TURNER

NSQF LEVEL - 4

2nd Year

TRADE THEORY

SECTOR : CAPITAL GOODS & MANUFACTURING

(As per revised syllabus July 2022 - 1200Hrs)

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 : **Capital Goods and Manufacturing**

Duration : **2 Years**

Trade : **Turner - 2nd Year Trade Theory - NSQF Level - 4 (Revised 2022)**

Developed & Printed by

National Instructional Media Institute
Post Box No.3142
Guindy, Chennai - 600032
INDIA
Email: chennai-nimi@nic.in
Website: www.nimi.gov.in

First Edition : November 2023 Copies : 1000

Rs.300/-

FOREWORD

The Government of India has set an ambitious target of imparting skills to 30 crores people, one out of every four Indians, 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 Media Development Committee members of various stakeholders 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 - 2nd Year - Trade Theory** in **CG & M Sector** under **Yearly Pattern.** The NSQF Level - 4 (Revised 2022) Trade Practical 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 - 4 (Revised 2022) trainees will also get the opportunities to promote life long learning and skill development. I have no doubt that with NSQF Level - 4 (Revised 2022) the trainers and trainees of ITIs, and all stakeholders will derive maximum benefits from these Instructional Media Packages IMPs and that NIMI's effort will go a long way in improving the quality of Vocational training in the country.

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

Jai Hind

ATUL KUMAR TIWARI, ***I.A.S***
Secretary
Ministry of Skill Development & Entrepreneurship,
Government of India.

November 2023
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 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.

In order to perform the skills in a productive manner instructional videos are embedded in QR code of the exercise in this instructional material so as to integrate the skill learning with the procedural practical steps given in the exercise. The instructional videos will improve the quality of standard on practical training and will motivate the trainees to focus and perform the skill seamlessly.

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.

Chennai - 600 032 **EXECUTIVE DIRECTOR**

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 2nd Year NSQF Level - 4 (Revised 2022)** under **Capital Goods & Manufacturing** Sector for ITIs.

MEDIA DEVELOPMENT COMMITTEE MEMBERS

Shri. A. Vijayaraghavan	-	Assistant Director of Training (Retd.), ATI, Chennai - 32.
Shri. Suriya Kumar	-	MTAB Technology Center (P) Ltd., Tamilnadu.
Shri. J.D. Dinesh Kumar	-	MTAB Technology Center (P) Ltd., Tamilnadu.
Shri. H.A. Manukumar	-	Training Officer, NSTI, Chennai.
Shri. K. Lakshminarayanan	-	Ex. Training Officer, DET, Tamilnadu.
Shri. M. Sampath	-	Training Officer (Retd.), CTI, Chennai - 32.
Shri. K. Karthi	-	Junior Training Officer, Govt ITI, Dharapuram.
Shri. V. Janarthanan	-	Assistant Professor, Rtd., MDC Member, NIMI, Chennai - 32.

NIMI CO-ORDINATORS

Shri. Nirmalya nath	-	Deputy Director of Training, NIMI, Chennai - 32.
Shri. V. Gopalakrishnan	-	Manager, NIMI, Chennai - 32.

NIMI records its appreciation of 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 staff who have contributed for the development of this Instructional Material.

NIMI is grateful to all others who have directly or indirectly helped in developing this IMP.

INTRODUCTION

TRADE PRACTICAL

The trade practical manual is intended to be used in practical workshop. It consists of a series of practical exercises to be completed by the trainees during the course. These exercises are designed to ensure that all the skills in compliance with NSQF Level - 4 (Revised 2022) syllabus are covered.

The manual is divided into Eight modules

Module 1	**Form Turning**
Module 2	**Lathe Accessories**
Module 3	**Turning with Lathe Attachments**
Module 4	**Boring**
Module 5	**Thread Cutting**
Module 6	**CNC Turning**
Module 7	**Advanced Turning**
Module 8	**Special Operation on Lathe**

The skill training in the shop floor is planned through a series of practical exercises centered around some practical project. However, there are few instances 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 trainee. However the development team accept that there is a scope for further improvement. NIMI looks forward to the suggestions from the experienced training faculty for improving the manual.

TRADE THEORY

The manual of trade theory consists of theoretical information for the Course of the **Turner 2nd Year NSQF Level - 4 (Revised 2022)** in **CG & M**. The contents are sequenced according to the practical exercise contained in NSQF Level - 4 (Revised 2022) syllabus on Trade Theory attempt has been made to relate the theoretical aspects with the skill covered in each exercise to the extent possible. This correlation is maintained to help the trainees to develop the perceptional capabilities for performing the skills.

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 exercises are given in every sheet of this manual.

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

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

CONTENTS

LEARNING / ASSESSABLE OUTCOME

On completion of this book you shall be able to

S.No	Learning Outcome	Ref. Ex.No.
1	**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.] (Mapped NOS: CSC/N0110)**	**2.1.88 - 2.1.94**
2	**Set & Produce components on irregular shaped job using different lathe accessories. [Different Lathe accessories: - Face plate, angle plate] (Mapped NOS: CSC/N0110)**	**2.2.95 - 2.2.96**
3	**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.] (Mapped NOS: CSC/N0110)**	**2.3.97 - 2.3.100**
4	**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] (Mapped NOS: CSC/N0110)**	**2.4.101 - 2.4.105**
5	**Calculate to set machine setting to produce different complex threaded component and check for functionality. [Different complex threaded component- Half nut, multi start threads (BSW, Metric & Square)] (Mapped NOS: CSC/N0110)**	**2.5.106 - 2.5.110**
6	**Set (both job and tool) CNC turn centre and produce components as per drawing by preparing part programme. (Mapped NOS:CSC/NO115)**	**2.6.111 - 2.6.148**
7	**Manufacture and assemble components to produce utility items by performing different operations & observing principle of interchangeability and check functionality. [Utility item: - screw jack/ vice spindle/ Box nut, marking block, drill chuck, collet chuck etc.; different operations: threading (Square, BSW, ACME, Metric), Thread on taper, different boring (Plain, stepped)] (Mapped NOS:CSC/NO115)**	**2.7.149 - 2.7.152**
8	**Make a process plan to produce components by performing special operations on lathe and check for accuracy. [Accuracy - ±0.02mm or proof machining & ±0.05mm bore; Special operation - Worm shaft cutting (shaft) boring, threading etc.] (Mapped NOS:CSC/NO115)**	**2.8.153 - 2.8.156**

SYLLABUS

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
Professional Skill 110 Hrs.; Professional Knowledge 30 Hrs.	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	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 indicator- construction & 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 (09 hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
	threading, knurling, grooving, chamfering etc.] (Mapped NOS: CSC/N0110)	91. Turn Morse taper plug (different number) and check with ring gauge / suitable MT sleeve. (20 hrs.) 92. Make revolving tail stock centre-Bush type (C-40). (Proof machining) (20 hrs.)	Cutting tool material-H.C.S., HSS, Tungsten. Carbide, Ceramic etc, - Constituents and their percentage. Tool life, quality of a cutting material. (13 hrs.)
		93. Make Morse taper sleeve and check by taper plug gauge. (25 hrs.)	Checking of taper with sin bar and roller-calculation involved (04 hrs.)
		94. Make mandrel/ plug gauge with an accuracy of ±0.02mm using tungsten carbide tools including throw-away tips. (20 hrs.)	Cutting speed, feed, turning time, depth of cut calculation, cutting speed chart (tungsten carbide tool) etc. Basic classification of tungsten carbide tips. (04 hrs.)
Professional Skill 40 Hrs.; Professional Knowledge 10 Hrs.	Set & Produce components on irregular shaped job using different lathe accessories. [Different Lathe accessories: - Face plate, angle plate] (Mapped NOS: CSC/ N0110)	95. Setting and turning operation involving face and angle plate (20 hrs.) 96. Make angle plate using face plate. (20 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. (10 hrs.)
Professional Skill 110 Hrs.; Professional Knowledge 30 Hrs.	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.] (Mapped NOS: CSC/ N0110)	97. Holding and truing of Crankshaft - single throw (Desirable). (45 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. (12 hrs.)
		98. Turning of long shaft using steady rest (within 0.1 mm). (20 hrs.)	Roller and revolving steadies, Necessary, construction, uses etc. (06 hrs.)
		99. Use of attachments on lathe for different operations. (20 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. (12 hrs.)
Professional Skill 80 Hrs.; Professional Knowledge 18 Hrs.	Set the machining parameters and produce & assemble components by performing different boring operations with an appropriate accuracy. [Different boring	101. Perform eccentric boringand 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. (05 hrs.)
		103. Boring and stepped boring (within ± 0.05 mm) (10hrs.)	

