence input is simple and the programming can be done quickly. The great benefits accrue from these patterns, i.e., the most effective blank and strip layout including information of the tool configuration can be done quickly [7].

Moreover, this study about Computer-aided progressive die design system and method. According to one embodiment of the invention, a computerized method for designing a progressive die used in the manufacturing of a part formed from sheet metal includes receiving, at a computer, information regarding one or more features of the part, and determining, by the computer, a blank layout for the part based on the features of the part and the number of parts desired. The computer further determines one or more details of a strip for the blank layout, information regarding a die base based on the details of the strip, and information regarding one or more inserts for die plates of the die base based on operations of the processes needed to form the features in the part. The computerized method further includes generating, by the computer, one or more outputs associated with the progressive die.[7] .

CHAPTER THREE

PROGRESSIVE DIE

3-1 Progressive die:

A progressive die is a metalworking device that is designed and built to convert a strip of metal (raw material) into parts or product that conform to blueprint specifications. The "dies" are placed into a stamping press. As the stamping press moves up, the die opens. As the stamping press moves down, the die closes. The raw material (strip) moves through the die while the die is open, being fed into the die a precise amount with each stroke of the press. When the die closes, the die performs its work on the strip and one or more finished parts are ejected (usually by gravity) from the die. All types of operations are done on sheet metal in single dies can be performed in a progressive die, that means a progressive die is many single dies placed in one frame.

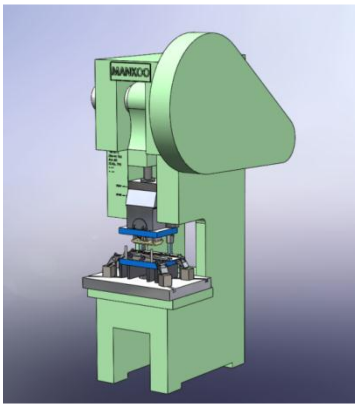


Fig. (3. 1) Stamping press with progressive die

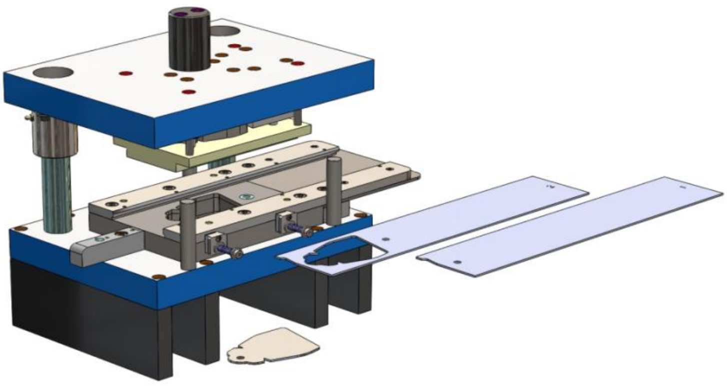


Fig. (3. 2) Component of progressive die And striper

The progressive die can modify the raw material in several ways, such as bending, coining, and punching. Holes that are cut into the raw material can be almost any shape. Progressive dies are widely used to produce components for various industries (civil and army) such as computer, electronics, automotive and electrical appliances. The requirements for progressive die to design and manufacturing are becoming more stringent as shorter delivery times, higher quality requirements and more innovative designs stretch abilities. A progressive die combines a number of forming and stamping functions such as blanking, forming, and flange forming, punching and trimming into a single die. Steel or aluminum is fed into the die, typically from a coil of material. Each time the die cycles a stamping operations is made on the strip and the strip is automatically advanced to the next position or station. Each station within the progressive die serves to progressively form, the final part, finally completed part is ejected from the end of the progressive die once all the operations have been completed. Progressive dies are ideal for economically mass producing small formed metal parts. Since additional work is done in each "station" of the die, it is important that the strip be advanced very precisely so that it aligns within a few thousandths of millimeters as it moves from station to station. Bullet shaped or conical "pilots" enter previously pierced round holes in the strip to assure this alignment since the feeding mechanism usually cannot provide the necessary precision in feed length. The key components of dies are made of tool steel to withstand the high shock loading involved, retain the necessary sharp cutting edge, and resist the abrasive forces involved.

The saving in time, cost, labor and high quality are the main feature of progressive die, the cost is determined by the amount of features, which determine what tooling will need to be used. It is advised to keep the features as simple as possible to keep the cost of tooling to a minimum. Features that are close together produce a problem because it may not provide enough clearance for the punch, which could result in another station. It can also be problematic to have narrow cuts and protrusions.

3-2 Design Steps and Analysis of progressive dies

3-2-1 Progressive Die Elements Designation

Die block, top plate, bottom plate and Guide pillars are the most basic elements of die set and these two plates are guided by the guide pillars. This alignment is done here for improving accuracy, part quality, die life and reduce the set up time. The lower plate support the die, die housing, raisers, and the top plate supports the punch, punch backup plate, guide pillar bush.

1. Die Block:-

The bottom assembly is the female part of the punch tool. The most important part of bottom assembly is die block. The cutting edge is given to the die block. The land is also provided on the die block. This land is provided for the proper cutting.

2. Bottom plate

The bottom plate is the base of lower assembly. There is die block, raiser, guide pillars are mounted on the bottom plate. This bottom plate is used to clamp the lower assembly of press tool over the bolster plate by using the clamping devices. The bottom plate is made from the mild steel material.

3. Punch

Punch is made part of the press tool. The cutting operation is carried out here. So the material required for manufacturing the punch is harder than the part material. Thickness of punch is depending on the punch alignment with stripper plate, compressed length of spring, and with punch travel. So here we take 71 mm as punch thickness. The clearance is given on punch in blanking operation. For the piercing punch there is its length is larger than its diameter. So the effect of length occurs on the performance on overall performance.

