Global Journal of Researches in Engineering: A
Mechanical and Mechanics Engineering
Volume 14 Issue 6 Version 1.0 Year 2014
Type: Double Blind Peer Reviewed International Research Journal
Publisher: Global Journals Inc. (USA)
Online ISSN: 2249-4596 & Print ISSN: 0975-5861

Importance of Lathe Machine in Engineering Field and its usage

By Jahnavi Madireddy
MLR Institute of Technology, India

Introduction- The lathe, probably one of the earliest machine tools, is one of the most versatile and widely used machine tool, so also known as mother machine tool.

An engine lathe is the most basic and simplest form of the lathe. It is called so because in early lathes, power was obtained from engines.

The job to be machined is held and rotated in a lathe chuck; a cutting tool is advanced which is stationary against the rotating job. Since the cutting tool material is harder than the work piece, so metal is easily removed from the job.

Some of the common operations performed on a lathe are facing, turning, drilling, threading, knurling, and boring etc.

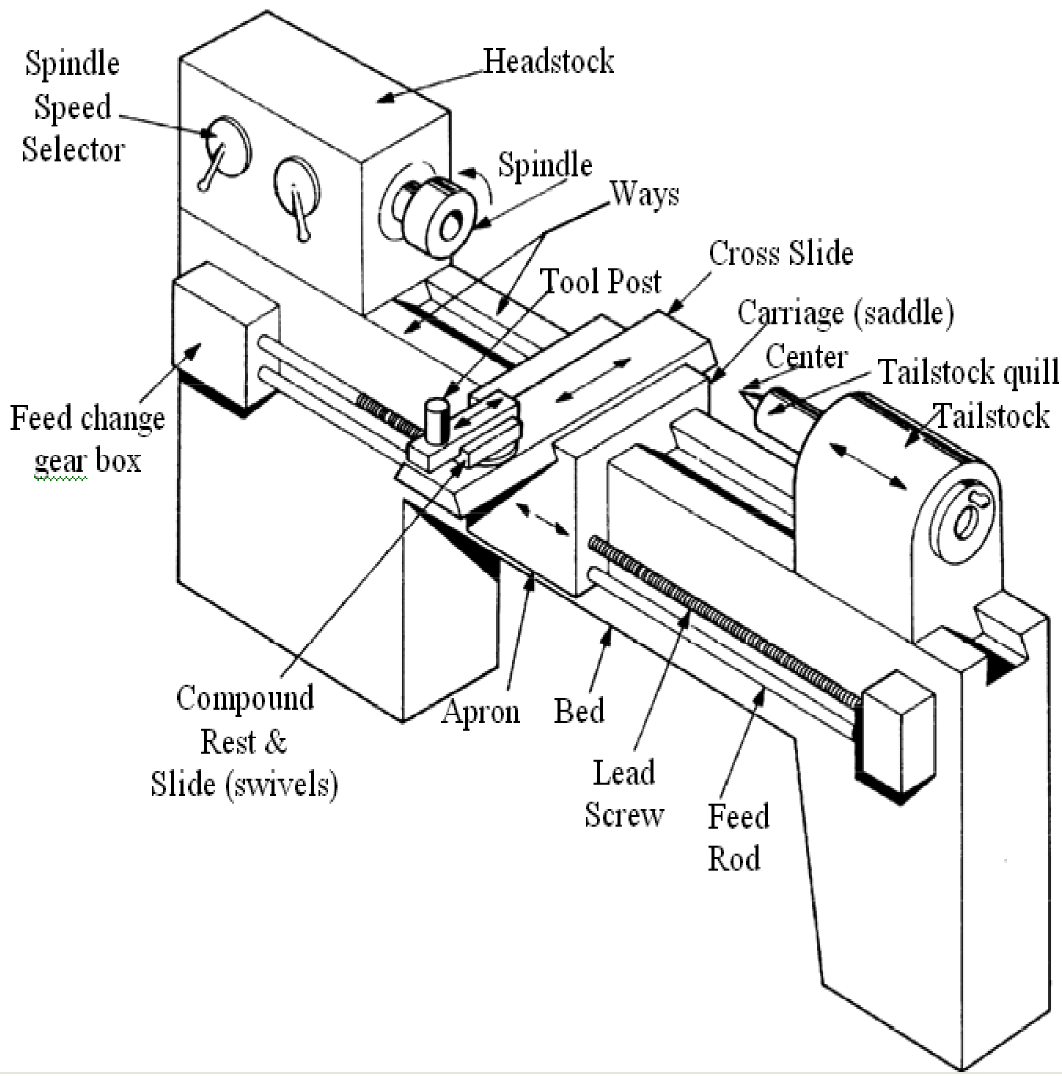
GJRE-A Classification : FOR Code: 290399p, 290501p