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
	operation - eccentric boring, stepped boring; appropriate accuracy - ±0.05mm] (Mapped NOS: CSC/N0110)	104.Cutting of helical grooves in bearing and bushes (Oil groove) (10 hrs.)	Inside micrometer principle, construction graduation, reading, use etc. (Metric & Inch.) (05 hrs.)
		105.Turning & boring of split bearing - (using boring bar and fixture) (35 hrs.)	Care for holding split bearing. Fixture and its use in turning. (8 hrs.)
Professional Skill 110 Hrs.; Professional Knowledge 28 Hrs.	Calculate to set machine setting to produce different complex threaded component and check for functionality. [Different complex threaded component- Half nut, multi start threads (BSW, Metric & Square)] (Mapped NOS: CSC/N0110)	106.Cutting thread of 8 and 11 TPI. (20 hrs.)	Calculation involving fractional threads. Odd & even threads. (04 hrs.)
		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. (04 hrs.)
		108.Multi start thread cutting (Metric) (External & internal). (20 hrs.)	Indexing of start - different methods tool shape for multi- start thread. Setting of a lathe calculation for required change wheel (06 hrs.)
		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.(05 hrs.)
		110.Make half nut as per standard lead screw. (20 hrs.)	Helix angle, leading angle & following angles. Thread dimensions-tool shape, gear, gear calculation, pitch, depth, lead etc. (09 hrs.)
Professional Skill 210 Hrs.; Professional Knowledge 62 Hrs.	Set (both job and tool) CNC turn centre and produce components as per drawing by preparing part programme. (Mapped NOS:CSC/NO115)	111.Personal and CNC machine Safety: Safe handling of tools, equipment and CNC machine. (2 hrs.) 112.Identify CNC machine, CNC console. (3 hrs.) 113.Demonstration of CNC lathe machine and its parts bed, spindle motor and drive, chuck, tailstock, turret, axes motor and ball screws, guide ways, LM guides, console, controlswitches, coolant system, hydraulic system, chip conveyor, steady rest. (6 hrs.) 114.Working of parts explained using Multimedia based simulator for CNC parts shown on machine. (3 hrs.) 115.Identify machine over travel limits and emergency stop. (2 hrs.)	CNC technology basics: Difference between CNC and conventional lathes. Advantages and disadvantages of CNC machines over conventional machines. Machine model, control system and specification. Axes convention of CNC machine - Machine axes identification for CNC turn centre. Importance of feedback devices for CNC control. Concept of Co-ordinate geometry, concept of machine axis. (05 hrs.)
		116.Conduct a preliminary check of the readiness of the CNC turning centre viz., cleanliness of machine, referencing - zero return, functioning	Programming - sequence, formats, different codes and words. Co-ordinate system points and simulations.

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		of lubrication, coolant level, correct working of sub-system. (2 hrs.) 117.Identification of safety switches and interlocking of DIH modes. (1 hr.) 118.Machine starting & operating in Reference Point, JOG and Incremental Modes. (6 hrs.) 119.Check CNC part programming with simple exercises and using various programming codes and words. (05 hrs.) 120.Check the programme simulation on machine OR practice in simulation software in respective control system. (05 hrs.) 121.Absolute and incremental programming assignments and simulations. (05 hrs.) 122. Linear interpolation, and Circular interpolation assignments and simulations on software. (6 hrs.)	Work piece zero points and ISO/ DIN G and M codes for CNC. Different types of programming techniques of CNC machine. Describe the stock removal cycle in CNC turning for OD / ID operation. L/H and R/H tool relation on speed. Describe CNC interpolation, open and close loop control systems. Co-ordinate systems and Points. Program execution in different modes like manual, single block and auto. Absolute and incremental programming. Canned cycles. Cutting parameters- cutting speed, feed rate, depth of cut, constant surface speed, limiting spindle speed, tool wear, tool life, relative effect of each cutting parameter on tool life. Selection of cutting parameters from a tool manufacturer's catalog for various operations. Process planning & sequencing, tool layout & selection and cutting parameters selection. Tool path study of machining operations Prepare various programs as per drawing. (15 hrs.)
		123.Perform Work and tool setting: - Job zero/work coordinate system and tool setup and live tool setup. (10 hrs.) 124.Carryout jaw adjustment according to Diameter and tooling setup on Turret. (10 hrs.) 125.CNC turning centre operation in various modes: JOG, EDIT, MDI, SINGLE BLOCK, AUTO. (10 hrs.) 126.Program entry. (2 hrs.) 127.Set the tool offsets, entry of tool nose radius and orientation. (8 hrs.) 128.Conduct work off set measurement, Tool off set measurement and entry in CNC Control. (8 hrs.) 129.Make Tool nose radius and tool orientation entry in CNC control. (5 hrs.)	Tool Nose Radius Compensation (G41/42) and its importance (TNRC). Cutting tool materials, cutting tool geometry - insert types, holder types, insert cutting edge geometry. - Describe Tooling system for turning - Setting work and tool offsets. - Describe the tooling systems for CNC TURNING Centers. - Cutting tool materials for CNC Turning and its applications - ISO nomenclature for turning tool holders, boring tool holders, indexable inserts. - Tool holders and inserts for radial grooving, face grooving, threading, drilling.(17 hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		130.Jaw removal and mounting on CNC Lathe. (5 hrs.) 131.Manual Data Input (MDI) and MPG mode operations and checking of zero offsets and tool offsets. (7 hrs.)	
		132.Program checking in dry run, single block modes. (5 hrs.) 133.Checking finish size by over sizing through tool offsets. (5 hrs.) 134.Part program preparation, Simulation & Automatic Mode Execution for the exercise on Simple turning & Facing (step turning) (6 hrs.) 135.Part program preparation, Simulation & Automatic Mode Execution for the exercise on Turning with Radius / chamfer with TNRC. (6 hrs.) 136.Part program preparation, Simulation & Automatic Mode Execution of CNC Machine for the exercise on Blueprint programming contours with TNRC. (6 hrs.) 137.Machining parts on CNC lathe with parallel, taper, step, radius turning, grooving & threading. (10 hrs.) 138.Carryout Drilling /Boring cycles in CNC Turning. (12 hrs.) (First 60 % of the practice ison CNC machine simulator, followed by 40 % on machine.)	Prepare various part programs as per drawing & check using CNC simulator. Processes and Tool selection related to grooving, drilling, boring & threading. (10 hrs.)
		139.Geometry Wear Correction. Geometry andwear offset correction. (4 hrs.) 140.Produce components on CNC Machine involving different turning operations viz., • Stock removal cycle OD • Drilling / boring cycles • Stock removal cycle ID • Carryout threading in different pitches. (12 hrs.) 141.Produce components by involving turning operation and part programme exercises of CNC turning viz., • Grooving and thread cutting OD • Grooving and thread cutting ID • Threading cycle OD • Sub programs with repetition	- Describe Tapping on CNC turning. - Programming for Grooving/ Threading on OD/ID in CNC Turning. - Trouble shooting in CNC lathe machine - Identify Factors affecting turned part quality/ productivity. - Parting off operation explanation. - Bar feeding system through bar feeder. - Input and Output of Data. - DNC system. Interlacing with PC. - Use of CAM Programme. (Optional) (15 hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		• Using Sub Programs & Cycles in the Main Program. (12 hrs.) 142.Part off: Part Prog. (3 hrs.) 143.Produce job involving profile turning, threading on taper, boring, etc. operations. (15 hrs.) 144.Demo on M/C on bar feeding system. (simulation/ video) (1 hr.) 145.DNC system setup. (Optional) 146.Run the machine on DNC mode. (Optional) 147.CAM programme execution. (Optional) 148.Data Input-Output on CNC machine. (2 hrs.)	
Professional Skill 80 Hrs.; Professional Knowledge 20 Hrs.	Manufacture and assemble components to produce utility items by performing different operations & observing principle of interchangeability and check functionality. [Utility item: - screw jack/ vice spindle/ Box nut, marking block, drill chuck, collet chuck etc.; different operations: threading (Square, BSW, ACME, Metric), Thread on taper, different boring (Plain, stepped)] (Mapped NOS:CSC/ NO115)	149.Thread on taper surface (Vee form). (40 hrs.)	Setting of tools for taper threads-calculation of taper setting and thread depth. Heat treatment - meaning & procedure hardening, tempering, carbonizing etc. Different types of metal used in engineering application. (8 hrs.)
		150.Manufacturing & Assembly of Screw jack/vice/Box nut by performing different lathe operation. (To use earlier produce screw jack). (20 hrs.)	Interchangeability meaning, procedure for adoption, quality control procedure for quality production. (06hrs.)
		151.Prepare different types of documentation as per industrial need by different methods of recording information. (4 hrs.) 152.Turn Bevel gear blank. (16 hrs.)	Importance of Technical English terms used in industry -(in simple definition only)Technical forms, process charts, activity logs in required formats of industry, estimation, cycle time, productivity reports, job cards. (06 hrs.)
Professional Skill 100 Hrs.; Professional Knowledge 28 Hrs.	Make a process plan to produce components by performing special operations on lathe and check for accuracy. [Accuracy - ±0.02mm or proof machining & ±0.05mm bore; Special operation - Worm shaft cutting (shaft) boring, threading etc.] (Mapped NOS:CSC/NO115)	153.Read a part drawing, make a process plan for turning operation and make arbor with clamping nut (hexagonal). (40hrs.)	Terms used in part drawings and interpretation of drawings - tolerances, geometrical symbols - cylindricity, parallelism, etc. (11 hrs.)
		154.Practice of special operations on lathes - worm gear cutting. (Shaft) (20 hrs.)	Automatic lathe-its main parts, types diff. Tools used-circular tool etc. (09 hrs.)
		155.Boring on lathe using soft jaws to make bush with collar (standard) on nonferrous metal andcheck with dial bore gauge to accuracy of +/- 0.05 mm. (25hrs.) 156.Make Arbor support bush. (Proof Machining) (15hrs.)	Related theory and calculation. (8 hrs.)