4- Top Plate:-

The top plate is the base of upper assembly. There is punch, springs, guide pillars bush are mounted on the top plate. This top plate is used to clamp the upper assembly of press tool over the bolster plate by using the clamping devices. The top plate is made from the mild steel material.

5-Guide Pillar:-

They are used for the alignment of upper assembly with lower assembly. It guides the punch through the die. These also manufacture by harden material to resist the buckling effect.

3-2.2 Material selection:

To specify the type of material for die, the designer must take in account the number of products see table (3-1), which specifies product material and quantity, and table (3-2)specify the plate's material up to product quantity. The whole press tool is not required high strength material. Because all parts of press tool does not involve in operation. Here the more important members are die, punch, and hitting pad. These parts are required more strengthful material. The main principle while selecting the material for that part is given as;

i. The tool material have more wear, abrasive or adhesive resistance than the part material. Also its friction force is more than part material.

ii. The hardness of material is more than the part material.

iii. Fatigue, shear, compressive strength is more than part material strength and plastic or elastic deformation strength is less than the part material strength.

Table (3.1) Die material selection

Work piece material	**Small**	**Medium (Thousand)**	**Large (Millions)**	**Very Large (Million)**
Copper / Al	1 – 50000	50 - 500	0.5-10	>10
SPCC	1 – 20000	20- 200	.2 – 5	>5
SUS	1 – 10000	10 - 100	0.1-1	>1
Hard Steel	1 – 5000	5-50	0.05-1	>1

The material of part is mild steel. So the type of steel is used for punch and die. It has more hardness and strength than the mild steel [3].

The progressive dies relies heavily on past experience and several prototype die tryouts. The most critical and challenging issues in this task are how to determine the:

a) Minimum number of required forming steps.
b) The corresponding tooling shapes while maintain the specified thickness distribution in the formed part.

This procedure requires extensive resources and increases the process development time and cost. A computer aided approach is highly desirable to design a robust progressive die sequence quickly and at low cost [3].

3.2.3 Clearance Calculations

Die clearance is depend on the part material property. If the material is ductile in nature then the clearance is small and for brittle material it is large clearance. If the clearance is given in reverse then there for ductile material it pass through die means here it draw from die instead of cutting. And in ductile material it damages the cutting edges of punch and die. Large clearance increases the tool life. So here take 10% of thickness per side [3].

Table (3.2) clearance of material

Thickness mm	**Mild steel 16-20%**	**Stainless steel 18-24%**	**Aluminium 12-16%**	**Copper 10-14%**
0,5 – 0,6	0,08-0,1	0,1- 0,12	0,06 – 0,08	0,05 – 0,06
0,8	0,14 – 0,16	0,15 – 0,2	0,1 – 0,14	0,08 – 0,1
1	0,16 – 0,2	0,18 – 0,24	0,12 – 0,16	0,1 – 0,14
1,2	0,2 – 0,24	0,24 – 0,3	0,15 – 0,2	0,12 – 0,15
1,5	0,25 – 0,3	0,27 – 0,35	0,18 – 0,24	0,15 – 0,2
2	0,34 – 0,4	0,36 – 0,45	0,24 – 0,3	0,2 – 0,25
2,5	0,45 – 0,5	0,45 – 0,55	0,32 – 0,35	0,25 – 0,3
3	0,5 – 0,6	0,6 – 0,7	0,35 -0,45	0,3 – 0,4

Table (3.3) Cutting clearance in percentages of material thickness per side [3]

All materials, up to 0.001 in. (0.03 mm) thickness	0.0001 in. per side 0.003 mm per side
All materials, 0.001–0.025 in. (0.03–0.65 mm) thickness	5% per side
Blue steel, up to 0.025 in. (0.65 mm) thickness	
Copper, brass, soft steel, 0.025 in. (0.65 mm) thick and up	5–8% per side 5% per side, ID
Medium hard steel, 0.025 in. (0.65 mm) thick and up	8% per side, OD 6% per side, ID
Stainless steel, carbon steel, Cr-Ni steel, 0.025 in. (0.65 mm) thick and up	9% per side, OD 8% per side, ID 10% per side, OD

Note: Clearances for aluminum to be used according to its comparable hardness.

Table (3.4) Plates Material [3]

Plate	Small	Medium	Large	Very Large
Die	SK3,SKS3	SK3,SKS3,SKD11	SKD11,SKH9D2 1.2379	WC-CO,FZ
Punch	SK3,SKS3	SK3,SKS3,SKD11	SKD11,SKH9D2 1.2379	WC-O,FZ,SKH9
Stripper	S50C,SK3	S50C,SK3,P20	HPM2-T,SKD11	HPM2-T,SKD11
Backing	--	--	SK3,SKS3	HPM2-T
Upper	St37,S45C, SS400	SS400,S50C	S50C	HPM2-T
Lower	SS400,S50 C	S50C	S50C,SK3	HPM2-T

3.2.4 Strip layout design

With every new part produced, a complete evaluation of the stamping method and parameters must be performed. Based on the part's flat layout, the sequence of tooling must be performed. The sequence of tooling must be designed, which in turn dictates the size of the die.

The economics of the strip must be assessed before the rest of the design is finalized. Seemingly small details such as the availability of strip material, the predetermined width, and its thickness and tolerance ranges may turn out to be of tremendous importance when it comes to production. Only then may the actual design be started, which always begins with the strip layout, and its projection into the cross section of a die. In other words this layout will fix all die parameters. Such a sequence of work process is intentional, as the cross-sectional view provides control of the placement of punches within the assembly.

Strip layout is the process of determining the sequences of operations doing on strip which is first and why, until reach to final product (piercing blanking bending cutting), through these operations may be some idle stations, because the implementing of two operations in one station may cause some problem.[3]

This layout may vary from designer to designer therefore it is preferred to be done in a group, and must obtain the effective use of the raw material that mean less scrap and must be easy to implement as you can. One of the main determinants in an assessment of appropriate strip layout is the production rate expected from the die. Where 100,000 pieces are to be delivered within a month is completely different from that where the same number of parts must be produced within a week or perhaps even a day.

