COMPUTERISED MACHINING OF COMPONENTS USING A CENTRE (ENGINE) LATHE MACHINE.

BY

MUHAMMAD SULAIMAN LIMAN

PGD/MCS/147

A PROJECT SUBMITTED TO THE DEPARTMENT OF MATHEMATICS AND COMPUTER SCIENCE, FEDERAL UNIVERSITY OF TECHNOLOGY, MINNA IN PARTIAL FULFILMENT OF THE REQUIREMENTS FOR THE AWARD OF POST GRADUATE DIPLOMA IN COMPUTER SCIENCE.

FEDERAL UNIVERSITY OF TECHNOLOGY, MINNA,

NIGER STATE, NIGERIA.

DECEMBER, '95.

DECLARATION

I HEREBY DECLARE THAT THIS RESEARCH PROJECT HAS BEEN CONDUCTED SOLELY BY ME UNDER THE GUIDANCE OF DR. S A REJU, AND I HAVE NEITHER COPIED SOMEONE'S WORK NOR HAS SOMEONE ELSE DONE IT FOR ME. WRITER'S WHOSE WORKS HAVE BEEN REFERED TO IN THE PROJECT HAVE BEEN ACKNOWLEDGED.

MUHAMMAD SULAIMAN LIMAN

SIGNATURE

28/02/97

DATE

CERTIFICATION

THIS PROJECT IS CERTIFIED TO BE CARRIED OUT BY MUHAMMAD SULAIMAN LIMAN (PGD / MCS / 147/93/94) . THE PROJECT MEETS THE REGULATIONS GOVERNING THE AWARD OF POST- GRADUATE DIPLOMA IN COMPUTER SCIENCE IN THE DEPARTMENT OF MATHEMATICS AND COMPUTER SCIENCE, FEDERAL UNIVERSITY OF TECHNOLOGY, MINNA, NIGER STATE.

____

DR. S. A. REJU
(PROJECT SUPERVISOR)

____

DR. K. R. ADEBOYE
(HEAD OF DEPARTMENT)

____

EXTERNAL EXAMINER

DEDICATION

IN THE NAME OF ALLAH, THE MOST BENEFICENT, THE MOST MERCIFUL.

THIS PROJECT IS SINCERELY DEDICATED WITH ALL RELEVANCE TO THE ALMIGHTY ALLAH, MAY HIS BLESSINGS BE ON US ALWAYS "AMEEN".

AND ALSO TO THE FOLLOWING:

MY FATHER:	ALH. SULAIMAN LIMAN AGAIE
MY MOTHER:	HAJIYA ZAINAB SULAIMAN LIMAN
MY LATE STEP-MOTHER:	HAJIYA HAUWAWU SULAIMAN
MY STEP-MOTHER:	HAJIYA AISHETU SULAIMAN

ACKNOWLEDGEMENT

I remain grateful to Almighty ALLAH for giving me the health, wisdom and courage to be able to pass through this course and carryout the project work successfully. May ALLAH continue to guide and protect us through our endeavours "AMEEN". I never forget my supervisor DR. S. A. Reju, infact I must specifically mention here that he has been so tolerant and also took his time not only to read, but made contributions and guidance. May ALLAH continue to crown his efforts with success "AMEEN". I wish to extend my gratitude to the following people for their individual contribution in one way or the other. Alh. Hassan Lemu (matawalin nupe), Alh. Abdullahi Nkochi, Sqn. Ldr. Onyeukwu, Alh. Yahaya Agaie, Alh. Abdullahi Alfa Bayiye, Alh. Mustapha Mahmud, Alh Abdulkadir Marafa, Mal. B. Hassan and Sqn. Ldr. Haruna Moh'd.. I shall also want to thank the entire members of NAF Rockets F.C. and to my classmates PGD/MCS/93/94, I would want to recognize the Cordial relationship that existed between us as a very memorable one.

Finally, my gratitude goes to the families of Alh. Hassan Lemu and Alh. Sulaiman Liman Agaie for all the support given to me during the duration of the course. I would want to thank Sqn Ldr. UH Bozegha, FS. Hussein Sulaiman and Miss Okon Helen for having much time to key in the project work.

Lastly, there are so many others whom space would not permit me to mention, specifically my close associates. I appreciate all you have done for me, thank you and May ALLAH reward you abundantly "AMEEN". And also may ALLAH continue to show us the right path and guide us through always "Ameen".

LIST OF ILLUSTRATIONS

CHAPTER ONE : BASIC THEORY OF LATHE MACHINES.

1.1 INTRODUCTION

The making or manufacturing of components of all kinds is the business of engineers. If these components are to be produced cheaply it is important that engineers choose the best possible methods, and at the same time produce the components as accurate as possible and as strong as possible.

Mechanical engineering is the broadest of all the engineering disciplines and plays some part in the production of virtually everything with which we come into contact in our daily lives. It is mechanical engineers who design, develop and build vehicles, engines, turbines and all manner of machines, infact anything that involves moving parts. They produce the equipment needed for extracting, processing and refining basic materials into useful metals and plastic and design and make the machine tools needed to shape them. Infact it could be said that mechanical engineers are perhaps the most important members of the community, for it is on them that practically everything depends in this highly mechanised world.

Over the last 40 years, we have moved steadily from an economy marked by an abundance of demand to a market dominated by supply. During what are sometimes called the '30 glorious years' (1945 - 1975), the manufacturing world organized itself to respond to growth with large scale production of a single product or similar products. Today, it has to satisfy a demand that has become increasingly complex and diverse : small scale production of many models with frequent and rapid changes. In such a context, reliability and flexibility becomes the key words in the production process.

Since the first use of machine tools there has been a gradual. but steady trend toward making machines more efficient by combining operations and by transferring more skill to the machine, thus reducing time and labour. To meet these needs, machine tools have become complex in design and in control. Automatic features have been built into many machines and some are completely automatic. This technical development has made it possible for industry to attain a high production rate with the accompanying low labour cost that is an essential development for any society wishing to enjoy high living standards.

Before the advent of computers into production processes, operating the lathe machines by human beings (hand) requires skill, and the talents of a highly skilled operator are poorly utilized in performing repetitive or variety of machining operations and also the operators job will be in fragments. This is due to the fact that the operator will have to reset the machine after one particular operation is carried out for another one to be undertaken. This brings about boredom or fatigue and it does have adverse effect on production output, inaccurate operations and there will be a lot of errors.

With the incorporation of computers into the production processes, on the Lathe machine is incorporated a computer interface with which the operator interacts in a conversational mode. For repetitive productions, it is done by giving to operator the responsibility for the first part of a new batch. The sub-sequent parts, and subsequent batches, can be made from the program generated for the first part.

And as regards variety of machining operations, the operator is asked, for example whether the next operation is turning, boring, facing or drilling, and presses the appropriate button against a dynamic menu. If he selects turning, he is asked for the initial and final diameters and the start and finish of the cut. Tapers, chamfers and radii can be easily defined. Speeds and feeds and cuts are optimised to make best use the available power. All of this is done in terms familiar to the operator and makes use of his existing skill. Once all the information for the next operation has been entered and checked, the operation can be performed with ease and accuracy.

This project 'computerised machining of components using a Lathe machine' signifies the importance of using computer on a lathe machine for repetitive or variety of machining operations compared to human beings (hand) operators. This system has technological and economic advantages in increasing the machine utilization and the speed of programming. It is a system which accepts and uses the existing skills of the machinists, and leaves room for it to develop. It does not fragments the operators job. The aim is not simply to preserve the operator's existing skills. It is rather to make those skills more productive and to allow it to develop in a natural way into a new skill - the skill of using a more highly develop system.

1.2 DEFINITIONS

1.2.1 MACHINE - A machine is a piece of equipment which does a particular type of work and usually used power from an engine or electricity, or a machine is anything which obeys casual laws. If men and women obeys casual laws, then they may be directly replaced by machines obeying the same or more desirable laws. They have no advantage over machines, which may be faster, more reliable, and not subject to boredom or fatigue.

1.2.2 MACHINE TOOL - It is defined as a power operated device designed to produce machined components having Linear and Angular dimensions within the limits laid down, such surfaces to be machined in the minimum of time and have an acceptable finish.

1.2.3 LATHE - A Lathe is a device in which the work is rotated against a cutting tool. As the cutting tool is moved length wise and crosswise to the axis of the workpiece, the shape of the work-piece is generated.

1.2.4 MACHINING - It is one of the four major metal - manufacturing methods. The others are hot forming, cold working and casting. It is a generic term, applied to all metal removal, while metal-cutting refers to processes in which the excess workpiece material is removed by a harder tool, through a controlled fracture process. Industry always uses the machining method when a very accurate and smooth surface is needed. In machining, metal is turned, shaped, milled, cut or otherwise reduced or changed by removing chips with machine tools to produce the shape and dimension wanted.

1.3 LATHE MACHINE

The most widely used machine tool is the centre (Engine) Lathe (fig 1.1) which provides a rotary primary motion while the appropriate feed motions are

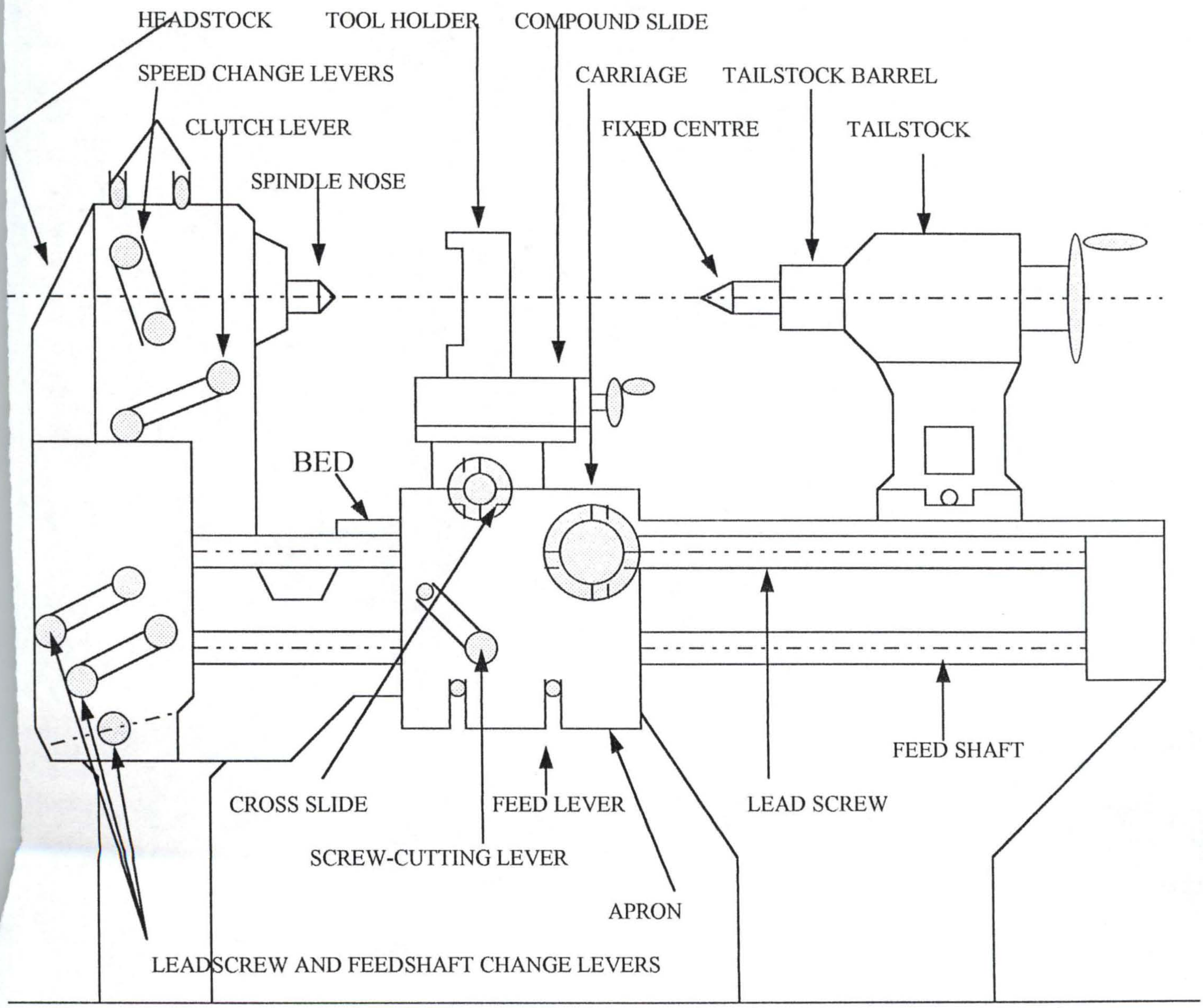


FIGURE 1.1 A TYPICAL LATHE MACHINE

imparted to the tool. It is considered to be the father of all machines because it was one of the earlier machines to be invented and also can be employ to do variety of operations that are carried out on other machines. It is one of the most useful and versatile machines employed in metal working . Modern Lathe are highly accurate and complex machines capable of performing a great variety of operations and a machinist depends on the Lathe to make precision parts.

The size of a lathe is determined by the swing and the length of the bed. Swing is the largest diameter that can be turned. It is twice the distance from the centre to the nearest interference on the bed of the Lathe. The bed length determines the distance between centres, or the longest piece that can be turned. Whichever way the size is expressed it must be remembered that the work has to clear the Lathe carriage when turning between centres.

The Lathe consists of the following major component groups:

1.3.1 BED - The bed is the foundation and backbone of a lathe. It's rigidity and alignment affect the accuracy of the parts machined on it.

On top of the bed are the ways, which usually consist of two inverted vees and two flat bearing surfaces made to support the carriage and tailstock. The ways of the Lathe are very accurately machined by grinding or by milling and hand scraping. Wear or damage to the ways will affect the accuracy of workpieces machined on them.

A gear rack is fastened below the front way of the Lathe. Gears that link the carriage handwheel to this rack make possible the lengthwise movement of the carriage by hand.

1.3.2 HEADSTOCK - The headstock is fastened to the left end of the bed. it consists of the headstock spindle and the mechanism for driving it.

The spindle is hollow to allow long slender workpieces to pass through, with a tapered hole at the front, or inner end. The spindle end facing the tailstock is called the Spindle Nose. Lathe chuck and other work holding devices are fastened and driven by the spindle nose. The spindle is supported by spindle bearings on each end. If they are sleeve - type bearings, a thrust bearing is also used to take up end play. Tapered roller spindle bearings are often used on modern lathes.

Spindle noses usually are of three designs : a long taper key drive, a cam lock type or a threaded spindle nose. The power for turning is provided by an electric motor. Spindle speed changes are also made in the headstock, either with belts or with gears. Speed changes are made in these lathes by shifting gears in much the same way as in a standard automobile transmission. To obtain more torque, or turning force, and slower speeds, the back gear is used. On belt-driven lathes, direct drive power is delivered through belts to a step pulley that turns the spindle. The speed is changed by moving the belts to different positions.

The hole in the spindle nose typically has a standard morse taper. The size of this taper varies with the size of the lathe. A sleeve fits into this tapered hole and then into the live, or headstock, centre.

A feed reverse lever, also called a leadscrew direction control, is located on the headstock. It's function is simply to control the direction of rotation of the leadscrew. This rotation determines the direction of feed and whether a thread cut on the lathe is left hand or right hand. The threading and feeding mechanism of the lathe are also powered through the headstock.

1.3.3 CARRIAGE - The distance from the top of the tool holder to the bed is known as the carriage, and it comprises of five parts.

a. The saddle is an h - shaped casting that fits over the bed, slides on the ways and supports the Cross slide and compound slide.

b. APRON - The apron is the part of the carriage facing the operator; it extends down the front of the machine to house the various feed gears and controls. these controls include a handwheel to move the carriage back and forth along the ways, levers to engage the longitudinal and cross feeds, a half-nut lever to start, stop and reverse the carriage travel. The handwheel is attached to a pinion that meshes with a rack under the front of the bed.

A thread dial is fastened to the apron (usually on the right side), which indicates the exact place to engage the half-nuts while cutting threads. The half-nut lever is used only for thread cutting and never for feeds for general turning.

The entire carriage can be moved along the ways manually by turning the carriage handwheel or under power by engaging the power feed controls on the apron. Once in position, the carriage can be clamped to the bed by tightening the carriage lock screw.

On some lathes a feed change lever (or plunger) or the apron is used to direct power from the feed mechanism to either the longitudinal (length wise) travel of the carriage or to the cross slide. On other lathes, two separate levers or knobs are used to transmit motion to the carriage and cross slide.

c. The cross slide is mounted on the saddle. It moves cross wise at 90" degrees to the axis of the lathe by manually turning the cross feed screw handle or by the cross feed lever also called power feed lever or on some lathes, the clutch knob, which is located on the apron for automatic feeds.

Hand wheels or ball cranks are provided for manual feeding of the cross-slide and compound slide. Graduated collars are mounted on the various handwheels for measuring the amount of feed.

d. The compound rest or slide is mounted on the cross slide and can be swiveled to any angle horizontal with the lathe axis in order to produce bevels and tapers. The compound rest can only moved manually by turning the compound rest feed screw handle.

Cutting tools are fastened on a toolpost that is located on the compound rest. It also has a slide in which the upper part of the casting can be moved in and out with the compound rest handle.

e. The tool post with the ring collar and rocker base slides in a T - slot on top of the compound rest. Sometimes the tool post sits on a compound tool rest which incorporates a slide that can be set at any angle; thus conical surfaces may be formed by hand feeding the tool. A four - way tool post allows quick changing of tools in transmiting power between the spindle and the carriage. By using the gear shift levers on the quick - change gear box, you can select different feeds. Power from the left end of the spindle is feed through the gears to this gearbox. The gearbox makes it possible to change the fed and the ratio between the revolutions of the headstock spindle and the movement of the carriage for thread cutting. There are usually two or three levers on the gearbox for controlling the feed and number of threads. An index chart, or plate, fastened to the gearbox tells you how to move the levers.

The leadscrew and feed rod transmit the power to the carriage for operating the feed and for thread cutting.

To get power for longitudinal, or backs-and-forth, feeding, the feed-change lever on the carriage is moved to the down (or up) position. Then the clutch handle, either a lever or a knob, is turned or moved.

To get power for cross feeding , put the feed - change lever in the opposite position. For thread cutting, the feed - change lever is put in the centre, or neutral, position to operate the half, or split nut lever. This unit closes over the threads of the leadscrew to move the carriage.

The base of the machine is used to level the lathe and to secure it to the floor. The motor of the lathe is usually mounted in the base.

1.4 WORKHOLDING DEVICES

Work must be efficiently held in lathe machine for safety operations. Work - holding and driving devices are fastened to the spindle nose and are very important to machining on lathes.

The methods used depends upon the workpiece, the machine, and the extent to which rapid production machines, such as turret lathes, the holding devices are usually actuated hydraulically, by air, electricity, or cam action in order that the clamping time and effort can be minimized.

In case of automated or numerically controlled machines, the holding devices may be programmed to release the workpiece at the conclusion of machining and to automatically clamp the next part fed to it. There are a number of workholding methods that have been devised to suit various types of jobs and to enable the accuracy requirements to be met because performing as many operations as possible at one setting, that is, without removing the work from it's setting in the machine is not always practicable.

The most commonly used work holding devices are :

1.4.1 CHUCKS

Chucks are used for holding large and irregular shaped parts and are either bolted or screwed to the spindle, making a rigid mounting.

Chucks are made in several designs and may be classified as

a. UNIVERSAL CHUCKS Universal chucks usually have 3 - jaws. All jaws are moved in or out equally in their slides by means of a scroll plate located at the back of the jaws. The scroll plate has a bevel gear on it's reverse side that is driven by a pinion gear. This gear extends to the outside of the chuck and is turned with the chuck key or wrench. The three jaw chuck or self - centering (fig 1.2) is undoubtedly the most convenient, and therefore most widely used. It is best suitable for gripping of round bars of accurate size and smooth finish. The chuck consists of a cylindrical body, across the face of which the three jaws move in equally spaced radical slots, driven by a scroll plate inside the body.

The chuck body is mounted on the back plate, which may be bored and threaded internally to screw on the nose of the lathe. A better form of attachment has the back plate recessed to engage a register on the spindle nose, and positive holding is then provided by four bolts. This type cannot come lose if the machine stops suddenly.

This chuck operates on a principle of holding and gripping of a cylindrical work piece so that it rotates about it's own axis. The three jaw chuck moves in unison towards the chuck key. the back plate of the scroll plate forms a large bevel gear which can be driven by any of three pinions which are let into the rim of the chuck. Each pinion has a square socket at its outer end into which the chuck key fits. Rotation of any pinion thus turns the scroll plate and moves the jaws.

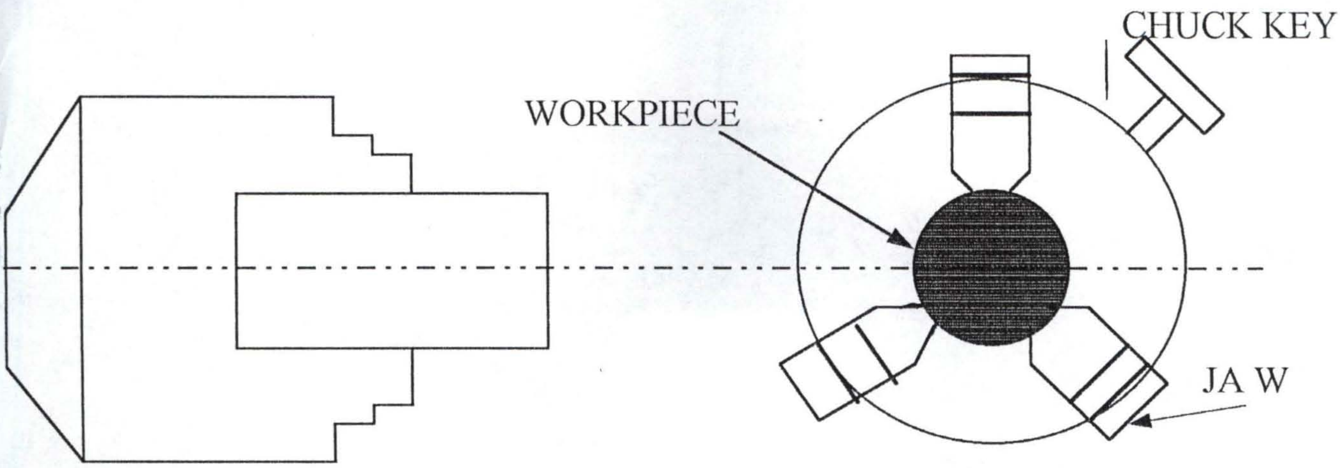


FIGURE 1.2 - 3 - JAW CHUCK OR SELF CENTERING CHUCK

The 2 - jaw chuck (fig 1.3) is used for holding casting and forging. All jaws maintain a concentric relationship when the chuck wrench is turned.