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.

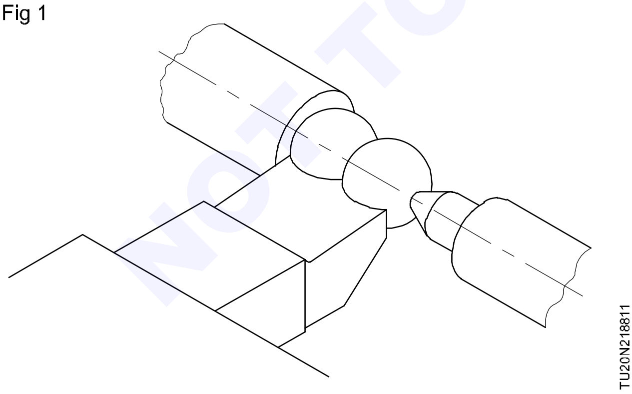
Form turning is largely employed in the manufacture of automobile engineering components.

Methods of turning formed surfaces

Formed surfaces can be turned by:

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

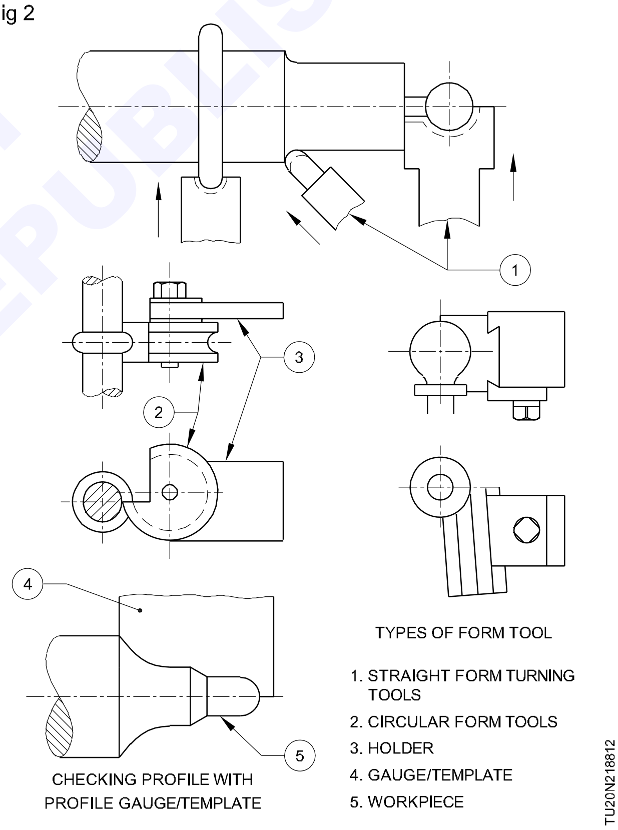
Fig 1



Form turning tools (Figs 1 & 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.

Fig 2



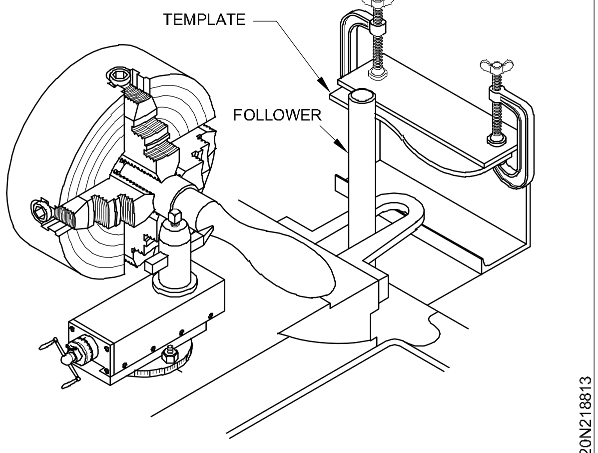
Form turning using a template (Fig 3)

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.

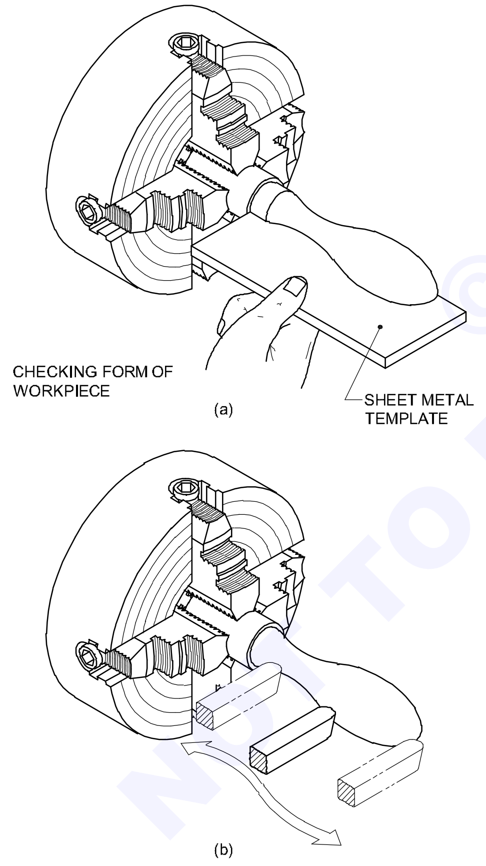
Fig 3



SETUP FOR PRODUCING A SPECIAL FORM

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

Fig 4



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.

