

Unit 1 Basics of Tool Engineering

Introduction

The process of metal removal, a process in which a wedge-shaped tool engages a workpiece to remove a layer of material in the form of a chip, goes back many years.

Even with all the sophisticated equipment and techniques used in today's modern industry, the basic mechanics of forming a chip remain the same.

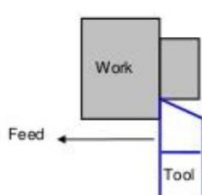
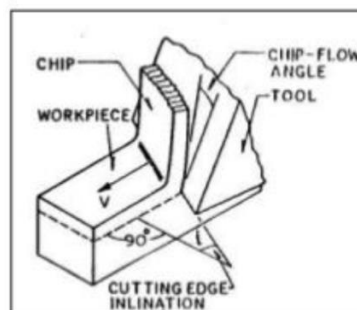
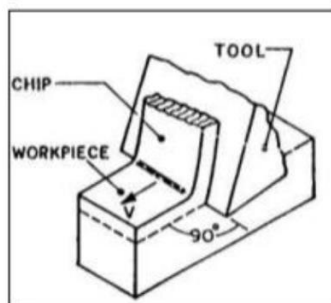
As the cutting tool engages the workpiece, the material directly ahead of the tool is sheared and deformed under tremendous pressure. The deformed material then seeks to relieve its stressed condition by fracturing and flowing into the space above the tool in the form of a chip.

Most of the engineering components such as gears, bearings, clutches, tools, screws and nuts etc. need dimensional and form accuracy and good surface finish for serving their purposes. Preforming like casting, forging etc. generally cannot provide the desired accuracy and finish. For that such preformed parts, called blanks, need semi-finishing and finishing and it is done by machining and grinding. Grinding is also basically a machining process. Machining to high accuracy and finish essentially enables a product

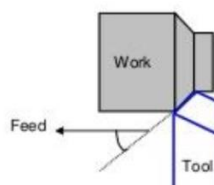
- fulfil its functional requirements
- improve its performance
- Prolong its service.

Methods of Machining

Basically, there are two methods of metal cutting depending upon the arrangement of the cutting edge with respect to the direction of the relative work tool motion. a. Orthogonal Cutting or Two Dimensional Cutting. b. Oblique cutting or three dimensional cutting.



a. Orthogonal Cutting



b. Oblique Cutting

Orthogonal Cutting:

- The cutting edge of the tool remains normal to the direction of tool feed or work feed.
- The direction of the chip flow velocity is normal to the cutting edge of the tool.
- Here only two components of forces are acting: Cutting Force and Thrust Force. So, the metal cutting may be considered as a two dimensional cutting.

Oblique Cutting:

- The cutting edge of the tool remains inclined at an acute angle to the direction of tool feed or work feed.
- The direction of the chip flow velocity is at an angle with the normal to the cutting edge of the tool. The angle is known as chip flow angle.
- Here three components of forces are acting: Cutting Force, Radial force and Thrust Force or feed force. So the metal cutting may be considered as a three dimensional cutting.
- The cutting edge being oblique, the shear force acts on a larger area and thus tool life is increased.

Cutting Tool and its requirements

Cutting tool is a device, used to remove the unwanted material from given workpiece. For carrying out the machining process, cutting tool is fundamental and essential requirement. A cutting tool must have the following characteristics:

- Hardness:** The tool material must be harder than the work piece material. Higher the hardness, easier it is for the tool to penetrate the work material.
- Hot hardness:** Hot Hardness is the ability of the cutting tool must to maintain its Hardness and strength at elevated temperatures. This property is more important when the tool is used at higher cutting speeds, for increased productivity.
- Toughness:** It should have enough toughness to withstand the impact loads that come in the start of the cut to force fluctuations due to imperfections in the work material. Toughness of cutting tools is needed so that tools don't chip or fracture, especially during interrupted cutting operations like milling
- Wear Resistance:** The tool-chip and chip-work interface are exposed to severe conditions that adhesive and abrasion wear is very common. Wear resistance means the attainment of acceptable tool life before tools need to be replaced.
- Low friction:** The coefficient of friction between the tool and chip should be low. This would lower wear rates and allow better chip flow.

•Thermal characteristics: Since a lot of heat is generated at the cutting zone, the tool material should have higher thermal conductivity to dissipate the heat in shortest possible time, otherwise the tool temperature would become high, reducing its life.

Types of cutting tools

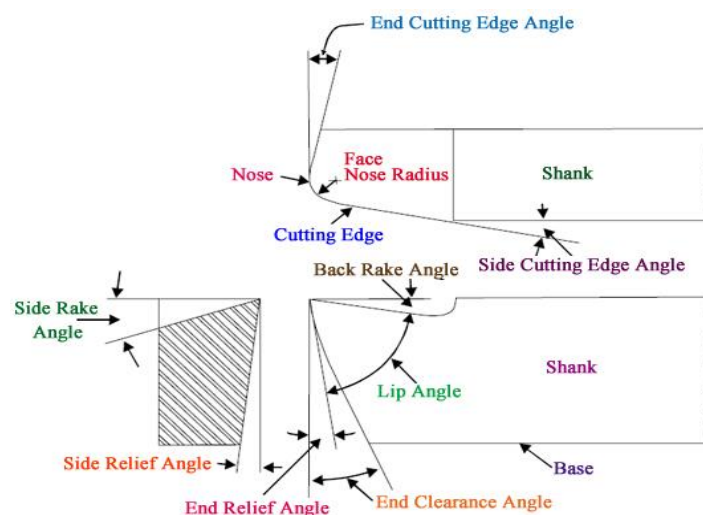
Cutting tools may be classified according to the number of major cutting edges (points) involved as follows:

Single point: e.g., turning tools, shaping, planning and slotting tools and boring tools.

Double (two) point: e.g., drills.

Multipoint (more than two): e.g., milling cutters, broaching tools, hobs, gear shaping cutters etc

Geometry of single point cutting tool



1. Shank: It is that part of single point cutting tool which goes into the tool holder. Or in simple language shank is used to hold the tool.

2. Flank: It is the surface below and adjacent of the cutting edges. There are two flank surfaces, first one is major flank and second one is minor flank. The major flank lies below and adjacent to the side cutting edge and the minor flank surface lies below and adjacent to the end cutting edge.

3. Base: The portion of the shank that lies opposite to the top face of the shank is called base.

4. Face: It is the top portion of the tool along which chips slides. It is designed in such a way that the chips slides on it in upward direction.

5. Cutting edge: The edge on the tool which removes materials from the work piece is called cutting edges. It lies on the face of the tool. The single point cutting tool has two edges and these are

- (i) Side cutting edge: The top edge of the major flank is called side cutting edge.
- (ii) End cutting edge: The top edge of the minor flank is called end cutting edge.

6. Nose or cutting point: The intersection point of major cutting edge and minor cutting edge is called nose.

7. Nose radius: It is the radius of the nose. Nose radius increases the life of the tool and provides better surface finish.

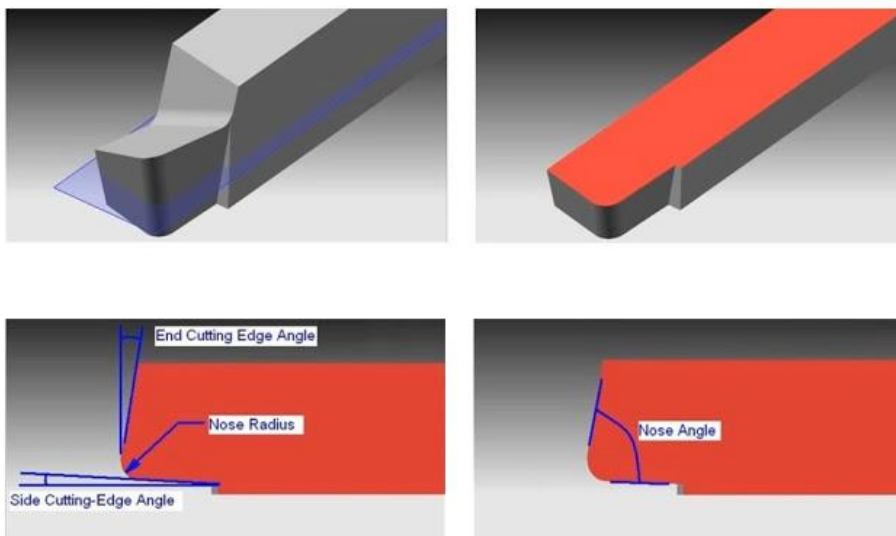
8. Heel: It is a curved portion and intersection of the base and flank of the tool.

Angles of Single Point Cutting Tool

The various angles of the single point cutting tool have great importance. Each angle has its own function and speciality.

1. End Cutting Edge Angle: The angle formed in between the end cutting edge and a line perpendicular to the shank is called end cutting edge angle.

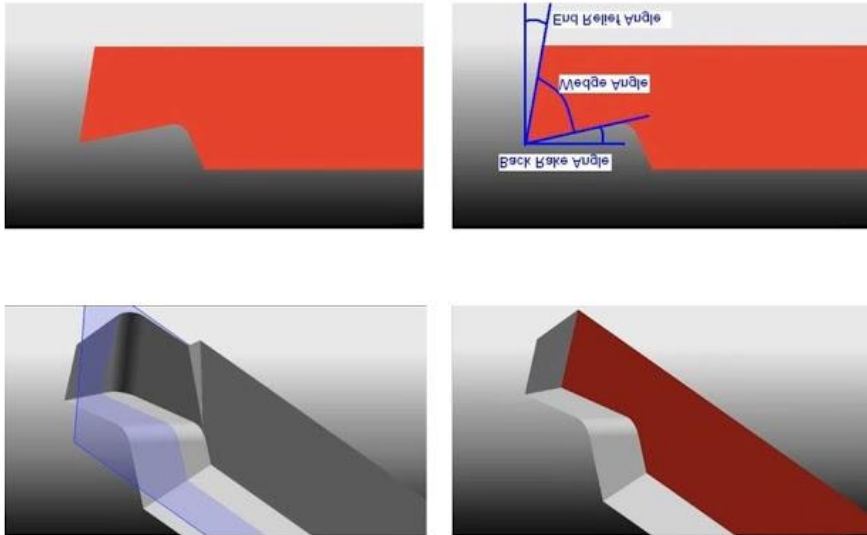
2. Side Cutting Edge Angle: The angle formed in between the side cutting edge and a line parallel to the shank.



3. Back Rack Angle: The angle formed between the tool face and line parallel to the base is called back rake angle.

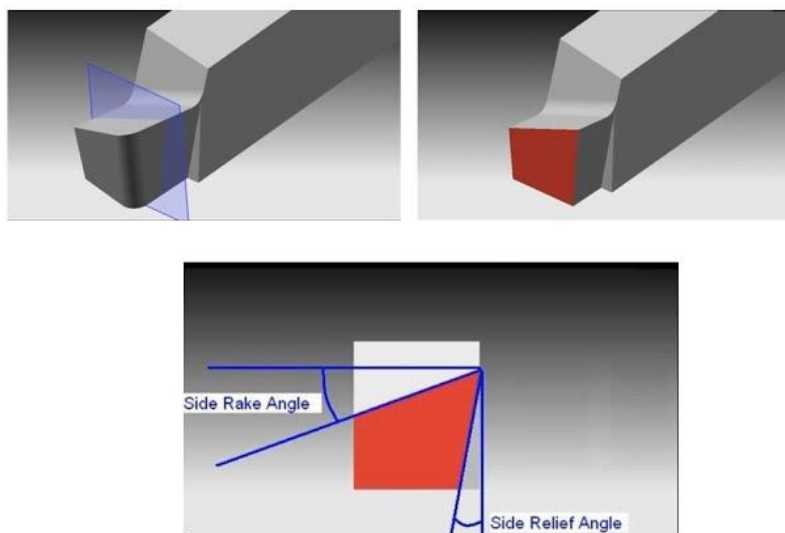
4. End Relief Angle: The angle formed between the minor flank and a line normal to the base of the tool is called end relief angle. It is also known as front clearance angle. It avoid the rubbing of the workpiece against tool.

5. Lip Angle/ Wedge Angle: It is defined as the angle between face and minor flank of the single point cutting tool.



6. Side Rake Angle: the angle formed between the tool face and a line perpendicular to the shank is called side rake angle.

7. Side Relief Angle: the angle formed between the major flank surface and plane normal to the base of the tool is called side relief angle. This angle avoids the rubbing between workpiece and flank when the tool is fed longitudinally.



Signature

The shape of a tool is specified in a special sequence and this special sequence is called tool signature. The tool signature is given below

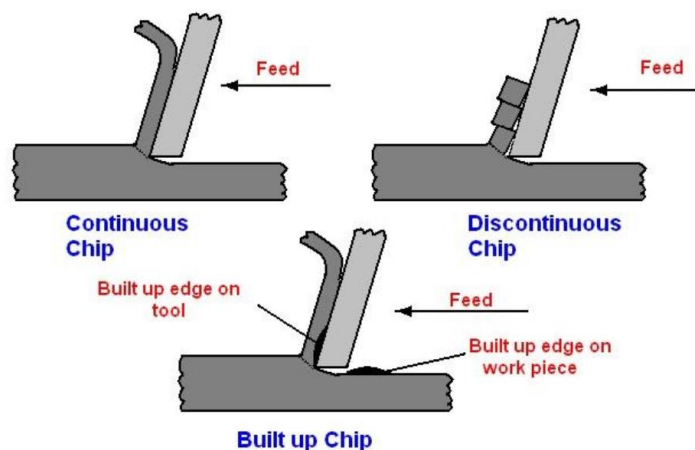
- (i) Back rake angle
- (ii) Side rake angle
- (iii) Clearance or End Relief angle
- (iv) Side Relief angle
- (v) End cutting edge angle
- (vi) Side cutting edge angle
- (vii) Nose radius

A typical tool signature of single point cutting tool is 0-7-6-8-15-16-0.8. Here this tool signature indicates that the tool has 0, 7, 6, 8, 15, 16 degree back rake, side rake, end relief, side relief, end cutting edge, side cutting edge angle and 0.8 mm nose radius.

Types of Chips

Whatever the cutting conditions can be, the chips produced may belong to one of the following three types.

1. Discontinuous chips:
2. Continuous chips.
3. Continuous chips with build up edge (BUE)



Discontinuous Chips

These types of chips are usually produced when cutting more brittle materials like grey cast iron, bronze and hard brass. These materials lack the ductility necessary for appreciable plastic chip formation. The material ahead of the tool edge fails in a brittle fracture manner along the shear zone. This produces small fragments of discontinuous chips. Since the chips break up into small segments, the friction between the tool and the chips reduces, resulting in better surface finish. These chips are convenient to collect, handle and dispose off.

Discontinuous chips are also produced when cutting more ductile materials under the following conditions:

- (i) large chip thickness.
- (ii) low cutting speed.
- (iii) cutting with the use of a cutting fluid.
- (iv) small rake angle of the tool.

Continuous Chips.

These types of chips are produced when machining more ductile materials. Due to large plastic deformations possible with ductile materials, longer continuous chips are produced. This type of chip is the most desirable, since it is stable cutting, resulting in generally good surface finish. On the other hand, these chips are difficult to handle and dispose off. The chips coil in a helix (chip curl) and curl around the work and the tool and may injure the operator when break loose. Also, this type of chip remains in contact with the tool face for a longer period, resulting in more frictional heat. These difficulties are usually avoided by attaching to the tool face or machine on the tool face, a 'chip breaker', The function of chip breaker is to reduce 1 curvature of the chip and thus break it.

The following cutting conditions also help in the production of continuous chips

- Small chip thickness.

- High cutting speed.

- Large rake angle of the cutting tool.

- Reducing the friction of the chip along the tool face, by: imparting high surface finish to the tool face, use of tool material with low co-efficient of friction, and use of good cutting fluid

Continuous chips with built up edge (BUE). –

When machining ductile materials, conditions of high local temperature and extreme pressure in the cutting zone and also high friction in the tool-chip interface, may cause the work material to adhere or weld to the cutting edge of then tool forming the built-up edge.

-Successive layers of work material are then added to the built up edge. When this edge becomes larger and unstable, it breaks up and part of it is carried up the face of the tool along with the chip while the remaining is left over the surface being machined, which contributes to the roughness of the surface.

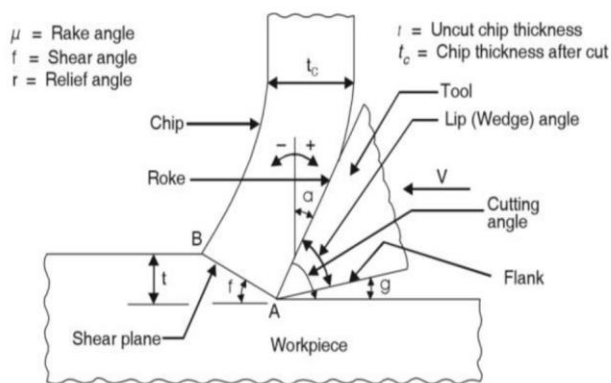
-The built-up edge changes its size during the cutting operation. It first increases, then decreases, then again increases etc. This cycle is a source of vibration and poor surface finish.

-Although, the built-up edge protects the cutting edge of the tool, changes the geometry of the cutting tool. Low cutting speed also contributes to the formation of the built-up edge.

-Increasing the cutting speed, increasing the rake angle and using a cutting fluid contribute to the reduction or elimination of the built-up edge.

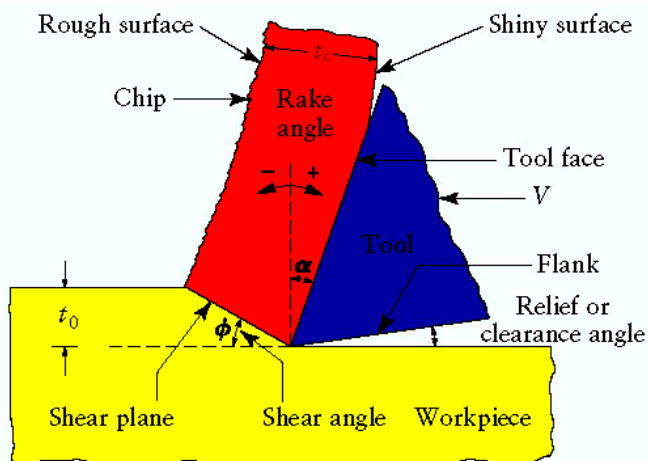
Mechanism of Chip Removal / Metal Cutting

A typical Metal cutting process can be schematically represented in Figure in which a wedge shaped tool is made to move relative to the work piece. As the tool makes contact with the metal; it exerts a pressure on it resulting in the compression of the metal near to the tool tip.



This includes shear type deformation with in the metal and it starts moving upward along the top face of the tool, as the tool advances, the material ahead of it is sheared continuously along a plane called the shear plane. This shear plane actually narrow zone. (About 0.025 mm) and extends from the cutting edge of the tool is formed by two intersecting surfaces. The surface along which the chip moves upwards is called “Rake Surface” and the other surface which is relieved to avoid rubbing with the machined surface is called “flank”. The angle between rake surface and the normal is known as “Rake Angle” (which may be positive or negative) and the angle between the flank and the horizontal machined surface is known as the “relief or clearance angle”.

Chip thickness ratio



The ratio of thickness of uncut chip to thickness of chip after cut is called chip thickness ratio.

The outward flow of the metal causes the chip to be thicker after the separation from the parent metal. That is the chip produced is thicker than the depth of cut.

The geometry of the chips being formed at the cutting zone follow a particular pattern especially in machining ductile materials. The major sections of the engineering materials being machined are ductile in nature; even some semi-ductile or semi-brittle materials behave ductile under the compressive forces at the cutting zone during machining. The pattern and degree of deformation during chip formation are quantitatively assessed and expressed by some factors, the values of which indicate about the forces and energy required for a particular machining work.

$$r = \frac{t_0}{t_c}$$

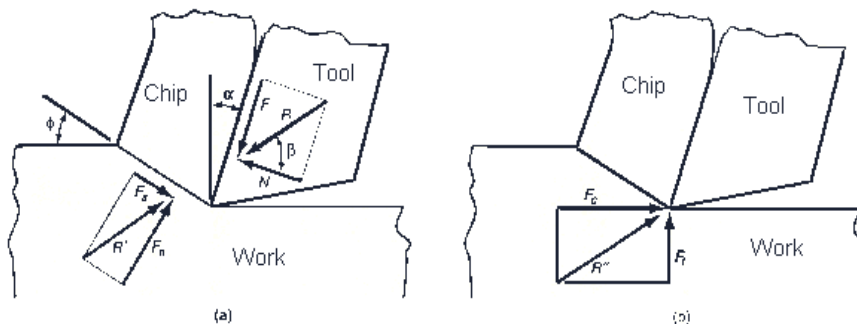
Cutting forces

Shear plane

Shear plane is the plane of separation of work material layer in the form of chip from the parent body due to shear along that plane.

Shear angle

Angle of inclination of the shear plane from the direction of cutting velocity



F_s = Shear Force, which acts along the shear plane, is the resistance to shear of the metal in forming the chip.

F_n = Force acting normal to the shear plane, is the backing up force on the chip provided by the workpiece.

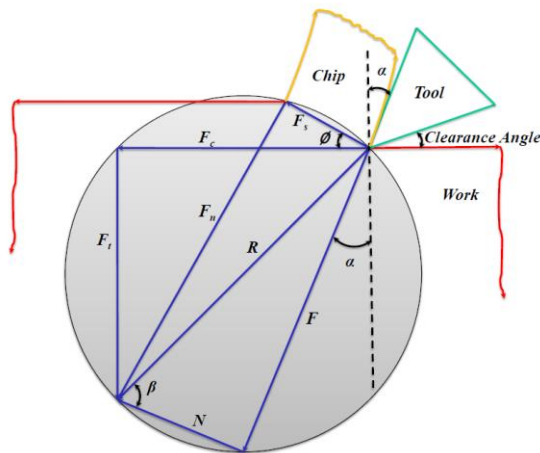
F = Frictional resistance of the tool acting against the motion of the chip as it moves upward along the tool.

N = Normal to the chip force, is provided by the tool.

It is assumed that the resultant forces R & R' are equal and opposite in magnitude and direction. Also they are Collinear. Therefore for the purpose of analysis the chip is regarded

as an independent body held in mechanical equilibrium by the action of two equal and opposite forces R , which the workpiece exerts upon the chip and R^1 which the tool exerts upon the chip.

Merchant's Circle



The following is a circle diagram. Known as Merchant's circle diagram, which is convenient to determine the relation between the various forces and angles. In the diagram two force triangles have been combined and R and R' together have been replaced by R . The force R can be resolved into two components F_c and F_t .

F_c and F_t can be determined by force dynamometers.

$$\vec{R} = \vec{F}_c + \vec{F}_t$$

The rake angle (α) can be measured from the tool, and forces F and N can then be determined. The shear angle (ϕ) can be obtained from its relation with chip reduction coefficient. Now F_s & F_n can also be determined.