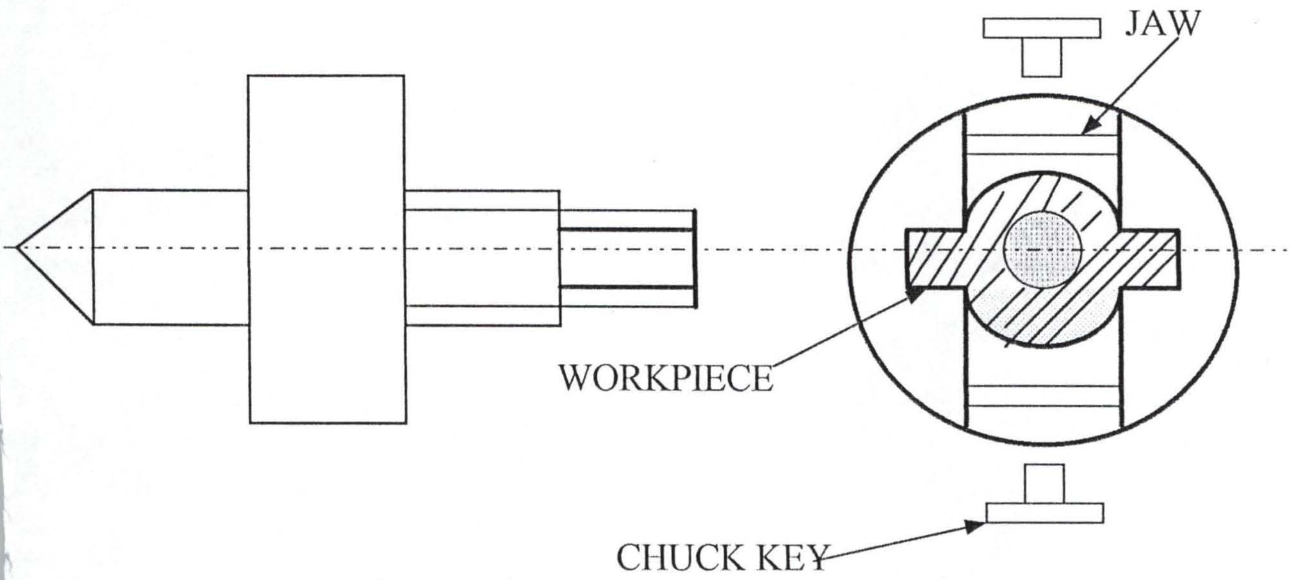


FIGURE 1.3 2 - JAW CHUCK

universal chucks provide quick and simple chucking and centering of round stock. Uneven or even irregularly shaped materials will damage these jaws. The jaws of universal chuck will not reverse as with independent chucks, so a separate set of reverse jaws are used to hold pieces with larger diameters. The chuck and each of its jaws are stamped with identification numbers, as such the jaws must be assembled in sequence in their individual slots or they will not centralise correctly.

One disadvantage of most universal chucks is that they loose their accuracy when the scroll and jaws wear, and normally there is no compensation for wear other than regrinding the jaws.

b. INDEPENDENT CHUCK - The independent 4 - jaw chuck (fig 1.4) is use for holding large and irregular components. Settings are not automatic and skill is required. It has four jaws each working independently of the others in it's own slot in the chuck body and actuated by it's own separate square - threaded screw. This makes for slower but more accuarate centering, and with a little care work can be made to run absolutely true.

It is a general requirements of cylindrical work that all diameter should be concentric, and all faced surfaces should be square to the cylinder axis. This is what we mean when we talk of work running true.

It provides the facility of deliberately setting work off centre to produce an eccentric work piece. The concentric diameters are turned first and the chuck is then adjusted for eccentricity by a dial gauge which for accuaracy must register at radial position. Note that the dial gauge readings must vary by twice the amount of eccentricity required. Very precise set ups can also be made with the 4 - jaw chuck by using a dial indicator, especially on round materials. Some types are fitted with top jaws that can be reversed after removal of bolts on the jaw. Jaws in the reversed position can grip larger diameter workpieces. The independent chuck will hold work more securely for heavy cutting than the 3 - jaw chuck.

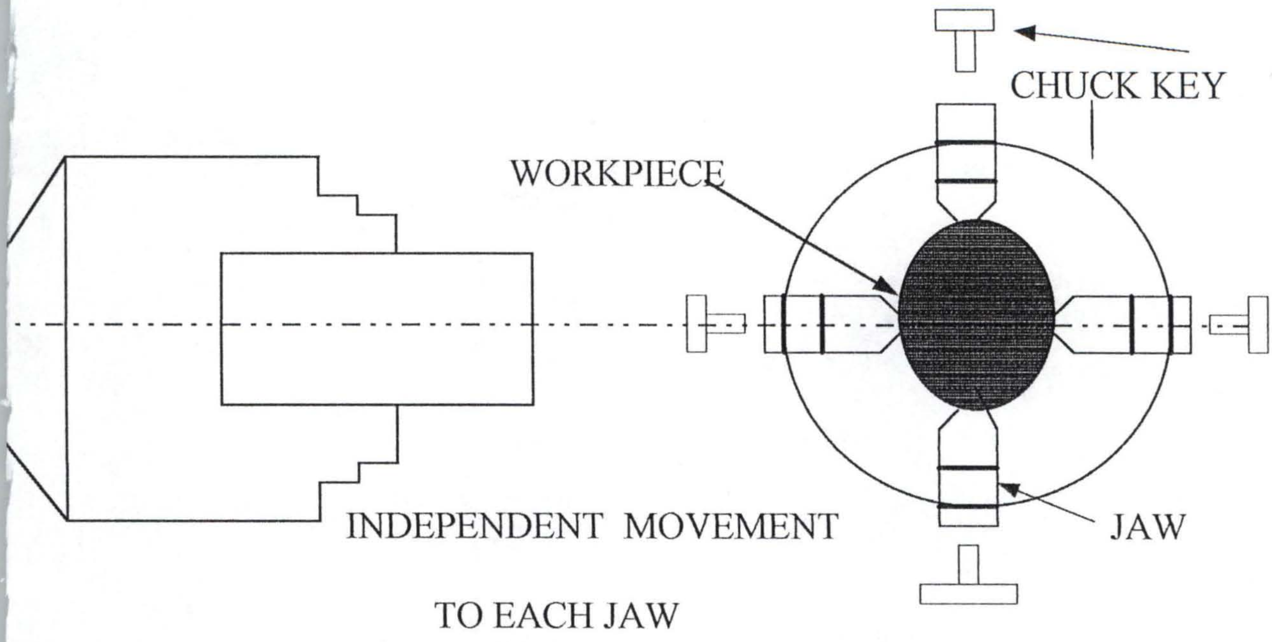


FIGURE 1.4 - 4 - JAW INDEPENDENT CHUCK

With both types of chucks, jaws are available with hard faces for gripping rough work, or soft faces for finished work.

c. **COMBINATION CHUCK** - Each jaw has an independent adjustment and, in addition, has a separate wrench connection that controls all jaws simultaneously.

1.4.2 COLLET CHUCK

These are tabular units made from medium carbon steel having slots or splits running about three - quarters of the length, hardened and spring tempered so that when unlocked it will spring open and release the work piece. Collet chucks (fig 1.5) are very accurate work - holding devices and are used in producing small precision part.They are commonly used for bar - stock materials or holding bars of standard sizes to accommodate round, square and hexagonal stock.

Each collet is made for a single diameter and a set of collets is provided for standard diameters of work. The outside shape is made to suit the particular machine tool builders specification. One end has a larger section with a conical surface which, when drawn into a mating taper surface, closes the collet on the work..

To enable it to close on the workpiece three or four narrow slots are cut from the body to the gripping end. The collet chuck which is tightened through being drawn into a tapered socket in the spindle nose by a draw tube which bears against the back end of the spindle. The outer taper of the gripping end matches the taper bore of the machine spindle nose and as the collet is drawn into the spindle it's gripping end closed on the workpiece. The draw tube and the back end of the collets are hollow to permit bars to be fed through the spindle for repetition turning and pating-off of workpieces.

For large stock, collets of the parallel closing type are sometimes used, but in most cases collets of spring type are recommended. The spring collets are solid at one end and split on the other end, which is tapered. Spring collets are made in three types, namely :- the push-out, the draw-back and the stationary.

Push-out collets are recommended for bar work, since the slight movement of the stock pushes it against the bar stop.

Draw-back collets are not widely employed for bar stock, but are useful when the collets are of extra capacity size and one utilized for holding short pieces.

With the both push and draw collets, there is a slight movement of the workpiece because the collet moves as it is tightened. In most cases this is not a disadvantage, but when it is, the stationary collet can be used. A shoulder on the collet comes to rest against the head to provide this endwise accuracy. Because there are more sliding surfaces in the stationary collet, the rotational concentricity is not as accurate as for other types of collets.

Some types, called master collets, have removal pads or jaws for holding various sizes and shapes of collet is made up of alternate steel and rubber segments bonded together. It is compressed to grip the work by turning a hand wheel on the spindle nose. This type has a wider gripping range than the conventional collets. There are a number of special collets, including those employing hydraulic force on a flexible steel sleeve in which the work piece is held. Such collets give a high torque connection with close accuracy.

Rough and inaccurate workpieces should not be held in collet chuck. Since the gripping surfaces of the chuck would form an angle with the workpiece.

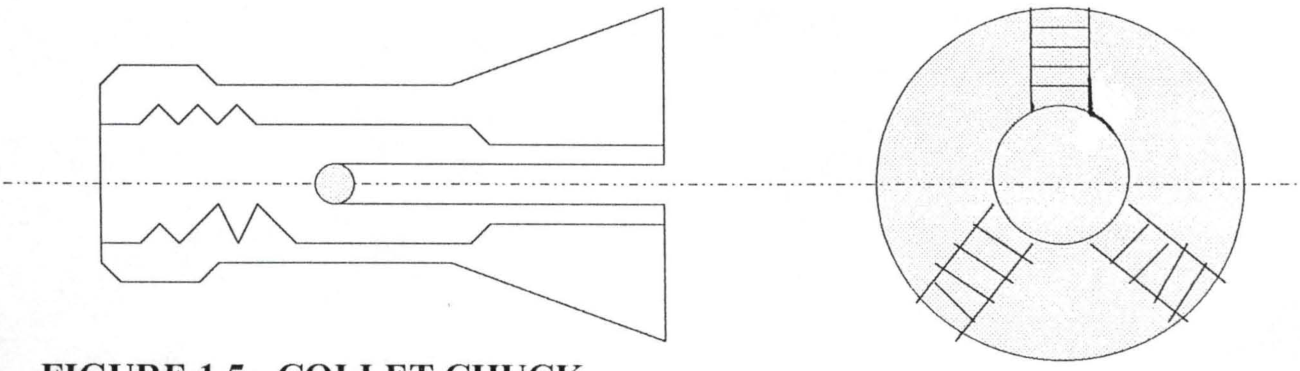


FIGURE 1.5 - COLLET CHUCK

1.4.3 DRILL CHUCK

A small universal screw chuck used principally on drill presses but frequently used lathes for drilling and centering.

1.4.4 POWER CHUCK

Operated by air, hydraulic means, or electricity, relieves the operator of the effect involved in tightening and loosening the work. Additional advantages of the power chuck are that it is quick acting; the chucking pressure can be regulated; and it's can be used for both bar and chucking work.

1.4.5 MAGNETIC CHUCK

Work can be held on surface grinders and other machine tools by means of magnetic chucks, this method of holding being both simple and rapid. Magnetic chucks are sometimes used for making light cuts on ferromagnetic material. They are useful for facing thin material that would be difficult to hold in conventional work holding devices. Parts to be held are placed on the chuck, which is energized by the turning of a switch. The two types of chucks used are permanent chuck and those magnetized by means of direct current. The direct-current chucks, made in both rectangular and circular shapes; have a pulling power of around 1mpa. The rectangular style is suitable for use on reciprocating grinders or for light milling machine work. Rotary chucks, designed for lathes and rotating table grinders.

Permanent magnetic chuck do not require any electrical equipment, and work can be held on these chucks without damage to work or chuck. The operation of this chuck is by a lever on one side. In the "off" position, the conductor bars and separators are shifted in such a way that magnetic flux passes through the top plate and isshort circuit from the work. when the handle is turned to" ON" the conductor bars and non magnetic seperators line up so that magnetic flux, in following the line of least resistance, goes through the work in completing the circuit. the holding power, obtained by the magnetic flux passes through, is sufficient to with stand the action of grinding wheels and other light machining operations.

All parts held on a magnetic chuck should be de-magnetized after the work is finished. Several types of de-magnetizers are available, operating on either alternating or direct current, which successfully removes the residual magnetism from knives, bearing races, blades, and many other parts. The permanent magnetic chuck and the direct current chuck may both be used for either wet or dry operations.

Chucks come in all diameters and are made for light, medium and heavy-duty uses. All chucks need frequent clearing of scrolls and jaws. These should be lightly oiled after clearing and chucks with grease fittings pressure lubricated.

1.4.6 ARBOR

Expanding or threaded arbors are used to hold short pieces of stock that have in them a previously machined and accurate hole. The action in holding the work is controlled by a mechanism very similar to that used with collets. The work is placed on the arbor against the stop plate and, as the draw rod is pulled, the tapered pin expands the partially split plug and grips the work.

The threaded arbor operates in a similar fashion except that the work is screwed on the arbor by hand until it is forced back against a stop tube or flange.

Both collets and arbors may be power operated by pneumatic, hydraulic, or electrical devices located at the end of the spindle. Such an arrangement is frequently used on high production work to provide a quicker and easier operation.

1.4.7 MANDRELS

Work which has a machined hole running through it's full length is frequently held on a mandrel. Mandrels (fig 1.6), in it's simplest form, is a hardened and ground shaft, with centered ends, and is slightly tapered, usually 0.006 in per ft.

The smaller end is placed in a hole, and pressed into work until it binds. The assembly is then held between centres when mounting work, be sure to press the mandrel in square with the work, and oil the surface so the parts can be disassembled without difficulty. When driving the mandrel out, be sure to drive it in the opposite direction from that in which it entered. If the diameter of the work is large in relation to the hole, fasten a pin or driver to the face plate so as to contact a lug or boss on the work to take part of the load, and prevent the work from slipping on the mandrel.

Because solid mandrels must be made to exact sizes and are subject to considerable wear, various types of expanding mandrels have been developed. These have a wider range and adapt themselves to take up wear. Other types include mandrels which grip the work between a shoulder and a washer by tightening a nut, and threaded mandrels on which work can be screwed up to a shoulder.

A great variety of special adapters can also be made for attaching the work to the spindle nose.

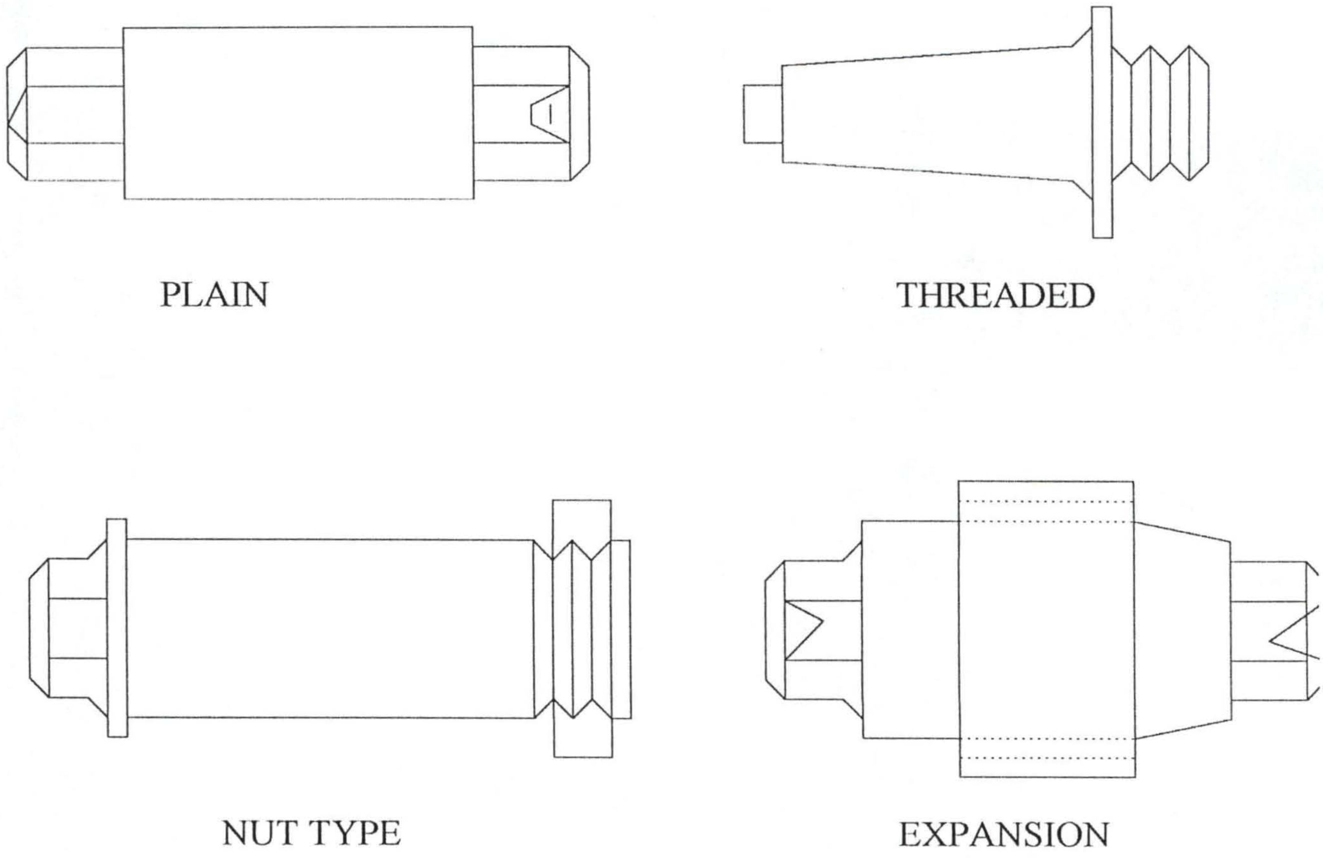


FIGURE 1.6 - COMMON TYPES OF MANDRELS

1.4.8 WORK HELD BETWEEN CENTRES

The most common way to support work, in the case of a rotating workpiece, is to mount it between centres (fig 1.7). If a workpiece is solid, that is not bored and can have a centre hole in each end, it can be supported between a running centre in the headstock spindle and a fixed centre in the tailstock or barrel. Convenient method for locating centre holes is with a combination square and scriber, followed by centre punching and drilling. This method has the advantage of being able to resist heavy cuts and is convenient for long parts. Headstock centre turns with the work; as such does not necessarily needs lubricantion and must be kept clean and free from damage. The tailstock centre or dead centre is stationary and does not affect the concentricity, but it must be hard to resist wear and kept clean and also lubricated to reduce friction because it acts as a conical bearing or else must be of the ball bearing type.

Care should be taken to provide for exp[ansion of the workpiece due to the heat generated by machine operations and rotation of the stock.

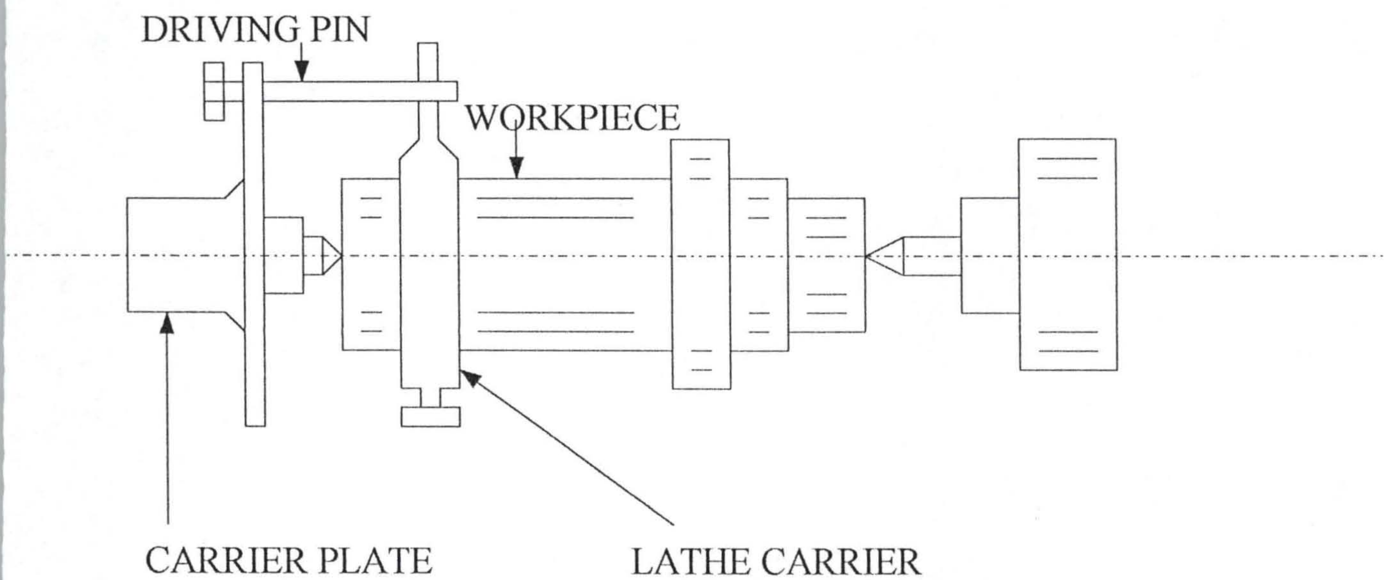


FIGURE 1.7 SHOWS WORK HELD BETWEEN CENTRES

1.4.9 FACE PLATE

The face plate (fig 1.8) is simply a circular worktable, mounted on the spindle nose. Large or awkwardly shaped workpiece may be held to the face by means of clamps, bolts, or else in a fixture or special holding device attached to it. Such mounting is suitable for flat plates and parts of irregular shape.