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 5 shows the application of a fillet gauge to check the radius formed externally. Fig 6 shows the application of a radius gauge to check the fillet formed on a turned component. The other typical applications are:

Fig 5

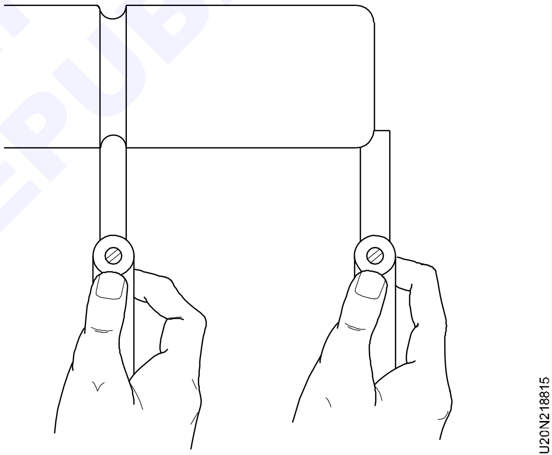
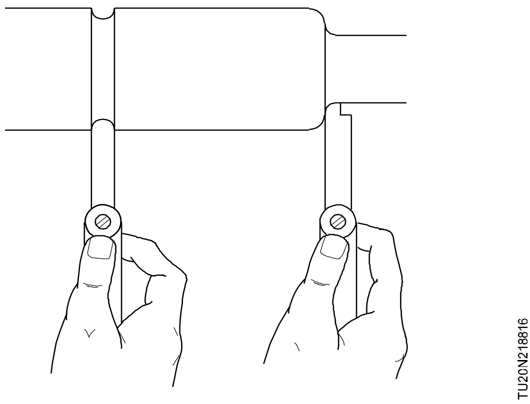


Fig 6



- Checking the corner radius of a part being filed to shape. (Fig 7)
- Checking a radius formed by a milling cutter. (Fig 8)

Fig 7

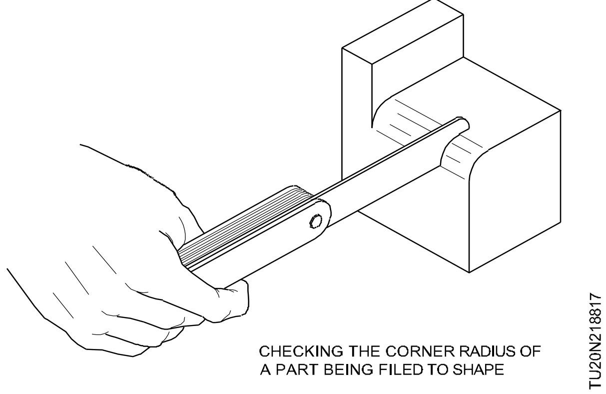
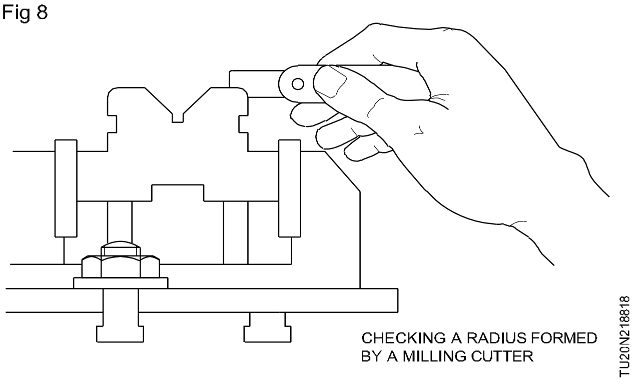
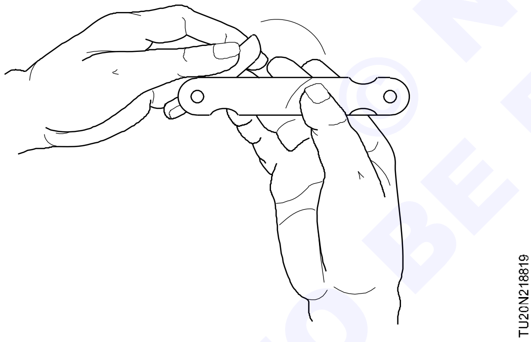


Fig 8



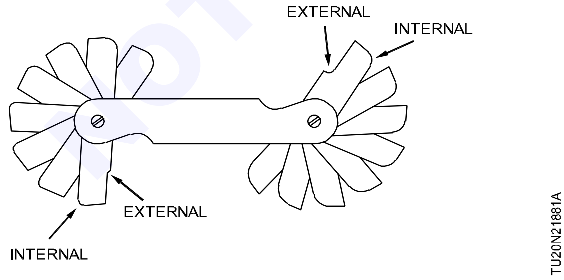
The radius and fillet gauges are available in sets of several blades which fold into a holder when not in use. (Fig 9)

Fig 9



Some sets have provisions to check the radius and fillet on each blade. (Fig 10)

Fig 10



And some sets have separate sets of blades to check the radius and fillet. (Fig 11)

Each blade can be swung out of the holder separately, and has its size engraved on it. (Fig 12)

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

Fig 11

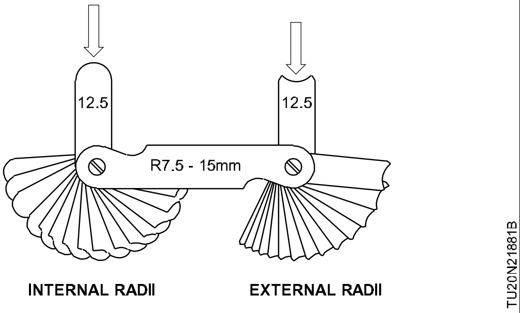
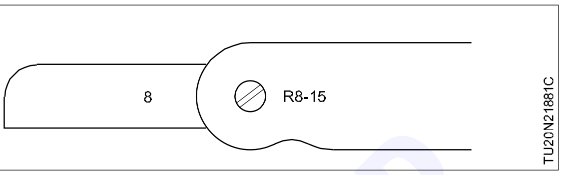


Fig 12



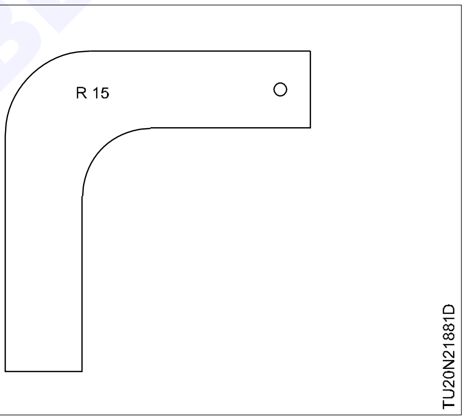
1 to 7 mm in steps of 0.5 mm

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 13)

Fig 13

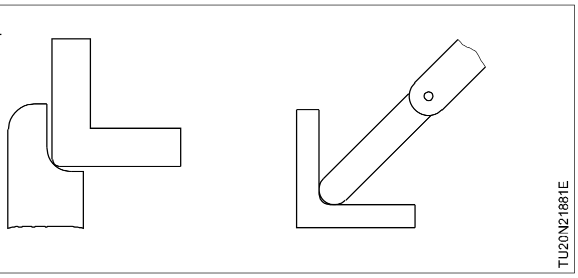


Before using the radius gauge:

- 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 14 shows that the radius of the fillet and that of the external radius are smaller than the gauge.

Fig 14

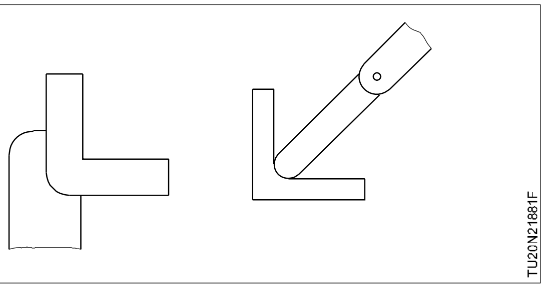


Try a smaller gauge to determine the radius dimension.

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

Fig 15 shows that the radius of the fillet and that of the external radius are larger than the gauge.

Fig 15

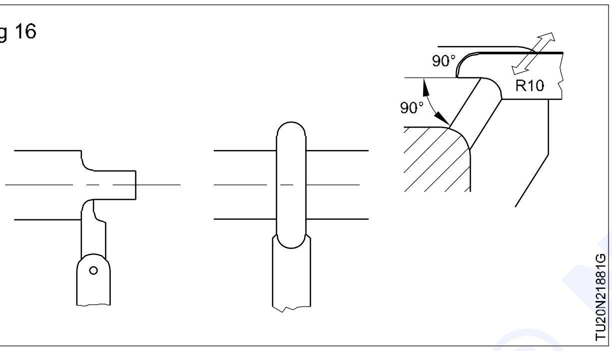


Try a larger gauge if you need to find the radius dimension.

