PRINCIPLES AND PRACTICES OF PRODUCTION TOOLING AND PRODUCT DESIGN

ABHILASH DALAL

Principles and Practices of Production Tooling and Product Design

Principles and Practices of Production Tooling and Product Design

Abhilash Dalal

THE INFO LIBRARY

Published by The InfoLibrary,
4/21B, First Floor, E-Block,
Model Town-II,
New Delhi-110009, India

Principles and Practices of Production Tooling and Product Design
Abhilash Dalal
ISBN: 978-93-5590-917-6

Table of Contents

PRODUCT DESIGN FOR SAND CASTING

Product design considerations, product planning, product development, value analysis, product specification. Role of computer in product design.

Product design for sand casting: design of gating system and risering.

1.1 Product design considerations

Design is a multifaceted process. The various considerations in a good design can be grouped into three categories:

1. Design requirements,

2. Life-cycle issues,

3. Regulatory and social issues.

1. Design requirements

It is obvious that to be satisfactory the design should be demonstrate the required performance. Performance measures each function and behaviour of the design, i.e., how well the device does what it is designed to do.

i) Performance requirements can be divided into following two groups.

• **Functional Performance Requirements:** They address capacity measures such as forces, power, strength, material flows, energy, deflection and efficiency of design, its accuracy, sensitivity etc.

• **Complementary Performance requirements:** They are concerned with the useful life of design, its robustness to factors in the service environment, its reliability and ease, economy and safety of maintenance. Issues such as built-in safety features, noise level of operation, all legal requirements and design codes should be considered.

ii) Physical Requirements: These pertain to such issues as size, shape, weight and surface finish.

iii) Environmental Requirements: There are two separate aspects. The first concerns the service conditions under which the product must operate. The extremes of temperature,noise, corrosive conditions, humidity,dirt,vibration, etc., should be predicted and allowed for in the design. The second aspect of environmental requirements pertains to how the product will behave with regard to maintaining a safe and clean environment, i.e., green design. Among these issues is the disposal of the product when it reaches its useful life.

iv) Aesthetic Requirements: They are concerned with how the product is perceived by a customer because of its colour,shape,surface texture and also include such factors as balance, unity and interest.

v) Manufacturing Technology: Should be intimately connected with product design. There may be restrictions on the manufacturing processes that can be used, because of either selection of material or availability of equipment within the company.

vi) Cost: The final major design requirement is cost. Every design has requirements of an economic nature. These include such issues as product development cost, life cycle product cost, initial product cost,tooling cost and return on investment.

Total Life Cycle

Producibility: Material selection can not be separated from Producibility. There is an intimate connection between design and material selection and the production processes. The objective in this area is a trade-off between the opposing factors of low cost and high durability.

Durability: It is concerned with the number of cycles of possible operation, i.e., the useful life of the product. The current societal issues of energy conservation, material conservation and protection of environment result in new pressures in selection of materials and manufacturing processes. Energy costs once nearly totally ignored in design are now among the most prominent design considerations. Design of materials recycling is becoming more and more important consideration.

Regulatory and Social Issues

Regulatory and Social Issues Specifications and standards have an important influence on design practice. The standards produced by such societies as ASTM, CE, ASME and BIS represent voluntary agreement among more elements of industry. Such as,they often represent minimum or least common denominator standards. When good design requires more than that, it may be necessary to develop company or agency standards. The code of ethics of all professional engineering societies requires the engineer to protect public health and safety.

Increasingly, legislation has been passed to require government agencies to regulate many aspects of safety and health. The designer has to develop the design in such a way to prevent hazardous use of the product in an unintended but foreseeable manner. When unintended use cannot be prevented by functional design, then clear and complete, unambiguous warnings should be permanently attached to the product.

In addition, the designer should be cognizant of all advertising materials, owner's manuals and operating instructions that relate to the product to ensure that the contents of the material are consistent with safe operating procedures and do not promise performance characteristics that are beyond the capability of the design.

An important design consideration is adequate attention to human factors engineering, which uses the sciences of ergonomics, biomechanics and engineering psychology to assure that the design can be operated efficiently by humans.

1.1.1 Product planning and Product development

Product design for both product has been developed in the design department.

• To convert the product design into a product, a manufacturing plan is required. Activity of developing such a plan is called process planning.

• Process planning consists of preparing sets of instructions that describe how to manufacture the product and its parts.

• The task of process planning consists of determining the manufacturing operations required to transform a part from a rough to the finished state specified on the engineering drawing. Also called as operations planning.

• It is the systematic determination of the engineering processes and systems to manufacture a product competitively and economically. It is a detailed specification which lists the operations, tools and facilities. It is commonly accomplished in manufacturing department.

Process Planning Definition

• It Can be defined as "an act of preparing a detailed processing documentation for the manufacture of a piece part or assembly." In according to the *American Society of Tool and Manufacturing Engineers.*

• A process planning is the systematic determination of the methods by which a product is to be manufactured economically and competitively.

• It Consists of devising and selecting , specifying of processes, machine tools and other equipment, transform the raw material into finished product as per the specifications called for by the drawings.

Production planning

• It is concerned with the logistics issues of making the product.

• It is concerned with ordering the materials and obtaining the resources required to make the product in sufficient quantities to satisfy demand for it.

The different steps or specific activities involved in process planning are:

1. Analysis of the finished part requirements as specified in the engineering design.

2. Determining the sequence of operations required.

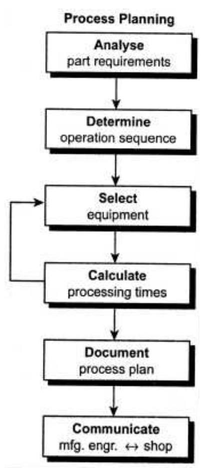
3. Selecting the proper equipment to accomplish the required operations.

4. Calculating the specific operation setup times and cycle times on both machine.

5. Documenting the established process plans.

6. Communicating the manufacturing knowledge to the shop floor.

The above process planning activities are diagrammatically presented in figure.



Process planning activities

1) Analyze Finished Part Requirements

• The process planning is to analyse the finished part requirements as specified in the engineering design.

• Engineering design should be shown either on an engineering drawing or in a CAD model format. Component drawing should be analysed in detail to identify its features, dimensions and tolerance specifications.

• A part's requirement defined by its feature, dimensions and tolerance specifications determines the corresponding processing requirements.

2) Determine Operating Sequence

• Let us determine the sequence of operations required to transform the features, dimensions and tolerances on the part from a rough (initial) to a finished state.

• Basic aim of this step is to determine the type of processing operation that has the capability to generate the various types of features, given the tolerance requirements.

• Two alternative ways of viewing the decision process in determining the sequence of operation are :

• The first view is to consider the processing evolution of the part from the rough (i.e., initial) state to the finished final state. Here like in conventional production shop material is removed or modified on the rough part in stages in order to transform it to the finished part.

• The second view is to consider part evolution from a finished state back to a rough/initial state.

• Here in contrast to the first view, the operation processing is planned by adding material back onto the part. It should be noted that the tolerance specifications are primary factors in determining the sequence of operations.

3) Select "Machines"

Once the appropriate type of manufacturing process has been determined, the next step in process planning is to select appropriate machines equipment and tools to accomplish the required operations.

The following considerations are to be made while selecting a machine:

(i) Economic considerations: Due analysis should be made with respect to the initial cost, maintenance and running cost.

An alternative which results in lower total cost should be selected.

(ii) Production rate and unit cost of production.

(iii) Durability and dependability.

(iv) Lower process rejection.

(v) Minimum set up and put away times.

(vi) Longer productive life of machines or equipment.

(vii) Functional versatility i.e. ability to perform more than one functions.

• Machine selection generally requires determining how the part would be processed on each of the alternative machines so that the best machine can be selected.

• At the machine selection phase, the firm has to decide whether to make (manufacture) or buy (purchase) the component part.

• Break-even analysis is the most convenient method for selecting the optimum method of manufacture or machine a mongst the competing ones.

4) Material Selection Parameters

Primary parameters affecting the choice of a material are given below:

1. Function: Some of the parameters developed for material selection are related to the functions the product must perform in terms of mechanical, physical, electrical and thermal properties of materials.

2. Appearance: The aesthetic value of the material must be considered while selecting the material.

3. Reliability: Important criterion for material selection because of increasing consumer demands for trouble free products.

4. Service life: The length of service life over which the material maintains its desirable characteristics is a very important consideration in material selection.

5. Environment: The environment to which the material is exposed during the product life is a very important consideration, depending on whether the environment is beneficial or harmful.

6. Compatibility: Important factor influencing material selection, especially whenever more than one type of material is used in a product or assembly.

7. Producibility: The ease of producibility of an item is an important parameter in the selection of material.

8. Cost: The cost of the material is a significant factor contributing to the overall cost.

5) Calculate Processing Times

• After an appropriate set of required machines is selected,to calculate the specific operation setup times and cycle times on each machine.

• Determination of setup times requires knowledge of available tooling and the sequence of steps necessary to prepare the machine for processing the given work piece.

• For establishing accurate setup times, detailed knowledge of equipment capability, tooling and shop practice is required.

• Calculation of part processing time requires the determination of the sequence of processing steps on each machine. This activity is often known as out planning.

• After the given machining or processing time is calculated, the appropriate times for part loading, part unloading, machine indexing and other factors involved in one complete cycle for processing a part must be included to compute the expected machine cycle time.

• With the calculated machine cycle time, allowances are added to calculate the standard cycle time for processing one piece.

• With the calculated cycle time appropriate machine rates i.e. the cost of machine ownership use in rupees per unit time are applied to calculate the expected standard cost for the given operation.

6) Document Process Planning

• Having selected the best processing alternatives and associated machines, the next step in process planning is to document clearly all the information in detail.

• Resulting process plan is generally documented as a job routing or operation sheet.Operation sheet is also called “route sheet”, “instruction sheet”, “traveller” or “planner”.

• Route sheet lists the production operations and associated machine tools for each component and sub assembly of the product.

1.1.2 Value analysis

Value Engineering is an organized creative approach to ensure that essential functions of a product or service are provided at minimum overall cost without sacrificing quality and reliability.

Value Analysis and Value Engineering

The difference between value analysis and value engineering lies in time and the phase of product life cycle at which the technique is applied. Value Analysis is applied to the existing product with a view to improve its value. It is analysis after the fact and it is a remedial procedure.

VALUE ENGINEERING PROCESS

Information Phase

This phase is the value engineering team gathers as much information as possible about the program requirements, background, project design, constraints and estimated or projected costs. The team performs functional analysis of systems and subsystems to identify the high cost areas.

The project designer provides additional design data and participates in the initial value engineering team conference.

The survival zone for a product source

Speculative or Creative Phase

The team uses a group interaction methods to identify alternative ideas for accomplishing the function of a system or subsystem.

Evaluation and Analytical Phase

The idea is generated during the speculative or creative phase are screened and evaluated by the team. The ideas showing the greatest potential for cost savings and project improvement.

Development/Recommendation Phase

The team researches the selected ideas and prepares descriptions, sketches and life cycle cost estimates to support the value engineering proposal (VEP) recommendations.

Report Phase

Presents the value engineering proposal is to the government during an oral presentation at the conclusion of the workshop. After the completion of the value engineering workshop, a preliminary value engineering report encompassing the entire value engineering effort is prepared by the value engineering group leader and submitted to the industry management.

Uses of value engineering

1. It is a cost prevention as well as cost elimination technique thus reducing cost of the product.

2. Balance of cost and performance.

3. Increase profit and reduce costs.

4. Helps employees for better understanding of their jobs and orients them towards creative thinking.

Benefits of Value Engineering

• Improving quality management.

• Lowering operation and maintenance costs.

• Simplifying procedures.

• Lowering staff costs.

• Minimizing paperwork.

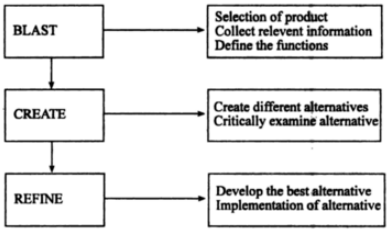
• Increasing procedural efficiency

• Developing value attitudes in staff

• Optimizing construction expenditures

• Competing more successfully in marketplace

Value Analysis procedure



Value Analysis procedure

Procedure of Value Analysis:

Following points should be considered for putting a scheme of value analysis in operation:

• Identification and definition of the problem, i.e. ascertaining whether the customer is being given the full use value and esteem value for the product be purchases and if not, what is required to be done.

• The feasibility of the alternatives and exploring the best method of performing the work at the minimum cost. For this purpose all relevant facts like drawing and design, labour, material specifications, material, overhead and other costs, market competition etc.

• The investment, if any required for the alternative.

• Percentage of the return on new investment. This return should be equal to or more than the expected return on investment.

• Costs resulting indirectly out of a decision to change to alternative like costs of items becoming obsolete cost of training, etc.

• The benefits from the alternative like reduction in costs and increased revenue.

• Recommendation of the final proposal for implementation after considering the above points which will increase use value and or esteem value.

• Value analysis requires a broad organisational framework, active involvement of various departments and a combination of initiative, creative approach, knowledge and mature personality in the person heading the value analysis team which generally includes a design engineer and production engineer, system expert, cost accountant, market analyst and experts from other functional areas.

• For its success, the value analysis team should be base its judge ment upon complete information from all areas of the organisation when cost is incurred, cost benefit analysis, standard costing, work study, market research etc.

• To get willing cooperation from everyone within the organisation, the value analyst should be invite suggestions for performance improvement and elimination of unnecessary costs which should be duly considered.

1.1.3 Product specification

In the exam you will be asked to address at least four of the following:

• Purpose/Function

• Performance

• Market

• Aesthetics/Characteristics

• Quality Standards Safety.

Purpose/function

The first product specification criteria asks we to define the purpose and function of the product by explaining:

• The aim or end-use of the product, e.g. 'A domestic iron with an electrically heated base, that converts water into steam. It can be used as a dry or steam iron to remove creases from clothes'

• How the product will be used or what it should do, e.g. 'A wind-up radio designed for use in less economically developed countries, to enable access to the media where there is no mains electricity and where batteries are too expensive.'

It is important that we fully explain (justify) the purpose and function of the product in order to gain the available marks.

Performance

In the product specification criteria we are asked to explain the performance requirements of the product, materials and components. Make sure that we read again the information given in the exam paper about the product and the materials and components it is made from.

• How the product should perform, example. 'The computer desk should hold a computer screen and peripherals on the desktop'

• How the materials/components should perform, example. 'The laminated board should be durable and robust to support the weight of the computer hardware.'

Market

In this next product specification criteria we need to think about two different aspects of the term 'market'; the retail market for the product and the target market group. The size of the market usually determines the level of production for a product. The retail market for the product has a considerable influence on how it is designed and manufactured.

An example, a tubular steel garden and with an 'engineered' design style might be batch produced for sale in an upmarket designer garden shop. On the other hand, a basic cheap and cheerful plastic biro, with a style little changed since its development in 1938, will be produced in high volume for sale in every High Street shop or supermarket world-wide.

When we analyse a product it is therefore important to look for design clues that can help us decide the product retail market and therefore the kind of user who might buy it. For example, the retail market for a hand blender is likely to be kitchen accessory outlets and the target market will be domestic users or chefs.

The second aspect of the term 'market' is the target market group (TMG), i.e. the users of the product. All products are designed with users in mind. Many manufacturers use extensive market research to establish their TMG, without which it would be almost impossible to design a product.

Establishing the TMG means knowing who we are designing for and what their needs are the users male or female ,able-bodied or disabled, expert or novice users of the product? The type of end-user will help determine the characteristics of a product to enable it to meet user needs and therefore have a good sales potential.

This is the science of designing products for human use, matching the product to the user. An ergonomically designed hand blender, for example, will be designed for comfortable use, to give a good speed of performance and to improve the user's satisfaction when using it. When we analyse a product we will need to decide if the product is made in a specific size to meet the needs of children, women or men, if it needs to be made in specific dimensions to make the product more usable or to improve the users' interaction with it.

Aesthetics/characteristics

Aesthetics play a very important role in the design and marketing of products as the main reason for buying a product is often how it looks and the image it can give the user. For many products appearance and design characteristics are based on the manufacturer's perception of user needs.

When we address the aesthetics/ characteristics of the product in the exam, we need to focus on the aesthetic properties and characteristics that the product should have in order to meet the TMG requirements. We will gain few marks for simply describing the product.

For example, the aesthetics and characteristics of a flat pack self-assembly computer desk include:

• It should be easy to assemble with basic tools.

• It should be easy to transport in flat pack form.

• It should be eye-catching with a contrasting colour scheme in line with design trends.

• It should have ergonomically designed handles so the drawers are easy to open.

• It should have wheels so its easy to move from room to room.

• It should have a laminated surface that is easy to wipe dean.

• It should have a pull-out keyboard desk at an ergonomic height.

• It should have a work area suitable for a range of computer peripherals.

In the Product Analysis exam we would not be expected to produce a long list of points about the aesthetics/characteristics of a product, because the maximum marks available for any of the specification headings is four. To achieve four marks for aesthetics/ characteristics, we would therefore need to provide four different, fully explained and appropriate specification points.

Quality standards:

When we explain how the product can meet the quality standards we need to refer to the size, dimensions and the use of tolerances. We should also refer to the relationship between quality, cost and scale of production. For example, a one-off product may use more expensive or unusual materials than a high volume product, although both levels of production would make use of a quality system of some kind.

All products need to be manufactured to the correct size and dimensions to ensure that, the component parts fit together and the product is capable of performing its function. Therefore, all the component parts need to be designed and manufactured to a high standard with precision and accuracy. This is where the standards such as British Standards (BS) comes in. There is an enormous range of published BS which are used by the manufacturers to help them produce a quality product. Although we are not expected to quote a BS number, it is always preferable to explain the relevance of a standard, such as the use of a BS to test the strength and stability of the furniture.

Successful manufacturers incorporate the use of a quality management system (QMS) in product design and manufacture in order to achieve a quality product. Therefore, we could refer to the use of a QMS or total quality management (TQM). One aspect of TQM is the use of quality control (QC) checks which use agreed tolerances to check the accuracy of dimensions and sizes in order to manufacture identical products.

We could also quote the use of ISO 9000 which is an internationally agreed set of standards for a QMS. ISO 9000 ensures that a manufacturing customer receives the product that has been agreed. For example, a domestic iron manufacturer that is ISO registered would only buy in component parts from a supplier that is also ISO registered because, they would know that the components would meet an agreed BS. Any manufacturer that has a QMS in place can apply for a KiteMark or offer a warranty or guarantee of product quality.

Safety:

Safety is an essential feature of all manufactured products, many of which have to comply with the safety legislation before they are sold. Safety standards are set by British Standards (BS) to ensure the safety of the user. Products such as toys, garden tools and children's outdoor play equipment are rigorously tested to ensure that they will not cause injury through normal use or misuse. This includes the safety of components like hinges, catches, paints and finishes which must be non-toxic.

Products like kettles and toasters are tested to ensure electrical safety. A domestic iron needs to have tamper proof screws to prevent access to electrical connections. A product that displays a Kite-Mark is independently tested at regular intervals to make sure it complies with a relevant safety standard.

The Kite-Mark symbol shows the potential customers that, the product is safe and reliable. When a product is sold in the European Union (EU) it has to meet the safety standards set by the EU. Manufacturers who claim to meet these standards can use the CE mark but, if the product is found not to comply with the safety standard then, it can be seized and the manufacturer, importer or supplier is prosecuted.

For example, British Standards (BS) in relation to food processors, might be related to the safety of the cutting blades or related to electrical safety when blending liquids. We will not be expected to quote a ES number but, we must make sure that, what we explain is appropriate to the product.

Other points about safety needs explaining, (e.g.) rather than stating that, the product needs to have a Kite-Mark, we could explain that, the product needs to meet a relevant safety standard and pass regular tests in manufacture to be awarded a Kite-Mark.

1.2 Role of computer in product design

Computers have made a massive impact on the accuracy and speed in which the products can be made. It is now possible to manufacture a high-quality outcomes in a short period of time when previously the people would have taken a number of days to make the same products with no guarantee that they would all be of same standard.

The conventional design process has been accomplished on drawing boards with the design being documented in the form of detailed engineering drawing.

The conventional design process, also known as Shigley model, consists of the following six steps/phases:

1.Recognition of need

2.Identification of problem

3.Synthesis

4.Analysis and optimization

5.Evaluation

6.Presentation

In CAD, the design related tasks are performed by a modern CAD system. The four stages or functional areas of a CAD design process are:

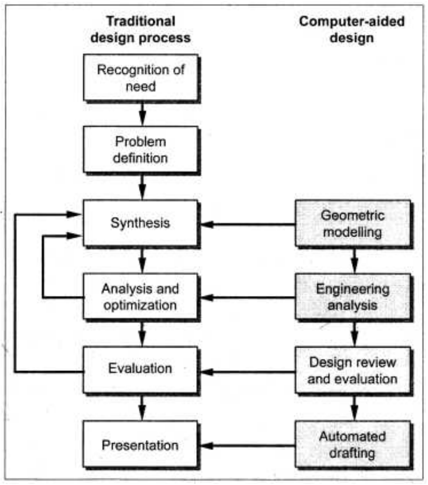
1.Geometric modelling,

2.Design analysis and optimization,

3.Design review and evaluation,

4.Documentation and drafting.