Face plate have T-slots and are more heavily built. Face plates are made of cast iron and so must be operated at relatively slow speeds. If the speed is too high, the face plate could fly apart. The method of setting the work on the face depends largely on the accuracy required.

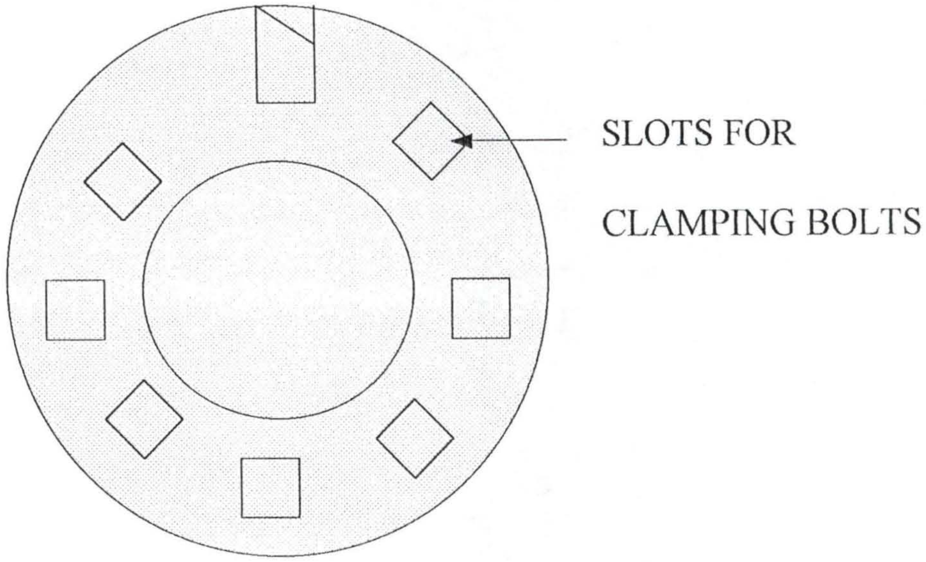


FIGURE 1.8 - FACE PLATE

1.4.10 EXTERNAL WORK SUPPORT

Apart from gripping and driving the work in a chuck, it is also necessary to ensure that the work is not deflected during cutting through cutting forces. Work with a large overhang is usually supported by the tailstock centre but if it is long and slender it requires additional support close to the tool.

Travelling steady (fig 1.9) is bolted to the cross slide on the saddle and travels along with tool as it feeds along the work. It is normally used for long - slender workpiece to stop them from vibration at high speed and also to prevent them from bending.

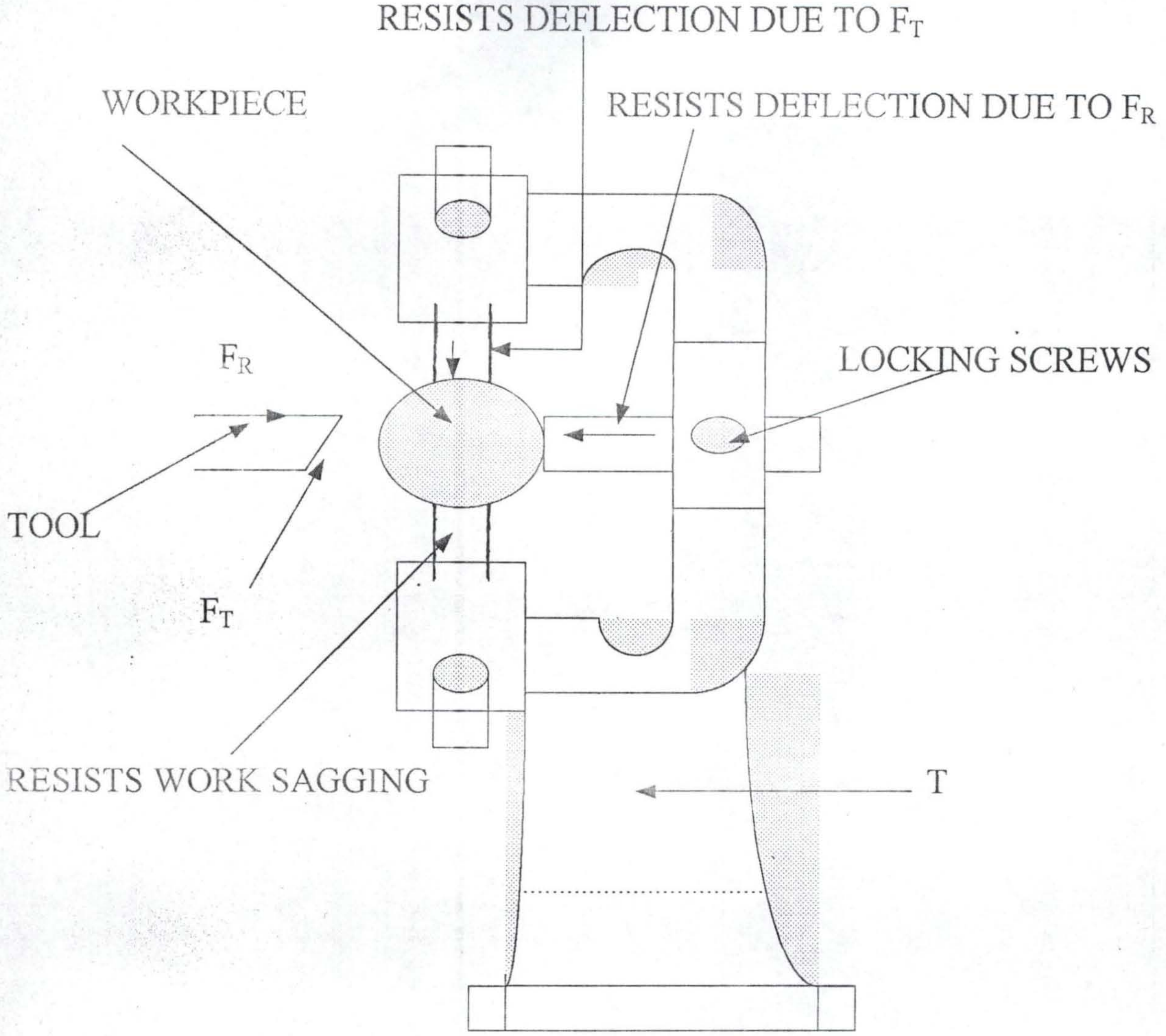


FIGURE 1.9 - TRAVELLING STEADY

The fixed steady (fig 1.10) is mounted on the lathe bed and does not travel, it is used for slender workpiece.

Note that the steady rests are positioned to withstand the two forces which would bend the work, that is the tangential cutting force F_I and the radial force T_R set up by the cutting action.

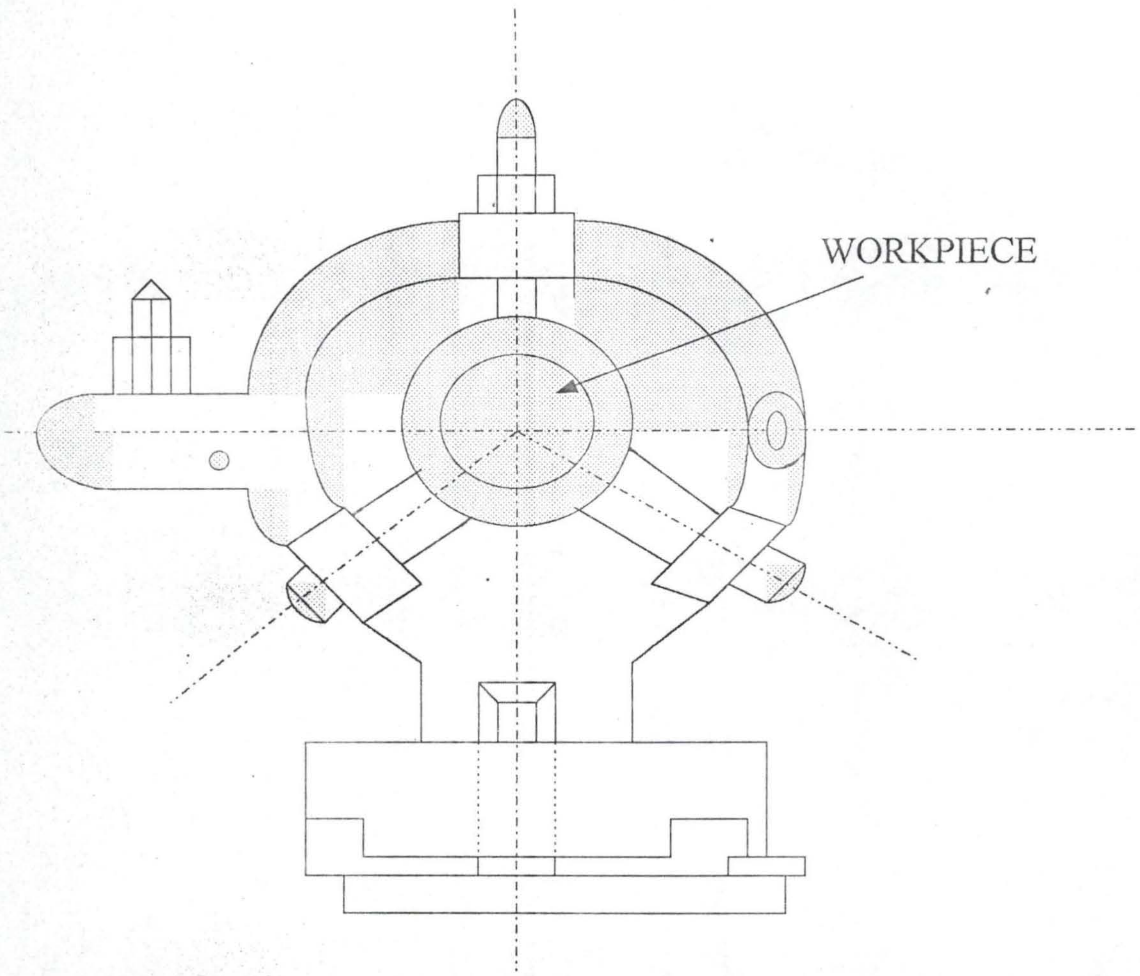


FIGURE 1.10 - FIXED STEADY

1.5 TYPE OF CUTTING TOOL The usefulness or efficiency of an expensive machine tool is measured in terms of the amount of metal removed, it is clear that the cutting tool usually plays a very important part in machining process.

On a lathe, metal is removed from a workpiece by turning it against a single - point cutting tool. In it's simplest form a single-point tool is usually reffered to as a standard straight-edge cutting tool, it is the basic lathe tool, and can be considered as a logical extension of the wedge - point principle underlying the removal of metal with hand tools. It is obvious that one of the most versatile and the easiest to visualize is a single-point cutting tool moved in a programmed fashion, and can be used for the following metal removing operations.

(a) Turning external cylindrical surface at a centre lathe.

(b) Boring internal cylindrical surface at a centre lathe.

(c) Facing plane surface at a centre lathe.

(d) Shaping plane surface at a shaping machine.

The work is seen to rotate while the cutting tool is stationary on the lathe.

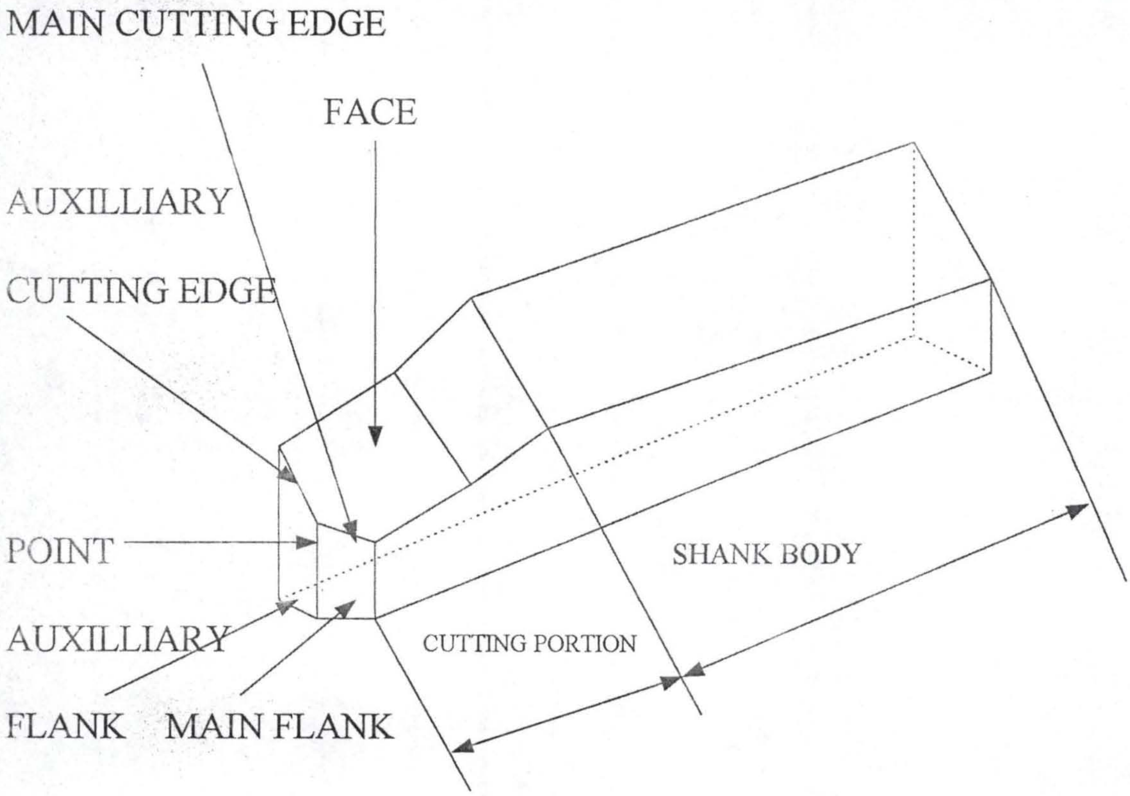


FIGURE 1.11 - TOOL ELEMENTS

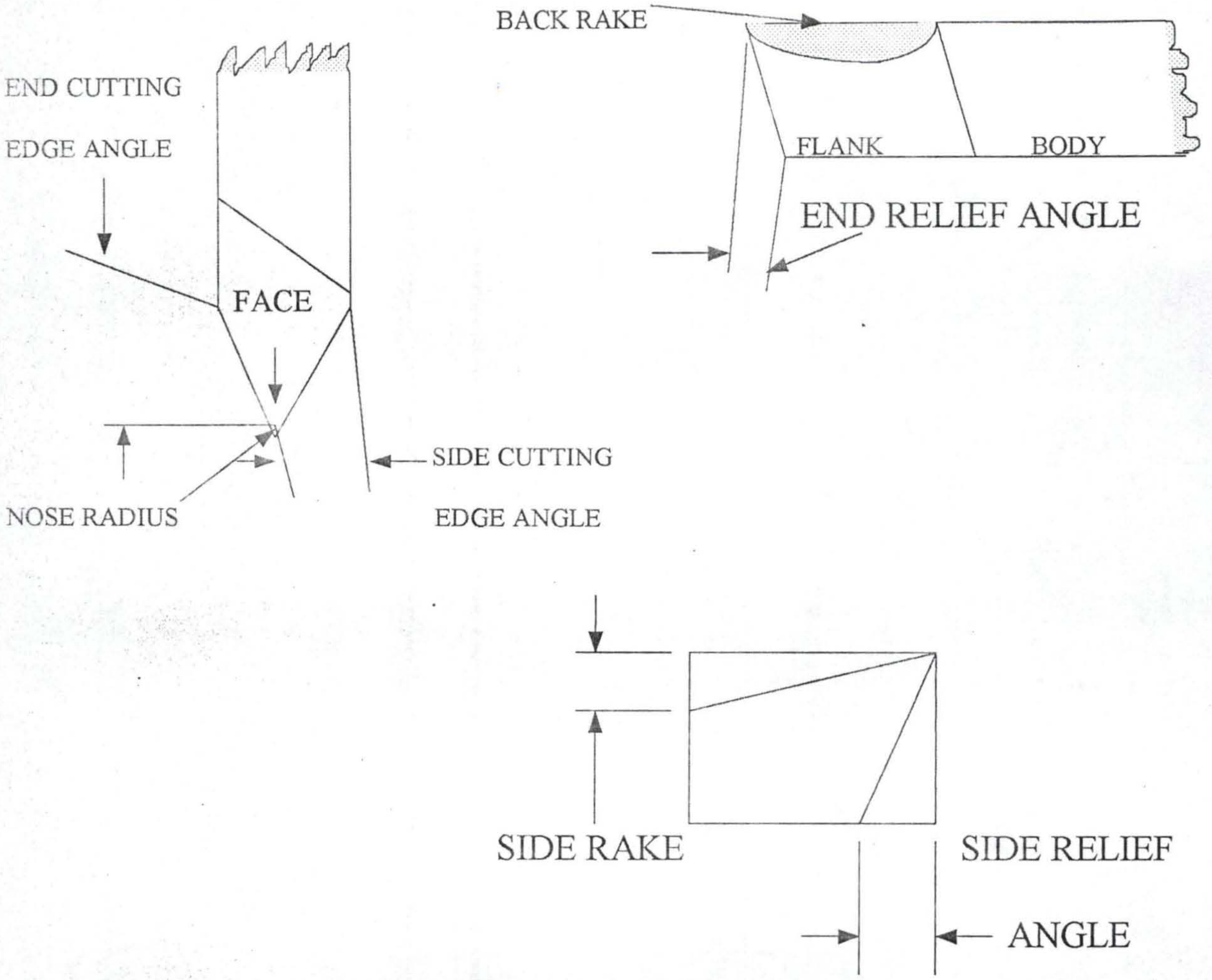


FIGURE 1.12 - THE PARTS AND ANGLES OF A TOOL

The cutting tools or tool bits, used are rectangular pieces of metal (fig 1.11), which must be very hard and it should not loose it's hardness from the heat generated by machining. The basic angles of a tool are shown in fig 1.12 namely side - relief, end - relief, back rake and side rake.

1. The tool shank is that part held by the tool holder.

2. Back rake is very important to smooth chip flow, which is needed for a uniform chip and a good finish, especially in soft materials.

3. The side rake directs the chip flow away from the point of cut and provides for a keen cutting edge.

4. The end relief angle prevents the front edge of the tool from rubbing on the work.

5. The side relief angle provides for cutting action by allowing the tool to feed into the work material.

6. The side cutting edge angle (SCEA) may vary considerably. For roughings, it should be almost square to be the work, usually about 5 degrees.

Tools used for squaring shoulders or for other light machining could have angles from 5 or 32 decrees depending on the application. This angle may be established by turning the toolholder or by grinding it on the toolbit or both. In finishing operations with a large nose radius light cut, SCEA is not an important factor. The side cutting edge angle directs the cutting forces back into a stronger section of the tool point. It helps to direct the chip flow away from the workpiece an also affects the thickness of the cut.

7. The nose radius will vary according to the finish required, the smallest nose radius that will give the desired finish should be used.

The cutting tool are ground to different shapes for different cutting operations, the most common tool shapes are illustrated in fig 1.13. It must accomodate not only the primary motion but also it must allow for feeding and chip disposal. Thus the cutting edge is inclined and the chip is wound into a helix rather than a spiral. The tool must be provided with sufficient clearance relative to the surface being machined, and if this is not achieved the tool will be rubbing against the metal. The amount of clearance should not be more than is necessary to permit the tool to cut cleanly, and 5 degrees to 10 decrees is usually sufficient. A little more clearance is generally necessary for flat facing, boring, shaping, and planning tools than for external turning. In any case, it must be recognized that the tool angles have meaning only in relation to the workpiece after installation in the machine tool.

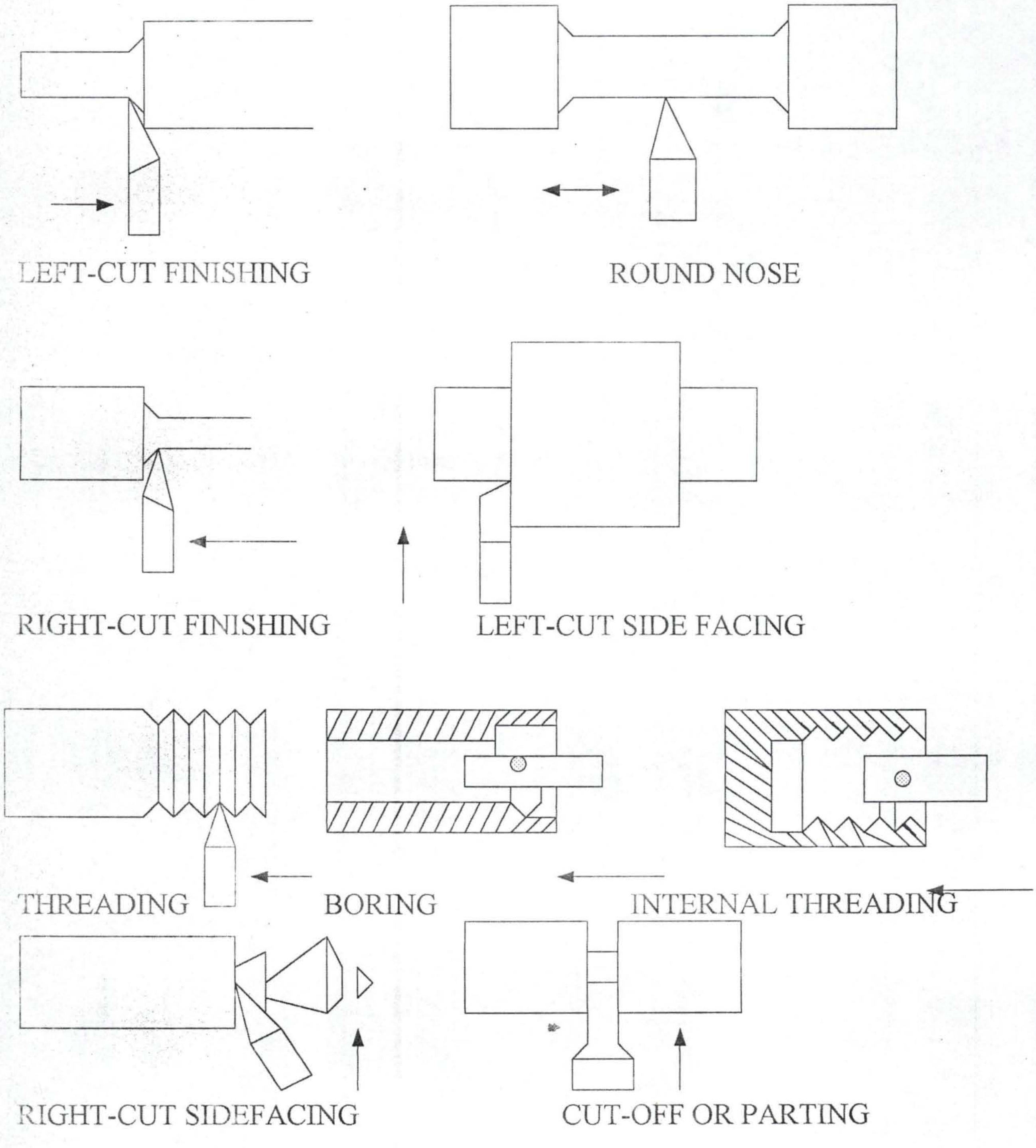


FIGURE 1.13 - COMMON TOOL SHAPES

Cutting action of the tool is supplied largely by the back and side rake angles. The slope at which the top surface is ground depends on the nature of the work material.

