TURNER

NSQF LEVEL - 5

3rd Semester

TRADE THEORY

SECTOR: Production & Manufacturing

Directorate General of Training

DIRECTORATE GENERAL OF TRAINING
MINISTRY OF SKILL DEVELOPMENT & ENTREPRENEURSHIP
GOVERNMENT OF INDIA

NATIONAL INSTRUCTIONAL
MEDIA INSTITUTE, CHENNAI

Post Box No. 3142, CTI Campus, Guindy, Chennai - 600 032

Sector : Turner

Duration : 2 - Years

Trade : Turner 3rd Semester - Trade Theory - NSQF level 5

First Edition : October 2018 Copies : 1,000

Rs.130/-

Published by:

NATIONAL INSTRUCTIONAL MEDIA INSTITUTE
P. B. No.3142, CTI Campus, Guindy Industrial Estate,
Guindy, Chennai - 600 032.
Phone : 044 - 2250 0248, 2250 0657, 2250 2421
Fax : 91 - 44 - 2250 0791
email : chennai-nimi@nic.in, nimi_bsnl@dataone.in
Website: www.nimi.gov.in

FOREWORD

The Government of India has set an ambitious target of imparting skills to 30 crores people, one out of every four Indians, by 2020 to help them secure jobs as part of the National Skills Development Policy. Industrial Training Institutes (ITIs) play a vital role in this process especially in terms of providing skilled manpower. Keeping this in mind, and for providing the current industry relevant skill training to Trainees, ITI syllabus has been recently updated with the help of Mentor Councils comprising various stakeholder's viz. Industries, Entrepreneurs, Academicians and representatives from ITIs.

The National Instructional Media Institute (NIMI), Chennai has now come up with instructional material to suit the revised curriculum for **Turner Trade Theory 3^{rds} Semester in Production & Manufacturing Sector .** The NSQF Level - 5 Trade theory will help the trainees to get an international equivalency standard where their skill proficiency and competency will be duly recognized across the globe and this will also increase the scope of recognition of prior learning. NSQF Level - 5 trainees will also get the opportunities to promote life long learning and skill development. I have no doubt that with NSQF Level - 5 the trainers and trainees of ITIs, and all stakeholders will derive maximum benefits from these IMPs and that NIMI's effort will go a long way in improving the quality of Vocational training in the country.

The Executive Director & Staff of NIMI and members of Media Development Committee deserve appreciation for their contribution in bringing out this publication.

Jai Hind

RAJESH AGGARWAL
Director General/ Addl. Secretary
Ministry of Skill Development & Entrepreneurship,
Government of India.

New Delhi - 110 001

PREFACE

The National Instructional Media Institute (NIMI) was established in 1986 at Chennai by then Directorate General of Employment and Training (D.G.E & T), Ministry of Labour and Employment, (now under Directorate General of Training, Ministry of Skill Development and Entrepreneurship) Government of India, with technical assistance from the Govt. of the Federal Republic of Germany. The prime objective of this institute is to develop and provide instructional materials for various trades as per the prescribed syllabi under the Craftsman and Apprenticeship Training Schemes.

The instructional materials are created keeping in mind, the main objective of Vocational Training under NCVT/NAC in India, which is to help an individual to master skills to do a job. The instructional materials are generated in the form of Instructional Media Packages (IMPs). An IMP consists of Theory book, Practical book, Test and Assignment book, Instructor Guide, Audio Visual Aid (Wall charts and Transparencies) and other support materials.

The trade practical book consists of series of exercises to be completed by the trainees in the workshop. These exercises are designed to ensure that all the skills in the prescribed syllabus are covered. The trade theory book provides related theoretical knowledge required to enable the trainee to do a job. The test and assignments will enable the instructor to give assignments for the evaluation of the performance of a trainee. The wall charts and transparencies are unique, as they not only help the instructor to effectively present a topic but also help him to assess the trainee's understanding. The instructor guide enables the instructor to plan his schedule of instruction, plan the raw material requirements, day to day lessons and demonstrations.

IMPs also deals with the complex skills required to be developed for effective team work. Necessary care has also been taken to include important skill areas of allied trades as prescribed in the syllabus.

The availability of a complete Instructional Media Package in an institute helps both the trainer and management to impart effective training.

The IMPs are the outcome of collective efforts of the staff members of NIMI and the members of the Media Development Committees specially drawn from Public and Private sector industries, various training institutes under the Directorate General of Training (DGT), Government and Private ITIs.

NIMI would like to take this opportunity to convey sincere thanks to the Directors of Employment & Training of various State Governments, Training Departments of Industries both in the Public and Private sectors, Officers of DGT and DGT field institutes, proof readers, individual media developers and coordinators, but for whose active support NIMI would not have been able to bring out this materials.

R. P. DHINGRA
EXECUTIVE DIRECTOR

Chennai - 600 032

ACKNOWLEDGEMENT

National Instructional Media Institute (NIMI) sincerely acknowledges with thanks for the co-operation and contribution extended by the following Media Developers and their sponsoring organisations to bring out this Instructional Material **(Trade Theory)** for the trade of **Turner (NSQF Level-5)** under the Production & Manufacturing Sector for ITIs.

MEDIA DEVELOPMENT COMMITTEE MEMBERS

Shri. A. Vijayaraghavan	_	Assistant Director of Training (Retd.) ATI, Chennai-32.
Shri. R. Purushothaman	_	Assistant Director (Retd.) (Mechanical Engg) MSME, Chennai-32.
Shri. M. Sampath	_	Training officer (Retd.) CTI, Chennai-32.
Shri. A. Natarajan	_	Training Officer Govt. ITI, Coimbatore.
Shri. N. Sampath	_	Assistant Training Officer Govt. ITI, Ambathur.
Shri. Dhayalamoorthy	_	Assistant Training Officer Govt. ITI, Tiruvannamalai.
Shri. Sampath Kumar	_	Assistant Training Officer Govt. ITI, Chennai-32.
Shri. S. Balasubramanian	_	Assistant Training Officer Govt. ITI, Chennai-32.
Shri. D.N. Srinivasa Shetty	_	Junior Training Officer Govt. ITI, Bangalore -29.
Shri. Chandrahasa	_	Junior Training Officer Govt. ITI, Mysore - 570007.
Shri. Srinivasa Naik	_	Junior Training Officer Govt. ITI, Mysore - 570007.
Shri. K. Srinivasa Rao	_	Joint Director, Co-ordinator, NIMI, Chennai-32.
Shri. V. Gopalakrishnan	_	Assistant Manager, Co-ordinator, NIMI, Chennai - 32.
Shri. N. Sundararajan	_	Assistant Manager, Co-ordinator, NIMI, Chennai - 32.

NIMI records its appreciation for the Data Entry, CAD, DTP operators for their excellent and devoted services in the process of development of this Instructional Material.

NIMI also acknowledges with thanks the invaluable efforts rendered by all other NIMI staff who have contributed towards the development of this Instructional Material.

NIMI is also grateful to everyone who has directly or indirectly helped in developing this Instructional Material.

INTRODUCTION

TRADE THEORY

This manual of trade theory consists of theorectical information for the First Semester couse of the Turner Trade. The contents are sequenced accoring to the practical exercise contained in the manual on Trade practical. Attempt has been made to relate the theortical aspects with the skill covered in each exercise to the extent possible. This cor-relation is maintained to help the trainees to develop the perceptional capabilities for performing the skills.

Module 1	**Form Turning**	125 Hrs
Module 2	**Turning with lathe attachements**	175 Hrs
Module 3	**Boring**	100 Hrs
Module 4	**Thread cutting**	125 Hrs
	Total	**525 Hrs**

The Trade Theory has to be taught and learnt along with the corresponding exercise contained in the manual on trade practical. The indications about the corresponding practical exercise are given in every sheet of the manual.

It will be preferable to teach/learn the trade theory connected to each exercise atleast one class before performing the related skills in the shop floor. The trade theory is to be treated as an integral part of each exercise.

The material is not the purpose of self learning and should be considered as supplementary to class room instruction.

TRADE PRACTICAL

The trade practical manual is intented to be used in workshop . It consists of a series of practical exercies to be completed by the trainees during the First Semester course of the Turner trade supplemented and supported by instructions/ informations to assist in performing the exercises. These exercises are designed to ensure that all the skills in the prescribed syllabus are covered.

The manual is divided into four modules. The distribution of time for the practical in the four modules are given below.

The skill training in the computer lab is planned through a seriess of practical exercises centred around some practical project. However, there are few instance where the individual exercise does not form a part of project.

While developing the practical manual a sincere effort was made to prepare each exercise which will be easy to understand and carry out even by below average traninee. However the development team accept that there is a scope for further improvement. NIMI, looks forward to the suggestions from the experienced training faculty sfor improving the manual.

CONTENTS

Lesson No.	Title of the Lesson	Page No.
	Module 1: Form Turning	
3.1.88	Form tools function - types and uses	1
3.1.89	Dial test indicator - construction - used for checking the forms	7
3.1.90	Jigs & fixtures - types and uses	11
3.1.91	Cutting Tool Material	15
3.1.92	Tool life and quality of a cutting tool material	17
3.1.93	Checking of taper with Sine bar and Roller	19
3.1.94	Cutting speed and feed, turning time, depth of cut calculation	24
	Module 2: Turning with lathe attachments	
3.2.95	Face plate - accessories used on face plate, angle plate	36
	Angle plates	37
	Balancing - its necessity	39
3.2.96	Surface finish symbol used on working blue prints	40
	Surface texture measuring instruments	43
	Machining symbols	44
	Lapping	46
3.2.97	Preventive maintenance - its necessity	49
	Documentations - 1	53
	Marking off and marking table	57
3.2.98	Roller and revolving steadies - necessity - construction - uses	59
3.2.99	Different types of attachements used in lathe	61
	Special attachments used in centre lathe	63
	Copying attachment	65
3.2.100	Various procedures of thread measurements	66
	Thread measurement (effective diameter)	68
	Thread measurement (minor diameter)	71
	Screw thread measurement (flank angle and form)	73

Lesson No.	Title of the Lesson	Page No.
	Module 3: Boring	
3.3.101	Toolmaker's button and its parts	75
3.3.102	Telescopic gauge - construction - uses	78
3.3.103	Inside micrometer - metric - construction	79
3.3.104	Inside micrometer - Inch	81
3.3.105	Care for holding split bearing, fixture and its uses	82
	Module 4: Thread cutting	
3.4.106	Calculation involving fractional thread (odd and even)	84
	Simple and compound gear trains	86
	Gear calculation for cutting metric thread on British lathe and vice versa	87
3.4.107	Multiple thread function, use	91
3.4.108	Multi-start thread and methods	93
3.4.109	Calculation involving shape of tool (Square thread tool)	96
3.4.110	Helix angle and its effects on threading tool clearance angles	100

LEARNING/ ASSESSABLE OUTCOME

On completion of this book you shall be able to

- **Recognise, understand typical turning operations like Form turning, taper turning, boring etc.,**
- **Draw and organise work to make Morse Taper plug, Taper sleeves, executing complex job involving face plate, angle plate etc.,**
- **Execute turning of crackshaft, turning of long shaft using lathe attachments such as revolving steady, roller steady etc.,**
- **Perform eccentric boring, stepped boring with in 50 micron accuracy level and use of inside micrometer, telescopic gauges etc for measurement.**
- **Execute metric and British standard thread cutting, multi start thread cutting, making use of change wheel calculation, and checking of threads.**
- **Understand the use and applications of all types of lathe attachments.**

SYLLABUS

Third Semester **Duration: Six Month**

Week No.	Ref. Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
53	Plan & set the machine parameter to produce precision engineering component to appropriate accuracy by performing different turning operation. *[Appropriate accuracy - ±0.02mm/ (MT - 3) (proof turning); Different turning operation – Plain turning, taper turning, boring threading, knurling, grooving, chamfering etc.]*	88. Form turning practice by hand. (8 hrs.) 89. Re-sharpening of form tools using bench grinder. (2 hrs.) 90. Tool machine handle turning by combination feed. (15 hrs.)	Form tools-function-types and uses, Template-purpose & use. Dial test indicatorconstruction & uses Calculation involving modified rake and clearance angles of lathe tool at above and below the center height. Subsequent effect of tool setting. Jig and fixture-definition, type and use. Chip breaker on tool-purpose and type
54-55	-do-	91. Turn Morse taper plug (different number) and check with ring gauge / suitable MT sleeve. (25 hrs.) 92. Make revolving tail stock centre- Bush type (C-40). (Proof machining) (25 hrs.)	Cutting tool material-H.C.S., HSS, Tungsten. Carbide, Ceramic etc, - Constituents and their percentage. Tool life, quality of a cutting material.
56	-do-	93. Make Morse taper sleeve and check by taper plug gauge. (25 hrs.)	Checking of taper with sin bar and rollercalculation involved
57	-do-	94. Make mandrel/ plug gauge with an accuracy of ±0.02mm using tungsten carbide tools including throw-away tips. (25 hrs.)	Cutting speed, feed, turning time, depth of cut calculation, cutting speed chart (tungsten carbide tool) etc. Basic classification of tungsten carbide tips.

58-59	Set & Produce components on irregular shaped job using different lathe accessories. *[Different Lathe accessories: - Face plate, angle plate]*	95. Setting and turning operation involving face and angle plate (25 hrs.) 96. Make angle plate using face plate. (25 hrs.)	Accessories used on face plate – their uses. Angle plate-its construction & use. Balancing-its necessity. Surface finish symbols used on working blueprints- I.S. system lapping, honing etc.
60-61	Plan and set the machine using lathe attachment to produce different utility component/ item as per drawing. *[Different utility component/ item – Crank shaft (single throw), stub arbour with accessories etc.]*	97. Holding and truing of Crankshaft – single throw (Desirable). (50 hrs.)	Preventive maintenance, its necessity, frequency of lubrication. Preventive maintenance schedule., TPM (Total Productive Maintenance), EHS (Environment, health, Safety) Marking table-construction and function. Angle plate-construction, eccentricity checking.
62	-do-	98. Turning of long shaft using steady rest (within 0.1 mm). (25 hrs.)	Roller and revolving steadies, Necessary, construction, uses etc.
63-64	-do-	99. Use of attachments on lathe for different operations. (25 hrs.) 100. Turning standard stub arbor with accessories collar, tie rod, lock nut. (25 hrs.)	Different types of attachments used in lathe. Various procedures of thread measurement thread screw pitch gauge. Screw thread micrometer, microscope etc.
65	Set the machining parameters and produce & assemble components by performing different boring operations with an appropriate accuracy. *[Different boring operation – eccentric boring, stepped boring; appropriate accuracy - ±0.05mm]*	101. Perform eccentric boring and make male & female eccentric fitting. (15 hrs.) 102. Position boring using tool maker's button. (10 hrs.)	Tool maker's button and its parts, construction and uses, telescopic gauge its construction and uses.

66	-do-	103. Boring and stepped boring (within ± 0.05 mm) (15 hrs.) 104. Cutting of helical grooves in bearing and bushes (Oil groove) (10 hrs.)	Inside micrometer principle, construction graduation, reading, use etc. (Metric & Inch.)
67-68	-do-	105. Turning & boring of split bearing – (using boring bar and fixture) (50 hrs.)	Care for holding split bearing. Fixture and its use in turning
69	-do-	106. Cutting thread of 8 and 11 TPI. (25 hrs.)	Calculation involving fractional threads. Odd & even threads
70	-do-	107. Multi start thread cutting (B.S.W.) external & internal. (25 hrs.)	Multiple thread function, use, different between pitch & lead, formulate to find out start, pitch, lead. Gear ratio etc.
71	-do-	108. Multi start thread cutting (Metric) (External & internal). (25 hrs.)	Indexing of start - different methods tool shape for multi-start thread. Setting of a lathe calculation for required change wheel
72	-do-	109. Multi-start thread cutting, square form (Male & Female). (25 hrs.)	Calculation involving shape of tool, change wheel, core dia etc. Calculation involving shape, size pitch, core dia. Etc.
73	-do-	110. Make half nut as per standard lead screw. (25 hrs.)	Helix angle, leading angle & following angles. Thread dimensions-tool shape, gear, gear calculation, pitch, depth, lead etc.
74-75		**Implant training / Project work (work in a team)** 1. Pedestal bearing 2. crank shaft 3. arbor with clamping nut 4. mandrel with jaw 5. Eccentric with connecting rod 6. Taper mandrel with sphere 7. Lever Handle	
76-77		Revision	
78		Examination	

13.	Plan and perform basic maintenance of lathe & grinding machine and examine their functionality.	85. Balancing, mounting & dressing of grinding wheel (Pedestal). (5 hrs.) 86. Periodical lubrication procedure on lathe. (10 hrs.) 87. Preventive maintenance of lathe. (10 hrs.)	Lubricant-function, types, sources of lubricant. Method of lubrication. Dial test indicator use for parallelism and concentricity etc. in respect of lathe work Grinding wheel abrasive, grit, grade, bond etc.
14.		**In-plant training / Project work** 1. Drill extension socket 2. conical brush 3. V-belt pulley 4. Tail Stock Centre (MT – 3) 5. Taper ring gauge 6. Sprocket 7. Socket spanner	
15.		**Revision**	
16.		**Examination**	

Note: -

1. Some of the sample project works (indicative only) are given against each semester.
2. Instructor may design their own project and also inputs from local industry may be taken for designing such new project.
3. The project should broadly covered maximum skills in the particular trade and must involve some problem solving skill. Emphasis should be on Teamwork: Knowing the power of synergy/ collaboration, Work to be assigned in a group (Group of at least 4 trainees). The group should demonstrate Planning, Execution, Contribution and application of Learning. They need to submit Project report.
4. If the instructor feels that for execution of specific project more time is required than he may plan accordingly to produce components/ sub-assemblies in appropriate time i.e., may be in the previous semester or during execution of normal trade practical.
5. More emphasis to be given on video/real-life pictures during theoretical classes. Some real-life pictures/videos of both conventional & CNC turning operation, production of different components, turning of complex job, etc., may be shown to the trainees to give a feel of Industry and their future assignment.

Form tools function - types and uses

Objectives: At the end of this lesson you shall be able to
- **distinguish between plain turning and form turning**
- **state the necessity of form turning**
- **brief the methods of form turning**
- **check forms with radious gauges**
- **understand use of templates**
- **learn the calculation of effective rake, clearance angle, setting**
- **understand type of chip breakers & it uses.**

Tools Function

The plain turning process is capable of generating cylindrical, conical and flat surfaces whereas the form turning process is intended for generating concave, convex profiles or the combination of both on the work piece. The figure shows the different types of forming obtained on the workpiece with the help of the form tools.

Purpose of form turning

Form turning is mainly used for making the handles to provide better grip for handling purposes.

It provides additional decoration on the product.

Concave forming is mainly used in ball bearing races, as a seat for ball or roller pin.

Form turning is largely employed in the manufacture of automobile engineering components.

Methods of turning formed surfaces

Formed surfaces can be turned by:

- using form tools
- using templates
- free hand form turning.

Form turning tools (Figs 1 & 2)

Fig 1

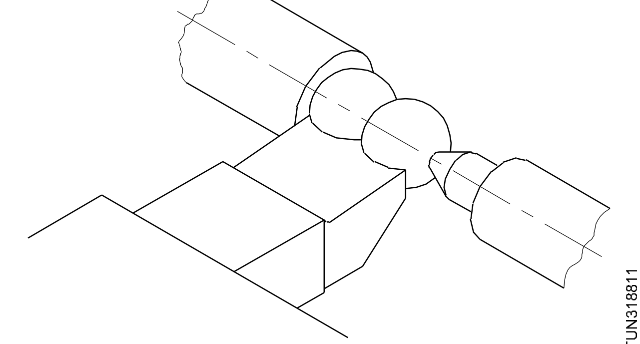
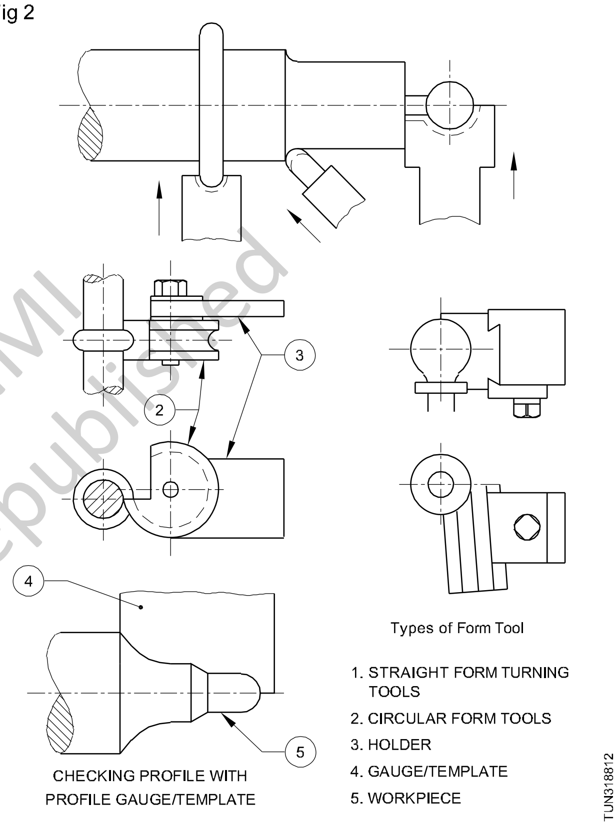


Fig 2



Form tools are ground so that the profile or contour of the cutting edge corresponds to the desired shape. If the tool bit is ground accurately, an accurate form is reproduced on the workpiece. If the form must be held to fine tolerances, it is wise to check the accuracy of the cutting edge on an optical projector. For mass production purposes, carbide tipped form tools are used. When a form tool requires sharpening, it is important that the grinding occurs only on the top of the cutting edge. Otherwise, the shape and accuracy of the form will be altered. When forms are produced manually, constant checking of parts against the master template is necessary.

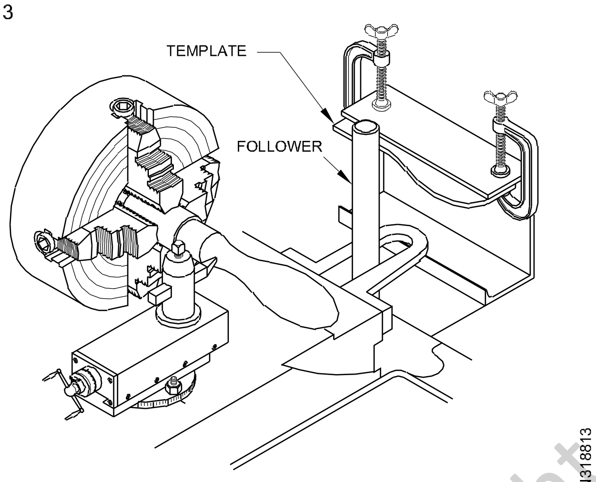
Form turning using a template (Fig 3)

Very accurate profiles or contours may be produced by using templates. The main parts involved in this method are the:

- cutting tool
- template
- follower.

A follower is fastened to the cross-slide to follow the contour of the template. The accuracy of the template determines the accuracy of the form produced. The template is mounted on the back of the lathe. With these arrangements, the tool has to be moved by hand using cross and longitudinal feeds.

Fig 3



SETUP FOR PRODUCING A SPECIAL FORM

Free hand form turning (Figs 4a & 4b)

Free hand turning is generally used only when a few parts are required and when it would be uneconomical to provide templates and follower. A very high skill is needed to produce accurate forms on the workpiece. This method involves simultaneous control over the carriage and the cross-slide. Also it involves coordination of both the hands of the operator.

Types of form Tool

1 Flat form tool

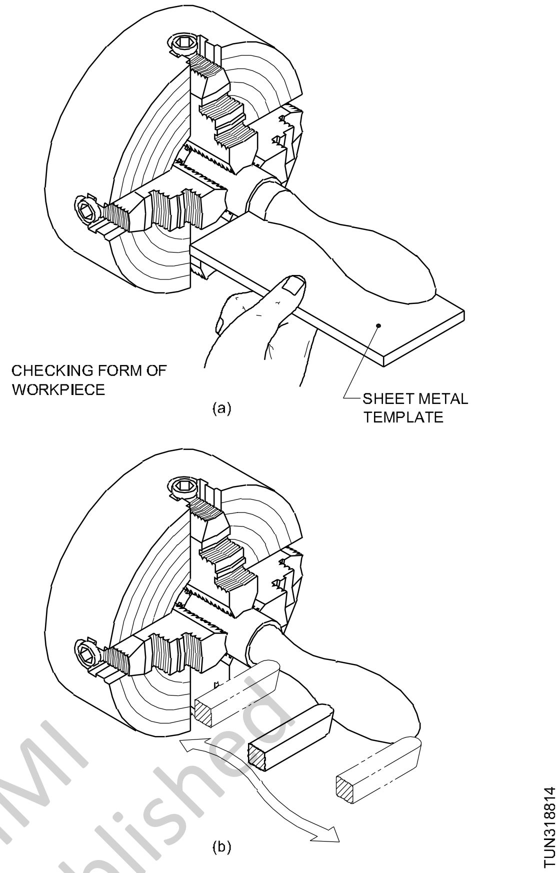
2 Circular form tool

3 End form tool

Radius and fillet gauges

Components are machined to have curved formation on the edges or at the junction of two steps. Accordingly they are called radius and fillets. The size of the radius and fillet is normally provided on a drawing. The gauges used to check the radius formed on the edges of diameters are called radius gauges and the radius formed at the steps are checked with fillet gauges . In other words these gauges can check the concave and convex forms on the component.

Fig 4



They are made of hardened sheet metal each to a precise radius. They are used to check the radii by comparing the radius on a part with the radius of the gauges.

Fig 1 shows the application of a fillet gauge to check the radius formed externally. Fig 2 shows the application of a radius gauge to check the fillet formed on a turned component. The other typical applications are:

Fig 1

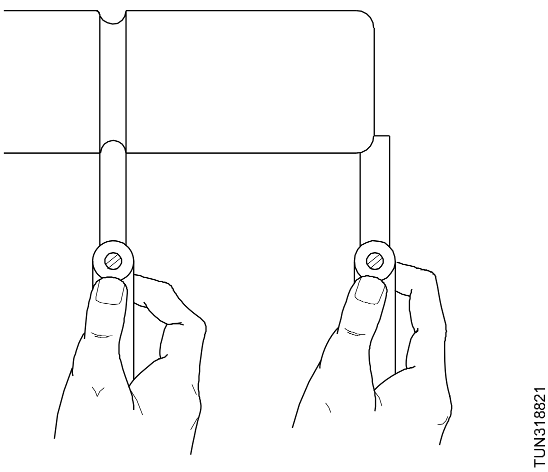
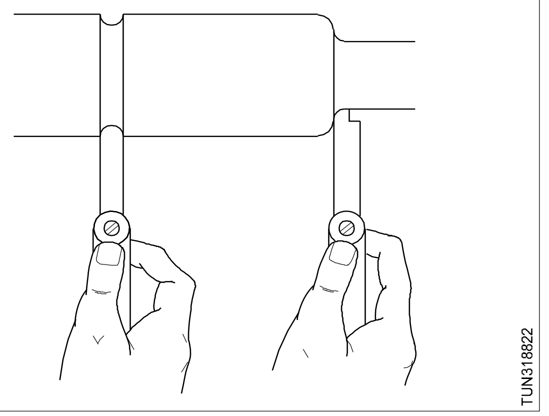
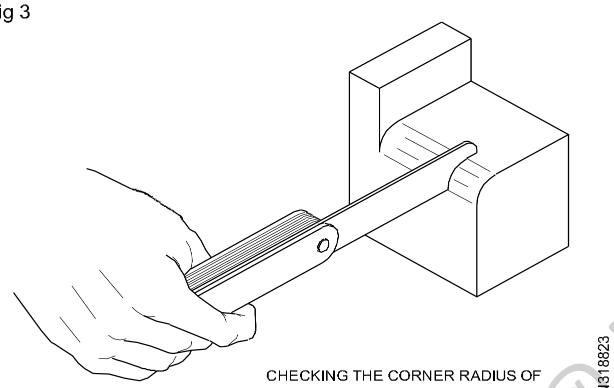


Fig 2



- Checking the corner radius of a part being filed to shape. (Fig 3)

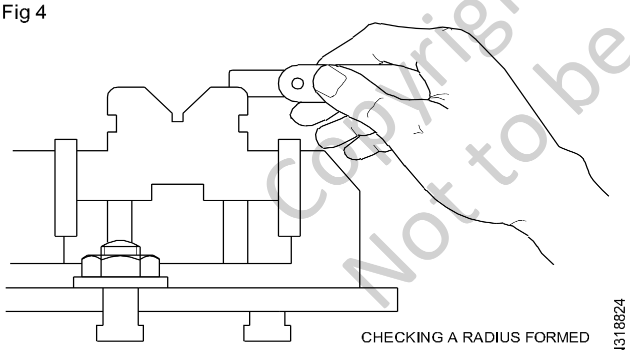
Fig 3



CHECKING THE CORNER RADIUS OF A PART BEING FILED TO SHAPE

- Checking a radius formed by a milling cutter. (Fig 4)

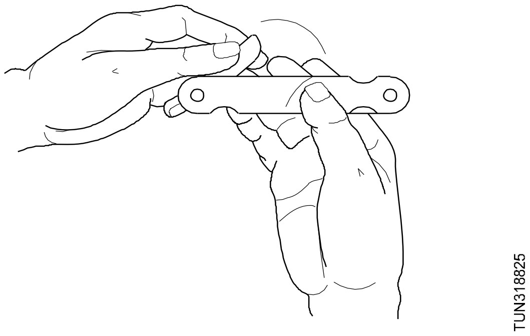
Fig 4



CHECKING A RADIUS FORMED BY A MILLING CUTTER

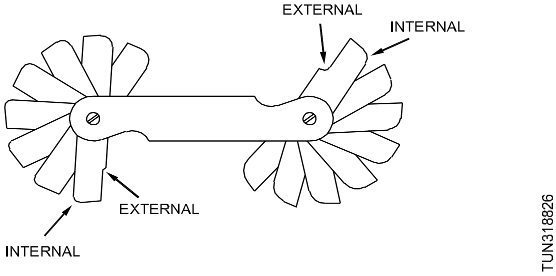
The radius and fillet gauges are available in sets of several blades which fold into a holder when not in use. (Fig 5)

Fig 5



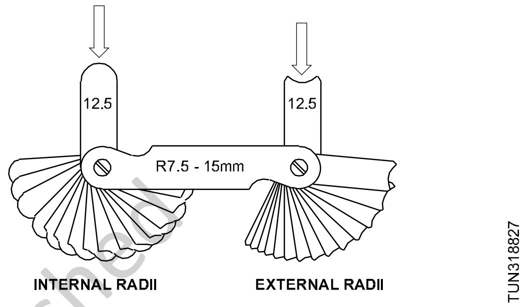
Some sets have provisions to check the radius and fillet on each blade. (Fig 6)

Fig 6



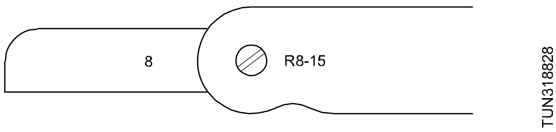
And some sets have separate sets of blades to check the radius and fillet. (Fig 7)

Fig 7



Each blade can be swung out of the holder separately, and has its size engraved on it. (Fig 8)

Fig 8



Fillet gauges are available in sets to check the radii and fillets from:

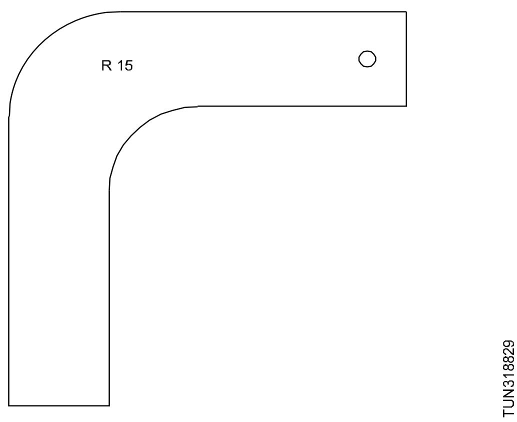
1 to 7 mm in steps of 0.5 mm

7.5 to 15 mm in steps of 0.5 mm

15.5 to 25 mm in steps 0.5 mm.

Individual gauges are also available. They usually have internal and external radii on each gauge and are made in sizes from 1 to 100 mm in steps of 1 mm. (Fig 9)

Fig 9

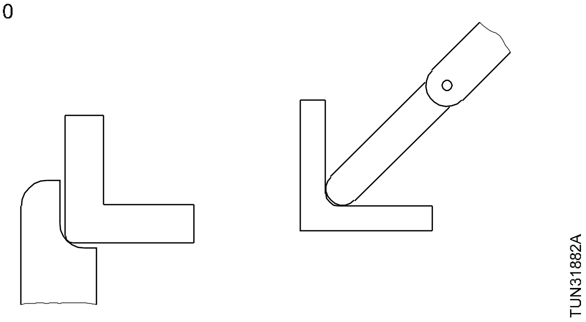


Before using the radius gauge:

- check that it is clean and undamage
- remove burrs from the workpiece
- select the leaf of the gauge from the set corresponding to the radius to be checked.

Fig 10 shows that the radius of the fillet and that of the external radius are smaller than the gauge.

Fig 10

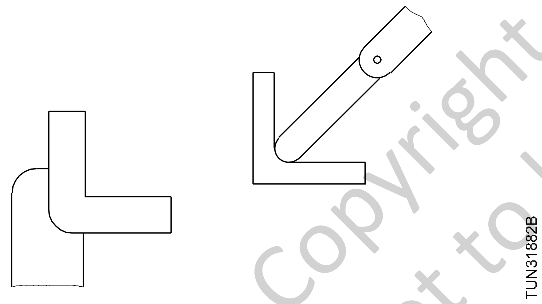


Try a smaller gauge to determine the radius dimension.

File or machine the workpiece if it has to be of the radius of the ga if you need to find the radius dimension.

Fig 11 shows that the radius of the fillet and that of the external radius are larger than the gauge.

Fig 11

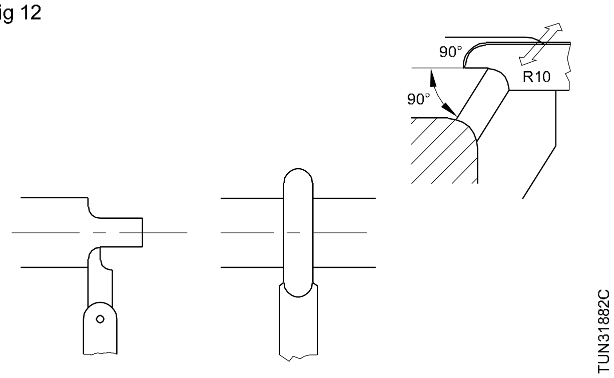


Try a larger gauge if you need to find the radius dimension.

If the workpiece has to be of the radius of the gauge you may have to reject the workpiece.

Fig 12 shows the workpiece having the same radius as that of the gauge that is being used for checking.

Fig 12



Templates (Profiles)

A template is made of good quality steel. It is used to mark and check profiles (Figs 1,2, & 3)

Templates are used for rapid standardised marking out of complicated shape or irregular shape It is also used while slotting or cutting complicated contours. The contoured of the template is case-hardened and can serve as a guide for marking.

Fig 1

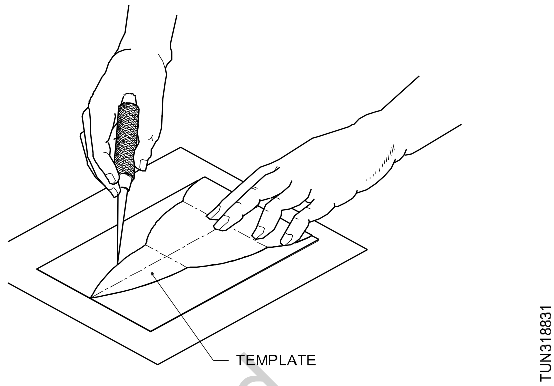


Fig 2

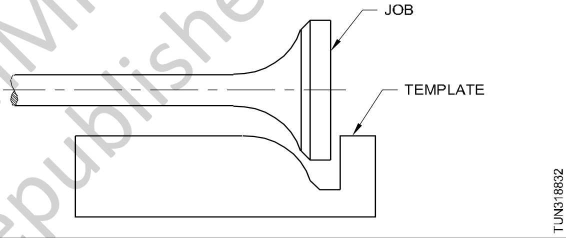
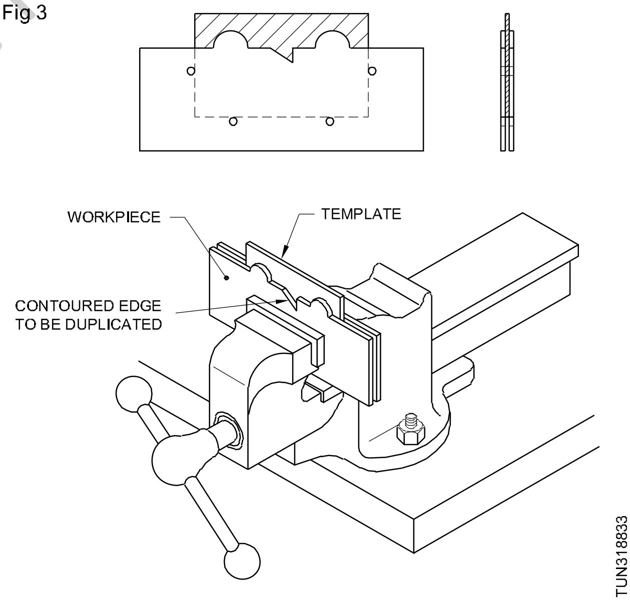


Fig 3



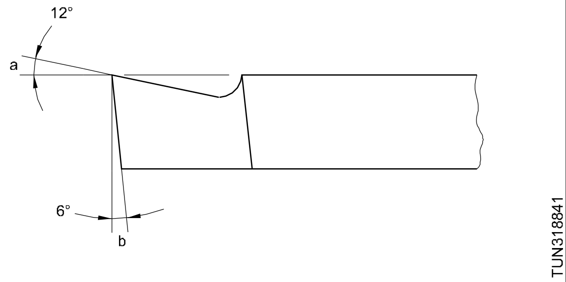
Templates save time in marking out and and result in standarisation of work.

A template is a negative replica of the profile it checks.

Calculation of the effective rake and clearance angles for tools set above or below the centre height

The setting of the tool with respect to the work axis affects the value of the rake angle (a) and the clearance angle (b) to which the tools is gound Fig 1.

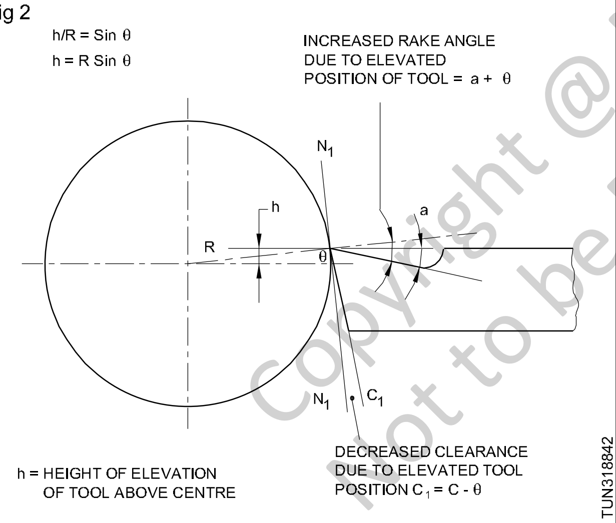
Fig 1



If it is above the work centre, the clearance angle diminishes and the rake angle increases, and there is a point at which clearance disappears completely and the tool will not cut but only rub on the surface of the work. Converely, if it below the centre, it will increase the clearance angle and decrease the rake angle.

Example Fig 2

Fig 2



To determine the effective rake and clearance angles when turning a job of ∅ 50mm, the tool being set 2mm above the centre height, with the ground angles being as shown in Fig 1, we have

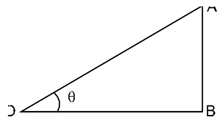
OA = 25 mm

AB = 2 mm

Sin θ = 2/25 = 0.080

θ = 4° 34'

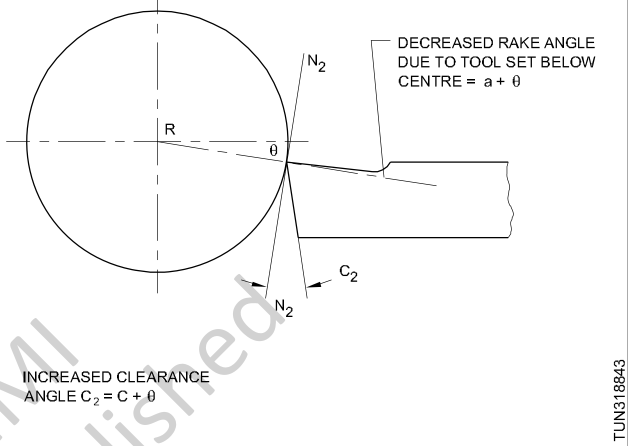
The effective top rake angle = 12° + 4° 34'
= 16° 34'



The effective front clearance angle = 6° + 4° 34'
= 1° 26'

Example Fig 3

Fig 3



Used on a job of 60 mm dia. The tool set at 2 mm below the centre.

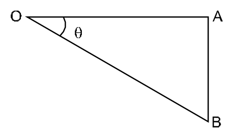
OB = 30 mm

AB = 2 mm

Sin θ = 2/30 = 1/15 = 0.066

θ = 3° 50'

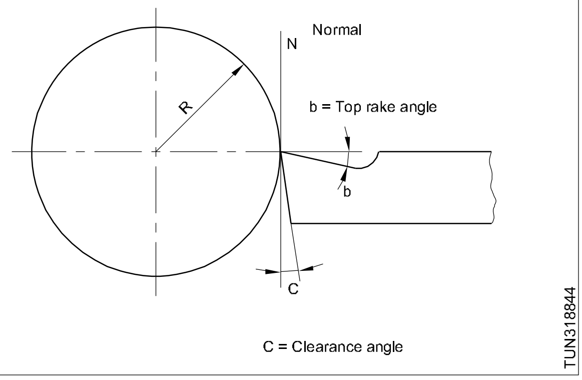
The effective top rake angle = 12° + 3° 50'
= 8° 10'



The effective front clearance angle = 6° + 3° 50'
= 9° 50'

So the tool must be set at the centre of the work piece.

Fig 4



Subsequent effect of tool setting:

Tool point above the centre height - Tool will not cut, only rubs.

Tool point below the centre height - Too digs, breaks, thro' away job, damage tools & job.

Tool point just in line with centre height - effective cutting, good finish and long life for tool.

Chip breaker

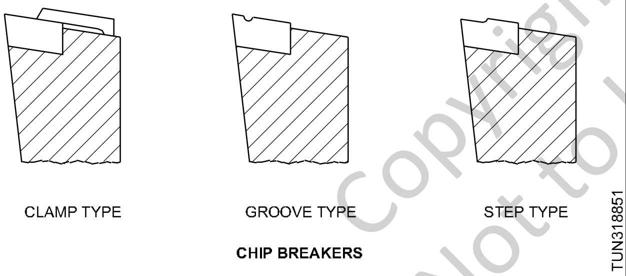
Chip breaker is a means with which the continuous long curly chips are broken into comparatively small pieces for easy handling, thus preventing it from becoming a work hazard.

Types of chip breakers (Fig 1)

- Step type - built in
- Groove type - built in
- Clamp type - mechanical

The common methods of breaking the chips in normal shop practice are summarized here.

Fig 1



CHIP BREAKERS

- By clamping a piece of sheet metal in the path of the coil.
- By a step type chip breaker in which a step is ground on the face of the tool, along the cutting edge.
- By a groove type chip breaker in which a small groove is ground behind the cutting edge.
- By a clamp type chip breaker in which a thin carbide plate or clamp is brazed or screwed on the face of the tool.

Throw-away tip tool-holders are provided with chip breakers.

Necessity for breaking the chips

Long and unbroken chips produced while turning ductile materials are difficult to handle and injurious to the operator. They should be broken into convenient lengths for easy disposal and also to protect the finished work-surfaces. Therefore, tools are provided with devices to curl and break the chips. These devices, which are called chip breakers are in the form of ground chip breakers in the case of brazed carbide tools, and external or pre–sintered chip breakers in the case of disposable, indexable inserts. However, with high speed steel tools, this problem may not arise because at low cutting speeds the chip has often natural curl and tends to be brittle enough to break on its own.

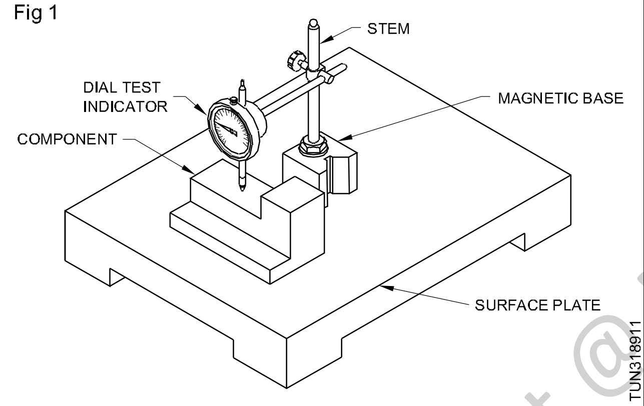
Dial test indicator - construction - used for checking the forms

Objectives: At the end of this lesson you shall be able to
- **state the working principle of a dial test indicator**
- **identify the parts of a dial test indicator**
- **state the important features of a dial test indicator**
- **state the functions of a dial test indicator**
- **identify different types of stands.**

What are dial test indicators

Dial test indicators are fine precision type of instruments used for comparing and determining the variation in the sizes of components. (Fig 1)

Fig 1



These instruments cannot give the direct reading of the sizes like micrometers and vernier calipers. A dial test indicator magnifies small variations in sizes by means of a mechanism & through a pointer on a graduated dial. This direct reading of the deviations gives an accurate picture of the conditions of the parts being tested.

Principle of working

The principle of a dial test indicator is the magnification of a small movement of the plunger by converting it into rotary motion of a pointer on a circular scale. (Fig 2)

For converting linear motion of the plunger into rotary motion of the pointer, a rack and pinion mechanism is used.