To evaluate the problem of production rate properly, a rough estimate of the tonnage and die size must be made for selection of a suitable press. These are preliminary assessments and need not be based on elaborate calculations or sketches. A hand sketch of the strip will often suffice, showing only the sequence of tooling and its location. Once a press of appropriate tonnage, bed size, stroke, and shut height is selected, scheduling of this machine has to be consulted to find out its availability. It is important to know what other jobs may be running at the time the new production is to begin, if such runs may be interrupted, or if a rigid schedule denominated by firm deliveries is to be observed. [2]

With regard to the number of parts to be produced, an accurate strip layout should be drawn next. It will help to evaluate the correctness of the first rough assumption, and it will also establish the exact location of blanks within the strip. Parts may be positioned horizontally, vertically, or at an angle. They may be placed beside each other or intertwined or we can produce two or three products in every stroke. The usages of computer software's (for example solid works) will simplify this layout.

CHAPTER FOUR

PROGRESSIVE TOOL DESIGN AND MODELING (CASE CTUDY)

4.1 Preface

This chapter discusses progressive Tool design and modeling for (metal part of fluorescent lamps (T-shape)) as below:

4.2 Case Study (Part Data):

The designer was select a product from Al Madinah Al Munawarah Factory for Electric Lamps products, this part serve as motion transmission produced by three single die , now the new design for a new tool (progressive die) by using it we can get the final product in one step.

Through the usage of progressive die the quality will improved and the cost will reduction and the number of labors will reduce from three to one.

Fig. (4. 1) Case study

Metal part of fluorescent lamps (T-shape)

4.3 Selecting Strip Material

The designer select mild steel for strip because Mild steel is the most commonly used steel. It is used in the industries as well in the different everyday objects we use. The mild steel is very important in the manufacturing of metal items. Almost 90% steel products of the world is made up of mild steel because it is the cheapest form of steel.

4-3-1 Efficiency of strip calculation:-

Figure (4-2) shows the efficient calculation minimize the material lost I the same time provide the essential sheet size with neither nether integrity nor geometrical defect. It’s possible to minimize width (b) by zigzagging the cutting process but this increased time required when considering the case of mass production.

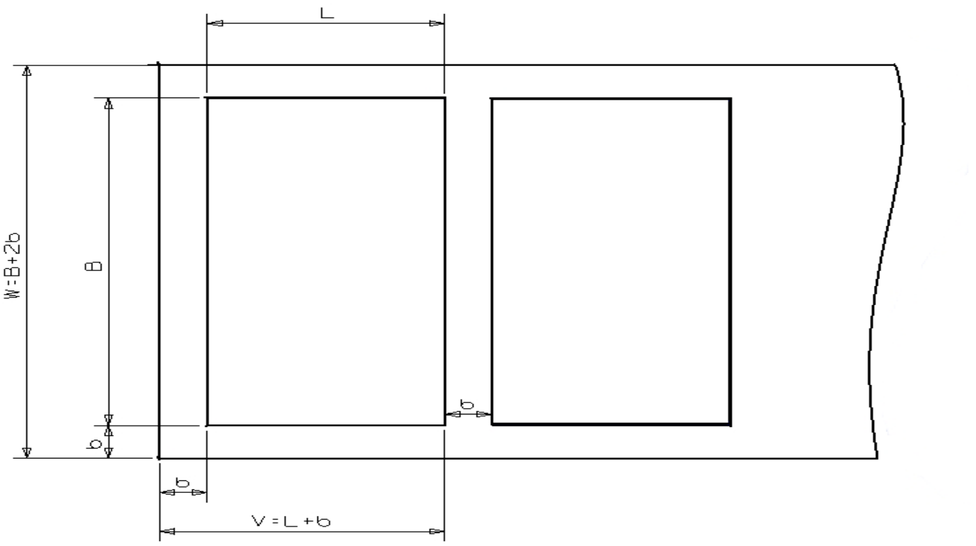


Fig. (4. 2) strip layout

Formula:

$$N = \frac{A_P}{A_S} \quad \text{.................. (4-1)}$$

$$Z_W = \frac{L_S}{V} \quad \text{...................... (4-2)}$$

$$N = \frac{(Z_W \times A_P)}{(L_S \times B)} \quad \text{...................... (4-3)}$$

$$V = f + e \quad \text{.............................. (4-4)}$$

$$A_s = B \times V \quad \text{.............................. (4-5)}$$

Where:

N= Efficiency of strip using

Ap =area of part in (mm2)

B= width of strip in(mm)

L=length of strip in (mm)

As = area of the strip in (mm2)   Zw=number of parts

V= feed in (mm).

4-4 Comparison between progressive die and three single dies:

The comparisons between the two types of dies include the following factors (see table (4-1):

1-Time factors: Setup time and production time.

Setup time: is the time needed to mount the die on the press, and set the stroke length. This time vary from fifteen to twenty minutes for both types of dies. Therefore to mount all the five single dies, around hundred minutes is needed.

Production time: is the net time to get final product. For progressive dies, production time starts at the first operation and finishes once product comes out of the die. Through experiments, production time in progressive die takes about five to ten seconds with manual feeding.

Then five single dies need twenty five second for one product.

2-Cost factors:

Number of labors: in dies production, a technician is needed to mount the dies and monitor their operation. In the case of three independent dies then three technicians are needed for each die.

Number of presses and Power consuming: the progressive die needs only one press, while three single dies need three presses (almost same force).

Maintenance cost: at same conditions the maintenance cost of progressive die equal 20% of three single dies.