IMPORTANCEOFLATHEMACHINEINENGINEERINGFIELDANDITSUSAGE

Importance of Lathe Machine in Engineering Field and its usage

Jahnavi Madireddy

Parts of Lathe Machine



I. Introduction

The lathe, probably one of the earliest machine tools, is one of the most versatile and widely used machine tool, so also known as mother machine tool.

An engine lathe is the most basic and simplest form of the lathe. It is called so because in early lathes, power was obtained from engines.

The job to be machined is held and rotated in a lathe chuck; a cutting tool is advanced which is stationary against the rotating job. Since the cutting tool material is harder than the work piece, so metal is easily removed from the job.

Some of the common operations performed on a lathe are facing, turning, drilling, threading, knurling, and boring etc.

Author: *Shaili Gardenia, Yapral, J. J. Nagar Post, Telangana, MLR Institute of Technology, India. e-mail: mjahnavi216@gmail.com*

II. Lathe Machine Parts

Bed: Supports all other machine parts.

Carriage: Slides along the machine ways.

Head stock: Power train of system (spindle included).

Tail Stock: Fixes piece at end opposite to the head stock.

Swing: Maximum diameter of the machinable piece.

Lead screw: Controls the feed per revolution with a great deal of precision.

Lathe Tools: Left handed, Right handed, Threading, Boring, Groove, Parting (Cut-Off)

a) Cutting Speeds

Typical Lathe Cutting Speeds:

Nominal cuts

-30 - 800 ft./min.

Roughing cuts

- Depth of cut greater then .02 in
- Feed speed of .008 - .08 in/rev.

Finishing Cuts

- Lower than roughing cuts

b) Turning Operations

Turning (Performed on lathe)

– Part is moving and tool is stationary.