Types

Two types of dial test indicators are in use according to the method of magnification.

- Plunger type (Fig 3)
- Lever type (Fig 4)

Constructional features

An important feature of the dial test indicator is that the scale can be rotated by a ring bezel, enabling it to be readily set to zero.

Fig 2

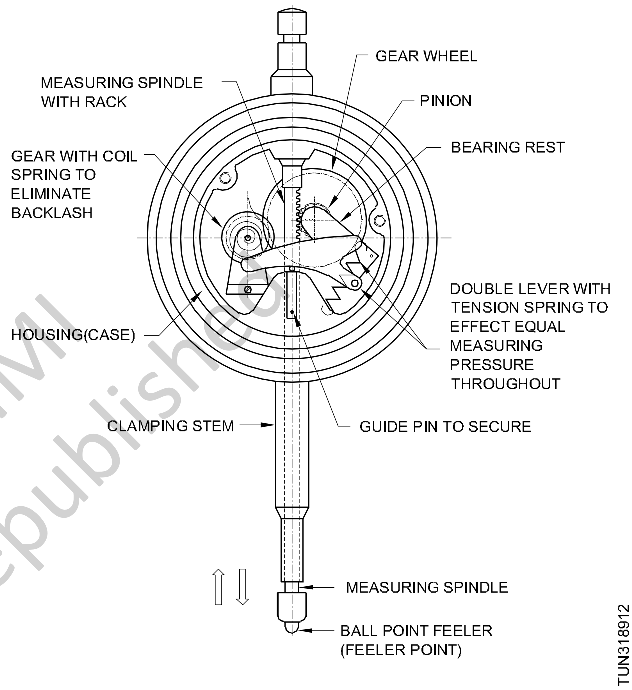
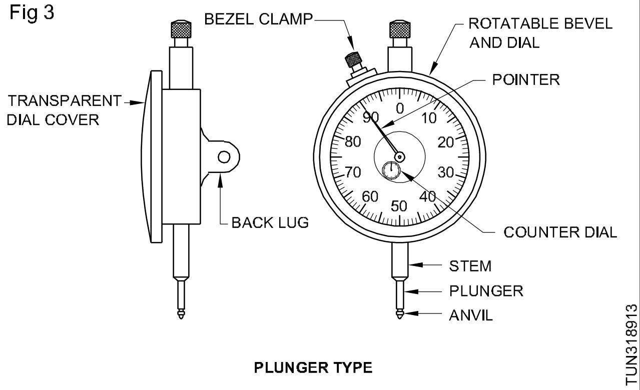
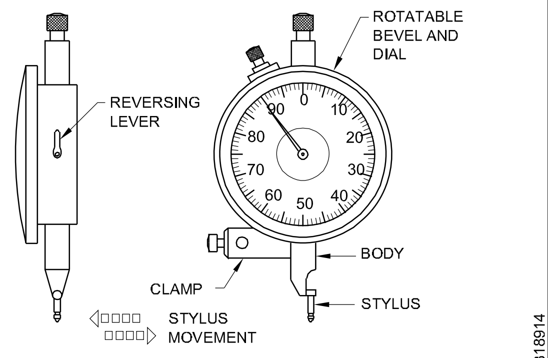


Fig 3



Many dial test indicators read plus in a clockwise direction from zero and minus in a counter clockwise direction to give plus and minus indications.

Fig 4

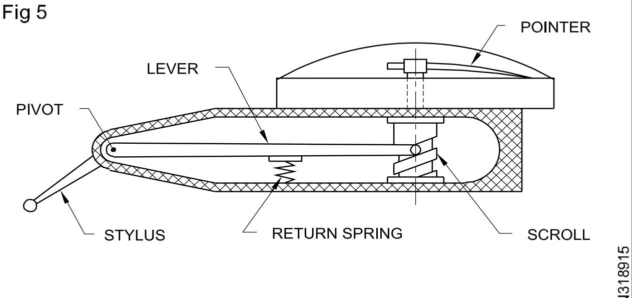


LEVER TYPE

The lever type dial test indicator

In the case of this type of dial test indicators the magnification of the movement is obtained by a mechanism of lever and scroll. (Fig 5)

Fig 5

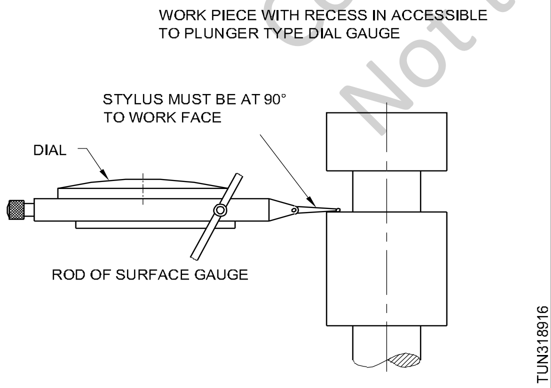


MECHANISM OF THE LEVER TYPE DIAL GAUGE

It has a stylus with a ball type contact operating in the horizontal plane.

This can be conveniently mounted on a surface gauge stand and can be used in places where the plunger type dial test indicator application is difficult. (Fig 6)

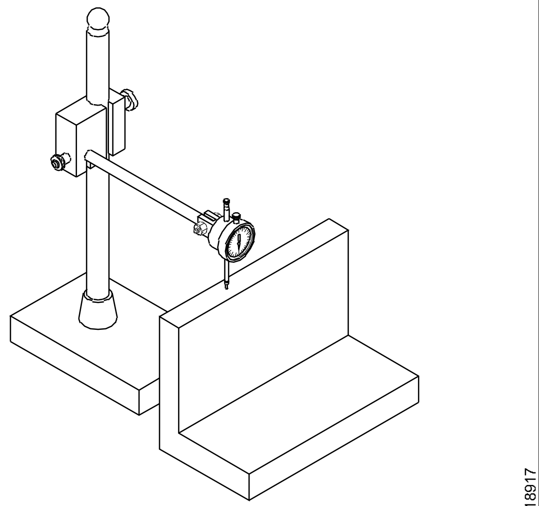
Fig 6



Uses (Figs 7 to 11)

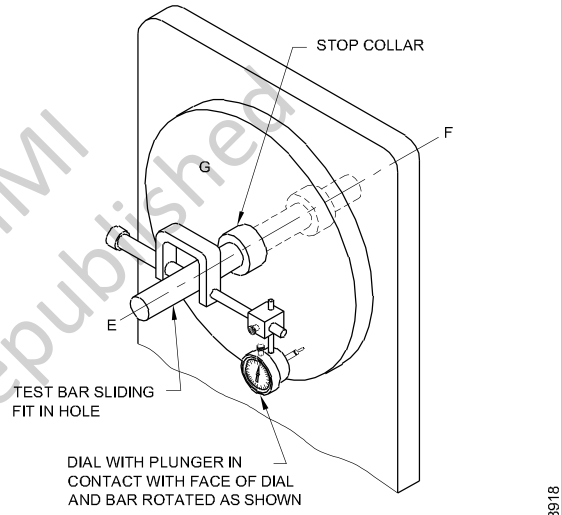
- To compare the dimensions of a workpiece against a known standard.
- To check plane surfaces for parallelism and flatness.
- To check parallelism of shafts and bars.
- To check concentricity of holes and shafts.
- To check the radial/axial runouts

Fig 7



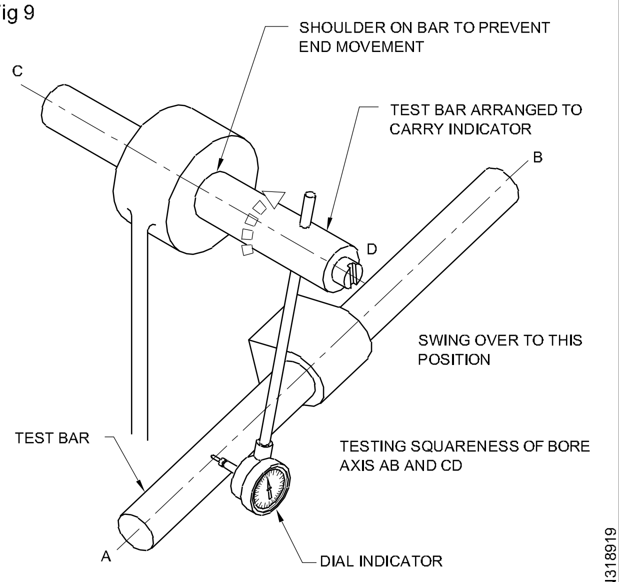
CHECKING FOR PARALLELISM

Fig 8



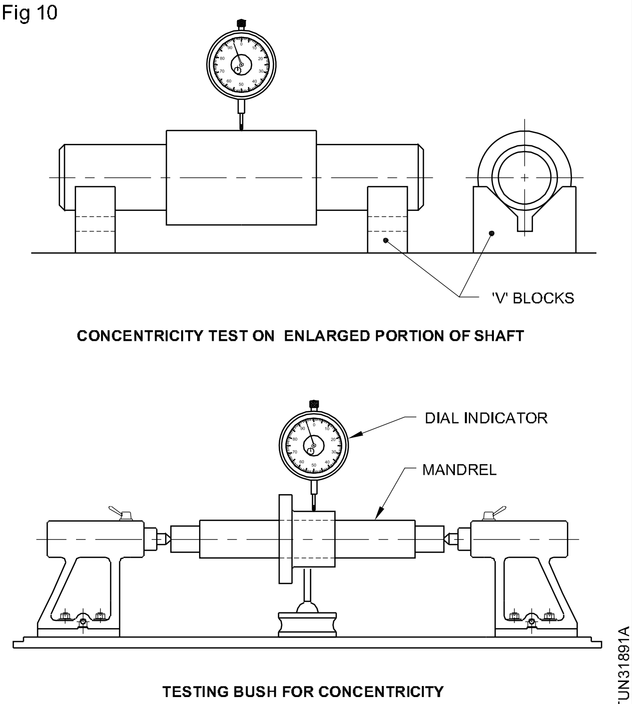
TESTING SQUARENESS OF FACE G WITH AXIS EF OF CENTRE HOLE

Fig 9



TEST SQUARENESS OF BORE AXIS USING TEST BAR

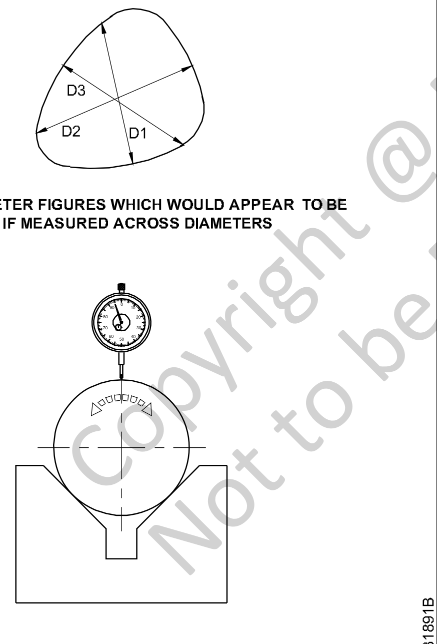
Fig 10



CONCENTRICITY TEST ON ENLARGED PORTION OF SHAFT

TESTING BUSH FOR CONCENTRICITY

Fig 11



CONSTANT DIAMETER FIGURES WHICH WOULD APPEAR TO BE ROUND IF MEASURED ACROSS DIAMETERS

TEST FOR ROUNDNESS WITH VEE BLOCKS AND CLOCK INDICATOR

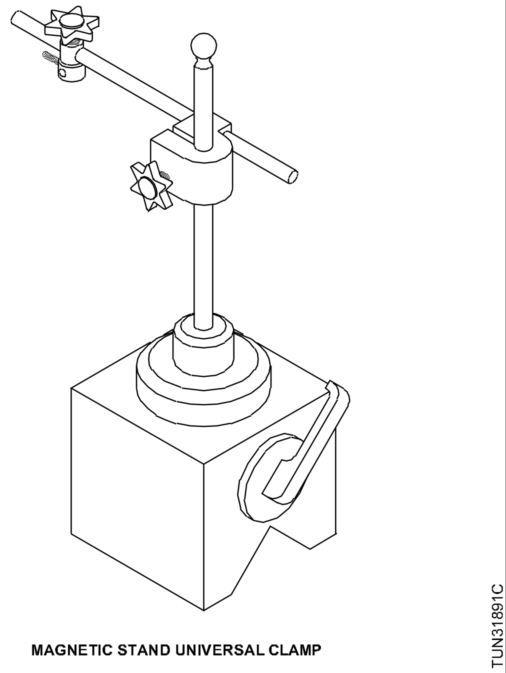
Stands

Dial test indicators are used in conjunction with stands for holding them so that the stand itself may be placed on the datum surface or machine tools.

The following are the three types of stands available.

Magnetic stand with universal clamp. (Fig 12)

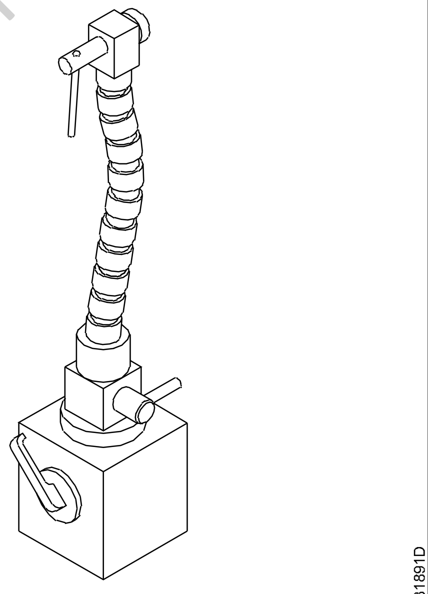
Fig 12



MAGNETIC STAND UNIVERSAL CLAMP

Magnetic stand with flexible post. (Fig 13)

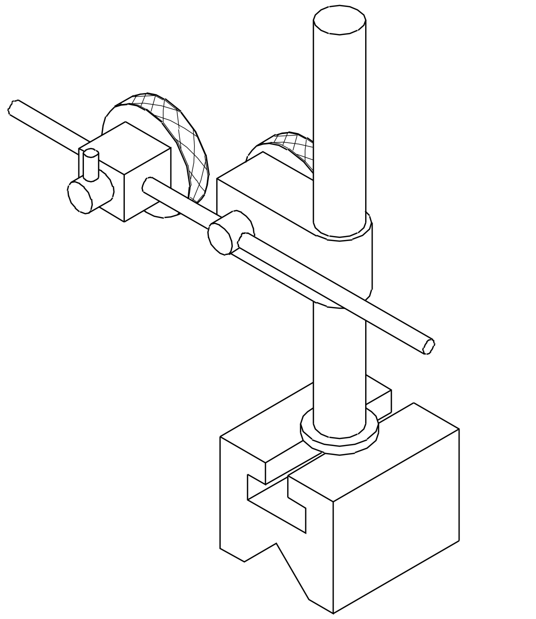
Fig 13



MAGNETIC STAND WITH FLEXIBLE POST

General purpose holder with cast iron base (Fig 14).

Fig 14



GENERAL PURPOSE HOLDER WITH CAST IRON BASE

Jigs & fixtures - types and uses

Objectives: At the end of this lesson you shall be able to
- **state the advantages of using jigs & fixtures**
- **state what is a jig and a fixture**
- **list the different types of jigs & fixtures**
- **state the features of a turning fixure.**

Definition - type - uses

Jigs and fixtures are the production tools used to manufacture duplicate parts accurately. (Figs 1 and 2)

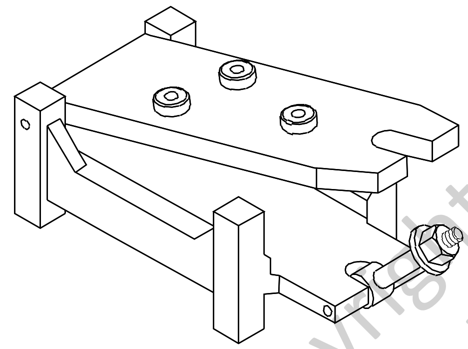
Advantages of using jigs and fixtures

Faster rate of production.

Easy to use even by an unskilled worker.

Layout and marking on individual parts eliminated.

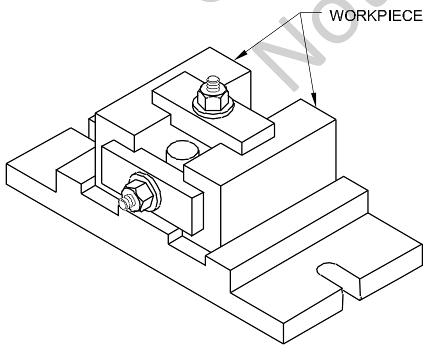
Fig 1



DRILL JIG

TUN319011

Fig 2



MILLING FIXTURE

TUN319012

Definition of a jig

A jig is a special device which holds, supports, locates and also guides the cutting tool during operation.

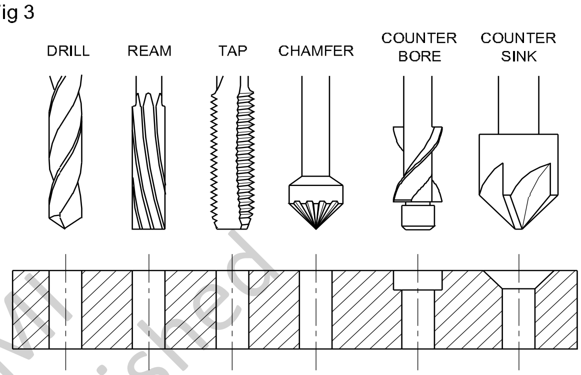
The two types of jigs are

– drilling jig

– boring jig.

Drilling jigs are used to drill, ream, tap and for other allied operations as shown on the component. (Fig 3)

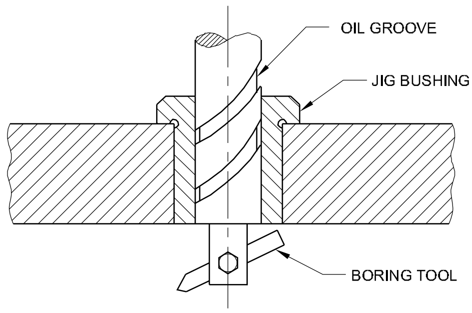
Fig 3



TUN319013

Boring jigs are used to bore holes which are either too large to drill or of odd size. (Fig 4)

Fig 4

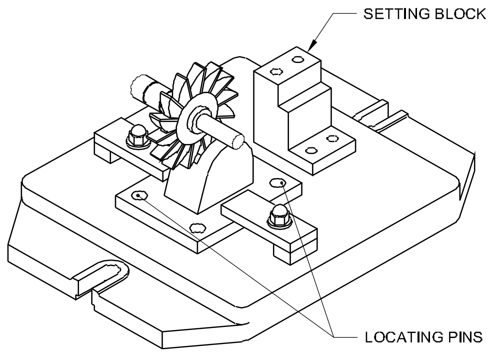


TUN319014

What is a fixture?

A fixture is a production tool that locates and holds the piece part. It does not control the cutting tools, but the tools can be positioned before cutting with the help of setting blocks and feeler gauges, etc. (Fig 5)

Fig 5



TUN319015

The commonly used fixtures are

- milling fixture
- turning fixture
- grinding fixture
- welding and assembly fixture, bending fixture, etc.

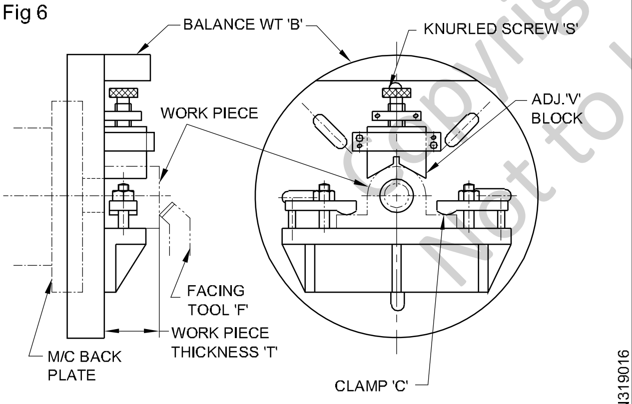
Features of turning fixtures

Some workpieces require special turning fixtures for quick location and clamping. These are generally special face-plates. Their swing should be lesser than the swing of the lathe machine. The clamp arrangement should be capable of withstanding the various forces developed during operations (i.e) cutting force tangential to cutting circle, axial and radial forces due to feed of the tool and bending forces due to the pressure of the tool on the workpiece.

Construction of boring and facing fixture (Fig 6)

The workpiece rests on the angle plate face and its boss is centralized with the machine axis by a sliding 'V' block which can be operated with knurled screw 'S'. The workpiece is clamped in position by two clamps 'C'. The height of the angle plate, sliding 'V' and other parts are kept less than the workpiece thickness 'T' to prevent obstruction to facing tool 'F'. The workpiece is bored through and one side of its hole is faced on this fixture. The eccentric masses due to the workpiece, angle plate and clamps are counter balanced by the balance weight 'B'.

Fig 6

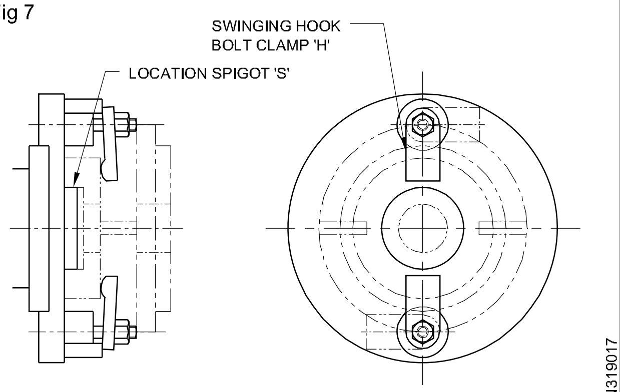


BORING AND FACING FIXTURE

Turning fixture (Fig 7)

The workpiece is located on the earlier machined spigot 'S' and clamped against the fixture face by two swing-ing hook bolt clamps 'H'. The clamps are loosened, and they swing anticlockwise to the position shown by chain-dotted lines to clear the path of the workpiece during loading and unloading. Because of even and symmetrical distribution of mass around the centre line of the fixture, no balance weights are necessary.

Fig 7



SECOND OPERATION TURNING FIXTURE

Classification of drill jig

Drill jigs are broadly classified as

- plate jig and channel jig
- solid jig
- post jig
- sandwich jig
- table jig
- box jig
- trunnion jig
- latch jig etc.

Selection of a particular type of jig will be based on the place where the drilling or its allied operation/operations are to be performed and the shape of the piece part.

Plate jig and channel

A jig consists simply of a drill plate which rests on the component to be drilled. For correct location, locating pins are clamped in position. At times on heavier piece parts even clamps will not be used. Generally a base plate will not be available for these types of jigs. (Figs 1, 2 & 3)

Fig 1

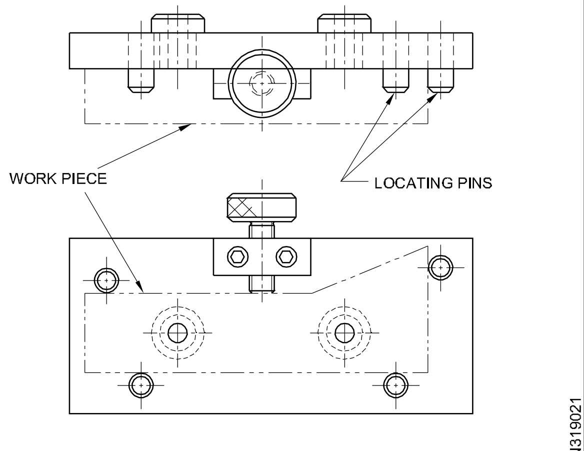


PLATE JIG

Fig 2

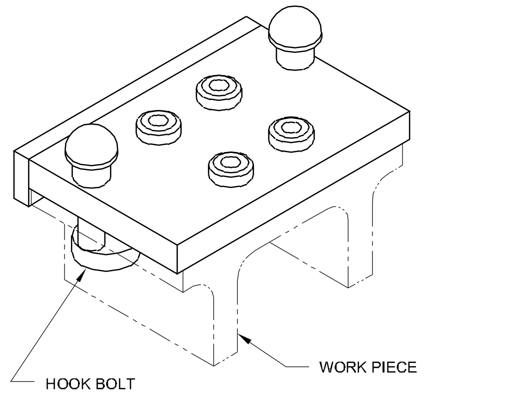
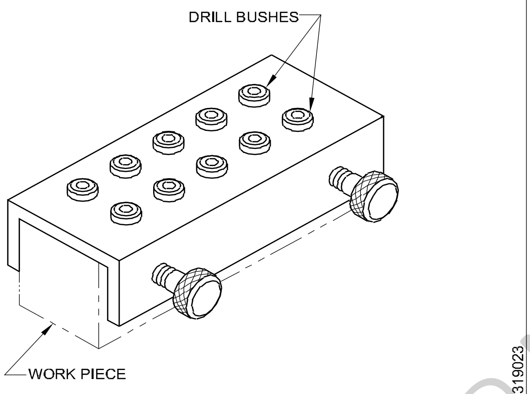


PLATE JIG

Fig 3

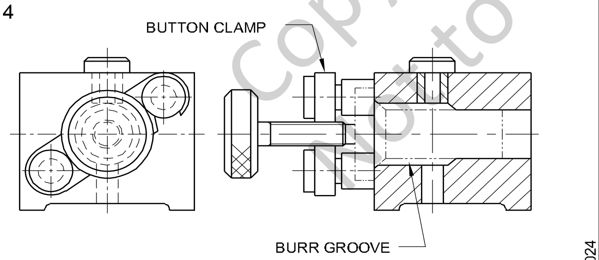


CHANNEL JIG

Solid jigs

These can be used while drilling on small piece parts. The body of these types of jigs is machined from solid blocks of steel. (Fig 4)

Fig 4



SOLID JIG

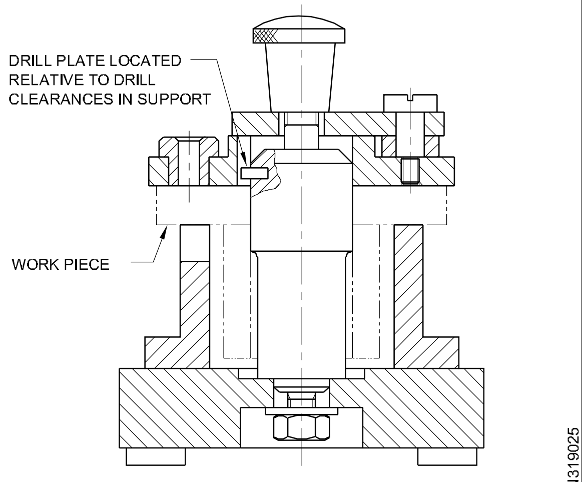
Post jigs

These are used for location from a bore. The post should be as short as possible to facilitate loading, and at the same time it must be long enough to support the work- piece. (Fig 5)

Sandwich jigs

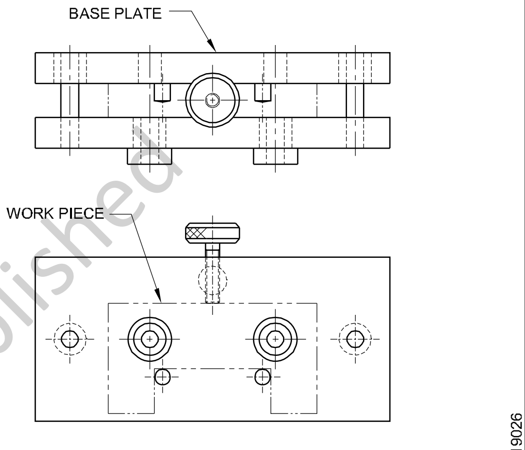
These are ideal for thin or soft piece parts which could bend or warp while machining. In this type of jigs, the piece part will be sandwiched between the base plate and the drill plate. The drill plate has to be located from a separate locator. (Fig 6)

Fig 5



POST JIG

Fig 6



SANDWICH JIG

Table jig (turn over jig)

This is used when it is necessary to locate the piece part from its face. For accurate seating of the jig on the machine table four legs will be provided on this type of jig. (Fig 7)

Fig 7

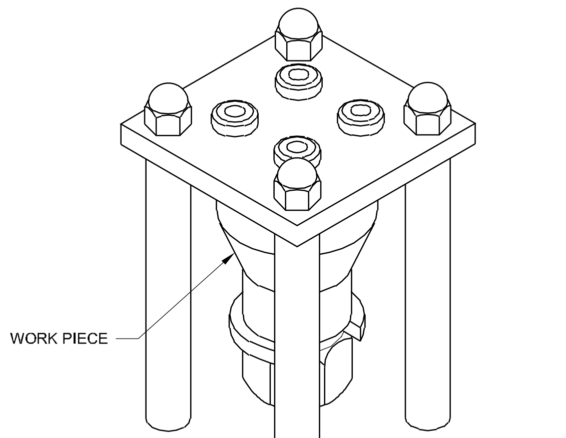
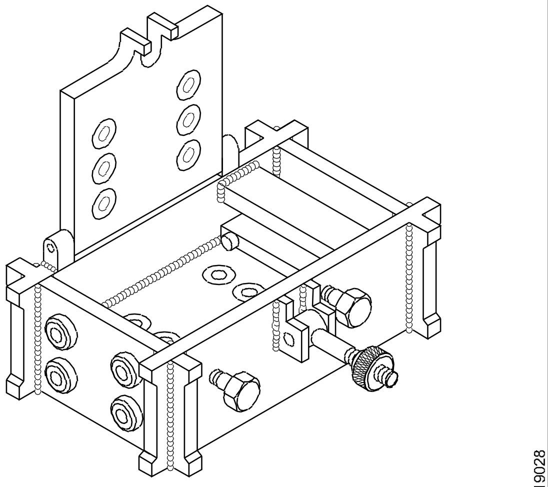


TABLE OR TURN OVER JIG

Box jig

This is made in the form of a box or frame work. The piece part will be located and clamped at one position when the piece part is to be drilled in many directions. This jig is meant for small piece parts only. (Fig 8)

Fig 8

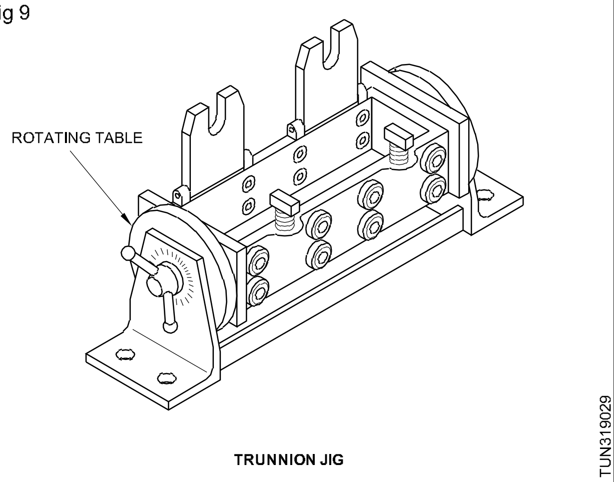


BOX JIG

Trunnion jig

This can be designed when a large or awkwardly shaped piece part is to be drilled in many directions. This is an extension of the box jig which is carried on trunnions and rotated from station to station and positioned using an indexing device. (Fig 9)

Fig 9

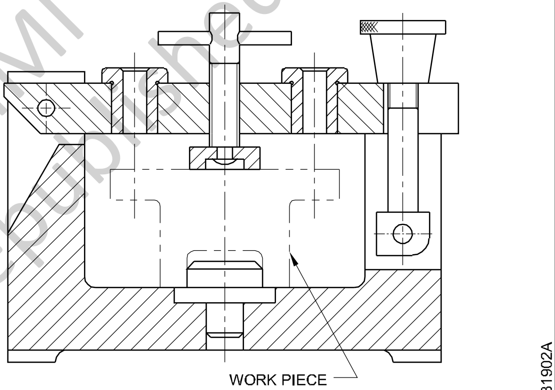


TRUNNION JIG

Latch jig

This is provided with latch clamps for easy loading and unloading. It is important that the latch must be positively located and clamped so that the bush bores are exactly perpendicular to the piece. (Fig 10)

Fig 10



LATCH JIG

Cutting Tool Material

Objectives: At the end of this lesson you shall be able to
- **state the properties and uses of high carbon steel, HSS, stellite, carbide, ceramic and diamond**
- **state the properties of cemented carbide**
- **select the lathe with reference to the use of carbide.**

Review of cutting tool materials

Various types of materials are used for making tools, and tips, each one having its advantages and disadvantages. They are dealt with below.

High carbon steel (0.9% to 1.5% carbon)

This is useful for making cutting tools for light finishing cuts and for machining soft materials. It is quite tough but the cutting edge softens and wears quickly, due to the heat generated whilst cutting (at 250°C), and so a fairly slow speed must be used.

High speed steel (HSS)

Besides carbon, it contains tungsten, chromium, vanadium, molybdenum as alloying metals. It loses its hardness at 600°C. It is probably the most popular type of tool material. It is tough enough to withstand most cutting shocks, and retains its hardness at higher speeds than high carbon steel. It will cut most materials quite satisfactorily, and is useful for general purpose work.

Stellite

This is a rather brittle, non-ferrous, cast alloy comprising of cobalt, chromium, tungsten and carbon (1.8 to 2.5%) but it is very hard and withstands heat up to 1000°C. It is useful for machining hard, chilled castings and similar materials.

Cemented carbide

It is a compound of carbon, cobalt and tungsten or titanium or tantalum or niobium. It is the hardest cutting tool material normally used. It is capable of withstanding temperature even above 1000°C. Several grades of cemented carbide tools are available, each one of which suits a particular material. It is important to select the correct grade of tool for the material to be turned; if not, an inferior surface texture may result.

The tools are either tipped with cemented carbide, which is brazed on to a carbon steel shank or the tips may be of the throw away type.

Coated carbides

A thin coating (extremely thin layer of 5 to 7 microns) of titanium carbide is deposited over processed inserts. So a good toughness is combined with very high wear resistance in the inserts. In the same working condition, the cutting edge of a coated carbide insert may last for 3 to 4 times longer than that of a conventional carbide insert. Also 40% higher cutting speeds can be used.

Ceramics

The latest development in the metal cutting tools is the use of aluminium oxide, generally referred to as ceramics.

Ceramic tools are made of aluminium oxide powder in a mould. Ceramic tool materials are made in the form of tips that are to be clamped on metal shanks.

These tools have very low heat conductivity and extremely high compressive strength, but they are quite brittle and have a low bending strength. For this reason, these materials cannot be used for tools operating in interrupted cuts with vibrations, as well as for removing a heavy chip. But they can withstand temperatures up to 1200°C and can be used at cutting speeds 4 times that for cemented carbides and up to about 40 times that of high speed steel tools. Heat conductivity of ceramics being very low, the tools are generally used without a coolant.

Diamonds

The diamond is the hardest known tool material and can be run at cutting speeds about 50 times greater than that for HSS tool and at temperatures up to 1650° C. In addition to its hardness, diamond is incompressible, is of a large grain structure, and readily conducts heat and has a low coefficient of friction.

Diamonds are suitable for cutting very hard materials such as glass, plastics, ceramics and other abrasive materials and for producing fine finishes. The maximum depth of cut recommended is 0.125 mm with feeds of, say 0.05 mm.

To summarise, the two most commonly used tool materials are the high speed steel and the cemented carbide.

High speed steel tools may be used when

- working to great accuracy on small diameters
- turning small diameters, if the machine is not capable of running at a high r.p.m.
- screw cutting
- intermittent cutting.

Cemented carbide tools may be used when

- a fast and higher rate of metal removal is needed
- cutting hard and non-ferrous materials
- high speed thread cutting is involved.

Comparative cutting speeds

Recommended speeds will vary according to the following factors: the kind and hardness of material being cut, the rate of feed, the depth of cut, the finish desired, the rigidity of the machine, the rigidity of the work set up, the type of cutting tool and the type of cutting fluid used.

The lathe must be capable of running at high speeds, since much higher speeds are used with carbide tools. To obtain a good surface texture on small diameter work, the machine must be rigid and in good condition.

H.S.S. Perccentage

H.S.S. (High speed steel)

18% Tungsten

4% Chromium

1% Vanadium

0.70% Carbon

Use of oil types of work on cutting tool material.

Ex; Lathe tool, Planer, Shaper drill, Milling cutters.

Molybdenum HIgh Speed Steel

6% Molybdenum 5% tungsten 4% chromium 2% Vanadium

This above percentage of drilling very strong and with good cutting ability.

Cobalt High Speed Steel

12% Cobalt, 20% Tungsten, 4% Chromium, 2% Vanadium

The following example is shown to determine the maximum cutting speed for a given tool life.

Relative Properties Of Cutting Tool Materials

Reduction in	**Tool material**	**Increase in**
Impact rupture strength	High carbon steel	Initial cost of tool.
(Shock resistance)	High speed steel	Red hardness temps.
Resilience		Wear resistance.
(Tool springs back to shape	Cast alloys	Brittleness.
under vibration.)		Cutting speeds used.
		Tool life between
	Cemented carbide	sharpenings.
		Tool chipping with vibration.
	Ceramics	
	Diamonds	

Tool life and quality of a cutting tool material

Objectives: At the end of this lesson you shall be able to
- **state the relationship between cutting speed and tool life**
- **explain tool life index equation**
- **determine the maximum cutting speed for a given tool**
- **state the quality of cutting tool material**
- **distinguish the charteristics of cold hardness, redhardness and toughness**
- **state the factors for selecting tool material**

Table

Tool life index

Material and conditions	Tool material	n
3 1/2% nickel steel	Cemented carbide	0.2
3 1/2% nickel steel (roughing)	Highspeed steel	0.14
3 1/2% nickel steel (finishing)	Highspeed steel	0.125
High carbon, high chromium die steel	Cemented carbide	0.15
High carbon steel	Highspeed steel	0.2
Medium carbon steel	High-peed steel	0.15
Mild steel	Highspeed steel	0.125
Cast iron	Cemented carbide	0.1

The following exampe is shown to determine the maximum cutting speed for a given tool life.

Example

The life of a lathe tool is 8 hours when operating at a cutting speed of 40 m/min. Given that Vt^n = C, find the highest cutting speed that will give a tool lige of 16 hours.

The value f n is 0.125.

(i) Determine the value of Log C from initial conditions.

C = Vt_1^n where

V = 40 m/min

t^1 = 480 min

n = 0.125

Log C = Log V + n Log t1

= log 40 + (0.125 Log480)

= 1.6021 + (0.125 x 2.681)

= 1.6021 + 0.3351

= 1.9372

(ii) Determine Vmax for revised conditions

Vmax

Where t_2 = 960 min

Log V max = Log C - n Log t_2

= 1.9372 - (0.125 x log960)

= 1.9372 - (0.125 x 2.9823)

= 1.9372 - 0.3728

= 1.5644

Vmax = 36.68 m/min

Quality of a cutting tool materials

Tool materials

Metal cutting tool materials perform the function of cutting. These materials must be stronger and harder than the material to be cut. They must be sufficiently tough to resist shock loads that result during cutting operations. They must have good resistance to abrasion and a reasonable tool life.

The three most important basic qualities that any cutting tool material should possess are:

- cold hardness
- red hardness
- toughness.

Cold hardness

It is the amount of hardness possessed by a material at normal temperature. Hardness is the property possessed by a material by which it can cut other metals, and has the ability to scratch on other metals.

When hardness increases, brittleness also increases, and a material which is having too much of cold hardness is not suitable for the manufacture of cutting tools.

Red hardness

It is the ability of a tool material to retain most of its cold hardness even at very high temperature. During machining, due to friction between tool and work, tool and chip, heat is generated, and the tool loses its hardness, and its efficiency to cut diminishes. If a tool maintains its cutting efficiency even when the temperature during cutting increases, then that metal possesses the property of red hardness.

Toughness

The property possessed by a material to resist sudden load that results during metal cutting is termed as 'toughness'. This will avoid the breakage of the cutting edge.

Points to be noted when selecting a tool material

- Condition and form of material to be machined.
- Material to be machined.
- Condition of the machine tool available.
- The total quantity of production and the rate of production involved.
- The dimensional accuracy required and the quality of surface finish.
- The amount of coolant applied and the method of application.
- The skill of the operator.

Tool life

The life of a cutting tool depends on so may factors like material hardness, toughness coolent used recomended cutting speed and feed. The table below indicates tool life index of various cutting tool.

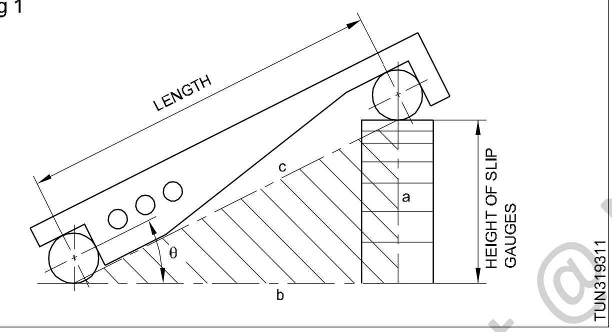
Checking of taper with Sine bar and Roller

Objectives: At the end of this lesson you shall be able to
- **check the correctness of the known angle of the work**
- **calculate the height of slip gauges to build up the height for a given angle**
- **name the features of a taper which can be measured using precision rollers and slip gauges**
- **state the formula for measuring the angle of the taper**
- **calculate the angle of the taper.**

A sine bar provides a simple means of checking angles to a high degree of accuracy.

The use of a sine bar is based on the trigonometrical function. The sine bar forms the hypotenuse of that triangle and the slip gauge height forms the opposite side of the angle Fig 1

Fig 1

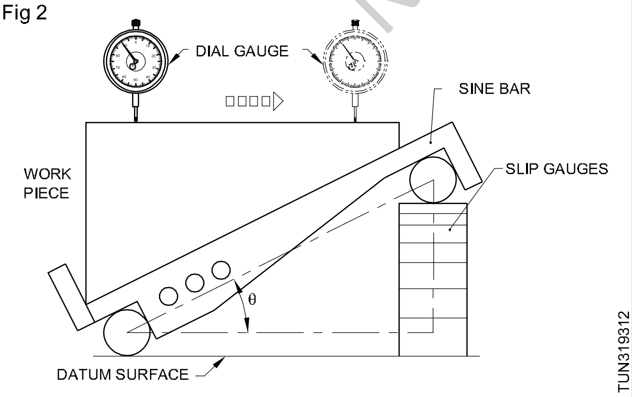


Checking the corrrectness of a known angle

For this purpose first choose the correct slip gauge combination for the angle to be checked.

The component to be checked should be mounted on the sine bar after placing the selected slip gauges under one roller, with the other resting on the datum surface Fig 2.

Fig 2



A dial test indicator is mounted on a suitable stand or vernier height gauge Fig 2. The dial test indicator is then set in first position as shown in the figure, and the dial is set to zero. Move th dial indicator to the other end of the component (second position). If there is any difference then the angle is incorrect. The height of the slip gauge pack can be adjusted until the dial test indicator reads the same reading at both ends. The actual angle can then be calculated and the deviation if any, will be the error.

Method of calculating the slip gauge height.

$$Sine\theta = \frac{a}{c} = \frac{84.52}{200}$$

$$Sine\theta = 0.4226$$

The value of sine of the angle is 0.4226

the angle = 25°.

Examples

1 What will be the angle of the workpiece is the slip gauge pack height is 17.36 mm and the size of the sine bar used is 100 mm? Fig 5

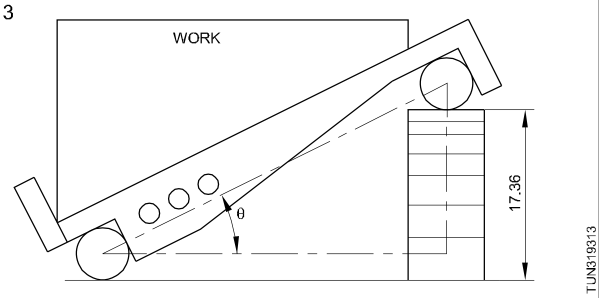
$$Sine\theta = \frac{a}{c} = \frac{17.36}{100}$$

$$= 0.1736$$

$$\theta = 10^0$$

To determine the height of slip gauges for an angle of 25 ° using a sine bar of 200 mm long Fig 3.

Fig 3



$$Sine\theta \frac{a}{c}$$

$$\theta = 25$$

$$a = C \sin e\theta$$

$$= 200x0.4226$$

$$= 84.52mm.$$

The height of the slip required is 84.52 mm.

Note

The value of sine θ can be seen form mathematical tables. (Natural Sine)

Use always accurate tables while working out sine bar constants for standard lengths of sine bars.

Calculating the anlge of tapered components.

The height of the slip gauge used is 84.52 mm. The length of the sine bar used is 200 mm.

What will be the angle of the component? Fig 4.

Fig 4

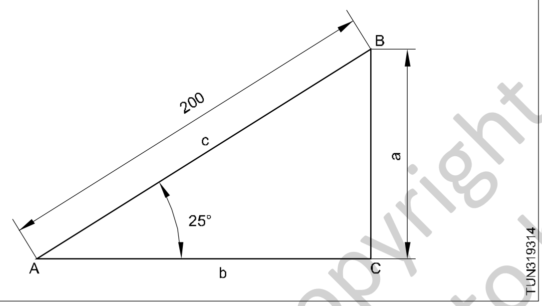
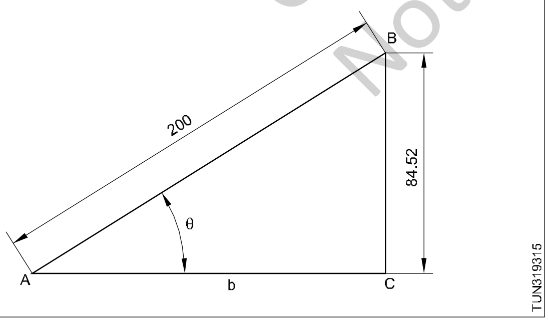


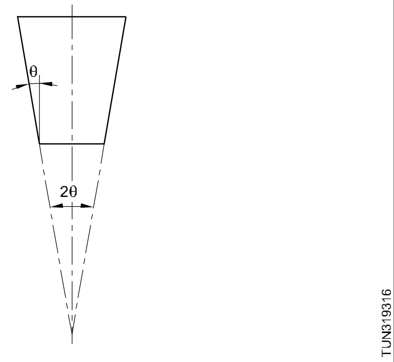
Fig 5



A method used for checking the dimensions of the tapered components is by using precision rollers or balls along with the slip gauges. Using this method the following elements of the tapers can be checked.