If it is hard and tough the rake angles are small so the cutting edge will have the maximum possible support. If it is soft, the angles may be large so the cutting edge has more of a knife - edge action. for brass and some similar materials, however, the tool is ground with no rake angle or a slight negative rake, to prevent it from digging into the work.

If the tool is held in a toolholder, it is not necessary to grind the full rake angle, because the tool is tilted up usually 15 or 20 degrees, so it is necessary to grind only enough to obtain the required variation from the angle.

High speed steel (HSS) is used for many tools as it fulfils the requirements for cutting tools and can easily be shaped by grinding. Other materials used are stelite, carbide, diamond, ceramics, or abrasive (including silicon, carbide and aluminum oxide). High speed tools have been largely replaced in production machining by carbide tools because of their higher metal rates.

However, in general machining operations high - speed tools are still used for special tooling. High speed steel tools are required for older lathes that are equipped with only low speed ranges. They are also useful for finishing operations, especially on soft materials.

The tools for lathes should have a solid shank and should be well supported by not protruding from the tool holder anymore than necessary.

1.6 TOOL HOLDER AND TOOL MOVEMENTS

One of the first rules and the most important, of efficient machine tools operation is rigid support of the cutting tool, together with secure holding or gripping of the workpiece.

A cutting tool is supported and held in a lathe by a tool holder that is screwed in the tool post of the lathe with a clamp screw. Tool height adjustments are made by swivelling the rocker in the tool post ring. Many toolholders are used with the standard tool posts.

A quick - change tool post, so called because of the speed with which tools can be interchanged, is versatile than the standard post illustrated in fig 1.14. The tool holders used on it are accurately held because of the dovetail construction of the post. This accuracy makes for more exact repetition of set ups. Tool height adjustments are made with a micrometer adjustment collar, and the height alignment will remain constant through repeated tool changes. An advantage of the quick - change tool post holder is that cutting tools of various shank thickness can be mounted in toolholders.

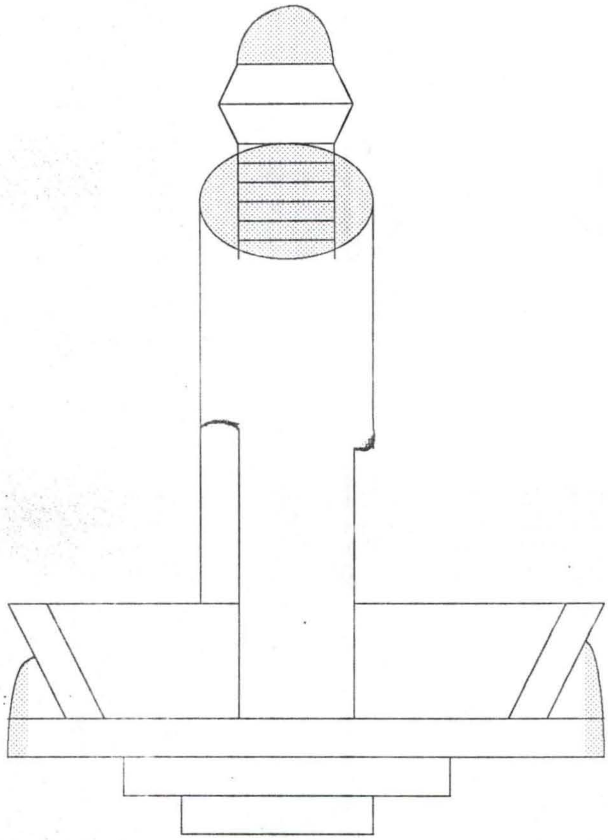
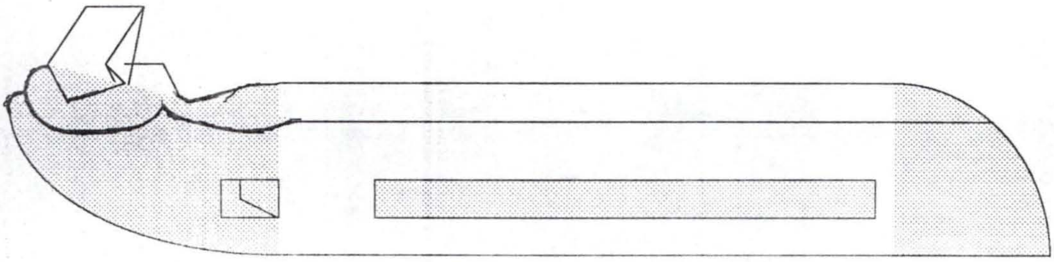


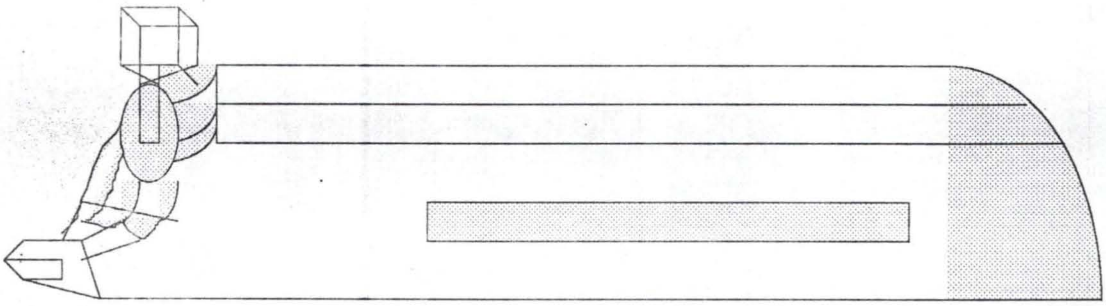
FIGURE 1.14 - STANDARD - TYPE TOOL POST WITH RING AND ROCKET

Toolholders for HSS or Carbide - tipped tools are made in variety of forms, they are shown in fig 1.15. The straight tool holder is used for external turning, but other forms have been developed for performing particular functions. The left-hand tool holder is intended for use with tools cutting fro right to left or toward the tailstock of the lathe. Right and left hand holder have the bit end bend at an angle to the shank to prevent interference with the work.

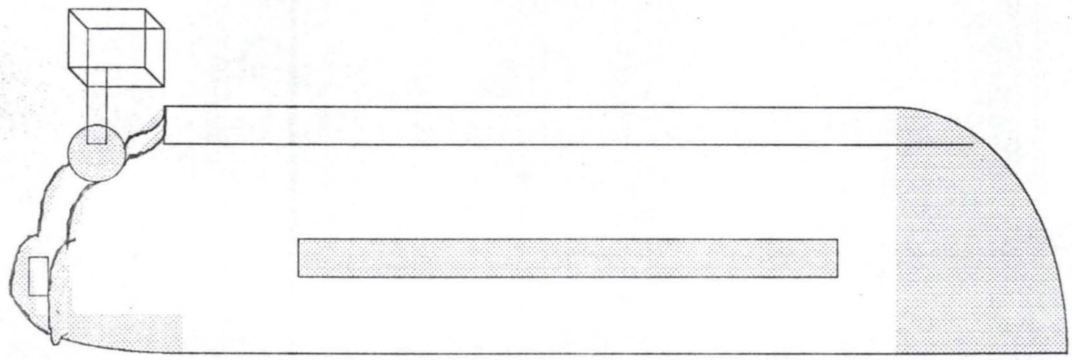
The most common types of tool holders are shown below :-



LEFT-HAND TOOL HOLDER



STRAIGHT TOOL HOLDER



RIGHT-HAND TOOL HOLDER

FIGURE 1.15 - COMMON TYPES OF TOOL HOLDERS

Many lathes can be equipped with a square turret - a special type of tool holder capable of holding four different tools. This can be turned (indexed) to twelve different positions to bring the various tools into play, as required. Off set tool holders allows machining close to the chuck or tailstock of a lathe without tool post interference.

Special toolholders are made for boring tools, cut-off tools, threading tools and knurling tools.

A tool holder should be selected according to the machining to be done. The set up should be rigid and the tool holder over hang should be kept to minimum to prevent chattering.

1.6.1 TOOL MOVEMENTS

If we remember that the cutting tool generates a cylindrical surface as it moves parallel with the lathe centre line, a plane surface as it moves at 90° to the lathe centre line, then clearly provision must be made to allow these tool movements or feeds to take place.

The versatility of the centre lathe, or its ability to machine several geometrical surfaces, arises from the varied movements of the cutting tool. We have seen that the tool needs to be set in a suitable toolpost; this toolpost is mounted on a compound slide, allowing movement of the tool at any desired angle. In its turn the compound slide is located on a cross slide, the purpose of which is to provide movement of the tool at 90° to the lathe centre. Finally the cross slide is integrated with the saddle which slide along the bed of the lathe parallel with the lathe centre line, and this allows the generating of cylindrical surfaces. The figure 1.16 illustrates in simple detail how the cutting tool is able to be moved not only parallel with the lathe centre but also at 90° to it and at a desired angle.

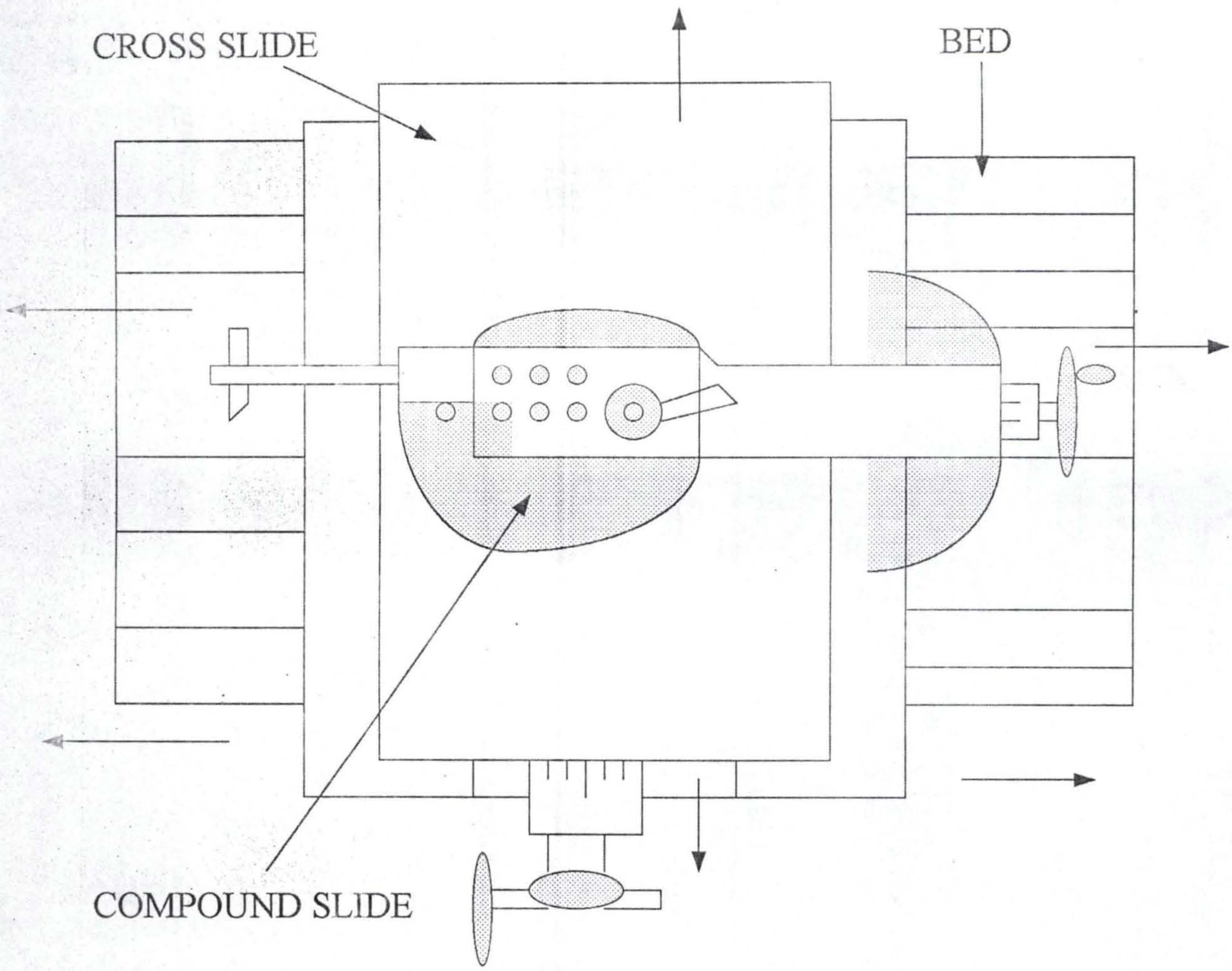


FIGURE 1.16 - TOOL MOVEMENTS ON THE CENTRE LATHE

1.7 CUTING SPEED AND FEEDS

1.7.1 CUTTING SPEED

The distance the workpiece move past the cutting point in one minute, as measured around the surface is reffered to as CUTTING SPEED. Cutting speed (CS) are normally given in tables for cutting tools materials and are based on surface metres per minute (smpm), that is, the number of metre of surface that pass the cutting tool in one minute. It can be determined from a flat surface or on the pheriphery of a cylindrical tool or workpiece. Thus a small diameter must turn or revolve faster than a larger one to cover an equal number of metres per minute.

The cutting speed is dictated mainly by the type of cutting tool material being used, the type of material being machined, rigidity of machine set up and the use of cutting fluid, but it is also influenced by the depth of cut and the feed rate. Since machine spindle speeds are given in revolutions per minute. To find the approximate number of revolutions per minute (rpm) necessary to give a certain diameter, multiply the cutting speed in surface metres per minute by 4, and divide by a diameter of the work.

$$RPM = \frac{CS \times 4}{D}$$

Where D = Diameter of workpiece

CS = Assigned Cutting speed for a particular material.

The above formula is certainly the most common one used in machine shop practice and it applies to the full range of machine tool operations, which include the lathe and milling machines, as well as the drill press.

The manufacturers of cutting tool materials give recommendation for the cutting speeds at which their tool materials will cut various work materials such as bras, alluminium, steel and so on.

A selection of these values is given for guidance in the table below :-

CUTTING SPEED (METRES/MINUTE)

Material being Cut	High speed Steel	Tungsten Carbide
Cast iron	20	160
Mid steel	28	250
Bronze	35	180
Hard brass	45	230
Copper	60	330
Aluminum	100	500

Where the workpiece rotates and the tool is stationary, as in lathes, the outer diameter (D) of the workpiece is used in the speed formula. Always observe the cutting action carefully, as the workpiece is reduced to a small diameter, make appropriate speed corrections by increasing it as needed. You should adopt the habit of carrying out the calculations before starting any machining operation, even if only as a rough mental calculation. This way you atleast have a basis for running the machine at a speed which is nearer the correct speed than if you guess.

If a machine cannot be set at the calculated rpm, use the next lower speed. If there is vibration or chatter, a lower speed will often eliminate it. As a rule, lower cutting speeds are used to machine hard or tough materials or where heavy cuts are taken and it is desirable to minimize tool wear and thus maximize tool life. Higher cutting speeds are used in machining softer materials in order to achieve better surface finish.

1.7.2 CUTTING FEEDS

The machine movement that causes a tool to cut into or along the surface of a workpiece is called FEED. This is the distance the tool advances for each revolution of the work. Feed is measured in millimetres per spindle revolution.

Generally, coarser feeds are used for rough turning and finer feeds are used for finished turning. These are set by changing the levers on the quick change gearbox. Coarser feeds, depending on the machine size, horse power, and rigidity, necessary for rapid stock removal, are called ROUGHING.

As long as the roughing dimension is kept well over the finish size, feeds should be set to the maximum the machine will handle. In roughing operations, finish is not important and no attempt should be made to obtain a good finish; the only consideration is stock removal and of course, safety. It is in the finishing cuts with fine feeds that dimensional accuracy and surface finish is of the greatest importance and stock removal is of little consideration.

If the machine labours or vibrates, it is overloaded, reduce the feeds. If it is cutting easily, it may be possible to increase the feed. The condition of the tool gives another indication, if the edge breaks down too rapidly or builds up excessively, reduce the speed or feed.

1.7.3 DEPTH OF CUT

This is the distance from the bottom of the cut to the un-cut surface measured at right angle to the machined surface.

To determine the depth of cut, the diameter of the work should be measured prior to and after removal of a metal layer by the cutting tool. The depth of cut equals half the diameter difference.

For most efficient production, run the lathe at the highest speed and coarsest feeds which will give the desired results. This is because feed and speeds determines the rate of metal removal.

1.8 CUTTING FLUIDS

A great deal of heat is generated in any machining operation, as such it is necessary to pour some sort of cooling solution over the tool and the work.. Otherwise, the work may be distorted and the edge of the tool spoiled. The machining of most metals is greatly improved by the use of a cutting lubricant.

The most common type of lubricant used when cutting most materials is a soluble oil, which when mixed with water, forms a white solution known as 'Suds' or 'Shurry'. This has better cutting properties than oil, but does not lubricate as well. The oil part of it is generally a mineral oil mixed with a soap solution, sometimes with an extreme pressure agent added.

Animal mineral lard oils, either straight or compounded with kerosene or parafin oil, are good for general machining operations. Extremely hard or tough materials usually require oils with a sulphur or chlorinated base. For the highest grade of work, many machinist prefer pure lard oil, though it is expensive.

The improvement effected using a lubricant are as follows :-

(a) The tool and work are cooled and higher cutting speeds and feeds may be used.

(b) The cutting fluid helps in lubricating the severe rubbing action taking place between the chip and the top face of the tool. The effects, some saving in power, prolongs the life of the tool and promotes a better finish.

(c) A heavy flow of lubricant helps to wash away the chips and keep the cutting point clear.

The ideal cutting fluid, in achieving the improvements above should :-

(a) Not corrrode the work or machine.

(b) Have a low evaporation rate.

(c) Be stable and not foam or fume.

(d) Not injure or irritate the operator.

Cast iron should always be machined dry, because the crumbly chips forms a paste with coolants and clog the system. Magnesium chips are prone to catch fire in contact with water, and machining should be done dry or with kerosene. Brass, bronze and aluminium may be cut dry or wet, whilst steel should always be machined with lubricant. Cutting lubricants may consist of a pure oil, a mixture of two or more oils - or a mixture of oil and water.

1.9 LATHE OPERATIONS

The centre lathe can be defined as a machine tool use to produce cylindrical, flat and conical surfaces by a generating process and the use of a single point tool. The shape of the surface produced by a generating process depends entirely on the relative motion between tool and work, not on the shape of the tool. All other operations carried out on the lathe are either the combination of or by product of these three major operations mentioned above.

In generating a surface, two motions are required, plus a further one to adjust the depth of cut, this adjustment remaining fixed while each cut is in progress.

Applying the above principle to the lathe, the three major operations on the lathe are illustrated below in fig 1.17

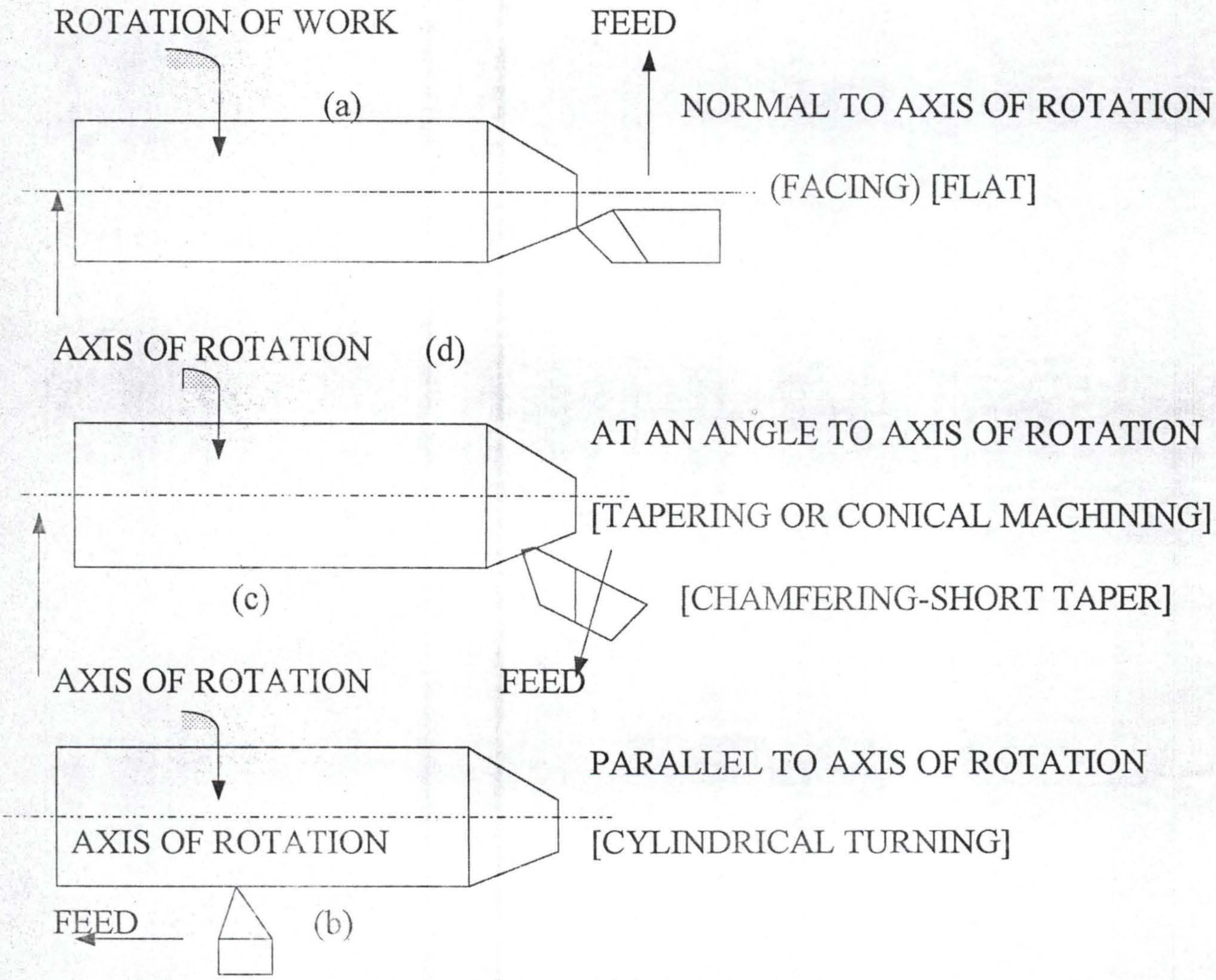


FIGURE 1.17- MAJOR OPERATIONS ON THE LATHE

The followings should be observed in the diagrams of operations on typical lathe machine.