Die design &machining cost: to design a progressive die, CAD and die design software is needed, also an expert designer and many standard elements. Then to manufacture those progressive dies, precision manufacturing machines are needed (CNC machines). Compared to single die design and manufacturing, progressive dies take longer time to design and they are more difficult to manufacture, and it also consumes time on the assembly and testing.

3- Quality:

The process by single dies has a bigger room for error because on each die the workpiece has to be set on the machine manually. Though a skilled labor will not find it difficult yet it is hard to predict and avoid errors. Where in progressive dies, the position of the workpiece is controlled by an automated and precise pilot which minimize the margin for error. Moreover, single dies products need further treatments (bench work)

4-Mass production:-

All the comparison factors mentioned above point to the fact that progressive dies work faster, produce high quality products in a shorter time than single dies and it uses less labor. This makes it more suitable for mass production. (See table (4-1)) [4].

Table (4. 1) Comparison between progressive die and three single dies [4]

Parameters		***Progressive die***	***three Single dies***
Time factors	Setup time	15-20 minutes	3(15-20)minutes
	Production time	5-10second	3(5-10)second
Cost factors	Number of labors	1	3
	Number of presses and Power consuming	1	3
	Maintenance cost	For only one die	For three dies
	Die design &machining cost	Costly	Less cost than progressive die
Quality		Very good product	Less quality than progressive die
Mass production		Excellent	Time consuming

4.5 Modeling the Strip Layout

Many commercial software package are available by which designs, analysis, assemblies, 3D and 3D drawings, rendering and simulation could be achieved, among which solidworks are sued at current work to execute the design and calculation at deferent levels accompanied with the illustration of the procedure following program. From the Strip Manager select Load Strip Icon and select the unfolded part. Then select an origin point on the unfolded part now become able to pick an application point on the screen. Place the part at X0, Y0, Z0.

Hit the “ESC” once to initiate your strip definition two unfolded parts on the screen side by side will appear.

Steps Number:

This option allow user to set the number of steps shown on the screen to help determine the various parameters.

- Step: is a distance to set the progression of the strip.
- Strip Width: determines the width of the strip.
- Upper Discard: is used to set the distance above the part to the upper side of the strip.
- Lower Discard: is used to set the distance below the part to the lower side of the strip.
- Minimum Distance between Parts: sets the smallest distance allowed between parts
- Rotation Angle: is used to rotate the parts on the strip to help while nesting parts of Unusual shapes allowing possible overlaps to reduce progression.
- Step Origin: display the current origin and allow the user to change its location.

Cutting the Punches

Select the "Cut Punch" icon from the main Progress toolbar to begin separating the punch into smaller ones.

Now using the "Set Drawing Filters" icon, turn on the layer "Punch Profiles". This will display some predefined profiles in new punch locations.

Strip Simulation:

Now it is time to run the strip through the punches and view the results. Using the "Rebuild the Strip" icon brings up the "Strip" dialogue, shown Fig. (4-3).

1. Restart: will rewind the strip to the beginning.
2. Automatic: will advance the strip through the punches and then display.
3. Strip Simulation: will advance the strip through the punches and display after each step.
4. Back one Step: will move the strip back one progression.
5. Advance a Step: will move the strip forward one progression.
6. Only Sheared Strip: will advance the strip without showing the bend stages.
7. Check Problems: will display any problem areas.
8. OK: Confirms your action, and closes the dialogue.

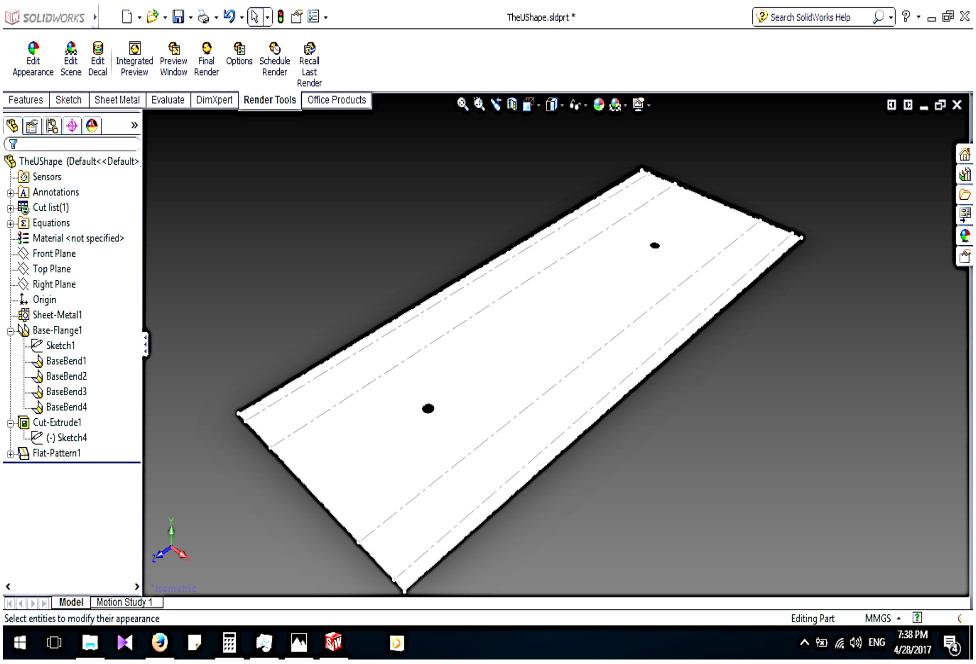


Fig. (4. 3) Steps of Creating strip layout

4-6 Computer software which used

The main advantage of computer aided design (CAD) of progressive die and machining is the ability to build precision tooling in less time and at lower cost. Integrating the part and the die design process with generation of cutter path information also greatly reduces the chance for error.The usage of computer aided design (CAD) software, computer aided engineering (CAE) software and computer aided manufacturing (CAM) software are recently the process of die design and making become easier and developed in all steps, the main factors which is developed is the time, cost and quality of products and dies, also the sheet metal fabrications become common and replaced many fabrications like cast, welding and rivet process.