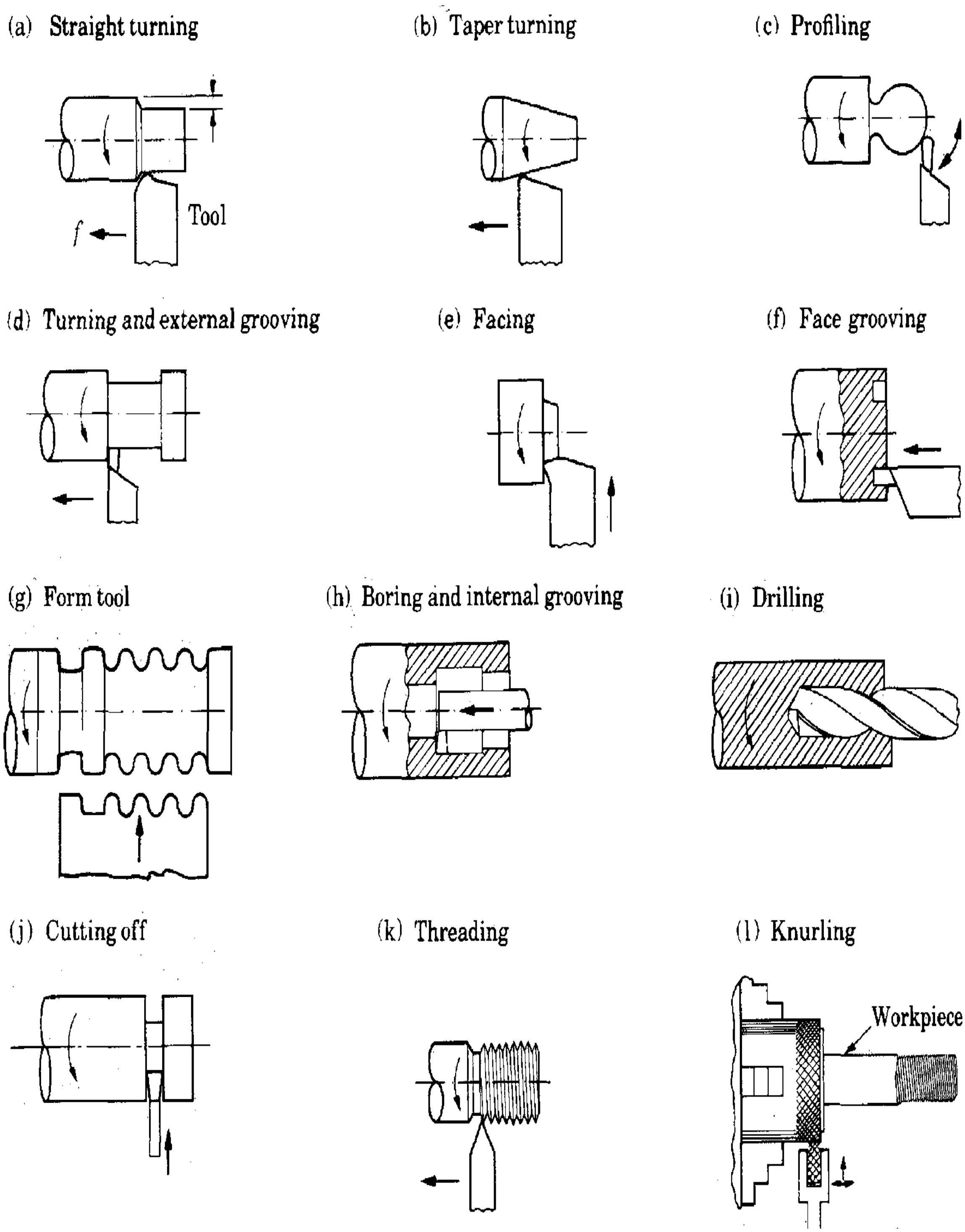
Used to make parts of round cross section

– Screws, shafts, pistons

Number of various lathe operations

– Turning, facing, boring, drilling, parting, threading

III. Lathe Operations



IV. Lathe -Plain Turing, Step Turning, Taper Turning, Knurling and Chamfering

In order perform various lathe operations such as plain turning, step turning, taper turning knurling and chamfering on a given material made of Mild steel.

We need mild steel bar of 22 mm diameter and 95 mm length.

a) Tools and Equipment used

H.S.S. single point cutting tool,

Parting tool,

Knurling tool,

Chuckey,

Tool post key,

Outside caliper,

Steel rule.

b) Operation Chart

S No.	Sequence of Operations	Cutting Tool Used
1.	Facing	H.S.S Single Point tool
2.	Rough turning	H.S.S Single Point tool
3	Finish turning	H.S.S Single Point tool
4	Step turning	Parting tool
5	Taper turning	H.S.S Single Point tool
6	Knurling	Knurling tool
7	Chamfering	H.S.S Single Point tool
7	Drilling	H.S.S Drill bit

V. Types of Operation

a) Facing Operation

Facing is the operation of machining the ends of a piece of work to produce a flat surface square with the axis. The operation involves feeding the tool perpendicular to the axis of rotation of the work piece.