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

Fig 16 shows the workpiece having the same radius as that of the gauge that is being used for checking.

Fig 16



Templates (Profiles)

A template is made of good quality steel. It is used to mark and check profiles (Figs 17,18, & 19)

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 17

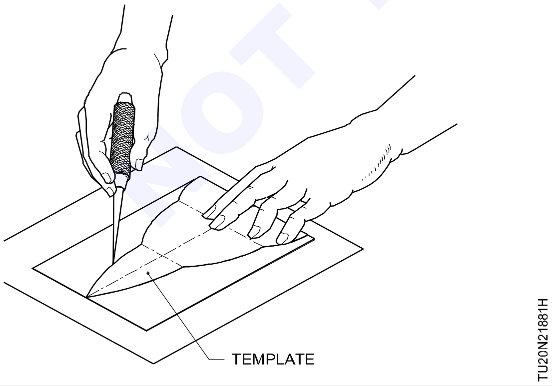


Fig 18

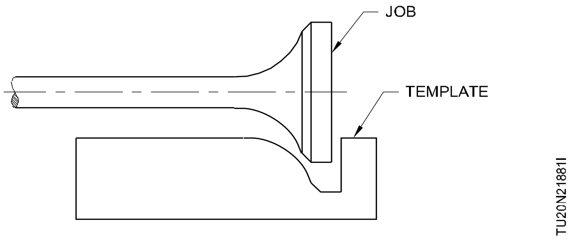
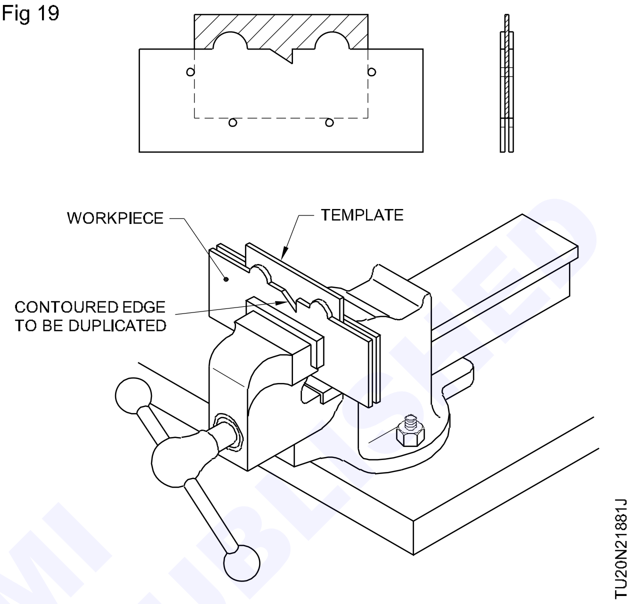


Fig 19



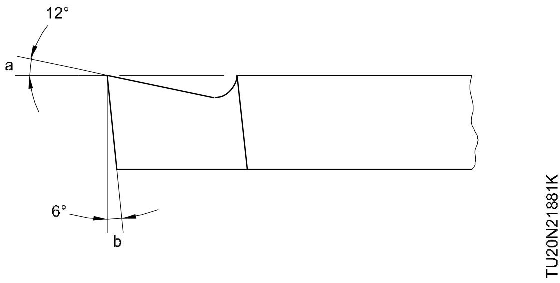
Templates save time in marking out and and result in standarisation of work.

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 20.

Fig 20

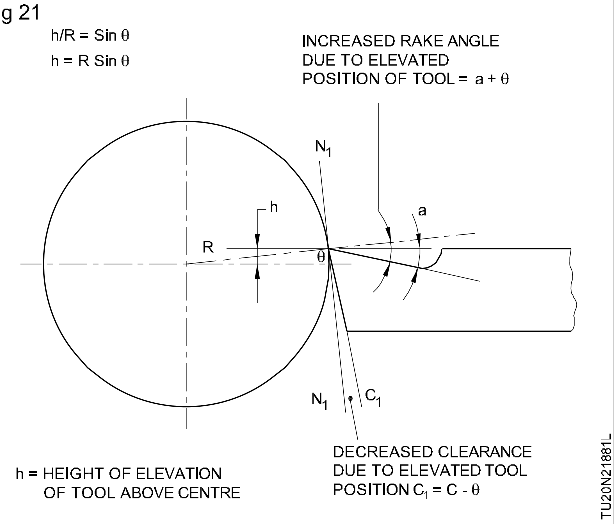


If it is above the work centre, the clearance angle decrease and the rake angle increases, and there is a point at which clearance disappears completely and the tool will not cut but only rub on the surface of the work. Converely, if it below the centre, it will increase the clearance angle and decrease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

Example (Fig 21)

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 21, we have

Fig 21



OA = 25 mm

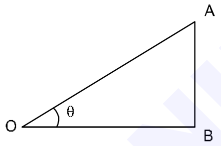
AB = 2 mm

Sin q = 2/25 = 0.080

q = 4° 34'

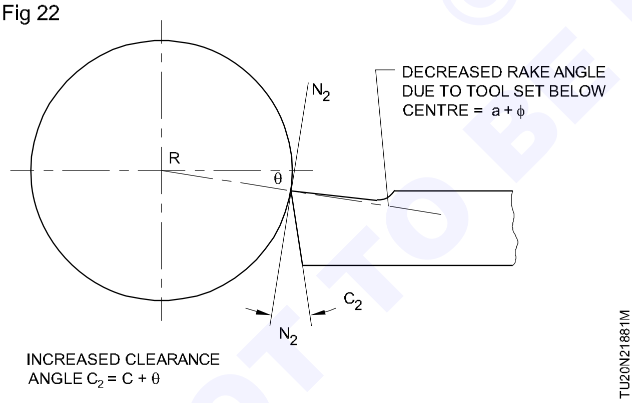
The effective top rake angle = 12° + 4° 34'
= 16° 34'

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



Example Fig 22

Fig 22



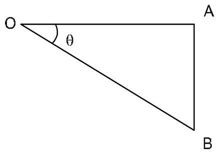
Used on a job of 60 mm dia. The tool set at 2 mm below the centre.

OB = 30 mm

AB = 2 mm

Sin q = 2/30 = 1/15 = 0.066

q = 3° 50'

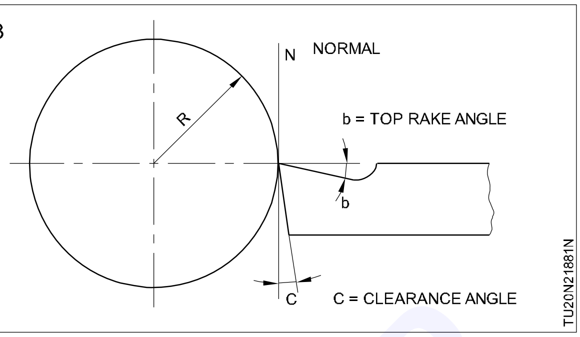


The effective top rake angle = 12° - 3° 50'
= 8° 10'

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

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

Fig 23



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

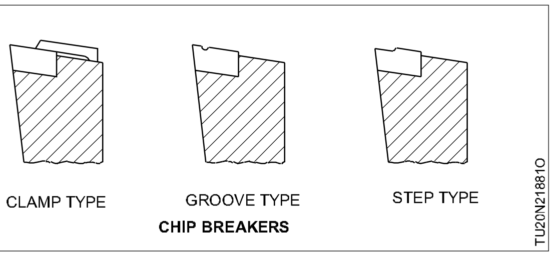
Chip breaker is a means with which the continuous long curly chips are broken into comparatively small pieces for easy handling, thus preventing it from becoming a work hazard.

Types of chip breakers (Fig 24)

- 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 24



- 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.

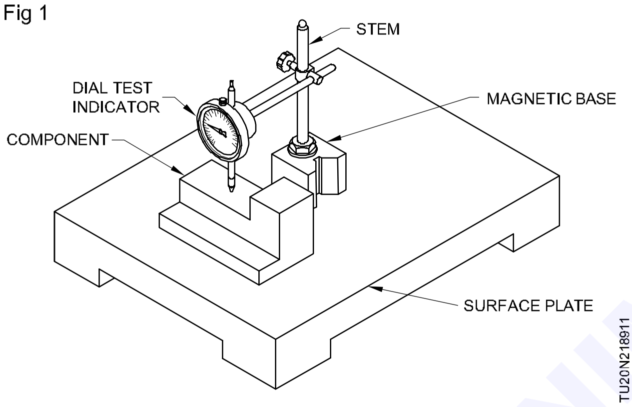
Dial test indicator - construction - uses

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.