(a) The work is provided with a rotary motion.

(b) The linear motion of the tool parallel to the axis of rotation of workpiece generates cylindrical turning

(c) The linear motion of the tool at an angle to the axis of rotation of workpiece generates tapering otherwise known as conical machining. Short tapers are known as chamfers.

(d) The linear motion of the tool at right angles to the axis of the workpiece generates a flat surface otherwise known as facing.

Power is transmitted from the spindle through a train of end gears to the leadscrew, an accurately made screw which runs parallel to the ways and engages in the carriage apron to produce the desired feed for turning and screw cutting or threading. On precision lathes, the leadscrew is reserved exclusively for cutting and, to avoid unnecessary wear, a separate feed rod is provided for longitudinal and cross feed.

Feed rate, in terms of thousandth an inch per revolution of the spindle, or in threads per inch, is usually controlled by a quick - change gearbox, although many older styles still make use of pick-off gears; that is, individual gears which must be combined in a suitable train to furnish the desired feed rate.

The rotary motion of the work necessitates for a source of power usually using an electric motor. Revolution on the lathe can vary from about 60 revolution per minute to about 1700 revolutions per minute. Other lathes can be designed to go faster. Medium speed 370 revolutions per minute is used generally for turning, facing etc.

The tailstock of a lathe can be replaced by a turret, when the surface to be generated or formed with relative simple motions requires a larger number of tools and operations (such as turning, facing, boring, and drilling) for completion. Equipped with a quick clamp device, a turret brings several (usually six) tools into position very rapidly. All six faces of this turret can hold tools of various kinds that can be used on the workpiece, one after another, for example, one face may hold a drill; the next, a reamer; the next, a tap; and so on. As the turret moves backward, it automatically rotates to the next tool. The order of operations needed determines the arrangement of tools in the turret. All tools are fed in the axial direction, by moving the turret on a slide (ram-type lathe) or, for heavier work, on a saddle which itself moves on the ways (Saddle-type lathe).

Before machining or undertaking machining operations, you should get to know the parts of the lathe and learn how to operate the controls.

(a) Try the levers and handles with the power-off.

i Move the carriage back and forth.

ii Move the cross slide in and out.

iii Slide the tailstock back and forth, and lock it in position.

iv Change the position of the belt on the pulley.

v Change the feed-reverse lever.

vi Disengage the bull-gear lock or pin. Pull the back gears in place.

vii Adjust the feed on the quick - change gearbox.

(b) Try adjustments after turning on the power. Operate the carriage with power feed. Also use the automatic cross-feed.

Clearly, the operations possible on a centre lathe are many and varied, and it has always been recognized as the most versatile of all machine tools available to the engineer.

1.10 MACHINING OPERATIONS

The major operations performed on a lathe are turning, facing, boring and threading or screw cutting. Minor ones include , taper-turning, knurling, drilling, cutting or parting-off, reaming, filing, polishing, centre-drilling etc.

1.10.1 TURNING - Turning applies specifically to the machining of external cylindrical surfaces. The work revolves about it's axis, while the single-point cutting tool follows a path parallel with the centre line of revolving work.

In this way a series of concentric circles are machined on the work as the tool is parallel with the axis of rotation of the work a cylindrical surface is machined. The most commonly used tool is the right - hand turning tool ground to suit the material to be cut.

1.10.2 FACING - Facing is the operation of cutting the end face of a section at right angles to the axis of the work using a single - point cutting tool, to obtain a flat surface on the end of cylindrical workpieces or on the face of parts clamped in a chuck or face plate.

1.10.3 BORING - The cutting of the internal surface of a hollow part using a single - point tool on a lathe machine is reffered to as BORING. It is a process whereby an existing hole is enlarged and finished to a standard size required. For example to trim out a hole in a casting, or to finish a very accurate hole of any size.

The tool use may be of one piece construction, forged to the required shape, but it is usually more convenient to employ a round bar with a small tool bit inserted near one end.

1.10.4 REAMING - This is a process that is done on a lathe to quickly and accurately finish drilled holes to size.

Best results are obtained if the tool is held in the tailstock in a floating holder to permit it to shift slightly and centre itself in the hole. It can be performed with standard reamers to smoothly and slightly enlarge drilled holes. Reduce speed to about one - half of the speed for drilling, and feed the reamer as fast as possible.

1.10.5 CUTTING OR PARTING - OFF - This is the process of cutting - off material to the correct or required length. Two surfaces perpendicular to the lathe axis are produced by moving the single - point tool in the carriage so that the feed motion is toward the centre of the lathe.

1.10.6 KNURLING - This is a process of creating a rough gripping surface on the circumference of work by pressing a straight or diamond shaped pattern. It is performed with a special knurling tool which is made in several patented styles.

Set the knurling tool in the tool post so that knurls are the same distance above and below the centre line of the work and parallel to it. Make sure both rolls are in contact with the work. Set the machine for a medium feed and speed, force the rolls into the work, and engage the longitudinal feed. Cut the full length without stopping the carriage, or rings will be left in surface.

Diamond knurling is used to improve the appearance of a part and to provide a good gripping surface for levers and tool holders. Rolls for straight line patterns are used to increase the size of a part for press fits in light - duty application. Either pattern can be had in coarse, medium or fine pitch.

1.10.7 TAPER - TURNING - A taper may be defined as a uniform increase in diameter on a workpiece for a given length measured parallel to the axis. Tapers are very useful machine elements that are used for many purposes.

Tapers are used on machines because of their capacity to align and hold parts and to re-align when they are repeatedly assembled and dis-assembled. Many parts and tools made in lathes have tapered surfaces, varying from the short steep tapers found on bevel gears and lathe centre ends to the long gradual tapers found on the mandrels.

There are several methods of turning a taper. The most commonly used methods are :

(a) Using the compound slide or rest.

(b) Off-setting the workpiece from the centre.

(c) Using the taper turning attachment.

And other methods include; the use of a form tool, and the use of a tracer or CNC lathe.

Each method has it's advantages and disadvantages, so the kind of taper needed on a workpiece should be the deciding factor in the selection of the method that will be used. Generally, only short tapers can be turned. Taper turning is performed like any other turning or boring operations.

1.10.8 FILING - It is performed on a lathe to obtain a fine finish or reduce a diameter slightly. It is difficult to produce a true cylinder by filing, so other factors being equal, adopt some other methods of finishing if possible.

Employ, a single - cut, smooth, mill file, and hold the tip between the thumb and fore finger of one hand, and handle with the other hand. Take light strokes at a slight angle to the work, and move along the work with overlapping strokes until the whole surface have been covered.

Don't hold the file still and rotate the work against it. Work just as in bench filing, but less pressure. This is rather a dangerous job, so take care that your arms or fore fingers don't touch the chuck or get jammed against the tool rest. Don't use a file without a handle, if it sticks and bites in, the file tang can be driven into your palm, causing serious injury.

Filing should be done at higher speeds than turning and the machine ways should be covered with paper or cloth as protection against the fine chips.

1.10.9 DRILLING - It is performed on a lathe with the same types of tools used in a drill press. In the lathe, however, the work, rather than the tool turns. Small straight-shank drills are held in drill chuck in the tailstock. Larger, taper-shank drills may be held directly in the tailstock spindle or in an adaptor. Some drilling may be done with the drill held in the headstock spindle and the work supported by a special tailstock centre.

Hold the work in a chuck if it is short, or in a chuck and steady rest if it projects far beyond the chuck, make sure it is faced square and running true before starting to drill. Start the hole with a centre drill at the centre of the face. Advance the tailstock until the drill almost touches the work, and clamp it to the ways.

Feed the drill with the tailstock hand-wheel. If the drill wobbles as it starts to cut, put a tool holder in the tool post and bring the butt end up very close to the drill point to hold it steady. A scale on the tailstock spindle enables the depth of full penetration to be measured; but be sure to note the scale reading at the start of the cut.

For deep holes, advance the entire tailstock beyond the range to prevent the spindle from being fed.

For large holes, drill a small pilot hole to the full length. This will prevent the large drill from running out.

For holes more than about 3 diameter deep, back the drill out of the hole at frequent intervals to clear the chips.

1.10.10 THREADING OR SCREW CUTTING - Thread cutting is an important function of the engine lathe and is performed by establishing a positive relationship between the rotation of the spindle and the feed of the carriage. For example, if a screw is to have 8 threads per inch (tpi), the spindle must rotate 8 times while the carriage moves exactly 1 inch. Spindle speed and carriage feed ratio is determined by a series of gears in the quick - change gearbox.

The nature of the thread to be cut determines the type of cutting tool to be used. Special forms are required for national standard, Acme, whit worth, square and other types.

Threads can be cut with a tool bit ground to a 60° point, with the face flat and the end relieved at both sides for external threading. And for internal threading, the same type of tool bit is used and held in a boring tool, setting the compound rest at

29° improves the operation. The tool is set squarely in the tool post, and with the compound rest set to feed squarely across the ways.

The tool for National Standard (NS) thread is ground to an accurate 60° V, and is relieved at the end and on both sides. The tool for square threads has no relief at the front end. A 90° angle at this point prevents digging in. Circular form tool may be used in a special tool holder for external threading.

To ensure an accurate thread, set the tool such that the centerline of the V is exactly perpendicular to the axis of the work. To cut a thread, set the speed at about one-third to one-quarter of the normal turning speed for the particular material, advance the tool until it just scratches the work, and engage the leadscrew by shifting the half-nut lever.

1.10.11 POLISHING - Polishing is sometimes performed on a lathe to obtain a fine finish or reduce a diameter slightly. It is done with a strip of abrasive cloth stretched over file or placed over the work and held with the fingers. The finer the abrasives, the slower the cut and the higher the finish. A drop of oil on the cloth will give a still finer finish.

Polishing should be done at higher speed than turning and cover the machine ways with paper or cloth as protection against the powdered abrasives.

1.10.12 CENTRE DRILLING - When work is held and turned between centres, a centre hole is required on each end of the work piece. The centre hole must have a 60° angle to conform to the centre and have a smaller drilled hole to clear the centre's point. This centre hole is made with a centre drill, often reffered to as a combination drill and counter sink.

These drills are available in a range of sizes from $^1/_8$ to $^3/_4$ inch body diameter and are classified by numbers from 00-8 which are normally stamped on the drill body. Facing the workpiece is almost always necessary before centre drilling because an uneven surface can push sideways on the fragile centre drill point and break it.

Centre drills are usually held in a drill chuck in the tailstock, while the workpieces are most often supported and turned in a lathe chuck for centre drilling. As a rule centre holes are drilled by rotating the work in a lathe chuck and feeding the centre drill into the work by means of a tailstock spindle.

1.11 LATHE SAFETY

The workpiece on the lathe rotates, usually at high speed and with a considerable amount of power behind it. As such when using the lathe, you must understand and obey the following safety rules.

(a) Only trained workers adequately briefed to operate the machine by the responsible person are allowed to work after they had participated in the obligatory training for prevention of accidents.

(b) Correct dressing is important :- It is strictly forbidden for the operator to wear wrist watches, rings, necklaces, neckties or dresses whose loose - fitting parts can be picked up and draw in. Roll up your sleeves. Wear your apron and put on your safety glasses to protect your eyes from swarf if the lathe has no chip guard.

(c) Make sure your workpiece is set up securely and tightly, when using chucks and collets. And the chuck key must not be left in the chuck, not even at it's steady position.

(d) The covers of the rotating parts must continuously be closed. All rotating and moving parts represent hazard, and the hands should be kept clear. Do not be tempted to touch swarf, stationary or moving with the bare hands or a deep cut can result; stop the machine and use a swarf - hook.

(e) When starting the operation of the lathe, the operator must check the right rotating sense. And always check in what direction and how fast the carriage or cross feed will move before you turn on the automatic feed.

(f) Only the operator may stay in the strict area of the machine when it is in operation. Accidents happen when your mind is distracted.

(g) Coolant should be fed to the job through pipes, not brushed on, and the machine should be allowed to stop itself - hands are too delicate and precious to use as brake bands on the chuck.

(h) Stop the lathe before making adjustments. And never measure workpiece while it is turning.

(I) It is unsafe to have small - diameter workpiece extended more than an inch or two (25mm to 50mm) from a chuck or collet unless it is supported by the tailstock centre.

(j) Keep rags, cotton waste, and brushes away from tools while turning.

(k) Do not run the carriage or compound slide into a turning chuck.

(l) When holding the workpiece between centres, make sure to use centres of the right size and with good points. Never use a soft centre in the tailstock. Apply oil or white lead to tailstock centre, and adjust it properly. If it is too tight, the point will heat up and burn off.

(m) Make sure the tool bits are sharp and ground to the correct shape. Set them at the proper height and angle to the work.

(n) After setting up the lathe, remove all wrenches, oil cans, and other tools from the work area. When the workpiece is held on a face plate, give the face plate one complete turn by hand to make sure the work will not strike any part of the lathe.

(o) Never remove the guards over belts and gears, shut off the power at the switch board before removing guards.

(p) In case of power cut, switch off the main switch. (Turn it to the "off " position). ATTENTION! If any disorder occur during the operation, the main switch must be turned off at once.

CHAPTER TWO: FACING MACHINING OPERATION

2.1 FACING

The material to be machined usually has been cut in a power saw, so the piece is not square on the end or cut to the specified length. Facing is done to obtain a flat surface on the end of cylindrical workpieces or on the face of parts clamped in a chuck or face plate.

Facing and centre drilling the workpiece are often the first steps taken in a turning project to produce a stepped shaft or a sleeve from solid material. Facing produces a plane perpendicular to the lathe axis by moving the single point tools in the carriage so that the feed motion is either towards or away from the centre of the lathe. If the tool moves towards the centre line or centre of the lathe, the operation is reffered to as **FACING INWARD**. If the tool moves away from the centre of the lathe the operation is reffered to as **FACING OUTWARD.**

Facing from the centre out produces a better finish, but it is difficult to cut on a solid face in the centre. Facing from the outside is more convenient since heavier cuts may be taken and it is easier to work to the scribed lines.

2.2 FACING TOOLS

Variety of tools are used depending upon the type of facing operation that is to be undertaken.

When facing from the centre out, a right - hand turning tool in a left - hand tool holder is the best arrangement, but when facing from the outside to the centre, a left - hand tool in a right - hand or straight tool holder can be used.

The most commonly used tool shapes for facing (Surfacing) are shown in fig 2.1 and they include the followings;

(a) Right - hand roughing and turning tool.

(b) Right - hand finishing tool.

(c) Left - hand roughing tool.

(d) Left - hand finishing tool.

(e) Right - cut and left - cut side facing tools.

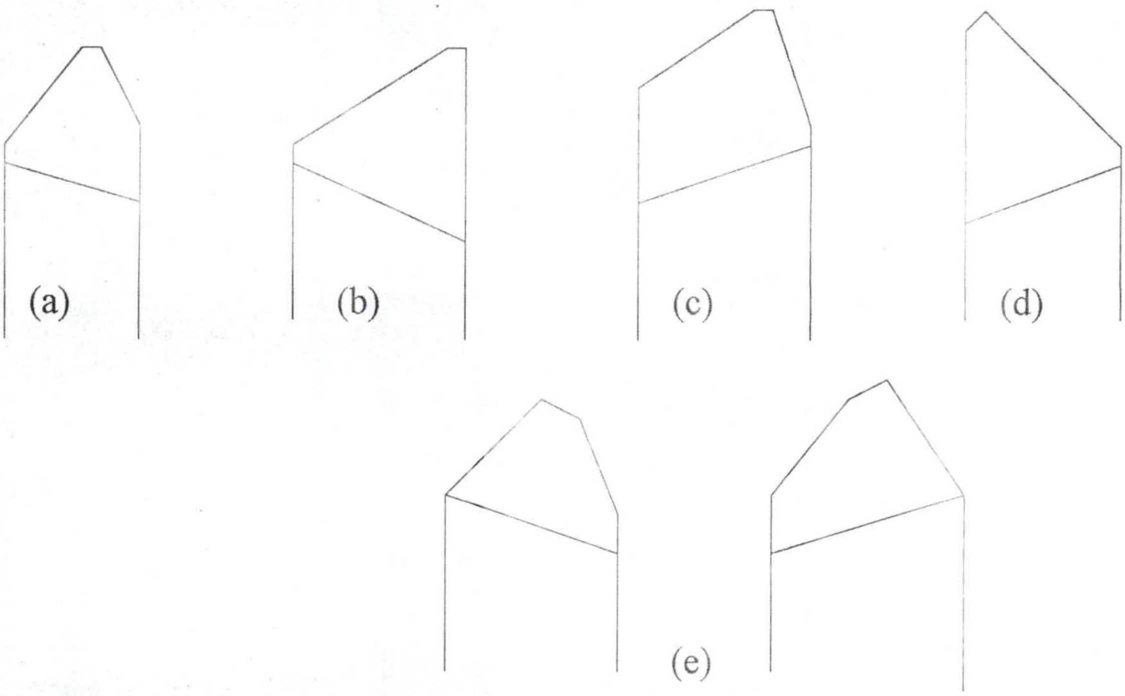


FIGURE 2.1 - COMMONLY USED TOOL SHAPES FOR FACING (SURFACING)

2.3 SPEED

Speeds (rpm) for lathe turning a workpiece are determined in essentially the same way as speeds for drilling tools. The only difference is that the diameter of the work is used instead of the diameter of the drill.

In facing operations, the outside diameter of the workpiece has greater surface speed than it's centres. For this reason the rpm should vary as the tool is moved in or out. This is easily done with a variable speed control on the machine, but when belts or gears must be changed, the feed must be disengaged and the lathe stopped.

Two or three speed changes on facing work may be required in order to get a uniform finish, depending on the size of the work. For facing work, the outside diameter is always used to determine maximum rpm.

$$\text{Thus RPM} = \frac{CS \times 4}{D}$$

where;

CS - Cutting Speed

D - Diameter of workpiece

RPM - Revolutions per minute

A right-hand (RH) or left-hand (LH) roughing or finishing tool often used for facing in chuck mounted workpieces. The right-cut or left-cut side facing tool is used for shaft ends and mandrel work, it has a very little nose radius and an inclined angle of 58' degrees. This facing tool is not used for chucking work, however, as it is a relatively weak tool.

The right-hand facing tool is specially ground or shaped to fit in the angle between the centre and the face of the workpiece mounted between centres to enable the workpiece to be faced. Both right- hand and left-hand facing tools are used for facing work held on mandrels. Care should be taken when machining pressure is toward the small end of a tapered mandrel (usually toward the tailstock). Excessive pressure may loosen the work piece on the mandrel.

Turning to size and facing on a shoulder or off set requires a tool that can cut on both the end and the side. Roughing should be done on both the diameter and face before finishing to size. The diameter is usually the critical dimension and should therefore be finished to size after the face is finished.

2.4 FACING OPERATION

The work most often is held in a three or - four jaw chuck. If the chuck is to be removed from the lathe spindle, a lathe board must be placed on the ways.

A facing cut (fig 2.2) is taken when the tool follows a path at 90^{o} to the lathe centre line, this is achieved through movement of the cross slide, either by hand or automatic traverse. Facing or other tool machining should not be done on workpieces extending more than five diameters from the chuck jaws. The carriage must be locked to bed with the carriage - lock screw when taking facing cuts as the cutting pressure can cause the tool and carriage to move away, which would make the faced surface curved rather than flat.

Facing operation can be undertaken through the following steps :-

(1) Move the tailstock until the dead centre just touches the live centre. Be sure the centres are aligned. If they are not, move the top casting of the tailstock toward or away from you.

(2) Fasten the lathe dog to one end of the workpiece.

(3) Move the tailstock assembly until the opening between centres is a little longer than the workpiece. Lock the tailstock in position.

(4) Slip the tail of the lathe dog in the opening in the drive plate. Place the workpiece between the centres. Apply a lubricant to the dead - centre hole, use white lead oil. Tighten the dead - centre until the workpiece is held snugly. It should not be so loose or so tight that the dead - centre becomes burned.

Remember that since metal expands when heated, you will have to re-adjust the dead-centre after two or three cuts.

(5) Place a facing tool in a toolholder with the point well extended. The shape of the tool depends upon the type of facing operation to be undertaken.

(6) Place the toolholder in the tool post, with the point cutting edge at right angles to the centre line. Adjust this by eye.

(7) Face the right end of the workpiece, placing your left hand on the carriage hand wheel and your right hand on the cross feed handle. This is done when a right-cut side facing tool is placed in the toolholder.

(8) For rough cuts, choose a left-cut roughing tool. The tool is fed from the outside of the workpiece toward the lathe centre. A roughing feed could be from .005 to .015 inch.

For finishing cuts, use a right-cut facing tool clamped in a left - hand holder. The tool is always fed from the centre hole toward the outside. A finishing feed is from .003 to .005 inch.

Finer feeds should be used for finishing than for roughing. Adjust the tool holder until the cutting edge is at an angle of about 8 to 10 degrees to the workpiece face. Use the power cross - feed for large diameter stock. The cross feed is one half to one third that of the longitudinal fed. The ratio is usually listed on the index plate of the quick - change gearbox. Use of cutting oils will help produce better finishes on finish facing cuts.