As mentioned before the integration between all these software's and human creativity the sheet metal become dominantand was used in all types of industrials.

SolidWorks is currently used by over 2 million engineers [2] and designers at more than 165,000 companies worldwide. In 2011–2012, the fiscal revenue for SolidWorks was reported $483 million.

SolidWorks is a solid modeler, and utilizes a parametric feature-based approach to create models and assemblies.

4.7 Progressive die Modeling:

The designer used the solid woks program for design the mold part because SOLIDWORKS 3D CAD enables product designers and mold makers to easily incorporate design changes throughout the development process, right up to final manufacturing. (Show the Appendices (A, B and C).

Steps of Progressive die Modeling design as below:

1- Punch

Figure (4-4) show defines the Punch and Striper with a detail dimension. The purpose of the punch to provide the sheet metal with it is final shape.

Driving force for operating the punch is hydraulic with and average shaping force 25Mpa.

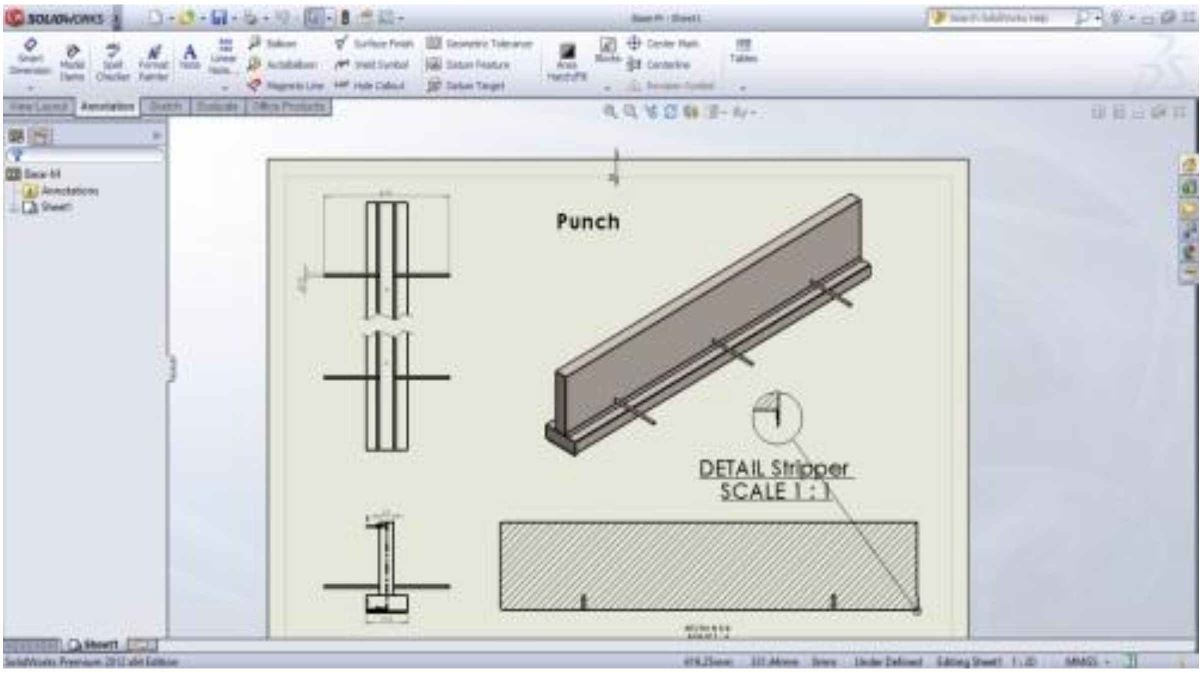


Fig. (4. 4) Punch model

2- Assembly of the Punch, die, Striper and U-shape:

Figure: (4-5) and Figure: (4-6), illustrate the assembly of the different part of Punch and die. Die material made of mild steel with yield strength 207 MPa, The marital Can Withstand frequently applied forces and can resist stress effectively. Additionally availability machinability and generally is low cost.