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 2

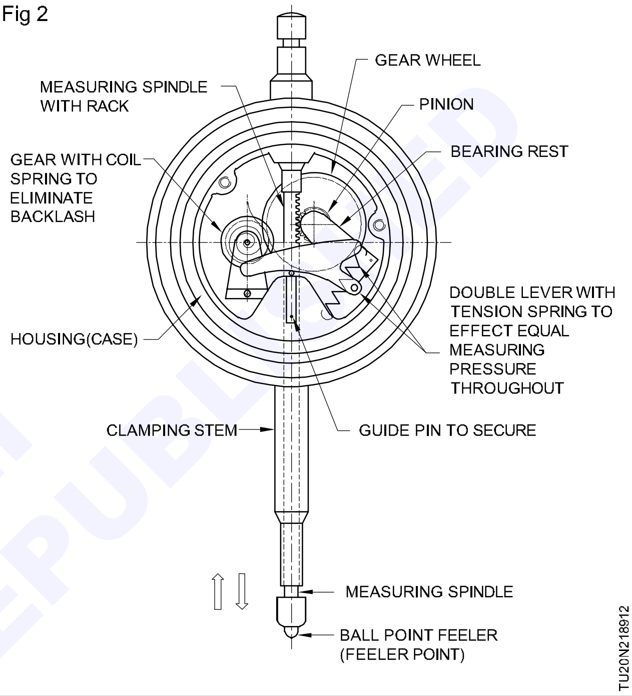
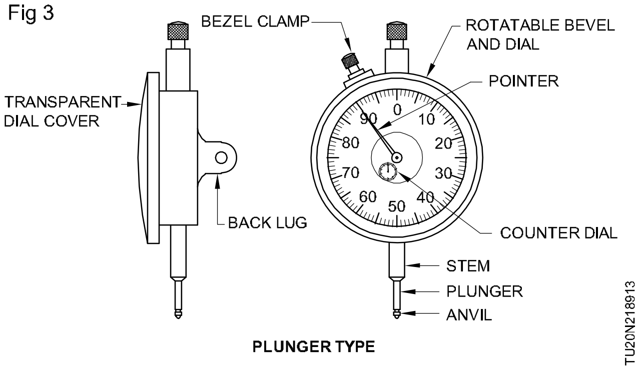
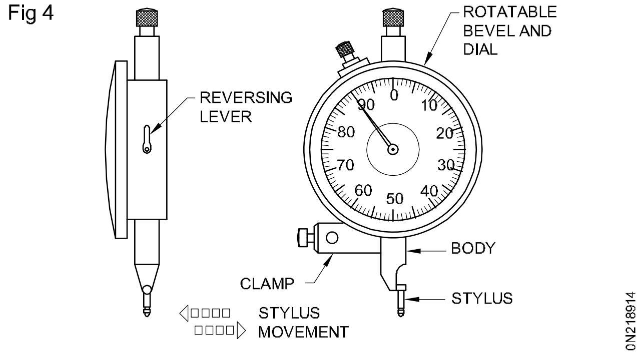


Fig 3



PLUNGER TYPE

Fig 4

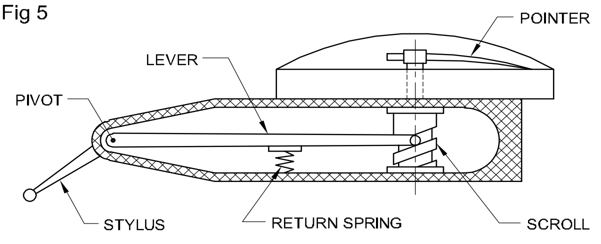


LEVER TYPE

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

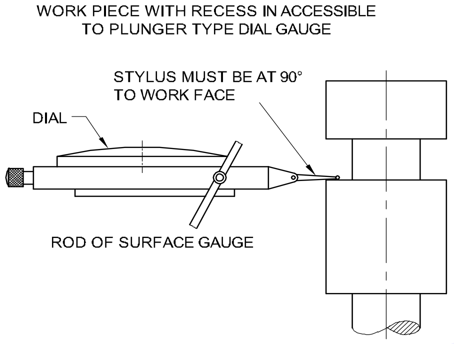


MECHANISM OF THE LEVER TYPE DIAL GAUGE

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)

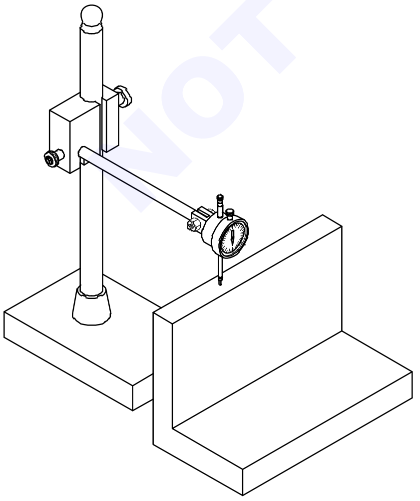
Fig 6



Uses (Figs 7 to 11)

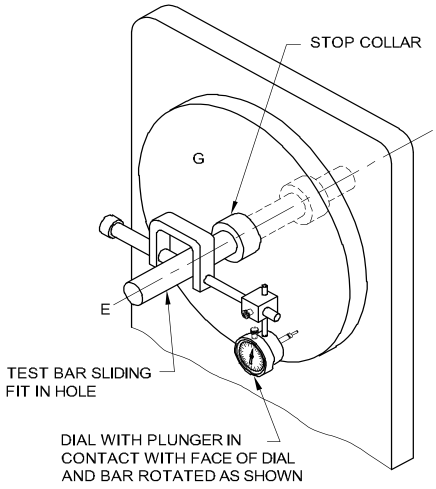
- To compare the dimensions of a workpiece against a known standard.
- To check plane surfaces for parallelism and flatness.
- To check parallelism of shafts and bars.
- To check concentricity of holes and shafts.
- To check the radial/axial runouts