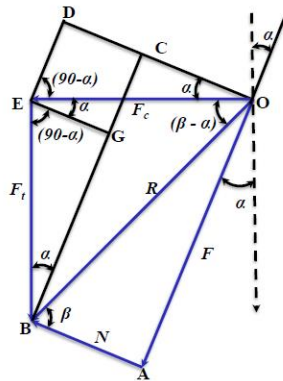
Procedure to Draw Merchants circle

Set up x-y axis labeled with forces, and the origin in the centre of the page. The cutting force (F_c) is drawn horizontally, and the tangential force (F_t) is drawn vertically. (Draw in the resultant (R) of F_c and F_t).

- Locate the centre of R , and draw a circle that encloses vector R . If done correctly, the heads and tails of all 3 vectors will lie on this circle.
- Draw in the cutting tool in the upper right hand quadrant, taking care to draw the correct rake angle (α) from the vertical axis.
- Extend the line that is the cutting face of the tool (at the same rake angle) through the circle. This now gives the friction vector (F).
- A line can now be drawn from the head of the friction vector, to the head of the resultant vector (R). This gives the normal vector (N). Also add a friction angle (β) between vectors R and N . Therefore, mathematically, $R = F_c + F_t = F + N$.
- Draw a feed thickness line parallel to the horizontal axis. Next draw a chip thickness line parallel to the tool cutting face.

- Draw a vector from the origin (tool point) towards the intersection of the two chip lines, stopping at the circle. The result will be a shear force vector (Fs). Also measure the shear force angle between Fs and Fc.
- Finally add the shear force normal (Fn) from the head of Fs to the head of R.
- Use a scale and protractor to measure off all distances (forces) and angles.

Frictional Force System

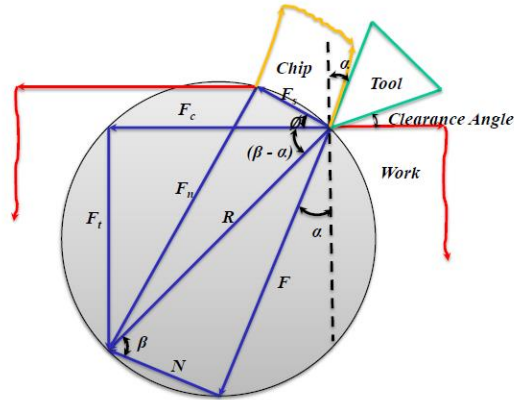


$$F = OA = CB = CG + GB = ED + GB$$

$$\Rightarrow F = F_c \sin \alpha + F_t \cos \alpha$$

$$N = AB = OD - CD = OD - GE$$

$$\Rightarrow N = F_c \cos \alpha - F_t \sin \alpha$$

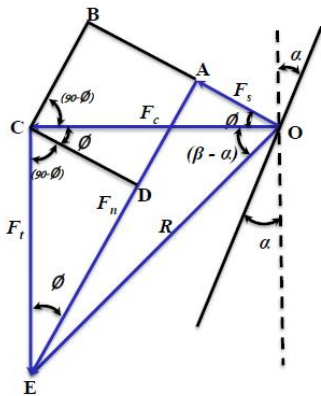


The coefficient of friction

$$\mu = \tan \beta = \frac{F}{N}$$

Where β = Friction angle

Shear Force System

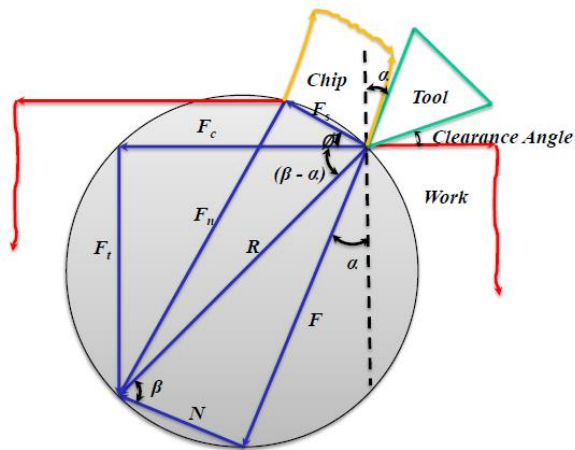


$$F_s = OA = OB - AB = OB - CD$$

$$\Rightarrow F_s = F_c \cos \phi - F_t \sin \phi$$

$$F_n = AE = AD + DE = BC + DE$$

$$\Rightarrow F_n = F_c \sin \phi + F_t \cos \phi$$



Also:

$$F_n = F_s \tan(\phi + \beta - \alpha)$$

$$F = F_C \sin \alpha + F_t \cos \alpha$$

$$N = F_C \cos \alpha - F_t \sin \alpha$$

$$F_S = F_C \cos \phi - F_t \sin \phi$$

$$F_N = F_C \sin \phi + F_t \cos \phi$$

$$F_N = F_S \tan(\phi + \beta - \alpha)$$

Unit 2: Cutting tool Material and Holding devices

Introduction

- The selection of cutting-tool materials for a particular application is among the most important factors in machining operations.
- The complex nature of this subject forces us to rely on guidelines and recommendations that have been accumulated in industry over many years.
- the cutting tool is subjected to:
 - a) High temperatures,
 - b) High contact stresses, and
 - c) Rubbing along the tool-chip interface and along the machined surface.

Carbon steels:

Carbon steels contain carbon in amounts ranging from 0.08 to 1.5 per cent. A disadvantage of carbon tool steels is their comparatively low heat and wears resistance. They lose their required hardness at temperature from 200 to 250 degree. Therefore, they may only be used in the manufacture of tools operating at low cutting speeds and of hand operated tools. But they are comparatively cheap easy to forge and simple to harden.

Medium Alloy steels:

The high carbon medium alloy steels have a carbon content into plain carbon steels, but in addition there is 5% alloy content consisting of tungsten, molybdenum, chromium and vanadium. Small additions of one or more elements improve the performance of the carbon steels in respect of hot hardness, wear resistance, shock and impact resistance and resistance to distortion during heat treatment. Therefore alloy carbon steels, occupy a midway performance position between plain carbon steels and high speed steels.

High Speed Steel (HSS)

Composition

The basic composition of HSS is 18% W, 4% Cr, 1% V, 0.7% C and rest Fe. Such HSS tool could machine (turn) mild steel jobs at speed only upto 20 ~ 30 m/min

Application

- The tool geometry and mechanics of chip formation are complex, such as helical twist drills, reamers, gear shaping cutters, hobs, form tools, broaches etc.
- Brittle tools like carbides, ceramics etc. are not suitable under shock loading
- The small scale industries cannot afford costlier tools
- The old or low powered small machine tools cannot accept high speed and feed.
- The tool is to be used number of times by resharpening.

Stellite

This is a cast alloy of Co (40 to 50%), Cr (27 to 32%), W (14 to 19%) and C (2%). Stellite is quite tough and more heat and wear resistive than the basic HSS (18 – 4 – 1) But such stellite as cutting tool material became obsolete for its poor grindability and specially after the arrival of cemented carbides.

Carbides (cemented or sintered carbides)

The proportion of cobalt (the usual matrix material) present has a significant effect on the properties of carbide tools. 3 - 6% matrix of cobalt gives greater hardness while 6 - 15% matrix of cobalt gives a greater toughness while decreasing the hardness, wear resistance and strength. Tungsten carbide tools are commonly used for machining steels, cast irons and abrasive non-ferrous materials.

- The two groups of tool materials described above cannot be used as effectively where high cutting speeds (High temp.) are involved.
- Introduced in the 1930's and have high hardness over a wide range of temperatures, so can be used for higher cutting speeds.
- High elastic modulus and thermal conductivity, and low thermal expansion.
- The most important, versatile, and cost-effective tool and die materials for a wide range of applications.
- Two major groups of carbides used for machining operations are tungsten carbides and titanium carbide.

Coated carbides

The properties and performance of carbide tools could be substantially improved by

- Refining microstructure
- Manufacturing by casting – expensive and uncommon
- Surface coating – made remarkable contribution.

Thin but hard coating of single or multilayers of more stable and heat and wear resistive materials like TiC, TiCN, TiOCN, TiN, Al₂O₃ etc on the tough carbide inserts (substrate) by processes like chemical Vapour Deposition (CVD), Physical Vapour Deposition (PVD) etc at controlled pressure and temperature enhanced MRR and overall machining economy remarkably enabling, • reduction of cutting forces and power consumption • increase in tool life (by 200 to 500%) for same VC or increase in VC (by 50 to 150%) for same tool life

- improvement in product quality
- effective and efficient machining of wide range of work materials
- pollution control by less or no use of cutting fluid
- reduction of abrasion, adhesion and diffusion wear
- reduction of friction and BUE formation
- heat resistance and reduction of thermal cracking and plastic deformation

Ceramics

Alumina

Introduced in the early 1950s, two classes are used for cutting tools: fine grained high purity aluminium oxide (Al₂O₃) and silicon nitride (Si₃N₄) are pressed into insert tip shapes and sintered at high temperatures. Additions of titanium carbide and zirconium oxide (ZrO₂) may be made to improve properties. But while ZrO₂ improves the fracture toughness, it reduces the hardness and thermal conductivity. Silicon carbide (SiC) whiskers may be added to give better toughness and improved thermal shock resistance.

The tips have high abrasion resistance and hot hardness and their superior chemical stability compared to HSS and carbides means they are less likely to adhere to the metals during cutting and consequently have a lower tendency to form a built up edge. Their main weakness is low toughness and negative rake angles are often used to avoid chipping due to their low tensile strengths. Stiff machine tools and work set ups should be used when machining with ceramic tips as otherwise vibration is likely to lead to premature failure of the tip. Typical cutting speeds: 150 - 650 m/min.

Silicon Nitride

In the 1970s a tool material based on silicon nitride was developed, these may also contain aluminium oxide, yttrium oxide and titanium carbide. SiN has an affinity for iron and is not suitable for machining steels. A specific type is 'Sialon', containing the elements: silicon, aluminium, oxygen and nitrogen. This has higher thermal shock resistance than silicon nitride and is recommended for machining cast irons and nickel based superalloys at intermediate cutting speeds

Diamond.

Diamond is the hardest cutting tool material which it can be applied in either the monocrystalline or the polycrystalline state. Monocrystalline diamond is used for fine machining and polycrystalline diamond enables machining with higher cutting depths. The typical applications of diamond tools are the machining of nonferrous metals, plastics, and other nonmetal materials (e.g., hard rubber, graphite, ceramics, glass, stone, and asbestos).

In the automobile industry polycrystalline diamonds are increasingly applied for machining of Al–Si alloys. Due to the affinity of carbon for ferrous materials, diamond cannot be used to machine these metals.

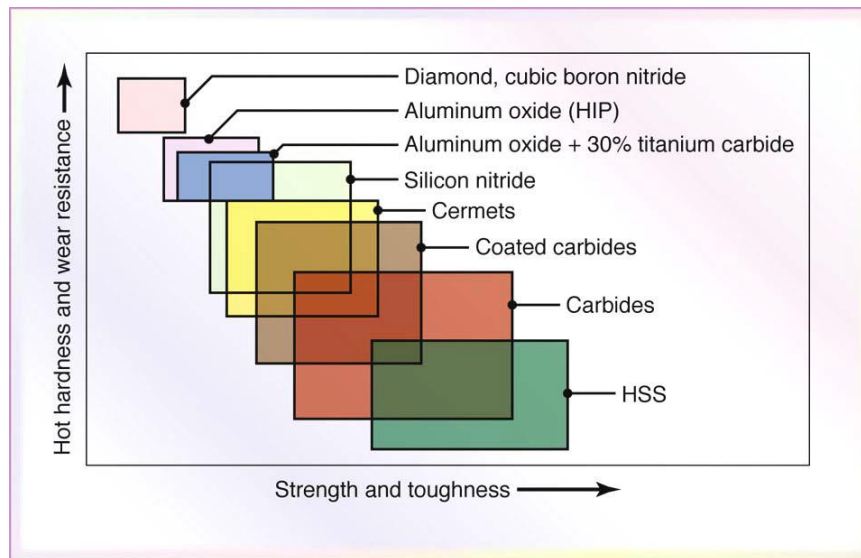
Cubic Boron nitride.

Next to diamond, cubical CBN is the hardest cutting tool material. It has a relatively high tensile strength and bending strength, a high thermal stability, and a high thermal resistance to iron. The superiority of CBN over other cutting tool materials such as hard metals and high-speed steels is obvious during the machining of steel at a hardness of 54–68 HRC or difficult-to-machine materials such as high-temperature cobalt and nickel-based alloys. By machining with CBN, surface qualities equivalent to those of fine machining processes can be realized.

Polycrystalline boron nitride (PCBN)

These superhard tool materials were developed mainly for applications where long tool life and high productivity are of main concern. Similar to ceramics, these materials are also made by sintering of the hard phase, provided in the form of micrograin particles or polycrystals, with or without a binder phase. The hard phase is synthetic diamond or cubic boron nitride grits that are created at high temperature and pressure. The binder phase, which could be ceramic or metallic, improves the toughness and manufacturability of the cutting tools. Sintering is performed at extremely high temperatures and high pressures to help consolidation and bonding of the tool material. This makes their production cost very high. PCBN tools are generally much harder than alumina-based ceramics and possess excellent

chemical wear resistance. But like ceramics, their toughness is poor. Therefore, they are more suited for finish turning of hardened materials and cast iron.



Classification of Carbides

The advent of sintered carbides made another breakthrough in the history of cutting tool materials.

Straight or single carbide

First the straight or single carbide tools or inserts were powder metallurgically produced by mixing, compacting and sintering 90 to 95% WC powder with cobalt. The hot, hard and wear resistant WC grains are held by the binder Co which provides the necessary strength and toughness. Such tools are suitable for machining grey cast iron, brass, bronze etc. which produce short discontinuous chips and at cutting velocities two to three times of that possible for HSS tools.

Composite carbides



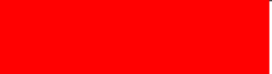
The single carbide is not suitable for machining steels because of rapid growth of wear, particularly crater wear, by diffusion of Co and carbon from the tool to the chip under the high stress and temperature bulk (plastic) contact between the continuous chip and the tool surfaces. For machining steels successfully, another type called composite carbide have been developed by adding (8 to 20%) a gamma phase to WC and Co mix. The gamma phase is a mix of TiC, TiN, TaC, NiC etc. which are more diffusion resistant than WC due to their more stability and less wettability by steel.

Mixed carbides Titanium carbide (TiC)

TiC is not only more stable but also much harder than WC. So for machining ferritic steels causing intensive diffusion and adhesion wear a large quantity (5 to

25%) of TiC is added with WC and Co to produce another grade called Mixed carbide. But increase in TiC content reduces the toughness of the tools. Therefore, for finishing with light cut but high speed, the harder grades containing upto 25% TiC are used and for heavy roughing work at lower speeds lesser amount (5 to 10%) of TiC is suitable.

Gradation of cemented carbides and their applications The standards developed by ISO for grouping of carbide tools and their application ranges are given in Table

ISO Code	Colour Code	Application
P		For machining long chip forming common materials like plain carbon and low alloy steels
M		For machining long or short chip forming ferrous materials like Stainless steel
K		For machining short chipping, ferrous and non-ferrous material and non-metals like Cast Iron, Brass etc.

K-group is suitable for machining short chip producing ferrous and non-ferrous metals and also some non metals.

P-group is suitably used for machining long chipping ferrous metals i.e. plain carbon and low alloy steels

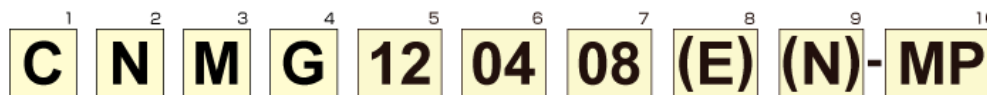
M-group is generally recommended for machining more difficult-to-machine materials like strain hardening austenitic steel and manganese steel etc.

Each group again is divided into some subgroups like P10, P20 etc., as shown in Table

ISO Application group	Material	Process
P01	Steel, Steel castings	Precision and finish machining, high speed
P10	Steel, steel castings	Turning, threading and milling high speed, small chips
P20	Steel, steel castings, malleable cast iron	Turning, milling, medium speed with small chip section
P30	Steel, steel castings, malleable cast iron forming long chips	Turning, milling, low cutting speed, large chip section
P40	Steel and steel casting with sand inclusions	Turning, planning, low cutting speed, large chip section
P50	Steel and steel castings of medium or low tensile strength	Operations requiring high toughness turning, planning, shapina at low cutting speeds
K01	Hard grey C.I., chilled casting, Al. alloys with high silicon	Turning, precision turning and boring, milling, scraping
K10	Grey C.I. hardness > 220 HB. Malleable C.I., Al. alloys containing Si	Turning, milling, boring, reaming, broaching, scraping
K20	Grey C.I. hardness up to 220 HB	Turning, milling, broaching, requiring high toughness
K30	Soft grey C.I. Low tensile strength steel	Turning, reaming under favourable conditions
K40	Soft non-ferrous metals	Turning milling etc.
M10	Steel, steel castings, manganese steel, grey C.I.	Turning at medium or high cutting speed, medium chip section
M20	Steel casting, austenitic steel, manganese steel, spherodized C.I., Malleable C.I.	Turning, milling, medium cutting speed and medium chip section
M30	Steel, austenitic steel, spherodized C.I. heat resisting alloys	Turning, milling, planning, medium cutting speed, medium or large chip section
M40	Free cutting steel, low tensile strength steel, brass and light alloy	Turning, profile turning, specially in automatic machines.

Turning Inserts Identification

1. Insert Shape
2. Relief Angle
3. Tolerance Class
4. Chipbreaker and Clamping System
5. Insert Size
6. Insert Corner Configuration
7. Insert Thickness
8. Cutting Edge Condition
9. Cutting Direction
10. Chip Breaker



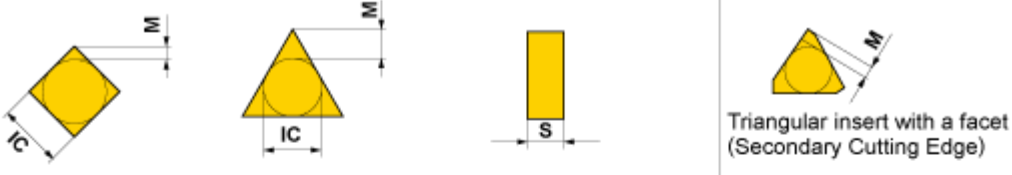
1. Insert Shape

1. Insert Shape		
Symbol	Insert Shape	
H	Hexagonal	
O	Octagonal	
P	Pentagonal	
S	Square	
T	Triangular	
C	Rhombic 80°	
D	Rhombic 55°	
E	Rhombic 75°	
F	Rhombic 50°	
M	Rhombic 86°	
V	Rhombic 35°	
W	Trigon	
L	Rectangular	
A	Parallelogram 85°	
B	Parallelogram 82°	
K	Parallelogram 55°	
R	Round	
X	Special Design	















2. Relief Angle

2. Relief Angle		
Symbol	Normal Clearance	
A	3°	
B	5°	
C	7°	
D	15°	
E	20°	
F	25°	
G	30°	
N	0°	
P	11°	
O	Other Relief Angle	
Major Relief Angle		








3. Tolerance Class

3. Tolerance Class										
			Triangular insert with a facet (Secondary Cutting Edge)							
3. Tolerance Class				Detail of M Class Insert Tolerance						
Symbol	Tolerance of Nose Height M (mm)	Tolerance of Inscribed Circle IC (mm)	Tolerance of Thickness S (mm)	●Tolerance of Nose Height M (mm)						
				D.I.C.	Triangular	Square	Rhombic 80°	Rhombic 55°	Rhombic 35°	Round
A	±0.005	±0.025	±0.025	6.35	±0.08	±0.08	±0.08	±0.11	±0.16	—
F	±0.005	±0.013	±0.025	9.525	±0.08	±0.08	±0.08	±0.11	±0.16	—
C	±0.013	±0.025	±0.025	12.70	±0.13	±0.13	±0.13	±0.15	—	—
H	±0.013	±0.013	±0.025	15.875	±0.15	±0.15	±0.15	±0.18	—	—
E	±0.025	±0.025	±0.025	19.05	±0.15	±0.15	±0.15	±0.18	—	—
G	±0.025	±0.025	±0.13	25.40	—	±0.18	—	—	—	—
J	±0.005	±0.05—±0.15	±0.025	31.75	—	±0.20	—	—	—	—
K*	±0.013	±0.05—±0.15	±0.025	●Tolerance of Inscribed Circle IC (mm)						
L*	±0.025	±0.05—±0.15	±0.025	D.I.C.	Triangular	Square	Rhombic 80°	Rhombic 55°	Rhombic 35°	Round
M*	±0.08—±0.18	±0.05—±0.15	±0.13	6.35	±0.05	±0.05	±0.05	±0.05	±0.05	—
N*	±0.08—±0.18	±0.05—±0.15	±0.025	9.525	±0.05	±0.05	±0.05	±0.05	±0.05	±0.05
U*	±0.13—±0.38	±0.08—±0.25	±0.13	12.70	±0.08	±0.08	±0.08	±0.08	—	±0.08
The surface of insert with * mark is sintered.				15.875	±0.10	±0.10	±0.10	±0.10	—	±0.10
				19.05	±0.10	±0.10	±0.10	±0.10	—	±0.10
				25.40	—	±0.13	—	—	—	±0.13
				31.75	—	±0.15	—	—	—	±0.15

4. Chipbreaker and Clamping System

4. Chipbreaker and Clamping System									
Metric									
Symbol	Hole	Hole Configuration	Chip Breaker	Figure	Symbol	Hole	Hole Configuration	Chip Breaker	Figure
W	With Hole	Cylindrical Hole + One Countersink (40–60°)	No		A	With Hole	Cylindrical Hole	No	
T	With Hole	Cylindrical Hole + One Countersink (40–60°)	One Sided		M	With Hole	Cylindrical Hole	Single Sided	
Q	With Hole	Cylindrical Hole + Double Countersink (40–60°)	No		G	With Hole	Cylindrical Hole	Double Sided	
U	With Hole	Cylindrical Hole + Double Countersink (40–60°)	Double Sided		N	Without Hole	—	No	
B	With Hole	Cylindrical Hole + One Countersink (70–90°)	No		R	Without Hole	—	Single Sided	
H	With Hole	Cylindrical Hole + One Countersink (70–90°)	One Sided		F	Without Hole	—	Double Sided	
C	With Hole	Cylindrical Hole + Double Countersink (70–90°)	No		X	—	—	—	Special Design
J	With Hole	Cylindrical Hole + Double Countersink (70–90°)	Double Sided						

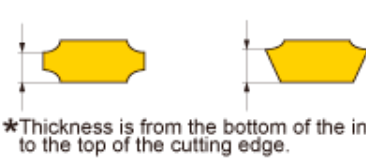
5. Insert Size

5. Insert Size							
Symbol							Diameter of Inscribed Circle (mm)
							
	02		04	03	03	06	3.97
	L3	08	05	04	04	08	4.76
	03	09	06	05	05	09	5.56
06							6.00
	04	11	07	06	06	11	6.35
	05	13	09	08	07	13	7.94
08							8.00
09	06	16	11	09	09	16	9.525
10							10.00
12							12.00
12	08	22	15	12	12	22	12.70
15	10		19	16	15	27	15.875
16							16.00
19	13		23	19	19	33	19.05
20							20.00
			27	22	22	38	22.225
25							25.00
25			31	25	25	44	25.40
31			38	32	31	54	31.75
32							32.00





7. Insert Corner Configuration

7. Insert Corner Configuration	
Symbol	Corner Radius (mm)
00	Sharp Nose
V3	0.03
V5	0.05
01	0.1
02	0.2
04	0.4
08	0.8
12	1.2
16	1.6
20	2.0
24	2.4
28	2.8
32	3.2
00 : Inch M0 : Metric	Round Insert




6. Insert Thickness

6. Insert Thickness	
	
Symbol	Thickness (mm)
S1	1.39
01	1.59
T0	1.79
02	2.38
T2	2.78
03	3.18
T3	3.97
04	4.76
06	6.35
07	7.94
09	9.52

8. Cutting Edge Condition

8. Cutting Edge Condition		
Figure	Cutting Edge	Symbol
	Sharp Cutting Edges	F
	Round Cutting Edges	E
	Chamfered Cutting Edges	T
	Chamfered and Rounded Cutting Edges	S
Mitsubishi Materials omit the honing symbol.		