A regular turning tool may be used for facing a large work piece. The cutting edge should be set at the same height as the center of the work piece. The tool is brought into work piece from around the center for the desired depth of cut and then is fed outward, generally by hand perpendicular to the axis of rotation of the work piece.

b) Rough Turning Operation

Rough turning is the operation of removal of excess material from the work piece in a minimum time by applying high rate of feed and heavy depth of cut. The depth of cut for roughing operations in machining the work ranges from 2 to 5 mm and the rate of feed is from 0.3 to 1.5 mm per revolution of the work.

c) Finish Turning Operation

It requires high cutting speed, small feed, and a very small depth of cut to generate a smooth surface. The depth of cut ranges from 0.5 to 1 mm and feed from 0.1 to 0.3 mm per revolution of the work piece.

d) Step Turning

Is the operation of making different diameters of desired length. The diameters and lengths are measured by means of outside caliper and steel rule respectively.

e) Taper Turning

A taper may be defined as a uniform increase or decrease in diameter of a piece of work measured along its length. In a lathe, taper turning means to produce a conical surface by gradual reduction in diameter from a cylindrical work piece.

A taper may be turned by any one of the following methods:

i. Form tool method
ii. Tail stock set over method
iii. Swiveling the compound rest and
iv. Taper turning attachment

Taper turning by swiveling the compound rest:

This method employs the principle of turning taper by rotating the work piece on the lathe axis and feeding the tool at an angle to the axis of rotation of the work piece. The tool mounted on the compound rest is attached to a circular base, graduated in degrees, which may be swiveled and clamped at any desired angle. Once the compound rest is set at the desired half taper angle, rotation of the compound slide screw will cause the tool to be fed at that angle and generate a corresponding taper.

The setting of the compound rest is done by swiveling the rest at the half taper angle. This is calculated by the equation.

$$\text{Tan } \alpha = (D-d) / 2L$$

Where α = Half taper angle

f) Knurling

Knurling is the process of embossing a diamond shaped pattern of the surface of a work piece. The purpose of knurling is to provide an effective gripping surface on a work piece to proven it from slipping when operated by hand. Knurling is performed by a special knurling tool which consists of a set of hardened steel rollers in a holder with the teeth cut on their surface in a definite pattern. The tool is held rigidly on the tool post and the rollers are pressed against the revolving surface of work piece to squeeze the metal against the multiple cutting edges, producing depressions in a regular pattern on the surface of the work piece.

Knurling is done at the slowest speed and oil is flowed on the tool and work piece. Knurling is done at the slowest speed and oil is flowed on the tool and work piece to dissipate heat generated during knurling. The feed varies from 1 to 2 mm per revolution.

g) Chamfering

Chamfering is the operation of beveling the extreme end of a work piece. This is done to remove the burrs, to protect the end of the work piece from being damaged and to have a better look. The operation may be performed after the completion of all operations. It is an essential operation after thread cutting so that the nut may pass freely on the threaded work piece.

VI. Metal Cutting Parameters

The cutting speed of a tool is the speed at which the metal is removed by the tool from the work piece. In a lathe, it is the peripheral speed of the work past the cutting tool expressed in meters/minute *a) Cutting speed (V) = π DN/1000, m/min*

Where, D = Diameter of the work in min

N = RPM of the work

b) Feed

The feed of a cutting tool in a Lathe work is the distance the tool advances for each revolution of the work. Feed is expressed in mm/rev.

c) Depth of cut

The depth is the perpendicular distance measured from the machined surface to the uncut surface of the work piece.

Depth of cut = (d1-d2) / 2

Where, d1= Diameter of the work surface before machining

d2= Diameter of the work surface after machining

While using HSS tool for turning mild steel work piece. The following parameters are to be chosen.

d) Rough Turning Operation

Cutting speed (V) = 25m/min,

feed(f) = 0.2 mm/rev,

Depth of cut(t) = 1 mm

e) Finish turning operation

Cutting speed (V) = 40m/min,

feed(f) = 0.1 mm/rev,

Depth of cut(t) = 0.2 mm

f) Tool geometry

Back rake angle = 7^0,

End relief angle = 6^0

Side relief angle = 6^0,

End cutting edge angle = 15^0

Side cutting edge angle = 15^0,

Nose radius = 2 mm

VII. PROCEDURE

a) The work piece and HSS single point cutting tool are securely held in the chuck and tool post respectively.
b) Operations such as facing, rough turning and finish turning are performed on a given mild steel bar one after the other in sequence up to the dimensions shown. Then the step turning is performed using parting tool.
c) Then the compound rest is swiveled by calculated half taper angle and taper is generated on the work piece. Rotation of the compound slide screw will cause the tool to be fed at the half-taper angle.
d) HSS single point cutting tool is replaced by the knurling tool and knurling operation is performed at the slowest speed of the spindle.
e) The knurling tool is replaced by the HSS single point tool again; the work piece is removed from the chuck and refixed with the unfinished part outside the chuck. This part is also rough turned, finish turned and facing is done for correct length.
f) Finally, the chamfering is done at the end of the work piece.