Angle of the taper Fig 1.

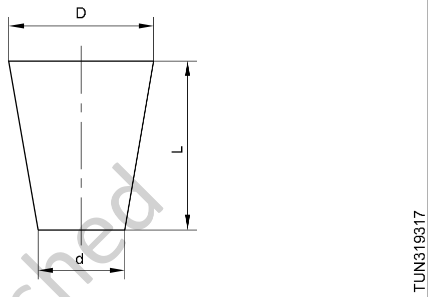
Fig 6



Small end diameter Fig 2.

Large end diameter Fig 2.

Fig 7



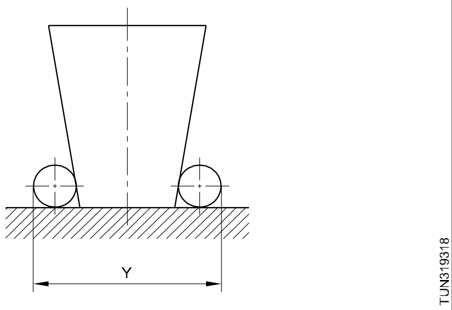
Check long the angle of the taper

For determining this angle two measurements are taken.

i.e. X and Y.

The meaurement Y is taken by placing the component against a datum surface like the surface palte or the marking table. Two precision rollers are then placed at the smaller end resting on the datum surface and contacting the workpeice Fig 3.

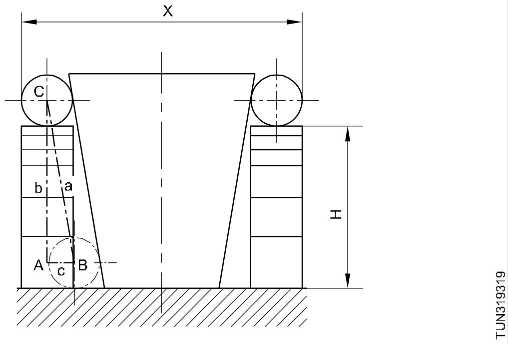
Fig 8



meaurement 'X' is taken by lifting and placing the rollers on both sides with the help of two sets of slip packs having the same size.

The measurement is then taken with a micrometer over the rollers Fig 4.

Fig 9



For Computing the taper angle the following trigonometrical ratio is applied.

$$Tangent\,\theta = \frac{AB}{AC}$$

From the two meaurements taken and the height of the slip packs the ratio is established by subtracting the measurement 'Y' from 'X' and dividing it by two. This corresponds to the distance AB.

The length AC corresponds to the size of the slip pack used on one side.

$$AB = \frac{x-y}{2}$$

Then the tangent of the taper angle is

$$Tan\,\theta = \frac{AB}{AC} = \frac{x-y}{2H}$$

Where X is the measurement ove rthe rollers placed on the slip gauge height, Y is the meaurement over the rollers at the smaller end and H is the slip gauge height.

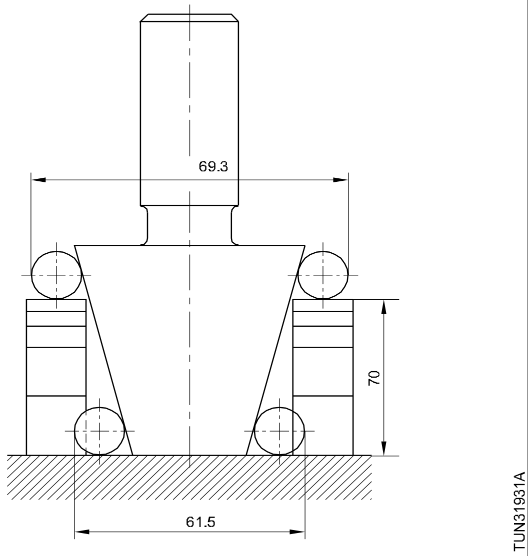
The included angle of the taper will be double the above angle.

Example

Calculate the included angle of the tapered component shown in Fig 5.

The measurement

Fig 10



$$X = 69.3mm$$
$$Y = 61.5mm$$
$$Height = 70mm$$
$$Tan\theta = (69.3) - (61.5) / 2x70$$
$$= 7.8 / 2x70$$
$$= \frac{3.9}{70} = \frac{0.39}{7} = 0.0557$$

Referring to the log table under Natural Tangents we fine θ = 3° 11'.

Hence included angle of the taper

$$2\theta = 3^0 11' x2 = 6^0 22'$$
$$2\theta = 6^0 22'.$$

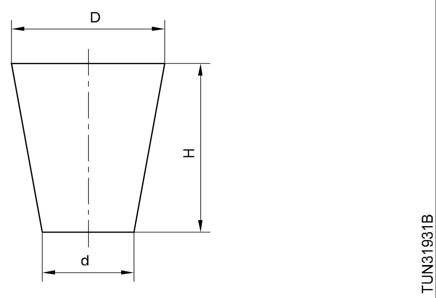
Diameters at any position of tapered components can be determined when the angle of taper is known.

For inspection of tapered components for dimensional quality the following diameters are measured.

Small end diameter d Fig 1.

Large end diameter D Fig 1.

Fig 11



Determining small end diameter Fig 2

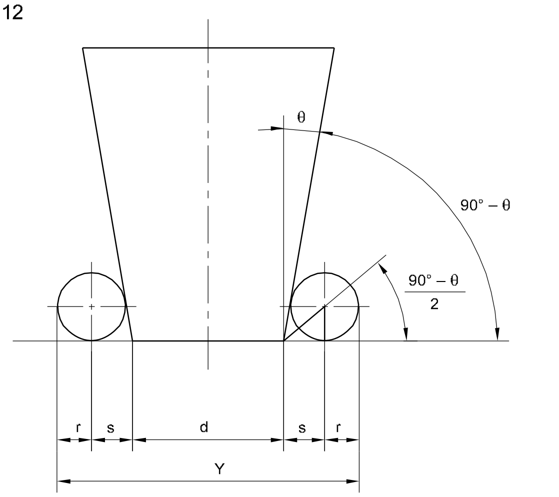
The small diameter 'd' is = Y - 2 (S + r).

Y - is the diameter over the two precision rollers.

r - is the radius of the roller.

S - is the distance from the centre of the roller to the end fo the component

Fig 12

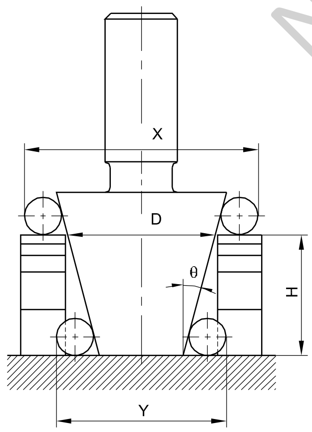


Calculating S Fig 3

$$Tan\left\{\frac{90-\theta}{2}\right\} = \frac{r}{s}$$

$$s = \frac{r}{Tan\left\{\frac{90-\theta}{2}\right\}}$$

Fig 13



$$d = Y - 2\left[\frac{r}{Tan\left\{\frac{90-\theta}{2}\right\}} + r\right]$$

$$= Y - 2r\left[Cot\left\{\frac{90-\theta}{2}\right\}\right] + 1$$

Example

0 = 3° 11"

Y = 61.5 mm

r = (radius of roller) 6 mm

$$Thend = 61.5 - 12\left[Cot\left\{\frac{90-3^0 11'}{2}\right\}\right] + 1$$

= 61.5 - 12 (1.0570+1)

= 61.5 - 12 x 2.0570

= 61.5 - 24.6840 = 36.3160 mm

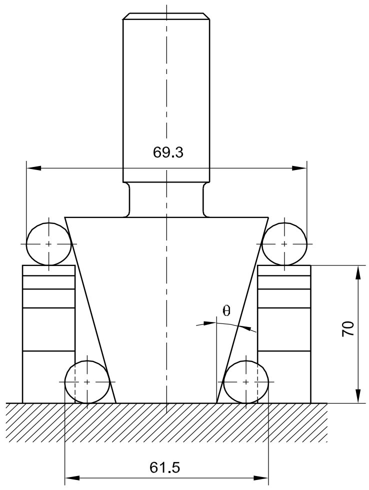
Determining the large diameter of taoer at any desired height (H for example)

The formula is derived by taking into consideration the meaurement over the rollers palced at a known height 'H', the diameter of the roller and the angle of taper. The diameter 'D' at larger end at height 'H'.

= X - 2 (S + r)

Example Fig 4

Fig 14



$\theta = 3° 11'$

X = 69.3 mm

H = 70 mm

r = (radius of the roller) 6 mm

Then the diameter of the taper at height H from the small end.

= 69.3 - 12(1+1.0570)

= 69.3 - 24.6840 = 44.6160 mm

The length of the taper can be directly meaured by using a vernier height gauge. Then the largest diameter of the taper is determined by computing the known values.

If 'M' is the maximum diameter of the taper, 'T' is the minimum diameter of the taper and L is the tapered length.

then $M = T + 2L \times \tan \theta$

Cutting speed and feed, turning time, depth of cut calculation

Objectives: At the end of this lesson you shall be able to
- **distinguish between cutting speed and feed**
- **read and select the recommended cutting speed for different materials from the chart**
- **point out the factors governing the cutting speed and calculation of machining time**
- **state the factors governing feed.**
- **understand features of carbide tip tools.**

Cutting speed (Fig 1)

The speed at which the cutting edge passes over the material, which is expressed in metres per minute is called the cutting speed. When a work of a diameter 'D' is turned in one revolution the length of portion of the work in contact with the tool is p x D. When the work is making 'n' rev/min, the length of the work in contact with the tool is p x D x N. This is converted into metres and is expressed in a formula form as

$$V = \frac{\pi \times D \times N}{1000} \text{ metre/min.}$$

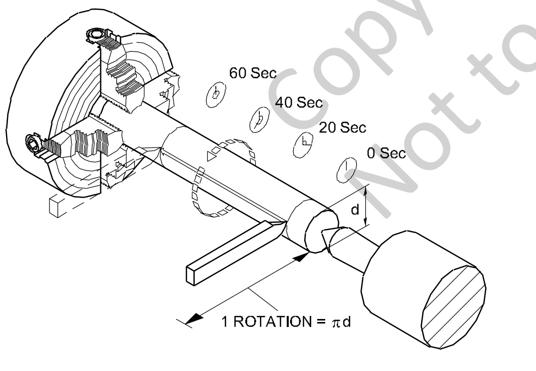
Where V = cutting speed in metre/min

p = 3.14

D = diameter of the work in mm.

N = r.p.m.

Fig 1



CUTTING SPEED FOR TURNING OPERATION

When more material is to be removed in lesser time, a higher cutting speed is needed. This makes the spindle to run faster but the life of the tool will be reduced due to more heat being developed. Recommended cutting speeds are given in a chart form which provides normal tool life under normal working conditions. As far as possible the recommended cutting speeds are to be chosen and the spindle speed calculated before performing the operation. (Fig 2)

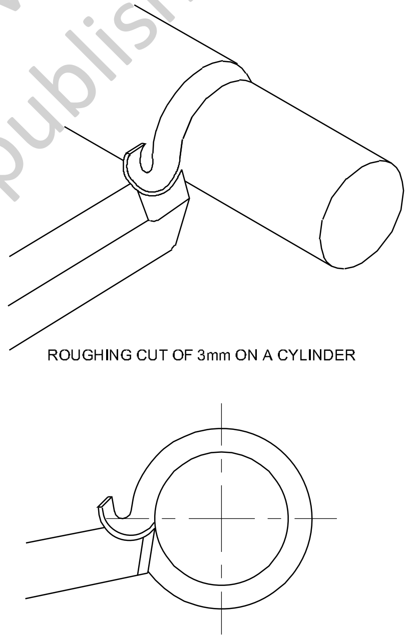
Example

Find out the rpm of the spindle for a 50 mm bar to cut at 25 m/min.

$$V = \frac{\pi DN}{1000} \qquad N = \frac{1000V}{\pi d}$$

$$\frac{1000 \times 25}{3.14 \times 50} = \frac{500}{3.14} = 159 \text{ r.p.m.}$$

Fig 2



DEPTH OF CUT

Factors governing the cutting speed

Finish required

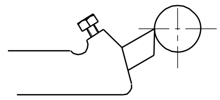
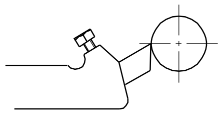
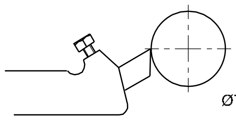
Depth of cut

Tool geometry

Properties and rigidity of the cutting tool and its mounting

Properties of the workpiece material

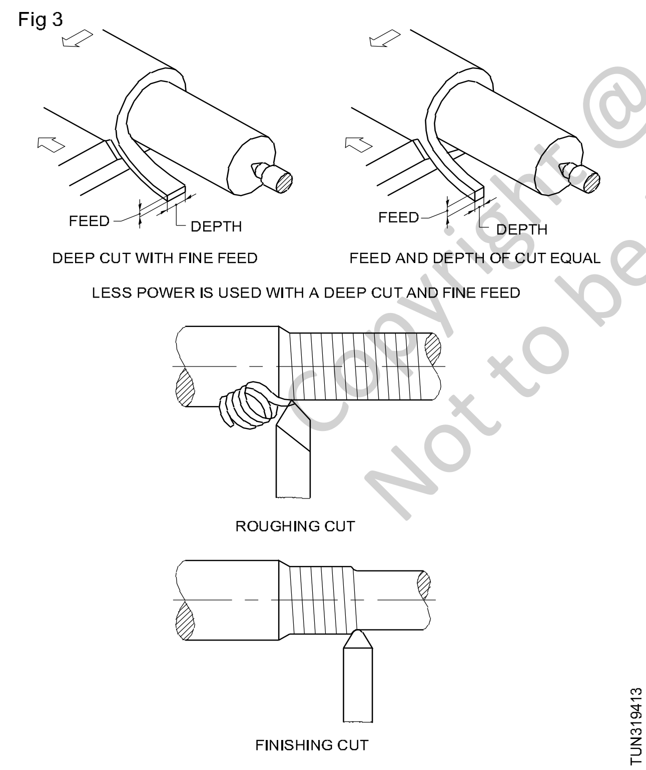
Rigidity of the workpiece

Cutting speed 120m/min	Length of metal passing cutting tool in I revolution	Calculated r.p.m. of spindle
	78.56 mm	1528
	157.12 mm	756
	235.68 mm	509.3

Type of cutting fluid used

Rigidity of the machine tool

Feed (Fig 3)



The feed of the tool is the distance it moves along the work for each revolution of the work, and it is expressed in mm/rev.

Factors governing feed

Tool geometry

Surface finish required on the work

Rigidity of the tool

Coolant used.

Rate of metal removal

The volume of metal removal is the volume of chip that is removed from the work in one minute, and is found by multiplying the cutting speed, feed rate and the depth of cut.

CUTTING SPEEDS AND FEEDS FOR H.S.S. TOOLS

Table

Material being turned	Feed mm/rev	Cutting speed m/min
Aluminium	0.2-1.00	70-100
Brass (alpha)-ductile	0.2-1.00	50-80
Brass (free cutting)	0.2-1.5	70-100
Bronze (phosphor)	0.2-1.00	35-70
Cast iron (grey)	0.15-0.7	25-40
Copper	0.2-1.00	35-70
Steel (mild)	0.2-1.00	35-50
Steel (medium-carbon)	0.15-0.7	30-35
Steel (alloy high tensile)	0.08-0.3	5-10
Thermosetting plastics	0.2-1.00	35-50

Note

For super HSS tools the feeds would remain the same, but cutting speeds could be increased by 15% to 20%.

A lower speed range is suitable for heavy, rough cuts.

A higher speed range is suitable for light, finishing cuts.

The feed is selected to suit the finish required and the rate of metal removal.

When carbide tools are used, 3 to 4 times higher cutting speed than that of the H.S.S. tools may be chosen.

Calculation involving cutting speed and feed

Objectives: At the end of this lesson you shall be able to

- **determine the spindle speed for turning jobs of different materials of different diameters with different tool materials**
- **determine the turning time with the given data.**

The selection of the spindle speed is one of the factors which decides the efficiency of cutting. It depends on the size of the job, material of the job and material of the cutting tool. The formula to determine cutting speed is

$$Cs = \frac{\pi \times D \times N}{1000} \text{ metre/min. where D is in mm.}$$

To determine the spindle speed(N)

$$N = \frac{CS \times 1000}{\pi \times D}$$

Example 1

Calculate the spindle speed to turn a MS rod of Ø 40 mm.

Using HSS tool data in the above problem, since the material is mild steel and tool is HSS, the recommended cutting speed from the chart is 30 m min.

Ø = 40 mm

$$N = \frac{CS \times 1000}{\pi \times D}$$

$$= \frac{30 \times 1000 \times 7}{22/7 \times 40}$$

$$= \frac{30 \times 1000 \times 7}{22 \times 40}$$

$$= \frac{30 \times 25 \times 7}{22}$$

= 238.6 r.p.m.

The spindle speed should be set nearest to the calculated r.p.m., on the lower side.

Example 2

Determine the spindle speed to be set for a hard cast iron round rod of Ø 40 mm using a HSS tool.

DATA: The cutting speed for hard cast iron from the chart is 15 m/min.

Ø = 40 mm

$$N = \frac{CS \times 1000}{\pi \times D}$$

$$= \frac{15 \times 1000 \times 7}{22/7 \times 40}$$

$$= \frac{15 \times 1000 \times 7}{22 \times 40}$$

$$= \frac{15 \times 25 \times 7}{22}$$

= 119.3 r.p.m.

The spindle speed should be set nearest to the calculated r.p.m., on the lower side.

Example 3

Calculate the spindle speed to turn a Ø40 mm MS rod using a cemented carbide tool.

DATA: The cutting speed recommended for turning mild steel using a carbide tool is 92 mtr/minute.

Ø of job = 40 mm

$$N = \frac{CS \times 1000}{\pi \times D}$$

$$= \frac{92 \times 1000 \times 7}{22/7 \times 40}$$

$$= \frac{92 \times 1000 \times 7}{22 \times 40}$$

$$= \frac{92 \times 25 \times 7}{22}$$

= 731.8 r.p.m.

The spindle speed should be set to the nearest calculated r.p.m..

Turning time calculation

The time factor is very important to decide the manufacturing of the component as well as to fix the incentives to the operator. If the spindle speed, feed and length of the cut are known, the time can be determined for a given cut. If the feed is 'f'and length of cut is 'l', then the total number of revolutions the job has to make for a cut is l/f.

If N is the rpm, the time required for a cut is found by

$$\text{Time to turn} = \frac{\text{Length of cut} \times \text{No. of cuts}}{\text{feed} \times \text{r.p.m}}$$

$$T = \frac{l \times n}{f \times N}$$

where 'n' is the number of cuts and 'N' is the r.p.m../

Example 1

A mild steel of Ø 40 mm and 100 mm length has to be turned to Ø 30 mm in one cut for full length using a HSS tool with a feed rate of 0.2 mm/rev. Determine the turning time.

$$\text{Turning time} = \frac{l \times n}{f \times N}$$

The r.p.m. for the above is calculated and found out as 238.6 r.p.m..

l = 100 mm

f = 0.2 mm

n = 1

N = 238.6 r.p.m.

$$\text{Time} = \frac{100 \times 1}{0.2 \times 238.6}$$

= 2.09 minutes

= 2 minute 5.4 seconds.

Calculation of the total machining time

Machining time

The machining time in a lathe work can be calculated for particular operation if the speed of the job, feed and length of the job is given.

Time taken for a complete cut

$$= \frac{l}{s \times n} \text{ minutes}$$

If 's' is the feed of the job expressed in mm per revolution and 'L' is the length of the job in mm, then number of revolutions of the job required for a complete cut will be: L ÷ s.

If the r.p.m. of the work is n, time taken to revolve the job through L/s number of revolutions for a complete cut will be

$$= \frac{l}{s \times n} \text{ minutes}$$

Example

Find the time required for one complete cut on a piece of work 350 mm long and 50 mm in diameter. The cutting speed is 35 metres per minute and the feed is 0.5 mm per revolution.

$$\text{Cutting speeds} = \frac{\pi DN}{1000} = \frac{\pi \times 50 \times n}{1000}$$

$$\text{or } n = \frac{1000 \times 35}{\pi \times 50} = 222.5$$

Number of revolutions required to cut the full length with the given feed $\frac{350}{0.5} = 700$

Time required for one complete cut

$$= \frac{700}{222.5} = 3.14 \text{ minutes}$$

Example

Find the machining time required for 2 rough cuts and 1 finish cut by carbide tool on a workpiece of Ø 80 mm, to a length of 350 mm. First depth of cut 7.5 mm, second depth of cut 5 mm and last finish cut 2.5 mm and feed is 0.05 mm per revolution.

Cutting speeds 120 m/min. 130 m/min. 140 m/min. respectively.

N = Rpm of 120 m/min.

$$N_1 = \frac{CS \times 100}{\pi \times D} = \frac{120 \times 1000}{3.14 \times 80} = 477.7 \text{ Rpm}$$

T_1 = First rough cut machining time

$$\frac{\ell}{s \times n} = \frac{350}{0.05 \times 477.7}$$

= 14.65 minutes

N_2 = Rpm of 130 m/min.

$$N_2 = \text{ for 2nd cut } = \frac{CS \times 1000}{\pi \times d}$$

$$= \frac{130 \times 1000}{3.14 \times 65} = 637 \text{ Rpm.}$$

T_2 = Second rough cut machining time

$$= \frac{l}{s \times n} = \frac{350}{0.05 \times 637}$$

= 11 minutes

N_3 = Rpm of 140m/min.

$$N_3 = \text{ for finish cut } = \frac{CS \times 1000}{\pi \times d}$$

$$= \frac{140 \times 1000}{3.14 \times 55} = 810 \text{ Rpm}$$

$$T_3 \text{ for the finishing cut} = \frac{l}{s \times n} = \frac{350}{0.05 \times 810}$$

= 8.64 minutes

Total machining time $= T_1 + T_2 + T_3$

= 14.65 min. + 11 min.+ 8.64 min.

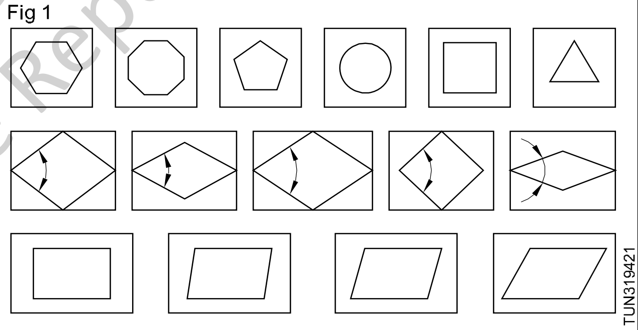
= 34.31 minutes.

Tungsten carbide tools

The throw-away carbide tool tips are the carbide inserts which are clamped mechanically. Throw-away tips are in different shapes such as round, polygon etc. When the cutting edge gets blunt a fresh cutting edge is obtained by indexing or replacing the insert.

Different shapes of throw-away tips are available to suit standard tool-holders, such as round, square, triangle, polygon etc. (Fig 1).

Fig 1



CLASSIFICATION:

Detail	Example	Alphabetical and numerical code	
Basic shape	S	C	Rhomboidal with 80° corner angle
		D	Rhomboidal with 55° corner angle
		E	Rhomboidal with 75° corner angle
		K	Parallelogram with 55° corner angle
		L	Rectangular
		R	Round
		S	Square
		T	Triangular
Clearance angle	P	C-7°, D-15°, E-20°, P-11°, 0°. (Clearance angle where special specification is necessary.)	
Tolerance class	U	Allowed variation ± on	

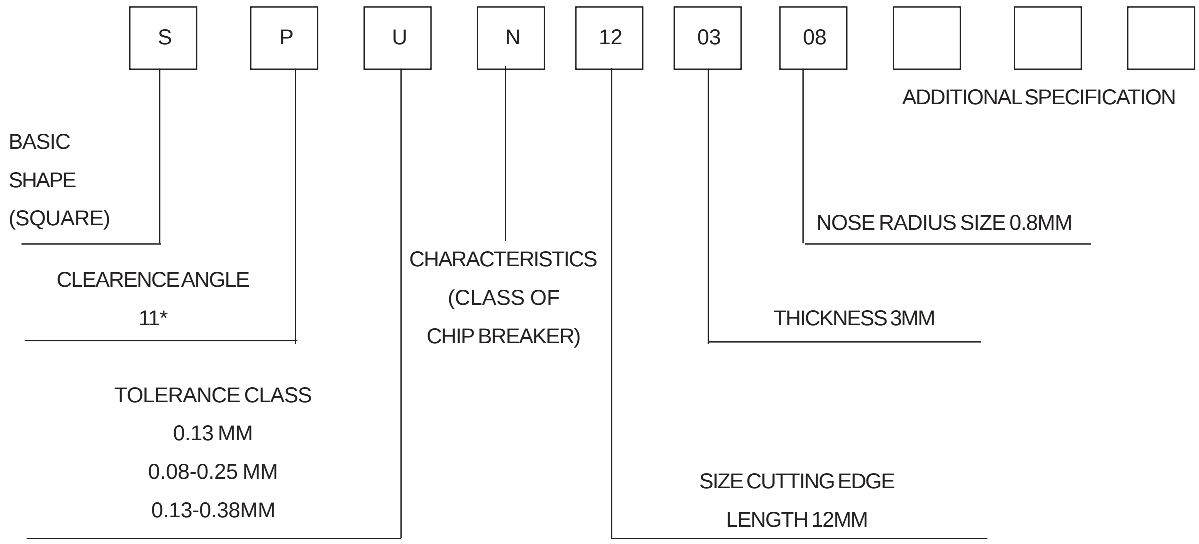
Tolerance class	**Insert thickness** ('s')	**Inscribed circle** ('d' dia).	**Control dimension** ('m')
A	0.025 mm	0.025 mm	0.005 mm
C	0.025 mm	0.025 mm	0.013 mm
E	0.025 mm	0.025 mm	0.025 mm
G	0.13 mm	0.025 mm	0.025 mm
H	0.025 mm	0.013 mm	0.013 mm
J	0.025 mm	*0.05–0.13mm	0.005 mm
K	0.025 mm	*0.05–0.13mm	0.013 mm
M	0.13 mm	*0.05–0.13mm	0.08–0.18
U	0.13 mm	*0.08–0.25mm	0.13–0.38

NOTE: This indicates that the tolerance is dependent upon the size of the insert and shape; hence no fixed values can be given.

Characteristics	N	A Without built-in chip breaker with central hole. F With built-in chip breaker on both sides without central hole. G With built-in chip breaker on both sides with central hole. M With built-in chip breaker on one side with central hole. N Without built-in chip breaker and without central hole. R With built-in chip breaker on one side without central hole. X Special, needs drawing or specification.
Size	12	Cutting edge length in mm. without decimal places. For single digit number '0' is prefixed.
Thickness	03	Thickness of insert in mm. without decimal place. For single digit number '0' is prefixed. For insert thickness 3.97 mm. symbol T3 will be used.
Nose	08	Insert with nose radius: Nose radius in 1/10 mm. For single digit number '0' is prefixed. For round inserts with diameter according to metric series symbol MO is used instead of "00". Inserts with planishing edge especially for milling: A-45°, E-75°, F-85°, P-90°, approach angle D-15°, E-20°, N-0°, P-11° clear ance angle on planishing edge. A = 3°, B = 5°, F = 25°, G = 30° For special nose design 'ZZ'
Cutting edge condition		This symbol is optional and need be specified only when necessary.
* Standard land on WIDIA inserts 0.1 to 0.2 mm x 20° (exceptions S...25) is 0.3 to 0.4 mm x 20°. Special land must be specified (e.g 0.3 mm x 20° abbreviation land 3020).	E F S T	Rounded cutting edge Sharp cutting edge Cutting edge with land and rounding Cutting edge with land *
Hand cutting	R L	Right hand cutting N Right and left hand cutting. Left hand cutting
Example	S P U N 12 03 08	Additional optional specification of cutting edge and direction of cutting.

Designation for carbide inserts

Example



Speed calculation for carbide tools

The cutting speeds and feeds for cemented carbide tools are about 3 to 4 times the cutting speeds and feeds for HSS tools.

An example of speed calculation

A workpiece with a diameter of 80 mm is to be turned with a cutting speed of v = 160 m/min.

What is the permissible headstock spindle speed?

$$V = \frac{dx\pi xn}{1000}, n = \frac{vx1000}{dx\pi} = \frac{160x1000}{80x3.14} = 636.6r.p.m.$$

If this speed cannot be obtained on the lathe, the nearest spped less than the calculated speed must be used.

Refer to Table 2 for the cutting speeds.

Table 2 shows the materials to be machined with different cutting tool material with recommended cutting speeds

TABLE 2

Cutting speeds for metals and plastics

Material	Cutting tool material	Heavy cut		Finishing cut	
		1 mpm	2 fpm	1 mpm	2 fpm
Free machining steels	HSS cast alloy carbide	35.0 76.2 122.0	115 250 400	91.4 144.8 205.7	300 475 675
Low carbon steels	HSS cast alloy carbide	30.5 65.5 106.7	100 215 350	79.2 129.5 190.5	260 425 625
Medium carbon steels	HSS cast alloy carbide	29.0 58.0 91.4	95 190 300	68.6 106.7 152.4	225 350 500
High carbon steels	HSS cast alloy carbide	24.2 53.3 76.2	80 175 250	61.0 91.4 137.2	200 300 450
Cast iron-soft grey	HSS cast alloy carbide	24.4 42.7 68.6	80 140 225	41.1 76.2 122	135 250 400
Brass and bronze - free machining	HSS cast alloy carbide	53.3 106.7 175.3	175 350 575	106.7 167.6 274.3	350 550 900
Aluminium	HSS cast alloy carbide	38.1 53.3 76.2	125 175 250	91.4 114.3 182.9	300 375 600
Plastics	HSS cast alloy carbide	30.5 45.7 61.0	100 150 200	76.2 114.3 152.4	250 375 500

Speeds should be adjusted ± 10 to 20% to suit the cutting conditions.

1 m.p.m. Metres per minute

2 f.p.m. Feet per minute

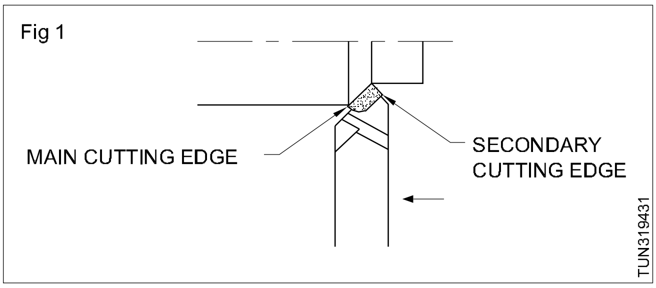
DIFFERENT TYPES AND SPECIFICATIONS OF CARBIDE TOOLS

Cemented carbide tools are available as brazed tipped tools and throw away tips held in specially designed tool holders.

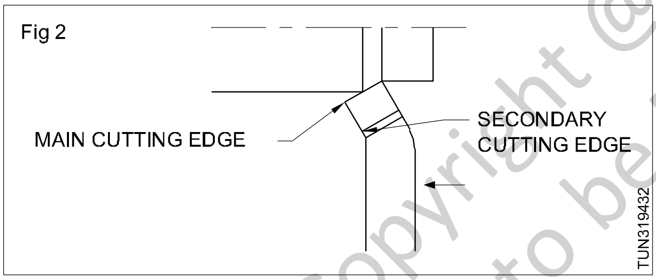
Standard shapes of carbide tipped turning and facing tools are shown in figures.(ISO 1-9) Carbide tipped cut off and boring tools are also available. These tools are re sharpened as needed using special silicon carbide and diamond wheels.

STANDARD TERMS FOR CARBIDE TOOLS AS SPECIFIED IN ISO

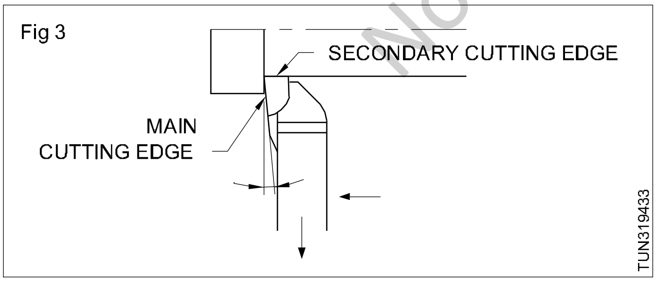
ISO 1 Straight turning tool



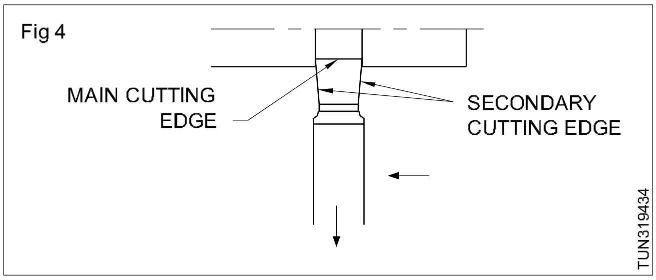
ISO 2 Cranked turning tool



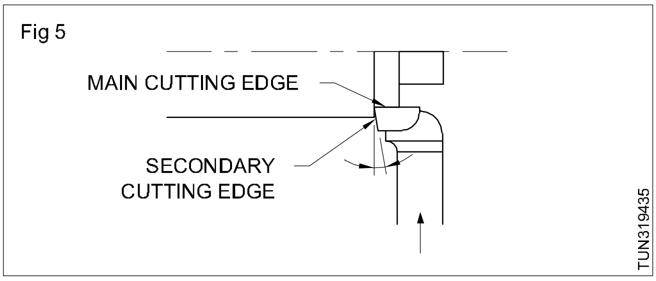
ISO 3 Offset facing tool



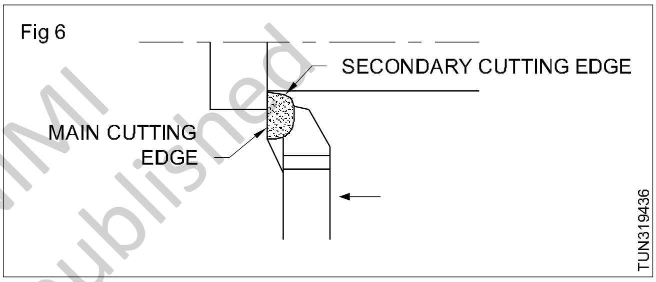
ISO 4 Wide nose square turning tool



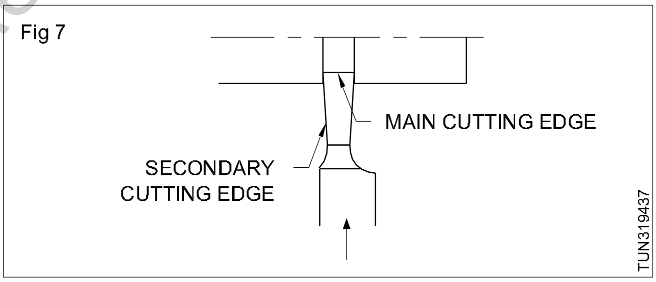
ISO 5 Offset turning and facing tool



ISO 6 Offset side cutting tool (Offset knife tool)



ISO 7 Recessing tool (parting tool)



ISO 8 Boring tool

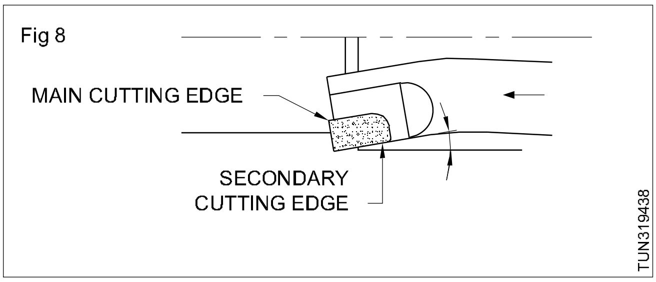


TABLE 1

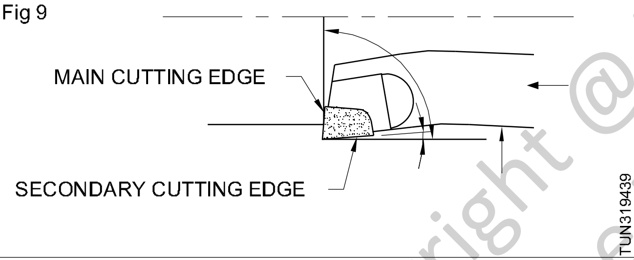
Classification of carbide tips according to their range of application. (IS: 2428 - 1964)

Designation		Increasing direction of the characteristic of		Range of application	
Identification colour		Carbide tip	Cutting	Material to be machined	Machining conditions
P01	BLUE	S — Resistance to wear; Toughness	S — Cutting speed; Feed	Steel, steel casting,	Precision turning and fine boring. Cutting speed: high. Feed : low
P10				Steel, steel casting,	Turning, threading and milling. Cutting speed: high. Feed: low or medium.
P20				Steel, steel casting, malleable cast iron, forming long chips.	Turning, milling. Cutting speed and feed: medium. Planing: with low feed rate.
P30				Steel, steel casting, malleable cast iron, forming long chips.	Turning, planing, shaping. Cutting speed: medium to low. Feed: medium to high even if operating conditions are unfavourable.
P40				Steel, steel casting with stand inclusions or shrinkage cavities.	Turning, planing, shaping. Cutting speed: low. Feed: high. Rake angle: high, for machining under unfavourable conditions and work on automatic machines.
P50				Steel, steel castings of medium or low tensile strength with sand inclusions or shrinkage cavities.	Turning, planing, shaping. Cutting speed: low. Feed: high Rake angle: high, for machining under under unfavourable conditions and work on automatic machines.
M10	YELLOW	S — Resistance to wear; Toughness	S — Cutting speed; Feed	Steel, steel castings manganese steel, grey cast iron, alloyed cast iron	Turning, cutting speed: medium to high. Feed: low to medium.
M20				Steel, steel castings, austenite, manganese steel, grey cast iron sphrodised cast iron and malleable cast iron.	Turning, milling. Cutting speed: medium. Feed: medium.
M30				Steel, steel casting, austenite, steel grey cast iron, heat resisting alloys.	Turning, milling. Cutting speed: medium. Feed: medium. Feed: medium or high.
M40				Free cutting steel, low tensile strength steel, brass and light alloy.	Turning , profile turning, parting off especially in automatic machines.
K01	RED	Resistance to wear — S; Toughness	S — Cutting speed; Feed	Very hard grey cast iron chilled castings of hardness up to 60 HRC. Aluminium alloys with high silicon content, hardened steel, plastics of abrasive type, hard board and ceramics.	Turning, precision, turning, boring and milling.

K10	S / S; Resistance to wear, Toughness, Cutting speed, Feed	Grey cast iron of hardness more than 220 HB, malleable cast iron forming short chips tempered steel, aluminium alloys containing silicon, copper alloys, plastics glass hard rubber, hard cardboard, porcelain, stone.	Turning, milling, boring, reaming, broaching.
K20		Grey cast iron of hardness up tp HB, non-ferrous metals such as copper, brass, aluminium; laminated wood of abrasive type.	Turning, milling, planing, reaming, broaching.
K30		Soft grey cast iron, low tensile strength steel, laminated wood.	Turning, planing, shaping, milling. Rake angle: large even under unfavourable conditions.
K40		Soft or hard natural wood, non-ferrous metals.	Turning, milling, planing, shaping. Rake abgle: large even under unfavourable machining conditions.

ISO 9 Corner boring tool (finishing)

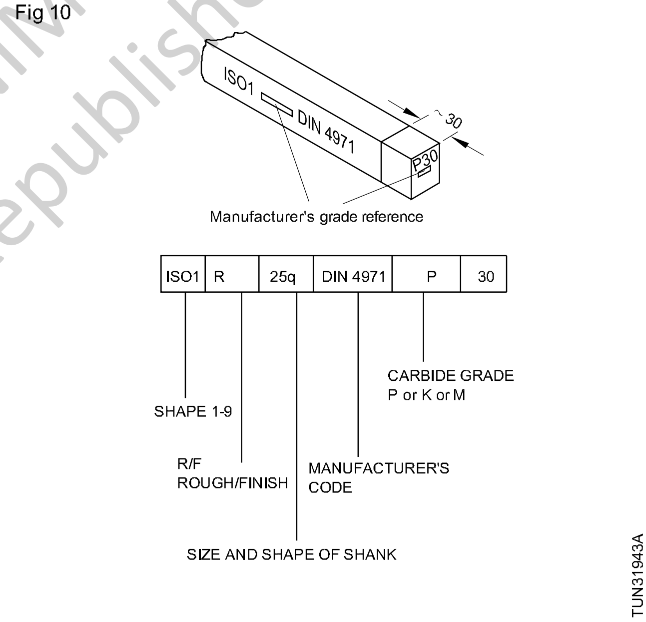
Fig 9



The carbide tools are specified according to (1) the operations (rough and finish) (2) right hand or left (3) material being turned and machining conditions. Refer to Table 1.

The method of referring to a straight ISO carbide toole by a manufacturer is given in Fig 10.

Fig 10



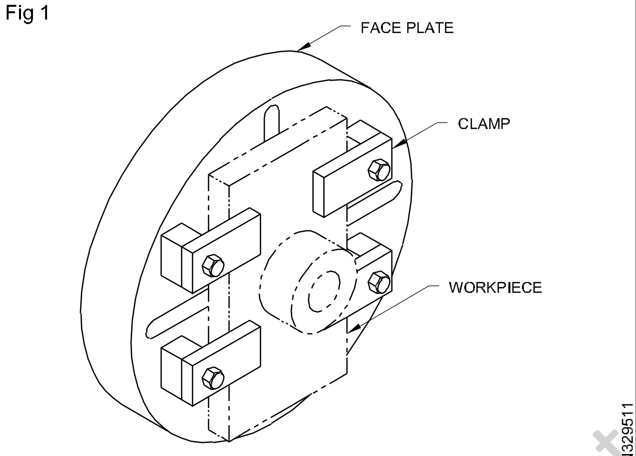
Face plate - accessories used on face plate, angle plate

Objectives: At the end of this lesson you shall be able to
- **state the necessity of a face-plate in lathe work**
- **list the face-plate accessories**
- **explain the truing of the work on a face-plate.**

Face-plate work

Large, flat, engine bodies, irregular shaped workpieces and castings, jigs and fixtures that cannot be gripped in a chuck may be clamped to a face-plate for machining operations. (Fig 1)

Fig 1

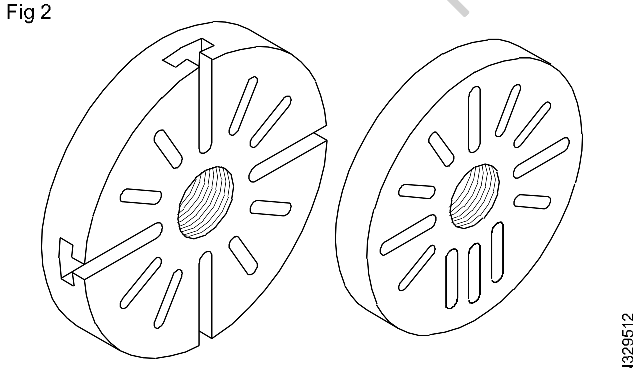


WORK CLAMPED DIRECTLY TO FACE-PLATE

Face-plate (Fig 2)

A face-plate is similar to a drive plate except that it is as large in diameter as the lathe will accommodate. It is fitted to the spindle nose and contains a number of Tslots or elongated holes to accommodate bolts and clamps. When the face-plate is mounted on the lathe spindle, its face is at right angles to the centre line of the lathe.

Fig 2



TYPES OF FACE-PLATES

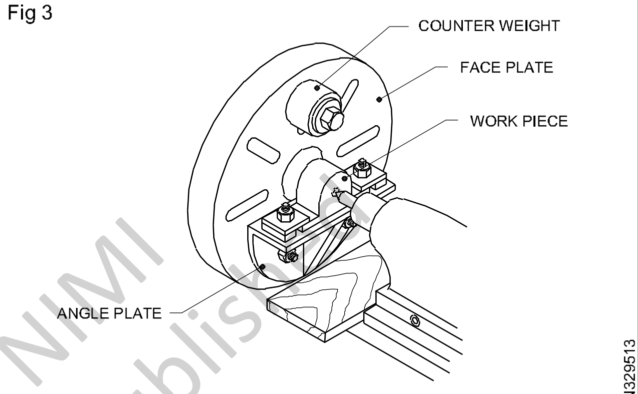
Face-plate accessories

Since the type and shape of workpieces vary greatly, a large number of face-plate accessories such as bolts, clamps, parallels, step blocks and counterweights are used to set up and fasten the work to the face-plate.

The machined surface of a workpiece can be clamped to an angle plate which is fastened securely to the face-plate.

Machining operations on the workpiece will then be parallel or square with the machined surface. (Fig 3)

Fig 3



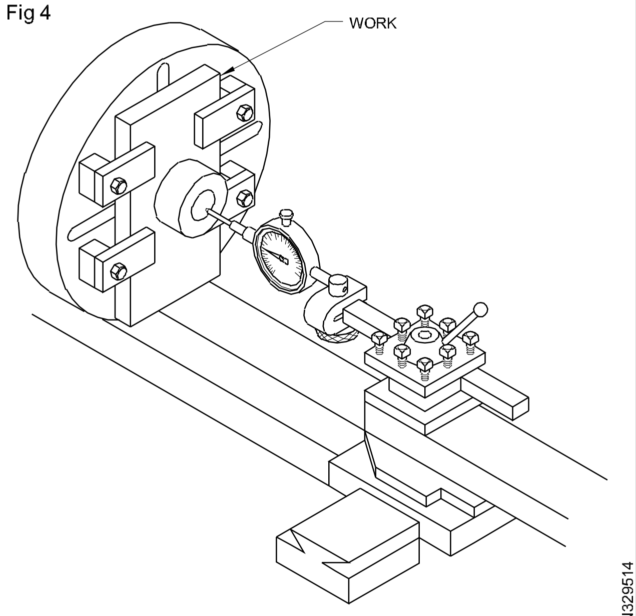
WORK HELD ON AN ANGLE PLATE

A `V' block can be bolted securely to the surface of the angle plate to hold round workpieces.