The above four areas correspond to the final four phases of Shigley's general design process, as illustrated in figure below.



Application of computers to the design process

1. Geometric Modelling

• The geometric modelling is concerned with computer compatible mathematical description of geometry of an object.

• The mathematic description of geometry should be such that:

(i)The image of the object can be displayed and manipulated in the computer terminal;

(ii)Modification on the geometry of the object can be done easily;

(iii)It can be stored in the computer memory; and (iv)It can be retrieved back on the computer screen for review, analysis or alteration.

• In geometric modelling, three types of commands are used. They are:

(i)Commands used to generate basic geometric entities like points, lines, circles, etc.

(ii)Commands used to do manipulation work like scaling, translation, rotation, etc.

• The models can be represented in three different ways:

(i)Wire-frame,

(ii)Surface,

(iii)Solid modelling.

2.Design Analysis and Optimization

• Once the graphic model is created, the design is subjected to engineering analysis. This phase may consist of analysing stresses, strains, deflections and other parameters.

• The analysis can be done either by using specific program generated for it or by using general purpose software commercially available in the market.

• Nowadays sophisticated packages (such as ANSYS, Pro-E, CATIA) having capabilities are available to compute the various performance parameters accurately.

• Because of the relative ease with which such analysis can be made, designers are increasingly willing to thoroughly analyse a design before it moves onto production.

• Experiments and field measurements may be necessary to determine the effects of loads, temperature and other variables.

3. Design Review and Evaluation

• The next phase is review and evaluation to check for any interference between various components in order to avoid difficulties during assembly or use of the part and whether the moving members such as linkages are going to operate as intended.

• By using the layering procedure, every stage of production can be checked; by using animation, the working of the mechanism can be checked. Also, during design review, the part is precisely dimensioned and toleranced as required for manufacturing it.

4.Documentation and Drafting

• After analysis and review, the design is reproduced by automated drafting machines for documentation and reference.

• In this phase, detailed and working drawing are developed and printed.

• The important features of automated drafting are automated dimensioning, scaling of the drawing, development of generating sectional views, enlargement of minute part details and ability to generate different views of the object (like orthographic, oblique, isometric and perspective views).

Computer Aided Design (CAD)

CAD is a system that allows designers to create solutions to problems within a computer program through the use of illustrations. Designs can be modelled in 3D and manipulated time and again from all angles.

Computer Aided Manufacture (CAM)

CAM is the term used to describe any activity where a machine is programmed with several instructions to produce a component from a raw material. CAD packages are commonly used through an interface software to drive the special machine codes that in turn tells the machine what to do and where to cut and shape the materials.

A car has many thousands of components that all need to behave in the specific ways. Cars have become increasingly complicated, but each small piece of the engine or controls is relatively simple to make. This is because the machines assemble and shape the raw materials or assist people in assembling them. Imagine how difficult it would be for one or two skilled people to make these cars without the assistance of machines, robots, computers and diagnostic systems.

It is easy to imagine how a craftsman, equipped with hand tools, can manipulate the hardwood into a well made table or use these metals to create some fine jewellery. In these cases, the human touch and sensitivity toward the aesthetics of the pieces is easy to appreciate. However, the computers are needed when manufacturing to consistently satisfy high demand and reproducible quality.

Sophisticated computer systems can be integrated together to the monitor in every aspect of a manufacturing process. Designs can be modified time and again without the need to repeat all the drawings and computers can hold vast amounts of technical data with great accuracy. This information can be fed into the manufacturing cell where several machines or robots can carry out the precise tasks time and again precisely and with accuracy.

Computer Aided Industrial Design (CAID)

In CAID, the computer designs are more commonly modeled in 3D and rendered to make the designs look as real as possible. The software is very sophisticated and more advanced than that found in schools.

Computer Aided Market Analysis (CAMA)

When companies wish to monitor the consumer behaviour, they may use the CAMA data to analyze their sales. For example, the use of bonus and reward cards in supermarkets provides the information about what products the people are buying and when they are buying them. This helps the designers target a new products.

Computer Numerical Control (CNC)

This is the control of machines using numbers or digital information. This can be provided manually or through a computer. Generally this is used for drilling and milling procedures.

Flexible Manufacturing Systems (FMS)

A flexible manufacturing system involves the use of pre-programmed machines and computers to carry out a series of tasks and operations. They can be programmed so that a different set of operations can be carried out as the designs change.

Computer Aided Administration (CAA)

Here data can be collected and accessed in a quick and effective way to assist manufacturing or information management. For example, our school may have an electronic registration system or the company may use clocking-in identification to monitor the whereabouts of its employees.

Automatic Guided Vehicle (AGV)

This is an unmanned vehicle that follows the pre-programmed route around a factory floor or warehouse.

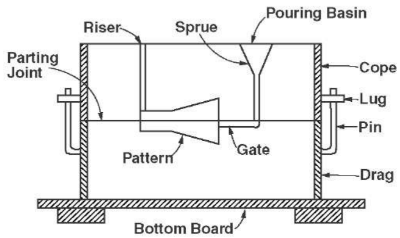
1.3 Product design for sand casting: Design of gating system

Casting is that the method of manufacturing metal elements by gushing liquified metal into the mold cavity of the desired form and permitting the metal to solidify. The coagulated metal piece is termed as “casting”

Sand casting is used to make large parts (typically Iron, but also Bronze, Brass, Aluminum). Molten metal is poured into a mold cavity formed out of sand (natural or synthetic).

The most generally used casting method, utilizes expendable sand molds to form complex metal parts that can be made of nearly any alloy. Because the sand mold must be destroyed in order to remove the part, called the casting, sand casting typically has a low production rate.

The sand casting method involves the utilization of a chamber, metal, pattern and sand mold. The metal is melted within the chamber so ladled and poured into the cavity of the sand mold, that is made by the pattern. The sand mold separates on a parting line and also the coagulated casting may be removed.



Sand casting

The sand casting is used to produce a wide variety of metal components with complex geometries. These parts can vary greatly in weight and size, ranging from a couple ounces to many tons. Some smaller sand cast parts include components as gears, pulleys, connecting rods ,crankshafts and propellers.

Many applications include housings for large equipment and heavy machine bases. Sand casting is also common in producing automobile components, this is engine blocks, like engine manifolds, cylinder heads and transmission cases.

Steps involved in making a casting

- Making mold cavity.
- Material is first liquefied by properly heating it in a suitable furnace.
- Liquid is poured into a prepared mold cavity.

• Allowed to solidify.

• Product is taken out of the mold cavity, trimmed and made to shape.

We should concentrate on the following for successful casting operation:

(i) Preparation of moulds of patterns,

(ii) Melting and pouring of the liquefied metal,

(iii) Solidification and further cooling to room temperature,

(iv) Defects and inspection.

Important casting terms

Flask:

A wood frame or metal without fixed to top or bottom, in which the mold is formed. In depending upon the position of the flask in the molding structure, it is referred to by different names such as:

• Cope – Upper molding flask.

• Drag – Lower molding flask.

• Cheek – Intermediate molding flask used in three piece molding.

Parting line:

The dividing line between the two molding flasks that makes up the mold.

Pattern:

It is the replica of the final object to be made. The mold cavity is made within the help of pattern.

Molding sand:

A molding sand, which binds strongly without losing its permeability to air or gases. It is a mixture of silica sand, clay and moisture in appropriate proportions.

Core:

A separate part of the mold, made of sand and commonly baked, which is used to create openings and various shaped cavities in the castings.

Facing sand:

The minimum amount of carbonaceous material sprinkled on the inner surface of the mold cavity to give a better surface finish to the castings.

Sprue:

The passage through which the molten metal, from the pouring basin, reaches the mold cavity. In many cases, it controls the flow of metal into the mold.

Pouring basin:

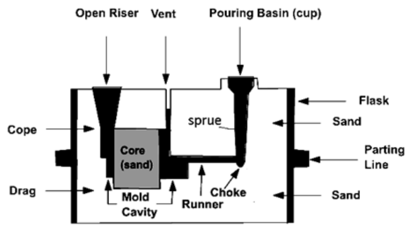
A minimum funnel shaped cavity at the top of the mold into which the molten metal is poured.

Gate:

A channel through which the molten metal enter the mold cavity.

Runner:

The molten metal is carried from the sprue to the gate.



Core Stand

Chaplets:

Chaplets are used to support the cores inside the mold cavity to take care of its own weight and overcome the metallostatic force.

Vent:

A small opening in the mold to facilitate escape of air and gases.

GATING DESIGN AND ANALYSIS:

A well-designed for gating system is a perquisite for achieving perfect casting. The key objective of a gating system is to ensure smooth and complete flow of molten metal from ladle to the casting cavity.

Elements of gating system

The main elements include well, pouring basin, sprue, sprue, runner and ingate. Gating systems can be classified based on the position of the parting plane and also on the basis of ingate position as follows,

i) Horizontal systems

Suitable for flat castings filled under gravity and commonly used in sand casting of ferrous metals and gravity die casting of non-ferrous metals.

ii) Vertical systems

A suitable for tall castings and used in high pressure sand and shell mold and die casting processes.

iii) Top gating systems

Wherein the molten metal enters at the top of the casting promoting directional solidification from bottom to top of the casting.

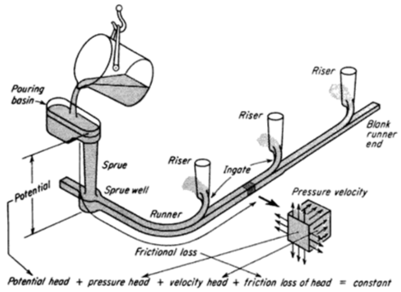
iv) Bottom gating systems

wherein the molten metal enters from the bottom and fills the mold slowly with minimal

v) Middle gating systems

These combine the features of top and bottom systems,disturbances.

The number and position of ingate is very important in designing the gating systems. The diagram given below describes the different considerations in the channel layout in horizontal gating system with side ingates.



Gating Design

Heuristics for ingate location

i) Side feeders

When side feeders are used their efficiency can be improved by filling the first stream of hot molten metal through ingates, thereby reducing the fettling effort and subsequent marks on the casting.

ii) Thick sections

This allows molten metal to flow to other sections with minimal cooling reducing breakage during fettling of ingates.

iii) Clear path

In case of sand casting it is essential to allow the metal to flow with minimal obstructions or direction changes to curb turbulence. In such systems ingates should never be placed opposite to a core.

iv) Low free fall

Here the ingates are located where the free fall of molten metal in the mold cavity is low thereby minimizing oxidation and erosion at the point of impact.

It is also essential to use adequate number of ingates ensuring that the distance of flow between the ingate and any point filled by that ingate is always less than the fluidity distance. Usually the sprue directs the flow of metal from the basin to the runners and the ingates.

The location of sprues depends on the following factors,

i) Flow distance

It should be minimize the total flow distance with the gating channel to give maximum yield and minimum heat loss.

ii) Heat concentration

It should be located away from hot spots in the casting

iii) Mold layout

It should be positioned to minimize the size of the bounding box, which encloses the entire casting.

Hick sections

This allows molten metal to flow to other sections with minimal cooling reducing breakage during fettling of ingates.

Optimal Filling Time

Filling time is very crucial in casting as it decides the final quality of product. A slow fill gives cold shuts and mis runs while a fast fill can lead to solid and gaseous inclusions. The ideal filling time is a function of cast metal, weight, minimum section thickness and pouring temperature and can be expressed as,

$t_f = K_0 (K_f L_f / 1000) (K_s + K_t t / 20) (K_w W)P$

Where,

t_f - Casting time,

W - The weight in kg,

t -Section thickness in mm.

L_f - Fluidity length in mm,

Where as K_0, K_f , K_s , K_t , K_w are all coefficients.

The value of these coefficients can be set to achieve a perfect metal-process combination in specific castings. The velocity of molten metal also crucial role in deciding the optimal filling time. The velocity usually varies within the gating channels and the mold cavity.

It depends majorly on two factors,

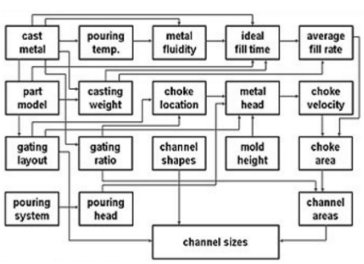
• Metallostatic pressure

• Gating ratio

Gating Element Design

A gating system is designed to fill the casting in prescribed time keeping a constant level of liquid metal in the basin to achieve a controlled flow rate via the choke, a small cross section in system that regulates the flow rate.

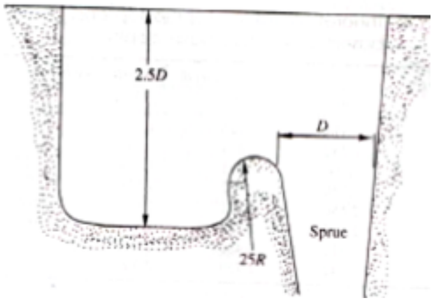
The dimensions and shape of different elements in the gating systems is determined as follows,



Gating systems

i) Sprue

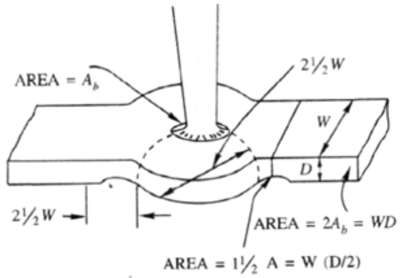
It is a circular cross-section minimizing turbulence and heat loss and its area is quantified from choke area and gating ratio. Ideally it should be large at top and small at bottom.



Sprue

ii) Sprue well :

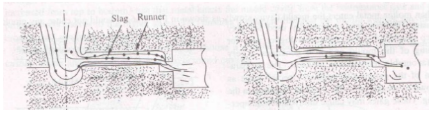
It is designed to restrict the free fall of molten metal by directing it in a right angle towards the runner. It aids in reducing turbulence and air aspiration. Ideally it should be shaped cylindrically having diameter twice as that of sprue exit and depth twice of runner.



Sprue well

iii) Runner

Mainly slows down the molten metal that speeds during the free fall from sprue to the ingate. The cross section are of a runner should be greater than the sprue exit. It should also be able to fill completely before allowing the metal to enter the ingates.

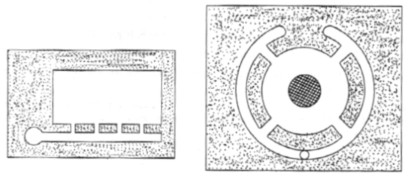


Runner full Runner partially full

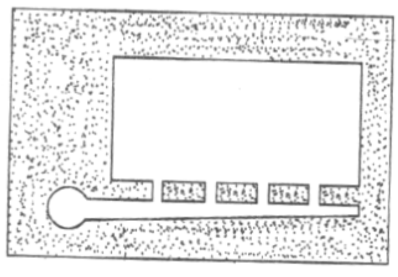
In systems where more than one ingate is present, it is recommended that the runner cross section area should be lowered after each ingate connection to ensure smooth flow.

iv) Ingate :

The ingate can be considered as a weir with no reduction in cross section of the stream at the gate. Then the rate of flow of molten metal through the gates depends on the free height of the metal in the runner and the gate area and the velocity with which metal is flowing in the runner.



Multi ingates feeding the various parts of a casting



Designed to uniform flow through all gates

The following points should be kept in mind while choosing the positioning of the ingates.

• Ingate should not be located near a protruding part of the mould to avoid the striking of vertical mould walls by molten metal stream.

• Ingates should be preferably be placed along the longitudinal axis of the mould wall.

• It should not be placed near a core print or a chill.

• Ingate cross sectional area should preferably be smaller than the smallest thickness of the casting so that the ingates solidify first and isolate the casting from the gating system. This would reduce the possibility of air aspiration through gating system in case of metal shrinkage.

• It is possible that the farthest gate from the sprue is likely to flow more metal than others,particularly in the case of unpressurized system. To make for more uniform flow through all the gates, the runner area should be reduced progressively after each ingate, such that restriction on the metal flow would be provided.

1.3.1 Risering

Riser

Riser is a source of extra metal which flows from riser to mold cavity to compensate for shrinkage which takes place in the casting when it starts solidifying. Without a riser heavier parts of the casting will have shrinkage defects, either on the surface or internally.

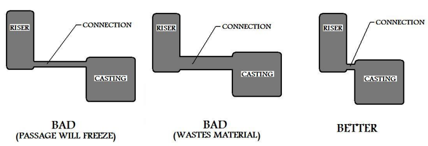
Risers are known by various names as metal reservoir, feeders or headers. Shrinkage in a mold, from the time of pouring to final casting, occurs in three stages.

• During the liquid state.

• During the transformation from liquid to solid.

• During the solid state.

The first stage of shrinkage is being compensated by the feeders or the gating system. For the second stage of shrinkage risers are required. Risers are normally placed at that portion of the casting which is last to freeze. A riser must stay in liquid state at least as long as the casting and must be able to feed the casting during this time.



Final riser casting

Functions Of Risers

Riser serves a dual purpose:

A provides reservoir of liquid metal, compensating for liquid contraction. (Pattern makers shrinkage allowance compensates for solid contraction, where as liquid contraction is compensated by the molten metal in the riser)

Acts as a reservoir of heat which aids in establishing proper temperature gradient so that directional solidification takes place. The riser size for a given application depends on the metal poured. For the example, grey iron requires little or no riser.

3. White iron, Steel and more non ferrous alloys, which have long freezing ranges require larger risers. To be effective, a riser must freeze later than the casting that it feeds and it must supply sufficient metal to compensate contraction from the liquid to solid state.

Types Of Riser

Risers are classified as open risers and blind risers.

Open riser:

The open riser is usually placed on the top of the casting or at the parting plane. The upper surface of riser is open to the atmosphere. So this riser derives feeding pressure from the atmosphere and from the force of gravity on the metal contained in the riser.

Blind riser:

Blind riser is recommended with skin forming metals and alloys. The riser is located by the side of the mould cavity, not necessarily at the top of it. The riser is not exposed to atmosphere and shape of the riser is such that it presents less surface area for a given volume, since it requires more then time to solidify or will be in fluid state for a longer time.

Once the metal is poured and a skin is formed, atmospheric effect is not present on the liquid metal present in the riser. Although a dry sand core is placed at the top of the blind riser and it provides access to atmospheric air to the molten metal.

As the casting starts solidifying, vacuum is created and draws liquid metal from the riser to compensate the same. The yield will be high with blind riser and it will have longer feeding distances. The dry sand core gets heated and will not allow the metal in the riser to solidify early and thus establishes atmospheric pressure on metal.

Design Requirements of Risers

Riser size:

A sound casting riser must be last to freeze. The ratio of the riser must be greater than that of the casting. Although, when this condition does not meet the metal in the riser can be kept in liquid state by heating it externally or using exothermic materials in the risers.

Riser placement:

The spacing of risers in the casting must be considered by effectively calculating the feeding distance of the risers.

Riser shape:

The cylindrical risers are recommended for most of the castings as spherical risers, it considers as best, are difficult to cast. To increase volume/surface area ratio the bottom of the riser can be shaped as hemisphere.

FORGING DESIGN AND SHEET METAL WORKING

Forging design: allowances, Die design for drop forging, Design of flash and gutter, Upset forging die design.

Sheet metal working: Design consideration for shearing, Blanking piercing, Deep drawing operation, Die design for sheet metal operations, Progressive and compound die, strippers , stops, strip layout.

2.1 Forging design: Allowances

Forging design consideration:

Parting Plane:

A parting plane is the plane at which the two die halves of the forging meet. It could be a simple plane or irregularly bent, depending on the shape of the forging.

• The parting plane should be the largest cross sectional area of the forging, since it is easier to spread the metal than to force into deep pockets.

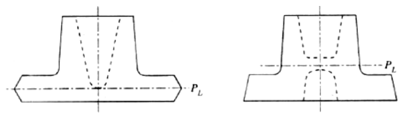
• A flat parting plane is more economical.

• It should be chosen in such a way that equal amount of material is located in each of the two die halves.

• It may be required to put more metal into the top die half since metal would flow more easily in top half.

• If the punching of hole, which is perpendicular to the parting plane, is required then it may be necessary to choose a parting plane which distributes the hole evenly and provides sufficient strength to the punch. The below mentioned figure shows one possible parting plane which simplifies the lower part of the die.

But the punch in the upper die half becomes excessively long and may buckle. By changing the parting line as shown in the next figure, it is possible to punch from both sides, thus reducing the machining. This also provides a smaller height to diameter for the punch increasing its rigidity.



Parting line to reduce the depth of a punched hole

Draft:

It is the taper put on all the forging sides arranged parallel to the travel of the press slide or hammer ram. This makes it easier for the metal to fill up the working volume of the die impressions and facilitates the removal of the forging.

Standard practice indicates the use of 2 to 12^0 draft angles depending on the type of die, rib height and the material to be processed. Internal surfaces require more draft than external surfaces. The forgings of non-ferrous alloys need smaller drafts than the steel ones.

In upset forgings, the draft problem is minimized because the part is held securely by the gripper die during the punch withdrawal and the gripper itself gets opened to release the component. So for upset forgings smaller value of draft angle is considered.