9. Cutting Direction

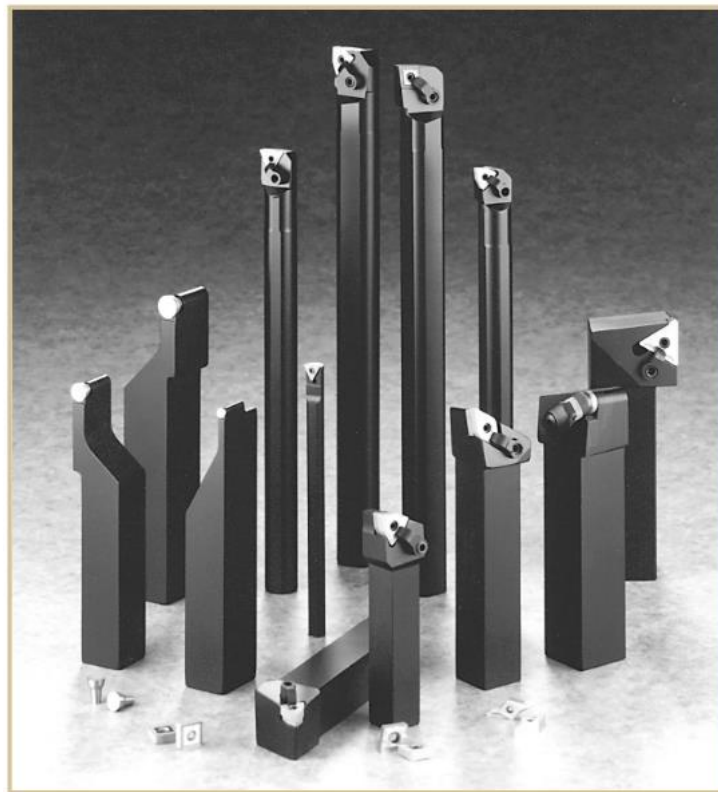
9. Cutting Direction		
Figure	Hand	Symbol
	Right	R
	Left	L
	Neutral	N

10. Chip Breaker

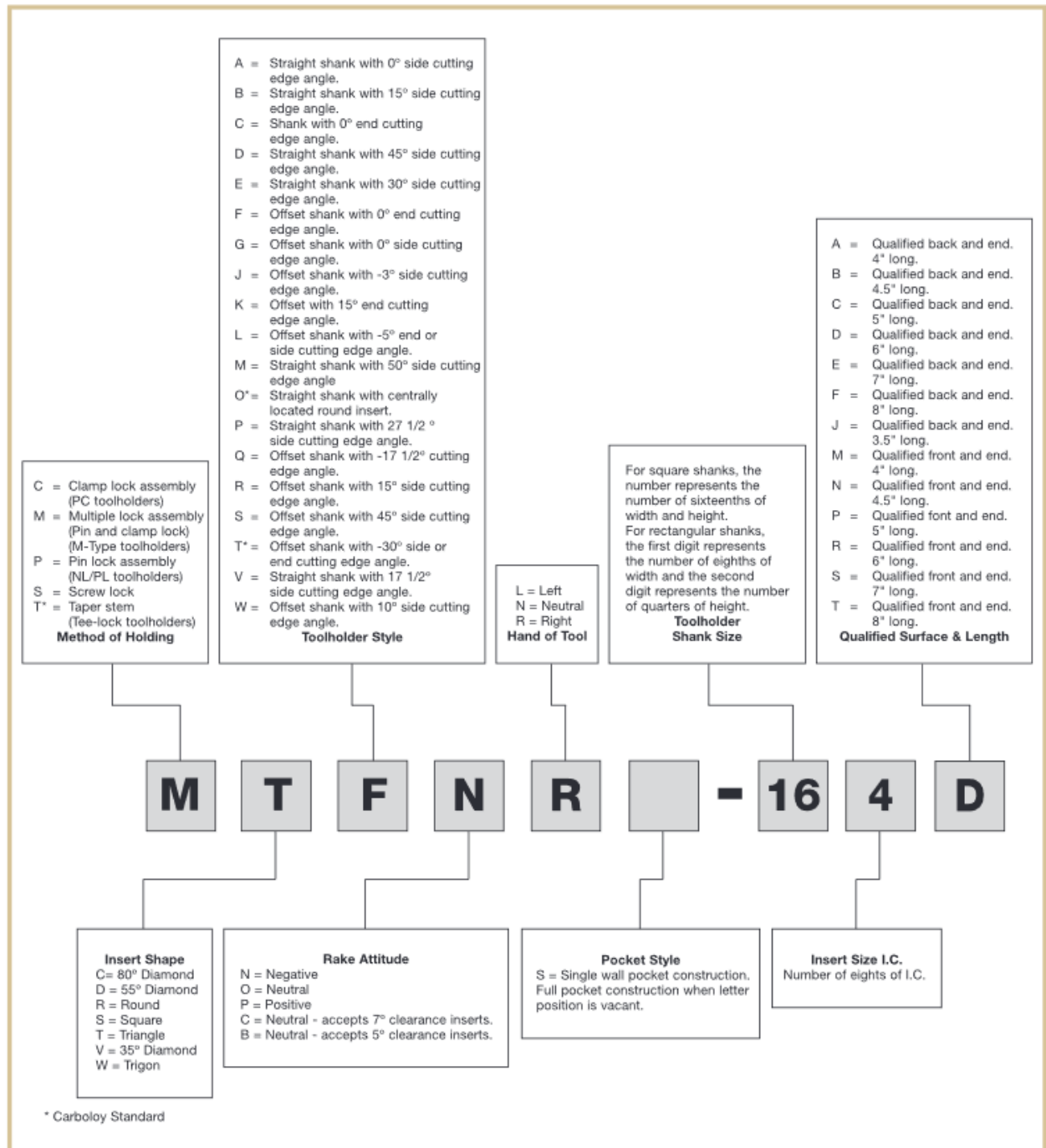
10. Chip Breaker		
LP	MP	RP
		
LM	MM	RM
		
LK	MK	RK
		
LS	MS	RS
		
FP	LP	MP
		
MA	SW	MW
		
HZ	HX	HV
		

Toolholders for Turning Operations

Toolholders for indexable inserts are made from steel that's machined to the desired configuration and hardened to add strength, toughness, and rigidity. Hardness values of a finished toolholder may be in the range of RC 44 to RC 48. Toolholders can be either external or internal types, as well as right- or left-handed. External types are used for machining out side diameters of a rotating part, while internal toolholders are used to remove material from inside diameters, such as boring operations. The selection of the toolholder depends almost entirely on the insert chosen, which is, in turn, determined by the machining operation to be carried out. The required feed, speed, depth of cut, and surface finish determines the type of insert. Once this is known, a toolholder design can be selected. Manufacturers' catalogues give very detailed information about selection of inserts and toolholders, and there's such a variety available that a standard configuration is usually available for almost any application.



ISO Designation of turning tool holder



Types of Toolholders



Collet Chuck:

A collet chuck includes a chuck body, shank and fastening unit. They are very versatile since different collet sizes can be used in one holder allowing for different size tools being held. They come in a variety of shank sizes and types.

Single Angle Collet System:

ER Collet Chuck: For most drilling and lighter higher speed milling applications. Very good concentricity and balance. Safe operational speed as high as 30,000 rpm. Larger sizes have a limit based on centrifugal forces.

TG/PG Series: For heavy drilling and some milling applications. The system provides good concentricity and good grip force. The nut system does not lend itself for good balance considerations at higher speeds.

Double Angle Collet System (DA Series):

For smaller drilling application where clearance is necessary. This system is the simplest to use but lacks the characteristics for concentricity and grip force. Not suitable for precision high speed applications

End Mill Holders:

For heavy machining. Used to hold end mills. End mill holders hold cutting tools with more rigidity than collets. They are available in Weldon type for using tools with Weldon flats.

Hydraulic Toolholders:

Hydraulic toolholders are the easiest holders for most operators to use and should be applied in close tolerance operations or where expensive round tools are being used and maximizing tool life is important. They minimize tool chatter with vibration damping and offer excellent balance characteristics, while delivering high runout accuracy.

Milling Chucks:

For precision in moderate and lower speed milling and nominal drilling applications. Very good concentricity and good side load capability. Movable large clamping nut restricts the balance and centrifugal force capability, limiting the speed to 8,000 to 12,000 rpm.

Shell Mill Holders:

For holding milling cutters that utilize the American National Standard mount for Shell Mills. Modern cutters are commonly indexable. Shell mill lock screw and socket head cap screws are commonly used depending on milling cutter manufacturer's specification. Either or both are provided with each holder depending on the size.

Shrink Fit Holders (Heat Shrink):

For nominal size drilling with higher speed and higher feed milling applications. Excellent concentricity and excellent balance. No moving features, and thin nose diameters, make it exceptional for high speed machining. Safe operating speeds are extremely high due to little impact from centrifugal forces. A Heat Shrink machine should be purchased to achieve optimal performance

Unit 3 Locating and Clamping Devices

A fundamental concern in metalworking is locating the part to be machined, punched, bent, or stamped relative to the work platform (fixture). For example, a CNC machine tool starts its process at a specific point relative to the fixture and proceeds from there. Hence, the accuracy with which a part is machined is quite dependent on the accuracy with which it is positioned in the fixture.

Accurate locating of not just one part, but each and every part that is loaded into the fixture is crucial. Any variation in part location on the fixture adds to the dimensional tolerance that must be assigned to the finished parts.

Additionally, the method of supporting and securing the part in the fixture affects not only dimensional tolerances, but surface finishes as well. This is true because improper supporting or clamping can temporarily or permanently deform the part. Hence, techniques for supporting and clamping must be considered together with the method of locating in order to assure repeatability from part-to-part.

Locating of a part to be machined is a three-step process:

1. Supporting
2. Locating (positioning)
3. Holding (clamping)

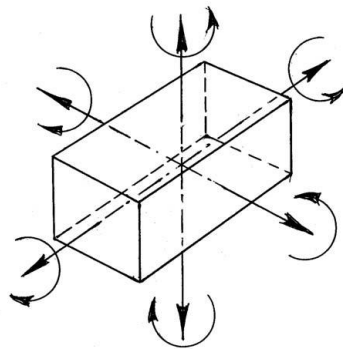
LOCATING PRINCIPLES

To position the work piece w.r.t. to tool, to ensure precision in machining

Locating: dimensional and positional relationship b/w work piece and tool

Locator: device to establish and maintain position of a part in a jig or fixture

Degrees of Freedom



In order to completely specify the position in space of a three-dimensional object (such as the cube that's shown), we refer to six coordinates:

1. Translational position along the X-axis
2. Translational position along the Y-axis
3. Translational position along the Z-axis
4. Rotational position about the X-axis

5. Rotational position about the Y-axis
6. Rotational position about the Z-axis

These six coordinates are known as the six degrees of freedom of a three-dimensional object. As the double-headed arrows indicate, the translational and rotational positions can vary in either direction with respect to each of the three axes.

To completely prevent movement, all six degrees of freedom must be restricted.

3-2-1 Principle of Location:

It is also known as six pin or six point location principle. In this, the three adjacent locating surfaces of the blank (work piece) are resting against 3, 2 and 1 pins respectively, which prevent 9 degrees of freedom.

The rest three degrees of freedom are arrested by three external forces usually provided directly by clamping.

3-2-1 principle states that the six locators are sufficient to restrict the required degree of freedom of any work piece. In this, motion is restricted using clamps and locators. A three pin base can restrict five motions and six pins restrict nine motions

Methodology of 3-2-1 Principle: For this, refer the below figure;

[1] The work piece is resting on three pins A, B and C which are inserted in the base of the fixed body.

[2] The work piece cannot rotate about the axes XX and YY and also cannot move downward.

[3] In this way, the five degrees of freedom 1,2,3,4 and 5 have been arrested.

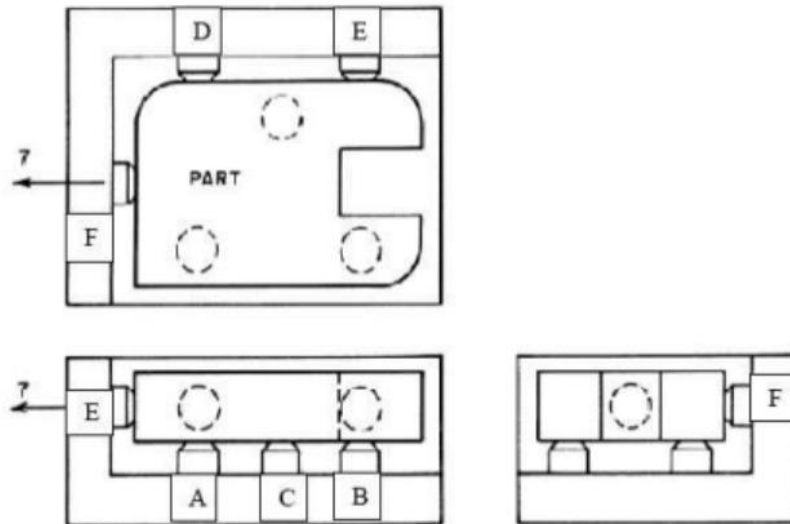
[4] Two pins D and E are inserted in the fixed body, in a plane perpendicular to the plane containing pins A, B & C.

[5] Now the work piece cannot rotate about the Z axis and also it cannot move towards the left.

[6] Hence the addition of pins D and E restrict three more degrees of freedom, namely 6, 7 and 8.

[7] Another pin F in the second vertical face of the fixed body, arrests degree of freedom 9.

The above method of locating a work piece in a fixture is called the 3-2-1 Principle.



Locating Devices

The pins of various design and made of hardened steel are the most common locating devices used to locate a work piece in a jig or fixture. The shank of the pin is press fitted or driven into the body of jig or fixture. The locating diameter of the pin is made larger than the shank to prevent it from being forced into the jig or fixture body due to the weight of the work piece or cutting forces. Depending upon the mutual relation between the work piece and the pin.

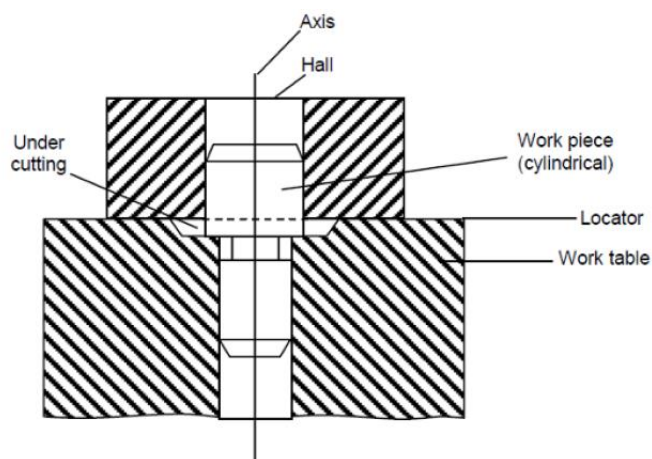
The pin may be classify as follows:

1. Locating Pins:

When reamed or finally finished holes are available in work piece, these can be used for locating purpose of the manner as shown , these are two types of locating pins:

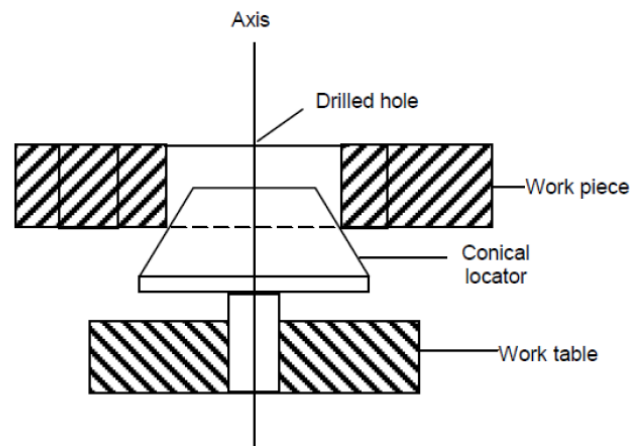
Cylindrical Locators

A cylindrical locator is shown in Figure. It is used for locating components having drilled holes. The cylindrical component to be located is gripped by a cylindrical locator fitted to the jig body and inserted in the drilled hole of the component. The face of the jig body around the locator is undercut to provide space for swarf clearance.



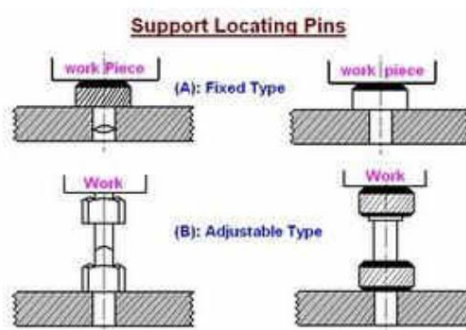
Conical Locator

A conical locator is illustrated in Figure. This is used for locating the workpieces having cylindrical hole in the workpiece. The workpiece is found located by supporting it over the conical locator inserted into the drilled hole of the workpiece. A conical locator is considered as superior as it has a capacity to accommodate a slight variation in the hole diameter of the component without affecting the accuracy of location. Degree of freedom along z-axis can also be restrained by putting a template over the workpiece with the help of screws



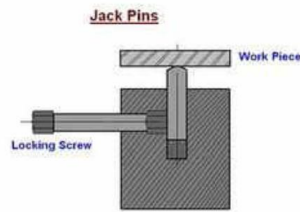
2. Support Locating Pins:

With these pins (also known as rest pins) buttons or pads the work piece with flat surfaces supported at convenient. In the fixed support pins the locating face is either ground flat or curved. Support pins with flat head are usually employed and provide location and support to machine surface, because more contact area is available during location. It would insure accurate and stable location. The spherical head or round head rest buttons are used for supporting rough surfaces (unmachined and cast surfaces) because they provide a point support which may be stable under these circumstances. Adjustable type support pins are used for work piece whose dimension can vary. For example sand casting, forging or unmachined faces.



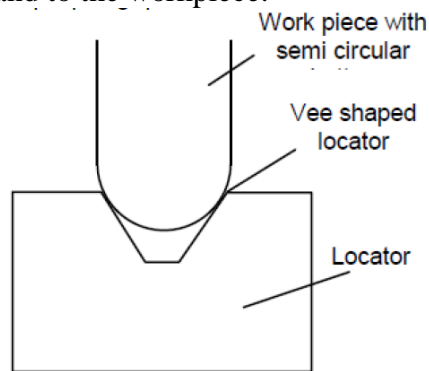
3. Jack Pins:

Jack pins or spring pins are also used to locate the work piece whose dimension are subjected to variation. The pin is allow to come up under spring pressure or conversely is pressed down by the work piece. When the location of the work piece is secured the pin is locked in this position by means of locking screw.



4. Vee Locators

This is quick and effective method of locating the workpiece with desired level of accuracy. This is used for locating the circular and semi-circular type of workpieces as shown in Figure. The main part of locating device is Vee shaped block which is normally fixed to the jig. This locator can be of two types fixed Vee locator and adjustable Vee locator. The fixed type locator is normally fixed on the jig and adjustable locator can be moved axially to provide proper grip of Vee band to the workpiece.



5. Diamond pin locator

- Work piece with the drilled holes use two round pins
- If dimension variation between centre to centre distance of the holes, one round and diamond pin locators are used
- Binding is eliminated
- Prevents movement around the pin and are relieved on two sides to allow variation

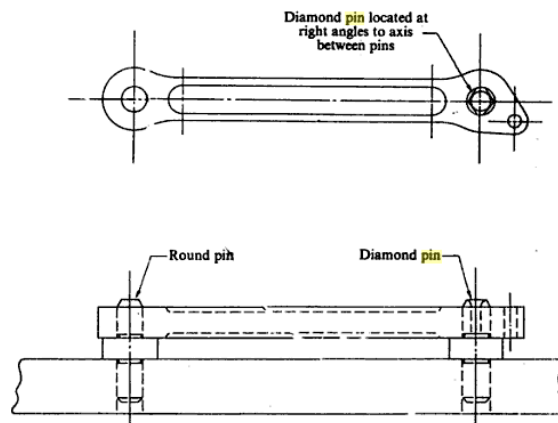
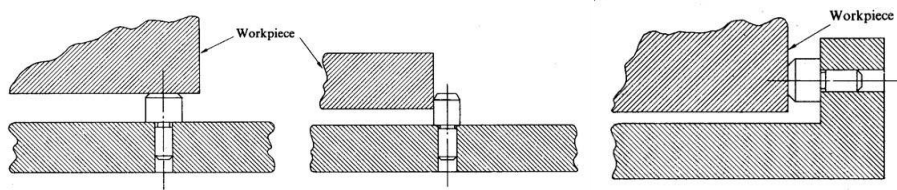


FIGURE 6-22
Use of one round and one diamond locating pin.