Fig 7



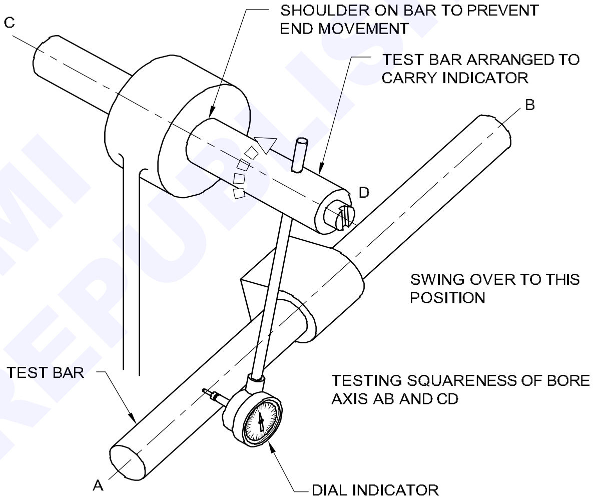
CHECKING FOR PARALLELISM

Fig 8



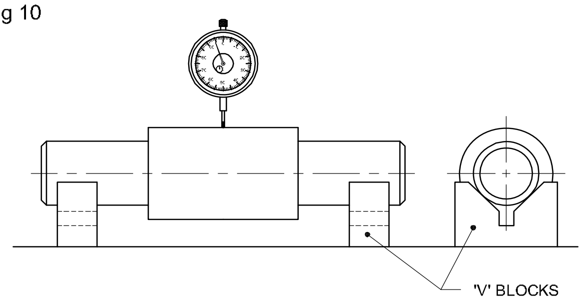
TESTING SQUARENESS OF FACE G WITH AXIS EF OF CENTRE HOLE

Fig 9

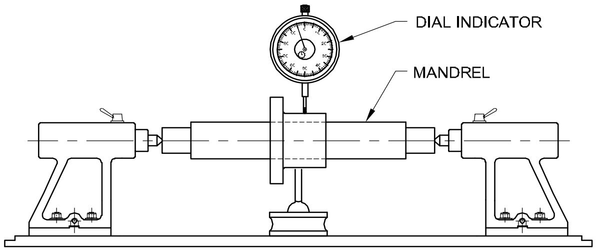


TEST SQUARENESS OF BORE AXIS USING TEST BAR

Fig 10

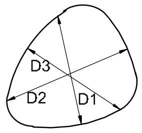


CONCENTRICITY TEST ON ENLARGED PORTION OF SHAFT

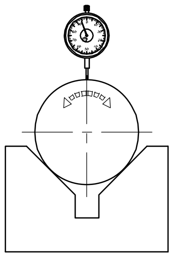


TESTING BUSH FOR CONCENTRICITY

Fig 11



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



TEST FOR ROUNDNESS WITH VEE BLOCKS AND CLOCK INDICATOR

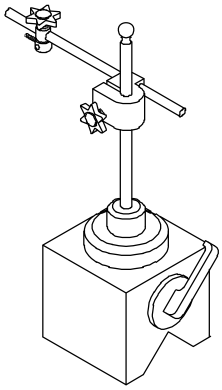
Stands

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)

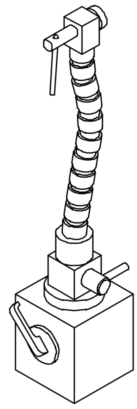
Fig 12



MAGNETIC STAND UNIVERSAL CLAMP

Magnetic stand with flexible post. (Fig 13)

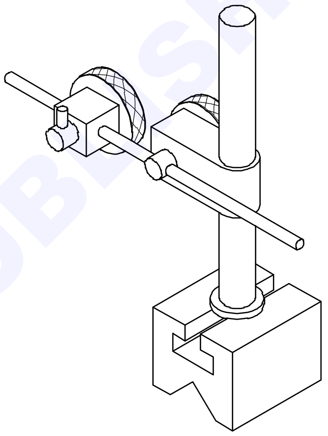
Fig 13



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

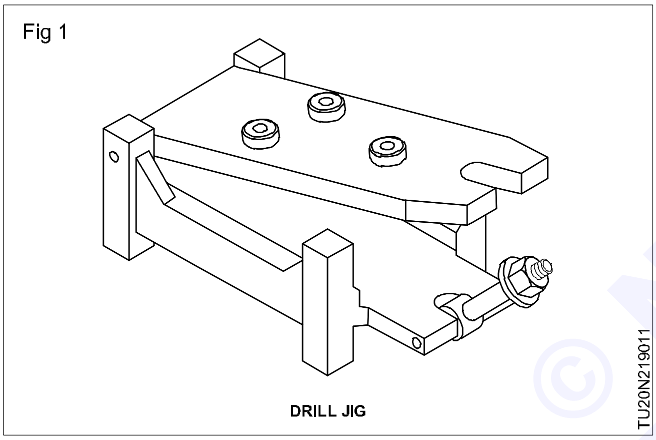
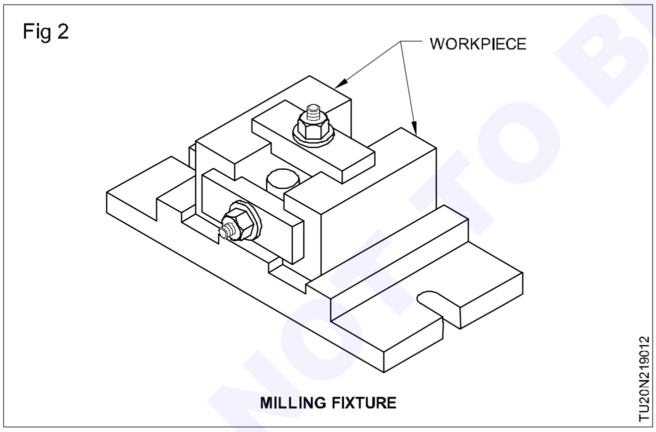


Fig 2



Definition of a jig

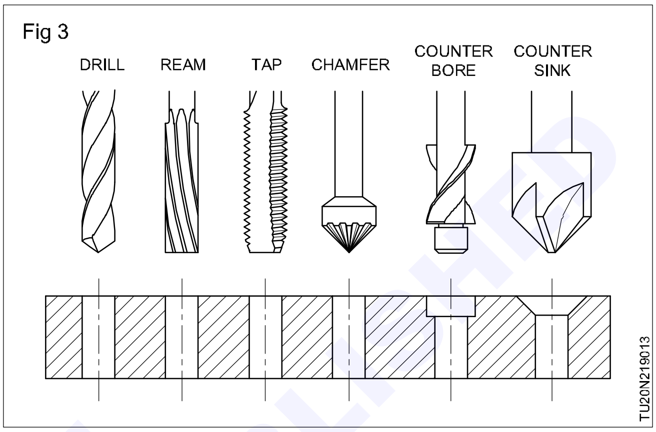
A jig is a special device which holds, supports, locates and also guides the cutting tool during operation.

The two types of jigs are

- drilling jig
- boring jig.

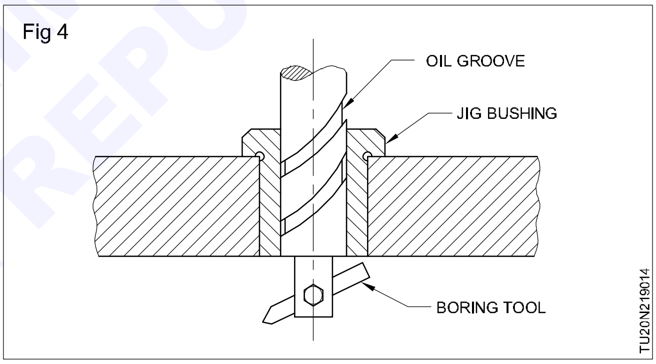
Drilling jigs are used to drill, ream, tap and for other allied operations as shown on the component. (Fig 3)

Fig 3



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

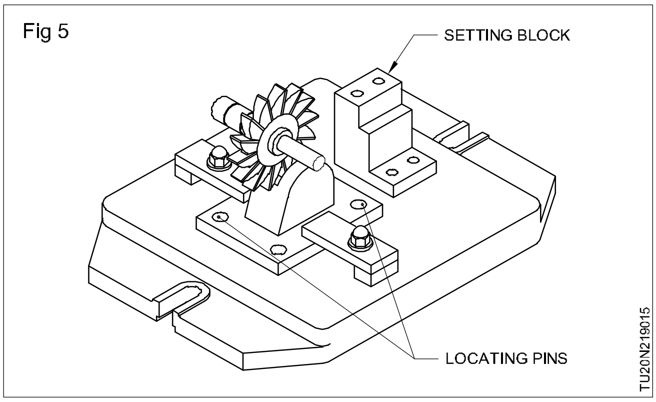
Fig 4



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



The commonly used fixtures are

- milling fixture

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

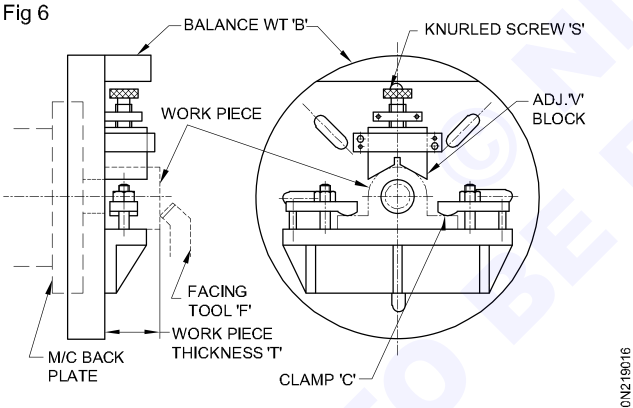
Features of turning fixtures

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)

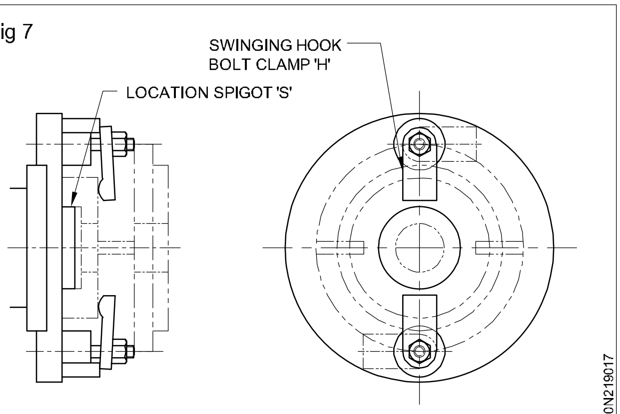
The workpiece is located on the earlier machined spigot 'S' and clamped against the fixture face by two swinging hook bolt clamps 'H'. The clamps are loosened, and they swing anticlockwise to the position shown by chain-dotted lines to clear the path of the workpiece during loading and unloading. Because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

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.