Table: Recommended draft angles

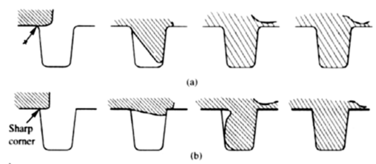
Draft position	**Height or Depth (mm)**	**Drop forgings**		**Upset forging**	
		Normal degree	**Close degree**	**Normal degree**	**Close degree**
Outside	Up to 25	5 to 7	3 to 7	3 to 5	2 to 4
	Above 25	5 to 10	3 to 7		
Inside	Up to 25	7 to 10	5 to 8	5 to 7	4 to 6
	Above 25	8 to 12	5 to 9		

Fillet and Corner Radii:

Since forging involves flow of metal in orderly manner, therefore it is necessary to provide a streamlined path for the flow of metal so that defects' free forging is produced. When two or more surfaces meet, a corner is formed which restricts the flow of metal. These corners are rounded off to improve the flow of metal.

Fillets are for rounding off the internal angles, whereas corner is that of the external angle. Let us consider the flow of metal over a corner as shown in the figure (a). Because of large corner radius provided, metal is allowed to flow smoothly into the pocket. But when corner radius is small or not provided as in figure (b), the metal flow is first hindered and when it finally enters the cavity, the metal would fold back against itself forming a defect called lap or cold shut.

Nominal fillets and corner radii are taken from the tables to suit the weight and required accuracy of the forgings. Sharp fillet sand radii increase the tendency towards forging defects and accelerate the die wear. To avoid this fillets are taken to be larger than corner radii.



Effect of corner radius on the flow of metal

Table: Recommended fillet and corner radii for drop forgings

Depth or Height, mm	Fillet, mm	Corner radius, mm
15	5	2.5
25	8	4.0
40	12	4.5
50	15	5.0
65	18	5.5
75	20	6.0

Table: Recommended fillet and corner radii for upset forgings

Upset diameter /Stock diameter	Filler, mm	Corner radius, mm
Up to 1.25	6.5	6.5
1.25 to 3.00	3.5	3.5
Over 3.00	3.0	3.0

Machining Allowance

● Design features that promote easy forging add to the metal that must be machined away.

● The machining allowance should allow for the worst case buildup of draft, radii and all tolerances.

● Extra metal is provided to keep critical machined surfaces away from the grain flow pattern that occurs in the flash region near the parting line.

● Machining or finishing allowances are added to external dimensions and subtracted from internal dimensions.

Machining allowances refer to the amount of material that is to be machined from the forging to obtain the finished part Allowances always mean extra material and they increase the costs. Machining allowances are dependent on the material forged on the type of production unit being used, on the maximum dimensions of the part and on customer requirements. Therefore an agreement must be made between the forger and the customer.

Machining allowance values are not given in the current EU standards. The recommended European values are between minimum 1 mm and maximum 5 mm, per surface.

In SKODA Auto forge shop practice, the amount of machining allowances is chosen according to the following considerations:

• Type of input bar (peeled bar - rolled bar).

• Tool design (accommodation of excess billet material, position of parting plane).

• Technical state of the forging machine (tolerances, mismatch,temperature stability).

• Billet shape, volume and precision of dimensions (method of bar separation).

Tolerances

The tolerances summarized below should be considered as guidelines. However, adjustment to these absolute values of tolerance can be made for reasons of either manufacturing economy or the component's function. These are applied to impression die forgings made in two-part die sets.

Length and width tolerances:

Dimensions parallel to the parting plane and perpendicular to die motion are provided with length and width tolerances. Length and width tolerances are commonly specified at +0.3 percent of each dimension, rounded off to the next higher 0.5 mm.

Die-wear tolerances:

These tolerances meant by only to dimensions generally parallel to the parting plane and perpendicular to die motion. The corresponding variations parallel to die motion are included in die-closure tolerances. Die-wear tolerances are generally additive to the external dimensions and subtractive to the internal dimensions. Table shows typical recommended tolerances.

To implement these tolerances in the design, each horizontal dimension is multiplied by the appropriate factor and rounded off to the next higher 0.5 mm.

Alloy family	Percentage (%)
Aluminum, 2014	0.4
Aluminum, 7075	0.7
Magnesium	0.6
Brass and copper	0.2
Mild steel	0.4
Alloy steel	0.5
Martensitic stainless steel	0.6
Austenitic stainless steel	0.7

Titanium	0.9
Super alloys	0.8
Refractory alloys	1.2

Die-closure tolerances:

Dimensions parallel to die motion between opposite sides of a forging are affected by failure of the two die halves to close precisely. The plus tolerances on such dimensions are shown in Table. There is no minus tolerance in this category.

	Forging size: Projected area at parting line, mm						
Alloy family	To 65 Cm^2	To $135cm^2$	To $320cm^2$	To 650 cm^2	To $3200cm^2$	To 6450 cm^2	Over $6450cm^2$
Aluminum, Magnesium, And brass	0.8	0.8	1.6	2.4	3.2	4.8	6.3
Steel	0.8	1.6	2.4	3.2	4.0	4.8	6.3
Martensitic Stainless steel	0.8	1.6	2.4	3.2	4.8	6.3	8.0
Austenitic Stainless steel	1.6	2.4	3.2	4.0	4.8	6.3	8.0
Titanium	1.6	2.4	3.2	4.8	6.3	8.0	9.5

Refractory alloys	2.4	3.2	4.0	4.8	6.3	8.0	9.5

Match tolerances:

A lateral shift of one die half with respect to the other half, generally moves all other features on opposite sides of the forging. Table summarizes typical tolerances with respect to piece weight and material.

	Forging size: Weight after trimming(mm)							
Alloy family	To 2.3 kg	To 11kg	To 23 kg	To 45kg	To 90kg	To 225 kg	To 450 kg	Over 450 kg
Aluminum, magnesium, brass, And steel	0.4	0.8	1.2	1.6	2.4	3.2	4.0	4.8
Stainless steel, Titanium and super alloys	0.8	1.2	1.6	2.4	3.2	4.0	4.8	6.3
Refraction alloys	1.6	2.4	3.2	4.0	4.8	6.3	8.0	9.5

Straightness tolerances: In relatively long, a thin part, a typical recommended straightness tolerance is 0.3 percent of the length.

Flash-extension tolerances: The most common flash-removal method is the use of punching operation in contoured dies. Recommended conventional flash-extension tolerances are given in table these are appropriate when this procedure is acceptable.

Forging size: Weight after trimming, mm							
Alloy family	To	To	To	To	To	To	To]	Over

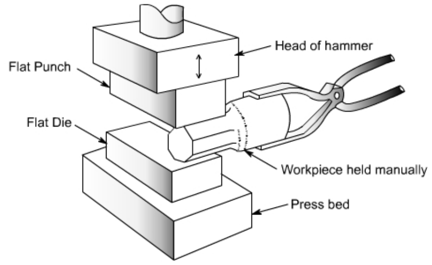
	5kg	10kg	25kg	50 kg	100 kg	250 kg	500 kg	500 kg
Aluminum, magnesium, brass and steel	0.8	1.6	2.4	3.2	4.8	6.3	8.0	9.5
Stainless steel, titanium and super alloys	1.6	2.4	3.2	4.8	6.3	8.0	9.5	12.7
Refractory alloys	3.2	4.8	6.3	8.0	9.5	12.7	15.8	19

Total tolerances: In a forging drawing, the tolerances provided for each dimension whether it is plus or minus, are arithmetic sums of all individual tolerances that apply to the surfaces involved. Dimensions for forging should be provided in such a way that enough metal will be available on every surface to satisfy all functional requirements of the finished part.

2.2 Die design for drop forging

This is the closed impression dies by means drop hammer here the force for shaping the component is applied in a series of blows.

A drop forging utilizes a closed impression die to obtain the desire shape of the component , the shaping is done by the repeated hammering given to the material in the die cavity. The equipment use for delivering for blows are called drop hammers.



Drop forging

The drop forging die consists of two halves. The lower halve of the die is fixed to the anvil of the machine , while the upper halve is fixed to ram. The heated stock is kept in the lower die, while the ram delivers 4-5 blows on the metal spreads and completely fills in the die cavity. When the two die of halves closed the complete is formed.

The typical products obtained in drop forging are cranks, crank shaft, connecting rods, wrench, crane hooks etc. The types of operations are fullering, bending, blocking ,edging, finishing and trimming etc.

Open-die drop forging

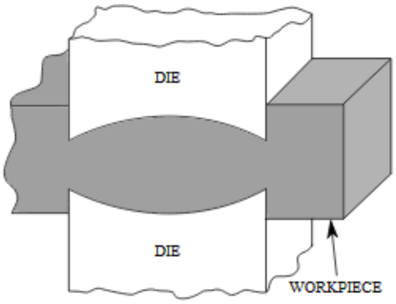
Open-die forging is also known as smith forging. In open-die forging, a hammer strikes and deforms the workpiece, which is placed on a stationary anvil. Open-die forging gets its name from the fact that the dies do not enclose the workpiece, allowing it to flow except where contacted by the dies. The operator therefore needs to orient and position the workpiece to get the desired shape.

The dies are usually flat in shape, but some have a specially shaped surface for specialized operations. For example, a die may have a round, concave or convex surface or be a tool to form holes or be a cut-off tool. Open-die forgings can be worked into shapes which include discs, hubs, blocks, shafts, sleeves, cylinders, flats, hexes, rounds, plate and some custom shapes.

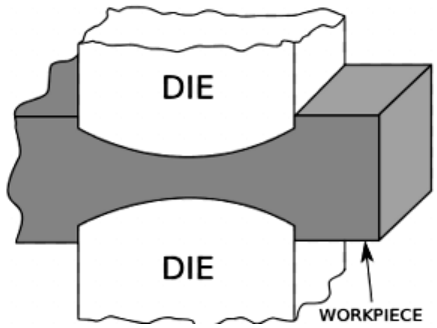
Open-die forging lends itself to short runs and is appropriate for art smithing and custom work. In some cases, open-die forging may be employed to rough-shape ingots to prepare them for subsequent operations. Open-die forging may also orient the grain to increase strength in the required direction.

A "Cogging" is the successive deformation of a bar along its length using an open-die drop forge. It is commonly used to work a piece of raw material to the proper thickness. Once the proper thickness is achieved the proper width is achieved via "edging". "Edging" is the process of concentrating material using a concave shaped open-die.

The process is called "edging" because it is usually carried out on the ends of the workpiece. "Fullering" is a similar process that thins out sections of the forging using a convex shaped die. These processes prepare the workpieces for further forging processes.



Edging



Fullering

Impression-die forging

Impression-die forging is also called "closed-die forging". In impression-die forging, the metal is placed in a die resembling a mold, which is attached to an anvil. Usually, the hammer die is shaped as well.

The hammer is then dropped on the workpiece, causing the metal to flow and fill the die cavities. The hammer is generally in contact with the workpiece on the scale of milliseconds. Depending on the size and complexity of the part, the hammer may be dropped multiple times in quick succession. Excess metal is squeezed out of the die cavities, forming what is referred to as "flash".

The flash cools more rapidly than the rest of the material, this cool metal is stronger than the metal in the die, so it helps prevent more flash from forming. This also forces the metal to completely fill the die cavity. After forging, the flash is removed. In commercial impression-die forging, the workpiece is usually moved through a series of cavities in a die to get from an ingot to the final form.

Design of impression-die forgings and tooling

Forging dies are usually made of high-alloy or tool steel. Dies must be impact resistant, wear resistant, maintain strength at high temperatures and have the ability to withstand cycles of rapid heating and cooling. In order to produce a better, more economical die the following standards are maintained:

• The dies part along a single, flat plane whenever possible. If not, the parting plane follows the contour of the part.

• The parting surface is a plane through the center of the forging and not near an upper or lower edge.

• Adequate draft is provided,usually at least 3° for aluminium and 5° to 7° for steel.

• Generous fillets and radii are used.

• Ribs are low and wide.

• The various sections are balanced to avoid extreme difference in metal flow.

• Full advantage is taken of fiber flow lines.

• Dimensional tolerances are not closer than necessary.

The dimensional tolerances of a steel part produced using the impression-die forging method are outlined in the table below. The dimensions across the parting plane are affected by the closure of the dies and are therefore dependent on die wear and the thickness of the final flash. Dimensions that are completely contained within a single die segment or half can be maintained at a significantly greater level of accuracy.

Dimensional tolerances for impression-die forgings

Mass [kg (lb)]	Minus tolerance [mm (in)]	Plus tolerance [mm (in)]
0.45 (1)	0.15 (0.006)	0.46 (0.018)
0.91 (2)	0.20 (0.008)	0.61 (0.024)
2.27 (5)	0.25 (0.010)	0.76 (0.030)
4.54 (10)	0.28 (0.011)	0.84 (0.033)
9.07 (20)	0.33 (0.013)	0.99 (0.039)
22.68 (50)	0.48 (0.019)	1.45 (0.057)
45.36 (100)	0.74 (0.029)	2.21 (0.087)

A lubricant is used when forging to reduce friction and wear. It is also used as a thermal barrier to restrict heat transfer from the workpiece to the die. Finally, the lubricant acts as a parting compound to prevent the part from sticking in the dies.

Advantages of open-die forging

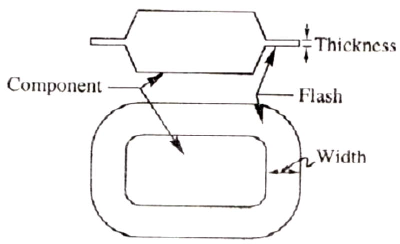
• Reduced chance of voids.

- Better fatigue resistance.
- Improved microstructure.
- Continuous grain flow.
- Finer grain size.
- Greater strength.

2.2.1 Design of flash and gutter

The excess metal added to the stock to ensure complete filling of the die cavity in the finishing impression is called Flash. It acts as a cushion for impact blows from the finishing impression and also helps to restrict the outward flow of metal, thus helping in filling of thin ribs and bosses in the upper die.

The amount of flash depends on the forging size and may vary from 10 to 50 %. The flash flows around the forging in the parting plane.

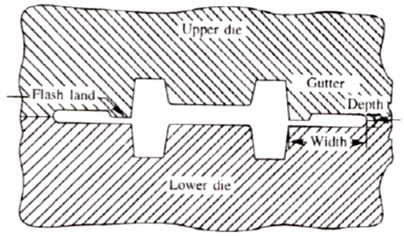


Proportion of flash in drop forging

Forging load is greatly influenced by the flash thickness and width. The load can be decreased by increasing the flash thickness but metal lost in the form of flash (it is a waste) also increases. The forging load also decreases with an increase in the average thickness of the component.

In addition to the flash,provision should be made in the die for additional space so that any excess metal can flow and help in the complete closing of the die. This is called Gutter. Without gutter, flash may become excessively thick, not allowing the dies to close completely.The flash land provided in the die should be about 3 % of the maximum forging thickness (0.5 to8.0 mm).

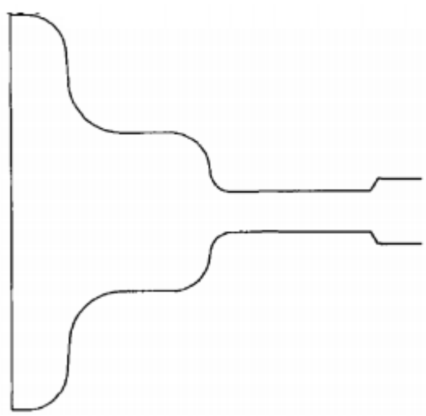
If the flash land is too small, then the energy required for the forging increases because of the excess metal trapped in the finishing impression and flash land wears out quickly. Similarly, too high a flash land lets the work metal to flow into the gutter and thus the die cavity gets unfilled.



Proportion of gutter in drop forging

The wider area of the gutter is usually made in the top section of the die. If the forging is of such a shape that it must be turned over through an angle of 180° when trimming the flash away, the magazine is made at the bottom. For greater slowing down of metal, the bridge of the gutter is made with a barrier. If much metal is expected to flow to flash, a larger magazine is provided. A wedge-type gutter decreases the flash.

The flash land and gutter used in dies perform two functions during forging. Firstly, the flash land restricts side ways metal flow and thus forces the material to fill die cavities by extrusion Secondly,



Flash and gutter

Dunning the final stages of forging when the cavity is filled the flash land allows metal to escape into the flash gutter. Using the flash command from the CAD menu, the user is asked to select the position where the flash has to be located. The calculation of the flash land and gutter depends on the mass of the forging figure shows in above the forging with flash land and gutter.

2.3 Upset forging die design.

Deep drawing is one of the most widely used processes in sheet metal forming. Apart from its use in many other sectors, it is applied in the automotive industry for the manufacturing of car body parts.

Process definition:

The deep drawing process is a forming process which occurs under a combination of tensile and compressive conditions. A flat sheet metal blank is formed into a hollow body open on one side or a hollow body is formed into a hollow body with a smaller cross-section.

Deep drawing processes are divided into three types:

• Deep drawing with tools.

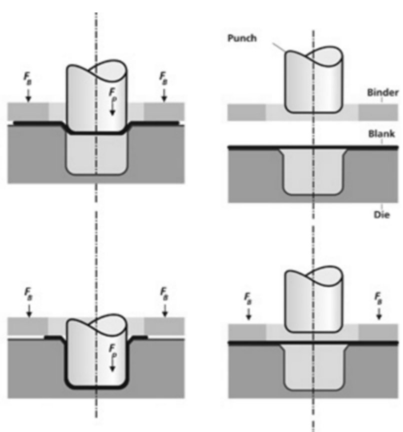
• Deep drawing with active means.

• Deep drawing with active energy.

In the automotive industry, deep drawing is usually carried out using rigid tools.

The figure illustrates the deep drawing process. The rigid tools consist of a punch, die and binder. In deep drawing, the plate holder closes after the metal sheet blank has been inserted.

Next the sheet is clamped between the die and binder. This process slows down the flow of the sheet while it is being drawn and thereby prevents wrinkles from forming under the binder. The punch stretches the sheet over the die radius and forms it in the die. The amount of punch force necessary for forming is thereby continually increased up to the lower dead center of the punch.



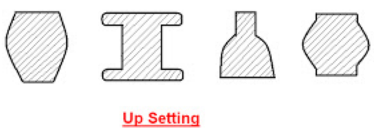
Upset forging die

Whereas in pure deep drawing there is no reduction of sheet metal thickness, forming is achieved in stretch forming purely as a result of a decrease in sheet metal thickness. Stretch forming is extensively used for the forming of only slightly curved parts with low depth of draw

When drawing complex car body parts in practice, there is usually a combination of stretch and deep drawing involved. It is necessary that the sheet metal is stretched as well as possible without reaching the material's limits.

There are approximately 300 to 400 sheet metal parts which fit together to form the body of a car. In order to manufacture high quality sheet metal parts optimally for their particular use, specialized software simulates the complete deep drawing process.

This is applied to increase the cross-sectional area of the stock at the expanse of the length. To achieve the length of upsetting force is applied in a direction parallel to the length axis, For the example forming of a bolt head.



Up setting

Upsetting force

$$F = A_1 \cdot k_{str1} \left(1 + \frac{1}{3} \cdot \mu \frac{d_1}{h_1}\right)$$

F= Upsetting force,

A_1= Surface after upset forging,

K_{str1}= Flow stress at the end of upsetting,

μ= Coefficient of friction (0.1 – 0.15),

d_1= Diameter after upset forging,

h_1= Height after upset forging,

Upset forging involves increasing the cross-section of a material at the expense of its corresponding length. Upset forging was initially developed for making bolt heads in a continuous manner, but presently it is the most widely used of all forging processes.

Parts can be upset forged from bars or rods upto 200 mm in diameter in both hot and cold condition. Examples of upset forged parts are fasteners, valves, nails and couplings.

The process uses split dies with one or several cavities in the die. Upon separation of split die, the heated bar is moved from one cavity to the next. The split dies are then forced together to grip the and a heading tool advances axially against the bar, upsetting it to completely fill the die cavity. Upon completion of upsetting process the heading tool comes back and the movable split die releases the stock.Upsetting machines, called upsetters, are generally horizontal acting.

When designing parts for upset forging, the following three rules must be followed.

1. The length of unsupported bar that can be upset in one blow of heading tool should not exceed three times the diameter of bar. Otherwise bucking will occur.

2. For upsetting length of stock greater than three times the diameter the cavity diameter must not exceed 1.5 times the dia of bar.

3. For upsetting length of stock greater than 3 times the diameter and when the diameter of the upset is less than 1.5 times the diameter of the bar, the length of unsupported stock beyond the face of die must not exceed diameter of the stock.

2.4 Sheet metal working: Design consideration for shearing

Basic Shearing Operations

The broad classification of sheet metal operations falls under the following two categories.

(i) Cutting operations

(ii) Forming operations

1. Cutting operations

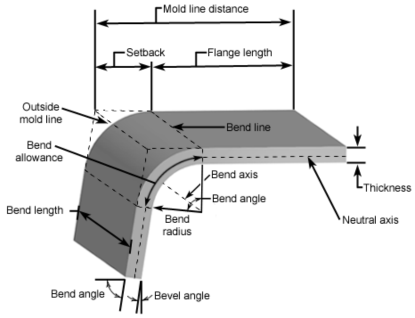
In cutting operations, the work piece is stressed beyond its ultimate strength and cut-off into two pieces. In forming operations, the stresses are below the ultimate strength of the metal. There is no cutting off-metal but only the contour of the work piece is changed to get the desired product.