(9) Move the carriage to the left until a light cut is taken. Then feed the tool. Reverse the workpiece in the lathe. When face the other end.

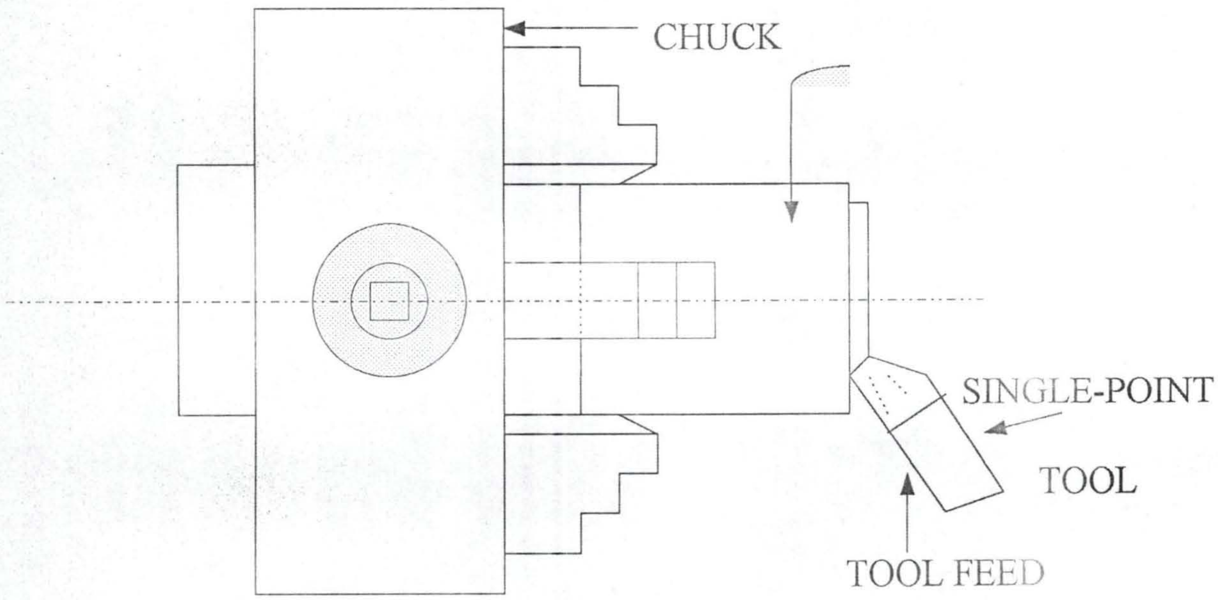


FIGURE 2.2 - FACING CUT

Facing to length may be accomplished by trying a cut and measuring with a hook rule or by facing to previously made layout line. A more precise method is to use the graduations on the micrometer collar of the compound. The compound is set so that it's slide is parallel to the ways. The carriage is locked in place and a trial cut is taken with the micrometer collar set on. The workpiece is measured with a micrometer and the desired length is subtracted from the measurement; the remainder is the amount you should remove by facing.

Half centres (fig 2.3) are made to make the job easier, but they should be used only for facing and not for general turning. If the tailstock is move off centre away from the operator, the shaft end will be covex; if it is moved toward the operator, it will be concave. The half centre resembles a regular centre, but has one side partly ground away to provide more clearance for the tool. Long workpieces, however are generally faced by chucking one end and supporting the other in a steady rest.

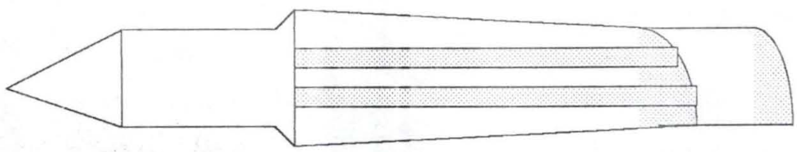


FIGURE 2 .3 - HALF CENTRE MAKE FACING SHAFT ENDS EASIER

CHARTER THREE : BORING AND CUTTING OR PARTING - MACHINING OPERATIONS

3.1 BORING

The term "BORING" on an engine lathe is applied to internal cutting with a single point tool. It is the chief means of enlarging and trueing an existing or drilled hole.

Boring can be used to correct errors in concentricity and alignment of a previously drilled hole. The hole can be finished to size by boring without the use of a reamer as would be the case when producing non - standard diameter for which a reamer was not available. Boring is also use to produce a recess which may not be practical by drilling and reaming. It is usually limited to a finishing operation after the hole must have been drilled.

3.2 BORING TOOL

The tool may be of one piece construction, forged to the required shape, but it is usually more convenient to employ a round bar with a small tool bit inserted near one end (fig 3.2)

Large variety of boring bars and holders are used. Boring bars designed for small holes (12.5 mm and smaller) are usually the forged type(fig 3.1). The forged end is sharpened by grinding when the bar gets ground too far back, it must be re-shaped or discarded.

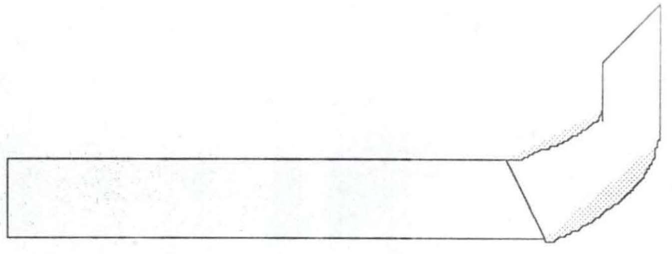


FIGURE 3.1 - FORGED BORING BAR

Boring for holes with diameter over 12.5mm use high speed tool inserts, which are typically hand ground in the form of a left-hand turning tool. These tools can be removed from the bar for re-sharpening when needed. The cutting tool can be held at various angles to obtain different result, which makes the boring bar useful for many applications. Standard bars generally comes with a tool angles of 30, 45 or 90 degrees. Some boring bars are made for carbide inserts.

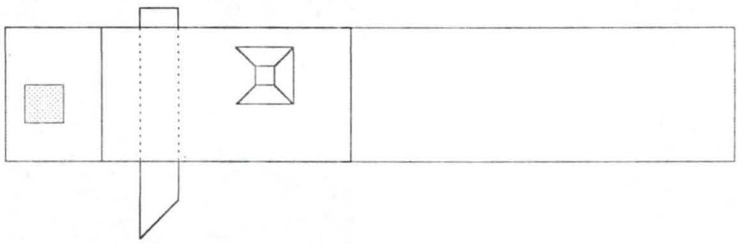


FIGURE 3.2 - BORING BAR WITH INSERTED TOOL BIT

Boring bar is clamped in a holder mounted on the carriage compound slide. Boring tool must be smaller then the bore it is producing, and this invariably results in a thin flexible tool. For this reason, it is not usually possible to take deep cuts, and care must be taken to avoid vibration. Boring tools must have sufficient side relief and end relief to be efficient cutting tools, but usually back rake is not normally used . The end relief should be between 10 and 20 degree. Insufficient end relief will allow the heel of the tool to rub on the workpiece. The machinist must use judgement when grinding the end relief because the larger the bore, the less end relief is required. If the end relief of the tool is relieved too much, the cutting edge will be weak and break down.

In selecting a boring tool, choose the thickest one which will enter the hole, to ensure maximum rigidity. Ensure also that adequate secondary clearance is provided in relation to the size of bore being produced.

3.3 SPEED

Speeds and feeds for boring are determined in the same way as they are for external turning. Such as facing, cutting, or parting - off etc.

3.4 BORING OPERATION

Most boring is performed on workpieces mounted in a chuck. But it is also done in the end of workpieces supported by a steady rest. A drilled hole for boring can be from $^1/_{32}$ to $^1/_{16}$ inch (.7938 - 1.5871 mm)

Boring to size predictable is also done in the same way as in external turning except that the cross feed screw is turned counter - clockwise to move the tool into the work. All boring work should have the edges and corners broken or chamfered. Boring operation is illustrated in fig. 3.3.

Steps through which boring operation can be undertaken on a lathe machine are as follows :-

1. Hold the boring tool in the holder so it projects only just far enough to complete the job, and use the longest tool that will clear the hole to obtain maximum rigidity.

2. For short lengths, the tool may be mounted cantilevered in the tool post. While for long lengths to avoid excessive vibrations, it is preferable to secure the work piece to the lathe bed while the boring bar is clamped in jaws at one end and supported in the tailstock at it's other end, is driven.

3. Set the cutting edge exactly on centre line of the workpiece, so the dial graduations on the cross feed will read accurately in thousandths of an inch.

4. Start the machine and use the calculated rpm of the spindle.

5. Take trial cuts and check the diameter carefully before engaging the longitudinal feed. It may be necessary to take several trial cuts to obtain the final diameter, but remember that it is easy to take a little more stock off, but there is no way of putting it back on, except by welding, metallizing or plating.

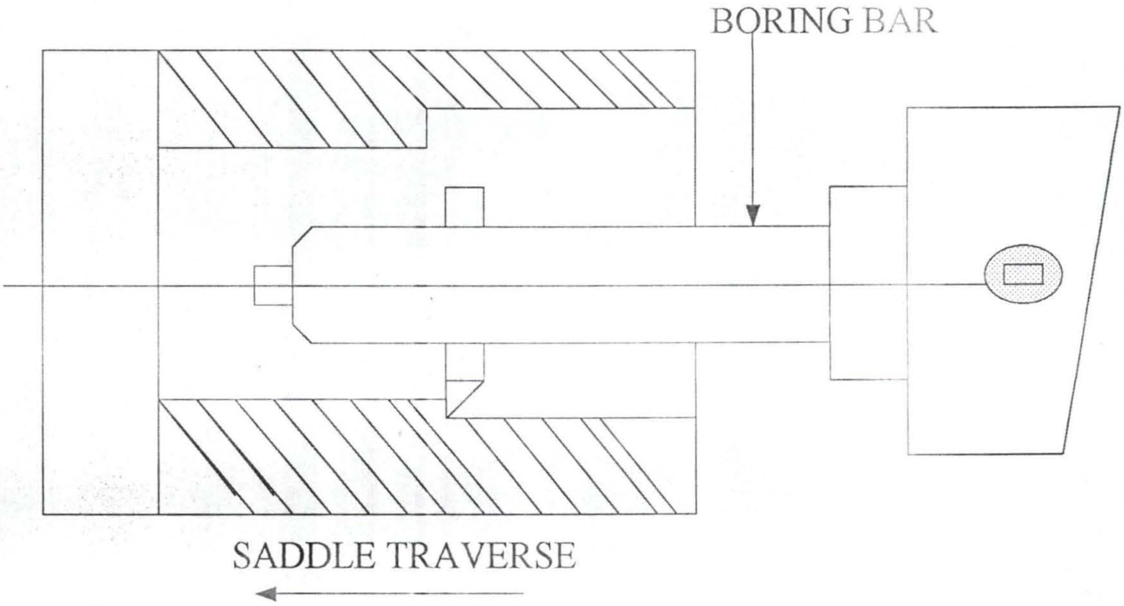


FIGURE 3.3 - BORING OPERATION

Vernier calipers are used by machinist for internal measuring, though the telescoping gauge and outside micrometer are most commonly used for the precision measurement of small bores because they can take a more accurate measurement. Inside micrometers can be used for bores over $1^1/_2$ inches (38 mm). Precision bore gauges are used where many bores are checked for similar size, such as for acceptable tolerance.

Chatter is the vibration between a workpiece and a tool because of the lack of rigid support for the tool. Chatter is a great problem in boring operations since the bar must extend away from the support of the compound. For this reason, boring bars should be kept back into their holders as far as practicable. Tuned boring bars can be adjusted so that their vibration is dampened.

If chatter occurs when boring, one or more of the following may help to eliminate the vibration of the boring tool.

1. Shorten the boring bar overhang, if possible.
2. Increase feed.
3. Make sure that the tool is on centre.
4. Reduce the spindle speed.
5. Use a boring bar as large in diameter aspossible without it binding in the hole.
6. Reduce the use radius on the tool.
7. Apply cutting oil to the bore.

Boring bars sometimes spring away from the cut and cause bell mouth, a slight taper at the front edge of a bore. One or two extra cuts (Called free cuts) taken without increasing the infeed will usually eliminate the problem.

Through boring is the boring of a work-piece from one end to the other or all the way through it. For through boring, the tool is held in a bar that is perpendicular to the axis of the workpiece. A slight side cutting edge angle is often used for through boring.

Counter boring in a lathe is process of enlarging a bore for definite length. The shoulder that is produced in the end of the counter bore is usually made square (90 degrees) to the lathe axis.

Boring and counter boring are also done on long workpieces that are supported in a steady rest.

3.5 CUTTING OR PARTING OFF MACHINING OPERATION

This is the process of cutting - off materials to the correct or required length. Two surfaces perpendicular to the lathe axis are produced by moving the single point tool in the carriage so that the feed motion is toward the centre of the lathe.

3.6 PARTING TOOLS

When a piece is to be cut - off from the stock a cut - off or parting tool is employed. Parting tools are also for necking or undercutting, external grooving and thread relief.

It may be a narrow, square - nosed tool, tapering back about 2^{o} on each side to prevent friction between tool and work, having a 2^{0} relief on each side, or it may be in the form of a narrow blade held in a special tool holder. Instead of being square, the cutting edge may be beveled to prevent leaving a burr on the piece of cut - off.

Parting tools (fig 3.4) are tools made in either straight or offset types. All parting and grooving tools have a tendency to chatter; therefore any set up must be as rigid as possible.

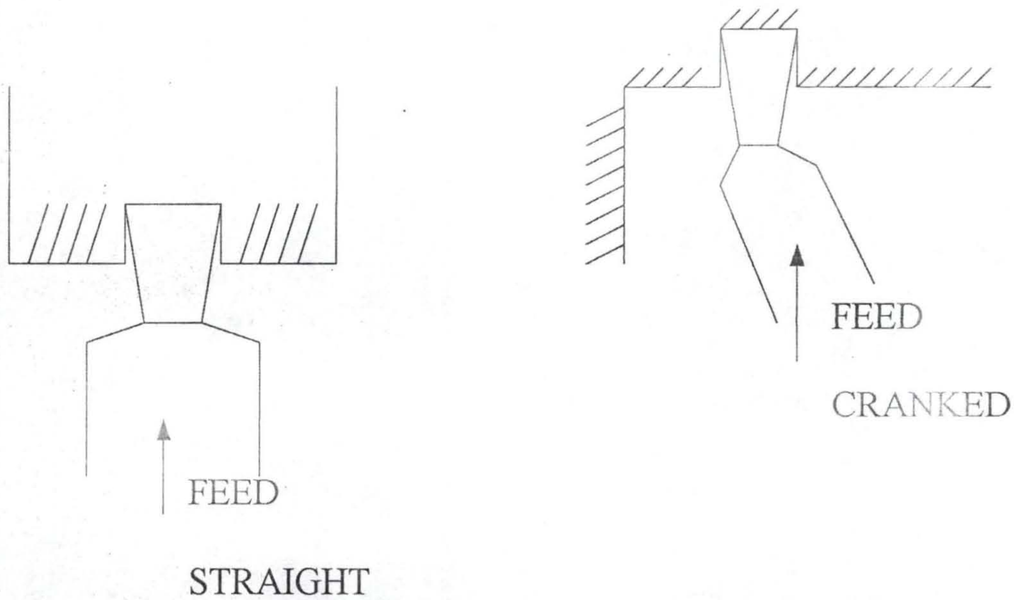


FIGURE 3.4 - PARTING TOOL

Parting or cut - off tools (fig 3.5) are designed to withstand high cutting forces, but if chips are not sufficiently cleared or cutting fluid is not used, these tools can quickly jam and break. The width of the tools will vary according to the size of work, but should be between 3mm and 8mm. Cutting - off tools are set exactly on the workcentre, clamped tightly and with the minimum of overhang.

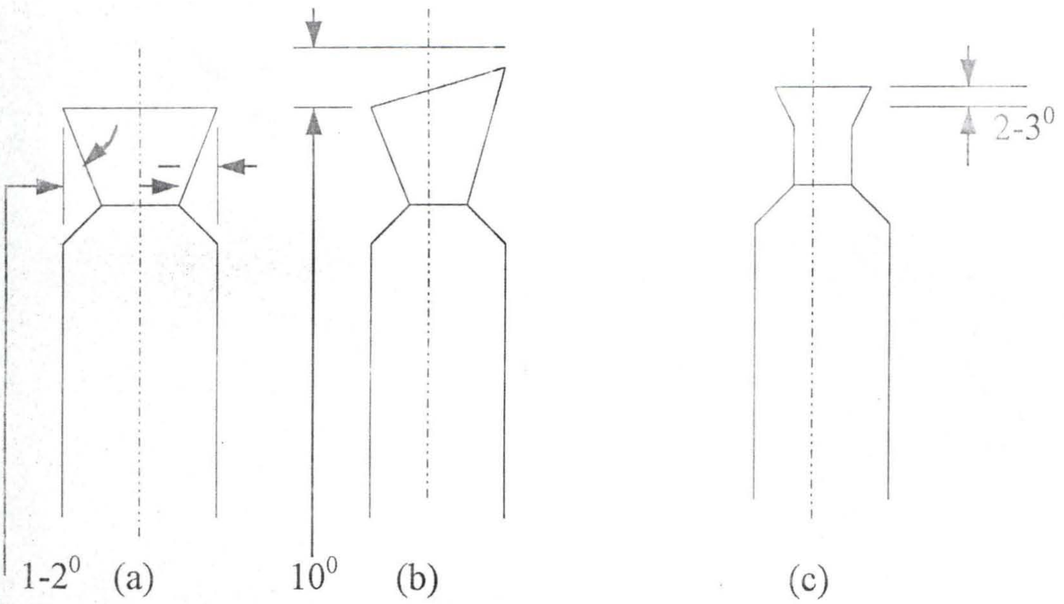


FIGURE 3.5 - STYLES OF EDGES OF CUTTING - OFF TOOLS.

Fig 3.5 (a), with this practice, the work cut breaks off before the tool reaches it's centre and thus receives a projection which has to be removed with a file or chisel. To eliminate the projection, the tool cutting edge should be ground so as to form a 10° bevel facing the chuck.

Fig 3.5 (b), with this style of cutting edge, the projection remains at the stock end and is removed by further advancing the tool. The auxiliary plan angles of cutting - off tools are 1 - 2°.

To obtain a good end surface of the cut work, the end tool edges are furnished with narrow lands 2 - 3mm long (fig 3.5 (c). The clearance of the cutting - off tool is 12°, the auxilliary clearance angles are about 2^0 . All the other angles and elements of the cutting - off tools correspond to those of the straight turning tools.

Lathe tools are often specially ground as parting tools for small or delicate parting jobs. These tools should be ground slightly wider at the cutting edge to clear the tool in the cut.

Diagonally ground parting tools leave no burr.

Cranked parting tools are used for parting - off near chuck; grooving near another face.

Straight parting tools are used in parting - off in lathe and cutting grooves.

3.7 SPEED

A low speed should be used for parting; if the tool chatters, reduce the speed. A feed that is too light can cause chatter, but a feed that is too heavy can jam the tool. The tool should always be making a chip. Hand feeding the tool is best at first, before you can later engage the automatic feed. The speed used could be one - half or two - thirds that for turning and feed in the tool slowly.

3.8 CUTTING OR PARTING - OFF OPERATION

Work should not extend very far from the chuck when parting or grooving, and no parting should be done in the middle of a workpiece or at the end near the dead centre. This is because the tool will bind in the cut when the material is almost cut through. When the workpiece is cut - off near the dead centre, the same binding problem exists withthe additional problem of the work climbing on the tool when it is cut - off. This could break the tool and possibly damage the machine.

Before parting - off, the carriage should be locked against lengthwise movement, and the compound slide set in line with the bed.

Parting or cutting off operation is illustrated in fig. 3.6 and can be undertaken through the following steps :

1. Turn the outside of the workpiece as you would if it were held between centres.

2. The cut - off tool is clamped in position.

3. Mark the location of the cut. Make sure that the blade is at right angle to the work and on centre.

4. The speed should be about two - thirds or one - half that for turning.

5. Carefully feed the cutting tool slowly into the work by hand.

When it has penetrated to about the depth of it's width, withdraw it, and move it a few thousandths sideways with the compound slide; feed in again, and after a further similar penetration into the solid metals, bring it out and move it sideways a slight amount to the other side of it's original position and so on.

Apply plenty of cutting lubricant. Sulphurized cutting oil works best for parting unless the lathe is equipped with a coolant pump and a steady flow of soluble oil is available.

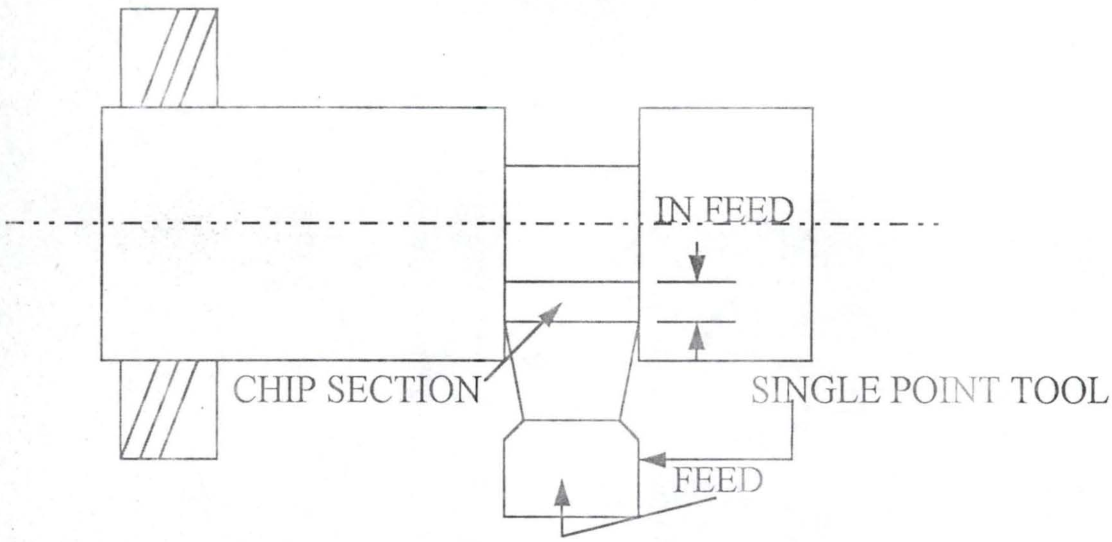


FIGURE 3.6 - PARTING OR CUTTING - OFF

Parting alloy steels and other metals is sometimes difficult, and step - parting may help in these case. When deep parting difficult materials, extend the cutting tool from the holder a short distance and part to that depth. Then back off the cross feed and extend the tool a bit further; part to that depth. Repeat the process until the centre is reached.