VIII. PRECAUTIONS

a) Operate the machine at optimal speeds
b) Do not take depth of cut more than 2 mm.
c) Knurling should be done at slow speeds and apply lubricating oil while knurling
d) Care should be taken to obtain the required accuracy.

IX. LATHE -THREAD CUTTING

In order to V-thread cutting on a lathe forming right hand and left hand metric threads as shown in fig.

We require Mild steel bar of 24 mm diameter and 100 mm length.

a) Tools and Equipment

H.S.S. single point cutting tool,

Grooving tool,

Threading tool thread gauge,

Outside caliper,

Chuck key,

Tool post key,

Steel rule.

X. OPERATION CHART

S no.	Sequence of Operations	Cutting tool used
1.	Facing	H.S.S Single Point cutting tool
2.	Rough turning	H.S.S Single Point cutting tool
3	Finish turning	H.S.S Single Point cutting tool
4	Step turning	H.S.S Single Point cutting tool
5	Grooving	Grooving tool
6	Thread cutting	Threading tool
7	Chamfering	H.S.S Single Point cutting tool

XI. PRINCIPLE OF THREAD CUTTING

The principle of thread cutting is to produce a helical groove on a cylindrical or conical surface by feeding the tool longitudinally when the job is revolved between centers or by a chuck. The longitudinal feed should be equal to the pitch of the thread to be cut per revolution of the work piece. The lead screw of the lathe, through which the saddle receives its traversing motion, has a definite pitch. A definite ratio between the longitudinal feed and rotation of the head stock spindle should therefore be found out so that the relative speeds of rotation of the work and the lead screw will result in the cutting of a screw of the desired pitch.

This is affected by change gears arranged between the spindle and the lead screw or by the change gear mechanism or feed box used in a modern lathe.

a) Calculation of change-wheels, metric thread on English lead screw

To calculate the wheels required for cutting a screw of certain pitch, it is necessary to know how the ratio is obtained and exactly where the driving and driven wheels are to be placed. Suppose the pitch of a lead screw is 12 mm and it is required to cut a screw of 3 mm pitch, then the lathe spindle must rotate 4 times the speed of the lead screw that is

$$\frac{\text{Spindle turn}}{\text{Lead screw turn}} = \frac{\text{Driven teeth}}{\text{Driver teeth}}$$

Hence we may say,

$$\frac{\text{Driven teeth}}{\text{Driver teeth}} = \frac{\text{Lead screw turn pitch of the screw to be cut}}{\text{spindle turn pitch of the lead screw}}$$

In British System

$$\frac{\text{Driven teeth}}{\text{Driver teeth}} = \frac{\text{Threads per inch on lead screw}}{\text{Threads per inch on work}}$$

Often engine lathes are equipped with a set of gears ranging from 20 to 120 teeth in steps of 5 teeth and one translating gear of 127 teeth. The cutting of metric threads on a lathe with an English pitch lead screw may be carried out by a translating gear of 127 teeth.

$$\frac{\text{Driven teeth}}{\text{Driver teeth}} = \frac{5\,p\,n}{127}$$

Where,

p = pitch of the thread to be cut and

N= threads per inch on lead screw

This is derived as follows:

$$\frac{\text{Driven teeth}}{\text{Driver teeth}} = \frac{\text{pitch of the work}}{\text{pitch of the lead screw}} = \frac{p}{(1/n) \times (127/5)} = \frac{5\,pn}{127}$$

$$\text{Since, pitch} = \frac{1}{\text{No. of threads per inch}}$$

XII. Thread Cutting Operation

In a thread cutting operation, the first step is to remove the excess material from the work piece to make its diameter equal to the major diameter of the screw thread. Change gears of correct size are then fitted to the end of the bed between the spindle and the lead screw.