2. Forming operations

(a) Bending

The bending is a metal forming process in which a force is applied to a piece of sheet metal, causing it to bend at an angle and form the desired shape. A bending operation causes deformation along one axis, but a sequence of several different operations can be performed to create a complex part.

Bent parts can be quite small, such as a bracket or up to 20 feet of length, such as a large enclosure or chassis. A bend can be characterized by several different parameters, shown in the figure below.



 Bending Diagram

Bend line - The straight line on the surface of the sheet, on either side of the bend, that defines the end of the level flange and the start of the bend.

Outside mold line - A straight line where the outside surfaces of the two flanges would meet, were they to continue. This line defines the edge of a mold that would bound the bent sheet metal.

Flange length - The length of either of the two flanges, extending from the edge of the sheet to the bend line.

Mold line distance - The distance from either end of the sheet to the outside mold line.

Setback - The distance from either bend line to the outside mold line. Also equal to the difference between the mold line distance and the flange length.

Bend axis - The straight line that defines the center around which the sheet metal is bent.

Bend length - The length of the bend, measured along the bend axis.

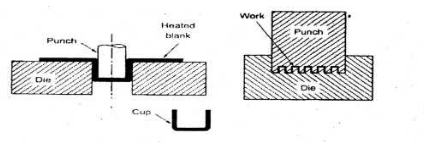
Bend radius - The distance from the bend axis to the inside surface of the material, between the bend lines. Sometimes specified as the inside bend radius. The outside bend radius is equal to the inside bend radius plus the sheet thickness.

Bend angle - The angle of the bend, measured between the bent flange and its original position or as the included angle between perpendicular lines drawn from the bend lines.

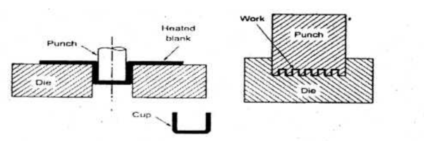
Bevel angle - The complimentary angle to the bend angle.

(b) Drawing: Drawing operations consists of a punch forcing a sheet metal blank to plastically into the clearances available between the punch and die surface so as to acquire top shape, a cylindrical shape or a box shape.

(c) Squeezing: In squeezing operation, the metal is caused to flow to all portions of cavity under the action of compressive force.



Drawing



Embossing

(d) Embossing:

It is the process of producing required shapes on sheet metal blanks of punches and dies.

(e) Nibbling:

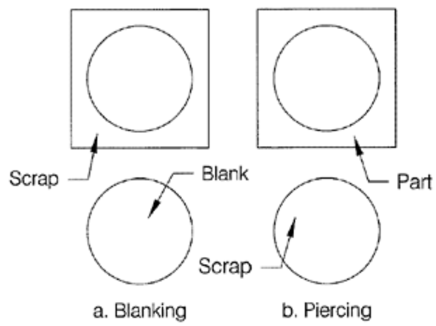
Nibbling is an operation of cutting any shape from sheet metal without metal tools. It is done on a nibbling machine. The required shape of is connected in the form of tracer or templates in nibbling machine. It time taken to cut the required shape is less when compared to other processes.

2.4.1 Blanking piercing

Blanking and piercing

The metal cutting is a process used for separating a piece of material of predetermined shape and size from the remaining portion of a strip or sheet of metal. It is one of the most extensively used processes throughout die and sheet-metal work. It is consists of different material-parting operations, such a piercing, perforating and shearing, notching, cutoff and blanking.

In blanking, the piece is cut off from the sheet and it becomes a finished part. In piercing, the cutout portion is scrap which gets disposed off while the product part travels on through the remainder of the die. The terminology is different here, though both processes are basically the same and therefore belong to the same category, which is the process of metal cutting figure given by,



Blanking and piercing differentiation.

The actual task of cutting is subject to many concerns. The quality of surface of the cut it condition of the remaining part, straightness of the edge, amount of burrs, dimensional stability all these are quite complex areas of interest, well known to those involved in sheet metal work.

Most of these concerns are based upon the condition of the tooling and its geometry, material thickness per metal cutting clearance, material composition and amount of press force, accurate locating under proper tooling and a host of additional minor criteria. These all may affect the production of thousands and thousands of metal stamped parts.

Blanking Tool

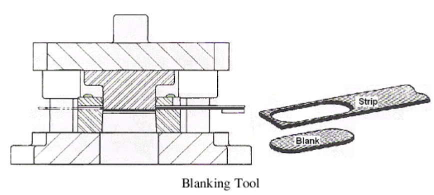
• When a component is produced with one single punch and die where the entire outer profile is cut in a single stroke the tool is called a blanking tool.

• The outer area of metal remaining after a blanking operation is generally discarded as waste.

• Blanking is the operation of cutting flat shapes from sheet metal.

• The size of blank or product is the size of the die and clearance is given on punch.

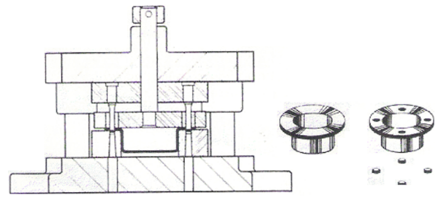
• One of the metal cutting operation.



Blanking Tool

Piercing tool

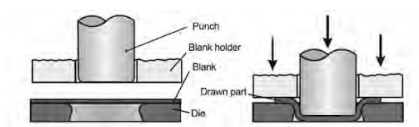
A piercing involves cutting of clean holes with a resulting scrap slug. The operation is known as die cutting and can also produce flat components where the die, the shaped tool is pressed into a sheet material employing a shearing action to cut holes. This process will be used to cut parts of various sizes and shapes in sheet metal, leather and many other materials.



Piercing tool

2.5 Deep drawing operation

A deep drawing is a sheet metal forming process in which a sheet metal blank is radially drawn into a forming die by the mechanical action of a punch. It is thus a shape transformation process with material retention. The process is considered "deep" drawing when the depth of the drawn part exceeds its diameter. This is achieved by redrawing the part through a series of dies.



Schematic outline of Deep Drawing Process

The metal flow during deep drawing is extensive and hence, requires careful administration to avoid tearing or fracture. Following are some of key issues affecting metal flow during deep drawing process and both of them should be considered when designing or troubleshooting sheet metal deep drawing stamping tools.

Causes of wrinkling in deep drawn parts: Several factors can cause wrinkles in deep drawn parts, including: • Blank holder pressure.

• Clearances between the blank, blank holder, punch and die cavity.• Final part geometry.• Friction between the blank, punch and die cavity,blank holder.• Blank shape and thickness.

• Die cavity depth and radius.• Punch speed.An other factors, such as die temperature and the metal alloy of the blank, can also affect the drawing process. A variation in any of these factors influences the potential for wrinkling or cracking in the deep-drawn part.The blank holder, as the name implies, holds the edges of the sheet metal blank in place against the top of the die while the punch forces the sheet metal into the die cavity, the sheet metal deforms into the proper shape, instead of simply being pulled into the die cavity. However, does not hold the edges of the blank rigidly in place. If this were the case, tearing could occur in the cup wall.

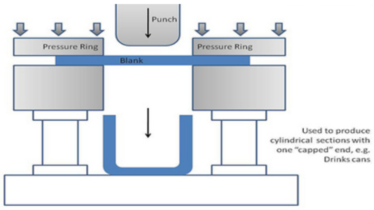
The blank holder allows the blank to slide somewhat by providing frictional force between the blank holder and the blank itself. Blank holder force can be applied hydraulically with pressure feedback, by using an air or nitrogen cushion or a numerically controlled hydraulic cushion.

The greater the die cavity depth, the more blank material has to be pulled down into the die cavity and the greater the risk of wrinkling in the walls and flange of the part. The maximum die cavity depth is a balance between the onset of wrinkling and the onset of fracture, neither of which is desirable.The radii degrees of the punch and die cavity edges control the flow of blank material into the die cavity. Wrinkling in the cup wall can occur if the radii of the punch and die cavity edges are too large. If the radii are too small, the blank is prone to tearing because of the high stresses.**Using a blank holder:**The simplest method for eliminating wrinkling in deep-drawn parts is using a blank holder. In most of deep drawing processes, a constant blank holder pressure is applied throughout the entire drawing action.Variable blank holder pressure, however, has been employed with some success. A pneumatic or hydraulic blank holder cushion can vary the blank holder pressure linearly over the stroke of the machine. This provides some increase in the allowable die cavity depth.A (NC) numerically controlled die cushion can be used to provide a variable blank holder pressure over the course of drawing action. In an optimal blank holder pressure force profile, the initial force is large so as to provide initial deformations.The cushion drops off to pull material into the die cavity and then slowly increases back up to ensure strain hardening in the drawn part. An NC die cushion can dramatically increase the allowable die cavity depth while preventing both wrinkling and cracking.**Die cavity design:**The design of the punch and die cavity can be optimized to reduce the probability of wrinkling. Choosing a flange radius that is just large enough to prevent cracking can minimize the potential for wrinkles. Additionally, considering minimizing the part complexity and any a symmetry can also help. Incorporating a multi-step drawing process offers a variety of advantages in preventing wrinkling in deep-drawn parts.Designing the blank geometry to minimize excess material can reduce the potential for wrinkling. The sheet metal blank has an inherent grain structure, so the stresses can vary depending on the design of the die and the orientation of the grain. Adjusting the grain in an asymmetrical design to minimize the compound of grain stresses and the general stresses of the deep draw process is something to take into consideration.

Other factors to consider: Surface conditions of both component can be tailored to improve overall performance. Lubricants reduce the friction between the blank and the punch and die cavity and can be liquid (wet) or films (dry).

Deep-drawing test

Deep drawing is steel metal forming method in which a sheet metal blank is radially drawn into a forming die by the mechanical action of a punch. It is thus a shape transformation process with material retention.

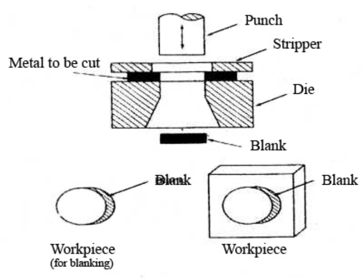


Deep drawing test

2.6 Die design for sheet metal operations

The common cutting operations are discussed below.

(a)Blanking:



Blanking and piercing or punching

Blanking is the operations of cutting a flat shape from the sheet metal. The metal that is punched out is called as 'blank' and the metal that is left out is called as scrap.

(b) Punching or Piercing:

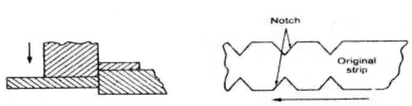
It is the operation of producing the hole on the work piece by a punch. In punching, the metal removed is called as a scrap and the metal that is left out called as work piece.

(c) Shearing:

It is the operation through which a metal is cut along a single line, usually a straight line.

(d) Parting:

It is the operation through which the metal is cut simultaneously along two parallel lines or contours or any other two lines which can balance each other to neutralize side thrust.



Shearing Notching

(e) Notching:

It is the operation through which metal pieces are cut from the edges of sheet, strip or blank.

(f) Trimming:

It is the operation used for removing excess metal, irregular outlines and waved edges etc., from the walls of drawn shells or the surfaces of waved and cast parts.

(g) Shaving:

It is also similar to trimming operations but here the amount of metal removal is usually about 10% of the thickness of the blank.

(h) Perforating:

In this operation, multiple holes which are very small and close together are cut in flat work material.

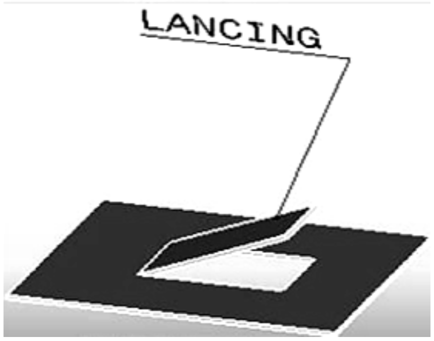
Slitting and perforating

(i) Slitting:

It is the operation of making an unfinished cut through a limited length only.

(j) Lancing:

Lancing consists of cutting the sheet metal through a small length and bending this small cut portion downwards.



Lancing

Sheet Metal Stamping Stamping presses and stamping dies are tools used to produce high volume sheet metal parts.

The press provides the force to close the stamping dies where they shape and cut the sheet metal into finished parts.

Production stamping is generally performed using sheet metal materials .020 to .080 thick, but the process also can be applied to foils as thin as .001 or to plate stock with thickness' approaching 1.000. Formability is the primary attribute of sheet metal material.

Formability is further defined as the materials ability to be:

• Bent

• Stretched

• Drawn

The metallurgical term for these qualities is “ductility”. Ductility is the materials ability to deform and elongate without fracture.

The extent to which a stamping is subjected to such deformation is directly related to the part’s overall shape and geometry. Other factors also influence the material’s formability.

They include:

• The die design.

• The press.

• The press speed.

• Lubrication.

• Sheet metal feeding mechanisms.

• Monitoring and control systems.

2.7 Progressive and compound die

Progressive dies:

A progressive die is also known as a follow on die. It consists of a series of stations. Thus, it is also known as a multi station progressive die. An operation is performed on the workpiece at each station during the stroke of the press. In between the stroke intervals, the workpiece, generally a metal strip, is transferred to the next station by suitable devices.

A simplified sketch of a progressive die is shown in figure. In the first stroke, the piercing punch cuts a hole in the stock. The strip is advanced for the blanking operation for the production of washer.

In a progressive die, the number of components produced is the same as the number of strokes.

1. Bottom bolster,

2 Thrust plate.

3. Die.

4. Nest plate.

5 Stripper plate,

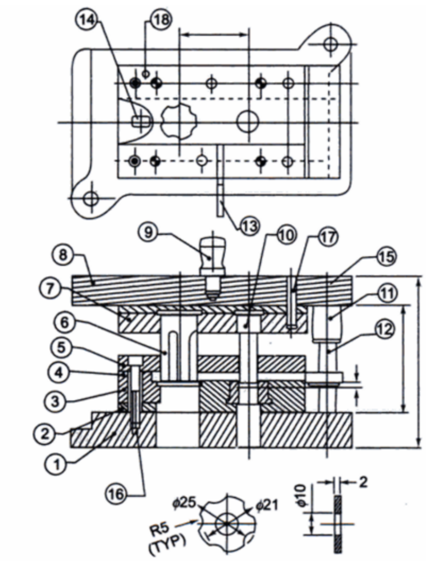
6. Punch.

7 Punch retainer.

8. Top bolster.

9. Shank.

10. Piercing punch.



Progressive die for a component

11. Guide bush.

12 Guide piler.

13. Finger stopper.

14. Running stopper.

15 Thrust plate,

16. Allen screw

17. Dowel pin.

18. Nest plate R.H

• These dies are made to cut and form a part in successive stages or stations of the die.

• The parts are held together by the strip Skelton or tabs until the last station or cut off. Force or movement comes from the strip feeder attached to the press.

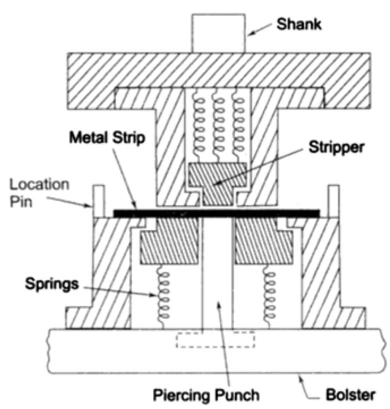
• A progressive die can perform very complex work, doing piercing, blanking, forming, lancing and notching.

• The advantage of tine type of die is that the tooling can be spread out.

• The nut of progressive die. is high and therefore they are usually limited to high production operation.

Compound dies

Dies used for performing two or more operations at one station are called compound dies. They are mainly used for cutting operation. figure shows the sketch of a compound cutting die used for cutting washers in one stroke of the press. The two operations performed for cutting a washer are blanking and piercing. Compound dies are accurate and economical for mass production.



Compound blank

• Compound dies make does tolerance and concentric parts, as all work is done in one stroke.

• Figure shows a compound blank and draw die. The metal enter. as a flat sheet, is cut to the right length and is then formed over a reverse-type punch. Spring action on a pressure plate stripe the part off the punch.

• The knockout pin, shown at the top, pushes the part out of the upper die if it stays on that side when the die opens.

2.7.1 Strippers

After a blank has been cut by the punch on its downward stroke the scrap strip has a tendency to expand. On the return stroke of the punch, the scrap strip has the tendency to adhere to the punch and be lifted by it.

This action interferes with the feeding of the stock through the die and some device must be used to strip the scrap material from the punch as it clears up the die block. Such a device is called stripper or stripper plate.

It is two types of strippers are;

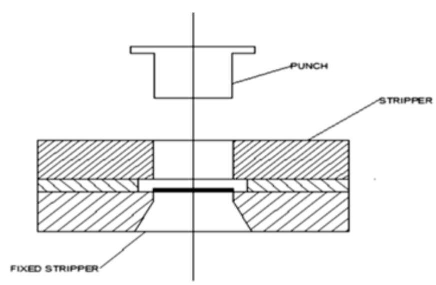
1. Fixed stripper

2. Sprig loaded stripper

1. Fixed stripper

Stripper is attached at a fixed height over the die block. The height should be sufficient to permit the sheet metal to be fed freely between the upper die surface and the under surface of the stripper plate. The stripper plate is usually of the same width and length as the die block in simple dies, it is fastened with the same screws and dowels which are used for die block.

In the complex dies, the strippers fastening will be independent of refastening. The thickness of the stripper plate should be sufficient to withstand the forces needed to strip the scrap strip from the punch. The usual value 9.5 mm to 16mm.



Stripper

2. Sprig loaded stripper

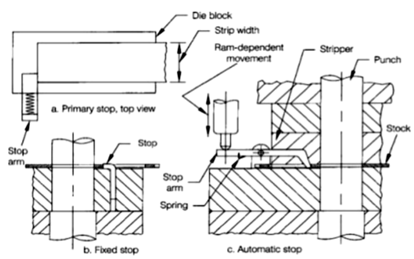
This type is used on large blanking operations and also on very thin and highly ductile materials where to utilize the pad pressure to hold the surrounding stock during the blanking operation. In this design the stripper plate is mounted over the compression springs and suspended by bolted from the punch holder, with the lower surface of the striper below the cutting end of the punch.

As the punch travels downward from the blanking operation, the stripper plate contacts the stock strip first and hold s it until the clears the strip on its return stroke. As the punch rises, spring pressure holds the strip, stripping it forces may vary from 2.5 to 20% of cutting force. However, the more common values for most of the applications are 5 to 10%.

2.7.2 Stops

Strip material, when first being guided into the die, should be stop somewhere for the sequence of die operations to begin successfully. It is obvious that the strip should not go as far as the forming tool, which may need some pre-blanking work performed at the beginning.

Advancing the strip too far may lead to greater than usual wear and tear of the tooling and its subsequent misalignment and breakage. For that purpose, stops are introduced in the die work. The first stop, which the strip meets on its way, is usually the first pierce and blank locator, which navigates the strip in such a way that all cutting is included prior to its arrival at forming and other stations.



Stops

The automatic stop is a device which slides up and down along with the movement of the ram and either:

1) Forces the nose of the stop lever up, to release its engagement of the strip for the latter's Progression.

2) Releases its pressure on the lever, thus allowing its nose to come down, pushed by a force of a spring. In such a position, the lever is ready for registration and retainment of the advancing strip.

2.8 Strip layout

It plays an important role especially in the case of the design of the press tool Strip decides the economic utilization of the work piece and helps in the decrease of cost of the job and reduction in the production time by increasing the number of components or layout the position of the work pieces in the strip and their orientation with respect to another. This is called 'strip layout'.

The factors which will influence the stock layout are:

1) Economy of material,

2) Direction of material grain or fibre,

3) Strip or coiled stock,

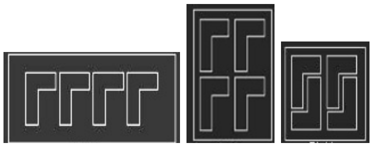
4) Press used,

5) Production required,

6) Die cost,

1) Economy of material

The different ways of arranging to blank the given work piece are shown.The arrangement of figure (a), the strip would either have to be fed twice,once for each row or double blanking will have to be employed The percent of material utilization may increase somewhat by the arrangement of figure (b), that is , by having two rows of blanks. Shown in the figure (c).



(a) (b) (c)

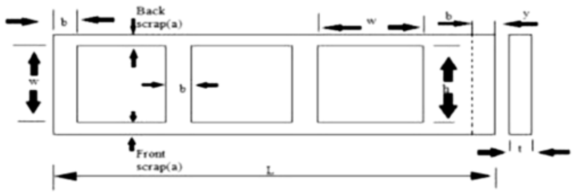
in a single row, double pass strip. This is called "stock nesting". Here, the strip will have to be passed through the dies once, turned over and passed through dies a second time. Nesting considerably reduces the scrap. However the strip layout with maximum material saving may not be the best strip layout, as the die constriction may become more complex which will offset the savings due to material economy unless a large number of parts are to be produced.