Truing a workpiece with a dial indicator (Fig. 4)

Mount a dial indicator with an internal or external attachment, as required on the tool post.

Fig 4



TRUING A BORE WITH A DIAL INDICATOR

Move the indicator into contact with the workpiece until the needle registers approximately one and a half turns.

Rotate the lathe spindle by hand and note the high reading on the dial indicator.

Tap the workpiece with a hardwood block or brass rod until the indicator registers half of the difference between the high and low readings. To prevent damage to the indicator, always tap the workpiece away from the indicator.

Continue tapping the workpiece until the indicator needle registers no movement when the lathe spindle is rotated by hand.

Tighten all bolts securely and recheck the accuracy of the set up.

Angle plates

Objectives: At the end of this lesson you shall be able to
- **state the constructional features of different types of angle plates**
- **list the different types of angle plates**
- **state the uses of different types of angle plates**
- **state the grades of angle plates**
- **specify angle plates.**

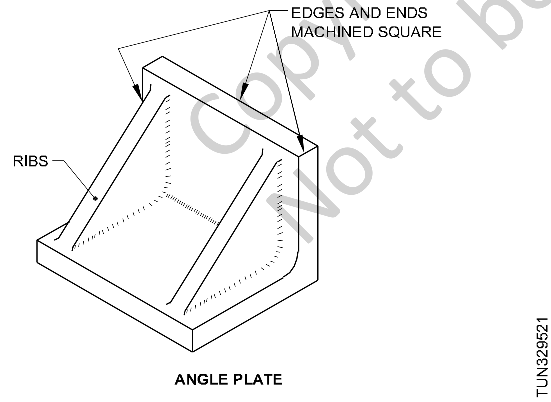
Constructional features

Angle plates have two plane surface, machined perfectly flat and at right angles to each other. Generally these are made of closely grained cast iron or steel. The edges and ends are also machined square. They have ribs on the unmachined part for good rigidity and to prevent distortion.

Types of angle plates

Plain solid angle plate (Fig 1)

Fig 1

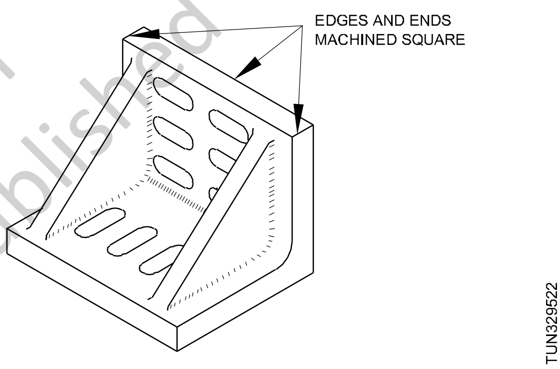


Among the three types of angle plates normally used. It plain solid angle plate is the most common. It has the two plane surfaces perfectly machined at 90° to each other such angle plates are suitable for supporting workpiece during layout work. They are comparatively smaller size.

Slotted type angle plate (Fig 2)

The two plane surfaces of this type of angle plate have their slots milled. It is comparatively bigger in size than the plain solid angle plate.

Fig 2



The slots are machined on the top plane surfaces to accomodating clamping bolts. This type of angle plate can be tilted 90° along with the work for marking and machining. (Figs 3 & 4)

Fig 3

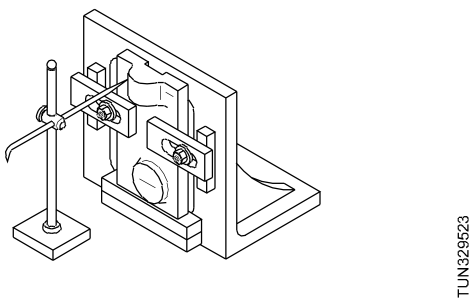
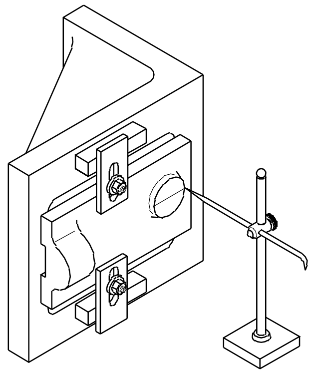


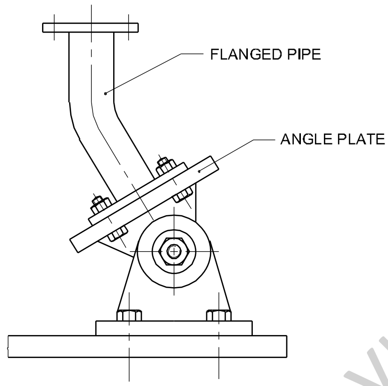
Fig 4



Swivel type angle plate (Fig 5)

This is adjustable so that the two surfaces can be kept at an angle. The two machined surfaces are on two separate pieces which are assembled. Graduations are marked on one to indicate the angle of tilt with respect to the other.When both zeros coincide, the two plane surface are at 90° to each other. A bolt and nut are provided for locking in position.

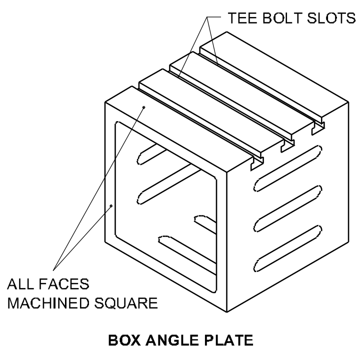
Fig 5



Box type angle plate (Fig 6)

They have applications similar to those of other angle plates. After setting, the work can be turned over with the box enabling further making out or machining. This is a significant advantage. This has all the faces machined square to one another.

Fig 6



Grades

Angle plates were available in two grades-Grade 1 and Grade 2. The grade 1 angle plates are more accurate and are used for very accurate tool room type of work. The grade 2 angle plates are used for general machine shop work. In addition to the above two grades of angle plates, precision angle plates are also available for insepction work.

Sizes

Angle plates are available in different sizes. The sizes are indicated by numbers. Table 1 gives the number of the sizes and the corresponding size proportions of the angle plates.

Specification of angle plates

Size 6 grade 1

The box plate wil be designated as

- box angle plate 6 grade 1 IS 623

Size 2 grade 2

This will be designated as angle plate 2 Gr 2 IS 623.

TABLE 1
(Grade 2 only)

Size No.	L	B	H
1	125	75	100
2	175	100	125
3	250	150	175
4	350	200	250
5	450	300	350
6	600	400	450
7	700	420	700
8	600	600	1000
9	1500	900	1500
10	2800	900	2200

Balancing - its necessity

Objectives : At the end of this lesson you shall be able to
- **state the methods used for balancing the work**
- **explain the methods of checking for balancing the job.**

Necessity of balancing the work

If the work is mounted on a face-plate in such a way that the centre of gravity of the work does not coincide with the lathe centre and if the lathe is operated, the out of balance forces, set up vibrations causing chatter and poor surface finish on the workpiece. To eliminate the vibration the work on the face-plate must be counter- balanced.

Balancing is accomplished by bolting a weight, or weights on the face-plate diametrically opposite. The amount of the weight and its position is varied until a balance is obtained.

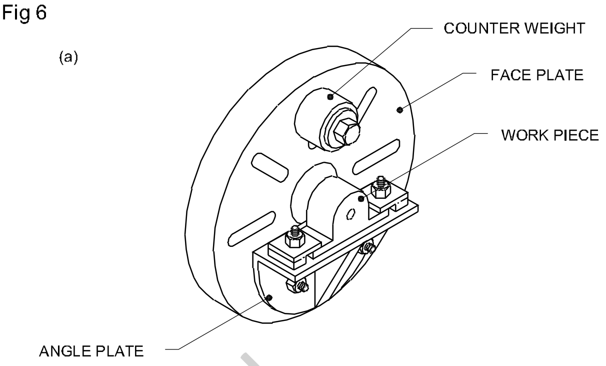
To check the balance, first disconnect the spindle drive. (Figs 1a and 1b)

Set the face-plate so that the balancing weights, the workpiece and the lathe spindle are approximately in a horizontal line.

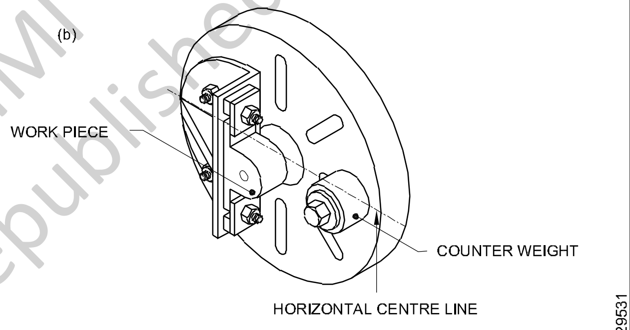
If the weight falls below the horizontal line, it is too heavy or too far out from the centre of the plate. If it rises, it is too light or too close to the centre. In either case an adjustment must be made until the balance weight remains in the horizontal position.

The degree of balance required will depend upon the accuracy desired and the speed of machining. Work machined at a low speed does not need to be balanced as accurately as work machined at a higher speed.

Fig 6



BALANCING WORK ON A FACE PLATE



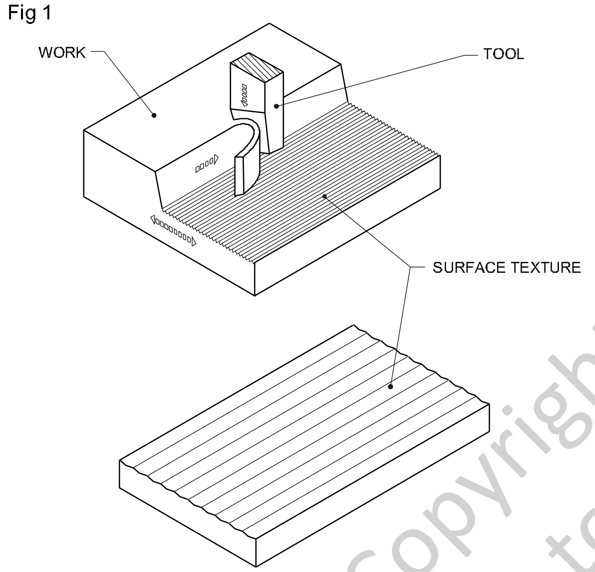
CHECKING BALANCE OF FACE PLATE

Surface finish symbol used on working blue prints

Objective: At the end of this lesson you shall be able to
- **state the meaning of surface texture**
- **distinguish between roughness and waviness**
- **state the need for different quality surface textures**
- **state the meaning of 'Ra' value**
- **interpret 'Ra' and roughness grade number in drawings.**

When components are produced either by machining or by hand processes, the movement of the cutting tool leaves certain lines or patterns on the work surface. This is known as surface texture. These are, in fact, irregularities, caused by the production process with regular or irregular spacing which tend to form a pattern on the workpiece. (Fig 1)

Fig 1



The components of surface texture

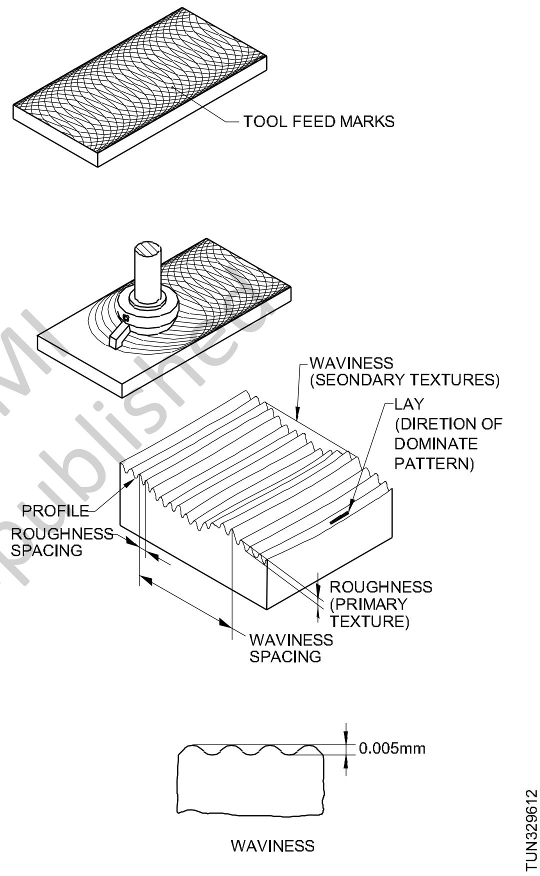
Roughness (Primary texture)

The irregularities in the surface texture result from the inherent action of the production process. These will include traverse feed marks and irregularities within them. (Fig 2a)

Waviness (Figs 2b & 2c)

This is the component of the surface texture upon which roughness is superimposed. Waviness may result from machine or work deflections, vibrations, chatter, heat treatment or warping strain.

Fig 2



The requirement of surface quality depends on the actual use to which the component is put.

Examples

In the case of slip gauges (Fig 3) the surface texture has to be extremely fine with practically no waviness. This will help the slip gauges to adhere to each other firmly when wrung together.

The cylinder bore of an engine (Fig 4) may require a certain degree of roughness for assisting the lubrication needed for the movement of the piston.

For sliding surfaces the quality of surface texture is very important.

Fig 3

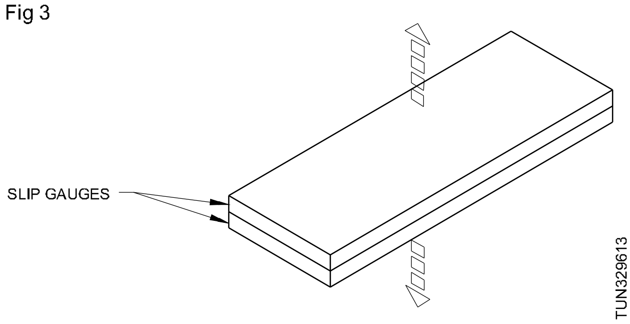
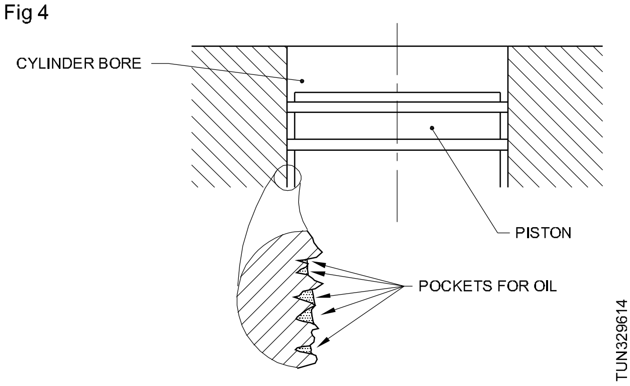
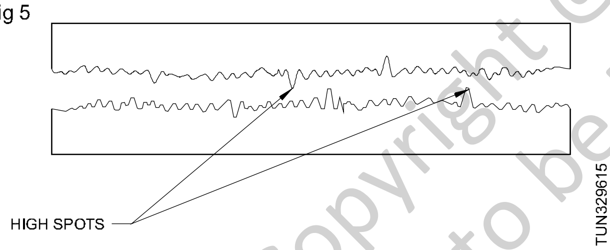


Fig 4



When two sliding surfaces are placed one over the other, initially the contact will be only on the high spots. (Fig 5) These high spots will wear away gradually. This wearing away depends on the quality of the surface texture.

Fig 5



Due to this reason it is important to indicate the surface quality of components to be manufactured.

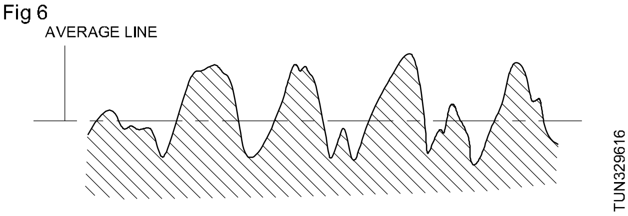
The surface texture quality can be expressed and assessed numerically.

'Ra' Values

The most commonly used method of expressing the surface texture quality numerically is by using Ra value. This is also known as centre line average (CLA).

The graphical representation of Ra value is shown in Figures 6 & 7. In Figure 6 a mean line is placed cutting through the surface profile making the cavities below and the material above equal.

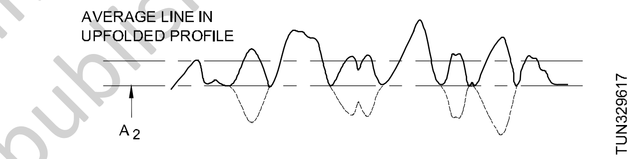
Fig 6



The profile curve is then drawn along the average line so that the profile below this is brought above.

A new mean line (Fig 7) is then calculated for the curve obtained after folding the bottom half of the original profile.

Fig 7



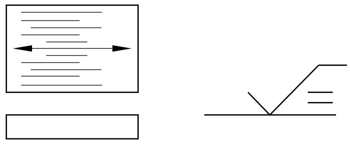
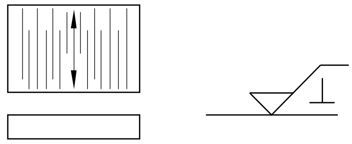
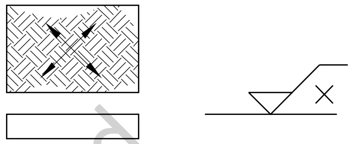
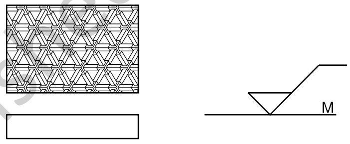
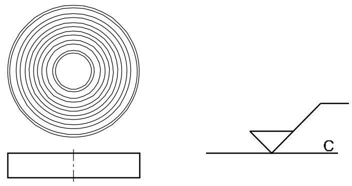
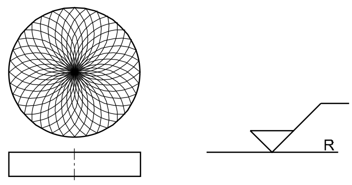
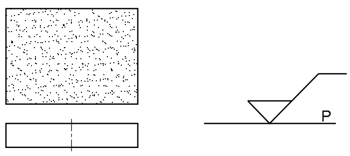
The distance between the two lines is the 'Ra' value of the surface.

The 'Ra' value is expressed in terms of micrometre (0.000001) or (m); this also can be indicated in the corresponding roughness grade number, ranging from N_1 to N_{12}.

When only one 'Ra' value is specified, it represents the maximum permissible value of surface roughness.

Lay: Symbols for designating the direction of lay are shown and interpreted in table 1.

TABLE 1

Example showing	Interpretation	Direction of tool marks
=	Lay approximately parallel to the line representing the surface to which, the symbol is applied.	
⊥	Lay approximately perpendicular to the line representing the surface to which the symbol is applied	
X	Lay angular in both direction to line representing the surface to which the symbol is applied.	
M	Lay multidirectional.	
C	Lay approximately circular relative to the centre of the surface to which the symbol is applied.	
R	Lay approximately radial relative to the centre of the surface of which the symbol is applied.	
P	Lay particulate, non-directional, or protuberant.	

Surface texture measuring instruments

Objectives: At the end of this lesson you shall be able to
- **distinguish the features of mechanical and electronic surface indicators**
- **name the parts of a mechanical surface indicator**
- **brief the features of electronic surface indicators (taly-surf).**

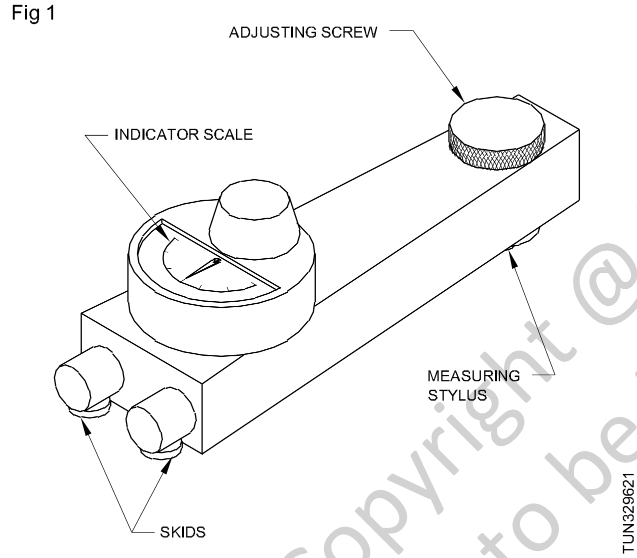
The use of surface finish standards which we have seen earlier is only a method of comparing and determining the quality of surface. The result of such measurement very much depends on the sense of touch and cannot be used when a higher degree of accuracy is needed.

The instruments used for measuring the surface texture can be of a mechanical type or with electronic sensing device.

Mechanical surface indicator:

This instrument consists of the following features. (Fig 1)

Fig 1



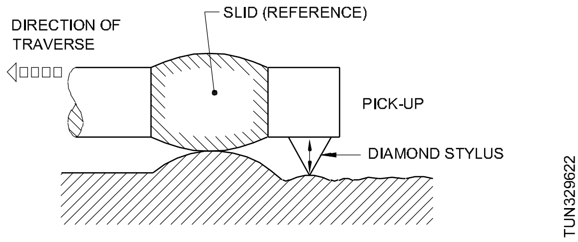
- **Measuring stylus**
- **Skids**
- **Indicator scale**
- **Adjustment screw**

The stylus is made of diamond, and its contact point will have a light radius.

When the stylus is slowly traversed across the test surface the stylus moves upward or downward depending on the profile of the surface. (Fig 2) This movement is amplified and transferred to the dial of the surface indicator. The pointer movement indicates the surface irregularities.

When using a mechanical surface indicator, measurement must be read as it is moved over the surface, and then a profile curve is drawn manually to compute the mean value.

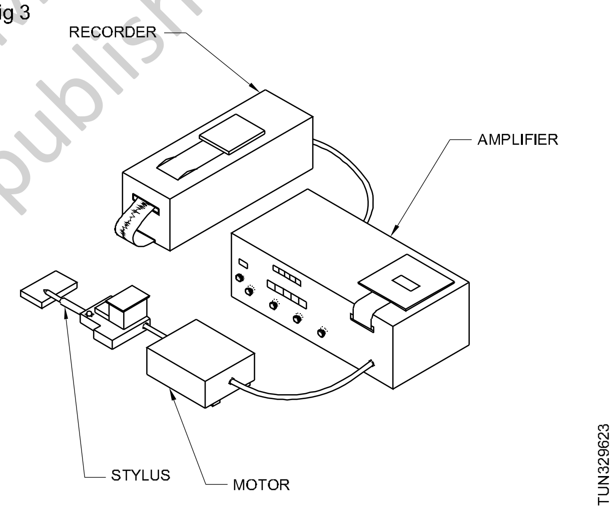
Fig 2



There are different types of electronic surface measuring devices; one type of such an instrument used in workshops is the taly-surf.

Taly-surf (Electronic surface indicator)**:** This is an electronic instrument for measuring surface texture. This instrument can be used for factory and laboratory use. (Fig 3)

Fig 3



The measuring head of this unit consists of a stylus (a) and a motor race (b) which controls the movement of the instrument head across the surface. The movement of the stylus is converted to electrical signals. These signals are amplified in the surface analyser/amplifier (c) which calculates the surface parameter and presents the result on a digital display or in the form of a diagram through a recorder (d).

Machining symbols

Objectives : At the end of this lesson you shall be able to
- **read the values of surface roughness**
- **interpret the surface roughness values.**

Letter symbols for tolerances:

Indication of surface roughness values in table 1.

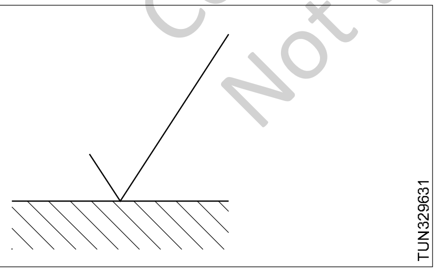
TABLE 1

Sl. No.	Roughness value Ra in microns	Roughness grade Number	Roughness Symbol	Manufacturing process
1.	50	N12	~	Flame cutting, hacksaw cut, bandsaw cut, shot blast etc.
2.	25.0 12.5	N11	∇	Sand casting, planning, shaping filling etc.
3.	6.3 3.2 1.6	N9 N8 N7	∇∇	Milling, drilling, die casting, turning, forging, boring etc.
4.	0.8 0.4 0.2	N6 N5 N4	∇∇∇	Centreless grinding, cylindrical grinding, cold rolling, internal grinding, extrusion, surface grinding, broaching, hobbing EDM, reaming etc.
5.	0.1 0.05 0.025	N3 N2 N1	∇∇∇∇	Super finishing, lapping honning etc.

Surface symbol indication:

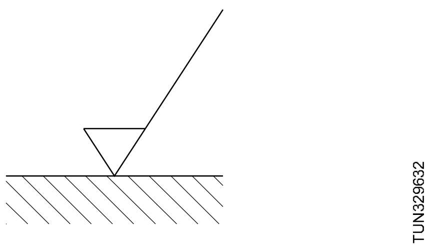
1 The basic symbol consists of two legs of unequal length inclined at approximately 60°. (Fig 1)

Fig 1



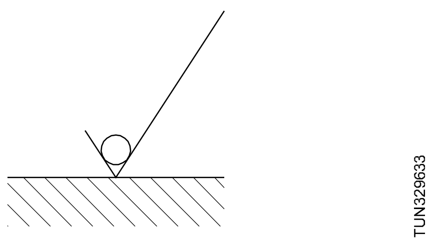
2 If the material removal by machining is required, a bar is added to the basic symbol. (Fig 2)

Fig 2



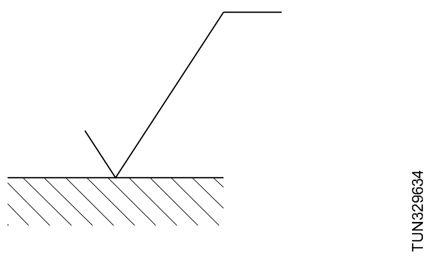
3 If the material removal is not permitted, a circle is added to the basic symbol. (Fig 3)

Fig 3



4 If some special characteristics have to be indicated, a line is added to the larger leg. (Fig 4)

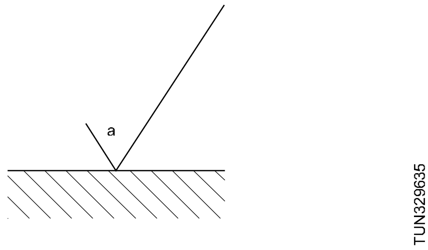
Fig 4



5 **Indication of surface roughness:**

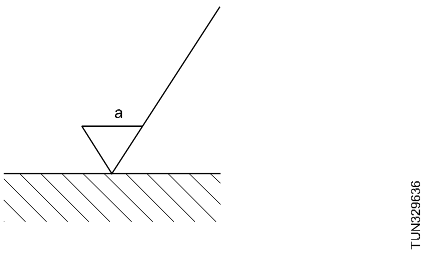
a Surface roughness obtained by any production method. (Fig 5)

Fig 5



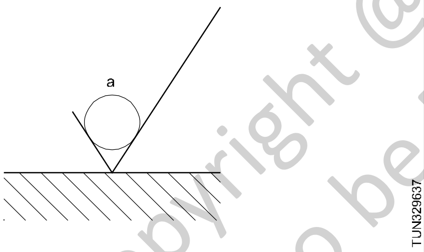
b Surface roughness obtained by removal of material by matching. (Fig 6)

Fig 6



c Surface roughness obtained by without of material removal. (Fig 7)

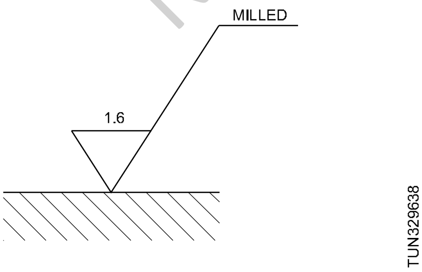
Fig 7



6 Indication of special surface roughness characteristics:

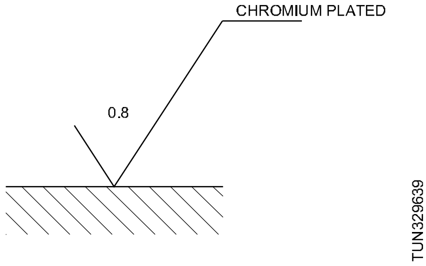
a Indicating the production method.(Fig 8)

Fig 8



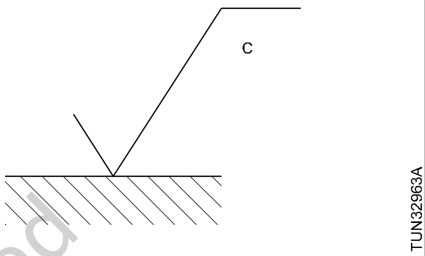
b Indicating the surface treatment or coating. Unless otherwise stated, the numerical value of the roughness, applies to the surface roughness after treatment of coating.(Fig 9)

Fig 9



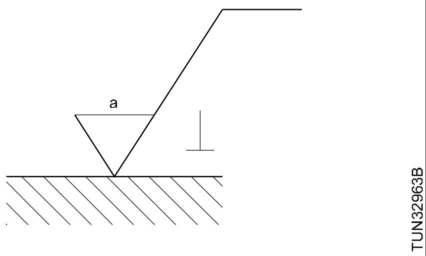
c Indicating the sampling length.(Fig 10)

Fig 10



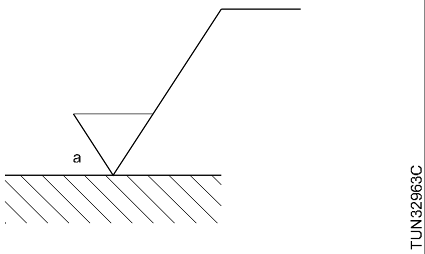
d Direction of lay, surface pattern by the production method employed.(Fig 11)

Fig 11



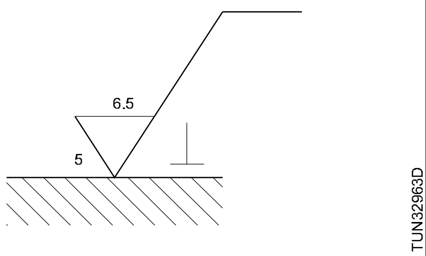
e Indication of allowance in mm.(Fig 12)

Fig 12



Surface texture:(Fig 13)

Fig 13



Lapping

Objectives: At the end of this lesson you shall be able to
- **state the purpose of lapping**
- **state the features of a flat lapping plate**
- **state the use of charging a flat lapping plate**
- **state the method of charging a cast iron plate**
- **explain between wet lapping and dry lapping.**

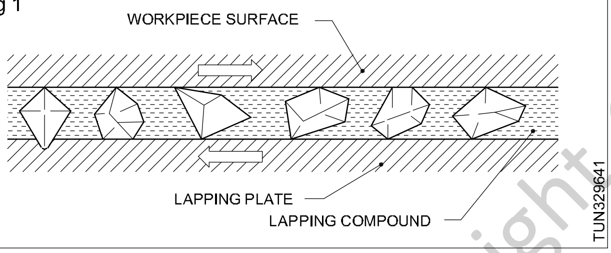
Lapping is a precision finishing operation carried out using fine abrasive materials.

Purpose: This process

- improves geometrical accuracy
- refines surface finish
- assists in achieving a high degree of dimensional accuracy
- improve the quality of fit between the mating components.

Lapping process: In the lapping process small amount of material are removed by rubbing the work against a lap charged with a lapping compound. (Fig 1)

Fig 1



Lap materials and lapping compounds

The material used for making laps should be softer than the workpiece being lapped. This helps to charge the abrasives on the lap. If the lap is harder than the workpiece, the workpiece will get charged with the abrasives and cut the lap instead of the workpiece being lapped.

Laps are usually made of:

- close grained iron
- copper
- brass or lead

The best material used for making lap is cast iron, but this cannot be used for all applications.

When there is excessive lapping allowance, copper and brass laps are preferred as they can be charged more easily and cut more rapidly than cast iron.

Lead is an inexpensive form of lap commonly used for holes. Lead is cast to the required size on steel arbor. These laps can be expanded when they are worn out. Charging the lap is much quicker.

Lapping abrasives: Abrasives of different types are used for lapping.

The commonly used abrasives are:

- Silicon Carbide
- Aluminium Oxide
- Boron Carbide and
- Diamond

Silicon carbide: This is an extremely hard abrasive. Its grit is sharp and brittle. While lapping, the sharp cutting edges continuously break down exposing new cutting edges. Due to this reason this is considered as very ideal for lapping hardened steel and cast iron, particularly where heavy stock removal is required.

Aluminium oxide: Aluminium oxide is sharp and tougher than silicon carbide. Aluminium oxide is used in un-fused and fused forms. Un-fused alumina (aluminium oxide) removes stock effectively and is capable of obtaining high quality finish.

Fused alumina is used for lapping soft steels and non-ferrous metals.

Boron carbide: This is an expensive abrasive material which is next to diamond in hardness. It has excellent cutting properties. Because of the high cost, it is used only in specialised application like dies and gauges.

Diamond: This being the hardest of all materials, it is used for lapping tungsten carbide. Rotary diamond laps are also prepared for accurately finishing very small holes which cannot be ground.

Lapping vehicles: In the preparation of lapping compounds the abrasive particles are suspended in vehicles. This helps to prevent concentration of abrasives on the lapping surfaces and regulates the cutting action and lubricates the surfaces.

The commonly used vehicles are

- water soluble cutting oils
- vegetable oil
- machine oils
- petroleum jelly or grease
- vehicles with oil or grease base used for lapping ferrous metals.

Metals like copper and its alloys and other non-ferrous metals are lapped using soluble oil, bentomite etc.

In addition to the vehicles used in making the lapping compound, solvents like water, kerosene, etc. are also used at the time of lapping.

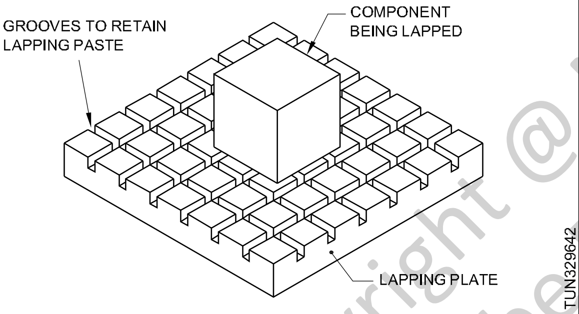
Abrasive of varying grain sizes from 50 to 800 are used for lapping, depending on the surface finish required on the component.

The lapping compound consists of fine abrasive particles suspended in a 'vehicle' such as oil, paraffin, grease etc.

The lapping compound which is introduced between the workpiece and the lap chips away the material from the workpiece. Light pressure is applied when both are moved against each other. The lapping can be carried out manually or by machine.

Hand lapping of flat surfaces: Flat surfaces are hand-lapped using lapping plate made out of close grained cast iron. (Fig.2) The surface of the plate should be in a true plane for accurate results in lapping.

Fig 2



The lapping plate generally used in tool rooms will have narrow grooves cut on its surface both lengthwise and crosswise forming a series of squares.

While lapping, the lapping compound collects in the serrations and rolls in and out as the work is moved.

Before commencing lapping of the component, the cast iron plate should be CHARGED with abrasive particles.

This is a process by which the abrasive particles are embedded on to the surfaces of the laps which are comparatively softer than the component being lapped. For charging the cast iron lap, apply a thin coating of the abrasive compound over the surface of the lapping plate.

Use a finished hard steel block and press the cutting particles into the lap. While doing so, rubbing should be kept to the minimum. When the entire surface of the lapping plate is charged, the surface will have a uniform gray appearance. If the surface is not fully charged, bright spots will be visible here and there.

Excessive application of the abrasive compound will result in the rolling action of the abrasive between the work and the plate developing inaccuracies.

The surface of the flat lap should be finished true by scraping before charging. After charging the plate, wash off all the loose abrasive using kerosene.

Then place the workpiece on the plate and move along and across, covering the entire surface area of the plate. When carrying out fine lapping, the surface should be kept moist with the help of kerosene.

Wet and dry lapping : Lapping can be carried out either wet or dry.

In wet lapping there is surplus oil and abrasives on the surface of the lap. As the workpiece, which is being lapped, is moved on the lap, there is movement of the abrasive particles also.

In dry method the lap is first charged by rubbing the abrasives on the surface of the lap. The surplus oil and abrasives are then washed off. The abrasives embedded on the surface of the lap will only be remaining. The embedded abrasives act like a fine oilstone when metal pins to be lapped are moved over the surface with light pressure. However, while lapping, the surface being lapped is kept moistened with kerosene or petrol. Surfaces finished by the dry method will have better finish and appearance. Some prefer to do rough lapping by wet method and finish by dry lapping.

Honing

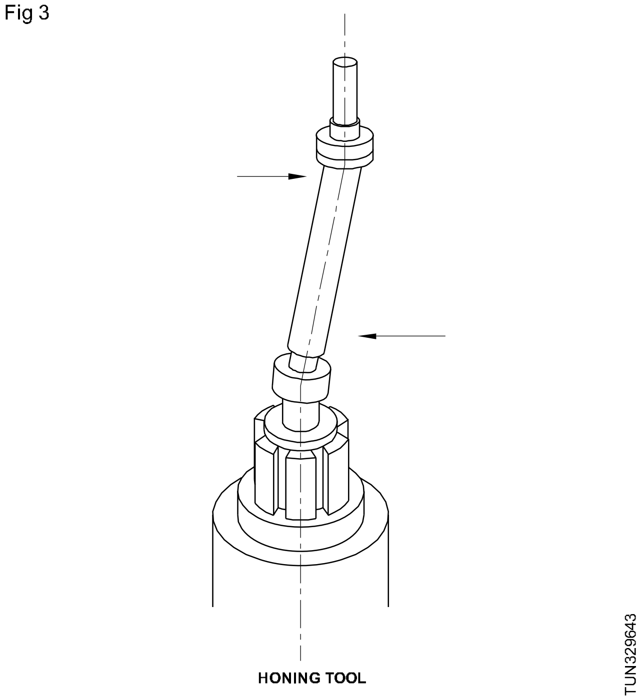
Honing is a finishing process, in which a tool called hone carries out a combined rotary and reciprocating motion while the work piece does not perform any working motion. Most honing is done on internal cylindrical surface, such as automobile cylindrical walls. The honing stones are held against the work piece with controlled light pressure. The honing head is not guided externally but, instead, floats in the hole, being guided by the work surface (Fig.3) It is desired that

1 Honing stones should not leave the work surface

2 Stroke length must cover the entire work length.

In honing rotary and oscillatory motions are combined to produce a cross hatched lay pattern as illustrated in Fig.3.

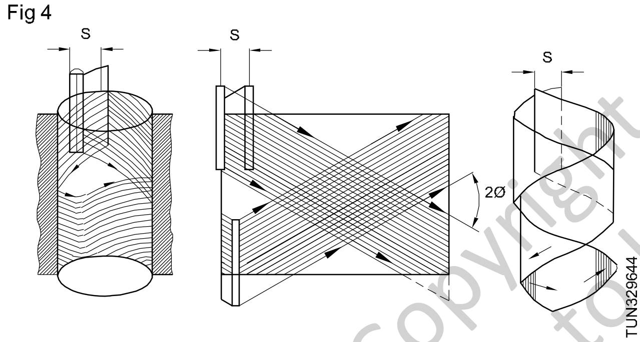
Fig 3



Honing tool

Lay pattern produced by combination of rotary and oscillatory motion. (Fig.4)

Fig 4



The honing stones are given a complex motion so as to prevent every single grit from repeating its path over the work surface.

The critical process parameters are

1 Rotation speed
2 Oscillation speed
3 Length and position of the stroke
4 Honing stick pressure

With conventional abrasive stick, several strokes are necessary to obtain the desired finish on the work piece. However, with introduction of high performance diamond and CBN grits it is now possible to perform the honing operation in just one complete stroke. Advent of precisely engineered microcrystalline CBN grit has enhanced the capability further. Honing stick with microcrystalline CBN grit can maintain sharp cutting condition with consistent results over long duration.

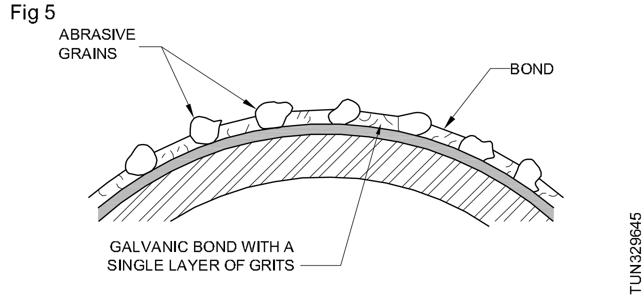
Super abrasive honing stick with monolayer configuration, where a layer of CBN grits are attached to stick by a galvanically deposited metal layer, is typically found in single stroke honing application.

With the advent of precision brazing technique, efforts can be made to manufacture honing stick with single layer configuration with a brazed metal bond. Like brazed grinding wheel such single layer brazed honing stick are expected to provide controlled grit density, larger grit protrusion leading to higher material removal rate and longer life compared to what can be obtained with a galvanically bonded counterpart.

The important parameters that affect material removal rate (MRR) and surface roughness (R) are:

1 Unit pressure, P
2 Peripheral honing speed, V_c
3 Honing time, T

Fig 5



Preventive maintenance - its necessity

Objectives: At the end of this lesson you shall be able to
- **state the need for preventive maintenance**
- **describe the functions of the P M department**
- **state the advantages of P M**
- **state the advantages of maintenance records and periodic inspection of machines.**

The machine tools are of high precision, and are sensitive and expensive.

They must be handled and maintained carefully in order to give good and long service.

The basic function of the maintenance department is the upkeep of the machines and equipments in good operating condition.

Earlier the maintenance of the equipment used to receive attention only when the equipment suffered some set-back or breakdown as a result of some minor/major fault. Such breakdowns not only brought a serious production hold-up but also used to upset the production flow of the industry where the other equipment also had to stand idle. This resulted in a more cautious approach to the maintenance of the equipment and this brought up the more scientific way of tackling the maintenance problem, through preventive maintenance.(P M)

Preventive maintenance

Preventive maintenance consists of a few engineering activities which help to maintain the machine tools in good working order.

The basic activities of preventive maintenance are the:

- periodic inspection of machines and equipments to uncover conditions leading to production breakdowns or harmful depreciation

- upkeep of machines and equipments to avoid such conditions or to adjust, repair or replace them while they are still in the initial stages.

Advantages of preventive maintenance system

Less down time in production.

Improves quantity and quality of product.

Standby equipment is not needed which saves capital investment.

Lower unit cost of manufacture.

Reduces major and repetitive repairs of machines.

P.M. helps in prolonging the life of the machines and reduction in un-expected breakdowns.

Functions of preventive maintenance department

Periodic inspection of machines and equipments as per

the ‘Check- lists’. (Annexure I)

Lubrication of machines and equipments as per the manufcaturer’s instruction manuals.

Servicing and overhauling of machine and equipment as per the P M schedule.

Keeping basic records of each machine and equipment. (Annexure II)

Analysis of inspection reports and systematic review of reports of machines and equipments.

Periodic inspection of machines and equipments as per the check-list

The check-list itemizes for the inspector all the points to be checked on individual machines. While preparing the check-list of the machine, make sure that no machine part or item that is omitted needs attention. The inspection of machine tools like lathe and drilling machine includes the following.

- Driving system and feeding system
- Lubricating and coolant system
- Slides and wedges and gibs
- Belts, bearings, clutch, brake and operating controls
- Guideways, lead screws and their mating parts

After inspection of each machine, the inspector has to make out the list of parts which need repairs or spares for replacement.

Frequency of inspection

The frequency of inspection depends on the age, kind of machine and its operating conditions. Frequent inspection of machines and equipment may be expensive and frequency with long intervals may result in more breakdowns. A good balance is needed to bring optimum savings.

Frequency of Lubrication

The length of time a machine will retain its accuracy and give satisfactory service depends on the lubrication and care it receives. It is essential that lubrication of machines should be carried out systematically at regular intervals as recommended in the service manual supplied by the machine manufacturer.

The manufacturer's manual contains all the necessary details like grade of oil, grease, oiling and greasing points and also indicates the time intervals of lubrication.

Maintenance records (Annexure III)

Keep a detailed record of faults, failures, repairs and replacements done for machines. It is useful to analyse the cause of a fault and rectification.

Maintenance records analysis

Systematic review and regular analysis of the equipment records will help to:

- re-design the weak part which gives repetitive trouble
- substitute with better material for high cost items
- minimise frequent breakdowns
- reduce the cost of production.

Preventive Maintenance Programme

Annexure - I

Name of the Machine :

Machine Number :

Model No. & Make :

Location of the machine :

Check-list for machine inspection

Inspect the following items and tick in the appropriate column and list the remedial measures for the defective items.

Items to be checked	Good working/satisfactory	Defective	Remedial measures
Level of the machine			
Belt and its tension			
Bearing sound			
Driving clutch and brake			
Exposed gears			
Working in all the speeds			
Working in all feeds			
Lubrication system			
Coolant system			
Carriage & its travel			
Cross-slide & its movement			
Compound slide & its travel			
Tailstock's parrallel movement			
Electrical controls			
Safety gaurds			

Inspected by

Signature

Name:
Date: Signature of in-charge

Equipment record Annexure - I

History sheet of machinery & equipment

Description of equipment:

Manufacturers' address:

Supplier's address:

Order No. and date:

Date on which received:

Date on which installed and placed:

Date of commissioning:

Size: Length X Width X Height

Weight:

Cost:

Motor particulars: Watts: r.p.m: Phase: Volts

Bearings/Spares record:

Belt specification:

Lubrication details:

Major repairs and overhauls
carried out with dates.

Maintenance Records

Sl.No	Name of the machine	Nature of fault rectified	Date	Signature of in-charge

Documentations - 1

Objectives: At the end of this lesson you shall be able to
- **describe work organisation**
- **name the aspects of organisation of work**
- **state the common technical terms used in industry.**

Total productive maintenance

Designed to eliminate all nonstandard, non-planned maintenance with the goal of eliminating unscheduled disruptions, simplifying (de-skilling) maintenance procedures and reducing the need for "just-in-case" maintenance employees.

Total quality management

This is aimed towards zero defect or elimination of poor quality in production. The quality concept of assuming the best quality from inception to implementation throughout the production process.