General methods of locating

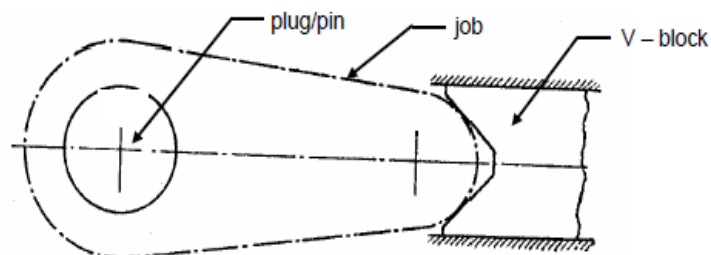
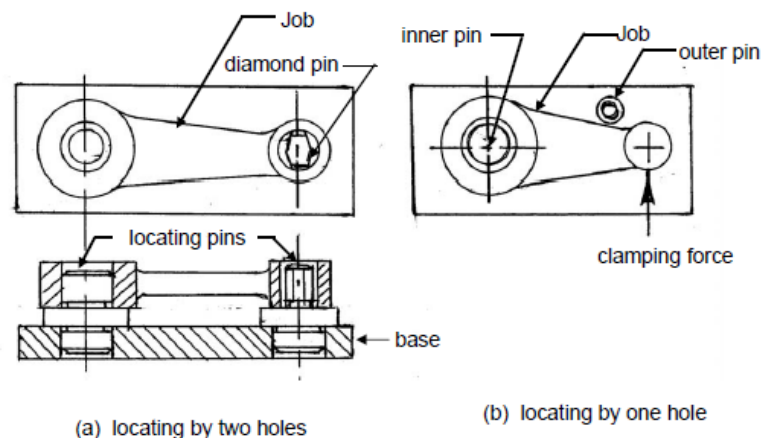
1. **Locating by flat surfaces** Fig. typically shows locating jobs by their flat surfaces using various types of flat ended pins and buttons.



2. **Locating by holes**

In several cases, workpieces are located by premachined (drilled, bored or pierced) holes, such as;

- Locating by two holes as shown in Fig.a where one of the pins has to be diamond shaped to accommodate tolerance on the distance between the holes and their diameters
- Locating by one hole and an external pin which presents rotation of the blank around the inner pin as indicated in Fig.b
- Locating by one hole and one Vee-block as shown in Fig.c



3. **Locating from Irregular Surfaces(External)**

Locating work from an external profile, or outside edge, is the most common method of locating work in the early stages of machining.

For simple components,a sighting plate may be used. Location is done by adjusting the workpiece in such a way that it has equal margins on all sides.

In large parts cylindrical pins can be used.

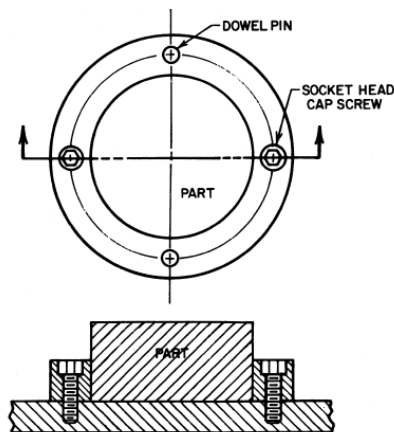


4. Nesting locator or cavity locator
used to position the work piece
Accurate method for profile location
No need of supplementary locating devices
But it is difficult to lift out of cavity

Common types:-

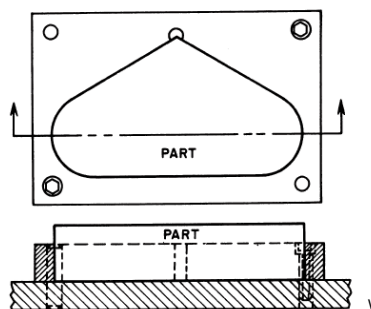
Ring Nest

Used for cylindrical workpiece.
It encloses the workpieces fully



Full Nest

For work pieces other than cylinders.
Encloses the work completely.



Profile locators

Locating work from an external profile or outside edge.

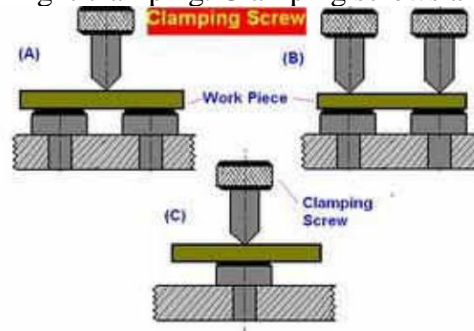
Clamping devices:

If the work piece can not be restrained by the locating devices or elements, it become necessary to clamp the work piece in jig or fixture body. The most common example of clamping devices is bench vice. The purpose of the clamping is to exert a pressure to press a work piece against the locating surfaces and hold it there in a position to the cutting forces. In bench vice the movable jaw of the vice exert force on the work piece , their by holding it in correct position of location in the fixed jaw of the vice.

The commonly used clamping devices are follows:

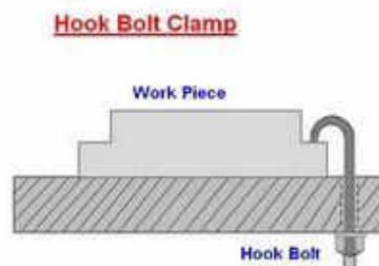
1. Clamping Screws:

Clamping screws are used for light clamping. Clamping screws are shown in fig.



2. Hook Bolt Clamp:

This is very simple clamping device and is only suitable for light work and where usual tip of the clamp is inconvenient. The typical hook bolt clamp is shown.

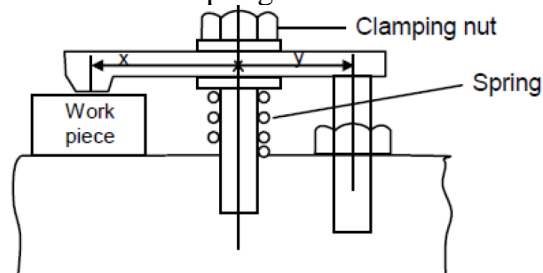


3. Bridge Clamp:

It is very simple and reliable clamping device. The clamping force is applied by spring loaded nut. Bridge Clamp

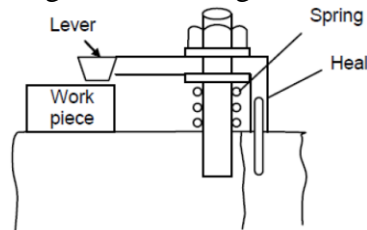
The bridge clamp is illustrated in Figure.

It applies more clamping pressure as compared to heel clamp. The clamping pressure experienced by the workpiece depends on the distances To release the workpiece the nut named as clamping nut is unscrewed. The spring lifts the lever to release the workpiece.



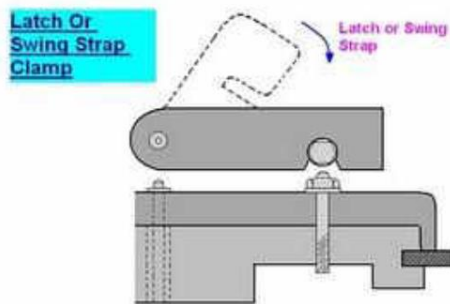
4. Heel Clamp:

These consist of a rusted plate, center stud and heel. This trap should be strengthened at the point where the hole for the stud is cut out, by increasing the thickness around the hole. The design differ from simple bridge clamp in that a heel is provided at the outer end of the clamp to guide its sliding motion for loading and unloading the work piece.



5. Swinging Strap(Latch Clamp):

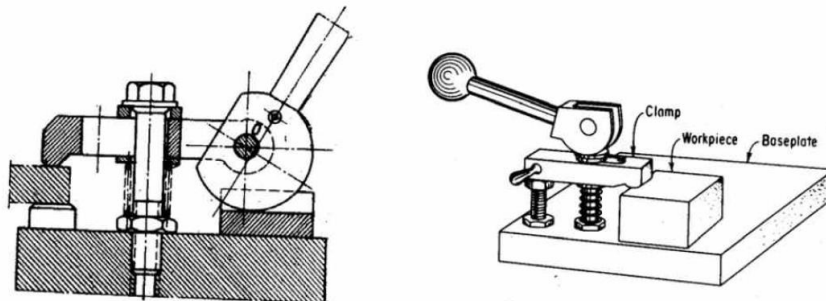
This is a special type of clamp which provides a means of entry for loading and unloading the work piece. For this the strap(latch or lid) can be swing out from the work piece. The typical swing strap or latch clamp is shown in figure



6. Quick clamps

Cam clamping:

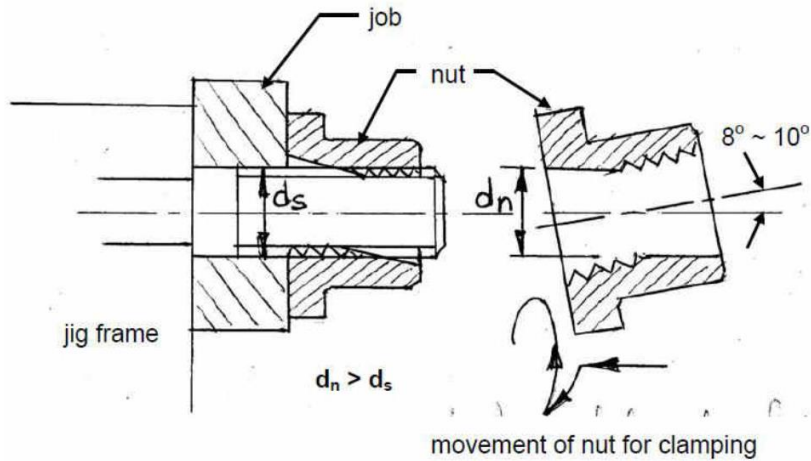
Quick clamping by cam is very effective and very simple in operation. Some popular methods and systems of clamping by cam are shown in Fig. The cam and screw type clamping system is used for clamping through some interior parts where other simple system will not have access.



(a) clamping by cam

Quick acting nut

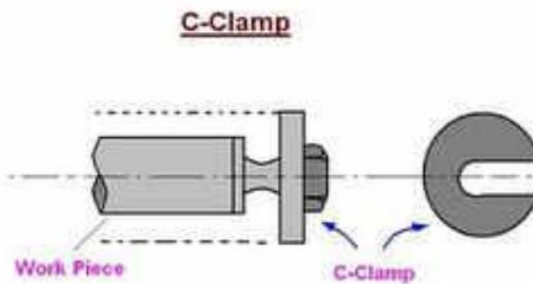
Use of quick acting nut – a typical of such nut and its application is visualized schematically in below figure.



Cam clamping Quick clamping by cam is very effective and very simple in operation. Some popular methods and systems of clamping by cam are shown in below Fig. The cam and screw type clamping system is used for clamping through some interior parts where other simple system will not have access.

C- Clamp

To unload the work piece, the locking nut is unscrewed by giving it about one turn and this releases the c- clamp. When the clamp is removed or swing away the work piece can freely pass over the nut. To reverse procedure is adopted for loading the work piece.



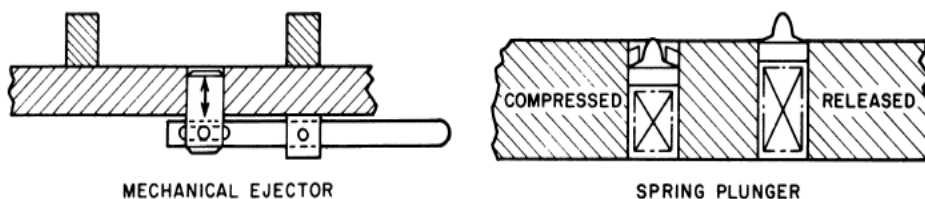
Ejectors

Used to remove the workpiece from close fitted locators and are present behind the workpiece

Speed up the operation by reducing unloading time

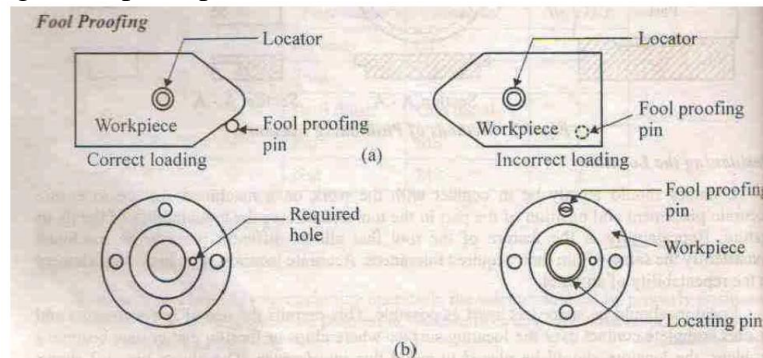
They are of 2 types:

- Mechanical type
- Spring plunger



Fool Proofing

Ensures that the part fits into the tool in its correct position only. The simplest and most cost effective method is positioning a fool proof pin.



Principles of Locating

- One or more surfaces (preferably machined) and / or drilled / bored hole(s) are to be taken for reference.
- The reference surfaces should be significant and important feature(s) based on which most of the dimensions are laid down
- For ease of cleaning, locating surface should be as small as possible.
- Locating surface should be raised above the surrounding surface.
- Sharp corners must be avoided for locating surface.
- Locating pin must easily accessible and visible to operator.
- Locating should be easy, quick and accurate
- In case of locating by pin, the pins and their mounting and contact points should be strong, rigid and hard
- A minimum of three point must be used to locate a horizontal flat surface
- The locating pins should be as far apart as feasible
- Vee block and cones should be used for self-locating solid and hollow cylindrical jobs.
- Sight location is applicable to first – operation location of blank with irregular surfaces produced by casting, forging etc. when the bracket is first located on two edges to machine the bottom surface which will be used for subsequent locating.

Principles of clamping.

- clamping need to be strong and rigid enough to hold the blank firmly during machining
- clamping should be easy, quick and consistently adequate
- clamping should be such that it is not affected by vibration, chatter or heavy pressure
- the way of clamping and unclamping should not hinder loading and unloading the blank in the jig or fixture
- the clamp and clamping force must not damage or deform the workpiece
- clamping operation should be very simple and quick acting when the jig or fixture is to be used more frequently and for large volume of work
- clamps, which move by slide or slip or tend to do so during applying clamping forces, should be avoided

- clamping system should comprise of less number of parts for ease of design, operation and maintenance
- the wearing parts should be hard or hardened and also be easily replaceable
- clamping force should act on heavy part(s) and against supporting and locating surfaces
- clamping force should be away from the machining thrust forces
- clamping method should be fool proof and safe
- clamping must be reliable but also inexpensive

UNIT 4: Jigs and Fixtures

Jigs and fixtures are devices used to facilitate production work, making interchangeable pieces of work possible at a savings in cost of production. Both terms are frequently used incorrectly in shops. A *jig* is a guiding device and a *fixture* a holding device.

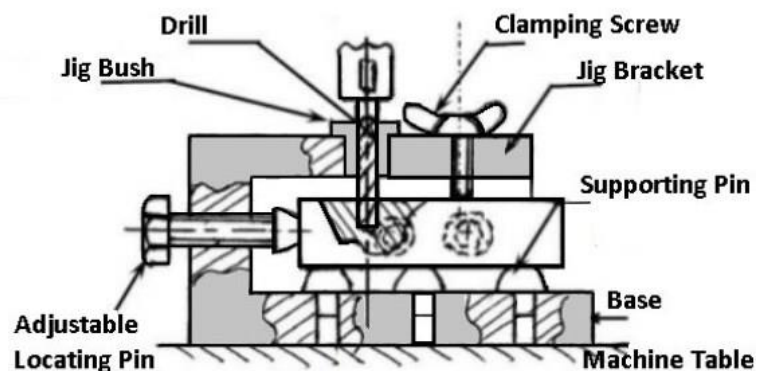
Jigs and fixtures are used to locate and hold the work that is to be machined. These devices are provided with attachments for guiding, setting, and supporting the tools in such a manner that all the workpieces produced in a given jig or fixture will be exactly alike in every way. The employment of unskilled labor is possible when jigs and fixtures can be used in production work. The repetitive layout and setup (which are time-consuming activities and require considerable skill) are eliminated. Also, the use of these devices can result in such a degree of accuracy that workpieces can be assembled with a minimum amount of fitting.

A jig or fixture can be designed for a particular job. The form to be used depends on the shape and requirement of the workpiece to be machined.

Mass production methods demand a fast and easy method of positioning work for accurate operations on it. Jigs and fixtures are production tools used to accurately manufacture duplicate and interchangeable parts. Jigs and fixtures are specially designed so that large numbers of components can be machined or assembled identically, and to ensure interchangeability of components. The economical production of engineering components is greatly facilitated by the provision of jigs and fixtures. The use of a jig or fixture makes a fairly simple operation out of one which would otherwise require a lot of skill and time. Both jigs and fixtures position components accurately; and hold components rigid and prevent movement during working in order to impart greater productivity and part accuracy. Jigs and fixtures hold or grip a work piece in the predetermined manner of firmness and location, to perform on the work piece a manufacturing operation.

The main elements of jigs and fixtures

Main elements of jigs and fixtures are:



Elements of Jigs and Fixtures

Body: It is a plate, box or frame type structure in which the components to be machined are located. It should be quite sturdy and rigid.

Locating elements: These elements locate the workpiece in proper position in relation to the cutting tool.

Clamping elements: These elements firmly secure the workpiece in the located position.

Grinding and setting elements: These elements guide the cutting tool in case of jig and help in proper tool acting in case of fixture.

Positioning elements: These elements include different types of fastning devices, which are used in securing the jig or fixture to the machine at proper position.

Indexing elements: They are not provided always. But, many workpieces may have to be indexed to different positions in order to perform machining operations on different surfaces or different locations. In such cases, these elements will have to be incorporated in the jig or fixture.

JIGS

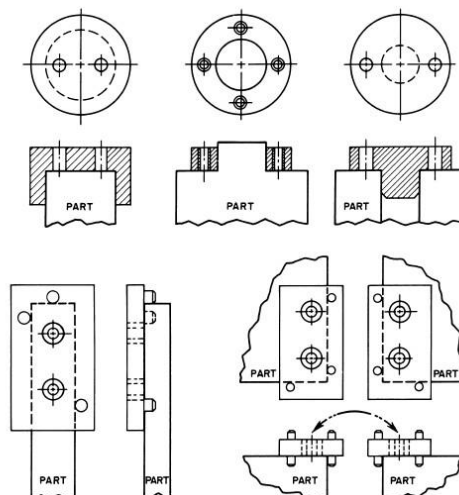
A jig is a special device that holds, supports, or is placed on a part to be machined. It is a production tool made so that it not only locates and holds the workpiece but also guides the cutting tool as the operation is performed. Jigs are usually fitted with hardened steel bushings for guiding drills or other cutting tools.

A jig is any of a large class of tools in woodworking, metalworking, and some other crafts that help to control the location or motion (or both) of a tool. Some types of jigs are also called templates or guides. The primary purpose for a jig is for repeatability and exact duplication of a part for reproduction. An example of a jig is when a key is duplicated, the original is used as a jig so the new key can have the same path as the old one. Since the advent of automation and CNC machines, jigs are often not required because the tool path is digitally programmed and stored in memory.

Types of jig

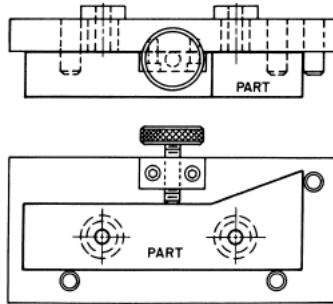
1. Template jigs

Template jigs are normally used for accuracy rather than speed. This type of jig fits over, on, or into the work and is not usually clamped. Templates are the least expensive and simplest type of jig to use. They may or may not have bushings. When bushings are not used, the whole jig plate is normally hardened.



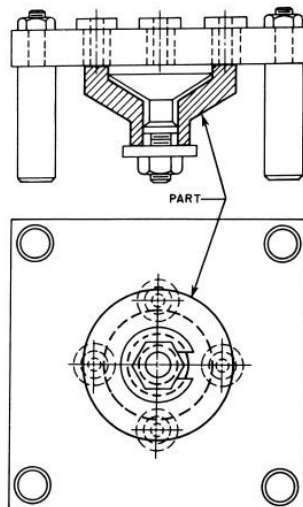
2. Plate jigs

Plate jigs are similar to templates. The only difference is that plate jigs have built in clamps to hold the work. These jigs can also be made with or without bushings, depending on the number of parts to be made.



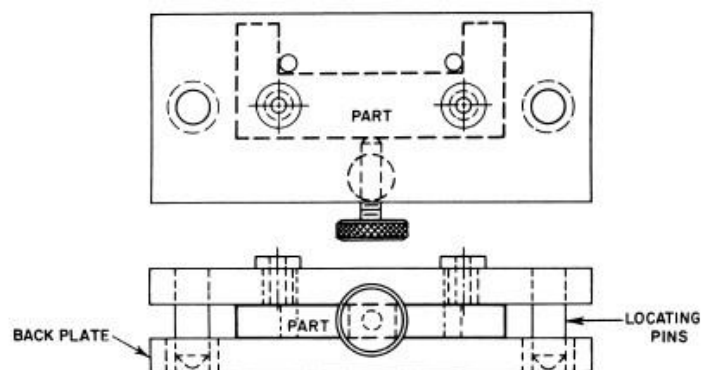
3. Table jig

Plate jigs are sometimes made with legs to raise the jig off the table for large work. This style is called a table jig .



4. Sandwich jigs

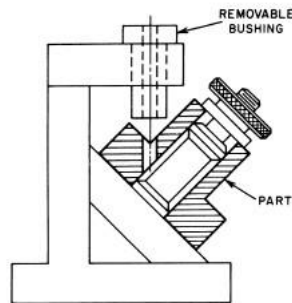
Sandwich jigs are a form of plate jig with a back plate .This type of jig is ideal for thin or soft parts that could bend or warp in another style of jig. Here again, the use of bushings is determined by the number of parts to be made.



5. Angle-plate jigs

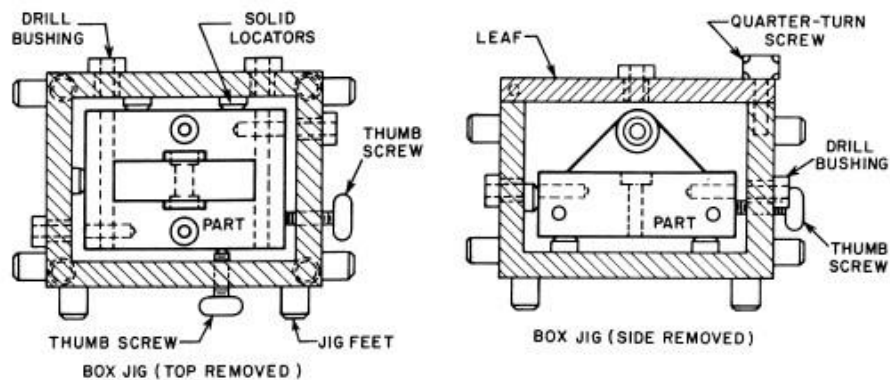
Angle-plate jigs are used to hold parts that are machined at right angles to their mounting locators. Pulleys, collars, and gears are some of the parts that use this type of jig.