Another important consideration in strip layout is the distance between the nearest points of blanks and between blanks and the edges of the strip. To prevent the scrap from twisting and wedging between blanks and the die, the distance must increase with material thickness. A general rule of thumb is to keep this distance, called web, at least 1.5 times the material thickness.

However, other factors such as strip thickness, hardness of the material, type of operation, shape of blank etc. may allow the web to be thinner. The various terms connected with strip layout are shown in below figure.

The distance between the blank and edge of strip, known as back scrap may be determined by the equation,

a=t +0.015h



The distance between successive blanks and also the scrap bride ,b, is given below,

Material thickness in mm	b in mm
0.8	0.8
0.8 to 3.2	t
Over 3.2	3.2.

In general softer materials requires larger spacing and thinner materials require larger spacing.

The feed or advance or the length o f one piece o f stock needed to produce one blank is,

S = w + b

The number of blanks which can be produced from one length of stock can be found out as,

N = (L - b)/s

The scrap remaining at the end of one length of strip maybe calculated from,

L (Ns + b)

Measure of material utilization:

η_m= Area of blank to be cut /area of material available

=B/A x 100

$$\% \text{ of scrap} = \frac{(A-B)}{A} * 100$$

Now area of material available per blank= Feed or advance x stock width

2. Direction of material grain or fibre

This factor is to be considered if the cut blanks have to undergo any subsequent operation, such as, bending or deep drawing. When the sheet metal strip is rolled in the mill, a fibre is produced in the direction of strip length.

During subsequent bending operation on the blank, to obtain maximum strength from bend parts, the bend should be made across the strip or at an angle of 90° to the fibre. Therefore some part prints specify that the fibre is to run in the direction of an arrow shown on the print.

In such cases, the blanks may cannot be tripped or rotated to just any position desired.

3) Strip or coiled stock

Another important consideration in the strip layout is whether the stock used will be in the form of a strip or coil. Whereas, the stock strip may be passed through the die more than once, the coiled stock is usually passed through the die only once.

When coiled stock is used, recoiling and recoiling of the stock is expensive.

Thus coiled stocked is used when:

1) Production is high

2) Thinner metal sheets are employed.

3) The stock needs to be passed through the die only once.

4) Strip stock is used when

5) Production is low

6) Thicker sheet metals are used

7) 7) The stock needs to be passed through the die more than once.

4. Direction of burr

When sheet metal is cut in a die, a burr is produced on the die side of the scrap strip and on the punch side of the blank. If the burr has to be on the hidden side, then the expensive operation of removing the burr need not be done. For this, a note is often placed on the part drawing which reads "burr down".

To control the position of the burr may limit the stock-layout arrangement. Scrap may not be reduced to a minimum.

5. Press used

During production planning, a press has been assigned to the operation and the die. Therefore, the stock layout has to be such that it allows the die to be designed within the press capacity. Shear may have provided on punch or die, to limit the maximum cutting force within the press capacity.

Another factor is the bed area of the press. The relation of the press bed area to the blank area is a definite factor controlling the stock layout. The third factor is to have the cutting forces of the die evenly balanced around the center line of the press ram.

6. Production requirement

The following guide lines may be followed when the production is the main consideration:

a) Low production -Thin material:

• Strip stock and a single-pass layout.

• Cutting of one or more blanks at a time.

b) Low production of thin material:

• Strip stock and a single or double-pass layout.

• Cutting one blank at a time.

c) High production -Thin material:

• Coiled stock and a single pass layout.

• Cutting of one or more blanks at a time.

d) High production - Thick material:

• Strip stock and a single or double-pass layout.

• Cutting of more than one blank at a time.

7. Die-cost

i) Higher productions.

ii) Cutting more than one time, particularly when cutting extremely complicated blank shapes or when cutting extremely accurate blank

However, for simple round or square edged blanks, multiple cutting at one time is often practical. Also, double-pass dies are less expensive than cutting two at a time. So, the designer has to decide while making the stock layout, as to which is preferred: more operator time per blank or more machine time per blank.

The first step in strip layout is defining the strip. This process involves naming the strip assembly and the strip part and defining the width and height of a station, the project shortcut, the number of stages and an offset before and after the strip. we can specify a prefix for the name that is generated for parts placed inside the strip assembly. Parts include instances of the article and stamp reference parts.

Note: Terms used in a strip layout

1) Scrap bridge:

a) this is the portion of the material remaining after blanks operation between one edge of the strip and the cutout portion.

b) Thee portion of material remaining between the two adjacent openings after blanking is also called as the scrap bridge.

2) Front Scrap:

This is the scrap bridge on that edge of the strip which is towards the operator In the design of blanking part from strip material, the first step is to prepare blanking layout, that is, to layout the position of the work pieces in strip and their orientation with respect to one another. While doing so, the major consideration is the Economy of material.

Another important consideration in strip layout is the distance between the blanks and the strip edge and distance between blank to blank. To prevent the scrap from twisting and wedging between the punch and the die. The distance must increase with material thickness. A general rule of thumb is to keep this distance equal to from 1 to 1.5 times the material thickness.

3

DESIGN OF JIGS AND FIXTURES, TOOLS, LIMIT GAUGES AND PROCESS PLANNING

Design of jigs and fixtures, principle of location and clamping, clamping methods, locating methods, Drill Jig bushing, Indexing type drilling Jig. Design of single point cutting tool, broach and form tool. Design of limit gauges.

Process Planning – selection of processes, machines and tools. Design of sequence of operations, Time & cost estimation, Tooling design for turret lathe and automats.

3.1 Design of jigs and fixtures

Fixtures:

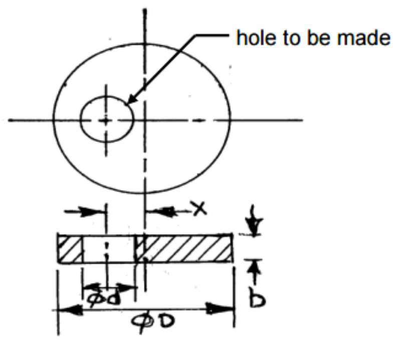
These strong and rigid mechanical devices used in machine shop enable easy, quick and consistently accurate locating, supporting and clamping, blank against cutting tools and result faster and accurate machining with consistent quality, functional ability and interchangeability. It holds the workpiece securely in the correct position with respect to the machine or cutter during the operation. There is sometimes a provision in the fixture for setting the tool with respect to the workpiece but tool is not guided. The fixtures are often clamped to the machine table.

Jigs:

It is a fixture with an additional feature of tool guidance such as drill bushes. These direct the tool to the correct position on the workpiece. Jigs are rarely clamped on the machine table because it is necessary to move the jig on the table to align the various bushes in the jig with the machine spindle.

Purpose Of Using Fixtures And Jigs

A machining work, like drilling a through hole of given diameter eccentrically in a pre-machined mild steel disk as shown in figure below in a conventional drilling machine without using any fixture or jig, the following elementary steps are to be sequentially followed.



(a) A through hole has to be drilled in a pre-machined mild steel disc

• Cleaning and deburring the blank (disc).

• Marking on the blank showing the location of the hole and its axis on the blank.

• Punch the centre at the desired location and prick punch the periphery of the hole to be made in the disc.

• Mount the blank in a drilling vice using parallel block, a small Vee block etc. to provide support and clamp the blank firmly.

• Position the vice along with the marked blank to bring the hole axis in alignment with the drill axis by,

• Either adjusting the vise position. The fixed drill axis.

• If moving the drilling machine table and then locking the table position

• If moving the radial arm and the drilling head, if it is a radial drilling machine.

• After fixing the blank, vise and the table, alignment is checked again.

• If error, like eccentricity, is found to occur then readjustment of location of the hole axis is to be done before and even after starting drilling.

• Drilling is accomplished.

Therefore it appears that more operations are needed to be carried out carefully and skilfully by the machinist or operator for such as simple job. Even after that there should be inaccuracies in machining. Such slow and time consuming manual work are eliminated or drastically reduced in mass production by automatic or special purpose machine tools.

But such a machine tools are quite expensive and hence are economically justified for only small mass production and not viable for small lot or batch production. For batch production proper design and use of simple but effective jigs and fixtures are appropriate and economically justified. This is schematically illustrated in figure below.

The common purposes of developing and using suitable jigs and fixtures for batch production in machine shops are :

• To eliminate marking, positioning, punching, alignments etc.

• It easy, quick and consistently accurate locating and supporting, clamping the blank in alignment of the cutting tool

• It increase in productivity and maintain product quality consistently

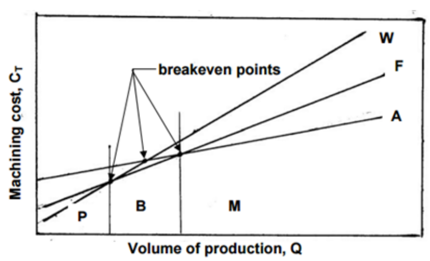
• A guidance to the cutting tool like drill, reamer etc.

• It reduction of overall machining cost and also increase in interchangeability.

• To reduce operator's labour and skill requirement

• Enhancing technological capacity of the machine tools.

• To reduce measurement and its cost.



Role of Jigs and Fixtures on machining cost

W – Without using jig and fixture.

A – Automatic (special purpose) machine.

F – Using jig and fixture.

P – Piece production.

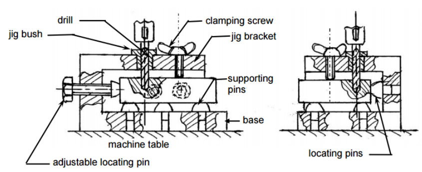
B – Batch production.

M – Mass production.

Design Considerations For Jigs And Fixtures

A Jigs and fixtures are manually or partially power operated devices. To fulfil their basic purposes, jigs and fixtures are comprised of many elements shown in below the figure.

- Base and body or frame with clamping feature.
- Indexing plates or systems, if necessary.
- Supporting surfaces and base.
- Clamping elements.
- Locating elements for proper positioning and orientation of the blank.
- Tool guiding frame and bushes.
- Auxiliary elements.
- Fastening parts.



Major elements of jig and fixtures

Therefore keeping in view increase in productivity, product quality,repeatability i.e. interchangeability and overall economy in batch production by machining,

The following factors are essentially considered during design, fabrication and assembly of jigs and fixtures :

- It is providing strong, rigid and stable support to the blank.
- It quick, strong and rigid clamping of the blank in the jig or fixture without interrupting any other operations.
- It is easy, quick and consistently accurate locating of the blank in the jig or fixture in reference to the cutting tool.
- Tool guidance for slender cutting tools like drills and reamers.
- It is easy and quick loading and unloading the job to and from the jig or fixture.

• If use of minimum number of parts for making the jig or fixture.

• If use of standard parts as much as possible.

• Easy and quick removal and replacement of small parts

• Reasonable amount of flexibility or adjustability, if feasible, to accommodate slight variation in the job - dimensions.

• It is easy, quick and accurate indexing system if required.

• Durability and maintainability.

• Easy and safe handling and moving the jig or fixture on the machine table, i.e., their shape, size, weight and sharp edges and corners.

• Prevention of jamming of chips, i.e. wide chips-space and easy chip disposal.

• Manufacturability i.e. ease of manufacture.

• Service life and overall expenses.

3.1.1 Principle of location and clamping

PRINCIPLES OF LOCATIONS

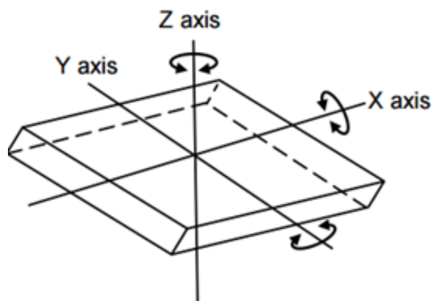
Any rectangular body have three axis along x-axis, y-axis and z-axis. It can more along any of these axes or any of its movement can be released to these three axes. At the same time the body can also rotate about these axis too. So total degree of freedom of the body along which it can move is six.

The body it is required to restrain all the (DOF) degree of freedom by arranging suitable locating points and then clamping it in a fixed and required position. The basic principle used to locate the points is given below.

Six Point Location of a Rectangular Block

Considering the six degree of freedom of a rectangular block as shown in the figure. It is made to rest on several points on the jig body. To provide a rest to workpiece on three points on the bottom x-y surface. This will stop the movement along z-axis, rotation with respect to x-axis and y-axis. Supporting it on the three points is considered as better support then one or two points.

Rest the workpiece on two points of side surface (x-z), this will fix the movement of workpiece along y-axis and rotation with respect to z-axis. Provide a support at one point of the adjacent surface (y-z) that will fix other remaining free movements. This principle of location of fixing points on the workpiece is also named as 3-2-1 principle of fixture design as number of points selected at different faces of the workpiece are 3, 2 and 1 respectively,

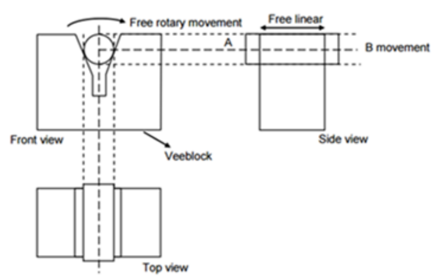


(a) Available Degree of Freedom of Rectangular Block

Body to be restrained (both of the axis can be divided into two halves positive and negative)

Location of a Cylinder on a Vee Block

The analysis of the principle of location of a cylinder on a Vee block is indicated in the figure (b). All the degrees of freedom of the cylindrical object are restrained. It is only fixed to move along axis AB. It can rotate about the axis AB. These free movements are also indicated in the figure. If the operation to be done on the cylindrical object requires restriction of the above mentioned free movements also than some more locating provisions must also in addition to the use of Vee block.



Locating a Cylinder on a Vee Block

CLAMPING

To restrain the workpiece completely a clamping device is required in addition to locating device and jigs and fixtures. A clamping device holds the workpiece securely in a jig or fixture against the forces applied over it during on operation.

Clamping device should be incorporated into the fixture, proper clamp in a fixture directly influence the accuracy and quality of the work done and production cycle time.

Basic requirement of a good clamping device are listed below :

- It should rigidly hold the workpiece.

• The workpiece being clamped should not be damaged due to application of clamping pressure by the clamping unit.

• The clamping pressure should be enough to over come the operating pressure applied on the workpiece as each pressure act on the workpiece in opposite directions.

• Clamping device must be capable to be unaffected by the vibrations generated during an operation.

• It should also be user friendly, like its clamping and releasing should be easy and less time consuming. Its maintenance should also be very easy.

• lamping pressure should be directed towards the support surfaces or support points to prevent undesired lifting of workpiece from its supports.

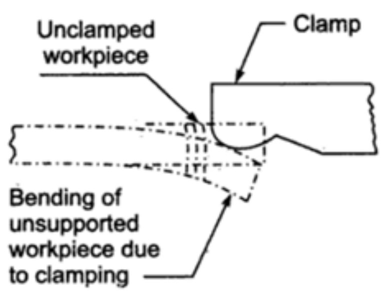
• Clamping faces should be hardened by proper treatments to minimize their wearing out.

• To handle the workpieces made of fragile material the faces of clamping unit should be equipped with fibre pads to avoid any damage to workpiece.

PRINCIPLES OF CLAMPING:

Position

Clamping should be positioned to direct the damping force on a strong, supported part of the workpiece. Clamping on unsupported part bends slender workpieces as is shown in figure.



(a) Distortion of unsupported workpiece

This affects the accuracy of the operation. A vertical hole drilled in the bent workpiece would become angular when the unclamped workpiece springs back to its original shape as is shown by the chair dotted lines in figure (a).

The clamping system should not obstruct the path of loading and unloading of the workpiece. The clamps in the path of loading should be retractable the path of loading and unloading of the workpiece. Clamps should not obstruct the path of the cutting tool. They should not get drilled, milled or welded during operation.

Strength:

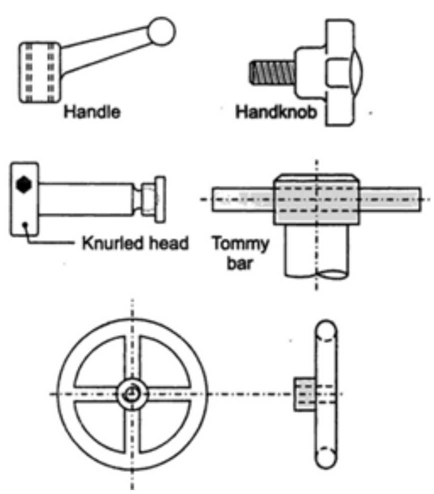
The clamping system should be capable of holding the workpiece security against the forces developed during operation. The clamping force should not dent or damage the workpiece with excessive pressure. For clamping weak or fragile workpieces, the clamping force should be distributed over a wider area of the workpiece. While clamping soft workpieces, clamps should be fitted with pads of softer materials, such as nylon or fibre to prevent damage and denting of the workpiece.

Productivity:

Clamping time should be minimised by using hand knobs, tommy bars, knurled screws, handwheels and handles shown in the figure(b), so that the clamp can be tightened or loosened manually without using spanners, as a spanner further adds motions of picking, aligning and laying it down.

Operator Fatigue:

Operator fatigue should be taken into account. If a considerable number of damps are to be tightened and loosened repeatedly, it is better to use pneumatic or hydraulic clamping which, in addition to reducing operator fatigue, also saves clamping time.Power damping facilitates tightening or loosening of many clamps simultaneously.



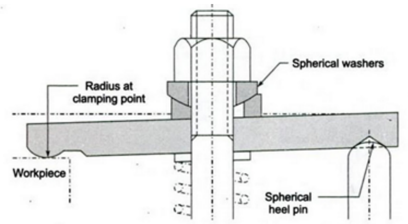
(b) Hand operated clamping device

Workplace Variation

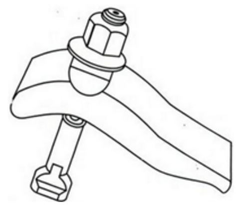
The damping points should be provided with ample radius to make the clamp operable even if there is variation in the workpiece. Heel pin pressure surface should also be made spherical to permit some tilting of the clamp shown in the figure (c).

Misalignment between the damp surface and the damping nut due to tilting of the clamp can be countered by use of spherical washers between the damp and the nut. As washers are used in pairs, the two washers have matching male and female spherical seats. The spherical bearing allows the washers to tilt with respect to each other.

The lower female washer tilts with the clamp while the upper male washer below the nut remains square to the nut. The spherical seat transmits the clamping pressure from the nut to the clamp.figure(d) shows a clamp with a cylindrical washer.

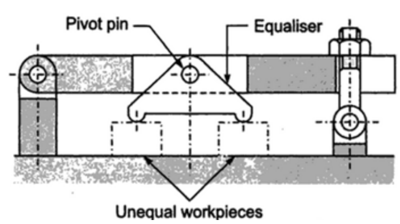


(c) Clamping variable workpieces

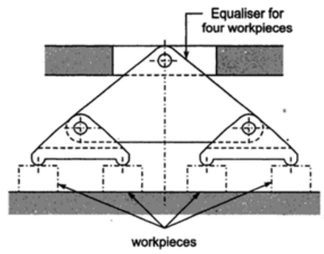


(d) Universal clamp with cylindrical washer

In multiple clamping, a pivoted equalizer is used for clamping two unequal workpieces simultaneously. The equalizer clamp pivots around the pin to suit the workpieces shown in the figure(e) . The equalizer principle can be extended to facilitate clamping of many even number of workpieces simultaneously by a single clamp shown in the figure (f) .



(e) Equalizer for two workpieces



(f) Equalizer for four workpieces

3.1.2 Clamping methods

Different types of clamps used with jigs and fixtures are classified into different categories.

STRAP CLAMP

This is also termed as edge clamp. This type clamping is done with the help of the lever pressure acting as a strap on the workpiece.

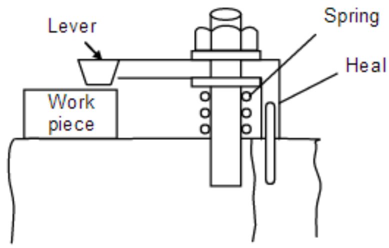
Types of strap clamps:

- Heel Clamp
- Bridge Clamp
- Edge Clamp or Side Clamp
- Screw Clamp
- Latch clamp
- Equalizing Clamps
- Power Driven Clamping

Heel Clamp

The simple form of a heel clamp is shown in the figure below. Rotation of the clamp in the clockwise direction is prevented and it is allowed in the anticlockwise direction.

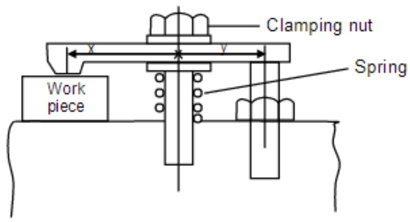
For releasing the workpiece, clamping nut is unscrewed. Free movement in the anticlockwise direction takes place before un-securing the nut to release the workpiece.



Heel Clamp

Bridge Clamp

The bridge clamp is shown in figure below. It applies more clamping pressure as compared to the heel clamp. The clamping pressure experienced by the workpiece depends on the distances “x” and “y” marked in the figure. To release the workpiece, clamping nut is unscrewed. The spring lifts the lever to release the workpiece.