An introduction to tatal productive maintenance (TPM)

What is total productive maintenenace (TPM)?

It can be considered as the medical science of machines. Total productive maintenance (TPM) is a mainteneace prgram which invloves a newly defiened concept for maintaining plants and equipment. The goal of the TPM program is to markedly increase production while, at the same time, increasing employee and unscheduled maintenance to a minimum.

Why TPM?

TPM was introduced to achieve the following objectives. The important ones are listed below.

- Avoid wastage in a quickly changing economic environment.
- Producing goods without reducing product quality.
- Reduce cost.
- Produce a low batch quantity at the earliest possible time.
- Goods send to the customers must be non defective.

Similarities and differences between TQM and TPM:

The TPM progran closely resembles the popular total quality management (TQM) program. Many of the tools such as employee empowerment, benchmarking, documentation, erc. used in TQM are used to implement and optimize TPM. Following are the similarities between the two.

1 Total commitment to the program by upper level mangament is required in both programmes.

2 Employees must be empowered to initiate corrective action, and

3 A long range outlook must be accepted as TPM may take a year or more to implement and is an on-going process. Changes in employee mind-set toward their job responsibilities must take place as well.

The differences between TQM and TPM is summarized below

Category	TQM	TPM
Object	Quallity (Output and effects)	Equipment (Input and cause)
Mains of attaining goal	Systenatize the management. It is software oriented	Employees participation and it is hardware oriented
Target	Quality for PPM	Elimination of losses and wastes.

Types of maintenance :

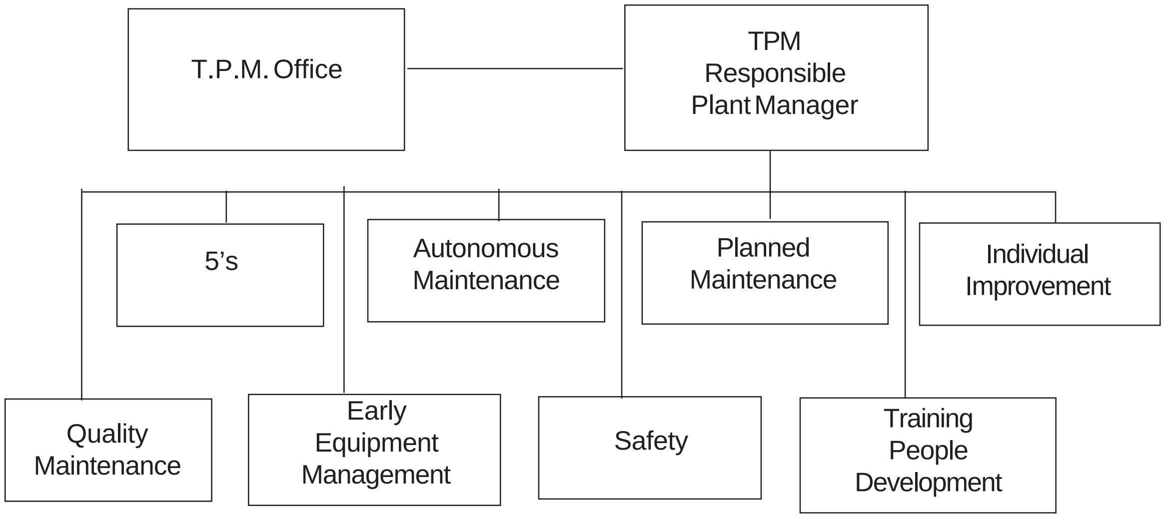
1 Breakdown maintenance :

An Introduction to Tatal Productive Maintenance (TPM)

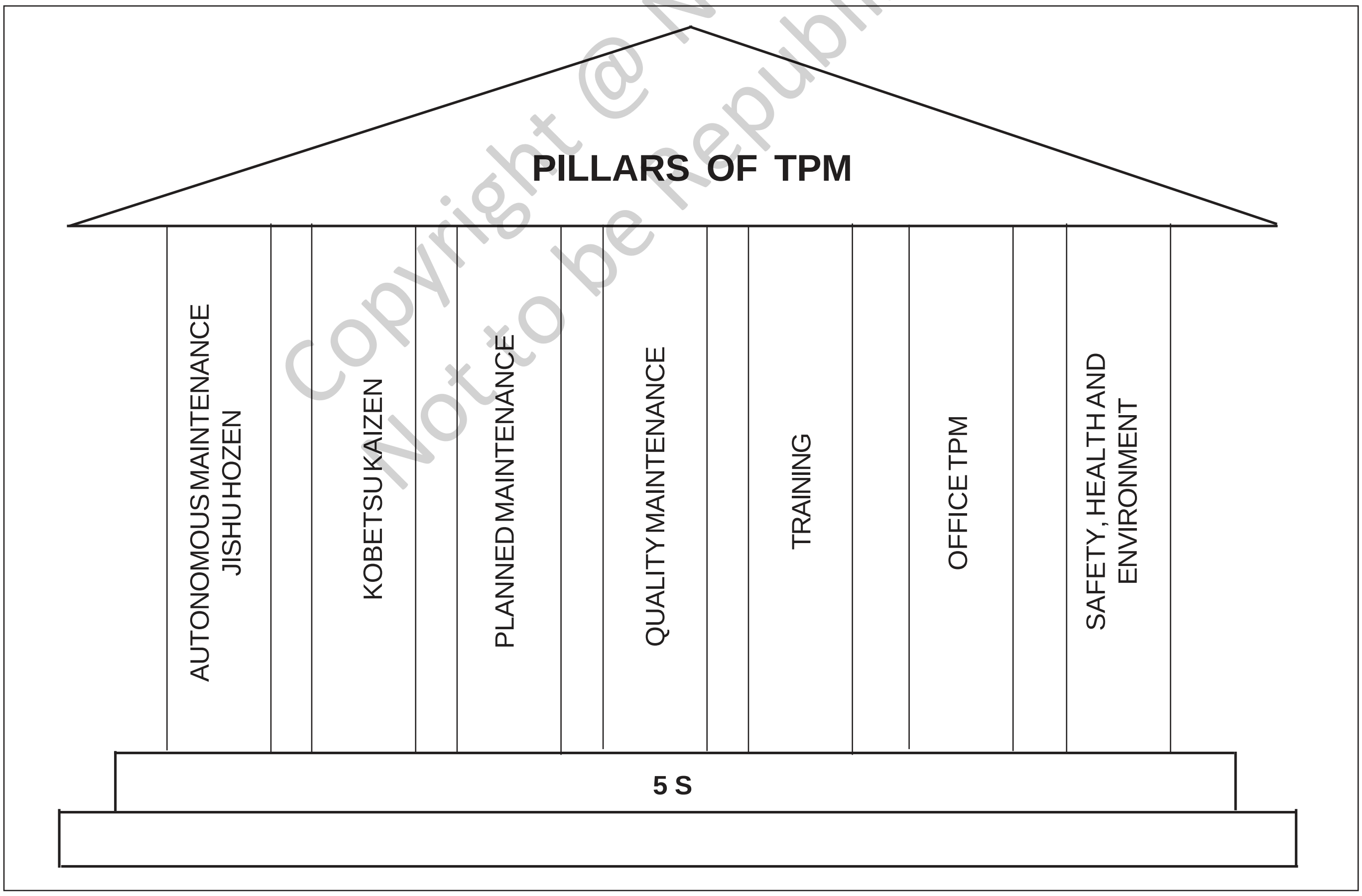
Increase the suggestions by 3 times. Develop Multi-skilled and flexible workers

Motives of TPM	1 Adoptation of life cycle approach for imporving the overall performance of prodiction equipment. 2 Improving productivity by highly motivatd workers which is achieved by job enlargement. 3 The use of voluntary small group activities for identifying the cause of failure, possible plant and equipment modifications.
Uniqueness of TPM	The major difference between TPM and other concepts is that the operators are also made to involve in the maintenance process. The concept of "/ production operators) Operate, You (Maintenance department) fix" is not followed.
TPM Objectives	1 Achieve Zero Defects, Zero Breakdown and Zero accidents in all functional areas of the organization. 2 Involve people in all levels of organization. 3 Form different teams to reduce defects and Self Maintenance.
Direct benefits of TPM	1 Increase productivity and OPE (Overall Plant Efficiency) by 1.5 or 2 times. 2 Rectify customer complaints. 3 Reduce tge manufacturing cost by 30%. 4 Satisfy the customers needs by 00% (Delivering the right quantity at the right time, in the required quality.) 5 Reduce accidents. 6 Follow pollution control measures.
Indirect benefits of TPM	1 Higher confidence level among the employees. 2 Keep the work place clean, neat and attractive. 3 Favorablechange in the attitude of the operators. 4 Achieve goals by working as team. 5 Horizontaldeployment of a new concept in all areas of the organization. 6 Share knowledge and experience. 7 The workers get a feeling of owning the machine.

T.P.M. PLANT WIDE STRUCTURE



An Introduction to Total Productive Maintenance (TPM)



Pillar 1 - 5S :

TPM starts with 5S, Problems cannot be clearly seen when the work place is unorganized. Cleaning and organising the workplace helps the team to uncover problems. Making problems visible is the first step of improvement.

Japanese Term	English Translation	Equivalent 'S' term
Seiri	Organisation	Sort
Seiton	Tidiness	Systematise
Seiso	Cleaning	Sweep
Seiketsu	Standardisation	Standardise
Shitsuke	Discipline	Self - Discipline

P Q C D S M in Office TPM :

P - Production output lost due to want of material, Manpower productivity, Production output lost due to want of tools.

Q - Mistakes in preparation of cheques, bills, invoices, payroll, Customer returns/warranty attributable to BOPs, Rejection/rework in BOP's/job work, Office area rework.

C - Buying cost/unit produced, Cost of logistics - inbound/ outbound, Cost of carrying inventory, Cost of communication, Demurrage costs.

D - Logistics losses (Delay in loading/unloading)

- Delay in delivery due to any of the support functions.
- Delay in payments to suppliers.
- Delay in information.

S - Safety in material handling/stores/logistics, Sagety of soft and hard data.

M - Number of kaizens in office areas.

Target : Environment health safety

1 Zero accident,

2 Zero health damage

3 Zero fires

In this area focus is on to create a safe workplace and a surrounding area that is not damaged by our process or procedures. THis pillar will play an active role in each of the other pillars on a regular basis.

A committee is constituted for this pillar which comprises representative of officers as well as workers. The committee is headed by Senior vice president (Technical). Utmost importance to safety is given in the plant. Manager (Safety) is looking after functions releated to safety. To create awareness among employees various competitions like safety slogans, Quiz, Drama, POsters, etc. related to safety can be organized at regular intervals.

Conclusion:

Today, with competition in industry at an all time high, TPM may be the only thing that stands between success and total failure for some companies. It has been provel to bea program that works. It can be adapted to work not only in industrial plants, but in construction, building maintenenace, transportation, and in a variety of other situations. Employees must be educated and convinced that TPM is not just an another "program of the month" and that mangement is totally committed to the program and the extended time frame necessary for full implementation. If everyone involved in a TPM program does his or her part, an unusually high rate of return compared to resources invested may be expected.

Marking off and marking table

Objectives: At the end of this lesson you shall be able to
- **why marking off is necessary**
- **the function of witness marks**
- **the features of marking tables**
- **the uses of marking tables**
- **the maintenance aspects concerning marking tables.**

Marking off

Marking off or layout is carried out to indicate the locations of operation to be done, and provide guidance during rough machining or filing.

Witness marks

The line marked on metal surfaces is likely to be erased due to handling. To avoid this, permanent marks are made by placing punch marks at convenient intervals along the marked line. Punch marks act as a witness against inaccuracies in machining and hence, they are known as witness marks.

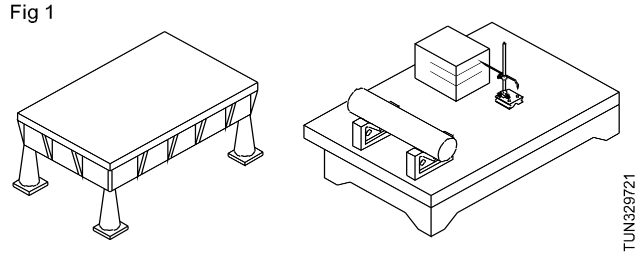
Marking table (Figs 1 and 2)

A marking table (marking-off table) is used as a reference surface for marking on workpieces.

Marking tables are of a rigid construction with accurately-finished top surfaces. The edges are also finished at right angles to the top surface.

Marking tables are made of cast iron or granite, and are available in various sizes. These tables are also used for setting measuring instruments, and for checking sizes, parallelism and angles.

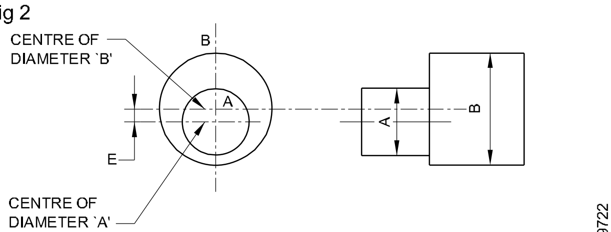
Fig 1



A marking table is very precise as an equipment, and should be protected from damage and rust.
After use, the marking table should be cleaned with a soft cloth.
The surface of the marking table, made of cast iron, should be protected by applying a thin layer of oil.

Indicator reads twice the reading of 'E'. The difference in the maximum and minimum readings of the dial test indicator is called 'throw' (i.e.) throw = 2 E. (Fig 2)

Fig 2



ECCENTRIC DIAMETERS (EACH DIAMETER HAS A DIFFERENT CENTRE)

Eccentric turning

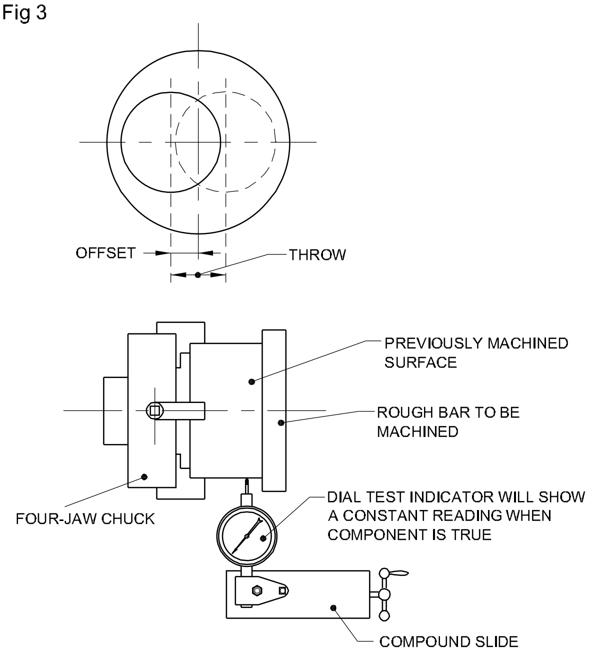
Objectives: At the end of this lesson you shall be able to
- **distinguish between eccentric turning and concentric turning**
- **identify the eccentricity of a turned job**
- **state the methods of turning eccentric jobs**
- **state the uses of eccentric turned jobs.**

Method of identifying eccentricity

The eccentricity of a turned job is tested with the help of a dial test indicator. It is possible to test the offset of the turned job when the job is being held on a 4 jaw chuck.

Fig 3 shows the method of using the dial test indicator for testing the trueness. If the diameters are eccentric, the dial test indicator gives different readings which amount to '2E'. Thus, eccentricity 'E' may be obtained from the two readings. (Fig 3)

Fig 3



TRUING-UP WITH THE DIAL TEST INDICATOR

The other method of testing eccentricity is using a ‘V’ Block and a dial test indicator. In this method, one of the diameters of the eccentric turning is supported in the ‘V’ block and the reading of the other diameter is obtained with the help of the dial test indicator. The difference in the readings gives the throw ‘2E’. Thus eccentricity ‘E’ may be determined by this method.

Method of eccentric turning

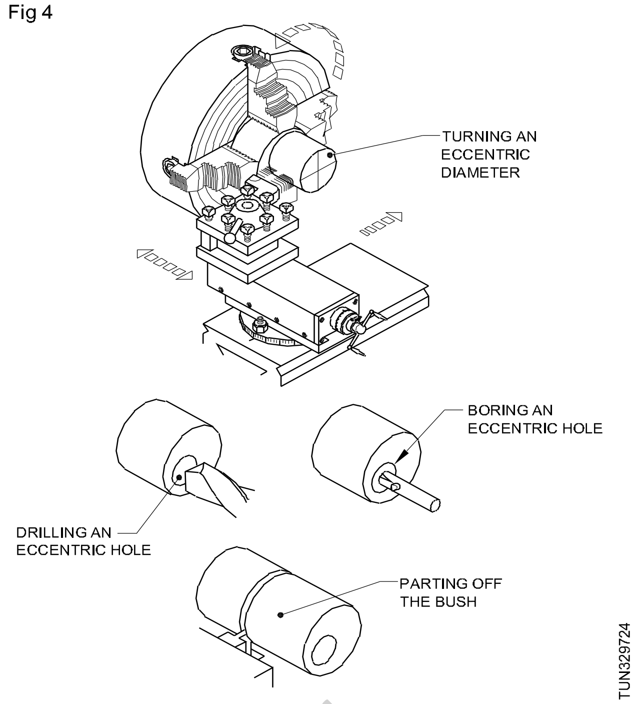
External eccentric turning as well as internal eccentric boring may be accurately carried out on a centre lathe. The figure shows the external eccentric and internal eccentric jobs.

It is possible to turn the eccentric turning with the help of a 4 jaw chuck as well as using a lathe carrier and centres, holding the work between centres.

When a 4 jaw chuck is used, a guide circle of the eccentric axis is essential for truing the eccentric axis. This requirement may be met by the marking process prior to the eccentric turning. With the help of this circle, the ‘Offset’ may be easily made by using a surface gauge. Thus the eccentric axis is located for external eccentric turning and eccentric boring. (Fig 4)

Eccentric turning by using a lathe carrier and centres is done with the help of accurate marking. Before using these accessories, the ‘offset’ has to be marked with the help of marking tools. Both the concentric and the offset centres have to be centre-drilled. By using these centre holes, it is possible to turn eccentric turning on the job.

Fig 4



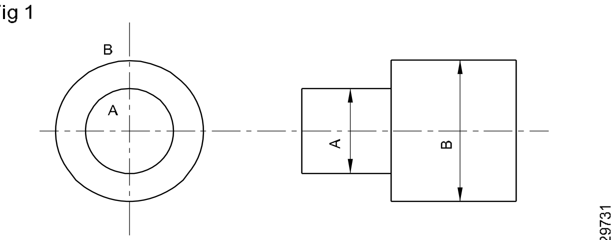
Uses of eccentric turned jobs

Eccentric turned jobs are largely used in automobile industry to convert rotary motion into reciprocating motion. An eccentric-turned job is used in crankshafts. It is used in power press, guillotine machines, and press brakes. It is also generally used in automatic controls.

Concentricity

When different diameters are turned in the same axis, it is said to be concentric turning. Figure 1 shows the two diameters A & B lie on the same axis having the same centre of rotation. If such jobs are tested with a dial test indicator and ‘V’ Block, the dial test indicator shows a constant reading.

Fig 1



CONCENTRIC DIAMETERS (BOTH DIAMETERS HAVE THE SAME CENTERS)

Eccentricity

When different diameters are turned on different axes, it is said to be eccentric turning. The figure shows that the diameters A & B lie on different centres and have a different centre of rotations. The distance E between the centre of rotation is the amount of ‘offset’ or ‘eccentricity’. If the diameter ‘A’ is tested with the dial test indicator by supporting the diameter ‘B’ in the ‘V’ Block, the dial test indicator reads twice the reading of ‘E’. The difference in the maximum and minimum readings of the dial test indicator is called ‘throw’ (i.e.) throw = 2 E. (Fig 1)

Roller and revolving steadies - necessity - construction - uses

Objectives: At the end of this lesson you shall be able to
- **state what is a steady rest**
- **identify and name the various types of steady rests**
- **state the uses of a steady rest**
- **identify the cat head and its use.**

Steady rest uses

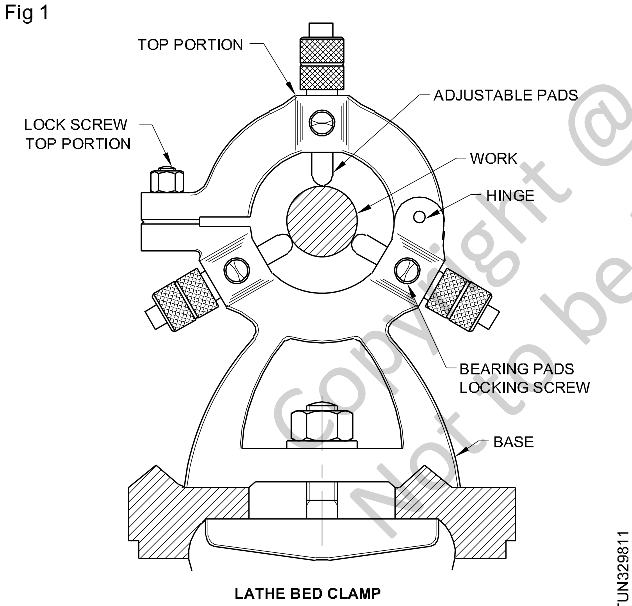
A steady rest is a lathe accessory used to give extra support for a long slender workpiece in addition to the centre support during turning.

The most common types of steady rests are:

- fixed steady rest
- follower steady rest (travelling steady).

Fixed steady rest (Fig 1)

Fig 1



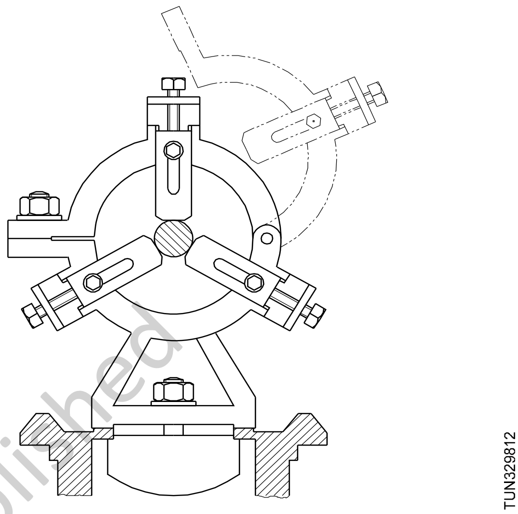
The figure shows the parts of a fixed steady rest.

A fixed steady rest is fixed to the lathe bed and it is stationary. It gives support at one fixed place only.

It consists of a frame containing three adjustable pads.

The base of the frame is machined to suit the inside ways of the lathe bed. The top portion is hinged at the back to permit the top to be lifted or assembled to the bottom half for allowing the work to be mounted or removed. A fixed steady can be clamped at any desired position on the lathe bed by the base clamping screw. (Fig 2)

Fig 2

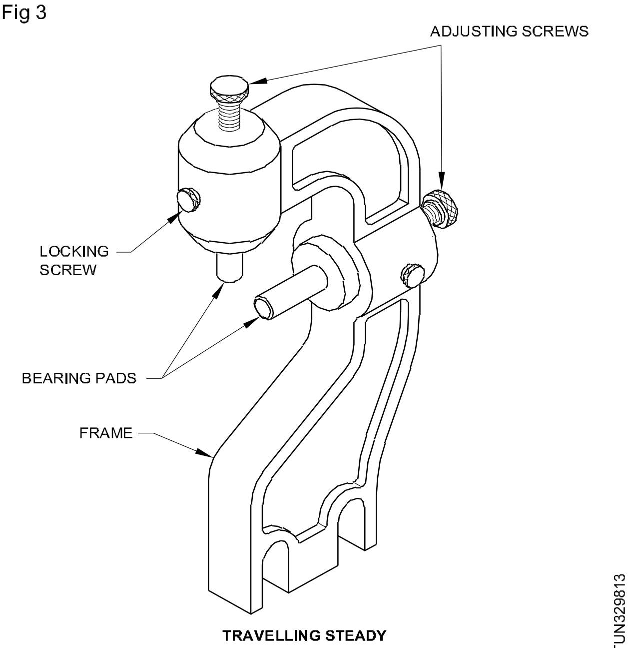


The three adjustable pads can be moved radially in or out by means of adjusting screws. The three pads are adjusted on a trued cylindrical face of the workpiece.

Follower steady rest (Fig 3)

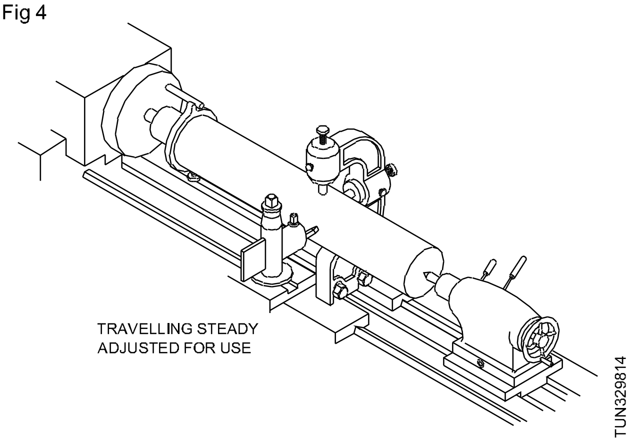
A follower steady is fixed to the saddle of the lathe. As it follows the tool, it gives support where cutting actually takes place. In the follower steady, the support is continuous to the entire length of cutting.

Fig 3



The follower steady rest has usually two pads. One pad is located opposite to the cutting tool and the other pad bears the top of the workpiece to prevent it from springing up. The figure shows a travelling steady rest in action .(Fig 4)

Fig 4



Cat head

If the job shape is not round or where we cannot turn a true cylindrical surface on the job, it is not possible to support the job, by a fixed steady rest. For these types of jobs, a device called cat head is fixed on the workpiece.

The cat head is a type of bush. Its external surface is round. Fig 5 shows a cat head. The middle portion is cylindrical and free to rotate. The two ends have the adjusting screws for holding and centering the work. After centering the work the fixed steady is positioned, and pads are adjusted to hold the cat head's centre portion. When the lathe is running the work revolves along with the ends of the cat head whereas the centre portion is stationary. (Fig 6) Another type of cat head, shown in Fig 7, is a single piece and it rotates along with the job.

Fig 5

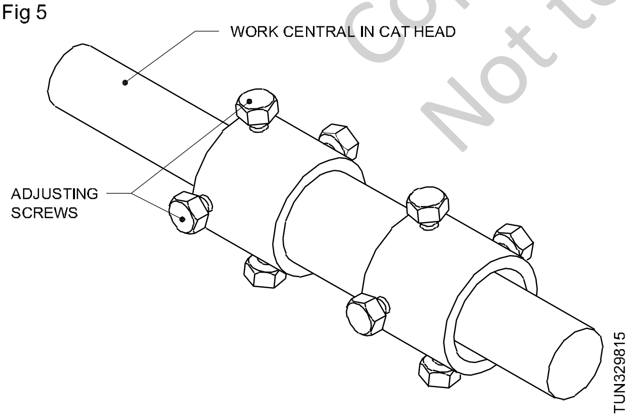
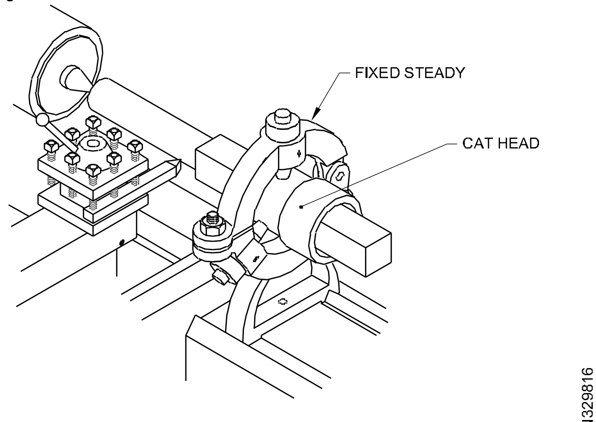
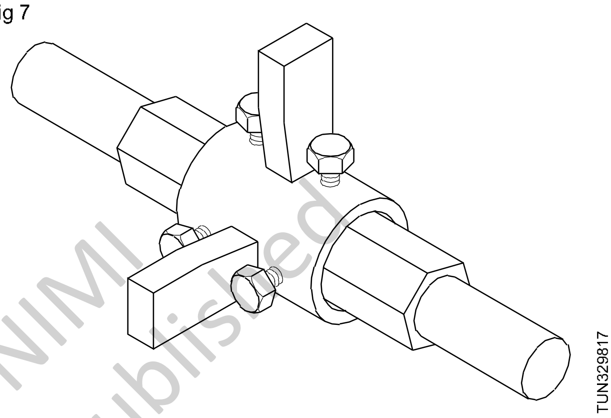


Fig 6



USING CAT HEAD AND FIXED STEADY REST

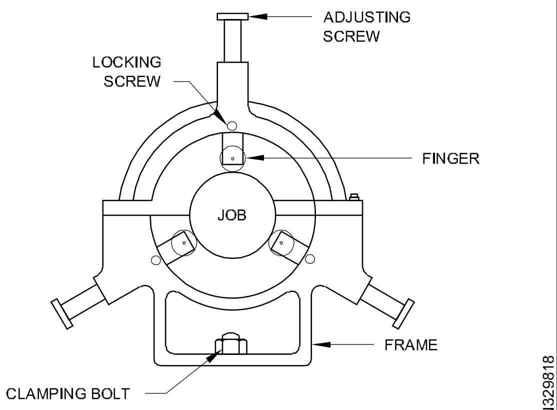
Fig 7



Roller steady (Fig 8)

This steady is similar to fixed steady. It is used to support long thin and thick rods. Difference is that it has wheels is place of three jaws where the job is set between the centres. This steady gives support to work and is easy to rotate due to rollers.

Fig 8



ROLLER STEADY

Different types of attachments used in lathe

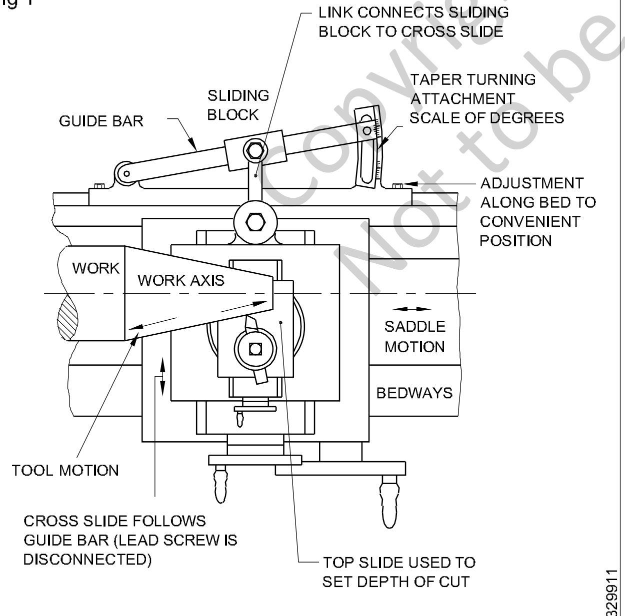
Objectives: At the end of this lesson you shall be able to
- **state the various operations performed in a centre lathe using different attachments**
- **state the features of the taper turing, milling and grinding attachments.**

Taper turning attachment

Many modern lathes have a taper bar fitted at the back of the bed. This can be set to different angles to the spindle axis. The bar carries a sliding block which, during taper turning, is attached by a link to the back of the cross-slide, as shown in Fig 1. The lead screw of the cross-slide is released so that it no longer controls the setting of the depth of cut and the slide is now free. When the saddle is moved along the bed, the cross-slide follows the taper bar, so that the tool moves parallel to the bar and a taper is produced. The top slide is swung through 90° to lie at right angles to the work so that it can be used to apply the depth of cut.

The length of the taper bar enables accurate settings to be carried out, with the help of the degree scale with an angle vernier incorporated. The taper is produced by the movement of the saddle under power feed, giving improved and controllable surface finish and a long taper is possible. It is, however, limited to the half-included angle of the taper of about 15° (30° included angle).

Fig 1



GENERATING A TAPER WITH A TAPER TURNING ATTACHMENT

Milling attachment

This attachment is fitted on to the cross-slide of a lathe in the place of the compound rest. Two types of milling attachments are illustrated in Figures 2 & 3. The attachment shown in Fig 2 holds the job at right angles to the milling cutter,which is mounted in the chuck or collet. In the other type of attachment , the workpiece is held between centres. The milling cutter and the indexing head are mounted on the compound rest. It is provided with a driving unit. Both these attachments have provisions to feed in all the three directions, and it is, therefore, possible to perform operations like keyway cutting, angular milling, Tee slot cutting, and thread milling etc.

Fig 2

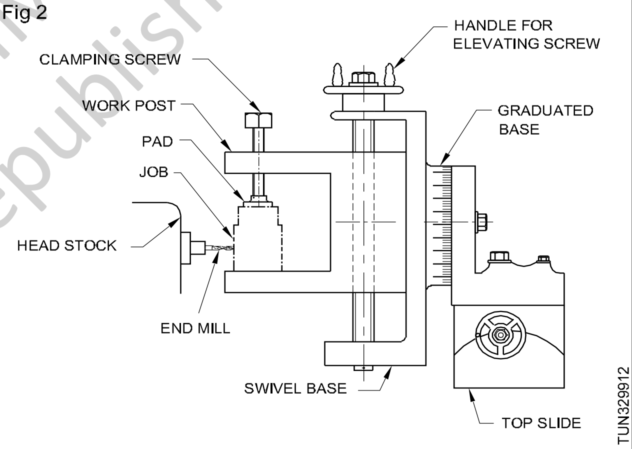
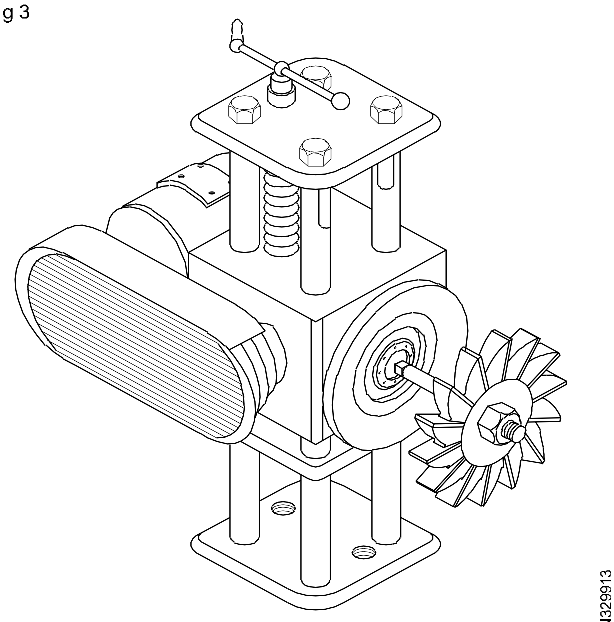


Fig 3



MILLING ATTACHMENT

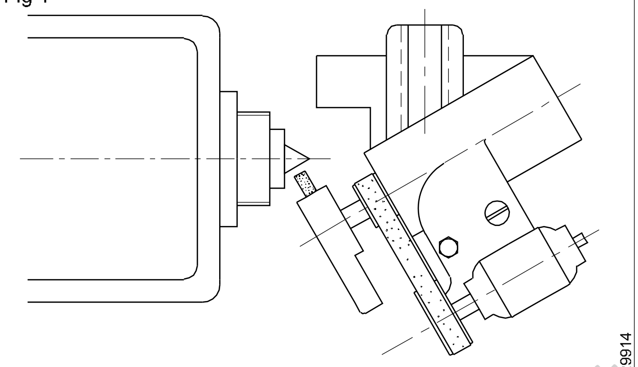
Grinding attachment

With the help of a good electric grinding attachment the lathe can be used for re-sharpening reamers and milling cutters, grinding hardened bushings and shafts, and many other grinding operations.

The V bed ways of the lathe bed should be covered with a heavy cloth or canvas to protect them from dust and grit from the grinding wheel, and the lathe spindle bearings should also be protected. A small pan of water or oil placed just below the grinding wheel will collect most of the grit.

A large, powerful grinder is most satisfactory for external grinding. The wheel should be atleast 100mm in diameter and the grinder should be mounted directly on the compound rest of the lathe, as shown in Fig 4.

Fig 4

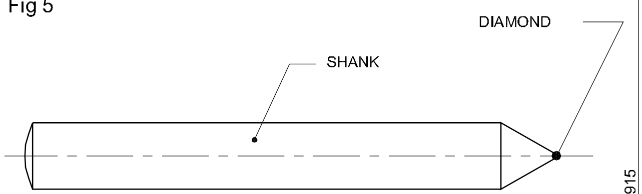


GRINDING A CENTER ON THE LATHE

To obtain a good finish, the grinding wheel must be balanced and must be dressed with a diamond dresser. The grinding wheel must be dressed to keep it true and free from particles of metal which become embedded in the periphery of the wheel.

The diamond dresser consists of a small industrial diamond mounted in a steel shank, as shown in Fig 5. The dresser must be rigidly supported in a fixture for truing the grinding wheel.

Fig 5



DIAMOND DRESSER

The diamond point of the dresser should be placed on the centre, or slightly below centre and the revolving grinding wheel passed across the diamond. Remove about 0.02mm from the wheel for each cut and dress the wheel till it runs true.

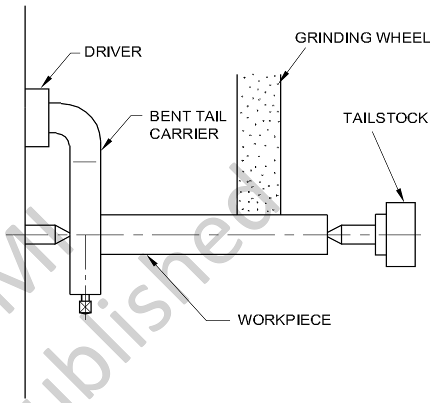
Grinding hardened steel parts

Hardened steel parts should be carefully ground in order to produce a smooth, accurate finish. The part should be machined to within a few microns of the finished size before it is hardened. After hardening, all the scales should be removed before grinding.

Remove only a few microns for every pass of the grinding wheel. If the part is ground too fast it may become overheated and warp, or the temper may be drawn.

Longer parts, such as, shafts, bolts, spindles, etc. are machined by longitudinal grinding. The workpieces are clamped between the centres. (Fig 6)

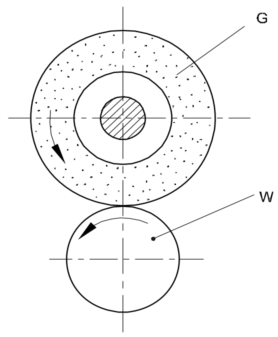
Fig 6



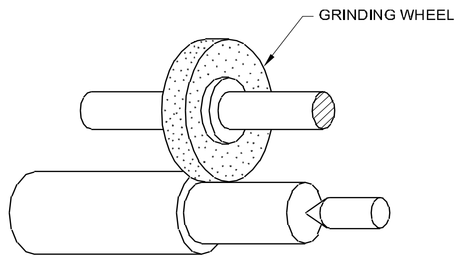
CLAMPING BETWEEN CENTERS

In addition to the proper selection of a grinding wheel, the following has to be observed for economical grinding; cutting speed of the grinding wheel, circumferential speed of the workpiece, feed, depth of cut for each pause, and coolant. (Fig 7)

Fig 7



DIRECTION OF ROTATION OF GRINDING WHEEL (G) AND WORKPIECE (W)



DEPTH ADJUSTMENT DURING GRINDING

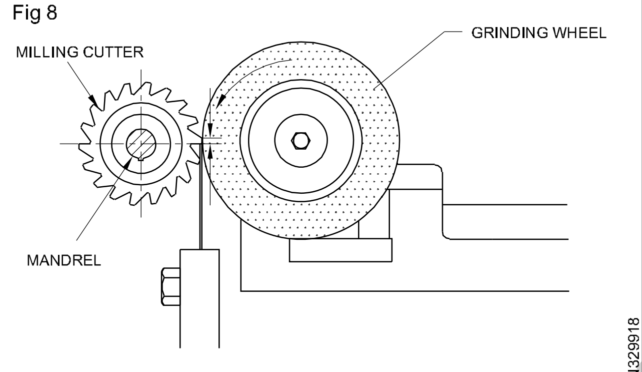
Selection of grinding wheel

Straight grinding wheels are used for grinding shafts. As a rule, soft wheels are more economical than hard ones. Soft wheels remain sharp and cut more in spite of faster wear.

Sharpening reamers and cutters

Reamers and milling cutters may be sharpened by grinding in the lathe. Some reamers are first cylindrically ground, then relieved by grinding with a tooth rest set slightly below centre, as shown in Fig 8 leaving a land 0.05mm to 0.125mm wide. Other reamers and most milling cutters are ground with about 2° relief.

Fig 8



RESHARPENING OF A MILLING CUTTER

Cutting speed and r.p.m. of the grinding wheel

The higher the cutting speed, the faster is the grinding work. The recommended speeds must, therefore, be followed. Higher speeds than the recommended ones are also to be avoided, because the wheels clog, and do not grind any more. The workpiece overheats and the surface becomes inaccurate. In addition, there is also the danger of accidents.

The r.p.m. can be calculated or can be selected from a table.

Circumferential speed and r.p.m. of the workpiece

The circumferential speed is designated in m/min. It affects the quality of grinding; if it is low, the cut will be fine, if it is too high the cut will be coarse.

Calculation of the r.p.m.

V = circumferential speed of the workpiece in m/min.

d = diameter of the workpiece in mm.

N = r.p.m. of the workpiece per min.

r.p.m. of the work piece = $N = \frac{V \times 1000}{\pi \times d}$

Example

A shaft of St 50 with a diameter of 50 mm can be ground. N is to be calculated.

Result : V = 15m/min. as per table.

$$N = \frac{V \times 1000}{\pi \times d} = \frac{15 \times 1000}{3.14 \times 20}$$

= 239 r.p.m.

The depth of cut for roughing may be 0.01 to 0.03 mm and for finishing 0.0025 to 0.005 mm.

Special attachments used in centre lathe

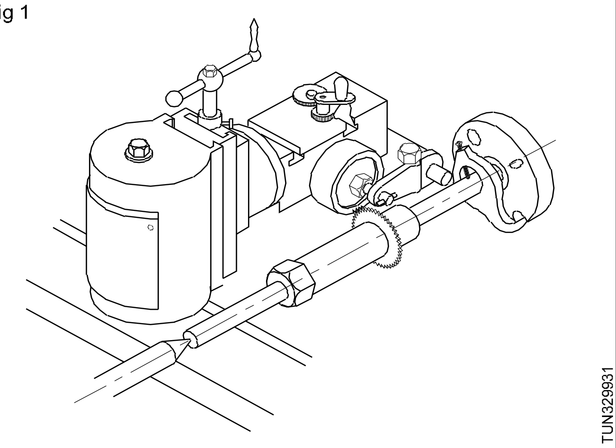
Objective: At the end of this lesson you shall be able to
- **state various operations performed in a centre lathe using the gear cutting, spherical cutting and relieving attachment.**

Gear cutting on centre lathe

The gear cutting attachment mounted on the lathe will cut spur and bevel gears. It may also be possible to do linear indexing, external keyway cutting, splining, slotting and all regular dividing head light milling works.

This attachment is very useful for cutting small gears and work involving light machining. (Fig 1)

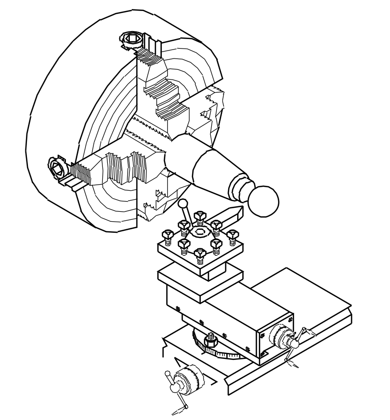
Fig 1



Spherical turning attachment

This attachment is mounted on the cross-slide, and operates by a hand wheel rotating the top slide and tool through a worm and worm wheel. The turning tool is set by using slip gauges between the tool and a test bar fitted.(Fig 2)

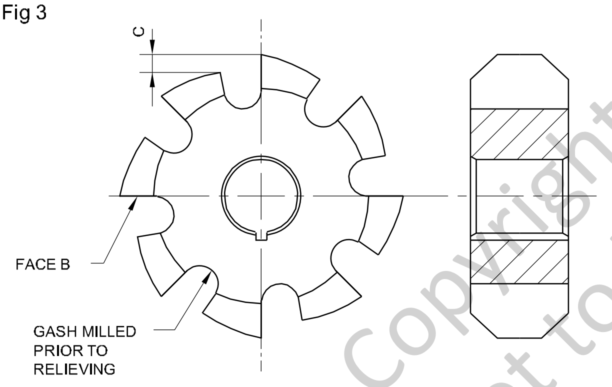
Fig 2



Relieving attachment

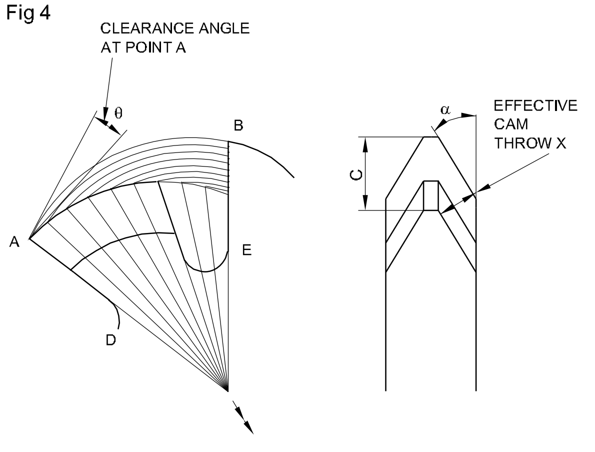
Figure 3 shows a form relieved milling cutter and Fig 4 exhibits the relieved portion in a cutter tooth.