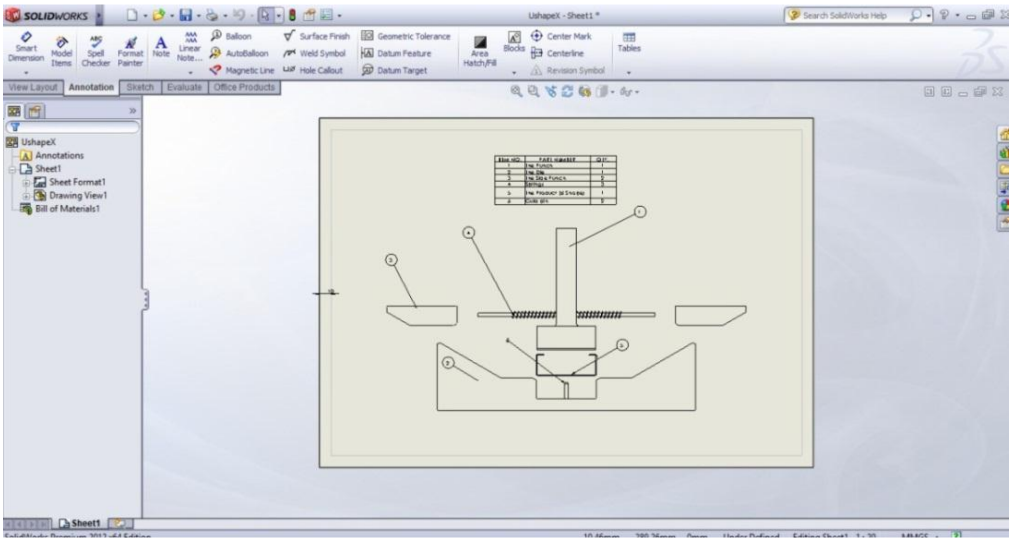


Fig. (4. 5) Assembly of the Punch, die, Striper and U-shape

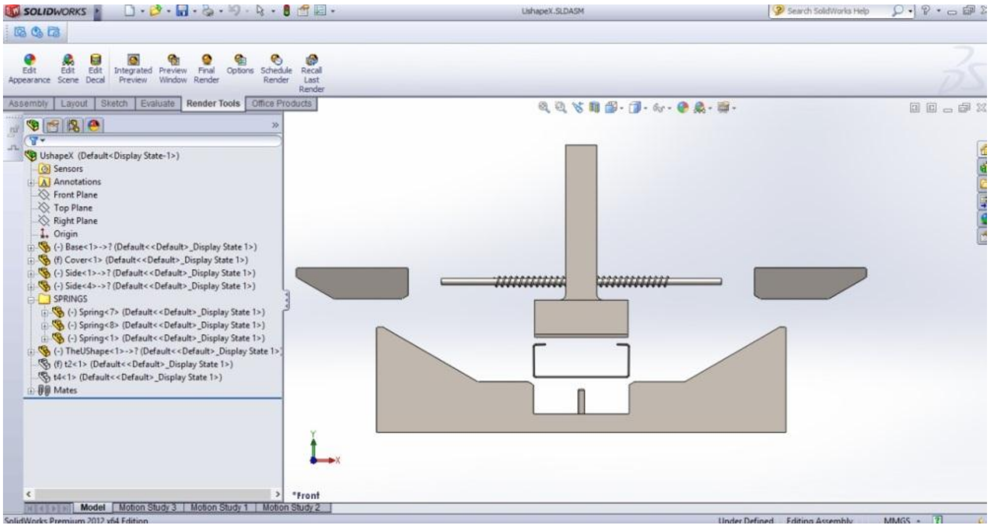


Fig. (4. 6) Exploded view of different dies and punch parts

3- Final Progressive die mold:

The figure (4-7) shows the rendering of the final assembly of the different parts of the progressive die which in names are: (punch striper and die base) in addition to the intended part U-shaped.

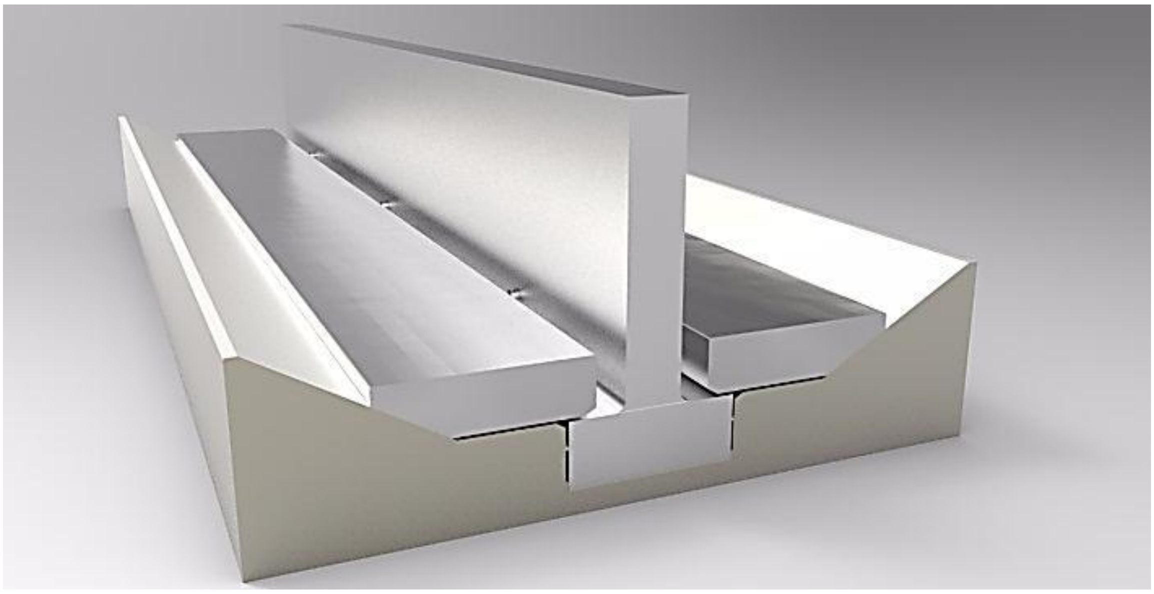


Fig. (4. 7) Progressive die assembly

CHAPTER FIVE

CONCLUSION AND RECOMMENDATION

5.1 Conclusion:

At the end of this research the following can be concluded:-

- The identification of new method for the production of sheet metal products by means of dies, especially (Progressive Die), also identifies the Die and how to design.
- For the completion of the design software (solid work) was used , a program dedicated to the design of moulds for plastic products moulds and sheet metal work has been accomplished with high quality and well-thought-out parts are grouped with each other and control tolerance.
- The difference is clear between the current production by the five dies and progressive die proposed by the study and that is to reduce the number of:
    - ❖ Cost factors.
    - ❖ Time factors.
    - ❖ Developed Quality control.
    - ❖ Mass production in a short time.

  These features are compared with the process as a whole at the time of utilization of the five stages. Therefore, it is recommended the use of progressive die for quantitative production.

- The Progressive die is expensive in the design and classified costly, needs experienced designer and computer software to assist the design process, analysis, and manufacture of special machines. Therefore, the choice of production in this way must be sensitive to the production of quantitative and high-definition.

- The use of standard parts reduces manufacturing time and increases the quality as well as to the age of the dies is therefore recommended amounts provided in the implementation of all designed Dies.

5.2 RECOMMENDATIONS:

It is recommended to:

Implement the (case study) design and compare the results to the desire specification.