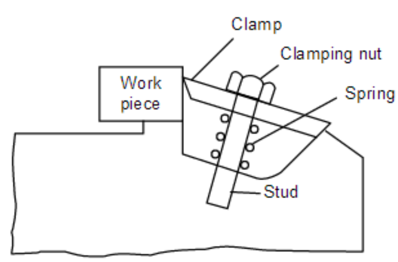


Bridge Clamp

Edge Clamp or Side Clamp

Side clamp is also termed as edge clamp. In this case, the surface to be machined is always clamped above the clamping device. This clamping device is recommended for fixed length workpiece.

The clamping device is illustrated in the below figure. Releasing and clamping of the workpiece is accomplished by unscrewing and screwing of the clamping nut respectively.



Edge clamp or side clamp

Screw Clamp

The screw clamp is shown in the below figure. It is also known as clamp screw. This clamping apply pressure directly on the side faces of the workpiece.

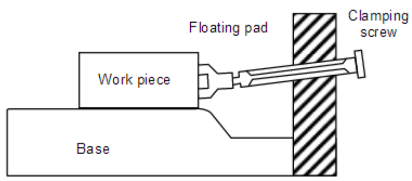
There is a floating pad at their end to serve the following purposes:

(a) It prevents the displacement of workpiece and slip.

(b) The available cushion prevents deflection of screw.

(c) It prevents the denting of clamping area of the workpiece.

There are some disadvantages associated with this method. The clamping pressure largely depends on the workpiece, it varies from one workpiece to other. It is more time consuming and more efforts are required.

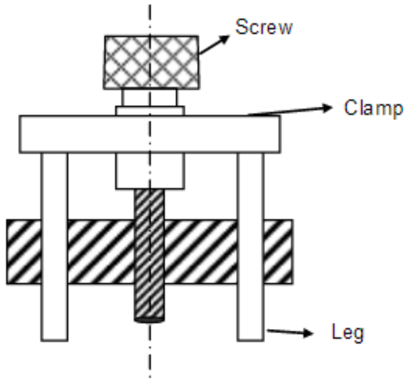


Screw clamp

Latch Clamp

Latch clamps are used to clamp the workpiece, clamping system is normally locked with the help of the latch provided. To unload the workpiece, tail end of the latch is pushed which causes the leaf to swung open, so releasing the workpiece. Here, time consumed in the loading and unloading is very less as no screw is tightened but the clamping pressure is not so high as in the other clamping devices. Life of this clamping device is small.

Equalizing Clamps



Equalizing clamp

Equalizing clamp is shown in the figure above . It is recommended to apply the equal pressure on the two faces of the work. The pressure applied can be varied by tightened or loosening the screw provided for the purpose.

Power Driven Clamping

Light duty clamps are used manually because small power is required to operate these clamps. Hand clamping leads to the application of variable pressure, operator’s fatigue and more time consumed. The power driven clamping overcomes the above mentioned problems of hand clamping. Power clamps are operated on the base of hydraulic or pneumatic power. Power clamps are high pressure clamping, these are quick acting, easily controllable, reliable and less time consuming.

3.1.3 Locating methods

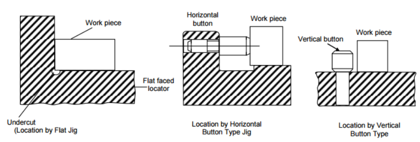
There are different methods used for location of a work. The locating arrangement .Type of operation, degree of accuracy required. Volume of mass production to be done also mattes a lot.

Different locating methods are described below.

- Flat Locator
- Cylindrical Locators
- Conical Locator
- Jack pin locator
- Drill Bush
- Vee bush locator

Flat Locator

Flat locators are used for location of flat machined surfaces of the component. Three various examples which can be served as a general principle of location are described here for flat locators. These examples are illustrated in figure given below.



(a) (b) (c)

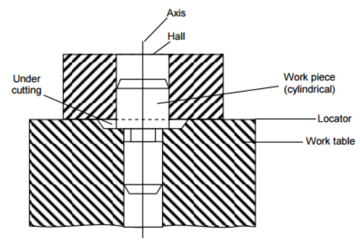
Method of Locating using flat locators

A flat surface locator can be used as shown in the figure(a). In this case an undercut is provided at the bottom where two perpendicular surfaces intersect each other. This is made for swarf clearance. The middle figure shows flat headed button type locator. There is no need to made undercut for swarf clearance.

The button can be adjusted to decide very fine location of the workpiece. There can be a vertical button support as shown in the figure(c), which is a better arrangement due to its capacity to bear end load and there is a provision for swarf clearance automatically.

Cylindrical Locators

A cylindrical locator is shown in figure below. 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's body and inserted in the drilled hole of the component. The face of the jig's body around the locator is undercut to provide space for swarf clearance.

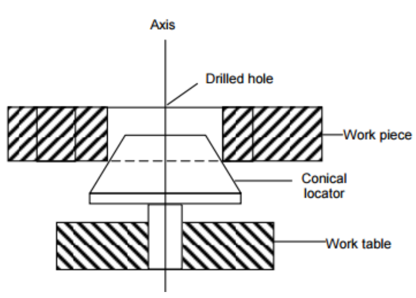


Cylindrical Locator

Conical Locator

A conical locator is shown in figure below. 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.

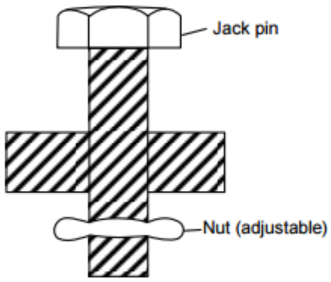
A conical locator is considered as superior as it has a capacity to accommodate a slight variation in the hole diameter of the component without affecting the accuracy of location. The degree of freedom along z-axis can also be restrained by putting a template over the workpiece with the help of screws.



Conical Locator

Jack Pin Locator

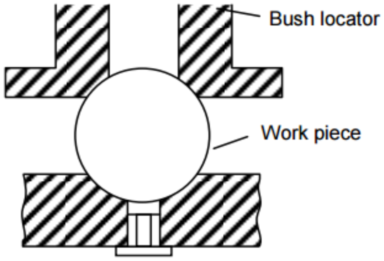
Jack pin locator is used for supporting rough workpieces from the button as shown in the figure below. Height of the jack pin is adjustable to accommodate the workpieces having variation in their surface texture. So this is a suitable method to accommodate the components which are rough and un-machined.



Jack Pin Locator

Drill Bush

The drill bush locator is illustrated in the figure below. It is used for holding and locating the cylindrical workpieces.The bush has conical opening for locating purpose and it is sometimes screwed on the jig"s body for the adjustment of height of the work.

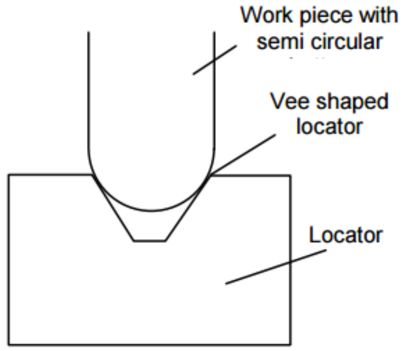


Drill Bush Locator

Vee Locators

This is effective and quick 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 the figure below. 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.



Fixed V Locator

3.2 Drill Jig bushing

Drill Jigs use bushes to guide drills, reamers and other cutting tools to the workpiece. Bushes are made of water hardening carbon steel with 0.85-1% carbon and 0.5-0.9% manganese and are hardened to Rc 60-64 to minimise wear due to contact with hard, rotating tools. Bushes are generally finished by grinding the inside and outside diameters within 0.001 mm concentricity.

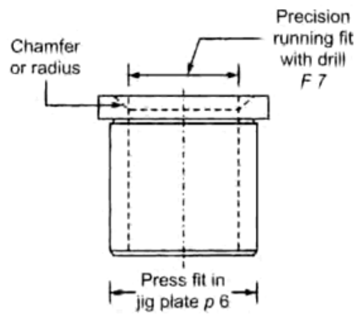
The inside diameter is ground precision running fit (F7) with the drill and reamer which needs to be guided whereas the outside diameter is made press fit (p6), precision location fit (h6) or precision running fit (fb), depending upon the function and application of the bush.

Press fit bushes an the most common type of bushes and are pressed interference fit in the bush plates also referred to as jig plates. These bushes are used in batch production where the bushes often outlast the life of the jig.

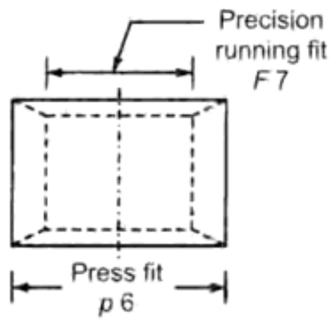
Headed bushes are preferable to headless bushes because the collar provides positive stop against the jig plate. Moreover, it is found that the chances of the bush getting loose in the jig plate and sliding axially with the drill are lesser in the collared bushes.

Hence, when the spacing of the bushes is close or the top surface of the jig plate is required to be free from the projecting collars, headless bushes are used Press fit bushes are also used as liners for renewable and slip bushes. Bushes fitted in soft materials are sometimes knurled instead of machining to p6 and s6 tolerance to provide more interference with the softer housing.

The knurling is also used to salvage interference fit parts whose external diameter is machined underside in advertently, due to error or lack of skill. Although far from good the remedy helps salvage, otherwise unusable, costly parts with many operations.



Headed collared press fit bush



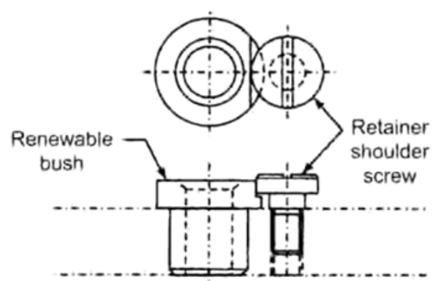
Headless press fit bush

Renewable Bushes

This is continuous or large batch production, the inside diameter of the bush is subjected to severe wear due to continuous contact with hard cutting tool. The guide bushes require periodic replacement. The replacement is simplified by making the outside diameter precision location fit (h6).

The bushes can then be assembled manually without any press. The use of liner in the jig plate provides hardened wear resistant mating surface to the renewable bush. The renewable bush must be prevented from rotating and moving axially with the cutting tool. This is accomplished by provision of a flat on the collar.

The fiat arrests with the collar of the retainer shoulder screw to prevent rotation. The bush flange below the collar of the shoulder screw prevents the bush from getting lifted with the cutting tool.



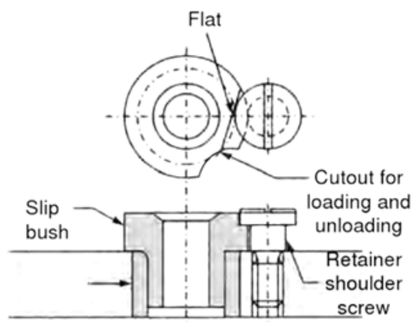
Renewable bush

Slip Bushes

When a hole in the workpiece requires two operations such as drilling and reaming, it is necessary to use two different guide bushes for the different tads. The hole is first drilled using a bush having a bore suitable for the drill. Alter drilling, the drill bush is removed and a reaming bush is used to guide the reamer. In mass production the changeover of these bushes should be effected quickly.This is accomplished by provision of slip bushes.

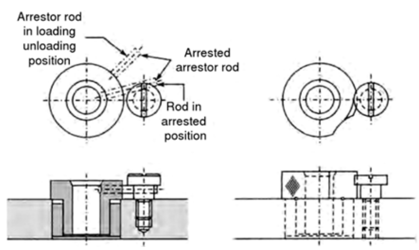
There are a number of different types of slip bushes uses. In the most common type, the bush is provided with a flat on the head similar to the renewable bush and a circular cutout in the flange to facilitate quick assembly and removal For loading or unloading of the slip bush, the cutout in the flange is aligned with the collar of the retainer shoulder screw. The bush can be moved freely axially in this position.

For assembly, the slip bush is aligned with the shoulder screw and insened into a liner. When the bush collar touches the jig plate, the slip bush is rotated clockwise to arrest the flat on the bush flange against the collar of the retainer shoulder screw. This prevents rotation of the bush during drilling.



Slip bush

The bush flange below the collar of the shoulder screw prevents the slip bush from rising up with the coning tool. For removal the slip bush is rotated anti-clockwise to align the bush cutout with the shoulder screw collar. Then the bush can be lifted axially out of the liner. In a variation of the slip bush, the straight flat is replaced by a circular step.



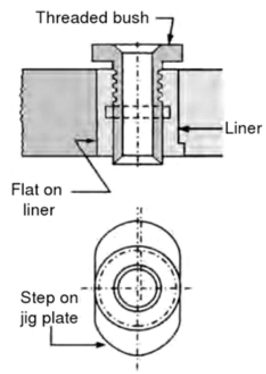
(a) (b)

Other type of slip bush

In another design, the head of the slip bush is fitted with a rod as shown in the above figure. For assembly, the rod is turned to arrest it against the shoulder of the retaining screw to prevent rotation. In this position, the collar of the shoulder screw prevents the rod and the bush from being lifted with the cutting tool. For removal, the bush is rotated anti-clockwise to turn the rod clear of the collar of the shoulder screw. Then, it can be removed axially from the finer.

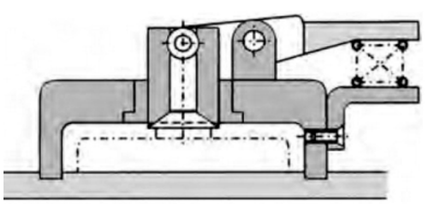
Threaded Bushes

The bushes used for clamping the workpiece are threaded on the outside. There should be another plain guiding diameter for accurate location of the bush shown in the figure below. The collar of the liner bush is usually placed on the opposite side to take the axial thrust of the screw. The liner bush should be prevented from rotation by a grub screw or a flat on the collar. The flat mates with a machined step on the jig plate.



Threaded bush

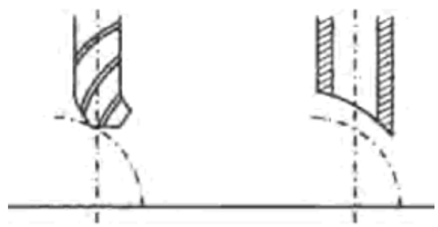
Alternatively, an unthreaded bush can be clamped with a spring-loaded lever. The pressure of the spring holds the bush pressed against the workpiece boss. For unclamping, the lever is pressed down to compress the spring. This talus the pivoted bush up, thus. releasing the hold on the workpiece.



Spring loaded clamping of threaded bush

Special Bushes

Some workpieces or operations require unusual types of bushes. In many instances, these involve simple modifications in the standard bush. A twist drill tends to slide down inclines curves. This causes bending and breakage of the drill. The problem can be countered by altering the shape of the drill bush to provide be user support and resistance against bending. This generally involves matching the end of the bush with the profile of the workpiece.



Drill bush for curved surface

Drill bush for inclined surface

Sometimes, the centre of the drilled holes are placed so close that it is just impossible to provide any drill bushes in the jig plate. The bushes shown by the chain dotted lines in figure obstruct both other. Consequently, there would be little wall material between the two holes for the bushes. Under such circumstances, a combined plate-type of bush is used.

Problems due to close centre distance in drilling holes

The plate bush A made of tool steel. A number of bushes can be combined into a single plate bush. The plate bush A screwed and do welled to the jig plate.

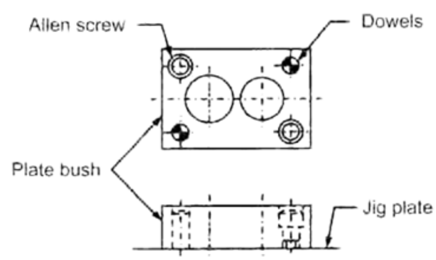


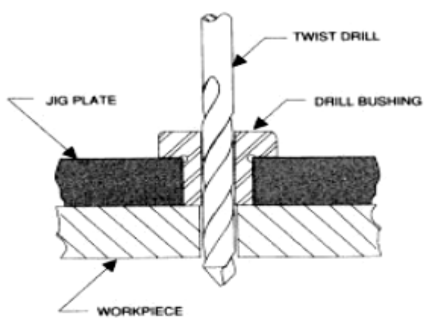
Plate bush

3.2.1 Indexing type drilling Jig

Indexing jigs are used to drill holes on periphery of cylindrical work at the required angular positions. An indexing device is provided in the jig.

Over the past century, manufacturing has made considerable progress. New machine tools, high performance cutting tools and modern manufacturing processes enable today's industries to make parts faster and better than even before. Although work holding methods have also advanced. Considerably the basic principles of clamping and locating are still the same. Jigs and fixtures form an important category of equipment that goes a long way in achieving productivity.

A fixture references the cutting tool. The differentiation between these types of work holders is in their relation to the cutting tool. As shown in the figure, jigs use drill bushings to support and guide the tool fixtures, figure use set blocks and thickness or feeler, gages to locate the tool relative to the work piece.



A jig guides the cutting tool, in this case with a bushing

In the shop, drill jigs are the most-widely used form of jig. Drill jigs are used for drilling, tapping, reaming, chamfering, counter boring, countersinking and similar operations.

Jigs are further identified by their basic construction. The two general forms of jigs are open and closed. Open jigs carry out operations on only one or sometimes two, sides of a work piece.

A closed jigs, on the other hand, operate on two or more sides. The most common open jigs are template jigs,table jigs, plate jigs, sandwich jigs and angle plate jigs. Typical examples of closed jigs include box jigs, channel jigs and leaf jigs.

Design Considerations:

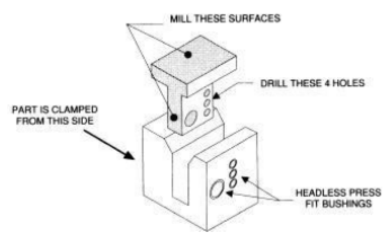
The principal considerations when choosing among work holder varieties fall into three general categories, tooling cost, tooling details and tooling operation. Although both of these categories is separated here in practice they are interdependent. The following are some design differences and considerations for permanent, general-purpose and modular work holders.

Tooling Details

Tooling details are the overall construction characteristics and special features incorporated into the jig or fixture. Permanent work holders are designed and built to last longer than temporary work holders. So permanent jigs and fixtures usually contain more elaborate parts and features than temporary work holders.

Tooling Operation

The performance of any work holder is critical to the complete usefulness of the tool. If the work holder cannot perform the functions desired in the manner intended, it is completely useless, regardless of the cost or the extent of the process.



A combination drill jig and milling fixture used for both types of operations on the same part.

The jigs and fixture must satisfy the following conditions:

Reduction Of Idle Time:

The design of jigs and fixtures should be such that the process of loading and unloading the component takes the minimum possible time and enables on easy loading and clamping should be such that idle time is reduce to minimum.

Provision For Coolant:

The jigs and fixtures must have adequate arrangement for the cutting edges of the tools so that the tool is cooled and at the same time the swarf or chips produced are washed away, so that the operator does not have to waste time in adjusting the coolant flows and cleaning of the swarf or chips.

Hardened Surfaces:

All locating and supporting surfaces such as faces of locating pins should be hardened materials as far as conditions permit, so that they are not quickly worn out and their accuracy is retained for a longer time.

Safety:

The design of jigs and fixtures should be such that it should not constitute a danger to operator.

Fool Proof:

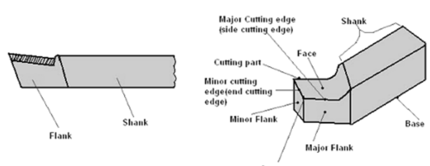
Since the use of jigs and fixtures allows for the employment of unskilled workmen, the design of such equipment should be such that it would not permit the work piece or the tool to be inserted in any position other than the correct one.

Indexing Type Of Jig;

These types of jigs are used to drill a series of holes in a circle, on the face of a work piece. The work piece is indexed and the next place the hole is to be drilled, comes under the jig bush, with the component clamp in one position of the jig, after each hole has been drilled there the single bush, etc. The work is indexed there 60 degree and the previously drilled hole located by the angular pin.

3.3 Design of single point cutting tool

A single point cutting tool consists of a sharpened cutting part and the shank and main parts or elements which are:



Nomenclature of single point cutting tool

1. Heel: It is the intersection of the flank and the base of the tool.

2. Flank: The surface or surfaces below the adjacent to the cutting edge is called flank of the tool.

3. Face: The surface on which the chip slides is called the face of the tool.

4. Shank: Is the main body of the tool

5. Nose: It is the point where the side cutting edge and end cutting edge intersect.

6. Cutting Edge:

It is the edge on the face of the tool which removes the material from the work piece. The cutting edge consists of the side cutting edge (major cutting edge) and cutting edge(minor cutting edge) and the nose.

Types and applications of different types of cutting tools.

All cutting tools can be divided into two groups. These are

• Single point tool

• Multi point tool

Single point cutting tools having a wedge like action find, a wide application on lathes and slotting machines, etc.

Multi-point cutting tools are merely two (or) more single point tools arranged together as a unit.

Types of Single Point Lathe Tool:

(1) According to the method of manufacturing the tool.

(a) Forged tool.