A variation in the angle-plate jig is called as modified angle-plate jig, which is used for machining angles other than 90 degrees . Both of these examples have clearance problems with the cutting tool. As the drill exits the product being drilled, it has little or no room for the drill point to clear the product completely, produce a round hole all the way through the part wall, and avoid drilling the part locator. This is most noticeable in Figure , where an angled hole requires additional clearance to the relieved portion of the part locator



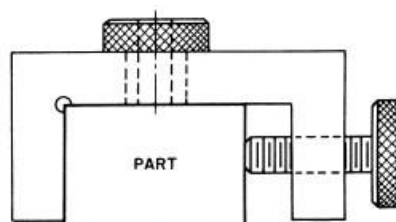
6. Box jigs

Box jigs, or tumble jigs, usually totally surround the part. This style of jig allows the part to be completely machined on every surface without the need to reposition the work in the jig.



7. Channel jigs

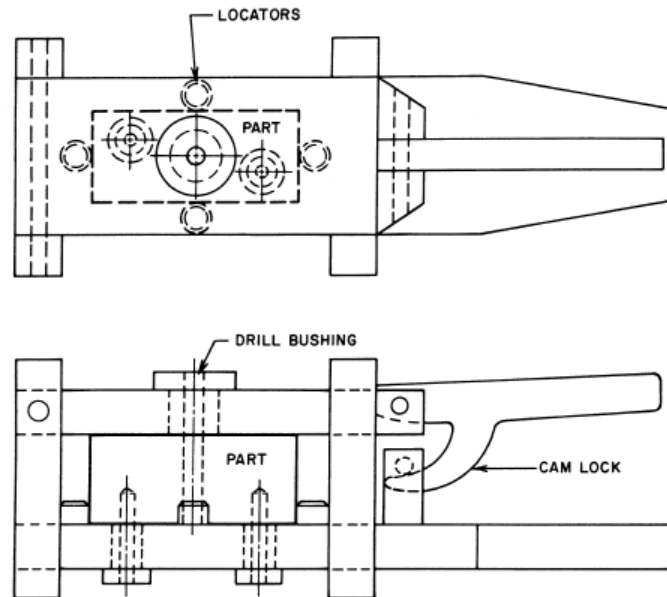
Channel jigs are the simplest form of box jig. The work is held between two sides and machined from the third side. In some cases, where jig feet are used, the work can be machined on three sides.



8. Leaf jigs

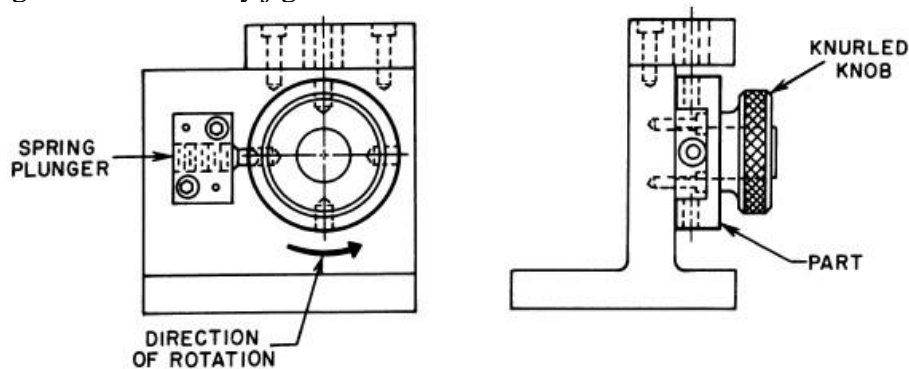
Leaf jigs are small box jigs with a hinged leaf to allow for easier loading and unloading. The main differences between leaf jigs and box jigs are size and part location. Leaf jigs are

normally smaller than box jigs and are sometimes made so that they do not completely surround the part. They are usually equipped with a handle for easier movement.



9. Indexing jigs

Indexing jigs are used to accurately space holes or other machined areas around a part. To do this, the jig uses either the part itself or a reference plate and a plunger. Larger indexing jigs are called rotary jigs.



Fixture

A fixture is a device which holds and locates a workpiece during an inspection or for a manufacturing operation. The fixture does not guide the tool.

In construction, the fixture comprises a different standard or specially designed work holding device, which is clamped on the machine able to hold the work in the position. The tools are set at the required positions on the work by using gauges or by manual adjustment.

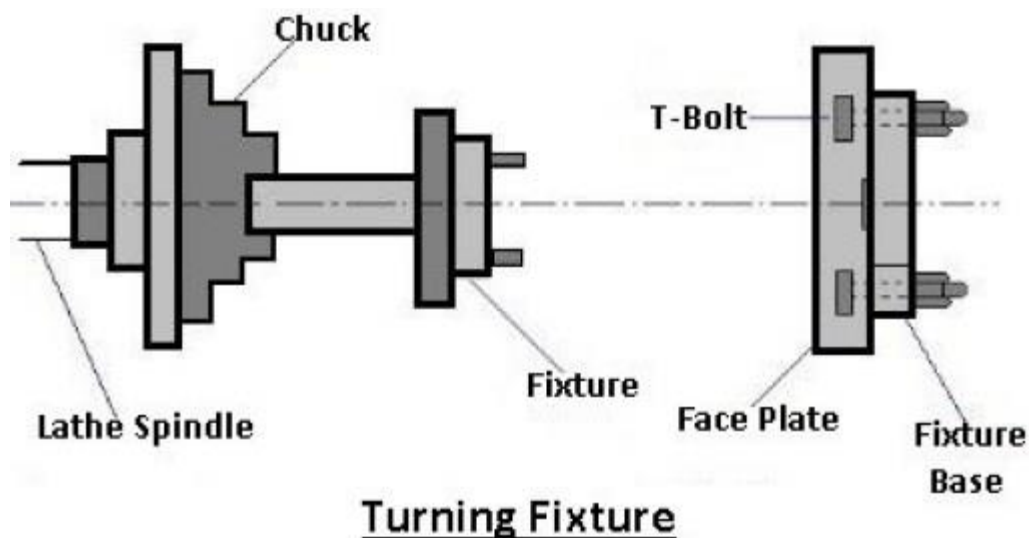
Types of Fixtures

Fixtures are usually named after the type of machining operation for which they are designed and employed. Some popular **types of fixtures** are the **following**.

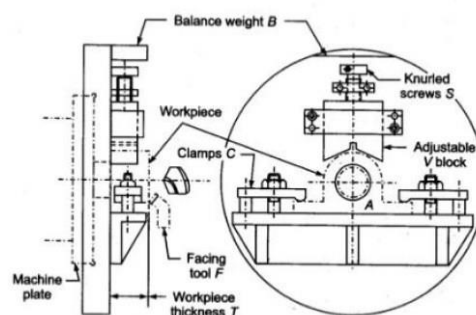
1. Turning fixtures.
2. Milling fixtures.
3. Broaching fixtures.
4. Grinding fixtures.
5. Boring fixtures.
6. Indexing fixtures.
7. Tapping fixtures.
8. Duplex fixtures.
9. Welding fixtures.
10. Assembly fixtures.

Turning fixtures

It is quite easy-to hold regular workpieces on lathes in standard job holding devices like chucks and collects, between the centre and on mandrels or faceplates. But irregularly shaped components offer a lot of difficulties in holding them correctly.



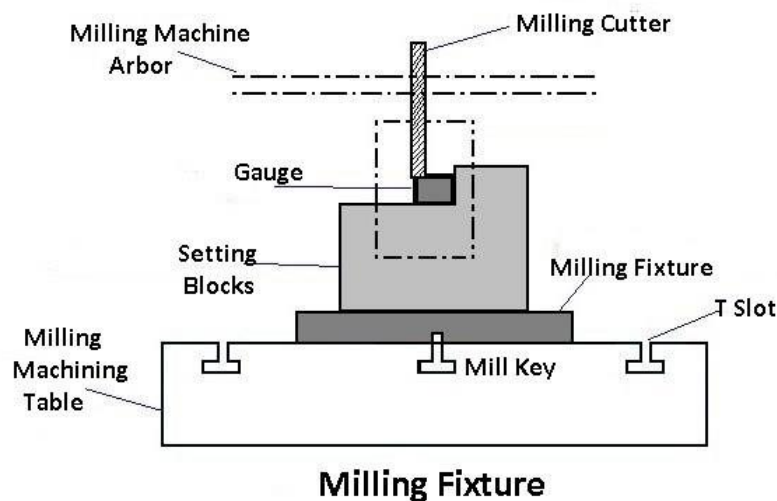
Simple odd shaped jobs can be held in chuck also, say by proper adjustment of jaws in a four-jaw chuck or by using shaped soft jaws. However, workpieces having complicated shaped have to be necessarily held in position with the help of turning fixtures. These fixtures are normally mounted on the nose of the machine spindle or on a faceplate and the workpieces held them.



whenever necessary, the fixture may have to be provided with a counterweight or balance the unbalance fixture. These fixtures should be carefully designed, to be rigid enough, with minimum overhang should grip the workpieces very firmly, should not carry any dangerous projections for the machine operator or tool or both and should be able to provide proper support to him to thin and weak sections of the workpieces during the operation.

Milling fixtures

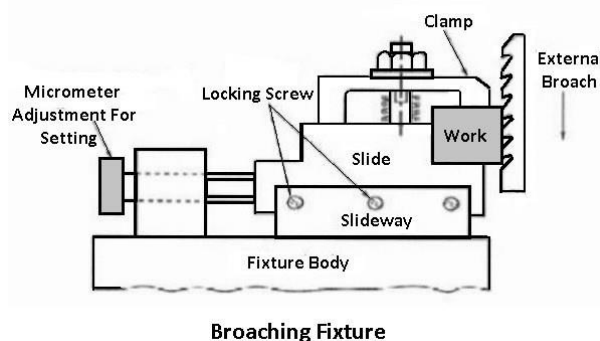
These fixtures are used on milling for carrying out different milling operations on workpieces. The fixture is properly located on the table of the machine and secured in position by means of bolts and nuts.



The table is shifted and set in proper position, in relation to the cutter. The workpieces are located on the base of the fixture and clamped before starting the operation.

Since the cutting forces involved are quite high, and also intermittent, the clamping elements. Proper locating of the fixture on the machine table is usually achieved with the help of two tenons provided under the fixture base. These tenons enter a T-slot of the table to provide the required location. The fixture base can then be secured to the table by means of T-bolts and nuts.

Broaching fixtures.



These fixtures are used on different types of broaching machines to locate, hold and support the workpieces during the operations, such as keyway broaching operations, such as keyway broaching, hole broaching, etc. The use of a clamping plate as a fixture for internal pull-type hole broaching.

Grinding Fixture.

Several different forms of fixtures are used on grinding machines to locate, hold and support the workpieces during the operations. These fixtures may be the standard work-holding devices, such as chucks, mandrels, chuck with shaped jaws, magnetic chucks, etc.

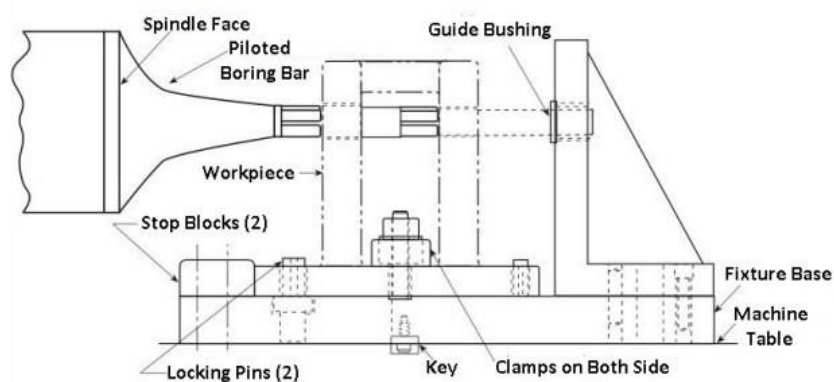
For example; a vertical surface grinder with a rotary table will normally have a rotary fixture secured to its table. Similarly, a plain or string fixture may be provided on a surface grinder with the reciprocating table. A drill grinding attachment is a glaring example of a standard fixture used for grinding of drill geometry. Whatever be the types of fixture designed for grinding operation, it should have provisions for supply and exit of coolant, mounting of wheel dressers and should be properly balanced, if it is a rotating fixture.

Boring Fixture.

The operation of bring can be performed in one of the following two ways:

- By keeping the boring bar (tool) stationary and feeding the rotating workpiece on to the bar.
- By keeping the workpiece stationary and feeding the rotating boring bar into the work.

Accordingly, therefore, the boring fixture are made in two common designs.



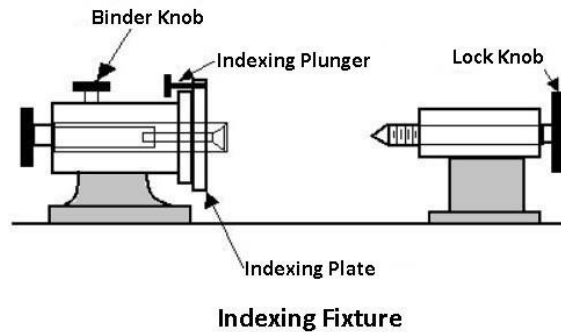
Boring Fixture

One of these incorporates the principle of a drilling jig, and in this, the boring bar (tool) is guided through a pilot bush. such fixture is also often referred to as boring jigs. the other design facilitates holding of the workpiece incorrect position, relative to the boring bar.

Although, this fixture incorporate almost all the common principles of jig and fixture design, their construction need not be as sturdy as that of the milling fixtures, because they never have to bear as heavy cutting loads as involved in milling fixtures, because they never have to bear as heavy cutting loads as involved in milling operations.

Indexing Fixture.

Several components need machining on the different surface such that their machined surface surfaces or forms are evenly spaced.



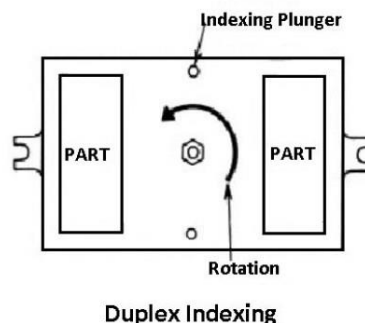
Such components are required to be indexed equally as many as the number of surfaces to be machined. Obviously, the holding devices (jigs or fixtures) used are made to carry a suitable indexing mechanism. A fixture carrying such a device is known as an indexing fixture.

Tapping Fixture.

tapping fixture is specially designed to position and firmly secure identical workpieces for cutting internal threads in drilled holes in them. Odd shaped and unbalanced components will always need the use of such fixtures, especially when the tapping operation is to be carried out repeatedly on a mass scale on such components.

Duplex fixtures.

It is the name given to the fixture which holds two similar components simultaneously and facilitates simultaneously machining of these components at two separate stations. While one workpiece is machined at one station.



Both the operations according to the requirement may be similar or different. Once machining at both the stations is over, the fixture is indexed through 180 degrees so that the first component is shifted to the second station for the second operation and the finished component to the first station.

The finished component is then uploaded here, replaced by a fresh component and the first operation performed on it. The cycle goes on repeating, which enables mass production of components at a fairly high speed.

Welding fixturesg

Welding fixtures are carefully designed to hold and support the various components to be welded in proper locations and prevent distortions in welded structures. For this, the locating element need to be carefully, clamping has to be light but firm, placement of clamping elements has to be clear of the welding area. The fixture has to be quite stable and rigid to withstand the welding stresses.

In many cases, a preferably and commonly used practice is to first tack weld the structure by holding it in welding jig and then transfer it to a holding fixture for full welding. This helps in reducing the chances of distortion considerably and the fixture is subjected to lesser stresses.

Assembly fixture

The function of these fixtures is to hold different components together in their proper relative position at the time of assembling them. For example, two or more steel plates can be held together in relative positions and riveted.

These fixtures, which are used for holding the components for performing mechanical operations, are known as mechanical assembly fixtures. Against this, there are other types of fixtures which the components are held for joining, such as the welding fixtures are also assembly fixtures, but for hot joining.

Difference between Jigs and Fixtures:-

Sr.no	Jigs	Fixtures
1	It is a work holding device that holds, supports and locates the workpiece and guides the cutting tool for a specific operation.	It is a work holding device that holds, supports and locates the workpiece for a specific operation but does not guide the cutting tool
2	jigs are used in unidimensional machining i.e. drilling,reaming, tapping, etc	fixtures are used in multidimensional machining i.e. milling,turning,grinding,etc .
3.	Jigs are light in weight.	Fixtures are rigid and bulky.
4.	Gauge blocks are not necessary.	Gauge blocks may be provided for effective handling.
5.	The jigs are special tools particularly used in drilling, reaming, tapping and boring operation.	Fixtures are specific tools used particularly in milling machine, shapers and slotting machine
6.	Usually it is not fixed to the machine table.	It is fixed to the machine table.
7.	Its cost is more.	Its cost is less as compared with the jig.
8.	Their designing is complex.	Their designing is less complex.

Advantages of Jigs and Fixtures:-

1. Increases the machining accuracy because the work piece is automatically located and the tool is guided without making any manual adjustment.
2. It increases the production capacity by enabling a number of workpieces to be machined in the single set up and in some cases a number of tools may be made to operate simultaneously.
3. The handling time is also greatly reduced due to the quick setting and locating of the work.
4. The speed, feed and depth of cut for machining can be increased due to high clamping rigidity of jigs and fixtures.
5. Enables the production of identical parts which are interchangeable. This facilitates the assembly operation.
6. It eliminates the marking out, measuring and other setting methods before machining.
7. These devices reduce the operator's labor and consequent fatigue handling operations, are minimized and simplified.
8. It reaches the semiskilled operator to perform the operation as the setting operations of the tool and the work are mechanized. This saves labor cost.
9. It reduces the expenditure on the quality control of the finished products.
10. Reduces the overall cost of machining by fully or partly automizing the processes.

Applications of Jigs and Fixtures:-

Applications of jigs are:

- Drilling
- Reaming
- Tapping

Applications of fixtures are:

- Milling
- Grinding
- Planing
- Turning
- Shaping

Design Consideration in Jigs and fixture.

(a)The main frame of jig must be strong enough so that the deflection of jig is as minimum as possible. This deflection of jig is caused due to the forces of cutting, clamping of the workpiece or clamping to the machine table. The mainframe of the jig should have the mass to prevent vibration and chatter.

(b)Frames should be built from simple sections so that frames can be fastened with screws or welded, whenever necessary. Those parts of the frame that remain intact with the jig may be welded. The parts needing frequent changing may be held with the screws. Where the body of jig or fixture has complex shape, it may be cast from good grade of cast iron.

(c)Clamping should be fast enough and require least amount of effort.

- (d)Clamps should be arranged so that they are readily available and may be easily removed.
- (e)Clamps should be supported with springs so that clamps are held against the bolt head wherever possible.
- (f)If the clamp is to swing off the work, it should be permitted to swing as far as it is necessary for removal of the workpiece.
- (g)All locators, clamps should be easily visible to the operator and easily accessible for cleaning, positioning or tightening.
- (h)Provision should be made for easy disposal of chip so that storage of chips doesn't interfere with the operation and that their removal during the operation doesn't interfere with the cutting process.
- (i)All clamps and support points that need to be adjusted with a wrench should be of same size. All clamps and adjustable support points should be capable of being operated from the fronts of the jig.
- (j)Workpiece should be stable when it is placed in jig. If the workpiece is rough, three fixed support points should be used. If workpiece is smooth, more than three fixed support points may be used. Support point should be placed as farthest as possible from each other.
- (k)The three support points should circumscribe the centre of gravity of the workpiece.
- (l)The surface area of contact of support should be as small as possible without causing damage to the workpiece. This damage is due to the clamping or work forces.
- (m)Support points and other parts are designed in such a way that they can be easily replaced on failure.

Fixture Design Procedure

In the design of a fixture, a definite sequence of design stages is involved. They can be grouped into three broad stages of design development.

Stage One deals with information gathering and analysis. These include product analysis such as the study of design specifications, process planning, examining the processing equipment and considering operator safety and ease of use. In this stage, all the critical dimensions and feasible datum areas are examined in detail.

Stage Two involves the consideration of clamping and locating schemes. A clamping scheme is devised in such a way that it will not interfere with the tools or cutters and are fully compatible with proposed locating surfaces or areas. The locating scheme, using standard elements such as pins, pads, etc. is designed to be consistent with clamping and tool-guiding arrangements.

Stage Three is the design of the structure of the fixture body frame. This is usually built around the workpiece as a single element which links all the other elements used for locating, clamping tool-guiding, etc. into an integral frame work.

The above procedures are quite general and can be modified depending on the relative importance of the various elements in providing for the required accuracy of the workpiece to be located and secured into the fixturing device. With the popular adaptation of modular fixturing elements, the fixture body frame is usually a standard block with fixed arrays of locating and fixing holes or slots.

Unit 5 Press Tool Design

Introduction to Press tools

Press working is used as general term to cover all press working operations on sheet metal. The stamping of parts from sheet metal is shaped or cur through deformation by shearing, punching, drawing, stretching, bending, coining etc. Production rates are high and secondary machining is not required to produce finished parts with in tolerance.

A pressed part may be produce by one or a combination of three fundamental press operations. They include:

1. **Cutting** (blanking, piercing, lancing etc) to a predetermined configuration by exceeding the shear strength of the material.
2. **Forming** (drawing or bending) whereby the desired part shape is achieved by overcoming the tensile resistance of the material.
3. **Coining** (compression, squeezing, or forging) which accomplishes surface displacement by overcoming the compressive strength of the material.

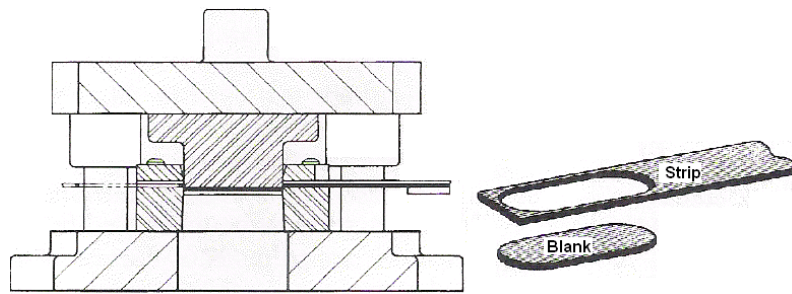
Whether applied to blanking or forming the under laying principle of stamping process may be desired as the use of force and pressure to cut a piece of sheet metal in to the desired shape. Part shape is produced by the punch and die, which are positioned in the stamping press. In most production operations the sheet metal is placed on the die and the descending punch is forced into the work piece by the press.

Inherent characteristics of the stamping process make it versatile and foster wide usage. Costs tend to be low, since complex parts can be made in few operations at high production rates.