Fig 3



TYPICAL FORM OF RELIEVED MILLING CUTTER

Fig 4



ENLARGED DETAIL OF A TOOTH

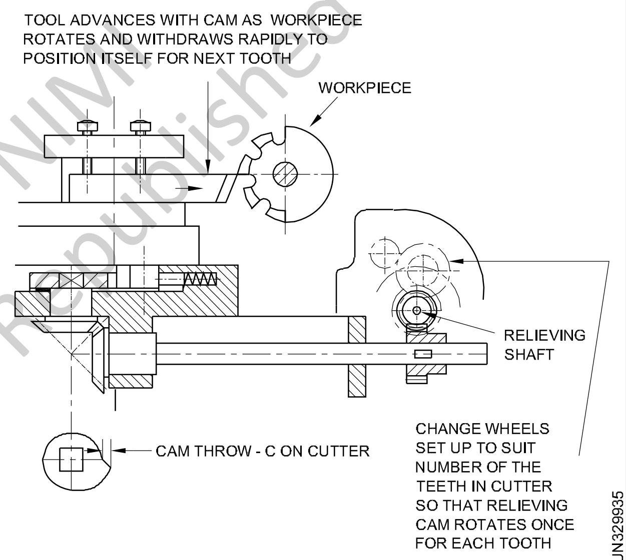
Relieving mechanism

The mechanism for relieving a gear cutter is shown in Fig 5. A relieving shaft, driven from the headstock, drives a cross-shaft which turns the cam located in the cross-slide. The relieving slide carries the tool and is mounted on the cross-slide. A projecting follower engages the cam and is spring-loaded. The rotation of the cam is timed according to the number of teeth in the cutter. The tool advancement is controlled by the cam profile. The tool produces the spiral form on the tooth periphery. When the tool reaches the end of its travel, it is returned to its starting position by the spring and the cycle is repeated.

The relieving shaft with its driving mechanism connected to the standard change wheels provides for dealing with cutters up to a maximum of 20 teeth.

The use of the lead screw is necessary when hobs are being relieved.

Fig 5



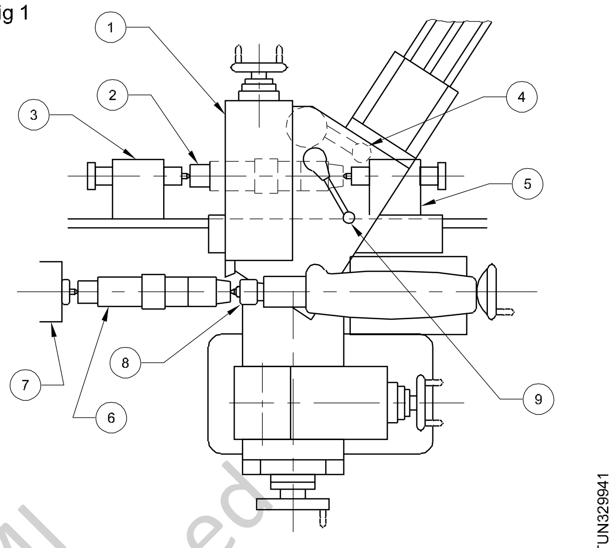
Copying attachment

Objectives: At the end of this lesson you shall be able to
- **identify the parts of a copying attachment**
- **state the function of the copying attachment**
- **state the uses of the copying attachment.**

Parts of the copying attachment (Fig 1)

1 Rear tool slide
2 Master profile
3 Adjustable master head centre
4 Stylus
5 Fixed master head centre
6 Workpiece
7 Headstock drive centre
8 Running centre
9 Control lever for hydraulic slide

Fig 1



The functions of the copying attachment

The copying attachment is generally fixed to certain standard centre lathes. This attachment works on the hydraulic system. Copying lathes are used to produce a particular type of jobs in large quantity.

The job is held on a chuck or between chuck and centre or in between centres. A masterpiece of the job to be produced is held separately parallel to the job axis. The cutting tool used for turning the job is connected to a stylus (tracer) which is switched on and the automatic feed is engaged when the stylus will move from the tail join stock to the headstock with an upward pressure. Since the stylus is in contact with the outer surface of the master piece, the movement of the stylus is guided by the shape of the masterpiece. Hence, similar pieces can be produced in large quantities with the help of the copying attachment.

Uses of the copying attachment.

- The attachment is used for form turning.
- It is used to produce a large number of duplicating parts.
- It is useful in production shops.

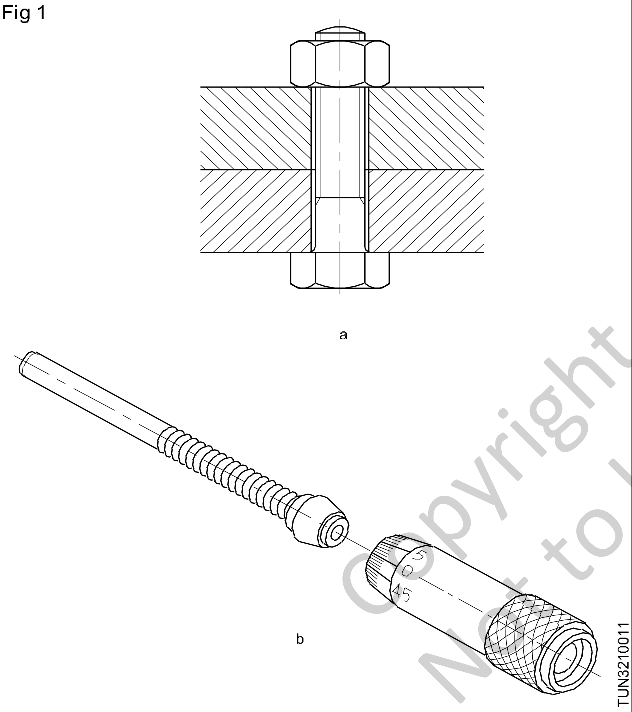
Various procedures of thread measurements

Objectives: At the end of this lesson you shall be able to
- **state the important elements of a thread to be considered while measuring/checking threads**
- **state the function of a screw pitch gauge**
- **identify the different types of thread gauges**
- **state the features checked by the 'Go' side of the screw plug gauge**
- **knowledge about screw thread micrometer and microscope.**

The selection of measuring instruments used for checking the threads depends very much on the accuracy requirement and the feature of the thread to be checked.

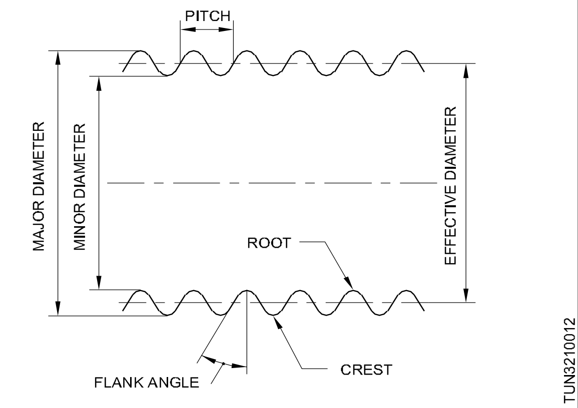
The accuracy requirement varies from a bolt used in structural work to threads of a fine measuring instrument. (Figs 1a and 1b)

Fig 1



The surface of a screw has a complex shape. The following elements (Fig 2) of a screw thread are to be considered in thread measurement

Fig 2



- Major diameter
- Minor diameter/root diameter
- Pitch
- Effective diameter
- Thread angle
- Form of root and crest

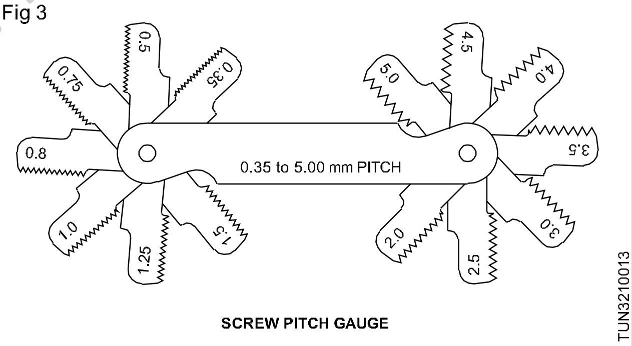
The above elements contribute to the strength and interchangeability of the threads.

The most important elements to be checked are:

- the pitch of the thread
- the angle and
- the effective diameter.

Screw pitch gauge (Fig 3)

Fig 3



This gauge is mainly used to check the pitch of external and internal threads. (Figs 4 & 5)

This consists of a number of blades with accurate notches made to the profile and pitch of the thread.

Fig 4

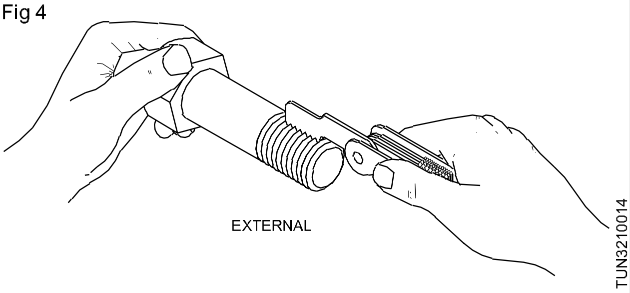
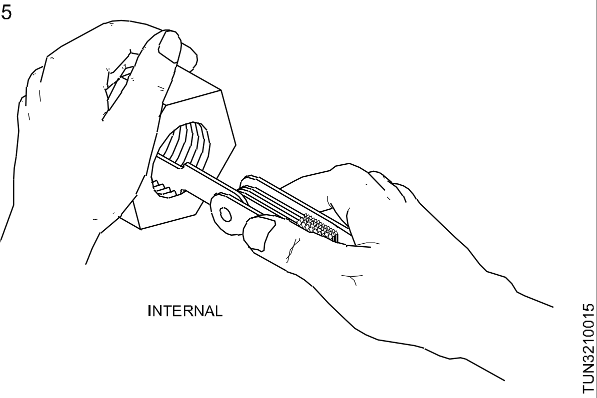


Fig 5

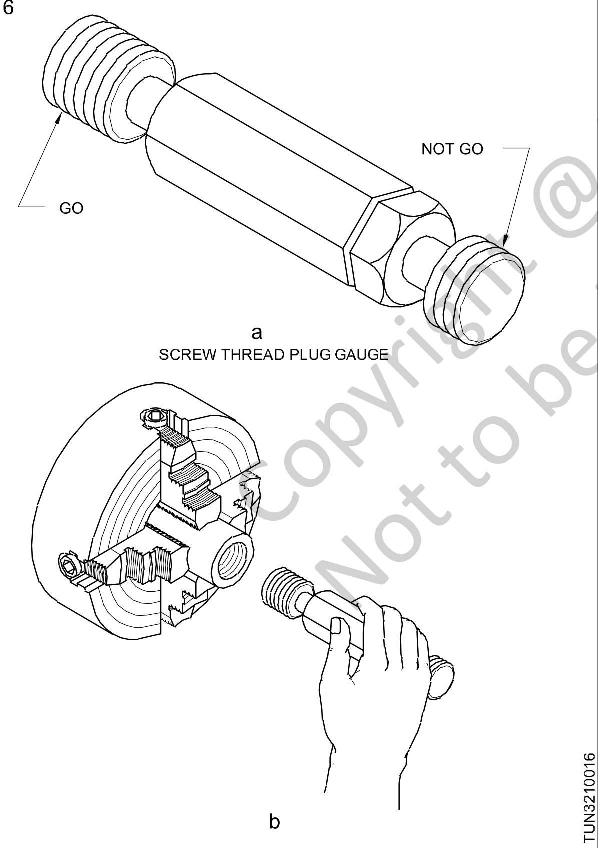


The decision about the correctness of the pitch is taken by comparing them by placing the appropriate blade on the screw. Each blade has an indication about the size of the screw and the pitch.

Example

Thread gauges M20 x 1.75 mm. (Fig 6a)

Fig 6



The screw thread plug gauge is used to check the internal thread. (Fig 6b) It checks whether a thread dimension is within its tolerance. The 'Go' side of the gauge checks the following. (Fig 7)

The profile angle (b)

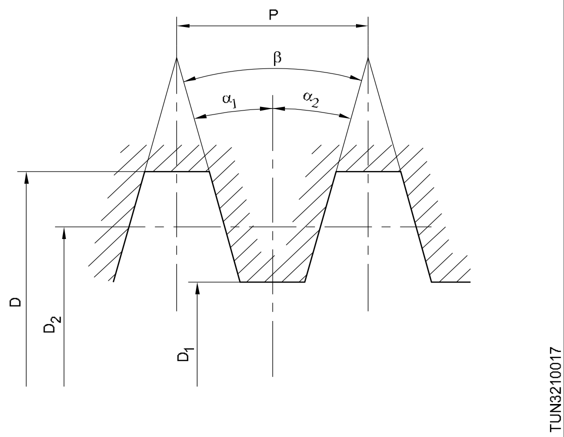
The pitch (P)

The major diameter (D)

The effective diameter(D_2)

The minor diameter (D_1)

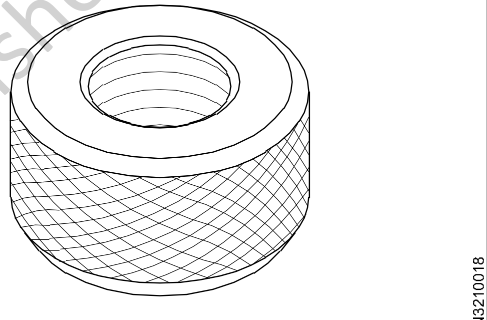
Fig 7



Thread ring gauge (Fig 8)

This is used to check the external thread for its accuracy. 'Go', 'No Go' gauges are used to check whether the thread is within tolerance.

Fig 8

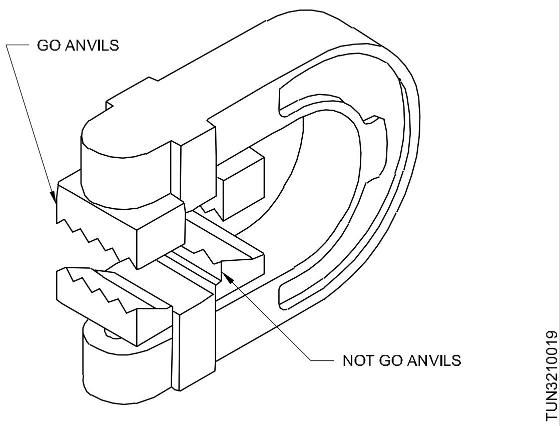


SCREW THREAD RING GAUGE

Screw thread caliper gauge

This is used for checking external threads. This gauge is a highly efficient type. This finds greater usage than the ring gauge for checking external threads. In this, the external threads are gauged with a caliper type gauge with two sets of anvils (Fig 9) representing the 'Go' and 'No Go' conditions.

Fig 9



The 'Go' anvils have full thread form (Fig 10) and are set to ensure no element of the thread is oversize. The 'No Go' anvils have truncated thread form (Fig 11) to ensure that the contact is made only on the flanks of the thread and checks that the effective diameter of the workpiece thread is not undersize. The gauges are adjustable and are set by means of master setting plugs. The gauges can be used for right or left hand threads.

Fig 10

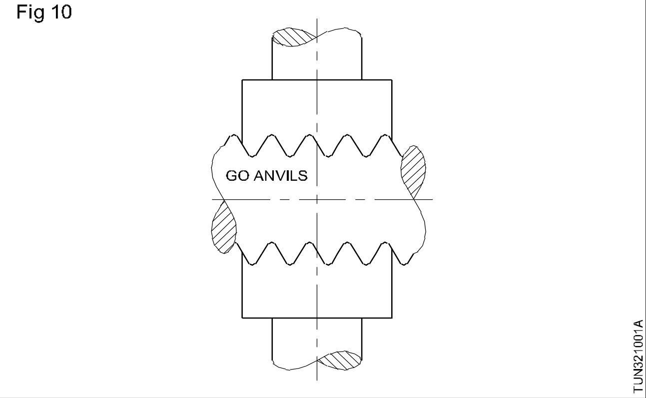
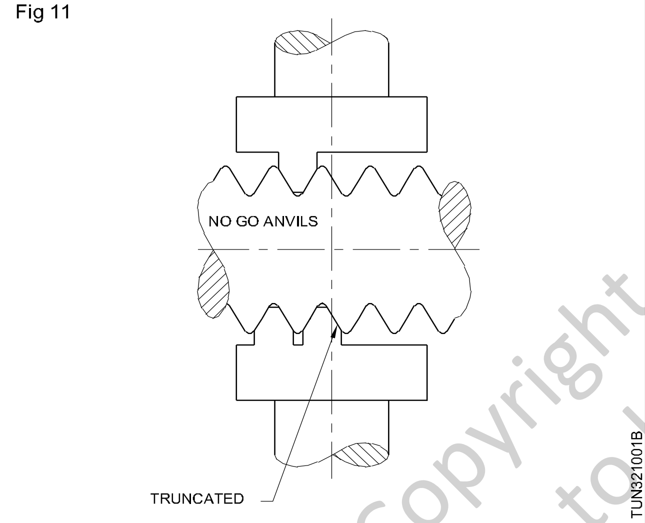


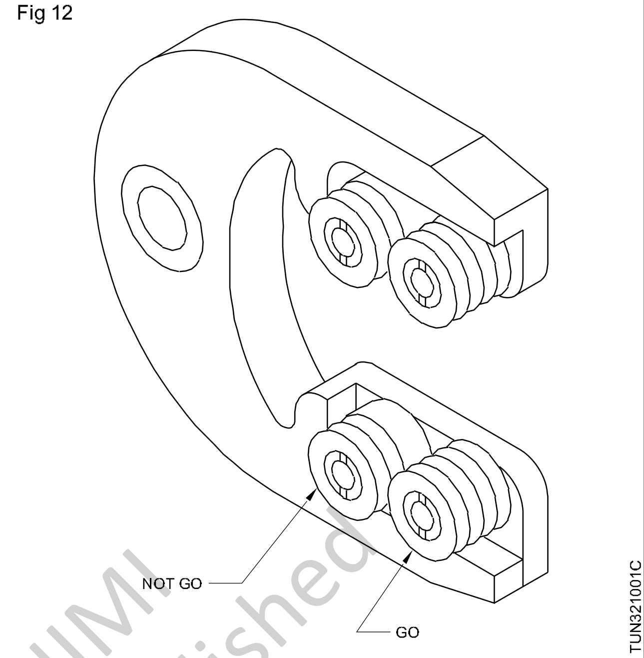
Fig 11



The caliper gauge with roller type anvils

This is another type of thread gauge. (Fig 12) The 'Go' rollers have a full form of thread while the 'No Go' rollers are truncated.

Fig 12



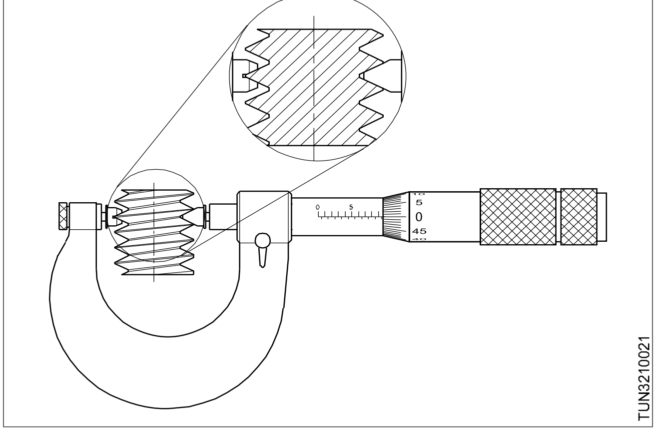
Thread measurement (effective diameter)

Objectives: At the end of this lesson you shall be able to
- **state the features of a screw thread micrometer**
- **state the features of the three-wire system of measurement with the help of tables**
- **select the best wire with the help of tables for using in the three-wire method.**

The screw thread micrometer

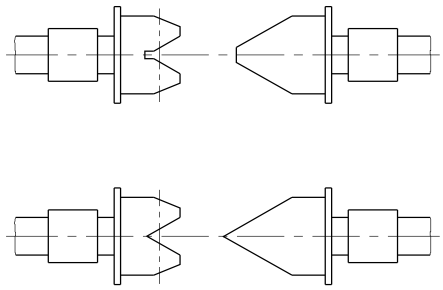
This micrometer (Fig 1) is used to measure the effective diameter of the screw threads. This is very similar to the ordinary micrometer in construction but has facilities to change the anvils.

Fig 1



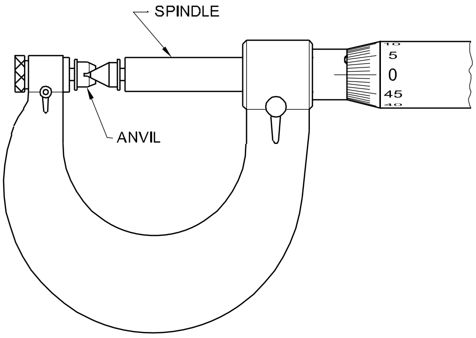
The anvils are replaceable and are changed according to the profile and pitch of the different systems of threads. (Figs 2 & 3)

Fig 2



INTERCHANGEABLE ANVILS

Fig 3



SPINDLE AND ANVIL OF MICROMETER AT ZERO SETTING

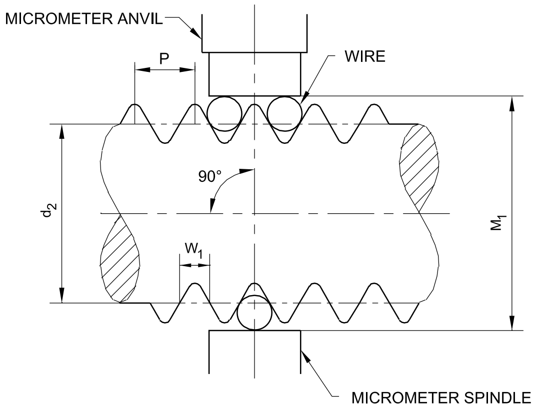
The three-wire method

This method uses three wires of the same diameter for checking the effective diameter and the flank form. The wires are finished with a high degree of accuracy.

The size of the wires used depends on the pitch of the thread to be measured.

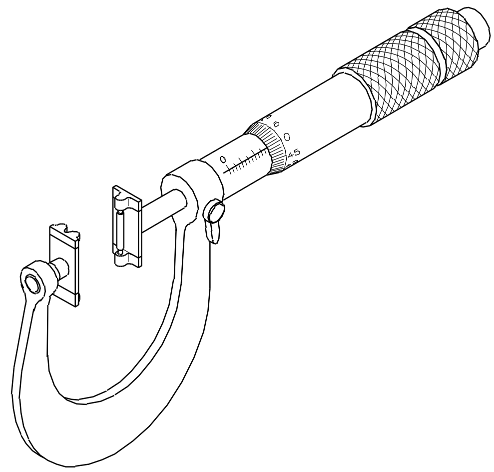
For measuring the effective diameter the three wires suitable for the thread pitch are placed between the threads. (Fig 4)

Fig 4



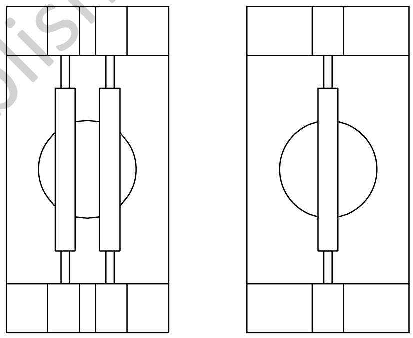
The measuring wires are fitted in wire-holders which are supplied in pairs. One holder has provisions to fix one wire and the other for two wires. (Fig 5)

Fig 5



While measuring the screw thread, the holder with one wire is placed on the spindle of the micrometer and the other holder with two wires is fixed on the fixed anvil. (Fig 6)

Fig 6

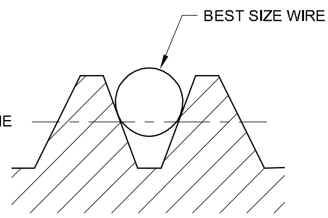


Selection of 'best wire' (Fig 7)

The best wire is the one which, when placed in the thread groove, will make contact at the nearest to the effective diameter. The selection of the wire is based on the type of thread and pitch to be measured.

The selection of the wire can be calculated and determined but readymade charts are available from which the selection can be made. (Table 1 to 4).

Fig 7



CONTACT AT PITCH LINE

Table 1

Measurement with measuring wires. Metric threads with coarse pitch (M)

Thread designa-tion	Pitch P mm	Basic measu-rement d_2 mm	Measuring wire dia. mean W_1 mm	Dimension over wire M_1 mm
M 1	0.25	0.838	0.15	1.072
M 1.2	0.25	1.038	0.15	1.272
M 1.4	0.3	1.205	0.17	1.456
M 1.6	0.35	1.373	0.2	1.671
M 1.8	0.35	1.573	0.2	1.870
M 2	0.4	1.740	0.22	2.055
M 2.2	0.45	1.908	0.25	2.270
M 2.5	0.45	2.208	0.25	2.569
M 3	0.5	2.675	0.3	3.143
M 3.5	0.6	3.110	0.35	3.642
M 4	0.7	3.545	0.4	4.140
M 4.5	0.75	4.013	0.45	4.715
M 5	0.8	4.480	0.45	5.139
M 6	1	5.350	0.6	6.285
M 8	1.25	7.188	0.7	8.207
M 10	1.5	9.026	0.85	10.279
M 12	1.75	10.863	1.0	12.350
M 14	2	12.701	1.15	14.421
M 16	2	14.701	1.15	16.420
M 18	2.5	16.376	1.45	18.564
M 20	2.5	18.376	1.45	20.563
M 22	2.5	20.376	1.45	22.563
M 24	3	22.051	1.75	24.706
M 27	3	25.051	1.75	27.705
M 30	3.5	27.727	2.05	30.848

Table 2

Measurement with measuring wires. Metric threads with fine pitch (M)

Thread designa-tion	Basic measure-ment d_2 mm	Measuring wire dia. mean W_1 mm	Dimension over wire M_1 mm
M 1 x 0.2	0.870	0.12	1.057
M 1.2 x 0.2	1.070	0.12	1.257
M 1.6 x 0.2	1.470	0.12	1.557
M 2 x 0.25	1.838	0.15	2.072
M 2.5 x 0.35	2.273	0.2	2.570
M 3 x 0.35	2.773	0.2	3.070
M 4 x 0.5	3.675	0.3	4.142
M 5 x 0.5	4.675	0.3	5.142
M 6 x 0.75	5.513	0.45	6.214
M 8 x 1	7.350	0.6	8.285
M 10 x 1.25	9.188	0.7	10.207
M 12 x 1.25	11.188	0.7	12.206
M 14 x 1.5	13.026	0.85	14.278
M 16 x 1.5	13.026	0.85	14.278
M 18 x 1.5	17.026	0.85	18.277
M 20 x 1.5	19.026	0.85	20.277
M 22 x 1.5	21.026	0.85	22.277
M 24 x 2	22.701	1.15	24.420
M 27 x 2	25.701	1.15	27.420
M 30 x 2	28.701	1.15	30.419

Thread measurement (minor diameter)

Objectives: At the end of this lesson you shall be able to
- **name the different methods used for checking the minor diameter of external threads**
- **name the methods of measuring the minor diameter of internal threads**
- **state the features of the methods adopted for measuring the minor diameters of the internal threads.**

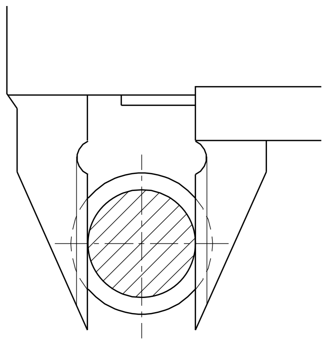
The measurement of the minor diameter and checking the form of the thread are important for producing accurate threads.

Checking the thread minor diameter

Use of the vernier caliper

The knife edge of a vernier caliper can be used for measuring the minor diameter - within a reasonable degree of accuracy. (Fig 1)

Fig 1



Checking with the micrometer

Two methods are adopted using the micrometers,

- Using a Vee piece/prism and ordinary micrometer.
- Using special micrometers with pointed anvils.

Using Vee piece/prism

In this method the minor diameter is determined by using a Vee piece/prism of known dimensions from the apex to the base. (Fig 2) The measurement is taken over the work and the Vee piece. (Figs 3 and 4)

Fig 2

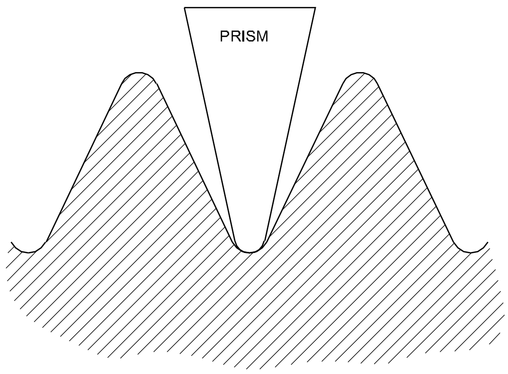


Fig 3

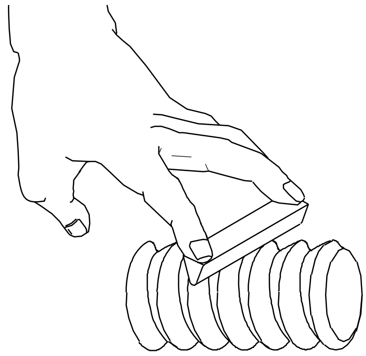
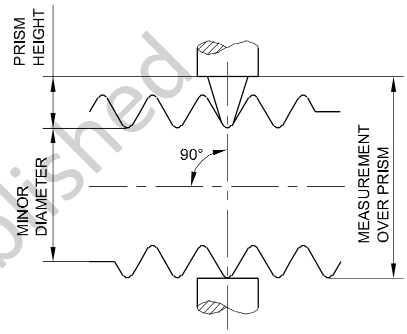


Fig 4



Selection of the size of the prism is very important for accurate measurement.

The sizes of the prisms are indicated by A, B, C and D.

Table 1 will help in determining the correct size of the prism for the different types and pitches of the threads.

For determining the minor diameter, first measure the major diameter of the threaded piece.

Then measure by placing one prism in the thread as shown in Fig 4. It may be noted that one of the anvils of the micrometer is on the prism and the other on the major diameter of the thread.

Table 1

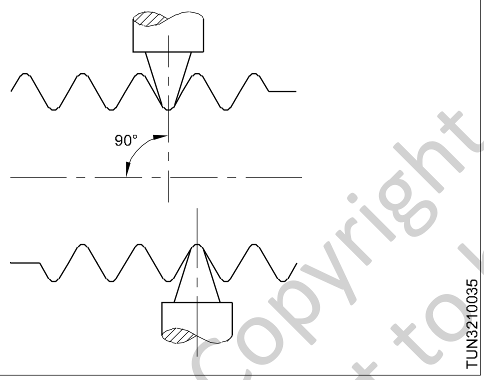
Prism designa-ting size	Thread form		
	Metric pitch in mm	Unified BSW threads/ inch	BA No.
A	1.0-1.25	56-44	9-16
B	1.5-2.25	40-28	3-8
C	2.5-4.5	26-14	0-2
D	5.0-6.0	12-4	

This will help to determine the core diameter of the thread. The core diameter = Measurement over the prism – height of the prism – single depth of the thread.

Using special micrometers

A special micrometer which can accommodate specially shaped anvils is used for this. This directly measures the minor diameter. It is important to ensure that the micrometer is placed perpendicular to the axis of the thread being measured (Fig 5) for accurate measurement.

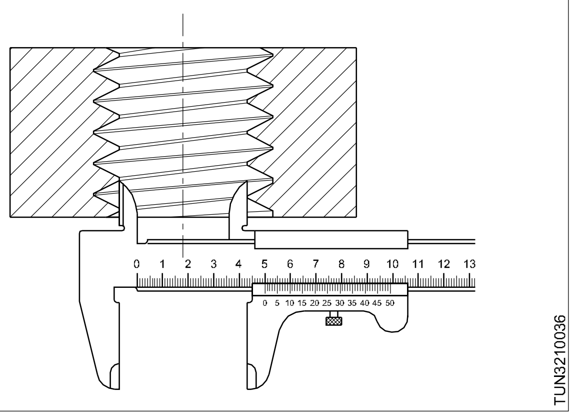
Fig 5



Measuring minor diameter of internal threads

The knife edge of a vernier caliper can be used to measure the minor diameter of an internal thread. This cannot be adopted when very accurate measurements are to be taken. (Fig 6)

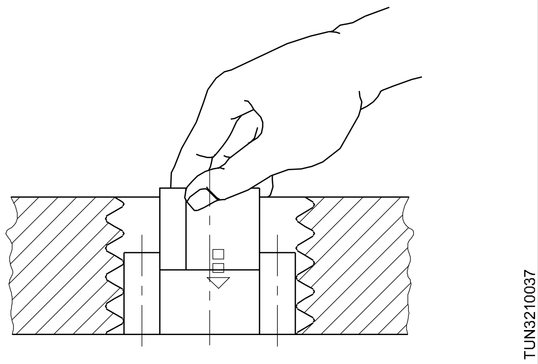
Fig 6



Direct accurate measurement of internal minor diameter is a difficult task.

The commonly used methods are by using slip gauges and precision rollers (Figs 7 & 8) and taper parallels and micrometer. (Fig 9)

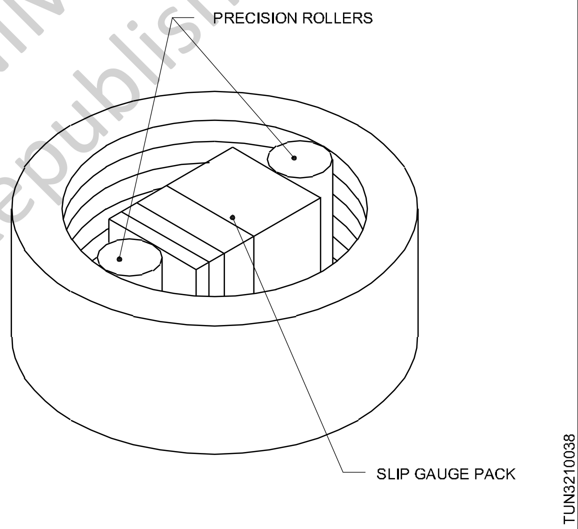
Fig 7



Using slip gauges and precision rollers

While using slip gauges and precision rollers of known diameter, the rollers are first placed diametrically opposite the bore. (Fig 8)

Fig 8



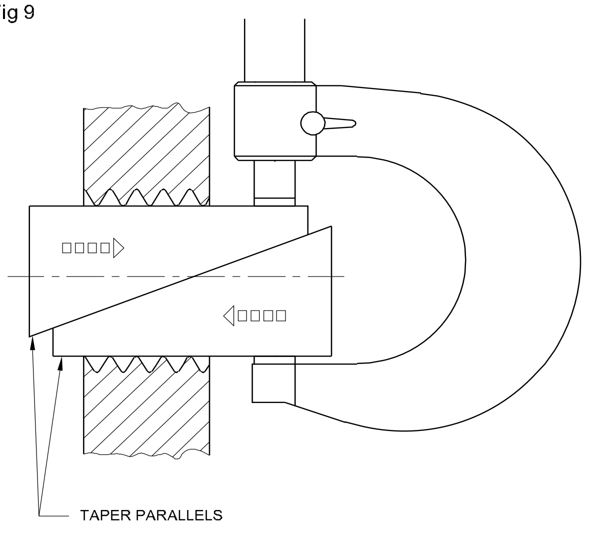
Then the slip gauge pack is selected until it just slides between the rollers.

The sum of the size of the slip gauge pack and the diameters of the rollers is the minor diameter.

Using taper parallels

Precision tapered parallels can be inserted as shown in Fig 9 and the measurement can be taken using a micrometer.

Fig 9



Screw thread measurement (flank angle and form)

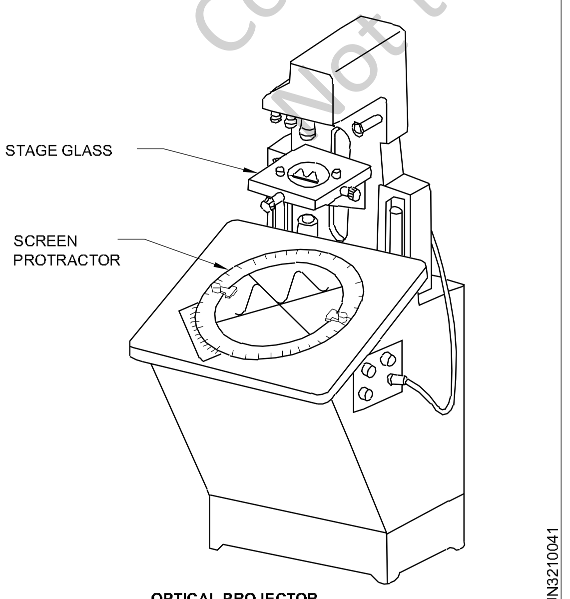
Objectives: At the end of this lesson you shall be able to
- **state the method of checking the flank angle and form of the external threads**
- **state the method of checking the form and features of the internal threads.**

The measurement of the flank angle and form of the threads is carried out using the optical projection method.

The profile of the external screw thread is projected on the screen (Fig 1) in a magnified form. The angle of the image can be measured using the screen with a protractor. The screen line and the flank image are accurately aligned and the angles are measured. (Fig 2)

The form of the thread can be compared against the projected image of the template (Fig 3) on the optical projector screen.

Fig 1



OPTICAL PROJECTOR

Fig 2

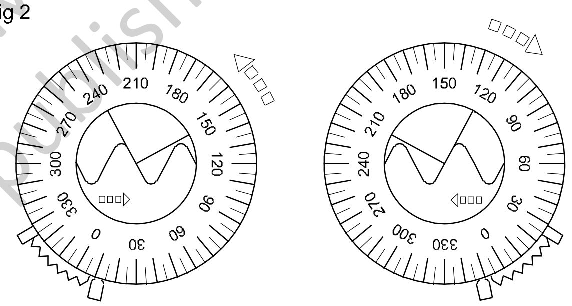
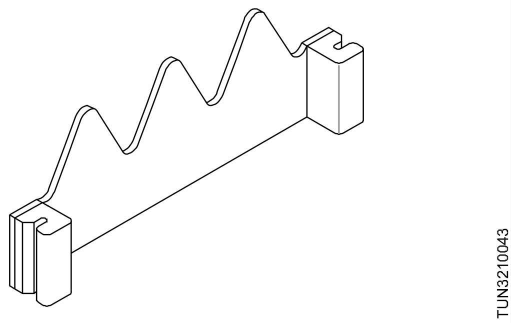


Fig 3



Checking flank angle and form of an internal thread

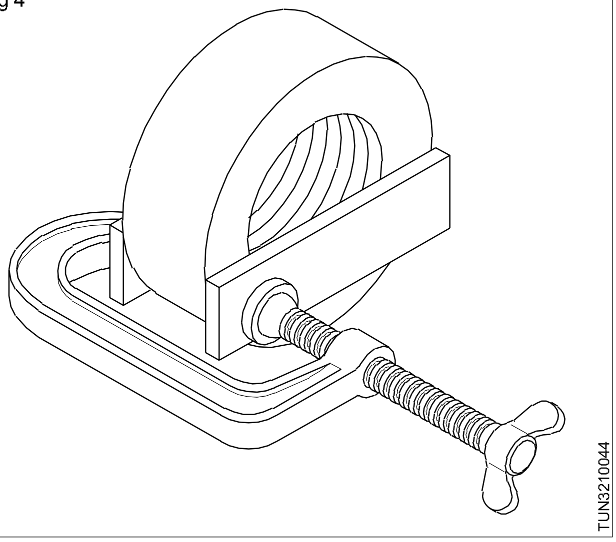
A method adopted for checking the flank angle and form of the thread is by preparing a cast form of the thread and comparing the cast using an optical projector.

Dental wax or superfine plaster of paris can be used for preparing the cast mould.

For preparing the cast form, the specimen is first cleaned and then a thin film of oil is applied on the thread.

Both ends are then blocked with metal pieces using clamps. (Fig 4)

Fig 4



The plaster which is mixed to a thin cream consistency is poured in the prepared thread cavity (Fig 5).

After the plaster is set the metal strips are removed and the cast taken out carefully.

The form of the thread thus obtained (Fig 6) is checked using an optical profile projector.

Microscope

For thread measuring inspection generally tool maker's microscope is used, with attachments of thread inlays. This instrument can read upto an accuracy of 10 microns. In Fig 7 indicates the elements of microscope used for thread inspection.

Fig 5

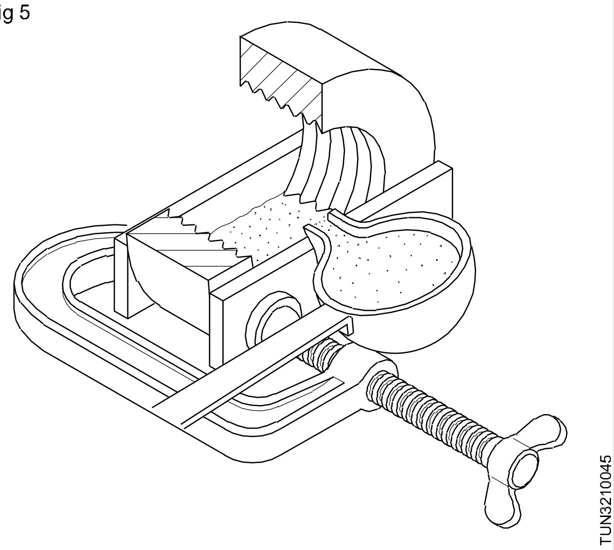
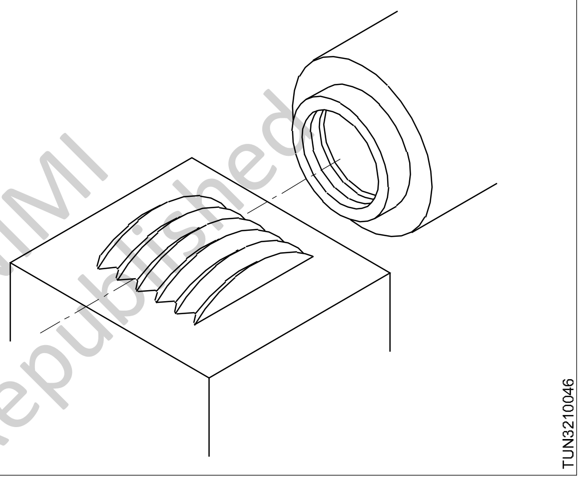
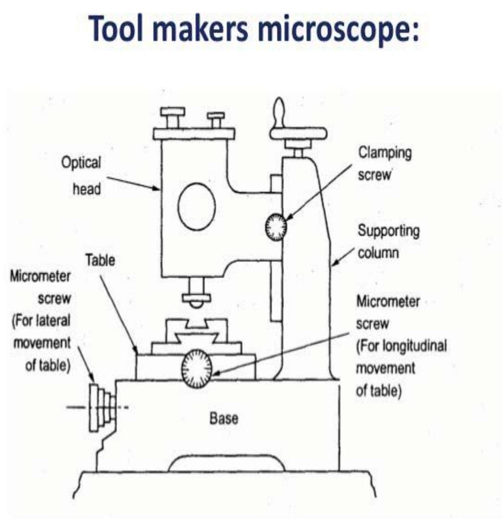


Fig 6





Toolmaker’s button and its parts

Objectives: At the end of this lesson you shall be able to
- **state the construction of a toolmaker’s button**
- **state the use of toolmaker’s button**
- **state the method of boring accurately with reference to a datum.**

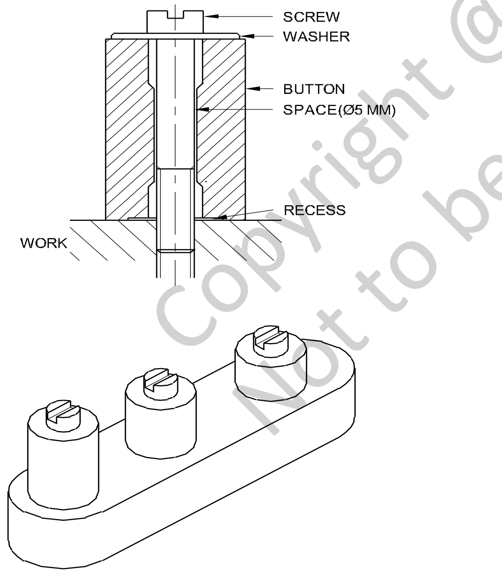
Tools marker’s button uses

The toolmaker’s button is used for producing a bore to a high degree of positional accuracy with reference to both the axes i.e. with reference to the two edges of the workpiece.

Construction: Tool makers’s button consists of a hardened cylindrical ends, a washer, screw as shown in Fig 1.

A toolmaker’s button (Fig 1) is a hardened and ground cylinder made of steel of 8, 10 or 12 mm diameter. The ends of the button are accurately square to its cylindrical axis. It has a 4 BA screw to hold the button in position on the workpiece, allowing sufficient clearance between the screw and the button hole, so that adjustments are possible during the alignment.

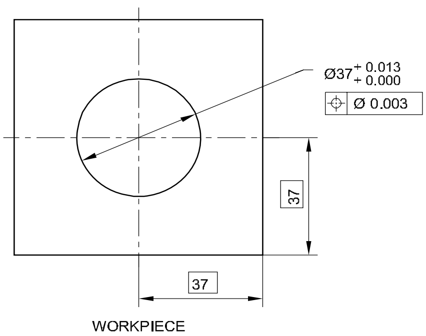
Fig 1



TUN3310111

Application (Fig 2)

Fig 2



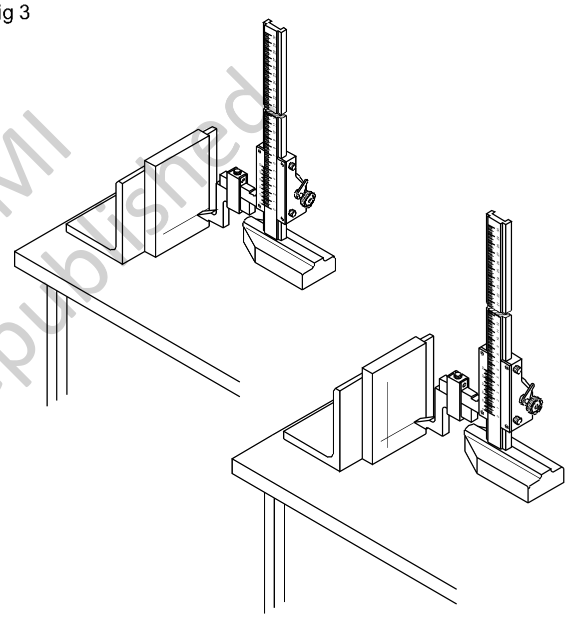
TUN3310112

In the component shown here, a hole of Ø 37 mm is to be bored to a positional tolerance of 0.003 mm. The following procedure is to be followed.