The shape or form of the thread depends on the shape of the cutting tool to be used. In a metric thread, the included angle of the cutting edge should be ground exactly 60^0. The top of the tool nose should be set at the same height as the center of the work piece. A thread tool gauge is usually used against the turned surface to check the cutting tool, so that each face of the tool may be equally inclined to the center line of the work piece as shown.

The speed of the spindle is reduced by one half to one – fourth of the speed require for turning according to the type of the material being machined and the half – nut is then engaged. The depth of cut usually varies from 0.05 to 0.2 mm is given by advancing the tool perpendicular to the axis of the work.

After the tool has produced a helical groove up to the desired length of the work, the tool is quickly withdrawn by the use of the cross slide, the half-nut disengaged and the tool is brought back to the starting position to give a fresh cut. Before re-engaging the half-nut it is necessary to ensure that the tool will follow the same path it has traversed in the previous cut, otherwise the job will be spoiled. Several cuts are necessary before the full depth of thread is reached arising from this comes the necessity to “pick-up” the thread which is accomplished by using a chasing dial or thread indicator.

a) Chasing dial or thread indicator

The chasing dial is a special attachment used in modern lathes for accurate “picking up” of the thread. This dial indicates when to close the split of half nuts. This is mounted on the right end of the apron. It consists of a vertical shaft with a worm gear engaged with the lead screw. The top of the spindle has a revolving dial marked with lines and numbers. The dial turns with the lead screw so long the half nut is not engaged.

If the half-nut is closed and the carriage moves along the dial stands still. As the dial turns, the graduations pass a fixed reference line. The half-nut is closed for all even threads when any line on the dial coincides with the reference line. For all odd threads, the half-nut is closed at any numbered line on the dial determined from the charts. If the pitch of the thread to be cut is an exact multiple of the pitch of the lead screw, the thread is called even thread, if otherwise the thread is odd thread.

In a chasing dial, the rule for determining the dial division is: In case of metric threads, the product of the pitch of lead screw and the no. of teeth on the worm wheel must be an exact multiple of the pitch of the threads to be cut. In case of English threads, the product of the threads per inch to be cut and the number of teeth on the worm wheel must be an exact multiple of the number of threads per inch of the lead screw. For example, if the pitch of the lead screw is 6 mm and the worm wheel has 15 teeth.

The product will be 90. so any pitch which is exactly divisible by 90, such as 1, 1.25, 2.25,3,3.75,4.5,5,6,7.5,9,10,15,30,45,90 may be picked up when any line of the dial coincides with the reference line.

b) Right hand and left-hand thread

If the bolt advances into the nut when rotated in clockwise direction, the thread is called right-hand thread. When cutting a right-hand thread the carriage must move towards the head stock.

If the bolt advances into the nut when rotated in counter-clockwise direction, the thread is called left-hand, for a left hand thread the carriage moves away from the head stock and towards the tail stock. The job moves as always in the anti-clock wise direction when viewed from the tail stock end. The direction at which the carriage moves in relation to lathe head stock is controlled by means of the tumbler gears or bevel gear feed reversing mechanism.

XIII. Procedure

a) The work piece and HSS single point cutting tool are fixed in chuck and tool post respectively.
b) Operations such as facing, rough turning finish turning and step turning are performed on the given mild steel bar one after the other in sequence up to the dimensions shown.
c) Single point cutting tool is replaced by a grooving tool and grooving operation is performed at half of the normal spindle speed.
d) The grooving tool is replaced by a threading tool. Right hand and left hand metric threads are cut on the work piece up to the required length at 1/4th of the normal speed of the spindle.
e) Threading tool replaced by a single point cutting tool again and finally chamfering is done at right end of the work piece at normal spindle speed.

XIV. Precautions

a) Low spindle speeds should be used for accurate threads in thread cutting operation.
b) Ensure correct engage and dis-engage of half-nut.
c) Plenty of oil should be flowed on the work and tool during thread cutting.