Various operations that are performed in Press tools are;

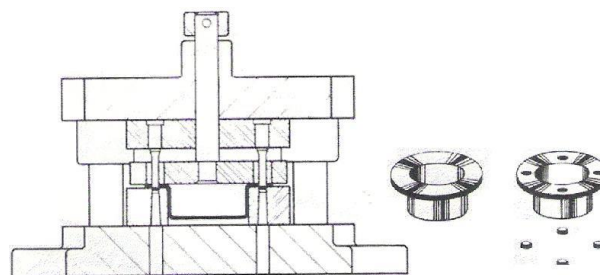
Blanking

When a component is produced with one single punch and die with entire periphery is cut is called Blanking. Stampings having an irregular contour must be blanked from the strip. Piercing, embossing, and various other operations may be performed on the strip prior to the blanking station.



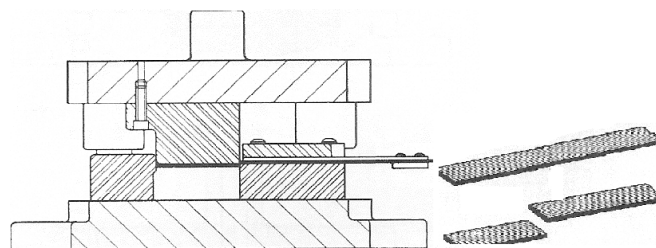
Piercing:

Piercing involves cutting of clean holes with resulting scrape slug. The operation is often called piercing, although piercing is properly used to identify the operation for the producing by tearing action, which is not typical of cutting operation. In general the term piercing is used to describe die cut holes regardless of size and shape. Piercing is performed in a press with the die.



Cut-off

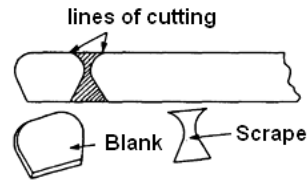
Cut off operations are those in which strip of suitable width is cut to lengthen single. Preliminary operations before cutting off include piercing, notching, and embossing. Although they are relatively simple, cut-off tools can produce many parts.



Parting off

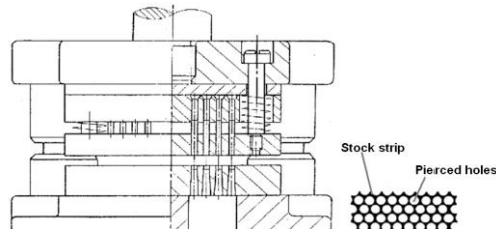
Parting off is an operation involve two cut off operations to produce blank from the strip. During parting some scrape is produced. Therefore parting is the next best method for cutting blanks. It is used when blanks will not rest perfectly.

It is similar to cut off operation except the cut is in double line. This is done for components with two straight surfaces and two profile surfaces.



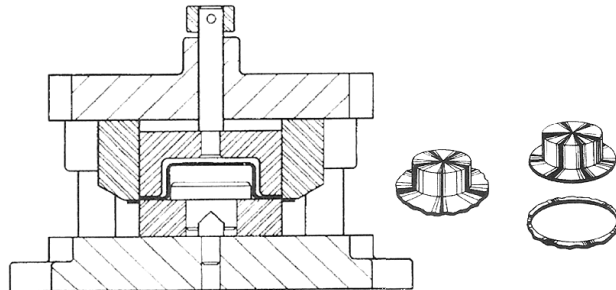
Perforating:

Perforating is also called as piercing operation. It is used to pierce many holes in a component at one shot with specific pattern.



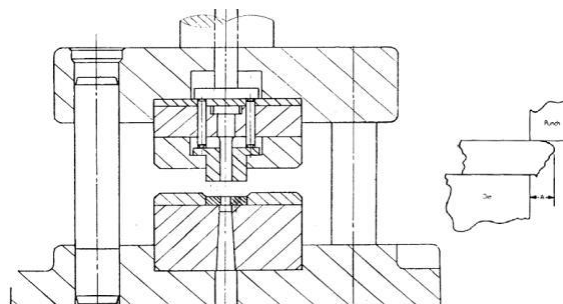
Trimming

When cups and shells are drawn from flat sheet metal the edge is left wavy and irregular, due to uneven flow of metal. This irregular edge is trimmed in a trimming die. Shown is flanged shell, as well as the trimmed ring removed from around the edge. While a small amount of Material is removed from the side of a component or strip is also called as trimming.



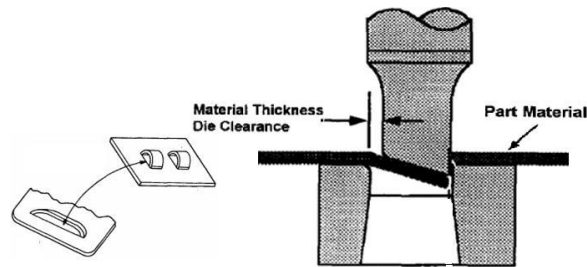
Shaving

Shaving removes a small amount of material around the edges of a previously blanked stampings or piercing. A straight, smooth edge is provided and therefore shaving is frequently performed on instrument parts, watch and clock parts and the like. Shaving is accomplished in shaving tools especially designed for the purpose.



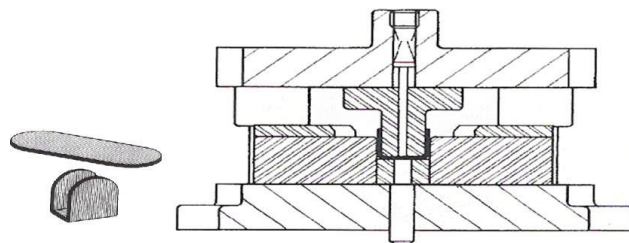
Lancing

Lancing is cutting along a line in a product without feeling the scrape from the product. Lancing cuts are necessary to create lovers, which are formed in sheet metal for venting function.



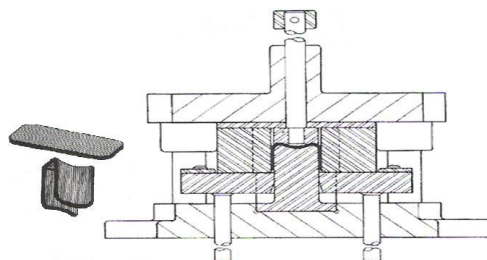
Bending

Bending tools apply simple bends to stampings. A simple bend is done in which the line of bend is straight. One or more bends may be involved, and bending tools are a large important class of pres tools.



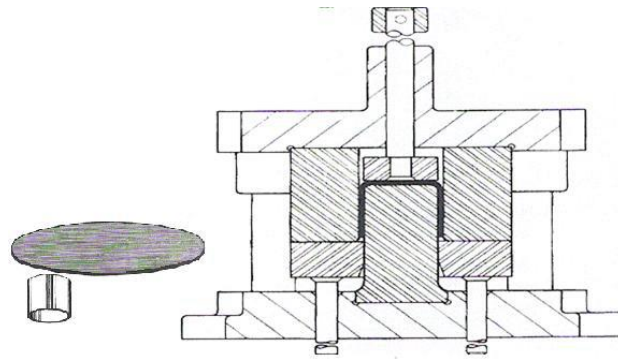
Forming

Forming tools apply more complex forms to work pieces. The line of bend is curved instead of straight and the metal is subjected to plastic flow or deformation.



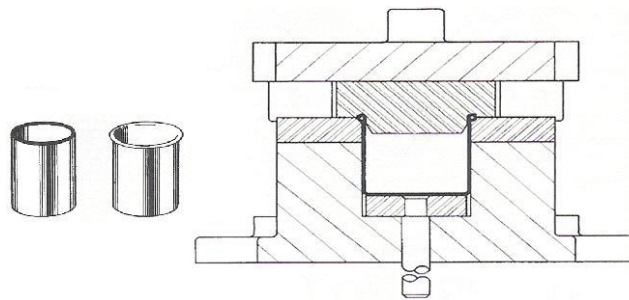
Drawing

Drawing tools transform flat sheets of metal into cups, shells or other drawn shapes by subjecting the material to severe plastic deformation. Shown in fig is a rather deep shell that has been drawn from a flat sheet.



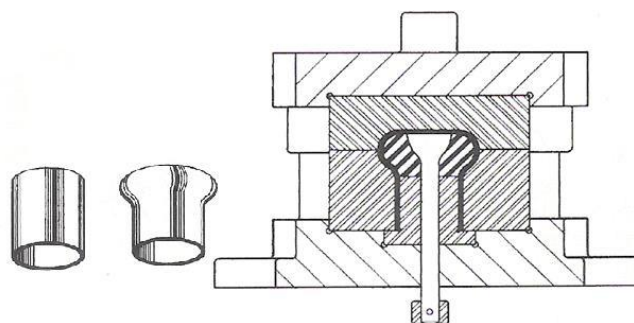
Curling

Curling tools curl the edges of a drawn shell to provide strength and rigidity. The curl may be applied over aware ring for increased strength. You may have seen the tops of the sheet metal piece curled in this manner. Flat parts may be curled also. A good example would be a hinge in which both members are curled to provide a hole for the hinge pin.

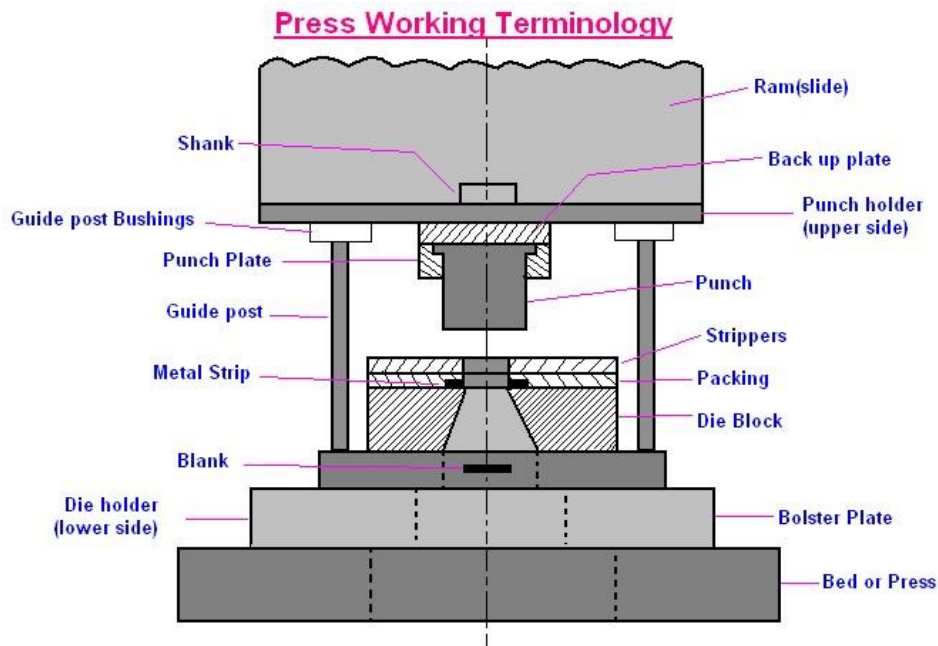


Bulging

Bulging tools expand the bottom of the previously drawn shells. The bulged bottoms of some types of coffee pots are formed in bulging tools



Elements of press tool



Different types of presses have almost common types of main parts. These parts are described below.

1: Bed:

The bed is the lower part of the press frame that serves as a table to which a Bolster plate is mounted.

2: Bolster Plate:

This is a thick plate secured to the press bed, which is used for locating and supporting the die assembly. It is usually 5 to 12.5 cm thick.

3: Die Set:

It is unit assembly which incorporates a lower and upper shoe, two or more guide parts and guide part bushings.

4: Die Block:

It is a block or a plate which contains a die cavity

5: Lower Shoe:

The lower shoe of the a die set is generally mounted on the bolster plate of a press. The die block is mounted on the lower shoe, also the guide post are mounted on it.

6: Punch :

This is male component of a die assembly, which is directly or indirectly moved by and fastened to the press ram or slide.

7: Upper Shoe:

This is the upper part of the die set which contains guide post bushings.

8: Punch Plate :

The punch plate or punch retainer fits closely over the body of the punch and holds it in proper relative position.

9: Back up Plate:

Back up plate or pressure plate is placed so that intensity of pressure does not become excessive on punch holder. The plate distributes the pressure over a wide area and the intensity of pressure on the punch holder is reduced to avoid crushing.

10: Stripper:

It is a plate which is used to strip the metal strip from cutting a non-cutting Punch or die. It may also guide the sheet.

SPECIFICATIONS OF A PRESS

- (a) **Maximum Force** : Maximum force that its ram can exert on the workpiece, this is expressed in tones and called tonnage. It varies from 5 to 4000 tonnes for mechanical press. It may be up to 50,000 tonnes by hydraulic press.
- (b) **Maximum Stroke Length** : Maximum distance traveled by the ram from its top most position to extreme down position. It is expressed in mm. the stroke length is adjustable so different values that can be obtained between minimum and maximum of stroke length, these are also the part of specifications.
- (c) **Die Space** : Total (maximum) surface area, along with $(b \times d)$, of bed, base, ram base. This the area in which die can be maintained.
- (d) **Shut Height** : Total opening between the ram and base when ram is at its extreme down position. This is the minimum height of the processed workpiece.
- (e) **Press Adjustments** : Different stroke lengths (already covered in point number 2). Different tonnage that can be set as per the requirement.
- (f) **Ram Speed** : It is expressed as number of strokes per minute. Generally it can be 5 to 5000 strokes per minute.

Press Shut Height

Shut height is defined as the distance from the bottom face of the ram (slide) to the top of the press bed, with the ram positioned at the very bottom of its stroke (Fig.) and ram adjustment set to the full-up position.

The press shut height can be found on a specification plate attached to the press frame and in the presses' technical manuals. Shut height may be expressed in inches, metric units or both.

The press shut height listed on the specification plate is its maximum height. The ram has an adjustment screw that can be turned manually or by an electric motor to adjust the ram position downward to reduce the press shut

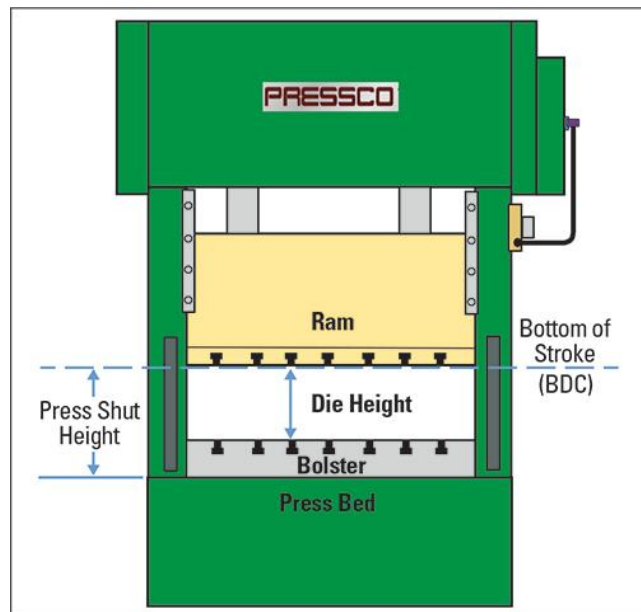


Fig.—Press shut height

height. The maximum amount of adjustment is determined by the adjustment-screw length. The maximum adjustment also is listed on the press-specification plate

Shearing Theory

The name shearing stands for the method of cutting sheet or stock without forming chips. The material is stressed in a section, which lies parallel to the forces applied. The force is applied either by means of shearing blade or punch and die.

The forces necessary to bring rupture of the material depends primarily upon the shearing strength (T_{max}). And the stressed section or the stressed area and secondly upon the shape, condition and position of the shear blades.

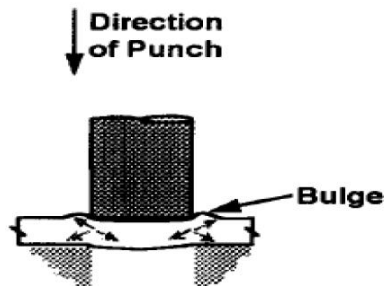
Critical stages in shearing

1. Plastic deformation.
2. Penetration.
3. Fracture.

1. Plastic deformation:

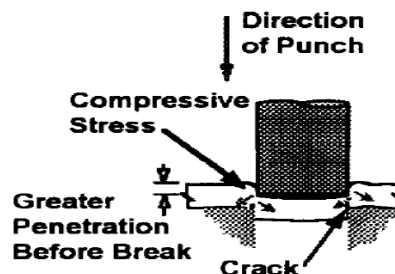
The pressure applied by the punch on the stock material tends to deform it into the die opening when the elastic limit is exceeded by further loading, a portion of the material will be forced into the die opening in the form of an embossed on the lower face of the material and will result in a corresponding depression on its upper face.

This stage imparts a radius on the lower edge of the punched out material. This is called the stage of “**plastic deformation**”.



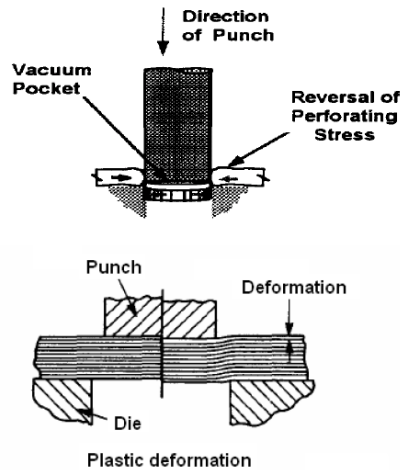
2. Penetration stage:

As the load is further increased, the punch will penetrate the material to a certain depth and force an equally thick portion of metal into the die. This stage imparts a bright polished finish on both the strip and the blank or slug. This is “penetration stage”.



3. Fracture stage:

In this stage, fracture will start from both upper and lower cutting edges. As the punch travels further, these fractures will extend towards each other and eventually meet, causing complete separation. This stage imparts a dull fractured edge. This is the “fracture stage”.



Cutting force

The Cutting force is the force which has to act on the on the stock material in order to cut out the blanker slug. This determines the capacity of the press to be used for particular tool. The first step in establishing the cutting force is to determine the cut length area. The area to be cut is found by multiplying the length of cut by stock thickness.

Formula for calculating the cutting force:

$$\text{Cutting force} = L \times S \times T \text{ max}$$

L = Length of periphery to be cut in 'mm'.

S = Sheet thickness in 'mm'

T max = Shear strength in N/mm²

Generally for calculation purposes Shear strength is considered as 80% of tensile strength.

Shear and tensile strengths for most materials are not the same. Shear strength for:

Aluminum is approximately 50% of its tensile strength

Cold roll steel is approximately 80% of its tensile strength

Stainless steel is approximately 90% of its tensile strength

Press force = Cutting force + Stripping force

Striping force = 10% to 20% of cutting force

Method of reducing Cutting force

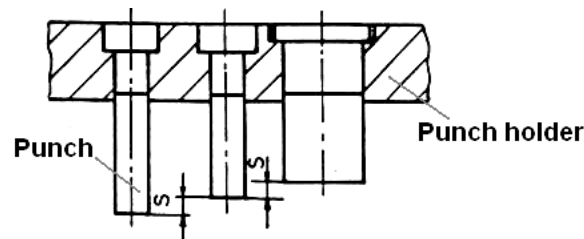
It sometimes becomes necessary to reduce the cutting force to prevent press over loading.

1. Stepped punches to be used.
2. Grinding the face of the punch or die to a small shear angle.

1. Stepped punches

The method of reducing cutting forces is to step punch length. Punches or group of punches progressively become shorter by about one stock material thickness. This will result in distribution of force during the blanking or piercing action on the punches internally reducing in total force.

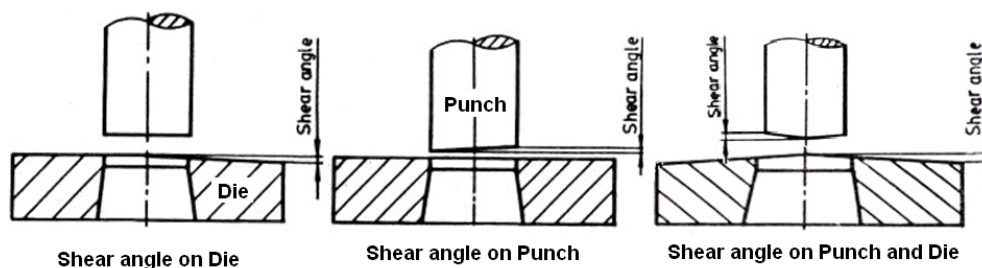
During shearing action lengthier punches will take cutting action at first time and once the sheet is pierced for a sheet thickness, other shorter punches will enter the sheet to get the required holes. This type is mainly used in piercing of more number of holes on the component.



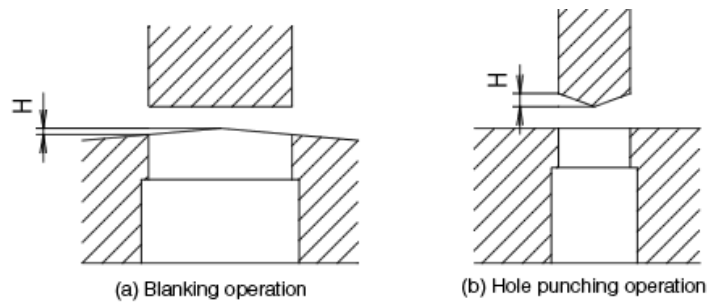
2. Grinding the face of the punch or die to a small shear angle.

A second method is to grind the face of the punch or die to a small shear angle with the horizontal. This has the effect of reducing the contact area while shearing at one time. Providing shear also reduces the shock to the press and smoothens out the cutting operation. The shear angle chosen should provide a change in punch from 1 to 1.5 sheet thickness. Various types of shear angle are shown in the figure. Double shear angle is preferred force acting on the punch. Double shear angle on punches should be concave to prevent the stretching of the material before it is cut. Shear angle may be applied either to the punch face or to the die face, depending on whether the operation is blanking or piercing because shear will distort the work material.

The shear angle for blanking operation will be on the die member, while, as the piercing operation the shear angle will be given on the punch member.

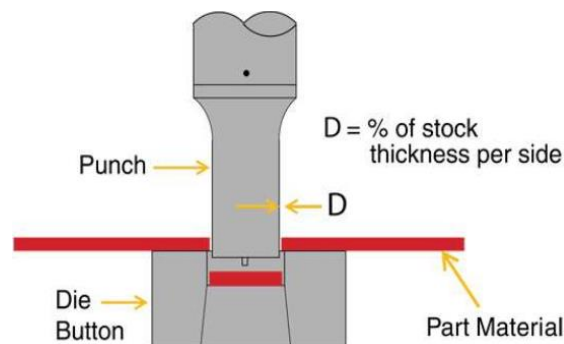


3. As a method of reducing the blanking force, there is a method of providing a shearing angle as shown in Fig. . The shearing angle is provided in the die in the case of blanking operations and in the punch in the case of hole punching operations. Most often the shearing angle is provided so that the dimension H is roughly equal to or more than the plate thickness. By the way, the blanking force can be reduced by about 30% when the dimension H of the shearing angle is made equal to the plate thickness.