(b) Tipped tool brazed to the carbon steel shank.

(c) Tipped tool fastened mechanically to the carbons tool shank.

2) According to the method of holding the tool

(a) Solid tool.

(b) Tool bit inserted in the tool holder.

3) According to the method of using the tool

(a) Turning

(b) Chamfering

(c) Thread cutting

(d) Facing

(e) Grooving

(f) Forming

(g) Boring

(h) Internal thread cutting

(i) Parting off

4) According to the method of applying feed.

(a) Right hand

(b) Left hand

(c) Round nose

Types of Multi point Tool:

- Plain milling cutter
- Side milling cutter
- Metal slitting saw
- Angle milling cutter
- End milling cutter
- T-slop milling cutter
- Wood ruff key slot milling cutter
- Fly cutter
- Formed cutter
- Tap and reamer cutter.

APPLICATION OF DIFFERENT TYPES OF CUTTING TOOLS

Operation	Application
1) Turning	Turning in a Lathe is to remove excess material from the work piece to produce a cone shaped (or) a cylindrical

	surface.
2) Chamfering	Chamfering is the operation of beveling the extreme end of a work piece. This is used to remove the burrs, to protect the end of the work piece from being damaged and to have a better book.
3) Facing	Facing is the operation of machining the ends of a piece of work to produce a flat surface .
4) Grooving	Grooving is the process of reducing the diameter of a work piece over a very narrow surface.
5) Boring	Boring is the operation of enlarging and truing a hole produced by drilling punching, casting, etc.
6) End milling	The end mills are used for light milling operations like cutting slots, machining. Accurate holes, producing narrow flat surfaces and for profile milling operations.
7) Plain milling	Used to produce flat, horizontal surfaces.

Design of single point cutting tool is an important aspect of tool engineering. This unit deals with the design of tool shank, design of single point cutting tool and various forces involved during machining of the workpiece. Strength and rigidity of tool is also taken into account while designing single point cutting tool.

Objectives

After studying this unit, we should be able to ,

• Design tool shank,

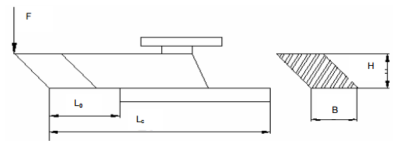
• Design single point cutting tool,

• Select appropriate tool material.

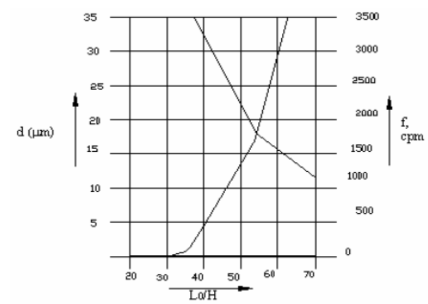
• Calculate and analyze the forces acting on tool.

DESIGN OF TOOL SHANK

The shank of a cutting tool is generally analyzed for strength and rigidity. Tool is assumed to be loaded as a cantilever by tool forces at the cutting edge as shown in the figure.



Forces Acting on Tool Shank



Deflections and Frequency of Chatter for Several Overhung Values

The notations used in design of shank is given below :

F = Permissible tangential force during machining, N

H = Depth of shank, mm

f = Chatter frequency, cycle per second (c.p.s)

B = Width of shank, mm

L_0 = Length of overhung, mm

σ_{per} = Permissible stress of shank material, N/mm^2

d = Deflection of shank, mm

I = Moment of inertia, mm^4

h_c = Height of centres, mm

E = Young's modulus of material, N/mm^2

$\sigma_u t$ = Ultimate tensile strength, N/mm^2

L_c = Length of centres, mm

The main design criterion for shank size is rigidity. The deflection at the cutting edge is limited to a certain value depending on the size of machine, cutting conditions and tool overhung. The tool overhung (L_0) is related also to the shank size as well as to the end support conditions.

The figure shows above the graph of amplitude and frequency of chatter for several overhung values. It is seen from Figure that only below L_0/H = 2, the amplitude is practically zero. The recommended value of (L_0/H) lies between 1.2 and 2. For the given value of chatter frequency f, the shank deflection can be calculated from the given as follows.

$$f = \frac{(15.76)}{\sqrt{d}} \text{ c.p.s.} \qquad \text{....(1)}$$

where,

d -Deflection in mm.

Now as chatter frequency ranges from 80 to 160 c.p.s.,

Let,

f = 100 c.p.s

d = $(15.76/100)^2$ = 0.025 mm....(2)

The permissible deflection of shanks ranges from 0.025 mm for finish cuts to 0.9 mm for rough cuts. Considering shank as a cantilever,

$$d = \frac{FL_0^3}{3EI}$$

$$d = \frac{FL_0^3}{3E}\left(\frac{12}{BH^3}\right) = \frac{4FL_0^3}{EBH^3} \qquad \text{...(3)}$$

It can be noted that the same value of d has been obtained from equation 2 also. The shank size can be estimated with respect to machine tool size by the following method :

(a) The force F for given size of lathe is given by,

F = f × t × C

where,

f - Feed in mm,

t - Depth of cut in mm,

C -Cutting force constant.

(b) Nicol sons Manchester experiments have set a standard area of cut for lathe design given by,

$A_c = f \times t$

Let, $f = h_c/180$ mm

$t = h_c/25$ mm

$$A_c = \frac{h_c}{180} \times \frac{h_c}{25}$$
$$= \frac{h_c^2}{4500} \text{ mm}^2$$

where,

h_c - Height of centre in mm,

Let,

$\sigma_{ut} = 440$ N/mm^2

C = 4σut

= 4 × 400

= 1760 N/mm^2

When

$$F = \frac{h_c^2}{4500} \text{ mm}^2 \times 1760 \text{ N/mm}^2$$

$$= 0.4\, h_c^2 \text{ N}$$

On substituting the value of $F = 0.4h_c^2$

$$d = \frac{4(0.4\, h_c^2)\, L_0^3}{EBH^3}$$
$$0.025 = \frac{4(0.4\, h_c^2)\, L_0^3}{EBH^3}$$

As, d = 0.025 from equation 2. Thus,

$$= \frac{(1.6\, h_c^2)\, L_0^3}{EBH^3}$$

B = 0.6 H for rectangular shanks

$$\text{Then, } \frac{h_c^2}{H^4} = 0.6\, \frac{ED}{L_3^0}$$

Let L_0 = 3 mm, E = 200 kN/mm^2

and d = 0.025 mm, (From equation 2)

On substituting these values in above equation, i.e.

$$\frac{h_c^2}{H^4} = 0.6\, \frac{ED}{L_3^0}\text{, we get}$$

$$\frac{h_c^2}{H^4} = 1000\ \text{mm}^{-2}$$

Table shows the standard shank size according to this rule.

Height of Centres Hc(mm)	**Shank size**	
	H (mm)	**B (mm)**
250	20	12
300	30	20
350	40	25

Usually the shank size is also checked for strength.

Noting,

$$FL_0 = \frac{1}{6} BH^2 \sigma_1$$

$$\sigma_1 = \frac{6FL_0}{BH^2}$$

When the effect of F_x included,

$$\sigma = \sigma_1 + \sigma_2 = \frac{6FL_0}{BH^2} + 6\, F_x \frac{L_0}{HB^2} \quad ....(4)$$

F_x = Component of force F acting in x direction (in Newton)

F_x = 0.3 to 0.40 F

Hence,

$$\sigma = \frac{6FL_0}{BH}\left(\frac{0.4}{B}\right) + \left(\frac{1}{H}\right) < \sigma_{per} \quad ....(5)$$

This can be expressed as,

$$F = \left\{ \frac{BH}{\left(\frac{0.4}{B}\right)} + \left(\frac{1}{H}\right) \right\} \frac{\sigma_{per}}{6L_0} \quad ....(6)$$

where,

F - Permissible tangential force during machining.

The maximum depth of shank (H_{max}) must be less than the value h_k as shown in table.

h_k(mm)	11	14	22	28	45	56
H_{max}(mm)	10	12	20	25	40	50

Problems :

1. The end of a pipe was orthogonally cut with a tool of 20° rake angle. The cut chip length was 85 mm corresponding to uncut chip length of 202mm. If the depth of cut was 0.5mm, let us determine the chip thickness and shear plane angle.

Solution:

Given:

Rake angle, $\alpha = 20°$

Cut chip length, $l_c = 85mm$

Un cut chip length, $l = 202mm$

Depth of cut, $t = 0.5mm$

To Find:

(i) Chip thickness.

(ii) Shear plane angle.

Formula To Be Used:

$$r = \frac{t}{t_c} = \frac{l_c}{l}$$

$$\frac{0.5}{t_c} = \frac{85}{202}$$

$$0.5 = 0.421 \times t_c$$

$$\Rightarrow t_c = \frac{0.5}{0.421}$$

$$\tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha}$$

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

$$= \frac{0.5}{1.188}$$

(i) Chip thickness ratio

$$r = \frac{t}{t_c} = \frac{l_c}{l}$$

$$\frac{0.5}{t_c} = \frac{85}{202}$$

$$0.5 = 0.421 \times t_c$$

$$\Rightarrow t_c = \frac{0.5}{0.421}$$

t_c =1.188 mm.

(ii) Shear phase angle:

Where,

$$\tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha}$$

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

$$= \frac{0.5}{1.188}$$

r = 0.421

$$\tan\phi = \frac{0.421 \times \cos 20^\circ}{1-(0.421 \times \sin 20^\circ)}$$

$$\tan\phi = \frac{0.3956}{0.856}$$

ϕ = 24,8°.

2. A specimen of 100 mm length along the stroke of shaper is machined with a tool with 15° rake angle. The uncut chip thickness is 1.5mm. If a chip length of 40 mm is obtained during one stroke of machining, let us determine the shear plane angle and the thickness of cut-chip.

Solution:

Given:

Specimen length, l = 100 mm

Rake angle, α = 15°

Un cut chip thickness, t = 1.5 mm

Chip length , l_c = 40 mm.

To find:

i) Chip thickness

(ii) Shear plane angle.

Formula To Be Used:

$$r = \frac{t}{t_c} = \frac{l_c}{l}$$

$$\frac{0.5}{t_c} = \frac{85}{202}$$

$$0.5 = 0.421 \times t_c$$

$$\Rightarrow t_c = \frac{0.5}{0.421}$$

$$\tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha}$$

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

$$= \frac{0.5}{1.188}$$

(i)Chip thickness ratio

$$r = \frac{t}{t_c} = \frac{l_c}{l}$$

where

t = Cuts chip thickness

t_c = Chip thickness

l_c = Length of cut chip

l = Length of incut chip

$$\Rightarrow \frac{t}{t_c} = \frac{l_c}{l}$$

$$\frac{1.5}{t_c} = \frac{40}{100}$$

Thickness of cut chip

$$t_c = \frac{100 \times 1.5}{40}$$

t_c = 3.75 mm ▯ r = 0.4

Shear plane angle

$$Tan\phi = \frac{r.\cos\alpha}{1 - r\sin\alpha}$$

$$Tan\phi = \frac{0.4 \times \cos 15^o}{1 - (0.4 \times \sin 15^o)}$$

$$Tan\phi = \frac{0.3864}{1 - 0.10353}$$

$$Tan\phi = \frac{0.3864}{0.8965}$$

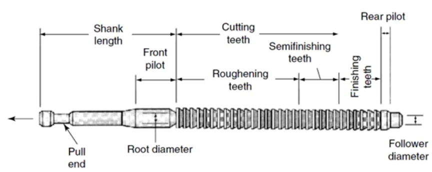
Shear Plane angle ϕ = 23.31°.

3.3.1 Broach and form tool

• Broaching is a process of machining a surface with a special multi point cutting tool called broach which has successively higher cutting edges in a fixed path.

• Broaching is a process of machining a surface with a special multi point cutting tool called broach which has successively higher cutting edges in a fixed path.

• Broaching is a machining process in which metal removal takes place with the help of a number of successive teeth incorporated on the broach. Cutting takes place by a transverse cutting action, by pushing or pulling the broach through the hole or surface. Broaching is an efficient and rapid process, because roughing and finishing operations can be done in a single pass.



Broaching machine

• Pull end for engaging the broach in the machine

• Neck of shorter diameter and length,where the broach is allowed to fail, if at all, under overloading

• Front pilot for initial locating the broach in the hole

• Roughing and finishing teeth for metal removal

• Finishing and burnishing teeth for fine finishing

• Rear pilot and follower rest or retriever

Broach Tool:

• It is a cutting tool having multiple transverse cutting edges.

• Wide range of application and several advantages over the machining process.

• Roughing and finishing operation in single pass.

• Close tolerances, smooth surface finished and higher accuracies are threaded advantages.

Form tool:

• It is a cutting tool having one or more cutting edges with define profile or contour that will be reproduced as the workpiece surface.

• The use of a form tool ensures high output, uniform contour and uniform dimensions.

• It is a tool adopted for mass production

Process of Broaching Tool: Broaching is the machining process that uses a toothed tool, called a broach to remove material.

Types of broaching machine

The broaching machine may be classified as follows

1) According to the nature and direction of primary cutting motion

a)Horizontal broaching machine.

b)Vertical broaching machine.

c)Continuous broaching machine.

2) According to the purpose

a.Internal broaching machine.

b.External surface broaching machine.

3) According to method operation

a)Pull broaching machine.

b)Push broaching machine

Method of operation :

Push up or down, pull up or down.

Construction:

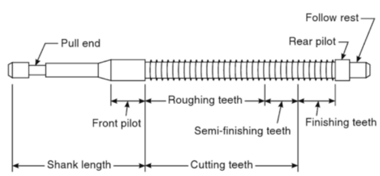
Solid, built-up, progressive, circular, inserted-tooth, rotary cut, overlapping tooth.

Internal broaching:

Shown in the figure the broaching of internal surfaces, such as the sides of holes. This method is known as internal broaching.

This can be done only when the surfaces to be broached are parallel to the direction of the broach cutting stroke and when there is no obstruction in the path of the broach.

A blind hole cannot be broached, because the bottom of the hole is an obstruction and does not allow the broach to pass completely through the hole. Openings of various shapes can be produced by internal broaching. Square holes, internal splines, holes with keyway and other holes which are not round can be produced from drilled holes by internal broaching. Broaching gradually changes the sectional shape of the initial round hole to the desired internal shape. A sectional broach is passed completely through the opening.



Internal broaching

At the beginning of internal broaching, the front end of the broach is inserted through the initial hole in the workpiece. The front pilot of an internal broach serves to locate the broach properly in the workpiece opening at the start of broaching. Some internal broaches are pulled or pushed through an opening in the workpiece at the start of broaching.

Some internal broaches are designed to fit the pull head of a broaching machine. Internal broaches, if they are not too slender, may be pushed through the workpiece opening. When an internal broaching is pulled or pushed through an opening in the workpiece, it is ordinarily free to follow the initial opening.

A rear pilot serves to keep the internal broaching straight during the operation until the last tooth has passed through the opening. After its cutting stroke, a broach should not be returned to its starting position by forcing it backwards through the workpieces. This would cause excessive wear on the broaching teeth.

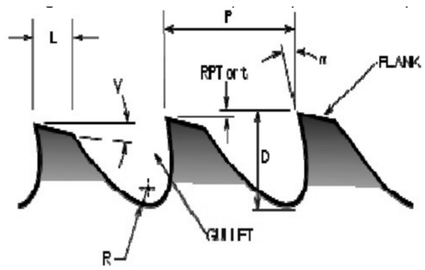
Function:

Keyway, square hole, round hole, serration, spline,combination round and spline, helical tooth, special contour and so on.

DESIGN OF BROACHES TOOL:

When compared to other cutting tools such as a milling cutter, a broach is many times costlier. Any small error committed in the design of a milling cutter or a turning tool may not result in the rejection of part or the tool. At the most it may result in reduced tool life or lesser productivity.

But in case of the broaches, such a mistake may result in the breakage of tool or rejection of parts.It is for this reason that broach design should be done more precisely and accurately.



Teeth of Broach Tool

Material to Be broached	**Depth of cut tooth (Inch)**		**Face angle and rake angle (degrees)**	**Clearance angle (Degrees)**	
				Roughing	**finishing**
	Roughing	**finishing**			
Steel, high Tensile Strength	0.0015-0.002	0.0005	10-12	1.5-3	0.5-1
Steel, Medium Tensile strength	0.0025-0.005	0.0005	14-18	1.5-3	0.5-1
Cast steel	0.0025-0.005	0.0005	5	1.5-3	0.5

Advantages:

Broach Tool

- Very high production rate (much higher than milling, planning, boring etc.)
- Roughing and finishing in single stroke of the same cutter.
- High dimensional and form accuracy and surface finish of the product.
- Extremely suitable and economics for mass production.
- Needs only one motion (cutting), design, construction, operation and control are simpler.

LIMITATIONS OF BROACH TOOL

- Only through holes and surface can be machined.
- Usable only for light cuts, i.e. low chip load and un hard materials.
- Cutting speed cannot be high.
- Design, manufacturing and restoration of the broaches are difficult and expensive.
- Economic only when the production volume is large.

FORM TOOL

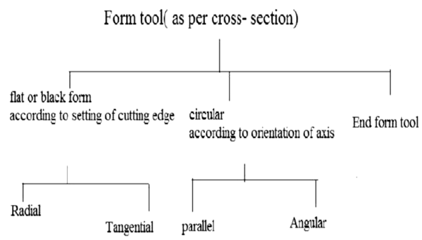
- A high output.
- Interchangeability of work pieces.
- High dimensional accuracy.

Form tools are the tools which is used to produce a complicated shape surface with the cross section outlined by curves or broken lines. The shape of the cutting edge of form tool is the mirror image of the profile required on the work piece.

Form tools are generally recommended for mass production industry.

form tool

Classification of Form Tool



DESIGN FEATURES OF FORM TOOL

Most of form tool are made of H.S.S. however, cemented carbides are increasingly being used for this purpose. The use of contoured cemented carbides tips for form tools enables productivity to be raised by 30 to 40 percent, as compared to H.S.S. form tools.

A form tool should have the proper rake and relief angles, the metal is cut under sufficiently advantageous condition.

The relief angles depend upon the type of form tool.

Relief angles is,

= 10 to 12° on circular form tool.

= 12 to 15° on flat tools.

= 25 to 30° on form tools for relieving from milling cutters.

LIMITATIONS OF FORM TOOL• When using form tools is that the feed into the work is usually slow, 0.0005" to 0.0012" per revolution depending on the width of the tool.

• Form tools wider than 2.5 times the smaller diameter of the part being turned have a greater risk of the part breaking off.

• Wide form tools create more heat and usually are problematic for chatter. Heat and chatter reduces tool life.

• From the above discussion we can conclude that Broach tool and form tool can be used in the different industries for getting high dimensional accuracy, better finish production with a higher production rate.

3.4 Design of limit gauges

A limit gauge is not a measuring gauge. They are just used as inspecting gauges. The limit gauges are used in inspection by methods of attributes. It gives the information about the products which may be either within the prescribed limit or not.

Limit Gauges

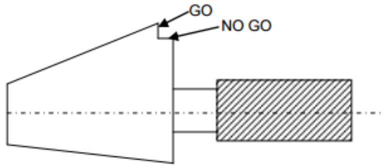
These are also called „go" and „no go" gauges. These are made to the limit sizes of the work to be measured. One of the sides or ends of the gauge is made to correspond to maximum and the other end to the minimum permissible size. The function of limit gauges is to determine whether the actual dimensions of the work are within or outside the specified limits.

A limit gauge may be either double end or progressive. A double end gauge has the „go" member at one end and „no go" member at the other end. The „go" member must pass into or over an acceptable piece but the „no go" member should not.

The progressive gauge has „no go" members next to each other and is applied to a workpiece with one movement. Some gauges are fixed for only one set of limits and are said to be solid gauges. Others are adjustable for various ranges.

The accuracy of a taper hole is tested by a taper limit gauge as shown in figure. This has two check lines „go" and „no go" each at a certain distance from the end of the face.

The go portion corresponds to the minimum and „no go" to the maximum dimension



Limited taper plug gauge

Common Types of Gauges:

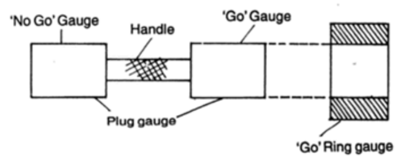
1) Plug gauges

2) Ring gauges

3) Snap gauges

1. Plug gauges:

Plug gauges are used for the measurement of internal dimensions such as cylindrical holes. The plug gauges are made of suitable wear resistant steel. The gauging surfaces are hardened, ground and lapped.



Plug and Ring gauge

The plug gauges for sizes upto 63 mm are normally double ended and single ended type for sizes above 63 mm. Smaller size plug gauges, upto 10 mm, are solid gauges while the 4arger size gauges are renewable end type gauges where the gauging portion and the handle are joined together to form a rigid assembly by a suitable locking device.

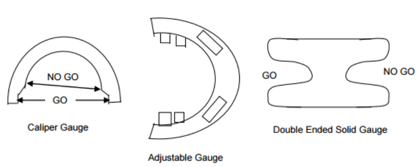
The plug gauges are designated as 'Go' and 'No Go' indicating permissible tolerance. The 'No Go' end of the gauges is always painted with a red band. While using single ended type gauges, separate gauges are used for permissible ± tolerances.