Often, difficulties in the operation of parting - off may be overcome by working with the tool upside down; and either locating the tool at the back of the work or running the lathe reversed. The tool should always be as near as possible to the chuck

CHAPTER FOUR: PROGRAM DESIGN FOR MACHINING OPERATIONS

When undertaking machining operations using the manual system, the talents of a highly skilled operator are poorly utilized due to the fact that performing repetitive or variety of operations will be in fragments, bringing about fatique or boredom which inturn does have adverse effect on production output.

From the analysis made above, it is very convincing now that, there is a problem at hand which need to be addressed. A system should be recommemded to take care of the existing problem inorder to solve the problems associated with using manual system in machining operations, to ensure a faster and more efficient service rate. After the recommendation and a conclusion has been reached that it is more desirable or better, the next important thing is to design a befitting system, which will be less tasking, ease of manipulation and other virtues that will give room for achievement for better results.

The choice of programming language is an important factor to be considered when developing a new system. Basic programming language has been choosen as the language to be use due to it vast feature ideal.

The features include the following :-

a. It is a simplified grammar and relatively less number of statements.

b. It is relatively easy to understand and quick to learn.

c. It is relatively easy to code.

d. It provides interactive computing.

e. It is the most commonly used programming language for mini and micro-computer system because of it's small interpreter and compiler.

4.1 MAIN MENU PROGRAM DESIGN

The main menu program incorporates the three machining operations namely Facing, Boring and Cutting or Parting-off. It is from this menu that you will choose which of the operation you would want to undertake. It has aflow chart, and shows the way in which the machining operations can be accessed.

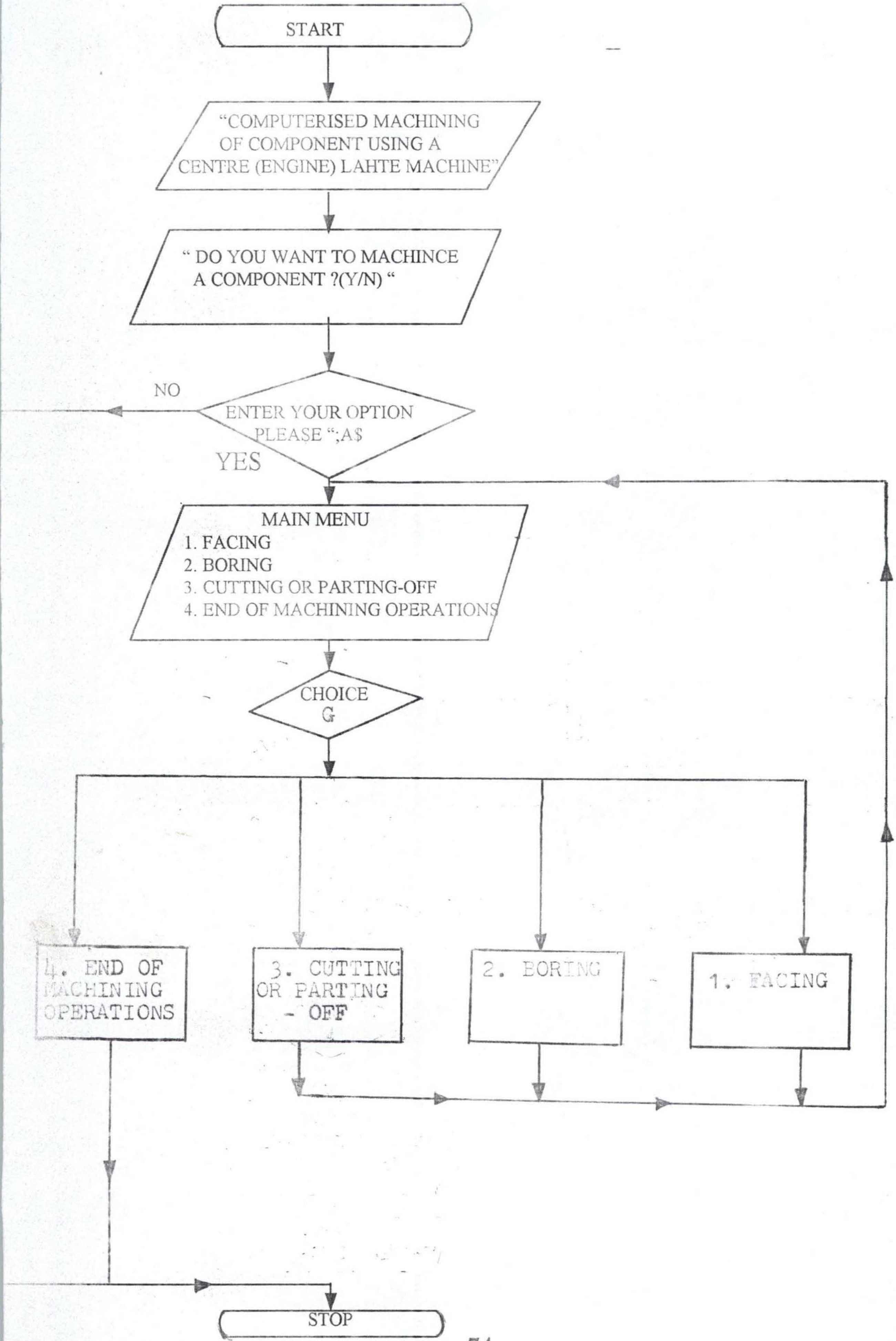


FIGURE 4.1 MAIN MENU FLOW CHART

4.2 FACING PROGRAM DESIGN

It comprises of a flow chart and set of instructions to follow when undertaking facing machining operation.

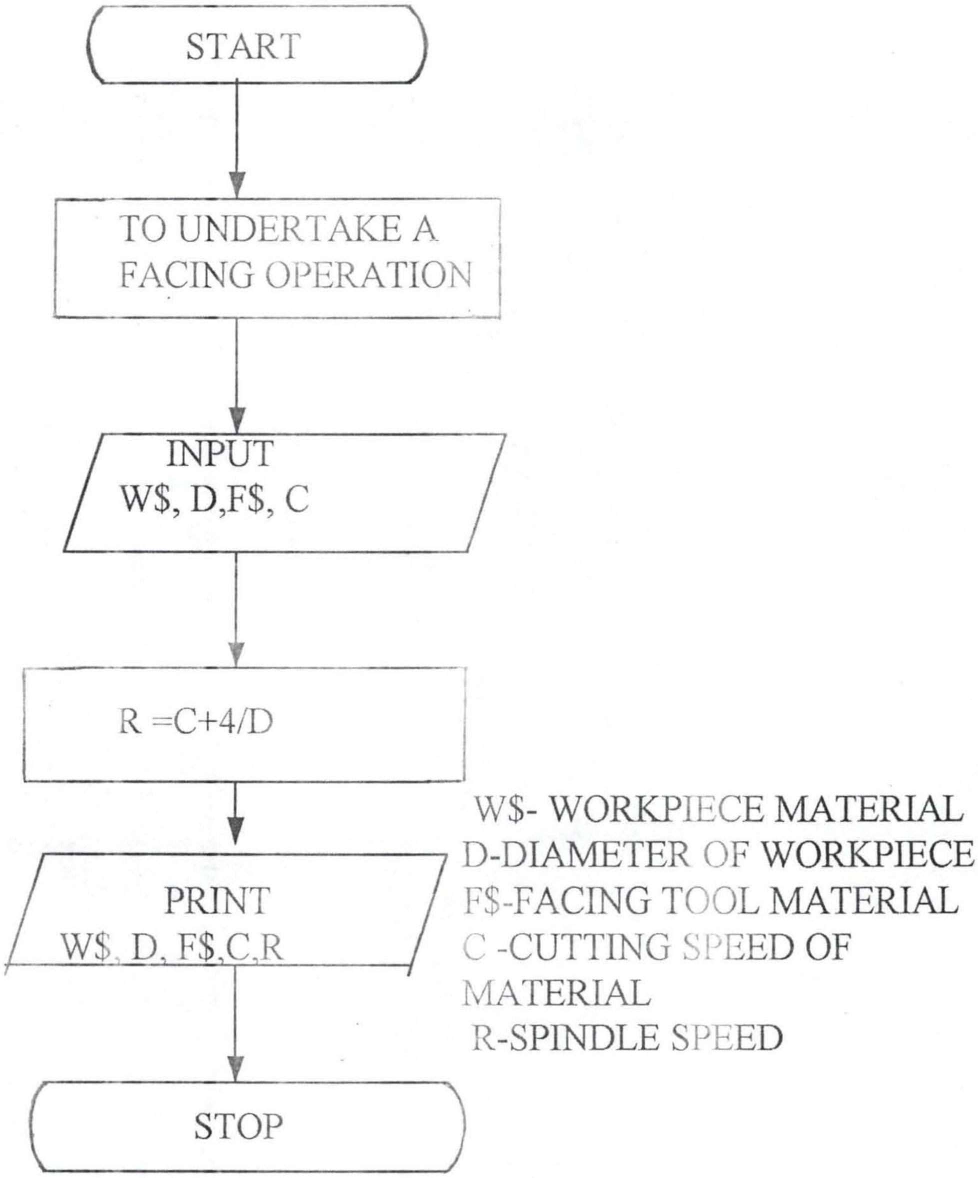


FIGURE 4.2 FACING OPERATION FLOW CHART

4.3 BORING PROGRAM DESIGN

It comprises of a flow chart and set of instructions to follow when undertaking boring machining operation

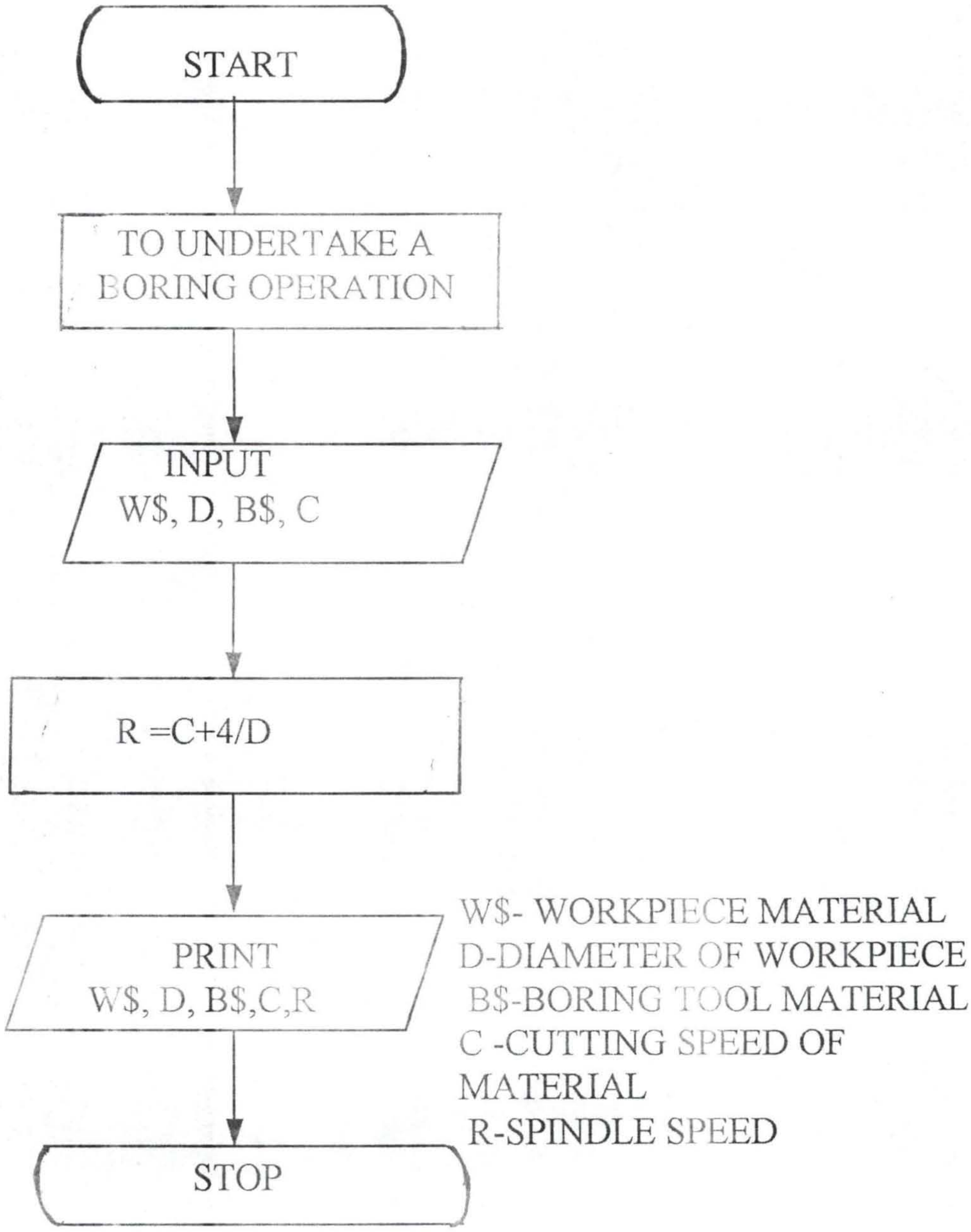


FIGURE 4.3 BORING OPERATION FLOW CHART

4.4 CUTTING OR PARTING-OFF PROGRAM DESIGN

It comprises of a flow chart and set of instructions to follow when undertaking cutting or parting-off machining operation.

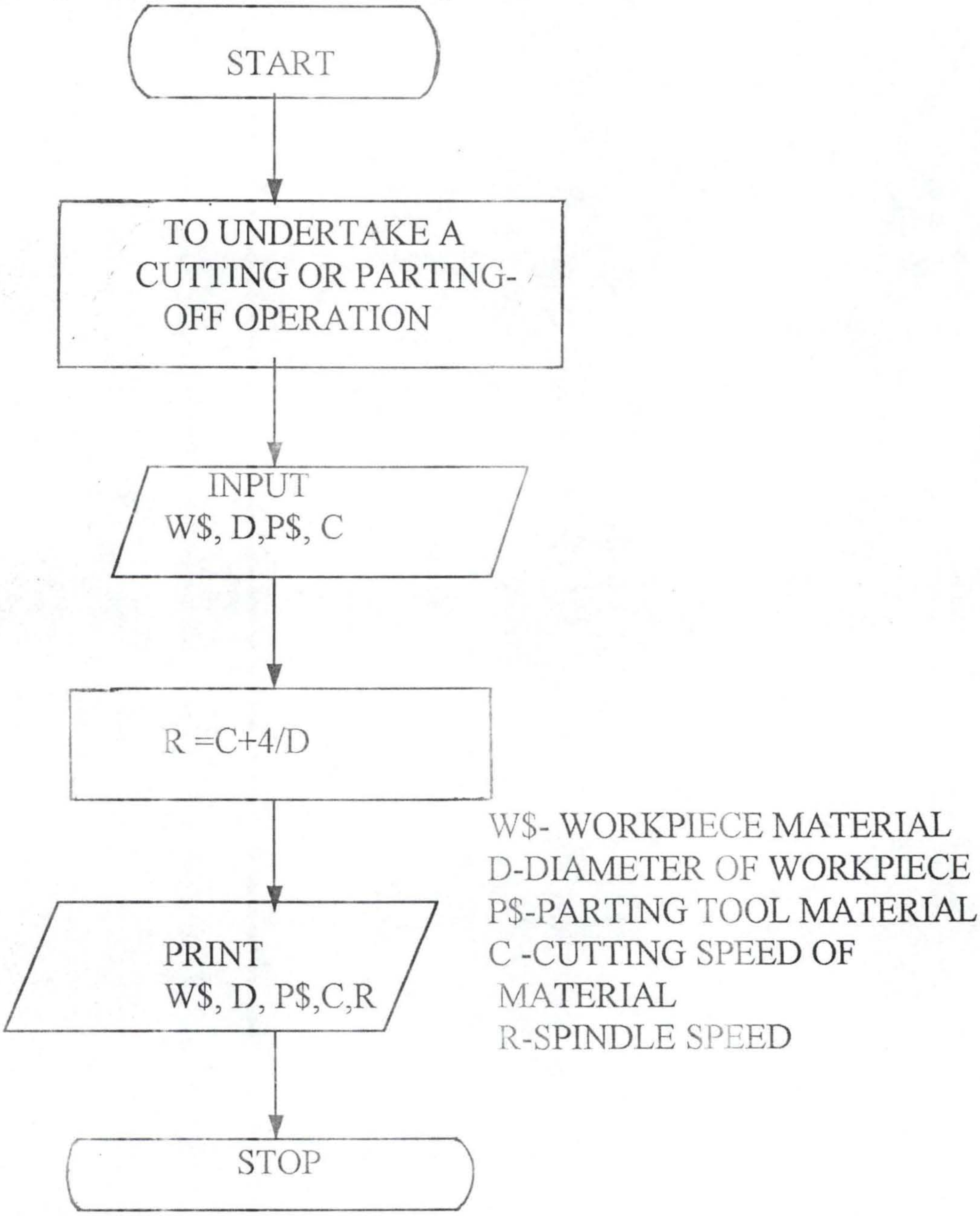


FIGURE 4.4 CUTTING OR PARTING-OFF FLOW CHART

CHAPTER FIVE: PROGRAM TESTING AND CONCLUSION

The basic programming language is used, which is contained in the diskette. The program file name used is SULE.

After booting the system, at the dot prompt you will see C:\> which implies that the computer is currently in drive C, that is the hard disk drive.

To change from the hard disk drive to drive A where the program file is, follow the steps below :

a. To change to drive A - Type A: at the dot prompt.

i.e **C:\>A :** ↵ (↵ **-Return key**)

A:\> will appear at the dot prompt which implies that the computer is in drive A.

b. To change directory to Basic

Type **CD BASIC** at the dot prompt

i.e **A:\>CD BASIC** ↵

A:\>BASIC> will appear at dot prompt which implies that the computer is in basic directory.

At the dot prompt type TB SULE

i.e **A:\> BASIC>**TB SULE ↵

5.1 MAIN MENU PROGRAM TESTING

The main menu program consists of facing, boring and cutting or parting-off machining operations and it is from this menu that the whole system is operated.

The first four statements or commands at the begining of the program are remarks used to provide information for you or any one else reading your program ; they provide no information to the computer.

PRINT Statement is an executable command, it displays program output on the screen and it also provides certain ways of controlling the alignment and spacing of print-outs in the terminal.

INPUT Statement allow the user to prepare the program for input from the keyboard during program execution which makes it suitable for interactive use.

During program execution when the computer gets to statement 60, it will display " DO YOU WANT TO MACHINE A COMPONENT ? (Y/N)" on the screen. It will then proceed to statement 80 to display " ENTER YOUR OPTION PLEASE"; A$ and wait for the option to be keyed in before continuing with program execution. If the option is "Y" it will proceed to statement 100 otherwise it goes to statement 90 and terminate the program.

If the option is "Y" the screen will be cleared (CLS) and the contents of statements 110 through 210 will be displayed on the screen. when a choice is selected the computer goes to statement 230 and carryout the machining operation in that particular subroutine.

Statement 210, the computer will only go to statement 230 if the choice entered is not greater than 4. If it is greater when the computer gets to statement 220, it will clear screen (CLS), an error message BEEP.BEEP and "WRONG CHOICE PLEASE TRY AGAIN" will be displayed on the screen. The computer will encounter GOTO 110 statement, which implies that it should go back to statement 110 and start program execution to statement 210 where it will wait for the correct choice to be made before proceeding.

GOSUB statement is used to branch to and return from a subroutine. Statement 230, those numbers after GOSUB are the subroutine numbers for machining operations. Statement 235 is a conditional transfer staetement that instructs the computer to go some where If (but only if) a certain condition is met . As such if the choice selected is 4 it proceeds to statement 240, clears screen and display "END OF MACHINING OPERATIONS" on the screen otherwise it goes back to statement 110 to undertake another machining operation. After displaying statement 240 it proceeds to statement 250 to display " END OF PROGRAM" on the screen.

Statement 260, the last line in your program should have the key word END. This helps the computer by indicating the physical end of the material you are giving it. Moreover, if computer reaches this line when running the program it will stop.

5.2 FACING PROGRAM TESTING

If the choice 1 is selected from the main menu program, the computer leaves statement 230 and proceeds to statement 300 to commence facing mahining operation.

Statement 305 tells the type of operation to be undertaken in this particular subrountine. Statements 325, 360, 415 & 425 will be displayed on the screen and the computer will continue with program execution on each line only if the required information have been keyed in by the user.

LET is an assignment statement , it assigns the value of an expression to a variable and it is optional in basic programming .The statement LET R = C+4/D instructs the computer to evaluate the expression on the right-hand side of the equal sign and then insert that value in the mail box (variable) indicated on the left-hand side of the equal sign.

Statements 455 through 495 displays those data needed for undertaking facing operation. Statement 510 tells the next step to be taken after getting the required informations for the operation and statement 525 tells how the cutting tool is used to undertake the operation. When the computer encounters statement 530 it will automatically go to the statement following the GOSUB from which it transferred, and continue program execution.

5.3 BORING PRAGRAM TESTING

If choice 2 is selected from the main menu program, the computer leaves statement 230 and proceeds to statement 540 to commence boring machining operation.

Statement 545 tells the type of operation to be undertaken in this particular subroutine. Statement 565, 600, 655 and 665 will be displayed on the screen and the computer will continue with program exection on each line only if the required information have been keyed in by the user.

LET is an assignment statement, it assigns the value of an expression to a variable and it is optional in basic programming. The statement LET R = C + 4/D instructs the computer to evaluate the expression on the right -hand side of the equal sign and then insert that value in the mail box (variable) indicated on the left-hand side of the equal sign.

Statements 695 through 735 displays those data needed for undertaking boring operation. Statement 750 tells the next step to be taken after getting the required informations for the operation and statement 765 tells how the cutting tool is used to undertake the operation. When the computer encounters statement 770 it will automatically go to the statement following the GOSUB from which it transferred, and continue program execution.