Procedure

Mark out the hole centre position with a height gauge. (Fig 3)

Fig 3

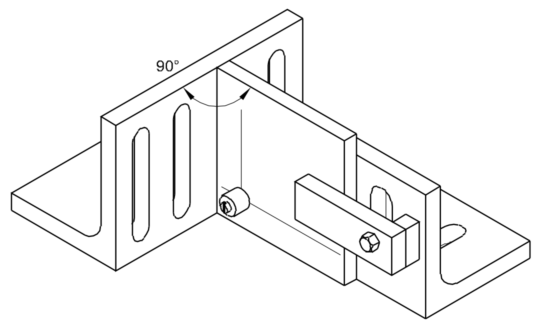


TUN3310113

At the marked centre position, drill and tap a 4 BA threaded hole.

Fit a toolmaker’s button over this hole, but do not fully tighten the holding screw. (Fig 4)

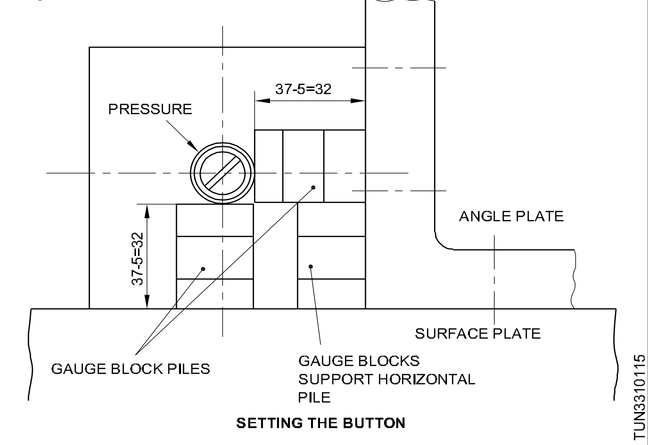
Fig 4



TUN3310114

Adjust the button position by piles of gauge blocks equal to T.P. (True position dimensions minus 5 mm button radius as shown in (Fig 5)

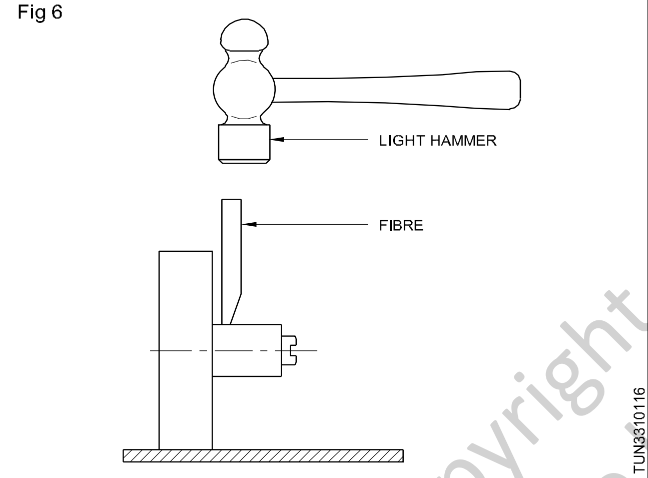
Fig 5



SETTING THE BUTTON

While positioning the button, tap gently as shown in (Fig 6).

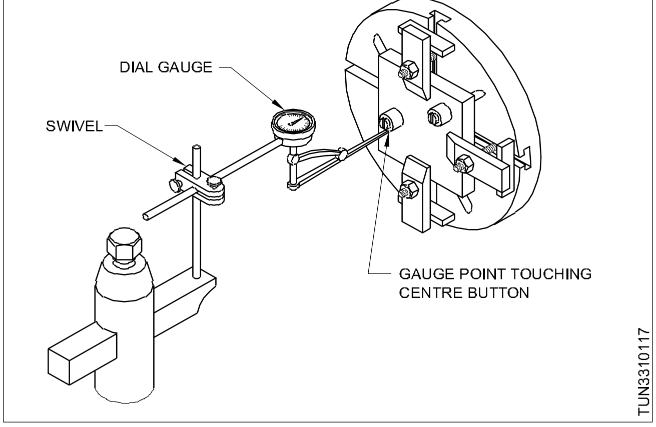
Fig 6



When the button position is correct, tighten the screw fully and ensure that the button position has not changed, due to tightening of the screw.

Mount the work on the face plate (or in the 4 - jaw chuck) and adjust the position of the work and not the button, until the button runs true. Use a dial test indicator for truing. (Fig 7)

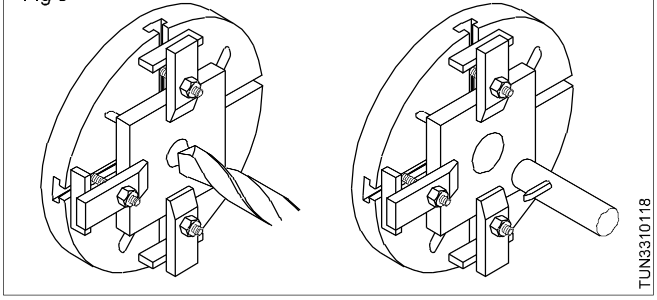
Fig 7



As the button has been positioned accurately on the axis of the required hole, the work is now rotating on this axis.

Remove the button and drill the tapped hole, which may be eccentric, to the required size, leaving machining allowance for boring. (Fig 8)

Fig 8



Now bore the hole to size i.e. Ø 37 mm

Performing this operation with the job held in a face plate is better than using a four-jaw chuck. It is because of the fact that the faceplate surface is square to the axis of rotation, the work face is set square to the axis of rotation with ease.

This method of button boring may be applied on to a job with a number of holes to be located accurately. The correct centres can all be preset with toolmaker's buttons and bored one at a time. (Figures 9 and 10) These figures illustrate the method of marking and setting a jig plate with two bores.

The job is first set on the face plate and hole No.1 is centered and button-bored.

To shift the hole No.2 to the centre position, the job is moved while strips remain fixed and slip gauges are inserted between locating edges and strips. Then the hole No.2 is button-bored to the size.

Fig 9

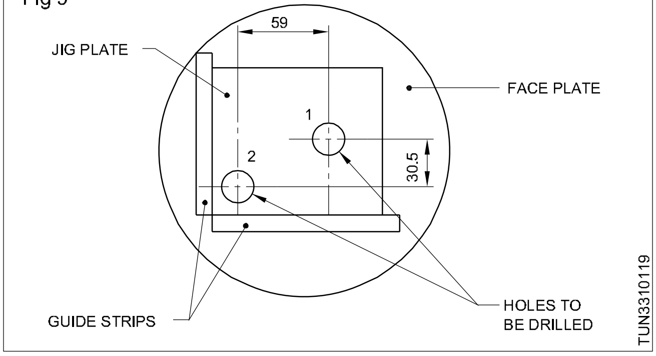
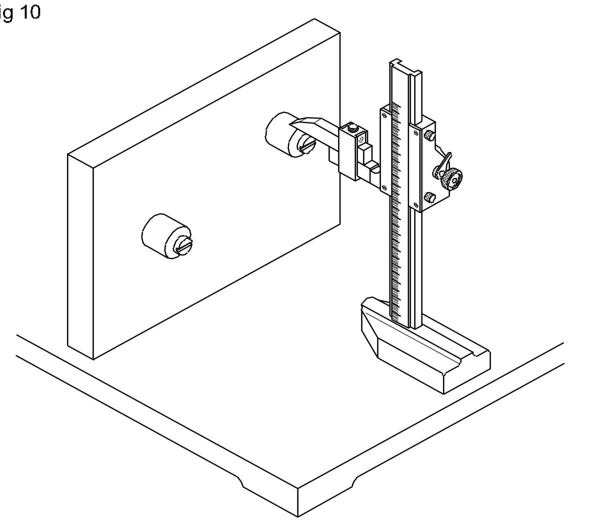


Fig 10



SETTING BUTTONS WITH HEIGHT GAUGE

Telescopic gauge - construction - uses

Objectives: At the end of this lesson you shall be able to
- **name the parts of a telescopic gauge**
- **state the constructional features of telescopic gauges.**

Telescopic gauges are popular for fine work as they are very rigid and have better 'feel'.

Uses

Telescopic gauges are used for checking the sizes of holes, slots and recesses.

Construction

Telescopic gauges are 'T' shaped. Each gauge consists of a pair of teleschopic tubes or plungers connected to a handle. (Fig 1) The plungers are spring - loaded to force them apart. After inserting it in a hole or slot, the gauge can be locked in position by turning the knurled handle. It may then be withdrawn from the hole and measured with a micrometer (Fig 2)

Fig 1

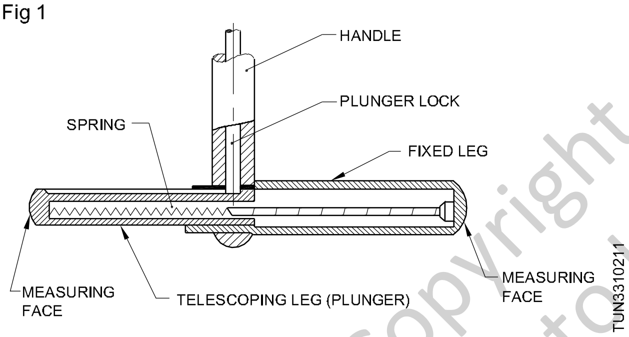
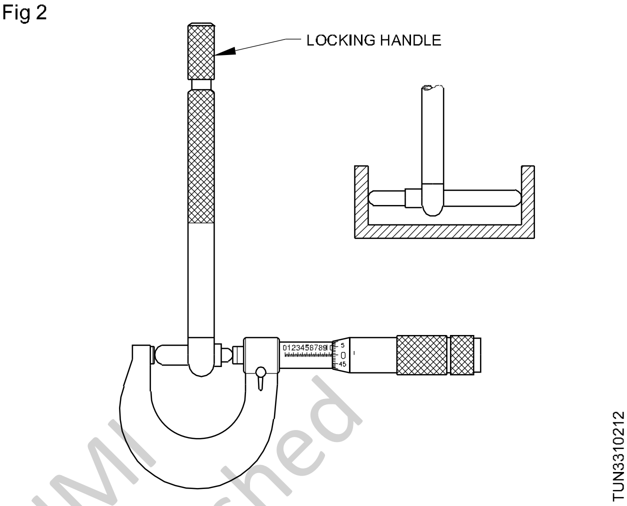
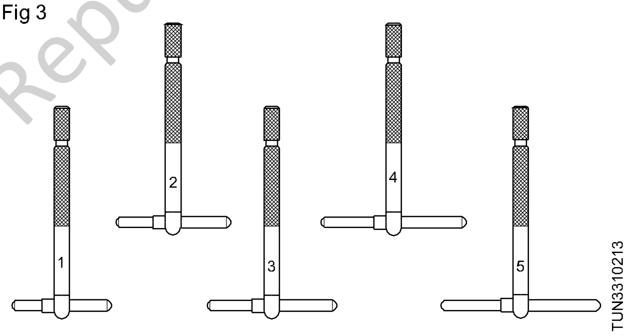


Fig 2



Telescopic gauges are available in a set of 5 Nos. to measure holes from 12.7mm to 152.4mm (Fig 3)

Fig 3



No 1	12.7mm to 19mm
No 2	19.0mm to 31.7mm
No 3	31.7mm to 53.9mm
No 4	53.9mm to 88.9mm
No 5	88.9mm to 152.4mm

Inside micrometer - metric - construction

Objectives: At the end of this lesson you shall be able to
- **name the parts of an inside micrometer**
- **determine the reading of the bore or hole**
- **determine the reading with a spacing collar & extension rods**
- **determine the distance between internal parallel surfaces.**

Construction

The inside micrometer is similar to an ordinary outside micrometer but without the 'U' frame. (Fig 1) The measurement is taken over the contact points. As the thimble opens or closes, the contact points get opened or closed. The inside micrometer consists of a sleeve, thimble, anvils, a spacing collar and extension rods. It is also equipped with a handle to measure deep bores. The least count of the instrument is also 0.01 mm. The inside micrometer is equipped with a 12 mm spacing collar and 4 extension rods for measuring holes of ranges 50-75mm, 75-100 mm, 100-125 mm and 125-150 mm. The sleeve is marked with the main scale and the thimble with the thimble scale. The barrel has a limited adjustment of 13 mm. When the inside micrometer is closed (when zero of thimble coincides with the zero of the barrel), it is capable of reading the minimum dimension of 25 mm. In addition to this, it is possible to read up to 38 mm with the thimble opening to the extreme right. In order to read further higher ranges, a standard spacing collar of 12 mm width is to be added. This facilitates the micrometer to read a maximum range of 50 mm. (Fig 2)

Similarly, each extension rod has to be used without the collar for measuring a minimum range up to 13 mm variation and with the collar for a maximum range of measurements. A clamping screw is also provided to clamp the extension rod firmly.

Fig 1

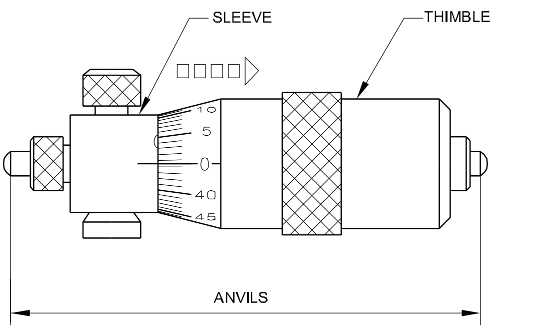
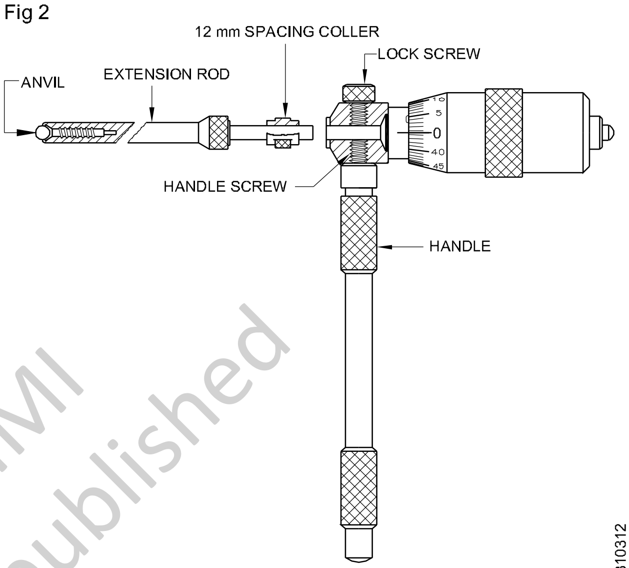


Fig 2

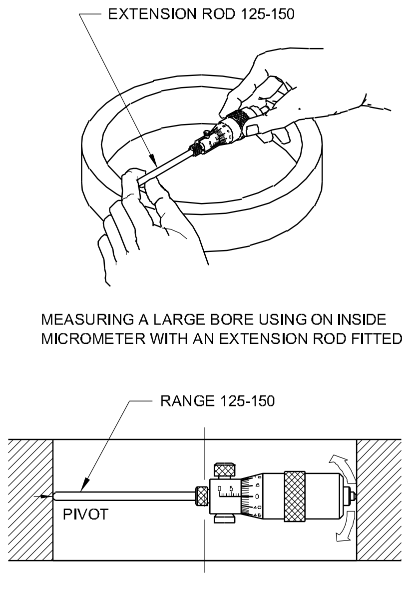


INSIDE MICROMETER

Determining the size of a bore or hole

Reading of inside micrometer Fig 3 shows an inside micrometer with a spacing collar and extension rod of 125-150 mm range. The size of the bore is 125 mm + 12 mm + barrel reading + thimble reading which is equal to 125 + 12 + 1.5 + 0.00 = 138.50 mm.

Fig 3



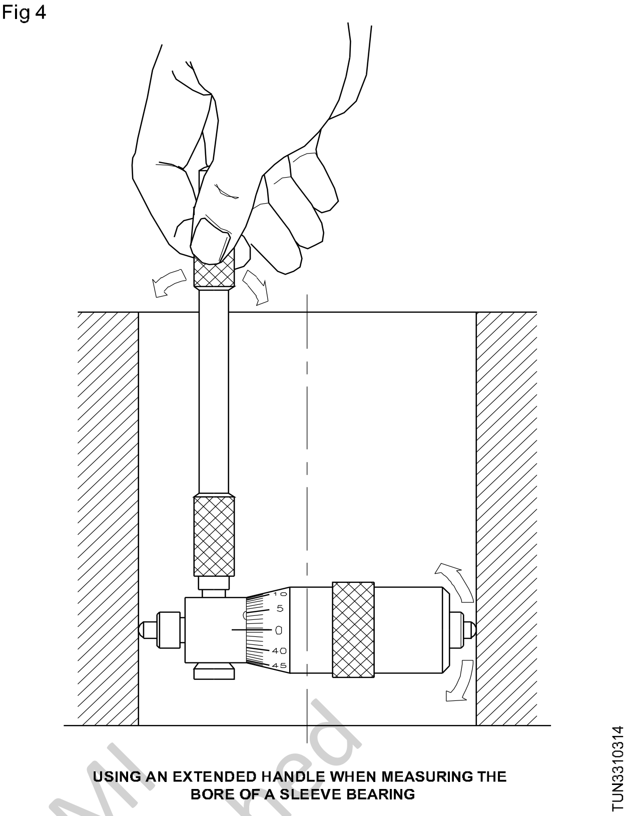
Determining the distance between internal parallel surfaces

While checking parallelism between two surfaces of a deep bore, a handle must be used along with the inside micrometer. The figure shows the inside micrometer with a handle. In order to ascertain the parallelism, a minimum of two readings has to be taken, i.e. one at the top surface of the deep bore and the other at the bottom surface of the bore. If there is no difference in the two readings, we may take it for granted that the surfaces are perfectly parallel. Any variation in the reading shows the bore has an error between the two surfaces. (Fig 4)

Uses:

This instrument is generally used to measure accurately the bored components, specifically it is used for checking cylinder bores. It is also used for measuring internal shoulders, checking ring gauges.

Fig 4



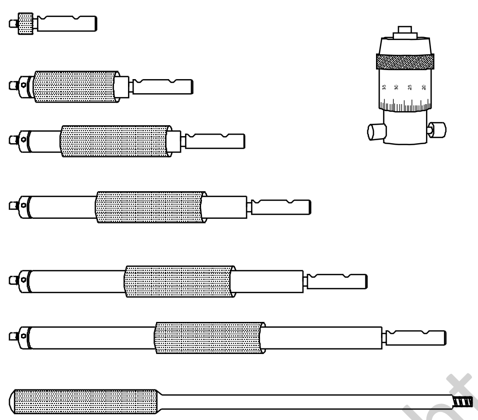
USING AN EXTENDED HANDLE WHEN MEASURING THE BORE OF A SLEEVE BEARING

Inside micrometer - Inch

Objectives: At the end of this lesson you shall be able to
- **determine the reading of inside micrometer inch**
- **calculate the least count of inside micrometer inch**
- **state the construction of inside micrometer inch.**

The inside micrometer inch is similar to inside micrometer metric. This consists of a sleeve, thimble, avoids a stiching coller and extension rods. It is also equiped with a handle to measure deep bores. The lease count of the instrument is 0.001".

Fig 1



TUN3310411

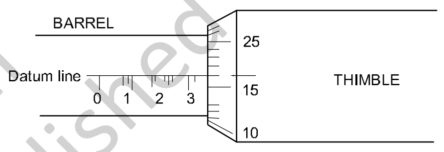
The inside micrometer inch is equiped with a 1/2" spacing coller an 4 extension rods for measuring holes of ranges 2" - 3", 4" - 5" and 5" - 6". The sleeve is marked with the main scale and the thimble with the thimble scale. The barrel has a limited adjustment 1/2". When the inside micrometer is closed (zero of thimble coincides with the zero of the barrel) it is capable of reading the minimum dimension of 1".

In addition to this, it is possible to read upto 1 1/2" with the thimble opening to the extreme right. In order to rad further higher ranges, a standard spacing coller of 1/2" width is to be added. This facilitates the micrometer to read a maximum range of 2".

Reading of inside micrometer

Graduation of inside micrometer inch.

Fig 2



TUN3310412

Value of one main scale division MSD = 0.100"

Vale of one sub division SD = 0.025"

Value of one thimble division TD = 0.001"

Reading

Main scale reading	=	3 x 0.00	= 0.300"
Sub division reading	=	1 x 0.025	= 0.025"
Thumble division coincide	=	15 x 0.001	= 0.015"
Total	=		0.340"

Care for holding split bearing, fixture and its uses

Objectives: At the end of this lesson you shall be able to
- **state split bearing**
- **list the type of split bearing**
- **explain how to set the split bearing**
- **state care for holding split bearing.**

Split bearing

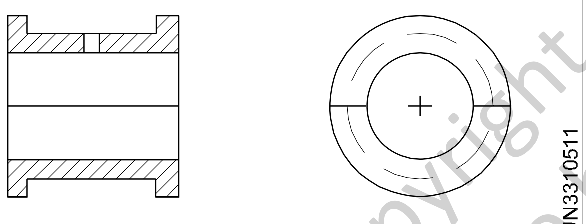
Split bearing are principally used where the replacement of spherical bearings would require costly additional work, involving the removal of gears or couplings, the dismounting of drives or the dismantling of shaft power trains. The use of split reduces the down time of machinery and plant.

Types of split bearing:

- Flanged pattern Fig 1

The flanged pattern are previously made out of brasses, in which the journal of a shaft runs direct. They may also be lined with white metal or babbit metal when they are used in car engines.

Fig 1

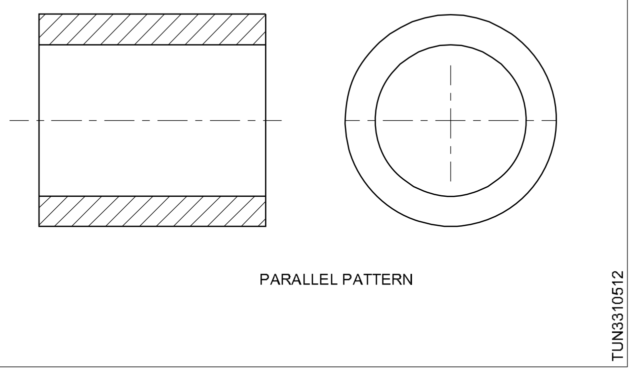


FLANGED PATTERN

- Parallel type Fig 2

The parallel type consist of steel shells lined with white metal or harder alloy for heavy duty. This type may be called shall bearings, thin - wall bearings or simply liners. They are circular outside to fit in accurately machined housings or the big ends of connecting rods.

Fig 2



PARALLEL PATTERN

Holding of split bearing

Methods of setting up to machine split bearings are broadly the same for all sizes for it is essential to maintain accuracy.

To make signal split bearings, can use drawn rectangular brass holding two piecces together in the four-jaw chuck with the joint line on the spindle axis. First each piece should be faced in the chuck. Then two face must be field smooth.

As shown in Fig 3 & 4 split bearing can be set up from their outside diameters to finish the bores.

Fig 3

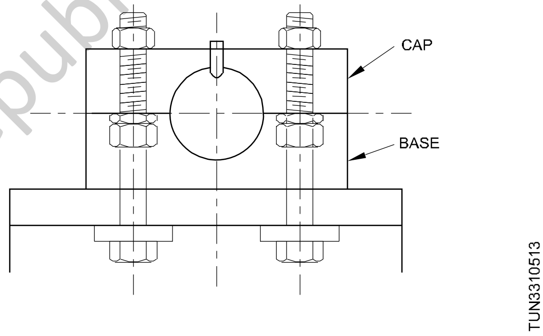
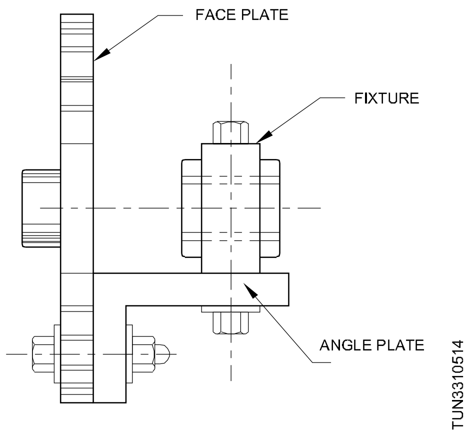


Fig 4



Care for holding the split bearing

The joint line of the halves must be on the diameter line.

The outside must be circular and the bore concentric with it.

While designing the turning fixtures most care should be taken to avoid projections for the operator's safety.

The accuracy of the machine tool must be protected by placing necessary balance weights in the fixture.

The over hang of the fixture should be minimum.

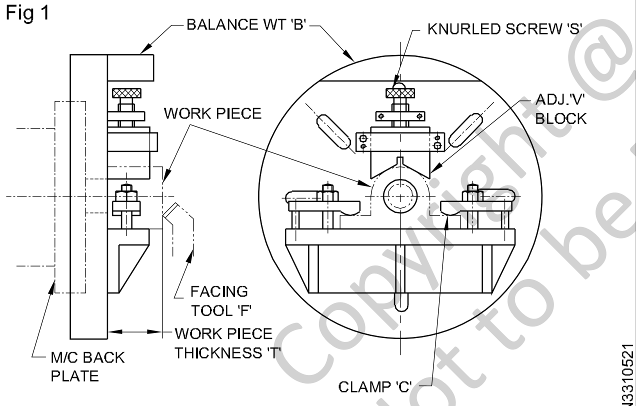
Lathe fixtures (Turning fixtures)

The standard work holding devices or fixures for lathe are:

- Three and four jaw chucks
- Collets
- Face plate
- Mandrels
- Milling vice

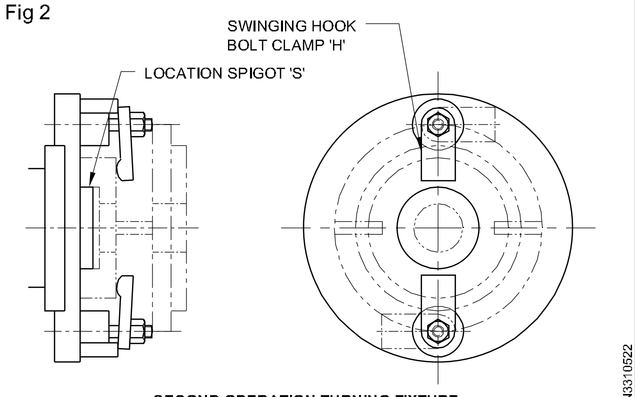
If the job can be held easily and quickly in the above mentioned standard devices, then there is no need for special work holding devices. However many jobs particuly casting and forging, because of their shapes, cannot be conveniently held by any of the standard devices. It then becomes necessary to build a special work holding devices for the jon. Such a device is called lathe fixture.

Fig 1



BORING AND FACING FIXTURE

Fig 2



SECOND OPERATION TURNING FIXTURE

A lathe fixture consists of a base, location and clamping devices. A lathe fixture can be fixed to the either by holding in the chuck jaws or fixing to a face plate.

Basic Deisgn Principles for Turning or Lathe Fixtures:

1 To avoid vibration while revolving, the fixture should be accurately balanced.

2 There should be no projections of the fixture which may causes injury to the operator.

3 The fixture should be rigid and overhanging should be kept minimum possible so that there is no bending action.

4 Clamps used to fix the fixture to the lathe should be designed properly so that they don't get loosed by centrifugal force.

5 The fixture should be as light weight as possible since it is rotating.

6 The fixture must be small enough to that it can be mounted and revolved without hitting the bed of the lathe.

Uses:

A fixture in mainly used to hold a non cylindrical job so that turning operation can be carried out.

It is mainly used for mass production purposes.

Calculation involving fractional thread (odd and even)

Objectives: At the end of this lesson you shall be able to
- **calculate change wheels for cutting vulgar fractional pitch threads (BritishSystem)**
- **calculate change wheels for cutting decimal fractional pitch threads (BritishSystem)**
- **calculate change wheels for fractional pitch threads by continued fraction method.**

It is necessary to calculate the ratio of change gears to cut fractional leads for worms, hobs etc. on a centre lathe at times.

To obtain a formula; suppose it is required to cut a lead of 1/4" on a lathe which has a lead of 1/2". If one to one ratio were used between the driver and the driven gears, the carriage would move 1/2" per revolution of the lathe spindle. Therefore, to cut a lead of 1/4" the ratio of the driver and driven gears must be as

$$\frac{1}{4} : \frac{1}{2}$$

$$\text{That is } \frac{1/4}{1/2} \text{ or } \frac{1}{2} = \frac{\text{Driver}}{\text{Driven}}$$

Expressed as a formula:-

$$\frac{DR}{DN} = \text{ratio of change gears} = \frac{\text{lead of screw to be cut}}{\text{lead of lead screw}}$$

or alternatively:-

$$\frac{\text{lead of screw to be cut}}{1} \times \frac{1}{\text{lead of lead screw}} = \frac{\text{Driver}}{\text{Driven}}$$

lead of screw to be cut x No. of threads/inch of lead screw

$$= \frac{\text{Driver}}{\text{Driven}}$$

Example

Calculate the change gears necessary to cut a thread of 7/16" lead on a lathe with a lead screw of 4 threads per inch.

lead of screw to be cut xNo. of threads/inch of lead screw

$$= \frac{\text{Driver}}{\text{Driven}}$$

$$= \frac{7}{16} \times 4 = \frac{28}{16} = \frac{7}{4}$$

$$= \frac{7}{4} \times \frac{10}{10} = \frac{70}{40} = \frac{\text{Driver}}{\text{Driven}}$$

If the lead to be cut is a whole number and a vulgar fraction, change it to an improper fraction and apply the above formula.

Example

Calculate the change gears required to cut an oil groove having 8 turns in 11 inches on a lathe with a lead screw of 4 threads per inch.

Pitch of the groove x No. of threads/inch of lead screw

$$= \frac{\text{Driver}}{\text{Driven}}$$

$$\text{Pitch groove} = \frac{\text{travel for given number of turns}}{\text{number of turns}}$$

$$= \frac{11}{8} \text{ inches}$$

$$\text{Gear ratio} = \frac{11}{8}$$

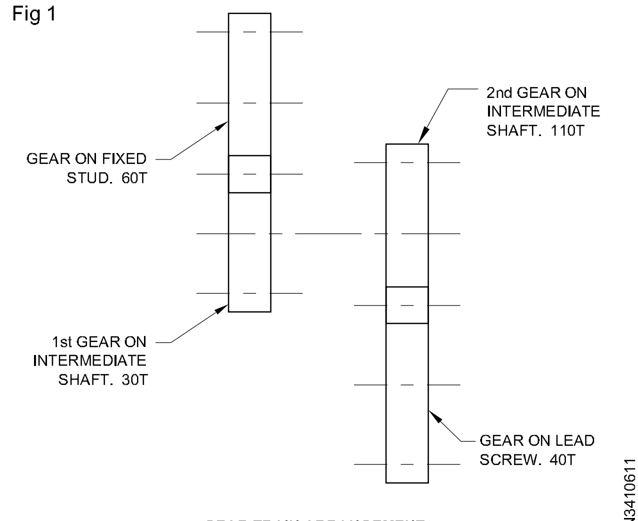
$$\frac{1}{4}$$

$$= \frac{11}{8} \times 4 = \frac{44}{8} = \frac{4 \times 11}{2 \times 4} = \frac{4}{2} = \frac{11}{4}$$

$$\text{First fraction} = \frac{4}{2} \times \frac{15}{15} = \frac{60}{30}$$

$$\text{Thus } \frac{DR}{DN} = \frac{60}{30} \times \frac{110}{40} \text{ (Fig.1)}$$

Fig 1



GEAR TRAIN ARRANGEMENT

Example

Calculate the change gears to cut a worm of 0.35 inches lead on a lathe with a lead screw having 4 threads per inch.

Lead to be cut x no. of threads/inch of lead screw

$$= \frac{DR}{DN} = 0.35 \times 4$$

$$= \frac{35}{100} \times \frac{4}{1} = \frac{7}{5} = \frac{10}{10} = \frac{70}{50} = \frac{Driver}{Driven}$$

When the lead occurs as a decimal, it may be necessary to use the method of continued fractions to obtain a suitable approximation of the change gear ratio, for which the change gears may be selected from the available set of gears.

Example

Calculate the change gears required to cut a worm of 0.55 inches lead on a lathe, with a lead screw of 6 threads per inch.

No. of threads/inch of lead screw $= \frac{Driver}{Driven}$

$$= 0.55 \times 6$$

$$= \frac{55}{100} \times \frac{6}{1}$$

1st fraction $= \frac{55}{100}$

2nd fraction $= \frac{6}{1} = \frac{20}{20} = \frac{120}{20}$

$$= \frac{driver}{driven} = \frac{55}{100} \times \frac{120}{20}$$

Example

Calculate the change gears required to cut a worm of 0.95 inches lead on a lathe with a lead screw of 6 threads per inch.

lead to be cut xNo. of threads/inch of lead screw

$$= \frac{Driver}{Driven}$$

$$= 0.95 \times 6$$

$$= \frac{95}{100} \times \frac{(6 \times 20)}{(1 \times 20)} = \frac{95}{100} \times \frac{120}{20}$$

$$= \frac{driver}{driven} = \frac{95}{100} \times \frac{120}{20}$$

Example

Calculate the change gears to cut 2BA threads (0.81mm pitch) on a lathe which has a lead screw of 1/4 inch - pitch by the continued fraction method.

Ratio : $\frac{driver}{driven} = \frac{0.81}{1/4 \times 25.4} = \frac{0.81}{6.35}$

$$\frac{driver}{driven} = \frac{81}{635} = \frac{1 \times 81}{5 \times 127}$$

This could be cut exactly if the 1/5 ratio were combined with a 81T driver and a 127 T driven change gears.

If special gears are not available we have to obtain the nearest fraction by the continued fraction method. For this nearest fraction gears may be selected from the available set of gears.

Determining the convergents by the continued fraction method.

81) 635 (7
567
___
68) 81 (1
68
___
13) 68 (5
65
___
3) 13 (4
12
___
1) 3 (3

		7	1	5	4	3
1	0	1	1	6	25	81
0	1	7	8	47	196	635
		7	1	5	4	3

The convergents are $\frac{1}{7};\frac{1}{8};\frac{6}{47};\frac{25}{196};\frac{81}{635}$

The 4th convergent $\frac{25}{196}$ may be written $\frac{5}{14}\times\frac{5}{14}$

$$\frac{driver}{driven}=\frac{25}{70}\times\frac{25}{70}\,\frac{25}{196}$$

and this could be obtained with duplicate 25 T and 70 T gears, a circumstance not unlikely, provided two similar lathes are available.

The actual pitch obtained from this driver and driven gears is:

$$\frac{25}{196}x6.35=\frac{158.75}{196}=0.80995mm$$

an error of 0.00005mm, which is equivalent to a total pitch error of about 0.0016 mm (0.00006 in) over a 1 in. length of the thread. This is well within the permissible limits of accuracy of an ordinary commercial lead screw.

Simple and compound gear trains

Objectives : At the end of this lesson you shall be able to
- **state what is a change gear train**
- **identify and name the different types of change gear trains**
- **distinguish between a simple gear train and a compound gear train.**

Change gear train

Change gear train is a train of gears serving the purpose of connecting the fixed stud gear to the quick change gearbox. The lathe is generally supplied with a set of gears which can be utilized to have a different ratio of motion between the spindle and the lead screw during thread cutting. The gears which are utilized for this purpose comprise the change gear train.

The change gear train consists of driver and driven gears and idler gears.

Simple gear train

A simple gear train is a change gear train having only one driver and one driven wheel. Between the driver and the driven wheel, there may be an idler gear which does not affect the gear ratio. Its purpose is just to link the driver and the driven gears, as well as to get the desired direction to the driven wheel.

Fig 1 shows an arrangement of a simple gear train.

Fig 1

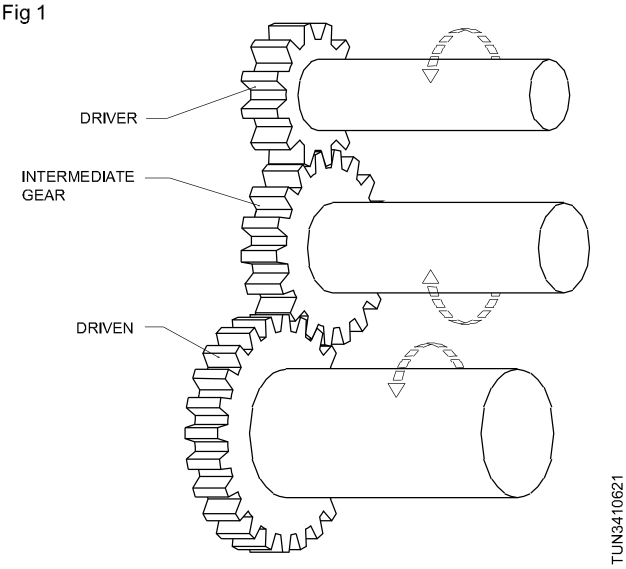
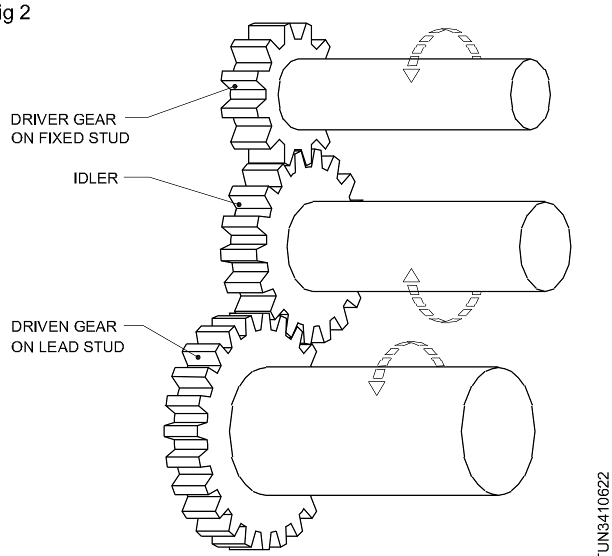


Fig 2 shows mountings of the driver and driven gears in a lathe.

The driver gear and the driven gear are changed according to the pitch of the thread to be cut on the job.

Fig 2

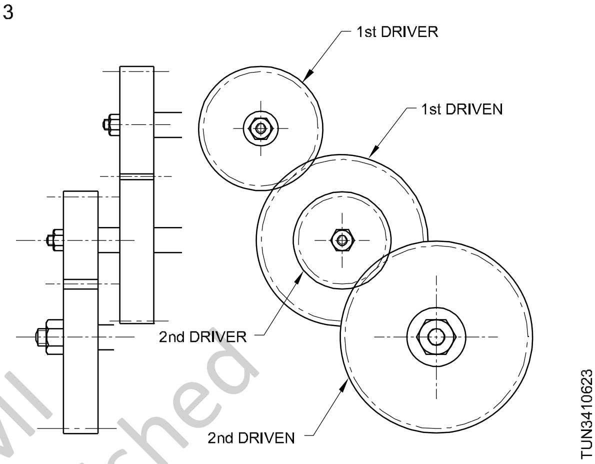


Compound gear train

Sometimes, for the required ratio of motion between the spindle and the lead screw, it is not possible to obtain one driver and one driven wheel. The ratio is split up and then the change gears are obtained from the available set of gears which will result in having more than one driver and one driven wheel. Such a change gear train is called a compound gear train.

Fig 3 shows the arrangement of a compound gear train

Fig 3



Gear calculation for cutting metric thread on British lathe and vice versa

Objectives : At the end of this lesson you shall be able to
- **state the formula of the gear ratio for cutting metric thread on a British lathe**
- **state the formula of the gear ratio for cutting British thread on a metric lathe**
- **solve the problems involving cutting metric thread on British lathe and vice versa.**

Gear ratio for cutting metric thread on British lathe

The formula of the gear ratio for cutting metric thread on a metric lathe is

$$\frac{\text{Driver}}{\text{Driven}} = \frac{\text{Lead to be cut on the job}}{\text{Lead of lead screw}}$$

Now, for cutting metric thread on a British lathe, the lead of the work to be cut in mm is converted to inches by multiplying with the constant 5/127.

Because 25.4 mm = l"

1 mm	=	1/25.4"
	=	10/254
	=	5/127"

Therefore,

Gear ratio

$$\frac{DR}{DN} = \frac{\text{Lead to be cut in mm on job} \times 1 \times 5}{\text{Lead of L.S.} \times 127}$$

$$\frac{DR}{DN} = \frac{\text{Lead to be cut in mm} \times \text{T.P.I on L.S} \times 5}{127}$$

A translating gear of 127 teeth is provided for cutting metric thread on a British lathe. This gear wheel is used as the driven wheel. The worked out example illustrates this statement.

Gear ratio for cutting British thread on metric lathe

The general formula for cutting British thread on a British lathe is

$$\frac{DR}{DN} = \frac{\text{Lead to be cut on job}}{\text{Lead of lead screw}}$$

Now for cutting British thread on a metric lathe the lead of the screw in mm is converted into inches by multiplying with a constant of 5/127.

$$\frac{DR}{DN} = \frac{\text{Lead to be cut in inch on job}}{\text{Lead of lead screw in mm} \times \frac{5}{127}}$$

$$\frac{DR}{DN} = \frac{\text{Lead to be cut in inch on job} \times 1 \times 127}{\text{Lead of lead screw in mm} \times 5}$$

$$\frac{DR}{DN} = \frac{1}{\text{T.P.I. to be cut}} \times \frac{1}{\text{Lead of lead screw}} \times \frac{127}{5}$$

As a practice, it is advisable to have a larger wheel as a driven gear as far as possible. But in this case the 127 teeth wheel has to be used as a DRIVER only.

Gear ratio for cutting metric thread on British lathe using 63 teeth as driver wheel.

Instead of taking the constant $= \frac{5}{127}$

$\frac{63}{1600}$ is taken because 1 metre = 39.37".

1 metre = 39.375" (approx.)

1000 mm = 39.375" = $39\frac{3}{8}$

1 mm $= \frac{315}{100 \times 8}$

$= \frac{63}{1600}$

Gear ratio

$$\frac{DR}{DN} = \frac{\text{Lead to be cut in mm} \times \text{TPI on LS} \times 63}{1600}$$

Gear ratio for cutting British thread on metric lathe using the 63 teeth wheel as the driven wheel:

$$\frac{DR}{DN} = \frac{1}{\text{T.P.I. to be cut}} \times \frac{1}{\text{Lead of lead screw in mm}} \times \frac{1600}{63}$$

Lathe constant

Lathe constant is the number of threads per inch that can be cut when the change gear ratio is 1 and the ratio between the main spindle gear and the fixed stud gear is also 1.

On some machines the ratio of the spindle gear to the fixed stud gear is more than 1 in which case the lathe constant is equal to:

$$\frac{\text{spindle gear} \times \text{T.P.I on lead screw}}{\text{fixed stud gear}}$$

When lathe constant is given

(gear ratio for cutting thread) $\frac{DR}{DN} = \frac{\text{Lathe constant}}{\text{T.P.I. to be cut}}$

Find the gears required to cut 4.5 mm pitch in a lathe having a lead screw of 6 T.P.I. Gears available from 20 to 120 teeth by 5 teeth range with a conversion gear of 127 teeth.

DATA

Lead of work = 4.5 mm

T.P.I of L/s = 6 T.P.I

Lead of L/s $= \frac{1}{\text{T.P.I}}$

Lead of L/s $= \frac{1}{6}$

$$\frac{DR}{DN} = \frac{5}{127} \times \frac{\text{Lead of work}}{\text{Lead of lead screw}}$$

$$= \frac{5}{127} \times \frac{4.5}{1/6}$$

$$= \frac{5 \times 6 \times 4.5}{127 \times 1}$$

Now it is not possible to have a change gear train with a simple gear train. So a compound gear train is used,

i.e. $\frac{30}{127} \times \frac{4.5}{1}$

$\frac{30}{127} \times \frac{45}{10}$

$\frac{45 \times (30 \times 2)}{127 \times (10 \times 2)} = \frac{45}{127} \times \frac{60}{20}$

45 T & 60 T are drivers.

127 T & 20 T are driven.

Problems involving cutting metric threads on British lathe and vice versa

Find the gears required to cut a 3 mm pitch in a lathe having a lead screw of 6 T.P.I. Gears available from 20 to 120 teeth by 5 teeth with a special gear of 127 teeth.

DATA

Lead of work = 3 mm

T.P.I on L/s = 6 T.P.I

Lead of L/s $= \frac{1}{6}$

Gear ratio $= \frac{DR}{DN} = \frac{5 \times \text{Lead of work}}{127 \times \text{Lead of lead screw}}$

$$= \frac{5}{127} \times \frac{3}{1/6}$$

$$= \frac{5}{127} \times \frac{3 \times 6}{1}$$

$$= \frac{90}{127}$$

90 teeth gear is driver.

127 teeth gear is driven.

Problems involving cutting British threads on metric lathe

Find the gears required to cut 6 T.P.I on job in a lathe having a lead screw of 6 mm pitch.

Gears available from-20 T to 120 by 5 teeth range with a special gear of 127 teeth.

DATA

Lead of work = 1/6"

Lead of L/S = 6 mm

Gear ratio $= \frac{DR}{DN} = \frac{127}{5} \times \frac{\text{Lead of work}}{\text{Lead of L/S.}}$

$$= \frac{127}{5} \times \frac{1/6}{6}$$

$$= \frac{127}{5} \times \frac{1}{6 \times 6}$$

$$= \frac{127}{30} \times \frac{1}{6}$$

$$= \frac{127}{30} \times \frac{(1 \times 20)}{(6 \times 20)}$$

$$= \frac{127}{30} \times \frac{20}{120}$$

127 T & 20 T are driver gears.

30 T & 120 T are driven gears.

At the end of the first cut, stop the lathe. Mark the position of the chuck and the lead screw with two fixed reference marks on the headstock and the norton gearbox with a chalk piece . Then disengage the half nut and bring back the tool to the starting point and give a depth of cut. Start the lathe and observe the instant at which both the marks of the lead screw and the chuck with their respective fixed reference lines marked on the norton gearbox and headstock coincide.

Engage the half nut when both the marks are coinciding simultaneously. The disadvantage of this method is it requires skill to watch both the marks simultaneously; it is time consuming.