Cutting Clearance

Cutting clearance is the gap between a side of the punch and the corresponding side of the die opening when the punch is entered into the die opening. Cutting clearance should always be expressed as the amount of clearance per side. Proper cutting clearance is necessary for the longer life of the tool. Quality of the piece part also depends on proper cutting clearance. A visual examination of the punched components will indicate the amount of clearance and whether the punch and die have optimum cutting clearance or excessive clearance or misalignment.



Clearance Equation The ideal clearance can be calculated using the following formula

$$\text{Clearance} = C * t \sqrt{\frac{T_{\max}}{10}} *$$

Where, C = 0.005 (accurate components)

= 0.01 (normal components)

t = sheet thickness in mm

T max = shear strength of stock material in N/mm²

CENTRE OF PRESSURE

When the shape of blank to be cut is irregular, the summation of shear forces about the centre line of press ram may not be symmetrical. Due to this, bending moments will be introduced in the press ram, producing misalignment and undesirable deflections. To avoid this the “centre of pressure” of the shearing action of the die must be found and while laying out the punch position on the punch holder, it should be ensured that the centre line of press ram passes exactly through the centre of pressure of the blank. This “centre of pressure” is

the centroid of the line perimeter of the blank. It should be noted that it is not the centroid of the area of the blank.

The centre of pressure can be found out by the following procedure:-

- 1.An outline of the piece part is drawn.
- 2.The X and Y axes are placed on it in a convenient position.
- 3.The outline of the piece part is divided into convenient line elements. These are numbered as 1,2,3 and so on.
- 4.The lengths l_1, l_2, l_3 etc. of these line elements are determined.
- 5.The centroids of these line elements are determined.
- 6.The distance of the centroids from the X and Y axes is determined. Let x_1, x_2, x_3 etc. and y_1, y_2, y_3 etc. be the distance of centroids of line elements l_1, l_2, l_3 etc, from the X and Y axes respectively.
- 7.The distance of the centre of pressure from each axis is determined by the method of centroids. i.e.,

$$X = \frac{l_1 x_1 + l_2 x_2 + l_3 x_3 + \dots}{l_1 + l_2 + l_3 + \dots}$$

And

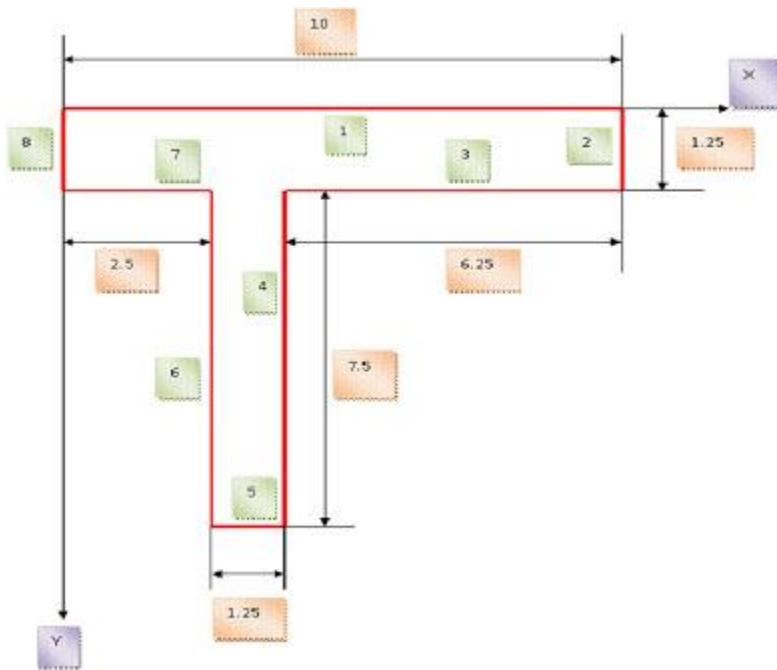
$$Y = \frac{l_1 y_1 + l_2 y_2 + l_3 y_3 + \dots}{l_1 + l_2 + l_3 + \dots}$$

Where,

X = x distance to centre of pressure

Y = y distance to centre of pressure

Example for how to calculate centre of pressure



All dimensions are in cm

Elements	l	x	y	lx	ly
1	10	5	0	50	0
2	1.25	10	0.625	12.5	0.781
3	6.25	6.875	1.25	42.969	7.813
4	7.5	3.75	5.2	8.125	37.5
5	1.25	3.125	8.75	3.906	10.937
6	7.5	2.5	5.1	8.75	37.5
7	2.5	1.25	1.25	3.125	3.125
8	1.25	0	0.625	0	0.781
<hr/>					
	37.5			159.375	98.437

$$\sum l = 37.5$$

$$\sum lx = 159.375$$

$$\sum ly = 98.437$$

There fore,

$$X = \frac{\sum lx}{\sum l} = \frac{159.375}{37.5} = 4.25$$

$$Y = \frac{\sum ly}{\sum l} = \frac{98.437}{37.5} = 2.625$$

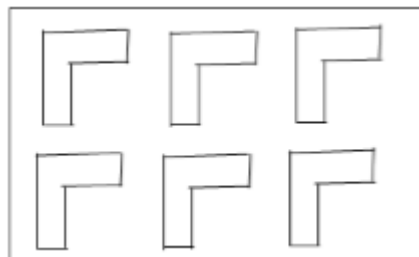
Strip layout

In the blanking die-set design, the first step is to prepare blanking layout i.e. the position of the work pieces in the strip and their orientation with respect to each other. This is known as Scrap Strip Layout. Following are the factors which affects the strip or stock layout.

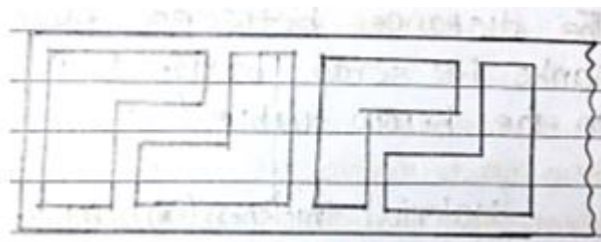
- Economy of Material:
 - As per arrangement in below fig. it can be worked at single row, single pass with a single punch.

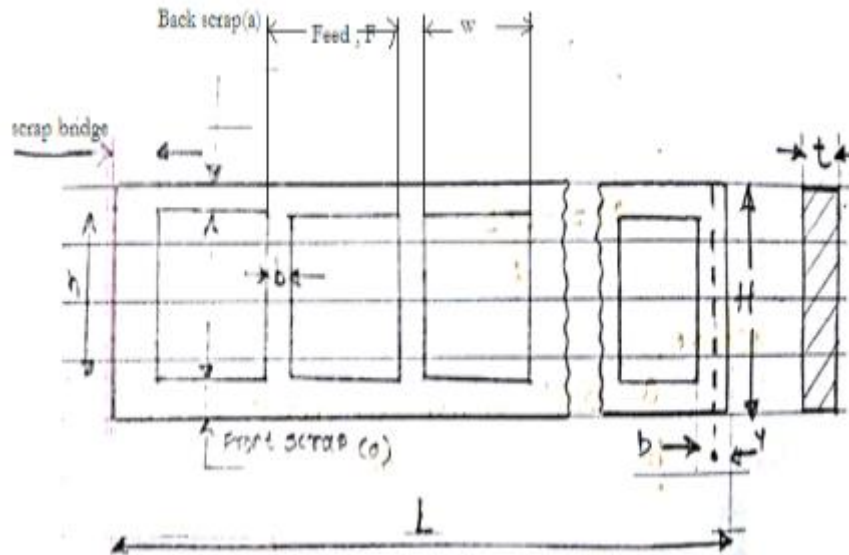


- By feeding the material as per below fig. there is increase in material utilization upto some extent.



- Below figure shows a single row, double pass strip. Here, strip will have to be passed through the dies once turned over and passed through dies second time. Hence, there is a maximum utilization of the material and reduction in scrap.





- The front scrap may be determined by the equation,
 - $a = t + 0.015h$.
- The distance between successive blanks i.e. scrap bridge b is given in the below table:

Material Thickness (mm)	B (mm)
0.8	0.8
0.8 to 3.2	t
Over 3.2	3.2

- The feed or advance or the length of one piece of stock required to produce one blank is ,

$$f = W + b$$

Where , a = back scrap or front scrap

b= scrap bridge

h & w= Height and Width of the component.

F=feed or advance.

T=thickness of the material.

L= length of the sheet.

N= number of blanks to be produced.

Y=scrap remaining at the end of the length.

Considerations in strip layout

- **Economy factor/ % stock utilization**

The designer should try out every possible means to attain a min 60% usage of any strip, without sacrificing the accuracy of the piece part.

Economy Factor =

$$\frac{\text{Area of the blank} \times \text{No of rows} \times 100}{\text{Width of the strip} \times \text{Pitch}}$$

- **Shape of the blank**

The contour of the blank is the main factor which decides the way in which it is to be positioned.

- **Grain direction**

When sheets are produced by rolling, the rolling direction orients the grains. Standard sizes of rolled sheets or strips will have the grains along its length. Bending the strip along the grain direction may result in cracks or fractures. If the blank is to be bent at a later stage the strip should take care of the grain direction.

The grain direction should be at right angles or at an angle more than 45 degree to the direction of the bend when harder varieties of strips are used.

- **Burr side**

In a blanking operation burr is formed on the face of the blank which comes in direct contact with the punch.

In piercing it appears on the face which comes in direct contact with the die.

In some piece parts the burr resulting from either blanking or piercing would be required to appear on a particular face of the blank in relation to details of the blank contour. While deciding the strip layout care must be taken to see that such requirements are met.

- **Stock material**

a comparative study of stock material conservation, tool cost and labour cost is necessary while the strip layout is made. If the stock material is precious every means to conserve the stock material should be employed. If double pass is a one complete pass of the strip, it is reserved fed again for maximum utilisation of stock strip. This is decided by the shape of the component (two pass or double pass both mean the same). Double pass method is employed, labour cost increases. So a double pass layout should justify the cost of stock material conserved.

Based on the above factors, different strip layouts are explained in following paragraphs.

- **Production requirement**

If lesser production is anticipated more emphasis should be given for material conservation without increasing the tool cost.

Cutting dies Types

Classified based on method of operation

- Simple dies
- compound dies
- combination dies
- progressive dies
- transfer dies
- multiple dies

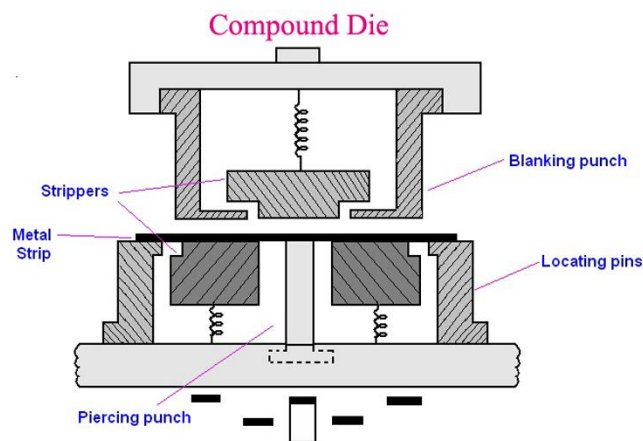
Simple die:

The simple die is also known as single action die. When the sheet metal process die perform only one operation in single stroke of press. Its operation may be as one type of cutting and forming operation. The die set are simple configuration.

Compound die:

The compound die are performed one (or) two more operation at single stroke of press. In compound die process used any one of cutting or forming type.

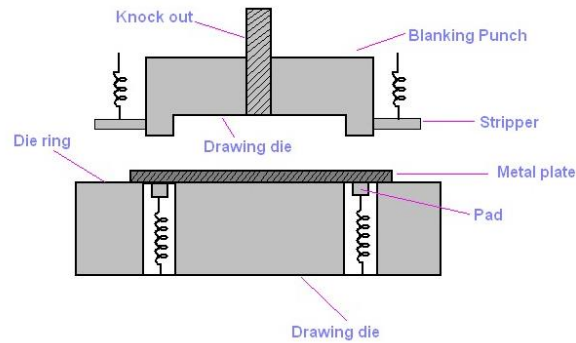
A washer is manufactured in single stroke of press as shown in figure. Including the operation piercing and blanking in one stroke of press movement. It die have more saving time as compared to simple die.



compound die assembly

Combination die:

The combination die can be performed one or more operation in single stroke of press. It is different from other operation. The cutting operations are combined with forging operation.

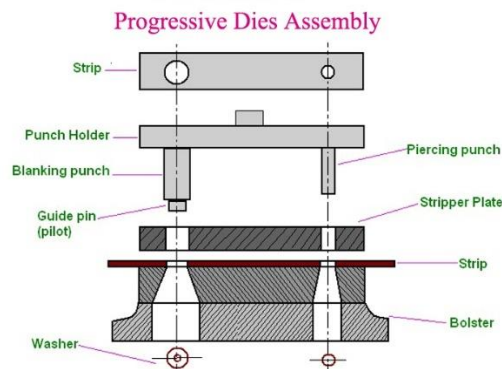


Combination Die

combination types of dies assembly

Progressive die:

More than one operation can be performed in single stroke of press by using progressive die. The die combination having series of die and punch as shown in figure. Where one set of die and punch is called “station”.



progressive types of dies assembly

Each station punch performs on the work piece during a single stroke of press. After that the work piece transfer to another next station of die set. Where next another operation carried out on the sheet metal. It can be made to cut and form a part in successive station of die.

The progressive die consists of as shown in figure. The piercing form performed on first station of press. Next, the metal piece is move to another operation. In between the distance of both operations is called advance.

Now the second stroke of metal blanking operation carried out in the sheet metal. For this blanking operation use of blanking punch and piercing operation done by use of piercing punch. The washer discards from the strip of blanking punch station and the piercing punch from a hole on strip for next washer. Against the metal is advanced and operation continued. Every stroke of press to be generates one component. The number of operation require on work piece deciding the number of station.

Transfer die:

The transfer types of dies are consist of multiple station dies in which metal blank moving from one station to next station like progressive die. The major differed is the progressive die having stroke move one station to another station. But transfer die having already cut blanks are transfer from station to station. The deep drawing operation carried by this die.

Multiple dies:

The multiple dies are performed produce two or more work piece at a time. It is also called as gang die.

Pilots

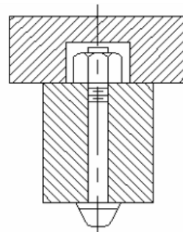
The main purpose of a pilot is to position the stock strip accurately. Pilots also bring the stock strip into proper position for blanking and piercing operation simultaneously. If strip is fed by hand, it may go beyond proper position due to strip stop. In such a situation, pilot takes it back to proper position in a direction away from the strip stop. Pilot also prevents buckling of strip. When the strip is fed by hand, under feeding of strip occurs and pilot pulls the strip forward. Diameter of pilot is generally 0.002 to 0.003 inch smaller than punch diameter for average work and 0.0005 to 0.001 inch smaller than punch diameter for precision work.

Classification of Pilot Pilots are classified as :

Direct pilot, indirect pilot and spring loaded pilot.

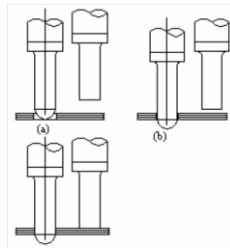
Direct Pilot

Direct pilots are mounted on punch face. A direct pilot is shown in Figure. It acts as a misfeed detector, which detects overfeeding or underfeeding of strip. If it finds misfeed, it actuates a switch to cut off the electric power to press.



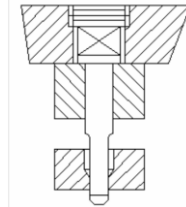
Indirect Pilot

Indirect pilot is used with previously pierced hole at a certain distance from blanking punches. Figure. shows typical indirect pilot. The distortion is avoided by using indirect pilot as it provides support to strip.



Spring-loaded Pilot

Spring-loaded pilot is generally used for workpiece having more than 1/16 inch thickness. A spring-loaded pilot is shown in Figure. Spring retracts the pilot from misfeed. This pilot pierces the hole if it is used for thin plate

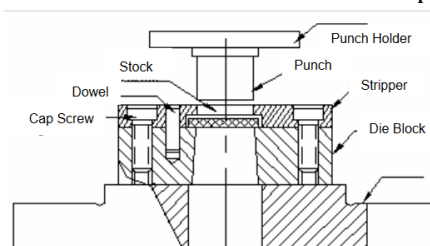


Strippers

The main purpose of stripper is to remove the stock from the punch after blanking or piercing operation.

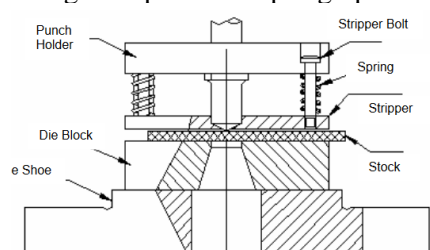
Fixed Stripper

A rectangular plate is mounted on the top of the die block. The strip is passed through the milled channel. In general, the height of the channel should be 1.5 times the stock thickness whereas width must be equal to the summation of strip width and clearance. The clearance is provided to take care of dimensional variation in the width of strip. Enough clearance is provided around the punch for its easy removal; but clearance should be less than 1.5 times the thickness of strip. The back edge of the channel is used as a back gauge to position the strip accurately. A stock pusher is used to hold the strip against the back edge. It is also desirable to help in reducing the wear. The wear resistance is also increased by using inserts of mild steel. The wear resistance is increased by pressing hardened dowel pin along with the back guide. This is shown in Figure. The thickness of channel stripper is found out from the size of the socket head cap screw which is used to hold the strip in position



Spring Operated Stripper

This type of stripper is also called pressure pad stripper. Its main advantage over other stripper, is that it holds the strip flat during the operation. Spring operated stripper is shown in Figure



Spring operated stripper is generally mounted on socket head stripper bolts. Spring stripper may also be mounted on rods. To minimize the bending, length of the rod should provide adequate support to the inner diameter of the spring. This is achieved by pressing a

dowel of proper diameter and length into one plate and drilling a clearance hole in other plate to provide passage for dowel to pass through other plate.

Stock Stops

The function of a stopper is to arrest the movement of the strip when it is fed forward to one pitch length. After each of the press stroke, the stock strip is stopped at definite position in order to perform cutting/non cutting operation on the strip to obtain the component correctly.

Types of stop

Solid stops

A hardened steel block is mounted at the required location

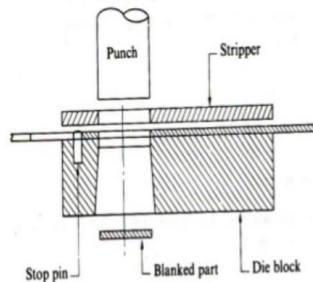
Plain pin stop

It is a plain cylindrical pin mounted in the die block. A clearance hole for the pin stop in the bottom plate is provided for the following reasons.

To permit adjustment of the height of the pin stop without removing the die plate from the assembly.

While resharpening the die the stop pin can be removed.

The pin can be driven down in the event of misfeed. This reduces the chance of damage to the tool.



Spring loaded pin stop

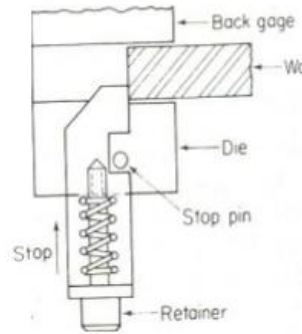
It is a spring pin located at the required stopping position. These stoppers do not require clearance in the opposing tool member. The pin is pressed down by the opposing tool member during operation.

Finger stops

In progressive tools designed for manual feeding finger stops locate the strip for each station except for the final station.

This is actuated manually. It is mounted in the stripper plate. Provision is made for moving the stopper through a predetermined distance. It is pushed inwards to enable the stock material to halt against it.

After the press stroke the stopper is released. When a new stock strip is fed the stopper has to be actuated again.



Trigger stops

For faster manual feeding, trigger stops are preferred.

There are two types of trigger stops.

- Front acting.
- Side acting.

The working mechanism for front acting and side acting trigger stops are same.

The front acting trigger stop is mounted in the front end of the tool and the side acting trigger stop is mounted in the side of the tool.

The lever shaped trigger stop fits freely in the slot milled in the guide plate. One side of the wall of the slot is machined with a taper angle. It gives the necessary movement to the trigger. An inclined set spring at the other end of the trigger holds the trigger in position.

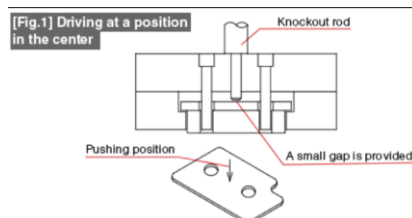
When the strip is pressed against the tip face of a trigger, the trigger moves back wards and stops against the non tapered wall of the slot.

This allows the strip to advance. This advancement is equal to one margin width. When the tool is tripped a knocker bar fixed to the top assembly of the tool comes down and knocks the free end of the trigger. This action lifts to clear the thickness and then jumps back to its old position to fall on the strip (blanked portion). The strip can be fed forward.

Knockouts

A compound punched product is in the state in which it has been pushed inside the die and is pushed up by the knockout, but is staying inside the die due to the force of friction with the die.

In the shape shown in Fig. , since there is a gap at the center of the knockout, it is possible to press the knockout directly with the knockout rod. If it is possible to adopt this method, it is possible to simplify the die structure.



Unit 6 Bending , Drawing and Forging Dies

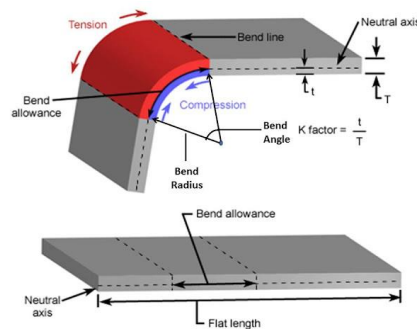
Bending

Sheet metal bending is the plastic deformation of the work over an axis, creating a change in the part's geometry. Similar to other metal forming processes, bending changes the shape of the work piece, while the volume of material will remain the same. In some cases bending may produce a small change in sheet thickness. For most operations, however, bending will produce essentially no change in the thickness of the sheet metal. In addition to creating a desired geometric form, bending is also used to impart strength and stiffness to sheet metal, to change a part's moment of inertia, for cosmetic appearance and to eliminate sharp edges.

Bending Processes

Neutral Axis:

It is an imaginary axis which does not undergo any stress during bending.



Bend Allowance:

The length of the neutral axis in the bend zone is known as bend allowance.

Bend Angle:

The angle formed by the bend area at the center of bend is known as bend angle.

Bend Radius:

Distance between bend center and neutral axis is known as bend radius. It is denoted by r .

Minimum Bend Radius:

The minimum bend radius is the bend radius at which a crack appears on the outer surface of the bend. It is usually expressed in terms of sheet thickness, such as $2T$, $3T$, $4T$ etc. It is different for different materials.