5.4 CUTTING OR PARTING-OFF PROGRAM TESTING

If choice 3 is selected from the main menu program, the Computer leaves statement 230 and proceeds to statement 780 to commence cutting or parting-off machining operation.

Statement 785 tells the type of operation to be undertaken in this particular subroutine. Statements 805, 840, 895 and 905 will be displayed on the screen and the computer will continue with program execution on each line only if the required information have been keyed in by the user.

LET is an assignment, it assigns the value of an expression to a variable and it is optional in basic programming. The statement LET R = C + 4 / D instructs the computer to evaluate the expression on the right-hand side of the equal sign and then insert that value in the mail box (variable) indicated on the left-hand side of the equal sign.

Statements 935 through 975 displays those data needed for undertaking cutting or parting-off operation. Statement 990 tells the next step to be taken after getting the required informations for the operation and statement 997 tells how the cutting tool is used to undertake the operation. When the computer encounters statement 999 it will automatically go to the statement following the GOSUB from which it transferred, and continue program execution.

5.5 CONCLUSION

The main aim of this project work is the computerisation of the steps taken when undertaking machining operations using a centre (engine) lathe machine.

When using the manual system in undertaking machining operations, the operators job use to be in fragments bringing about boredom or fatique, which in-turn has adverse effect on production output.

This system is designed to ease some of the difficulties encountered by the operator when using the manual system. Some of the advantages that can be achieved when using this system includes - the technological and economic advantages in increasing the machine utilization and the speed of programming. It is a system which accepts and uses the skills of the machinists, and leaves room for it to develop, it doesn't fragment the operator's job and also it ensures a faster and more efficient service rate.

The aim is not simply to preserve the operator's existing skills. It is rather to make those skills more productive and to allow it to develop in a natural way into a new skill - the skill of using a more highly developed system.

REFERENCE TEXT BOOKS

1) Amstead, B.H; Begeman, M. L., (1979): Manufacturing processes; Canada: John Wiley and Sons, Incorporation, Seventh edition.

2) Chapman, W.A.J, (1981): Workshop Technology Part 1, 2 and 3; London: Edward Arnold (Publishers) Ltd., Third edition.

(3) Chernov, N; (1989): Machine tools; Moscow: MIR Publishers, Second edition.

(4) Fomin, S; (1985): Handbook for Lathe operators and foremen; Moscow: MIR Publishers, Fifth edition.

(5) Kibbe, R.R; Meyer, R.O; ; (1979): Machine tool practices; New Jersey, Fourth edition.

(6) The Editors of American Machinist; (1982): Tools of our (Metal working) trade; New York: Mc Graw - Hill Publications Company.

```
     A:SULE.BAS     Line 1     Col 1   Insert Indent Tab
CLS : COLOR 15, 4
10 REM "COMPUTERISED MACHINING OF COMPONENTS USING A CENTRE (ENGINE) LATHE MACH
20 REM **************** " PROGRAMMER MUHAMMAD SULAIMAN LIMAN " **************
30 REM *********************** " PGD/MCS/93/94 " *****************************
40 REM ;  ER = 0
50 PRINT
60 PRINT "COMPUTERISED MACHINING OF COMPONENTS USING A CENTRE (ENGINE) LATHE MA
70 LOCATE 12, 15:   PRINT " DO YOU WANT TO MACHINE A COMPONENT ?(Y/N)"
80 LOCATE 23, 25: COLOR 15, 3:    :  INPUT "   ENTER YOUR OPTION PLEASE  "; A$
CLS
90 IF A$ = "Y" OR A$ = "y" THEN 110 ELSE 100
100 IF A$ = "N" OR A$ = "n" THEN END
CLS
110  CLS:LOCATE 8, 35: COLOR 15, 6: PRINT "  MAIN  MENU  "
120  LOCATE 9, 15: COLOR 15, 3: PRINT
130  LOCATE 11, 15: PRINT "         1.  FACING                        "
140  LOCATE 12, 15: PRINT
150  LOCATE 13, 15: PRINT "         2.  BORING                        "
160  LOCATE 14, 15: PRINT
170  LOCATE 15, 15: PRINT "         3.  CUTTING  OR  PARTING-OFF      "
180  LOCATE 16, 15: PRINT
190  LOCATE 17, 15: PRINT "         4.  END OF MACHINING  OPERATION "
200  PRINT
210   LOCATE 23, 35: COLOR 15, 6: INPUT " ENTER  CHOICE "; G

     A:SULE.BAS     Line 47    Col 1   Insert Indent Tab
220 IF G > 4 THEN
     CLS
     COLOR 4, 7
     BEEP
     BEEP
     PRINT "   WRONG CHOICE PLEASE TRY AGAIN "
     GOTO 110
ELSE
230        ON G GOSUB 300, 540, 780, 1000
240  GOTO 110
END IF
260        END
300  CLS : COLOR 15, 3
REM    *********    SUBROUTINE FOR FACING COMPONENTS    ***********
305 LOCATE 5, 15: PRINT "   TO UNDER TAKE A FACING  OPERATION          "
310 LOCATE 6, 15: PRINT "                                              "
315 LOCATE 7, 15: PRINT "   MOUNT THE WORKPIECE IN THE CHUCK           "
320 LOCATE 8, 15: PRINT "                                              "
325 LOCATE 9, 15: COLOR 15, 6: INPUT "   ENTER WORKPIECE MATERIAL        "; W$
CLS : COLOR 15, 3
330 PRINT
335 PRINT
340 LOCATE 10, 15: PRINT "   CLAMP THE FACING  TOOL IN THE TOOL HOLDER      "
345 LOCATE 11, 15: PRINT "                                                 "

     A:SULE.BAS     Line 68    Col 1   Insert Indent Tab
335 PRINT
340 LOCATE 10, 15: PRINT "   CLAMP THE FACING  TOOL IN THE TOOL HOLDER      "
345 LOCATE 11, 15: PRINT "                                                 "
350 PRINT
355 PRINT
360 LOCATE 14, 15: COLOR 15, 6: INPUT "  ENTER  FACING  TOOL  MATERIAL  "; F$
CLS : COLOR 15, 3
365 PRINT
370 PRINT
375 LOCATE 8, 15: PRINT "   MARK THE LOCATION OF THE CUT MAKING SURE    "
380 LOCATE 9, 15: PRINT "   THAT THE BLADE IS  AT RIGHT ANGLE TO THE    "
385 LOCATE 10, 15: PRINT "   WORKPIECE  AND ON CENTRE.                  "
```

```
390 PRINT
395 PRINT
400 LOCATE 15, 15: PRINT "    ********* TO CALCULATE THE SPINDLE SPEED ******
405 PRINT
410 PRINT
415 LOCATE 18, 15: COLOR 15, 6: INPUT "  ENTER DIAMETER OF WORKPIECE MATERIAL
420 PRINT
425 LOCATE 20, 15: INPUT "  ENTER CUTTING SPEED OF MATERIAL     "; C
LET R = C + 4 / D
REM    **********   OUTPUT OF RESULT     ********************************
CLS : COLOR 15, 3
430 PRINT

    A:SULE.BAS     Line 92    Col 1   Insert Indent Tab
435 PRINT
440 LOCATE 4, 20: PRINT "  OUTPUT  OF  RESULTS  "
445 LOCATE 5, 20: PRINT "  ===================  "
450 PRINT
455 LOCATE 7, 10: PRINT "  WORKPIECE  MATERIAL :        "; W$
460 PRINT
465 LOCATE 9, 10: PRINT "  DIAMETER  OF WORKPIECE :    "; D
470 PRINT
475 LOCATE 11, 10: PRINT " FACING  TOOL  MATERIAL  :   "; F$
480 PRINT
485 LOCATE 13, 10: PRINT " CUTTING SPEED OF MATERIAL : "; C
490 PRINT
495 LOCATE 15, 10: PRINT " SPINDLE    SPEED   :              ";
498 LOCATE 15,35:PRINT USING "#############.####";R
500 PRINT
505 PRINT
510 LOCATE 18, 5: PRINT "  START  THE  MACHINE  AND  USE  THE  CALCULATED
515 LOCATE 19, 5: PRINT "  SPINDLE  SPEED  FOR  THE  OPERATION .
520 PRINT
525 LOCATE 21, 5: PRINT "  FEED THE  CUTTING  TOOL  SLOWLY  INTO THE  WORKPIECE
526 LOCATE 24,10: COLOR 0: INPUT"PRESS ENTER KEY TO EXIT TO MAIN MENU",E$:COLOR
530 RETURN
REM ***********************END OF  SUBROUTINE 1********************************
REM *************************************************************************

    A:SULE.BAS     Line 116   Col 1   Insert Indent Tab
REM *************************************************************************
540 CLS : COLOR 15, 3
REM *************************************************************************
REM ************** " SUBROUTINE FOR BORING COMPONENTS " **********************
545 LOCATE 5, 15: PRINT " TO UNDERTAKE A BORING OPERATION "
550 LOCATE 6, 15: PRINT "                                 "
555 LOCATE 7, 15: PRINT " MOUNT THE WORKPIECE IN THE CHUCK "
560 LOCATE 8, 15: PRINT "                                  "
565 LOCATE 9, 15: COLOR 15, 6: INPUT " ENTER WORKPIECE MATERIAL    "; W$
CLS : COLOR 15, 3
570 PRINT
575 PRINT
580 LOCATE 10, 15: PRINT " CLAMP THE BORING TOOL IN THE TOOL HOLDER "
585 LOCATE 11, 15: PRINT "                                          "
590 PRINT
595 PRINT
600 LOCATE 14, 15: COLOR 15, 6: INPUT " ENTER BORING TOOL MATERIAL "; B$
CLS : COLOR 15, 3
605 PRINT
610 PRINT
615 LOCATE 8, 15: PRINT " MARK THE LOCATION OF THE CUT MAKING SURE    "
620 LOCATE 9, 15: PRINT " THAT THE BLADE AT RIGHT ANGLE               "
625 LOCATE 10, 15: PRINT " TO THE WORKPIECE AND ON CENTRE.            "
630 PRINT
```

```
A:SULE.BAS     Line 140   Col 1   Insert Indent Tab
635 PRINT
640 LOCATE 15, 15: PRINT " ****** TO CALCULATE THE SPINDLE SPEED ****** "
645 PRINT
650 PRINT
655 LOCATE 18, 15: COLOR 15, 6: INPUT " ENTER DIAMETER OF WORKPIECE MATERIAL ";
660 PRINT
665 LOCATE 20, 15: INPUT " ENTER CUTTING SPEED OF MATERIAL                    ";
LET R = C + 4 / D
REM ********************* OUTPUT OF RESULT ***********************************
CLS : COLOR 15, 3
670 PRINT
675 PRINT
680 LOCATE 4, 20: PRINT " OUTPUT OF RESULTS "
685 LOCATE 5, 20: PRINT " ================= "
690 PRINT
695 LOCATE 7, 10: PRINT "  WORKPIECE MATERIAL :                   "; W$
700 PRINT
705 LOCATE 9, 10: PRINT "  DIAMETER OF WORKPIECE:                 "; D
710 PRINT
715 LOCATE 11, 10: PRINT " BORING TOOL MATERIAL:                  "; B$
720 PRINT
725 LOCATE 13, 10: PRINT " CUTTING SPEED OF MATERIAL:             "; C
730 PRINT
735 LOCATE 15, 10: PRINT " SPINDLE SPEED:                         ";

A:SULE.BAS     Line 161   Col 1   Insert Indent Tab
725 LOCATE 13, 10: PRINT " CUTTING SPEED OF MATERIAL:             "; C
730 PRINT
735 LOCATE 15, 10: PRINT " SPINDLE SPEED:                         ";
738 LOCATE 15,35:PRINT USING "###########.####";R
740 PRINT
745 PRINT
750 LOCATE 18, 5: PRINT " START THE MACHINE AND USE THE CALCULATED "
755 LOCATE 19, 5: PRINT " SPINDLE SPEED FOR THE OPERATION.         "
760 PRINT
765 LOCATE 21, 5: PRINT " FEED THE CUTTING TOOL SLOWLY INTO THE WORKPIECE "
768 LOCATE 24,10:COLOR 0:INPUT"PRESS ENTER KEY TO EXIT TO MAIN MENU", E$ :COLOR
770 RETURN
REM ************************** END OF SUBROUTINE 2 *****************************
REM ***************************************************************************
REM ***************************************************************************


780 CLS : COLOR 15, 3
REM ********* " SUBROUTINE FOR CUTTING OR PARTING-OFF COMPONENTS " ************
785 LOCATE 5, 15: PRINT " TO UNDERTAKE A CUTTING OR PARTING-OFF OPERATION "
790 LOCATE 6, 15: PRINT "                                                 "
795 LOCATE 7, 15: PRINT " MOUNT THE WORKPIECE IN THE CHUCK                "
800 LOCATE 8, 15: PRINT "                                                 "

A:SULE.BAS     Line 185   Col 1   Insert Indent Tab
805 LOCATE 9, 15: COLOR 15, 6: INPUT " ENTER WORKPIECE MATERIAL       "; W$
CLS : COLOR 15, 3
810 PRINT
815 PRINT
820 LOCATE 10, 15: PRINT " CLAMP THE PARTING TOOL IN THE TOOL HOLDER  "
825 LOCATE 11, 15: PRINT "                                            "
830 PRINT
835 PRINT
840 LOCATE 14, 15: COLOR 15, 6: INPUT " ENTER PARTING TOOL MATERIAL   "; P$
CLS : COLOR 15, 3
845 PRINT
850 PRINT
```

```
855 LOCATE 8, 15: PRINT " MARK THE LOCATION OF CUT MAKING SURE THAT "
860 LOCATE 9, 15: PRINT " THE BLADE IS AT RIGHT ANGLE TO THE         "
865 LOCATE 10, 15: PRINT " WORKPIECE AND ON CENTRE.                  "
870 PRINT
875 PRINT
880 LOCATE 15, 15: PRINT " ******* TO CALCULATE THE SPINDLE SPEED *********** "
885 PRINT
890 PRINT
895 LOCATE 18, 15: COLOR 15, 6: INPUT " ENTER DIAMETER OF WORKPIECE MATERIAL ";
900 PRINT
905 LOCATE 20, 15: INPUT " ENTER CUTTING SPEED OF MATERIAL "; C
LET R = C + 4 / D

    A:SULE.BAS     Line 208   Col 1   Insert Indent Tab
LET R = C + 4 / D
REM ******************** OUTPUT OF RESULT **********************************
CLS : COLOR 15, 3
910 PRINT
915 PRINT
920 LOCATE 4, 20: PRINT " OUTPUT OF RESULTS "
925 LOCATE 5, 20: PRINT " ================= "
930 PRINT
935 LOCATE 7, 10:  PRINT " WORKPIECE MATERIAL :        "; W$
940 PRINT
945 LOCATE 9, 10:  PRINT " DIAMETER OF WORKPIECE :    "; D
950 PRINT
955 LOCATE 11, 10: PRINT " PARTING TOOL MATERIAL :    "; P$
960 PRINT
965 LOCATE 13, 10: PRINT " CUTTING SPEED OF MATERIAL: "; C
970 PRINT
975 LOCATE 15, 10: PRINT " SPINDLE SPEED :                  ";
978 LOCATE 15,35:PRINT USING "#############.####";R
980 PRINT
985 PRINT
990 LOCATE 18, 5: PRINT " START THE MACHINE AND USE THE CALCULATED "
995 LOCATE 19, 5: PRINT " SPINDLE SPEED FOR THE OPERATION.         "
996 PRINT
997 LOCATE 21, 5: PRINT " FEED THE CUTTING TOOL SLOWLY INTO THE WORKPIECE. "

    A:SULE.BAS     Line 226   Col 1   Insert Indent Tab
980 PRINT
985 PRINT
990 LOCATE 18, 5: PRINT " START THE MACHINE AND USE THE CALCULATED "
995 LOCATE 19, 5: PRINT " SPINDLE SPEED FOR THE OPERATION.         "
996 PRINT
997 LOCATE 21, 5: PRINT " FEED THE CUTTING TOOL SLOWLY INTO THE WORKPIECE. "
998 LOCATE 24,10:COLOR 0:INPUT"PRESS ENTER KEY TO EXIT TO MAIN MENU",E$:COLOR 1
999 RETURN
REM ************************ END OF SUBROUTINE 3 ***************************
REM ************************************************************************
REM ************************************************************************
1000 CLS : LOCATE 20, 25: PRINT " END OF MACHINING OPERATIONS "
1005 PRINT
1007 LOCATE 24, 25: PRINT "  END OF PROGRAM  "
1008   END
RETURN
```

COMPUTERISED MACHINING OF COMPONENTS USING A CENTRE (ENGINE) LATHE MACHINE

DO YOU WANT TO MACHINE A COMPONENT ?(Y/N)

ENTER YOUR OPTION PLEASE ? y

MAIN MENU

1. FACING
2. BORING
3. CUTTING OR PARTING-OFF
4. END OF MACHINING OPERATION

ENTER CHOICE ? 1

TO UNDER TAKE A FACING OPERATION

MOUNT THE WORKPIECE IN THE CHUCK

ENTER WORKPIECE MATERIAL ? HARD BRASS

CLAMP THE FACING TOOL IN THE TOOL HOLDER

ENTER FACING TOOL MATERIAL   ? TUNGSTEN CARBIDE

MARK THE LOCATION OF THE CUT MAKING SURE
THAT THE BLADE IS AT RIGHT ANGLE TO THE
WORKPIECE AND ON CENTRE.

********* TO CALCULATE THE SPINDLE SPEED **********

ENTER DIAMETER OF WORKPIECE MATERIAL? 20

MARK THE LOCATION OF THE CUT MAKING SURE
THAT THE BLADE IS AT RIGHT ANGLE TO THE
WORKPIECE AND ON CENTRE.

********* TO CALCULATE THE SPINDLE SPEED **********

ENTER DIAMETER OF WORKPIECE MATERIAL? 20

ENTER CUTTING SPEED OF MATERIAL ? 80

OUTPUT OF RESULTS
===================

WORKPIECE MATERIAL : HARD BRASS

DIAMETER OF WORKPIECE : 20

FACING TOOL MATERIAL : TUNGSTEN CARBIDE

CUTTING SPEED OF MATERIAL : 80

SPINDLE SPEED : 80.2000

START THE MACHINE AND USE THE CALCULATED
SPINDLE SPEED FOR THE OPERATION .

FEED THE CUTTING TOOL SLOWLY INTO THE WORKPIECE

PRESS ENTER KEY TO EXIT TO MAIN MENU

MAIN MENU

1. FACING

2. BORING

3. CUTTING OR PARTING-OFF

4. END OF MACHINING OPERATION

ENTER CHOICE ? 2

TO UNDERTAKE A BORING OPERATION

MOUNT THE WORKPIECE IN THE CHUCK

ENTER WORKPIECE MATERIAL ? ALUMINIUM

CLAMP THE BORING TOOL IN THE TOOL HOLDER

ENTER BORING TOOL MATERIAL ? HIGH SPEED STEEL

CLAMP THE BORING TOOL IN THE TOOL HOLDER

ENTER BORING TOOL MATERIAL ? HIGH SPEED STEEL

MARK THE LOCATION OF THE CUT MAKING SURE
THAT THE BLADE AT RIGHT ANGLE
TO THE WORKPIECE AND ON CENTRE.

****** TO CALCULATE THE SPINDLE SPEED ******

ENTER DIAMETER OF WORKPIECE MATERIAL ? 60

MARK THE LOCATION OF THE CUT MAKING SURE
THAT THE BLADE AT RIGHT ANGLE
TO THE WORKPIECE AND ON CENTRE.

****** TO CALCULATE THE SPINDLE SPEED ******

ENTER DIAMETER OF WORKPIECE MATERIAL ? 60

ENTER CUTTING SPEED OF MATERIAL ? 100

OUTPUT OF RESULTS
=================

WORKPIECE MATERIAL :	ALUMINIUM
DIAMETER OF WORKPIECE:	60
BORING TOOL MATERIAL:	HIGH SPEED STEEL
CUTTING SPEED OF MATERIAL:	100
SPINDLE SPEED:	100.0667

START THE MACHINE AND USE THE CALCULATED
SPINDLE SPEED FOR THE OPERATION.

FEED THE CUTTING TOOL SLOWLY INTO THE WORKPIECE

PRESS ENTER KEY TO EXIT TO MAIN MENU

MAIN MENU

1. FACING

2. BORING

3. CUTTING OR PARTING-OFF

4. END OF MACHINING OPERATION

ENTER CHOICE ? 3

TO UNDERTAKE A CUTTING OR PARTING-OFF OPERATION

MOUNT THE WORKPIECE IN THE CHUCK

ENTER WORKPIECE MATERIAL ? MILD STEEL

CLAMP THE PARTING TOOL IN THE TOOL HOLDER

ENTER PARTING TOOL MATERIAL ? TUNGSTEN CARBIDE

CLAMP THE PARTING TOOL IN THE TOOL HOLDER

ENTER PARTING TOOL MATERIAL ? TUNGSTEN CARBIDE

MARK THE LOCATION OF CUT MAKING SURE THAT
THE BLADE IS AT RIGHT ANGLE TO THE
WORKPIECE AND ON CENTRE.

******* TO CALCULATE THE SPINDLE SPEED ***********

ENTER DIAMETER OF WORKPIECE MATERIAL ? 80

ENTER CUTTING SPEED OF MATERIAL ?

MARK THE LOCATION OF CUT MAKING SURE THAT
THE BLADE IS AT RIGHT ANGLE TO THE
WORKPIECE AND ON CENTRE.

******* TO CALCULATE THE SPINDLE SPEED ***********

ENTER DIAMETER OF WORKPIECE MATERIAL ? 80

ENTER CUTTING SPEED OF MATERIAL ? 250

OUTPUT OF RESULTS
================

WORKPIECE MATERIAL : MILD STEEL

DIAMETER OF WORKPIECE : 80

PARTING TOOL MATERIAL : TUNGSTEN CARBIDE

CUTTING SPEED OF MATERIAL: 250

SPINDLE SPEED : 250.0500

ART THE MACHINE AND USE THE CALCULATED
INDLE SPEED FOR THE OPERATION.

ED THE CUTTING TOOL SLOWLY INTO THE WORKPIECE.

PRESS ENTER KEY TO EXIT TO MAIN MENU

MAIN MENU

1. FACING

2. BORING

3. CUTTING OR PARTING-OFF

4. END OF MACHINING OPERATION

ENTER CHOICE ? 4

END OF MACHINING OPERATIONS

END OF PROGRAM