Prepare cost analysis model in separate study.

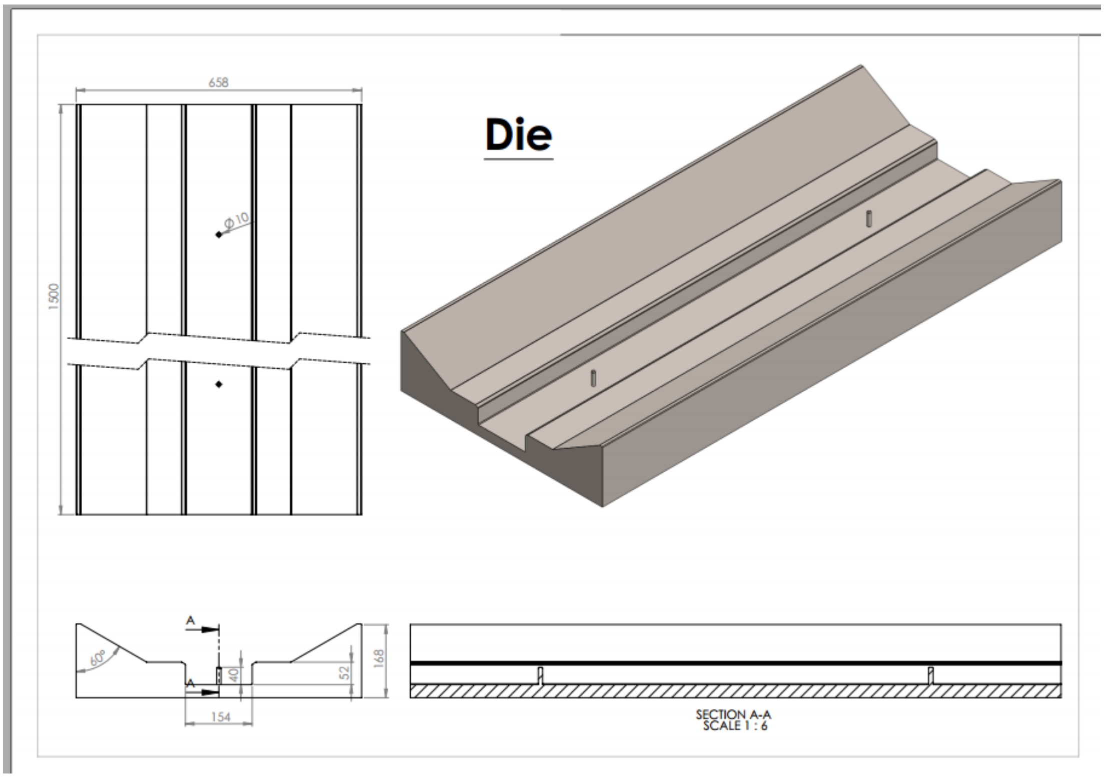
The study was limited to piercing, blanking and bending operations, thus it is recommend continuing on research including other methods.

REFERENCES:

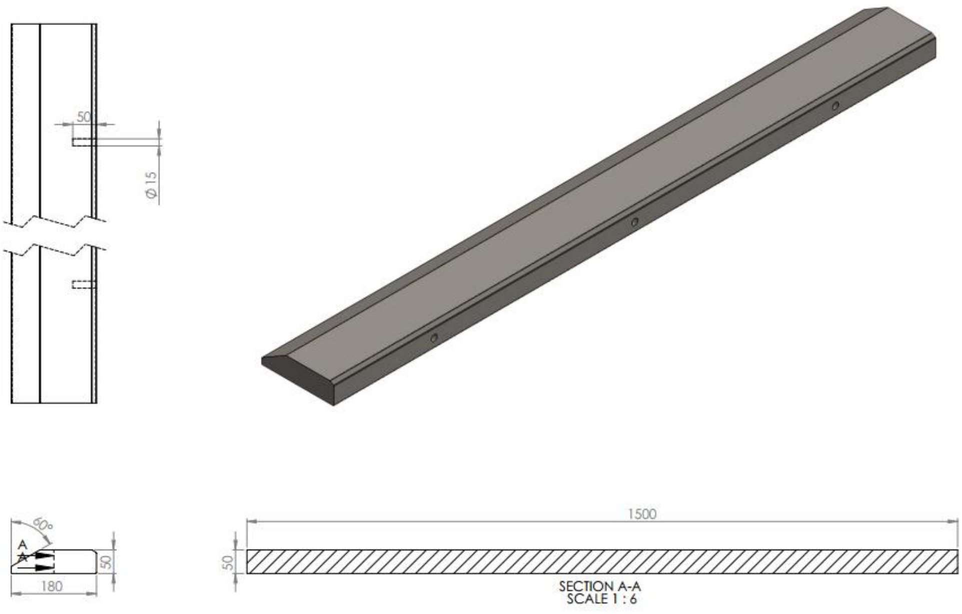
1. IVANA SUCHY, HANDBOOK OF DIE DESIGN, McGraw-Hill (USA) 1997.
2. DAVID A. SMITH, DIE DESING HANDBOOK (Third Edition), McGraw-Hill (USA) 1990 BY (SME).
3. J.R.RAQUIN, R. E. CROWLEY, DIE DESIGN FUNDEMENTALS (Second Edition), McGraw-Hill (USA) New York 1997.
4. MISUMI (Standard components for press dies), catalog, Edition 2007.
5. Ch. Mastanamma1, K. Prasad Rao2, Dr. M. Venkateshwara Rao3, " Design and Analysis of Progressive die", Vol. 1, Issue 6, Aug. 2012 IJERA.
6. Seiji Nakahara, Toshio Kojima, Kimio Tamura, Fumio Asuke, Choichiro Soda, Tatsuya Nakamura" Computer Aided Progressive Die Design" Proceedings of the Nineteenth International Machine Tool Design and Research Conference, pp 171-176.
7. https://www.google.ch/patents/US7716019