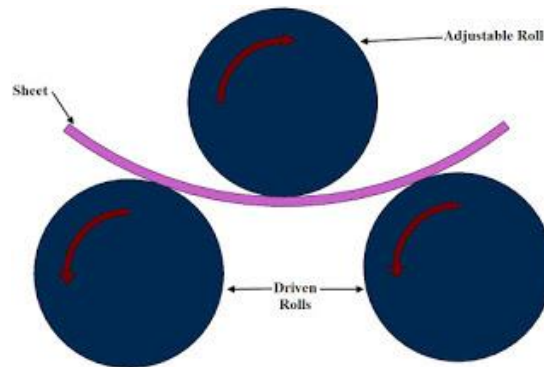
Springback:

When load is removed, the sheet metal shows some elastic recovery and tends to achieve its original position. This phenomenon is called springback. It will increase the final bend radius and decrease the bend angle after springback.

Bending Types

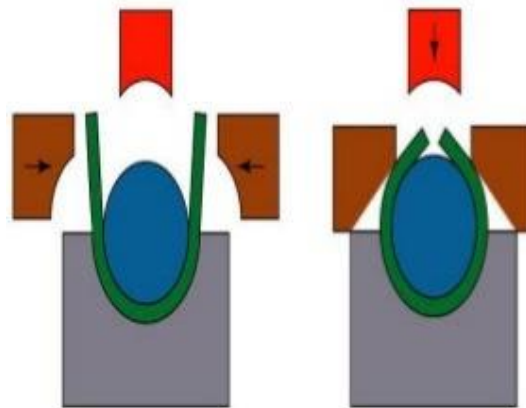
Roll Bending:

This process uses a three rollers set to bend a sheet by adjusting the distance between rolls. This process can utilize to form various curvatures.



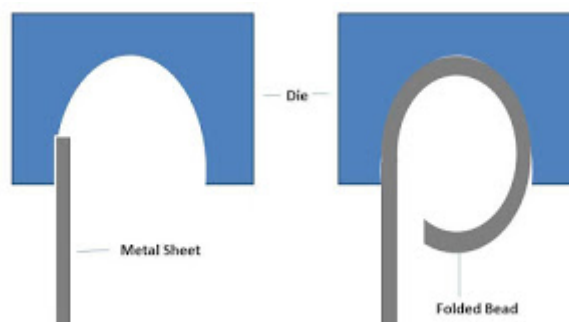
Four Slide machine Bending:

This process uses three movable and one stationary slide to bent a work sheets as shown in figure. This is used to bent small work pieces.



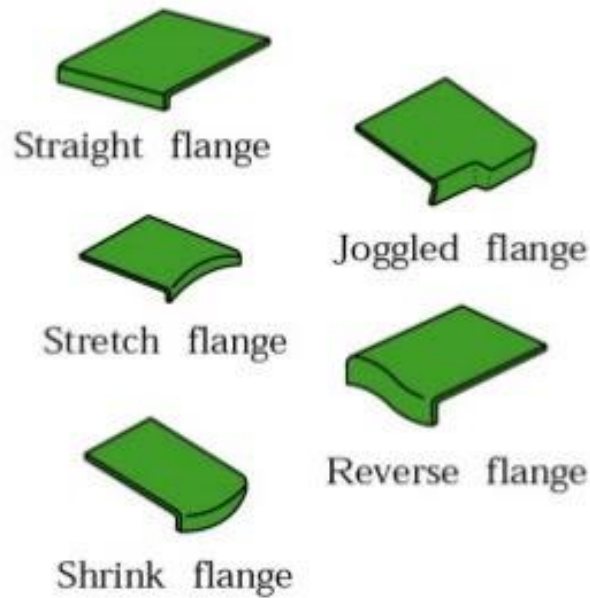
Beading:

It is a process in which the periphery of the sheet metal is bent into the cavity of a die. It increases moment of inertial of the section and stiffness. It also eliminates exposed sharp edges.



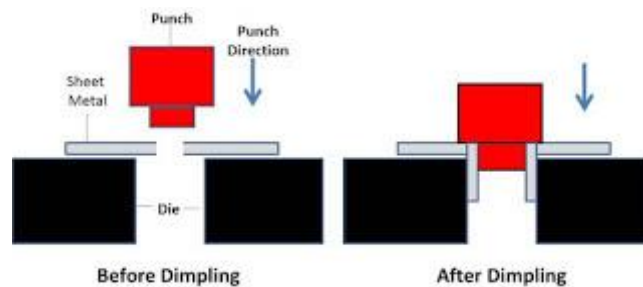
Flanging:

It is a process of bending the edges of the metal sheet at perpendicular to the length. It can be further divided according the shape like straight flange, stretch flange, joggled flange, shrink flange etc. as shown in figure.



Dimpling:

It is a process in which first a hole is made into the sheet metal and then it is expanded into a flange using punch die system.



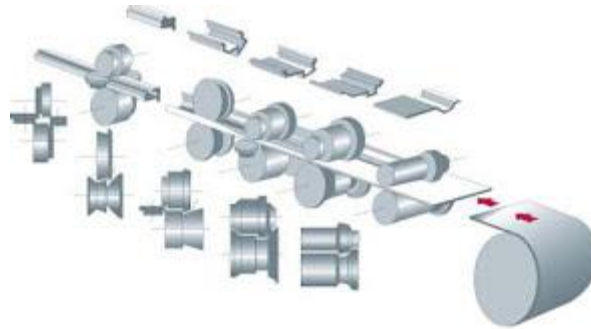
Hemming:

It is an operation in which the edge of the sheet is folded over itself. This process increases stiffness of the part and eliminate sharp edges.



Roll Forming:

It is an continuous bending process used for long work piece. This process uses a series of rolls of various shapes according to the bending process. The metal sheet allows to pass through these rolls and the part is sheared and stacked continuously.



Bend Allowance

Consider a sheet with a 20 mm thickness and a length of 300 mm as shown in Figure 1. We are going to review three bending scenarios with three different bending angles; 60, 90 and 120, and we will calculate K-Factor, Bend Allowance and Bend Deduction for them. The bending tool has a radius of 30 mm which means that our Inside Bend Radius (R) is 30 mm. Let's start with 90 degrees bend which is the most simple scenario.

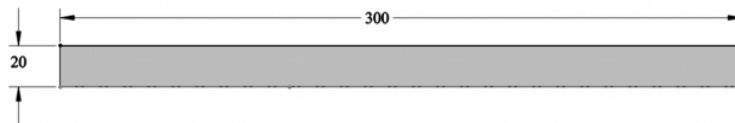
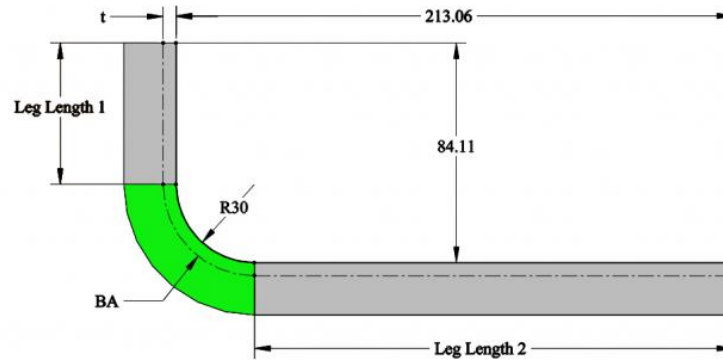


Figure 1: Flat sheet

90 Degrees Bend Angle

Figure 2 illustrates the sheet that is bent with the bend angle of 90 degrees. We will start by calculating the Bend Allowance. From there we can calculate the K-Factor and the Bend Deduction. After bending the sheet we need to do some measurements as shown in Figure 2.



We can calculate the Leg Length 1 and 2 as follows:

$$\text{Leg Length 1} = 84.11 - R = 84.11 - 30 = 54.11$$

$$\text{Leg Length 2} = 213.06 - R = 213.06 - 30 = 183.06$$

At the neutral axis we have:

$$\text{Initial Length} = \text{Leg Length 1} + BA + \text{Leg Length 2}$$

Bend allowance is calculated by using formula:-

Sheet Metal Bend Allowance

Formula

$$BA = A \frac{\pi}{180} (R + KT)$$

BA – Bend Allowance

A – bend angle in deg

R – inside bend radius in m

K – constant

T – material thickness in m

K values:

$$k = 0.33 \text{ for } IR < 2t$$

$$k = 0.50 \text{ for } IR > 2t$$

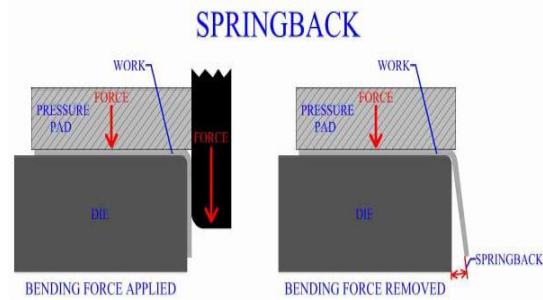
Spring Back:

When the force used to create the bend is removed, the recovery of the elastic region results in the occurrence of **spring back**.

Spring back is the partial recovery of the work from the bend to its geometry before the bending force was applied.

The magnitude of spring back depends largely on the modulus of elasticity and the yield strength of the material.

Typically the results of spring back will only act to increase the bend angle by a few degrees, however, all sheet metal bending processes must consider the factor of spring back.



Methods of Preventing Spring Back:

❖ Over bending :

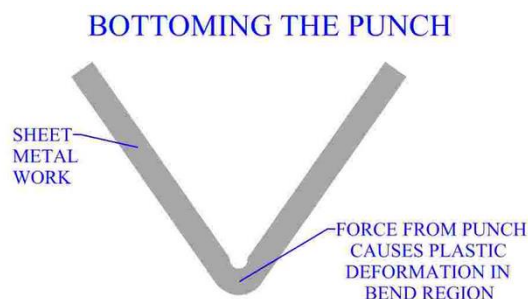
It is used for V Die air Bending . Some more bending is made than required , then due to spring back it results in desired bending.



❖ Corner Setting/ Bottoming :

Spring back is prevented by coining (squeezing) the metal slightly at the corner in order to relieve elastic stresses.

Localized compressive forces between the punch and die in that area will plastically deform the elastic core, preventing spring back. This can be done by applying additional force through the tip of the punch after completion of bending. A technique known as bottoming, or bottoming the punch.



Bending Pressure /Force:

Bending pressures The amount of pressure required to bend the workpiece material depends upon the thickness of the stock, die opening, length of bend, and the amount of coining, bottoming, or ironing used. Air dies require the least pressure of all bending methods. To estimate the pressure required for bending with air die, the following formula, derived from the beam formula may be used:

$$F = (kLst^2) / W$$

F = Bending force required (in tons)

L = Length of the bend part (in inch)

s = Ultimate tensile strength (in ton / in²)

t = Metal thickness

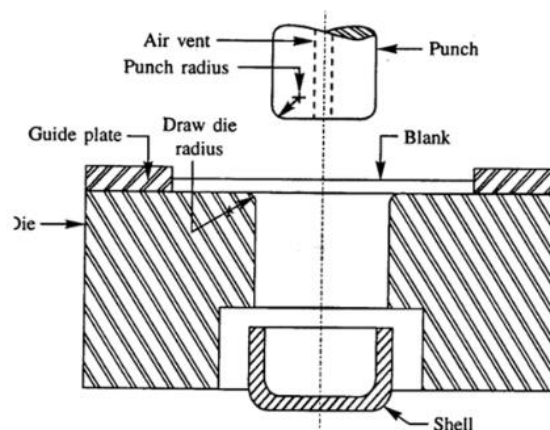
k = Die opening Factor

w =Width between contact points on die

- For U bending
 $k = 2.67$; when Die opening = $8 \times$ Material thickness
 $k = 2.40$; when Die opening = $16 \times$ Material thickness
- For U bending and channel bending
 $k = 1.33$; when Die opening = $8 \times$ Material thickness
 $k = 1.20$; when Die opening = $16 \times$ Material thickness
- For Edge bending
 $k = 0.67$; when Die opening = $8 \times$ Material thickness
 $k = 0.60$; when Die opening = $16 \times$ Material thickness

Drawing

- Drawing is the process of making cups, shells, and similar articles from metal blanks.
- Typical tools used for drawing are shown.



- The set up is similar to that used in blanking except that the punch and die are provided with the necessary rounding at the corners to allow for the smooth flow of metal during drawing.
- The blank is first kept on the die plate. The punch slowly descends on the blank and forces it to take the cup shape formed by the end of the punch, by the time it reaches the bottom of the die.
- When the cup reaches the counter bored portion of the die, the top edge of the cup formed around the punch expands slightly due to the spring back.
- When the punch moves in the return stroke, the cup would be stripped by this counter bored portion.

Shallow Drawing

Shallow drawing is defined as that where the cup height is less than half the diameter.

Deep Drawing

For drawing deeper cups it is necessary to make specific provisions to confine the metal in order to prevent excess wrinkling of the edges.

For this purpose, a blank holder is normally provided on all deep drawing dies.

Drawing Ratio

Measurement of the amount of drawing performed on a sheet metal blank can be quantified. This can be done with the *drawing ratio*. The higher the drawing ratio, the more extreme the amount of deep drawing. Due to the geometry, forces, metal flow and material properties of the work, there is a limit to the amount of deep drawing that can be performed on a sheet metal blank in a single operation. Drawing ratios can help determine the maximum amount of deep drawing possible. The drawing ratio is roughly calculated as,
 $DR = D_b/D_p$.

D_b is the diameter of the blank and D_p is the diameter of the punch. For shapes that are noncircular the maximum diameter is sometimes used, or occasionally drawing ratio is calculated using surface areas. The limit to the drawing ratio for an operation is usually 2 or under. Actual limits to the amount of drawing possible are also dependant upon the depth of drawing, punch radius, die radius, anisotropy of the sheet and the blank's material.

Reduction

Another way to express drawing ratio is the *reduction (r)*. Reduction is measured using the same variables as drawing ratio. Reduction can be calculated by $r = (D_b - D_p)/(D_b)$.

D_b and D_p being blank and punch diameters respectively. Reduction should be .5 or under. Often expressed as the percent reduction $r = (D_b - D_p)/(D_b) \times 100\%$. In this case the reduction should be 50% or under.

Types of Drawing Dies :

- Shallow Drawing when $h/d \leq 0.5$
- Deep Drawing when $h/d > 0.5$

Where ,

h = outside height

d = outside diameter

Percent Reduction and depth of draw :

Percent reduction , $P = 100 (1 - d/D)$

where P = Percentage reduction, d =ID of drawn shell, D = OD of blank

Maximum allowable reduction in

one stroke / one draw ,

Theoretical $\approx 50 \%$

Practical $\approx 40 \%$

Example: The drawing of a 3 inch dia cup from a 5 in dia blank results in a 40% reduction.

Percent reduction , $P = 100 (1 - d/D) = 100 (1 - 3/5) = 40 \%$

Possible number of reduction for a given ratio of shell height to diameter :

Ratio h/d	Numbers of reduction	Reduction			
		1st Draw	2nd Draw	3rd Draw	4th Draw
Upto 0.75	1	40			
0.75 – 1.5	2	40	25		
1.5 – 3.0	3	40	25	15	
3.0 – 4.5	4	40	25	15	10

Blank size determination :

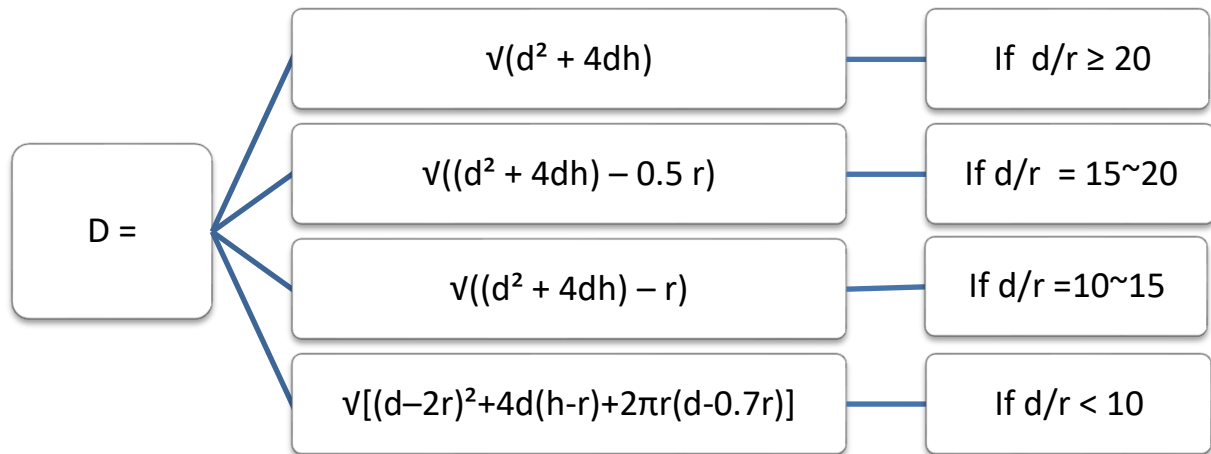
The following equations can be used to calculate the blank size for cylindrical shells of relatively thin metal.

D = Blank Diameter ,

d = Cup/Shell diameter (outside)

h = Cup/Shell height (outside) ,

r = Corner radius of punch



Drawing Force :

The amount of force required to shape a symmetrical cup by drawing can be calculated from the following formula from the ASTM "Die design handbook".

$$P = \pi d t s (D/d - c)$$

Where ,

P = Drawing force/pressure.

d = Cup/shell outside diameter.

t = Metal thickness.

s = Yield strength of the material.

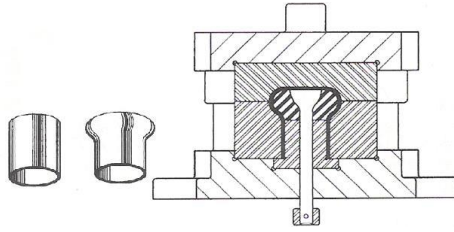
D = Blank diameter.

C = Constant to cover friction and bending(0.6 to 0.7 for ductile materials)

Drawing operations

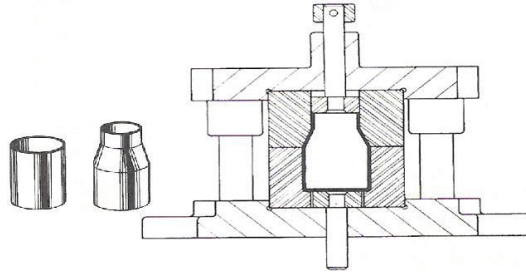
Bulging

Bulging tools expand the bottom of the previously drawn shells. The bulged bottoms of some types of coffee pots are formed in bulging tools



Swaging

In swaging operations, drawn shells or tubes are reduced in diameter for a portion of their lengths.



Flanging

Flanging is a process of bending the sheet edges

The workpiece may have a plane or a curved surface.

The edge may be a straight line or a curve.

Coining

Coining is a very basic type of bending in which the work piece is stamped between the punch and die. The material is put under enough pressure that the punch tip penetrates the material and it begins to flow into the die. This method produces excellent accuracy and repeatability, and does not require sophisticated machines to execute

Embossing

Embossing uses tension to stretch metal into a shallow depression. The die set primarily is composed of a punch and a cavity. The metal's thickness and mechanical properties, along with the forming punch geometry, determine the depth that can be achieved

Forging

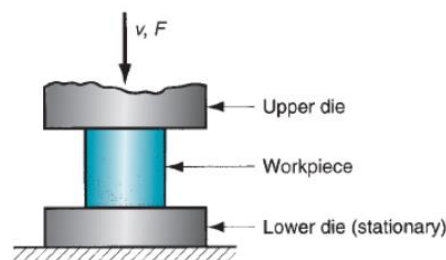
- It is a deformation process in which the work piece is compressed between two dies, using either impact load or hydraulic load (or gradual load) to deform it.

- It is used to make a variety of high-strength components for automotive, aerospace, and other applications. The components include engine crankshafts, connecting rods, gears, aircraft structural components, jet engine turbine parts etc.
- Category based on temperature :cold, warm, hot forging
- Category based on presses: impact load => forging hammer; gradual pressure => forging press
- Category based on type of forming:

Open die forging, close die forging

Open die forging

In open die forging, the work piece is compressed between two flat platens or dies, thus allowing the metal to flow without any restriction in the sideward direction relative to the die surfaces.



A simplest example of open die forging is compression of billet between two flat die halves which is like compression test. This also known as upsetting or upset forging. Basically height decreases and diameter increases.

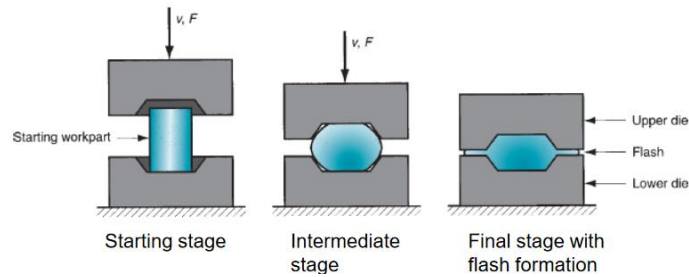
Under ideal conditions, where there is no friction between the billet and die surfaces, homogeneous deformation occurs. In this, the diameter increases uniformly throughout its height.

Closed die forging

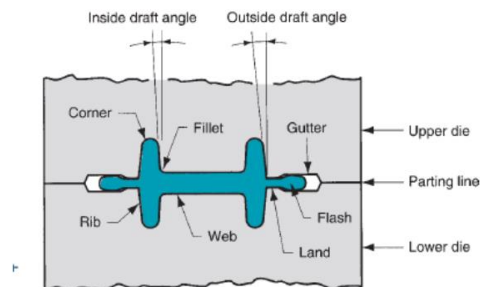
Closed die forging called as impression die forging is performed in dies which has the impression that will be imparted to the work piece through forming.

In the intermediate stage, the initial billet deforms partially giving a bulged shape. During the die full closure, impression is fully filled with deformed billet and further moves out of the impression to form flash.

In multi stage operation, separate die cavities are required for shape change. In the initial stages, uniform distribution of properties and microstructure are seen. In the final stage, actual shape modification is observed. When drop forging is used, several blows of the hammer may be required for each step.



Forging dies:



Parting line: The parting line divides the upper die from the lower die. In other words, it is the plane where the two die halves meet. The selection of parting line affects grain flow in the part, required load, and flash formation.

Draft: It is the amount of taper given on the sides of the part required to remove it from the die.

Draft angles: It is meant for easy removal of part after operation is completed. 3° for Al and Mg parts; 5° to 7° for steel parts.

Webs and ribs: They are thin portions of the forging that is parallel and perpendicular to the parting line. More difficulty is witnessed in forming the part as they become thinner.

Fillet and corner radii: Small radii limits the metal flow and increase stresses on die surfaces during forging.

Flash: The pressure build up because of flash formation is controlled proper design of gutter and flash land.