Predetermined travel of carriage

Predetermined travel of the carriage means the shortest distance the carriage has to move to engage the half nut so that the thread on the lead screw and the thread on the job are in unison.

The following example shows the calculation of predetermined travel of the carriage.

To cut 8 tpi on a job in a lathe having 6 tpi lead screw

If the job makes 4 revolutions, the lead screw makes 3 revolutions (8/6 = 4/3). For every 3 revolutions of the lead screw the thread on the lead screw will be in unison with the thread on the job. For 3 revolutions of the lead screw the carriage travels to a distance equal to 3 x pitch = 3 x 1/6" = 1/2 inch.

This 1/2" is the shortest distance the carriage can travel to engage the half nut. This is the predetermined travel. Depending upon the threading length, an exact multiple of the predetermined travel of the carriage is chosen for marking on the bed. The carriage is allowed to travel only between these two marks for engaging and disengaging the half nut.

By thread chasing dial

The chasing dial indicates the relationship between the ratio of the number of turns of the work and the lead screw with respect to the position of the cutting tool and thread groove.

Determine the predetermined travel of the carriage, then interpret this in terms of graduation on the dial. Allow this predetermined travel of the carriage movement to occur between the position of the graduations of the dial at which the half nut can be engaged.

A detailed description of the chasing dial is dealt with, in the next lesson.

Multiple thread function, use

Objectives : At the end of this lesson you shall be able to
- **understand about a multistart thread and its application**
- **understand about pitch and lead in a multistart thread.**

Multiple thread function

Multi-start threads are generally used, for a greater nut advancement for each rotation. The advancement is linked to number of pitches and the number of threads per inch. Hence it is possible to tighten and close the fasteners with lesser nut movement.

Multistart thread user

1 Such threads are used on pen cap, flypress, thermoflask cap, hand presser, telescopes, and camera focusing devices.

2 Wherever higher mechanical advantage is required to be obtained then the output should be more than the input (E.g. Screw jacks used to lift vehicles).

3 Wherever leak proof applicatons is needed Eg: Liquid container cap).

4 Tight fitting applications.

Difference between Pitch & Lead

Pitch	Lead
1 **Pitch** is defined as the distance between a point of thread to the another coreesponding point on the adjacent thread. 2 In single start pitch is equal to lead divided by no of start $P = \frac{1}{No}$ of start	Lead is defined as the advancement of mating part (next to bolt) in one complete (360°) rotation. In the multistart thread lead is the product of the pitch and number of starts of thread. I = P x No of start

Difference between pitch and lead

Pitch is the distance from a point on one thread to the corresponding point on the next thread.

Lead is the distance that a screw thread advances axially in one rotation in a single start thread.

Formulae

$$\text{Start} = \frac{\text{lead}}{\text{pitch}}$$

$$\text{Pitch} = \frac{\text{lead}}{\text{No of start}}\left(\frac{\text{Distance moved}}{\text{No of rotation given}}\right)$$

Lead = Pitch x No of start

Gear ratio

1 Metric thread on metric lead screw.

$$\text{Gear Ratio } \frac{DR}{DN} = \frac{\text{Pitch to be cut}}{\text{Pitch of lead screw}}$$

2 Inch thread on Inch lead screw (T.P.I)

$$\frac{DR}{DN} = \frac{\text{Pitch to be cut}}{\text{Pitch of lead screw}}$$

3 Metric thread on Inch (or) British lead screw formula.

$$\frac{DR}{DN} = \frac{5PN}{127}$$

P = pitch to be cut in m

N = No of T.P.I on lead screw.

127 = Transmitting gear (or) special gear.

4 Inch (or) British thread on metric lead screw formula

$$\frac{DR}{DN} = \frac{127}{5PN}$$

5 British lead screw on metric thread change calculation formula. (Translating gear 63 teeth)

$$\frac{DR}{DN} = \frac{63PN}{1600}$$

P = pitch to be cut mm.

N = No of T.P.I on lead screw

6 Metric lead screw on british. Thread change wheel calculation formula (Translating gear 63 teeth) .

$$\frac{DR}{DN} = \frac{1600}{63PN}$$

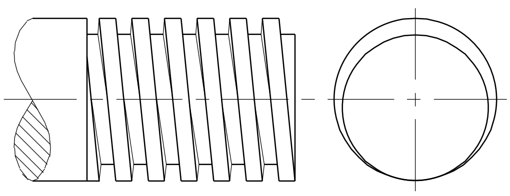
Multi-start thread and methods

Objectives: At the end of this lesson you shall be able to
- **state the purpose of multi-start threads**
- **mention the various methods of identifying multi-start threads**
- **state the methods of cutting multi-start threads**
- **understand multi-start thread elements**
- **calculate gear ratio.**

Metric multi-start threads are used where quick transmission is required. Such threads are used on pen cap, fly press, thermos flask cap, hand presses, telescopes and camera focusing devices.

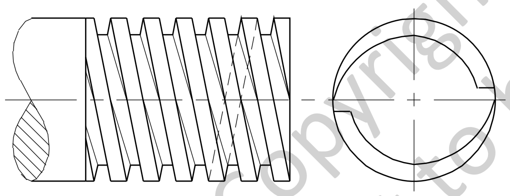
The lead on a single start thread is equal to the pitch. (Fig 1)

Fig 1



The lead on a double start thread is twice the pitch. (Fig 2)

Fig 2

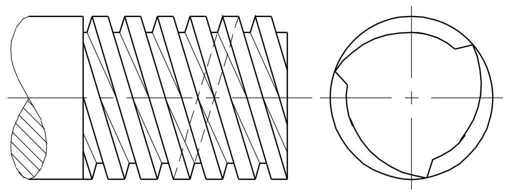


The lead on a triple start thread is three times the pitch. (Fig 3)

A triple start thread will advance 3 times the distance of a single start thread for a single turn.

The threads are specified by stating the diameter, pitch and number of starts.

Fig 3



Methods of cutting multi-start thread

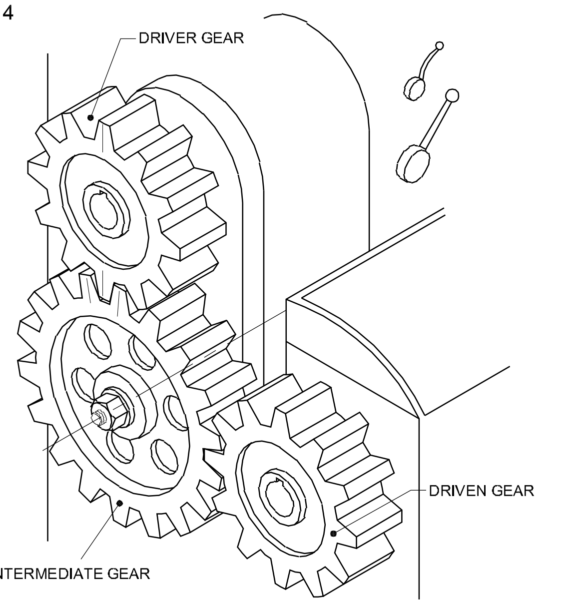
- Dividing the 1st driver of the change gear train
- Using slotted face plate
- Moving the top slide to a new position (compound slide)
- Using thread chasing dial

1 Dividing the first driver method

As regards the gear train it becomes necessary to arrange the layout so that the first driver is a multiple of the number of starts required. Thus for a double start thread, the gear teeth must be divisible by two.

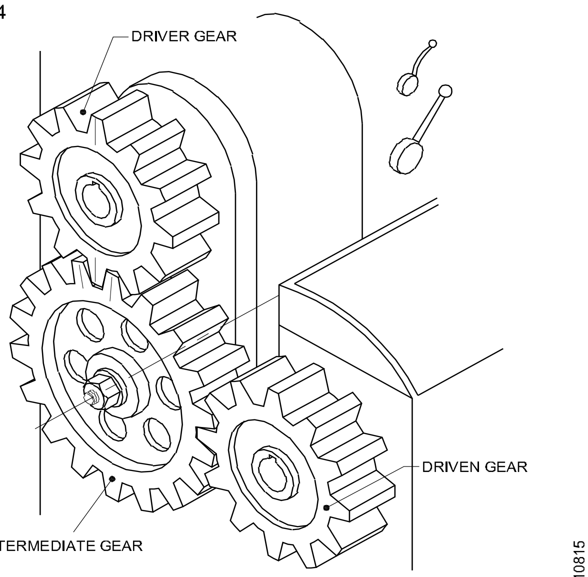
After finishing the first start, the lathe is stopped. One tooth of the 1st driver and the space of the first driven gear in which it is seating are marked. By counting the number of teeth from the marked tooth of the 1st driver, make another mark on the tooth which is exactly 180° away. Loosen the swing plate and disengage the idler gear from the 1st driver. Rotate the spindle by hand to bring the second mark of the first driver to mesh in the previously marked space of the 1st driven gear. The lathe is now ready for cutting the 2nd start. This procedure is applicable to cut threads of more than two starts also. Figs 4 and 5 illustrate marking on change gears.

Fig 4



GEAR TEETH MARKED FOR INDEXING

Fig 4



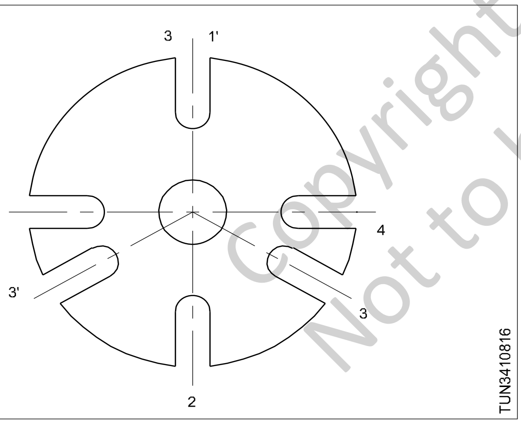
GEAR TEETH MARKED FOR INDEXING

2 Method using slotted face-plate (Fig 6)

A slotted face-plate illustrated is used to cut threads of 2 starts, 3 starts, 4 starts etc.

Slots are provided on the face-plate at convenient distances. Two opposite slots to cut double start thread, 3 slots at 120 degree apart to cut 3 start thread and 4 slots at 90 degree apart to cut 4 start thread and so on and so forth.

Fig 6



Using indexing drive–plates

A graduated indexing drive-plate is used (Fig 7) to cut multi-start threads on the job. Here also the job is held between centres with the help of a straight tailed dog carrier.

3 Method by moving the top slide (compound rest) (Fig 8)

The top slide may be used (Fig 8) for adjusting the tool to have the correct spacing while cutting multi-start threads. After one start of the thread has been cut the top slide is moved a distance equal to the pitch of the thread for the 2nd start. When this method is followed the top slide must be parallel to the axis of the workpiece.

The backlash must be eliminated in the top slide. After rotating the hand wheel through the number of graduations which is equal to the pitch, it is advisable to set the graduated collar again to zero.

Fig 7

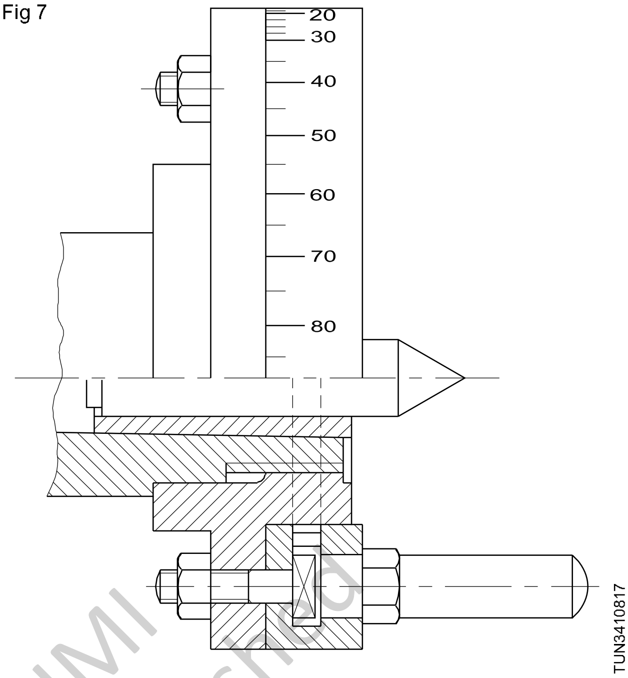
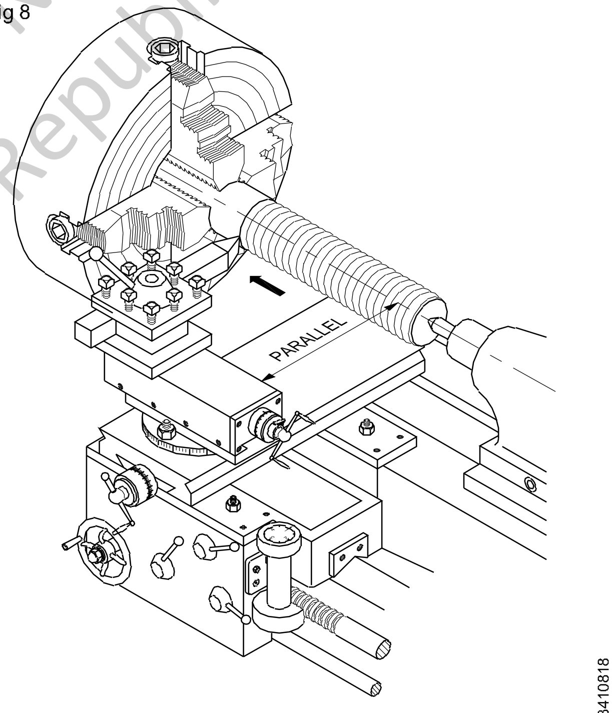


Fig 8



CUTTING A MULTIPLE THREAD BY COMPOUND RESET METHOD

4 The thread chasing dial method

The construction of the thread chasing dial enables to cut 2-start, 4-start, 8-start and 16-start threads. This purely depends upon the graduations marked on the dial and the number of teeth of the worm wheel.

Change wheel calculation for multi–start threads

In multi-start threads, the lead of the thread is equal to the pitch of the thread multiplied by the number of starts. For example, in a double start thread, the lead of the screw = 2 x pitch. In a triple start thread, the lead of the screw = 3 x pitch and in a quadruple start, the screw lead = 4 x pitch.

Example

Calculate the change gears to cut a 3-start thread having a pitch of 1.5 mm; the lead screw has a pitch of 6 mm.

Lead of thread = pitch x number of starts

= 1.5 x 3 = 4.5 mm.

$$\text{Gear Ratio} = \frac{\text{driver}}{\text{driven}} = \frac{\text{lead of the thread}}{\text{lead of the lead screw}}$$

$$\frac{\text{driver}}{\text{driven}} = \frac{4.5 \times 10}{6 \times 10} = \frac{45}{60}$$

Example

Calculate the change gears to cut a 4-start thread having 12 TPI. The lead screw has 4 TPI

$$\text{Lead of the work} = \frac{1}{12} \text{x4} = \frac{45}{60}$$

$$= \frac{4/12}{1/4} = \frac{4 \times 4}{12 \times 1} = \frac{16}{12}$$

$$\frac{16}{12} = \frac{4 \times 10}{3 \times 10} = \frac{40}{30} = \frac{Dr}{Dn}$$

The number of threads for which gears are to be determined } = 3.

40 is the driver and 30 is the driven.

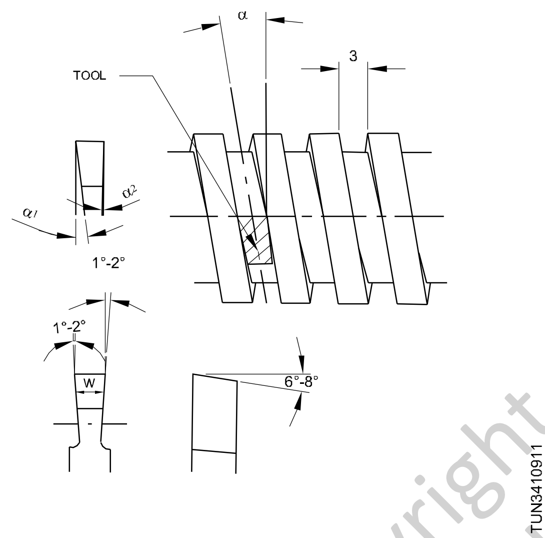
Calculation involving shape of tool (Square thread tool)

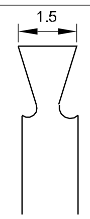
Objectives: At the end of this lesson you shall be able to
- **understand the parameters for shaping the tool for square thread**
- **identify square thread and its elements**
- **identify different type of trapezoidal threads.**

Determine width and angles required for grinding the external square threading tool

The side clearance of the square threading tool is of prime importance to prevent the tool from interfering or rubbing against the vertical flank the thread.

Fig 1





Calculation of tool shape

The width of the nose of the square threading tool. Should be equal to half of the pitch of the square thread.

w = 0.5 x p

= 0.5 x 3

Width of tool = 1.5 mm

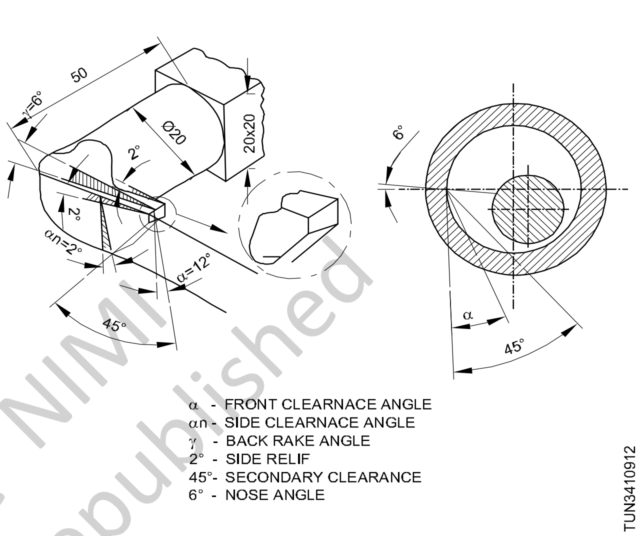
Front clearance angle = 6° - 8°

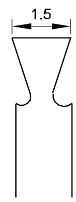
Side relief angle

1°° to 2° = a_1 = a + (1° + 2°)

Calculation involving shape of tool [Internal square thread tools]

Fig 2





1 Calculate the change gears to cut 2-start square thread having a pitch of 3mm the lead screw has a pitch 5mm.

lead = pitch x No of start

= 3 x 2

= 6 mm

$$\frac{DR}{DN} = \frac{\text{lead of the thread}}{\text{lead of the lead screw}}$$

$$= \frac{6 \times 10}{5 \times 10} = \frac{60}{50}\text{ Ans}$$

Core dia

Core diameter formula

Core diameters = Major dia - 2 x single depth

1 Given data

Major dia = 30mm

Pitch = 4mm

To find

Core dia

single depth = 0.6134 x pitch

= 0.6134 x 4

= 2.4536mm

Core diameter = Major dia - 2x single depth

= 30 - 2 2.4536

= 25.09 mm.

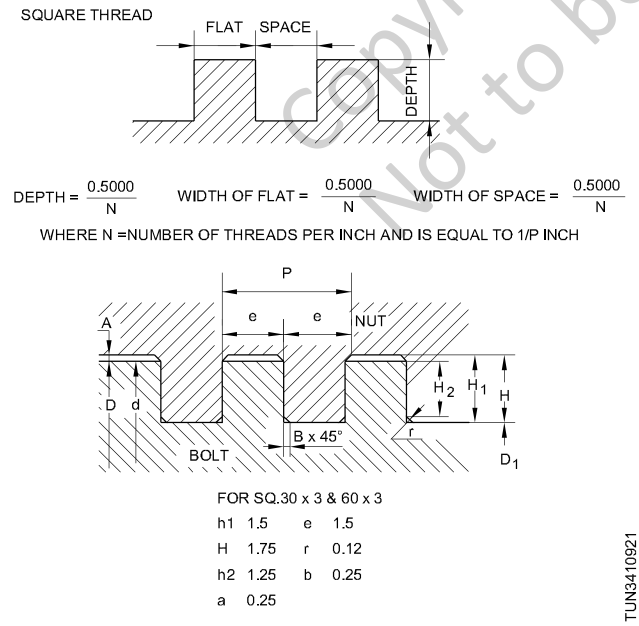
Square and trapezoidal threads

Square and trapezoidal threads have more cross-sectional area than ‘V’ threads. They are more suitable to transmit motion or power than ‘V’ threads. They are not used for fastening purposes.

Square thread

In this thread the flanks are perpendicular to the axis of the thread. The relationship between the pitch and the other elements is shown in Fig 1.

Fig 1



Square threads are used for transmitting motion or power. *Eg.* screw jack, vice handles, cross-slide and compound slide, activating screwed shafts.

Designation

A square thread of nominal dia. 60mm and pitch 9mm shall be designated as Sq. 60 x 9 IS: 4694-1968. The dimensions a, b, e, p, H_1, h_1, h_2 & d_1 are changed as per thread series (fine, normal & coarse).

Modified square thread

Modified square threads are similar to ordinary square threads except for the depth of the thread. The depth of thread is less than half pitch of the thread. The depth varies according to the application. The crest of the thread is chamfered at both ends to 45° to avoid the formation of burrs. These threads are used where quick motion is required.

Trapezoidal threads

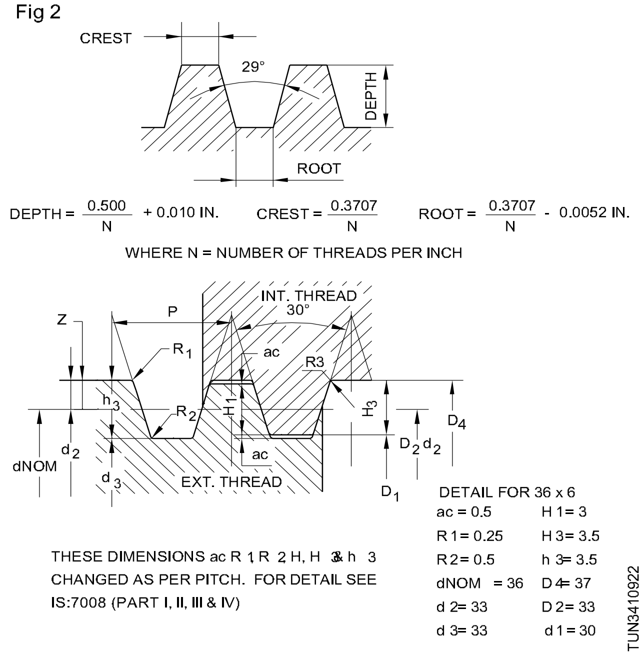
These threads have a profile which is neither square nor ‘V’ thread form and have a form of trapezoid. They are used to transmit motion or power. The different forms of trapezoidal threads are:

- acme thread
- buttress thread
- saw-tooth thread
- worm thread.

Acme thread (Fig 2)

This thread is a modification of the square thread. It has an included angle of 29°. It is preferred for many jobs because it is fairly easy to machine. Acme threads are used in lathe lead screws. This form of thread enables the easy engagement of the half nut. The metric acme thread has an included angle of 30°. The relationship between the pitch and the various elements is shown in the figure.

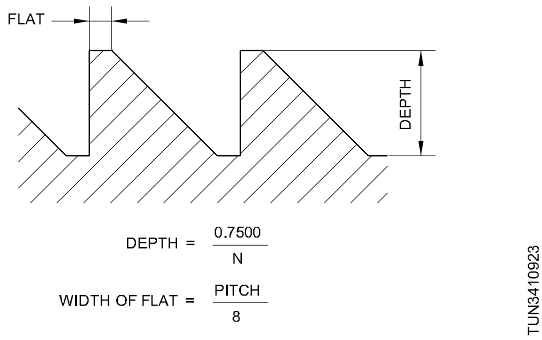
Fig 2



Buttress thread (Fig 3)

In buttress thread one flank is perpendicular to the axis of the thread and the other flank is at 45°. These threads are used on the parts where pressure acts at one flank of the thread during transmission. Figure 3 shows the various elements of a buttress thread. These threads are used in power press, carpentry vices, gun breeches, ratchets etc.

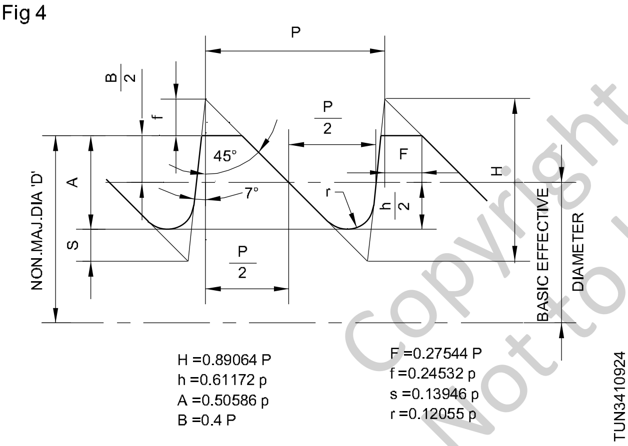
Fig 3



Buttress thread as per B.I.S. (Fig 4)

This is a modified form of the buttress thread. Figure 4 shows the various elements of the buttress thread. The bearing flank is inclined by 7° as per B.I.S. and the other flank has a 45° inclination.

Fig 4



Saw-tooth thread as per B.I.S. 4696

This is a modified form of buttress thread. In this thread, the flank taking the load is inclined at an angle of 3°, whereas the other flank is inclined at 30°. The basic profile of the thread illustrates this phenomenon. (Fig 5) The proportionate values of the dimensions with respect to the pitch are shown in Figs 6 and 7.

Fig 5

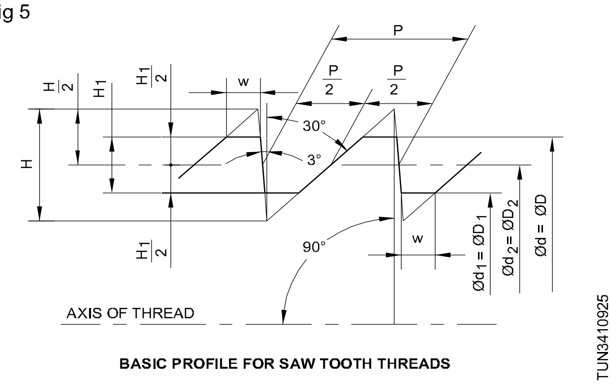


Fig 6

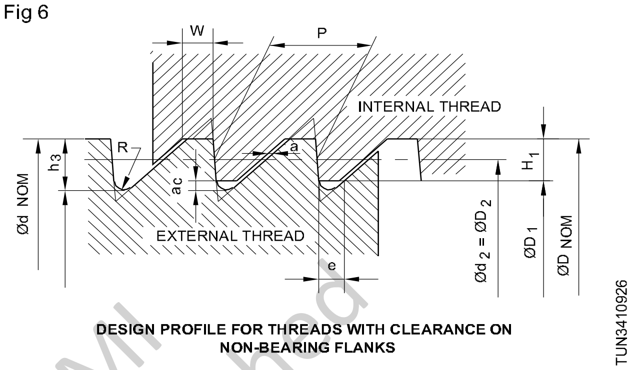
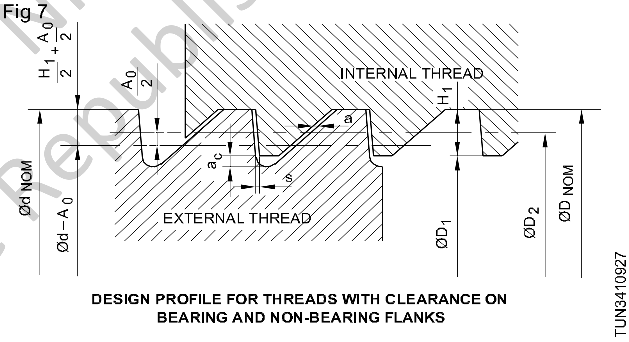


Fig 7



The equations associated with the dimensions indicated in the two figures (Figs 6 and 7) are given below.

H_1 = 0.75 P

$h_3 = H_1 + a_c$ = 0.867 77 P

a = 0.1 P (axial play)

a_c = 0.117 77 P

W = 0.263 84 P

e = 0.263 84 P - 0.1 P = W - a

R = 0.124 27 P

$D_1 = d - 2 H_1$ = d - 1.5 P

$d_3 = d - 2 h_3$

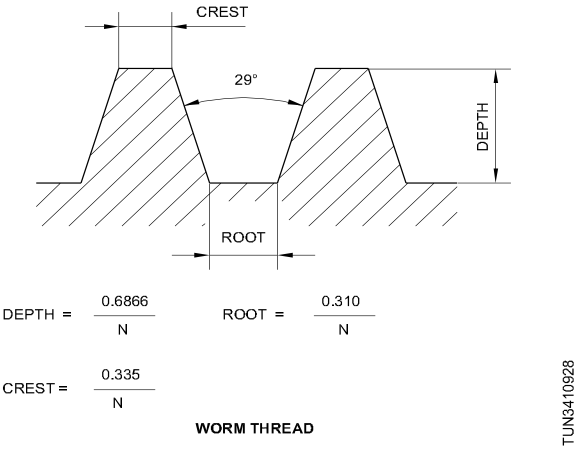
$d_2 = D_2$ = d - 0.75 P

S = 0.314 99 A_o, where A_o = basic deviation (= upper deviation) for external thread in the pitch diameter.

Worm thread

This is similar to acme thread in shape but the depth of thread is more than that of acme thread. This thread is cut on the worm shaft which engages with the worm wheel. Figure 8 shows the elements of a worm thread.

Fig 8



The worm wheel and worm shaft are used in places where motion is to be transmitted between shafts at right angles. It also gives a high rate of speed reduction. The worm wheel is generally cut by diametral pitch (D.P) or module pitch cutters. Diametral pitch (D.P) is the ratio between the number of teeth to the pitch diameter (P.D.) of the gear.

Module is the ratio between the pitch diameter of the gear and the number of teeth of the gear.

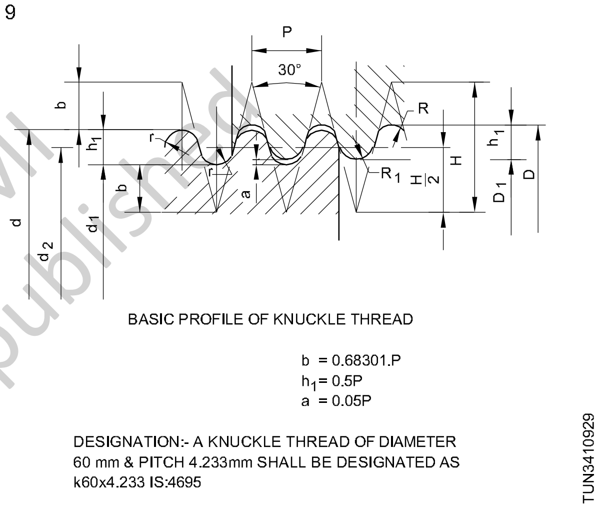
The linear pitch of the worm thread must be equal to the circular pitch of the worm gear. When the worm gear is of D.P. then the linear pitch of the worm thread in mesh is equal to p/DP. When the worm gear is of module teeth, then the linear pitch of the worm thread is equal to module x p.

In some of the lathes, a chart illustrates the position of levers of the quick change gearbox together with the change gear connections for cutting D.P. or module worm threads.

Knuckle threads

The shape of the knuckle thread is not trapezoidal but it has a rounded shape. It has limited application. The figure shows the form of knuckle thread. It is not sensitive against damage as it is rounded. It is used for valve spindles, railway carriage couplings, hose connections etc (Fig 9).

Fig 9



Helix angle and its effects on threading tool clearance angles

Objectives: At the end of this lesson you shall be able to
- **state the features of a helix angle**
- **state the effect of a helix angle**
- **calculate the leading and following angles for a square threading tool taking the helix angle into consideration.**

The helix angle is the angle included between the direction of the thread crest and the plane perpendicular to the axis. (Fig 1) This angle can be determined from the following formula.

$$\text{Tan } \theta = \frac{L}{\pi \times d}$$

Outside diameter

where q = Helix angle in degrees

L = Lead of the thread in millimetres (or) inches

π = Constant (3.14 or 22/7)

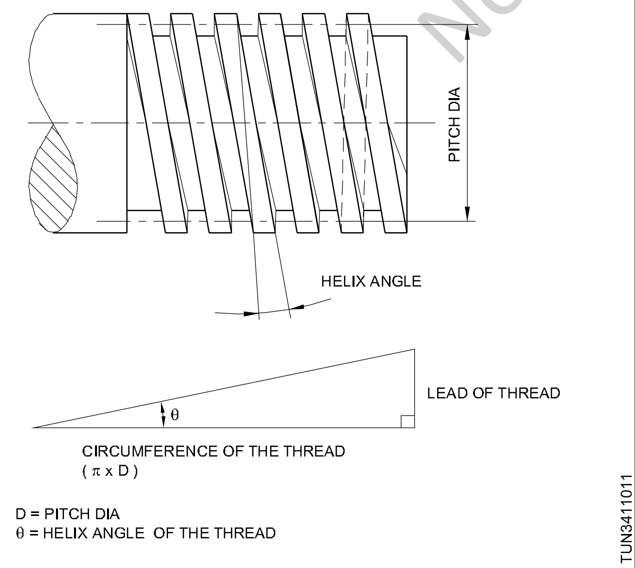
d = Pitch diameter of the thread in millimetres (or) inches.

Calculations

Example

Calculate the helix angle of the threads cut on a workpiece with the following data.

Fig 1



Data given

$$\text{Out side diameter} = 1\frac{1''}{2}$$

$$\text{Pitch} = \frac{1''}{4}$$

Number of starts = 4

$$\text{Pitch diameter of the job} = 1\frac{1''}{2} - \frac{1''}{8}$$

$$= 1\frac{3''}{8}$$

$$\text{Tangent of helix angle} = \frac{L}{\pi \times d}$$

$$\frac{1''}{\frac{22}{7} \times 1\frac{3''}{8}}$$

$$= \frac{1}{\frac{22}{7} \times \frac{11}{8}} = \frac{7 \times 8}{22 \times 11}$$

$$\text{Tan } \theta = \frac{56}{242} = 0.2314.$$

Helix angle = 13° approx.

Example

Calculate the helix angle of the thread cut on a job of 2" diameter, with 1/8" pitch, 3-start square thread.

Data given

Diameter of the job = 2"

$$\text{Pitch diameter of the job} = 2'' - \frac{1''}{16} = 1\frac{15''}{16}$$

$$\text{Tan of helix angle} = \frac{L}{\pi \times d}$$

$$= \frac{\frac{3}{8}}{\frac{22}{7} \times 1\frac{15}{16}}$$

$$= 3/8 \times \frac{7}{22} \times \frac{16}{31} = \frac{3 \times 7}{11 \times 31}$$

$$= \frac{21}{341} = 0.0615.$$

The helix angle = 3° 30'.

Helix angle of a thread and its effects

The helix angle of a thread and the angle of the square threading tool, depends upon two factors.

- The helix angle changes for different leads on a given diameter. The greater the lead of the thread, the greater will be the helix angle.
- The helix angle changes for each different dia-meter of thread for a given lead. The larger the diameter, the smaller will be the helix angle.

Helix angle (Fig 2)

The helix angle is the angle included between the direction of the thread crest and the plane perpendicular to the axis.

This angle can be determined from the formula,

$$\text{the tangent of helix angle} = \frac{\text{lead}}{\pi \times \text{pitch dia.}}$$

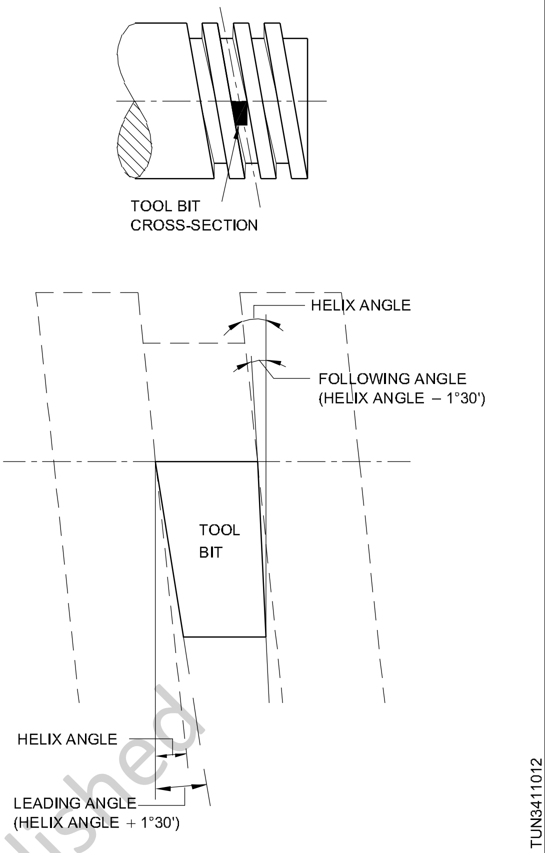
When grinding a tool for cutting multi-start threads, the helix angle of the thread is to be taken into consi-deration. The side clearances on the leading and the following sides of the tool will have to be of different values. The side clearance on the leading side of the tool is always greater than that ground on the following side.

The side clearance on the leading side of the tool = the helix angle of the leading side of the thread + the normal side clearance that will be ground on the tool (1° approximately).

Lead angle

Lead angle of a square thread is obtained by adding 1°30' as clearance to the helix angle. This follows the helix of a thread and always greater than helix angle.

Fig 2



The side clearance on the following side of the tool = the helix angle of the following side of the thread – the normal side clearance that will be ground on the tool. (1° approximately). The following example illustrates this. (Fig 2 may also clarify the above stated points.)

Example

Solution

Lead or pitch = 1/4" = 0.250 in.

Single depth = 0.5 x pitch

$$\text{Single depth} = \frac{0.500}{4}$$

= 0.5 x 0.25 = 0.125 in.

Double depth = 2 x 0.125

= 0.250 in.

Minor diameter = 1.250 – 0.250

= 1.000 in

Tan of the helix angle of the thread at the leading side

$$= \frac{\text{Lead}}{\text{Minor dia. circumference}}$$

$$= \frac{0.250}{1.000 \times \pi} = \frac{0.250}{3.1416}$$

= 0.0795.

The helix angle = 4° 33'

The tool bit's leading side clearance angle
= 4° 33' + 1° = 5° 33'.

Tan of the helix angle of the thread at the following side of the thread

$$= \frac{\text{Lead}}{\text{Major dia. circumference}}$$

$$= \frac{0.250}{1.250 \times \pi} = \frac{0.250}{3.927} = 0.0636.$$

Therefore the helix angle of the thread at the following side = 3° 38'.

Therefore the tool bit's following side clearance angle = 3° 38'–1° = 2° 38'.

Calculation involving change wheel, core dia in multi start thread cutting.

1 To be cut = 10 mm pitch square thead.

2 Dia = 62 mm

3 No of start = 2

4 Lead screw pitch = 6 mm pitch.

Find

a Depth of thread to give 0.12 mm clearance

b Lead of thread

c Core diameter

d Helix angle at the core diameter

e Helix angle of thread

f Gear ratio between the head stock spindle and the lead screw.

g Tool with angles at leading and trailing edges of the tools. Its main dimension and general shapes.

Ans :

a) depth of square thread

$$= \frac{\text{pitch}}{2} = \frac{10}{2} = 5 \text{ mm}$$

Depth of thread to give a clearance of 12 mm = 5+0.12 = 5.12 mm

b) Lead of thread= pitch x no of start

= 10 x 2 = 20 mm

c) Core diameter = out dia - 2 depth of thread

= 62 - (2x 5.12)

= 62 - 10.24 = 51.76 mm

d) Helix angle at core of dia thread

$$= \tan^{-1} + \frac{\text{Lead}}{\text{Core circumference}}$$

$$= \tan^{-1} + \frac{20}{51.76\,\pi} = 7°(\text{app})$$

e) Helix angle of thread

$$= \tan^{-1} + \frac{\text{Lead to be cut}}{\text{Mean circumference of work}}$$

$$= \tan^{-1} \frac{20}{(62-5)\,\pi} = \tan^{-1} \frac{20}{57\,\pi} = 6°\,20\,(\text{app})$$

f) Gear ratio

$$\frac{\text{Driver}}{\text{Driven}} = \frac{\text{Lead to be cut}}{\text{Lead of lead screw}}$$

$$= \frac{20}{6} = \frac{100}{30}$$

100 teeth gear wheel may be keyed to the lath spindle and 30 teeth gear wheel to the lathe lead screw.

Width of tool for a square thread

$$= \frac{\text{Pitch of thread}}{2}$$

$$= \frac{10}{2} = 5 \text{ mm}$$

British standard withworth thread

Angle = 55°

$$\text{Pitch} = \frac{1}{\text{thread per inch}}$$

Depth = pitch x 0.6403

Radious = pitch x 0.1373

Core dia = D - (1.28 x pitch)

D = outside dia. of bolt.

Acme thread (Fig 3)

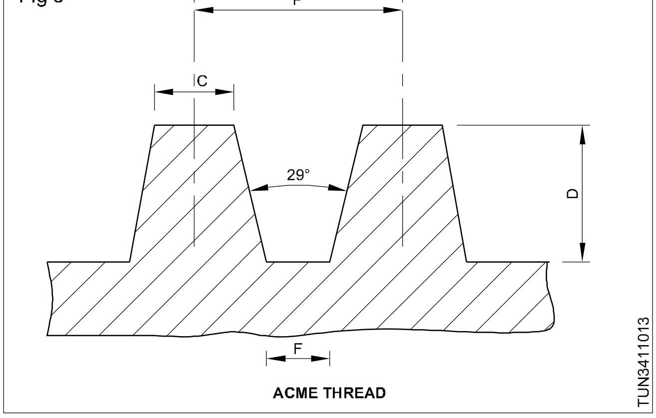
Angle= 29°

Depth = 0.5 x P + 0.01

Plain portion of crest = 0.317 x P

Plain portion at depth = 0.335 x P

Fig 3



Buttress thread (Fig 4)

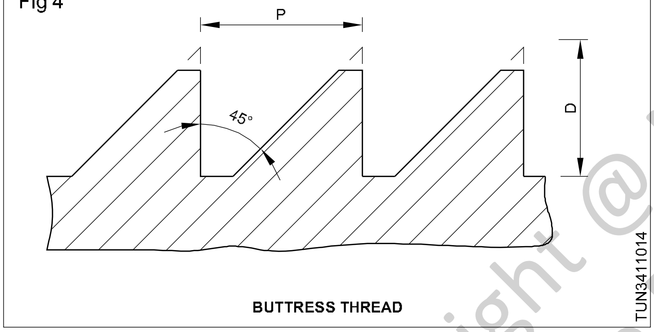
Angle = 45°

One side = 90°

Depth=0.75 x pitch

width of crest and root = 0.125x Pitch

Fig 4

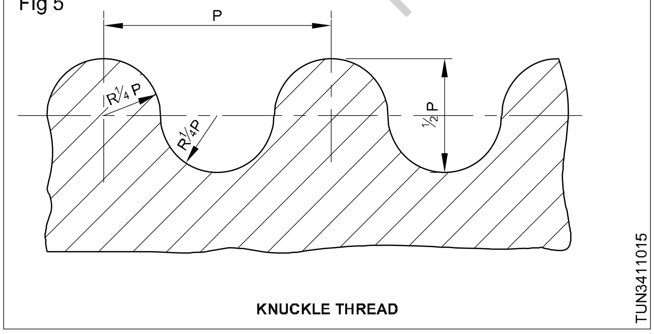


Knuckle thread (Fig 5)

$$R = \frac{1}{4} \times \text{pitch}$$

$$\text{Depth} = \text{pitch}$$

Fig 5



Worm thread or brown and sharp worm thread(Fig 6)

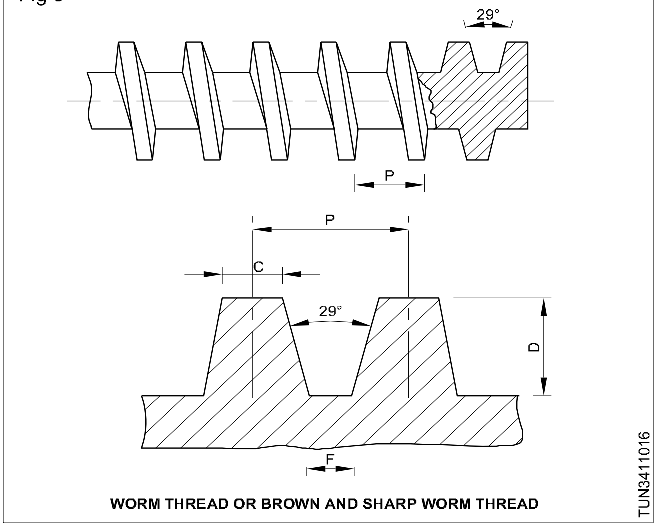
Angle = 29°

Depth = 0.6866x pitch

Plain portion of crest = 0.31 x pitch

Plain portion at depth = 0.335 x pitch

Fig 6



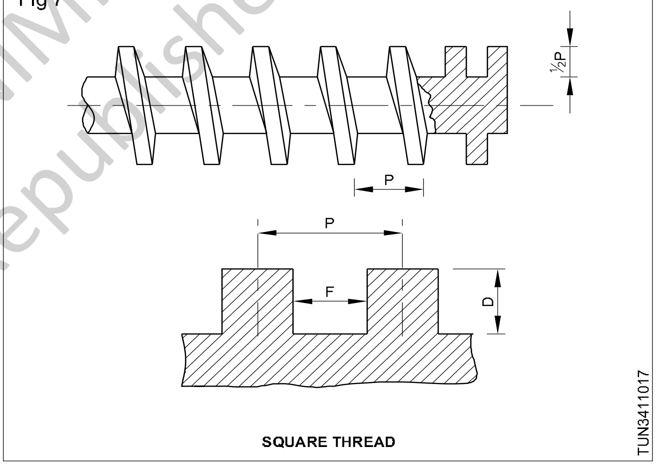
Square thread (Fig 7)

Angle = 90°

Width of tool = 0.5 pitch

Depth = 0.5 pitch

Fig 7



Metric 'V' thread (Fig 8)

Angle - 60°

Core dia

= Major dia - [2 x depth of thread]

Fig 8