Appendixes

(A)

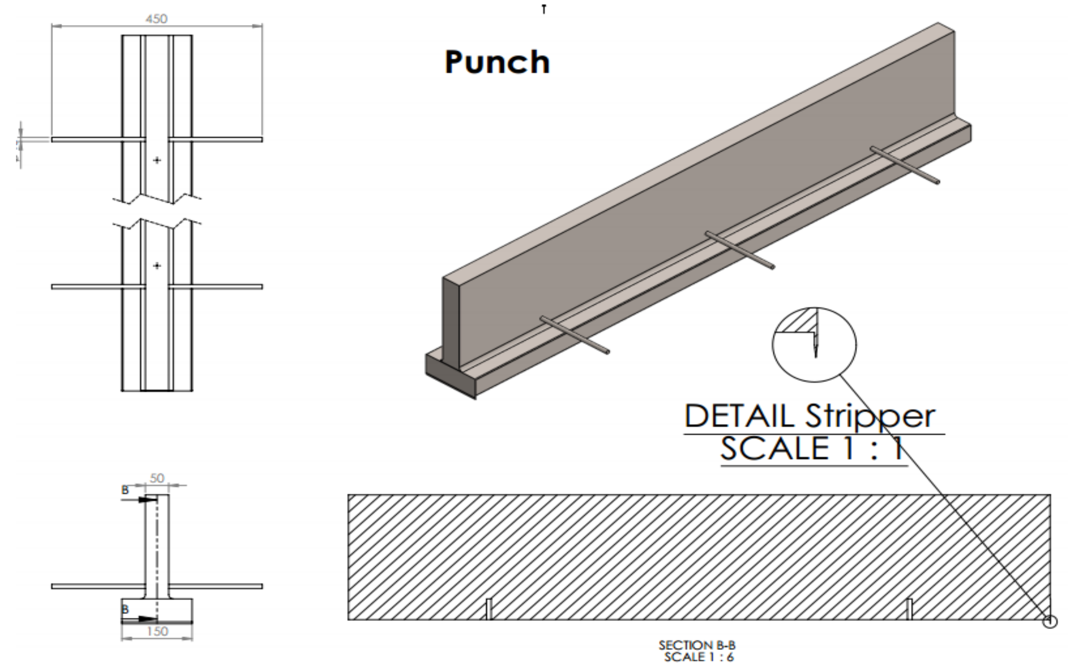


658
Die
Ø10
1500
A
A
60°
40
52
168
154
SECTION A-A
SCALE 1 : 6

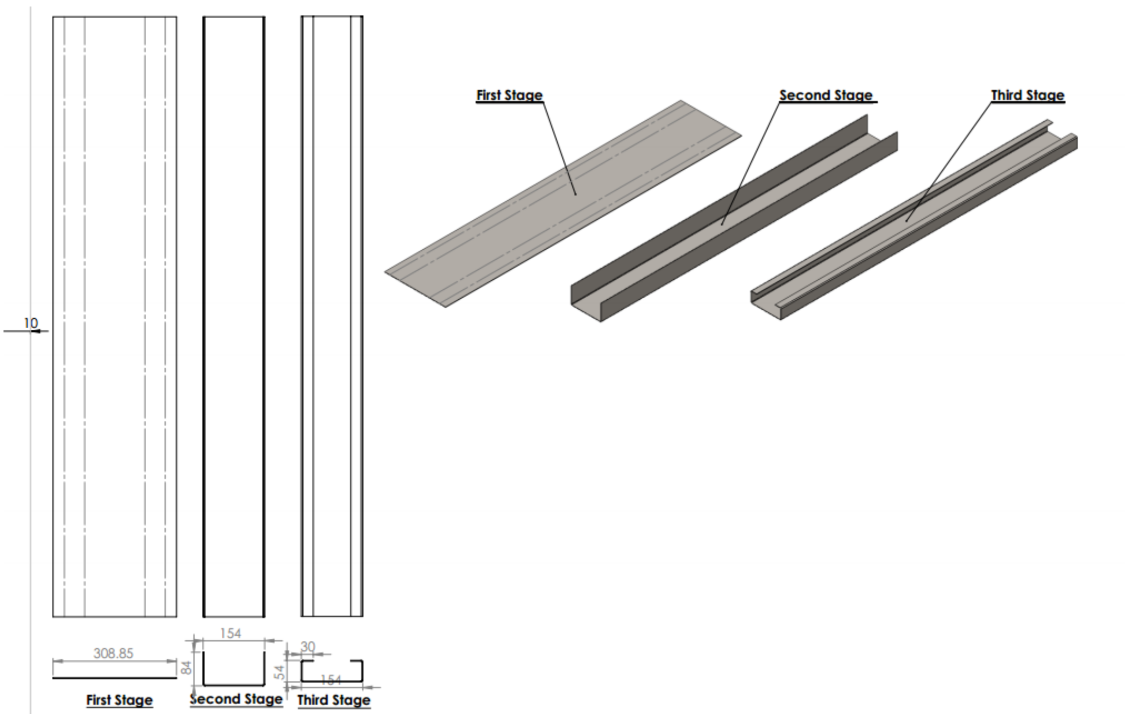


50
Ø15
60°
A
A
50
180
1500
50
SECTION A-A
SCALE 1 : 6

(B)



450
Punch
50
B
B
150
DETAIL Stripper
SCALE 1 : 1
SECTION B-B
SCALE 1 : 6



10
308.85
First Stage
154
84
Second Stage
30
54
154
Third Stage
First Stage
Second Stage
Third Stage

(C)

ITEM NO.	PART NUMBER	QTY.
1	The Punch	1
2	The Die	1
3	The Side Punch	2
4	Springs	3
5	The Product (U Shape)	1
6	Guid pin	2