2. Ring Gauge

These are employed for the measurement of out side dimensions such as shafts. The ring gauges are generally of non-adjustable type and separate gauges are used for 'Go' and 'No Go' gauging. These are also made of wear resistant steel and the gauging surface is hardened, ground and lapped.

3. Snap Gauges

These gauges are used for checking external dimensions. Shafts are mainly checked by snap gauges. They may be solid and progressive or adjustable or double-ended. The most usual types are shown in the figure.



Snap Gauges

i) Solid or non-adjustable caliper or snap gauge with „go" and „no go" each is used for large sizes.

(ii) Adjustable caliper or snap gauge used for larger sizes.

This is made with two fixed anvils and two adjustable anvils, one for „go" and another for the „no go".

The housing of these gauges has two recesses to receive measuring anvils secured with two screws. The anvils are set for a specific size, within an available range of adjustment of 3 to 8 mm.

The adjustable gauges can be used for measuring series of shafts of different sizes provided the diameters are within the available range of the gauge.

(iii) Double-ended solid snap gauge with „go" and „no go" ends is used for smaller sizes.

The precautions to be taken while using slip gauges are:

- Slip gauges should be stored carefully in a box.
- They are stored in a temperature control system.
- Resistant to impact.
- Water resistant.
- Resistant to wear

Grades of slip gauges:

As regards grades or classes of slip gauges, these could also be designed in five grades as under:

Grade 2:

This is the workshop grade. Typical uses include setting up machine tools, positioning milling cutters and checking mechanical widths.

Grade 1:

Used for more precise work, such as that carried out in a good-class toolroom. Typical uses include setting up sine bars and sine tables, checking gap gauges and setting dial test indicators to zero.

Grade 0:

This is more commonly known as the Inspection grade and its use is confined to tool room or machine shop inspection. This means that it is the Inspection department only who has access to this grade of slips. In this way, it is not possible for these slip gauges to be damaged or abused by the rough usage to be expected on the shop floor.

Grade 00:

This grade would be kept in the Standard Room and would be kept for work of the highest precision only. A typical example would be the determination of any errors present in the workshop or Grade 2 slips, occasioned by rough or continual usage.

Calibration grade:

This is a special grade, with the actual sizes of the slips stated or calibrated on a special chart supplied with the set. This chart must be consulted when making up a dimension and because these slips are not made to specific or set tolerances, they are not as expensive as the Grade 00.

Taylors Principle:

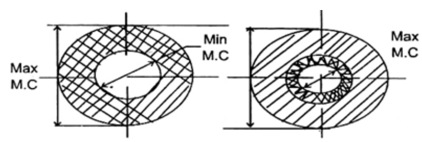
It states that Go gauge should check all related dimensions simultaneously. No Go gauge should check only one dimension at a time.

Maximum Metal Condition:

It refers to the condition of hole on shaft when maximum material is left on. i.e., high limit of shaft and low limit of hole.

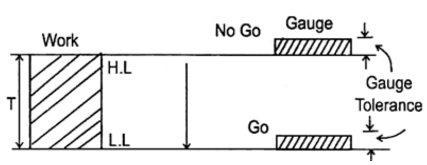
Minimum Metal Condition:

t refers to the condition of hole or shaft when minimum material is left on such as low limit of shaft and high limit of hole.



Diagrammatic representation of Minimum and Maximum metal condition

Allowances



Diagrammatic representation of Gauge allowances

Design of gauges should have manufacturing tolerance and wear allowances,

1) Manufacturing tolerance is nearly 10% of work tolerance (T).

2) Allowance varies 5.15% of work tolerance.

Statements of Taylors Principle:

1) The Go gauge must be made to check the maximum metal condition.

2) The No Go gauge must be made to check the minimum.

3) The Go gauge should be as far as possible the geometrical shape of the components.

4) For circular holes, the Go gauge should be plug gauge having length equal to length of the hole components.

5) For circular holes, the No Go gauge should be a pin gauge.

6) For circular shaft the Go gauge should be a ring gauge with length of the shaft.

7) The circular shaft, the No Go gauge should be in the form of gap gauge.

Precautionary measures to be taken at various stages of using slip gauges are:

Interferometers are used to check the surface quality of the slip gauges. The instruments consist essentially of a mercury vapor lamp whose radians are passed through a green filter, thus removing all other, colors and leaving green monochromatic light whose wavelength is very closer to 0.5µm.

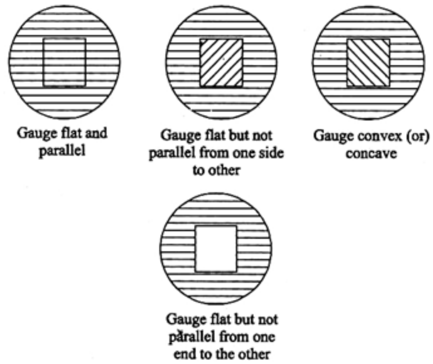
This light is focused onto a pinhole, giving an intense point source of monochromatic light which is in the focal plane beam of light. This beam is directed onto the gauge to be tested via an optical flat so that interference fringes are formed across the face of the gauge, the glass plate semi reflector set at 45° to the optical axis.

It is noted the optical flat is mounted on the adjustable tripod: Independent of the gauge base plate is designed to be rotated so that the fingers can be oriented to the best advantage.

An advantage of this instrument is that it can also be used for testing the parallelism between gauge surfaces. Two methods are used for testing the parallelism.

(a) A gauges below 25 mm in length.

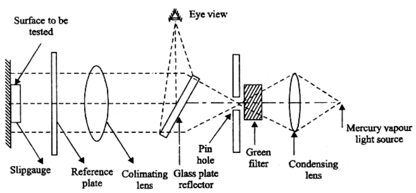
(b) A gauges greater than 25 mm in length.



Diagrammatic representation of Slip gauges types

When shorter gauges are used, interference fringes will be focused both on the gauges surface and the base plate. As the gauge is wrong on the base plate, its underside is parallel with its base plate. It means that if the gauge faces are parallel, the fringes on the base plates should be equally spaced and parallel with the fringes on the gauge surface.

If the gauge being tested is more than 25 mm in length the fringe pattern on the base plate is difficult to observe but the base plate is rotary and its underside is rapped truly parallel with its working surface. Therefore, if a non-parallel gauge is viewed the angle it makes with the optical flat. This is shown in the figure below. If the table is turned through 180°. The surface is now less parallel with the optical flat and a greater number of fringes are observed.



Diagrammatic representation of Interferometer

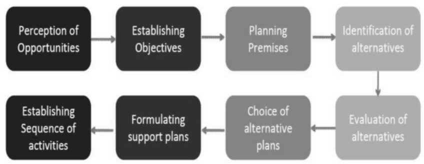
Wrapping:

Wringed or slipping is nothing but the process of combining the faces of slip gauges one over the other.

3.5 Process Planning – selection of processes

Planning process:

Planning is the management function that involves setting of goals and deciding the best method to achieve them. The various steps involved in planning are given below :



Planning process

Establishing objectives:

Planning itself is to establish objectives for the entire enterprise and then for each subordinate unit. Objectives specifying the results expected indicates the end points of what to be done, where the primary emphasis is to be placed and what is to be accomplished by the network of policies, strategies, rules, procedures, budgets and programs.

Enterprise objectives should give direction to the nature of all major plans which by reflecting these objectives, define the objectives of major departments, in turn control the objectives of subordinate departments and so on down line.

Considering the planning premises:

Another logical step in planning is to establish, obtain agreement to utilize and disseminate critical planning premises. These includes forecast data of a factual nature, applicable basic policies and existing company plans.

Premises, are planning assumptions in other words, the expected environment of plans in operation. It leads to one of the major principles of the planning. The more the individuals understand planning and agree to utilize consistent planning premises, the more coordinated the enterprise planning will be a planning premises include far more than the usual basic forecasts of population, production, prices, costs, markets. Because the future environment of plan is very complex, it would not be profitable or realistic to make assumptions about every detail of the future environment of a plan.

Since agreement to utilize the given set of premises is very important to coordinate the planning, it becomes a major responsibility of managers starting with those at the top to make sure that the subordinate managers understand the premises upon which they are expected to plan.

Identification of alternatives:

If the organizational objectives have been clearly stated and the planning premises have been developed, the manager should list as many available alternatives for reaching the objectives.The focus of this step is to search for and examine alternative courses of action particularly those not immediately apparent.

The more common problem is not finding the alternatives but reducing the number of alternatives so that the most promising may be analyzed. Even with mathematical techniques and the computer, it has a limit to the number of alternatives that may be examined. It is therefore necessary for the planner to reduce by preliminary examination of the number of alternatives to those promising the most truthful possibilities or by mathematically eliminating by the process of approximation, the least promising ones.

Evaluation of alternatives:

Having sought out alternative courses and examined their strong and weak points, the following step is to evaluate them by weighing the various factors in the light of the premises and goals.

If the only objective were to examine profits in a certain business immediately, In case if the future were not uncertain, if cash position and capital availability were not worrisome and if most factors could be reduced to the definite data, it should be relatively easy.

But typical planning is replete with uncertainties, problems of capital shortages and intangible factors and so evaluation is usually very difficult.

A company may wish to enter a new product line primarily for purposes of prestige, the forecast of expected results may show a clear financial loss but the question is still open as to whether the loss is worth the gain.

Choice of alternative plans:

An evaluation of alternatives must include an evaluation of the premises on which the alternatives are based. A manager usually finds that some premises are unreasonable and can therefore be excluded from further consideration. This process helps the manager determine which alternative would best accomplish organizational objectives.

Formulating of supporting plans:

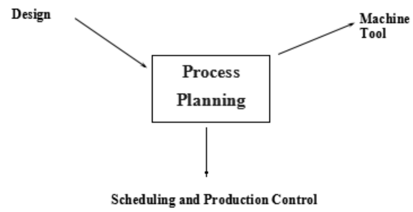
After decisions are made and plans are set, the final step to give them meaning is to number them by converting them to budgets. The overall budgets of an enterprise represent the sum total of income and expenses with resultant profit or surplus and budgets of major balance sheet items such as cash and capital expenditures.

Each department or program of a business or other enterprise can have its own budgets, usually of capital expenditures and expenses, which tie into the overall budget.

If this process is done well, budgets become a means of adding together the various plans and also important standards against which planning progress can be measured.

Establishing sequence of activities:

Once plans that furnish the organization with both long-range and short range direction have been developed they must be implemented.



3.5.1 Machines and tools.

MACHINE TOOL SELECTION

• Machine availability, cost.

• Machine capability.

Also called : operation sheet, route sheet, operation planning summary or another similar name.

• The detailed plan contains:

• Route

• Processes

• Process parameters

• Machine and tool selections

Fixtures

• How detail the plan is depends on the application.

• Operation: a process

• Operation Plan (Op-plan): contains the description of an operation, includes tools, machines to be used, process parameters, machining time, etc.

• Op plan sequence: Summary of a process plan.

EXAMPLE PROCESS PLANS

PROCESS PLAN — ACE Inc.

Part No. S0125-F — Material: steel 4340Si
Part Name: Housing
Original: S.D. Smart Date: 1/1/89 — Changes: Date:
Checked: C.S. Good Date: 2/1/89 — Approved: T.C. Chang Date: 2/14/89

No.	Operation Description	Workstation	Setup	Tool	Time (Min)
10	Mill bottom surface1	MILL01	see attach#1 for illustration	Face mill 6 teeth/4" dia	3 setup 5 machining
20	Mill top surface	MILL01	see attach#1	Face mill 6 teeth/4" dia	2 setup 6 machining
30	Drill 4 holes	DRL02	set on surface1	twist drill 1/2" dia 2" long	2 setup 3 machining

Detailed Process Plan

FACTORS AFFECTING PROCESSPLAN SELECTION

- Shape
- Tolerance
- Surface finish
- Size
- Material type
- Quantity
- Value of the product
- Urgency
- Manufacturing system itself, etc.

Process Planning Classification

- Manual
- Computer-Aided Variant
- GT based
- Computer aids for editing
- Parameters selection

GENERATIVE

- Some kind of decision logic
- Decision tree/table

• Artificial Intelligence

• Objective-Oriented

• Still experience based

Automatic

• Design understanding

• Geometric reasoning capability

3.6 Design of sequence of operations

Quite often the tool designer must design more than one tool for a part. When this is the case, the sequence of operations should be determined as well as which tool to design first. An example, if a drill jig for a part is designed first, then the holes provide an excellent location for the milling fixture that is needed in the next operation.

3.6.1 Time & cost estimation.

Quality time and cost estimates are the bedrock of project control. Past experience is the best starting point for these estimates. The quality of estimates is influenced by other factors such as people, technology and downtimes.

The key for getting estimates that represent realistic average times and costs is to have an organization culture that allows errors in estimates without incriminations. If times represent average time, we should expect that 50% will be less than the estimate and 50% will exceed the estimate. The use of teams that are highly motivated can help in keeping task times and costs near the average. For this reason, it is crucial to get the team to buy into time and cost estimates.

Using top-down estimates is good for initial and strategic decision making or in situations where the costs associated with developing better estimates have little benefit. However, in most cases the bottom-up approach to estimating is preferred and more reliable because it assesses each work package, rather than the whole project, section or deliverable of a project.

Estimating time and costs for each work package facilitates development of the project schedule and a time-phased budget, which are needed to control the project as it is implemented. Using the estimating guidelines will help eliminate many common mistakes made by those unacquainted with estimating times and costs for project control. Establishing a time and cost estimating database fits well with the learning organization philosophy.

The level of time and cost detail should follow the old saying of "no more than is necessary and sufficient." Managers must remember to differentiate between committed outlays, actual costs and scheduled costs. It is well known that upfront efforts in clearly defining project objectives, scope and specifications vastly improve time and cost estimate accuracy.

Finally, how estimates are gathered and how they are used can affect their usefulness for planning and control. The team climate, organization culture and organization structure can strongly influence the importance attached to time and cost estimates and how they are used in managing projects.

Cost Estimation

An approximation of the probable cost of a product, program or project, computed on the basis of available information.

The four common types of cost estimates are:

Planning estimate:

A rough approximation of cost within the reasonable range of values, prepared for information purposes only. Also called ball park estimate.

Budget estimate:

The approximation based on well-defined (but preliminary) cost data and established ground rules.

Firm estimate:

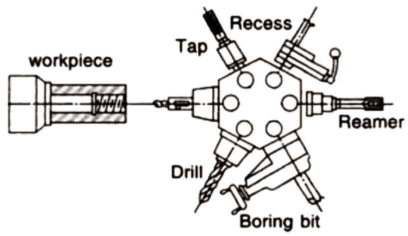
A figure which is based on cost data which is sound enough for entering into a binding contract.

Not-to exceed /Not-less-than estimate:

The maximum or minimum amount which is required to accomplish a given task, based on a firm cost estimate.

3.7 Tooling design for turret lathe and automats.

The cutting occasion for a certain process is mostly controlled by correct tooling, speed with feed. Greatly time is saving by intriguing joint or multiple cuts. In bar work, mutual cuts give extra support to work with eliminates springing action with chatter. The process of tooling and sequence of operation for creation internal threads on a component it occupy the following steps.



Capstan And Turret Lathes of tool layout

1. Proceed the bar stock beside combined stock stop. Place the correct position by drill and clamp the job in collect. Advance the start drill in job after centering work piece.

2. Recess a groove for thread consent. The process is performing by a rapid acting slide tool mount on boring bar.

3. Bore thread diameter to necessary size.

4. Drill the job to the necessary length.

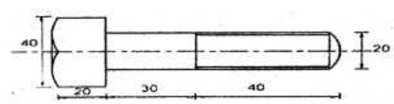
5. Part off job by a parting tool.

6. Ream diameter to precise sizes.

7. Cut the threads by the tap.

Tool layout of a turret lathe

A tool layout is prepared for the manufacture of square headed bolt from a square bar stock



Using a turret lathe

Stage-I

1. The component drawing is drawn

2. The tool length of the work is calculated and 10mm is added to provide clearance.

3. The number of operations involved is roughly listed.

4. The sequence of operation is assigned.

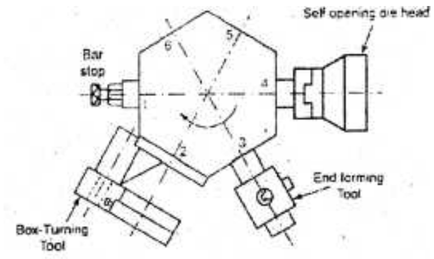
5. The proper machine of 75mm turret lathe is selected.

6. The proper material of mild steel square bar is selected.

7. All the tools and equipment's as per operation sequence are collected and fitted on turret faces or on cross-slides as per our convenience.

Stage-2

1. The tool layout is drawn as shown in figure.



Tool layout

Note: Number tools fitted in the turret face are only four. So, for providing uniform balancing tools are arranged like the first two are in Successive faces and other two are in next successive faces by leaving one lace left to free.

Stage 3

Tooling schedule chart (to machine square bolt)

MACHINE: 75mm turret lathe.

MATERIAL: Square mild steel bar.

Description of operations to be performed:

Operation sequence **Description of operation**	**Tool position**	**Tool**
1.Holding the square bar in collet and setting the required length of 100mm (90 + 10).	Turret positions-1	Bar stop
2.Turn to 20mm. diameter to a length 70mm (from the right end)	Turret positions-2	Roller steady box

3. Form the right end of the bolt.	Turret positions-3	Roller steady bar ending tool
4.Make the external thread cutting of 20mm diameter to a length of 40mm (from the right end)	Turret positions-4	Self opening die head with chases of 20mm
5.Chamfering the bolt head.	Front cross slid tool post-1	Chamfering tool
6.Parting off the work piece	Rear tool post.	Parting tool is place in inverted position (making the rotation of work anticlockwise with respective to tool)

(i) Setting the bar stop:

The bar stop (1) is set at the distance of 100mm from the collet face by using slip gauge. An extra length of 10mm is allowed for parting off (4mm) and clearance of the collet face (6mm). This clearance is allowed to penetrate the parting tool deep into the work piece without any Inference.

(ii) Setting of the roller steady box turning rod:

This tool is set on turret face of 2. This tool (2) is used for turning the work piece to 200mm diameter and 80mm long from the right end.

(iii) Setting of bar ending tool:

This tool is set (3) on fourth turret face but turret position-3. This is used to chamfer the right end of the work piece.

(iv) Setting self opening die head:

This tool (4) is set on the fifth face of the turret. The proper blades of chasers are selected and fitted into the die head to cut a thread of 20mm diameter.

(iv) Setting of Chamfering tool:

This tool (5) is set on the cross-slide front-end position-I used to chamfer the bolt head edges by giving cross feed.

(vi) Setting of parting tool:

This tool is set on the rear end of cross-slide. It is used to part off. The work piece after completing all operations.

Tool layout for machining a product in semi-automatic and automatic lathes.

The procedural steps to be followed in sequence for batch or lot production of a job by machining in semi-automatic and automatic general purpose machine tools are

(a) Thorough study of the job to be produced: in respect of :

• Volume of production, i.e., number of pieces of the specific job to be produced.

• Material and its properties

• Size and shape

• Surfaces to be machined

• Required dimensions with tolerances and surface finish

• End use of the product.

(b) Selection of machine tool (after studying the job) in respect of:

• Type

• Size

• Precision

• Kind and degree of automation

(c) Selection of blank (based on job and machine selected) in respect of:

• Bar chucking or housing type

• Preformed by; casting, forging, rolling etc.

• If bar type; cross section (circular, tubular, square, hexagon etc.)

• Nominal size based on largest dimensions and availability

• Preformed by hot working or cold working

(d) Identification and listing of the elementary machining operations required, depending upon the product configuration

(e) Combine elementary machining operations as much as possible for saving time.

(f) Sequence the operations (after combining)

(g) Select cutting tools in respect of:

• Type

• Material

• Size

• Geometry

• Availability depending upon the machining operations (after combining) and work material

(h) Work scheduling or preparation of the instruction sheet or operation chart giving column-wise :

• Description of the machining work to be done in sequence

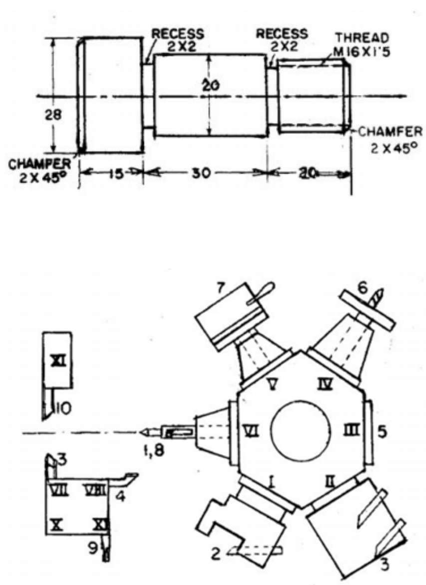
• Cutting tools : type and location

• Speed and feed for each operation

• Length of travel of the tools

Tool layout :

Schematically showing the type and configuration of a typical tool layout for a particular job being machined in a single spindle automatic lathe is schematically shown in the figure.



Tool layout for a typical job in single automatic lathe