Fig 7



SECOND OPERATION TURNING FIXTURE

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 8, 9 & 10)

Fig 8

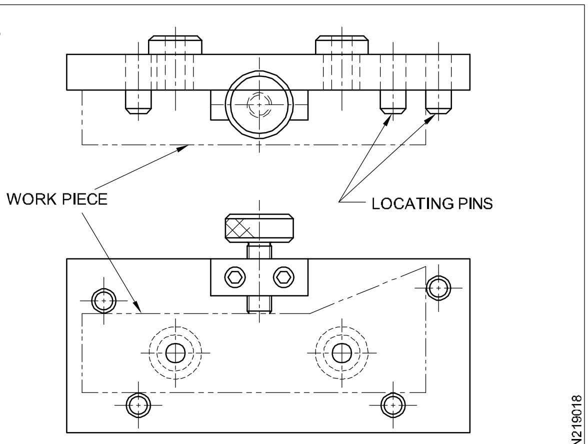


PLATE JIG

Fig 9

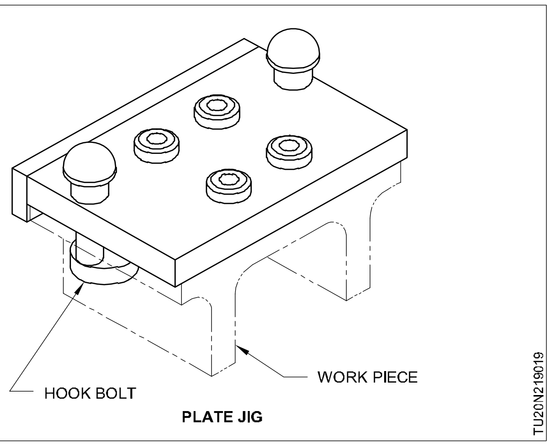
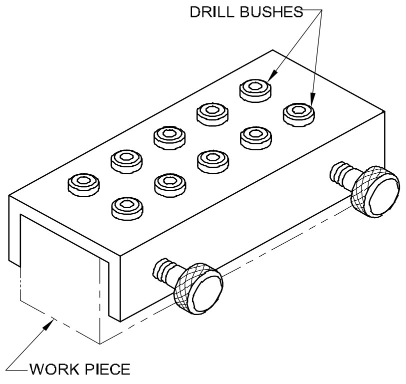


PLATE JIG

Fig 10

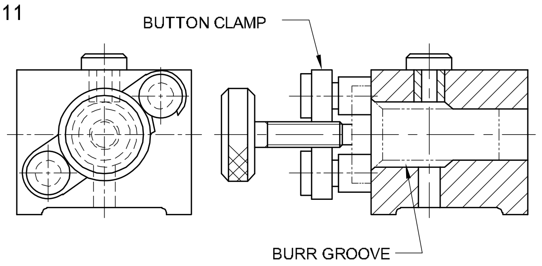


CHANNEL JIG

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 11)

Fig 11

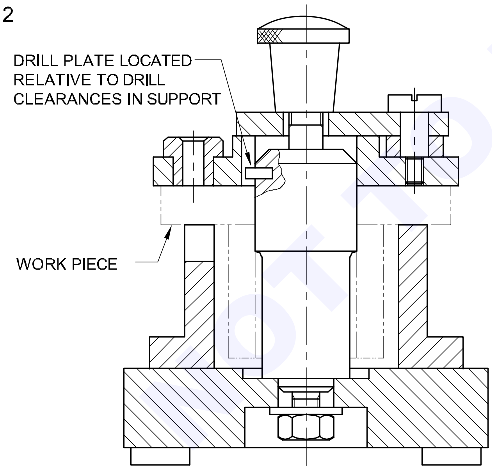


SOLID JIG

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 12)

Fig 12

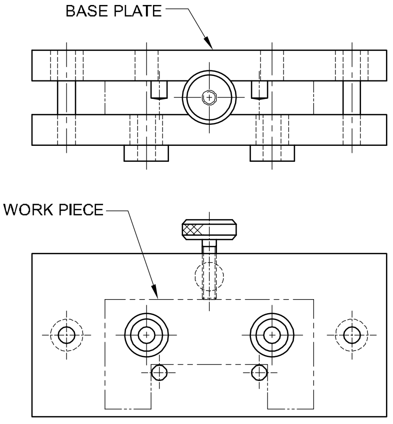


POST JIG

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 13)

Fig 13



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 14)

Fig 14

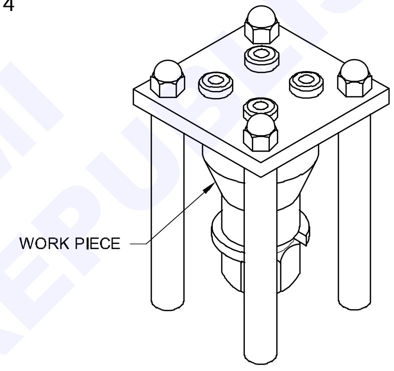
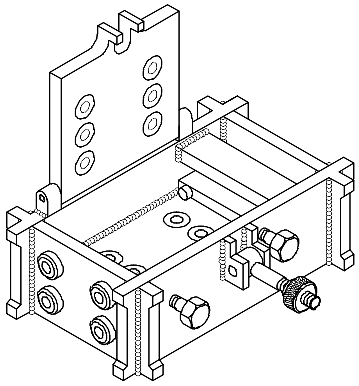


TABLE OR TURN OVER JIG

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 15)

Fig 15

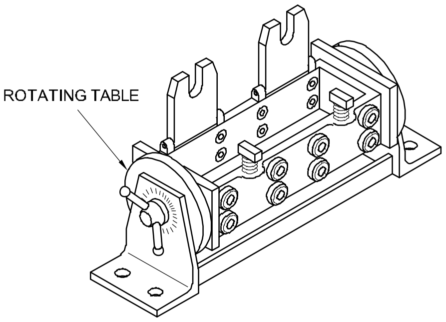


BOX JIG

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 16)

Fig 16

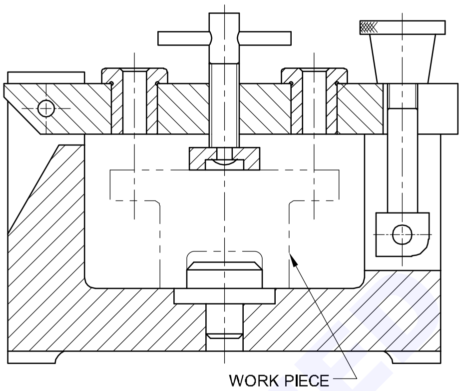


TRUNNION JIG

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 17)

Fig 17



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) (HCS)

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.

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.

Tungsten Carbide

Carbide tools are the most common choice for CNC machining due to their exceptional heardness and wear resistance.

Tungsten carbide (WC) is the primary constituent, with cobalt (Co) acting as a binder material.

Carbide tools can withstand high cutting speeds and temperatures, making them ideal for machining hard materials such as stainless steel, cast iron, and exotic alloys.

They are more expensive than HSS tools but have a longer tool life and offer improved productivity.

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.

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

Impact rupture strength
(Shock resistance)
Resilience
(Tool springs back to shape
under vibration.)

Tool material

High carbon steel
High speed steel
Cast alloys
Cemented carbide
Ceramics
Diamonds

Increase in

Initial cost of tool.
Red hardness temps.
Wear resistance.
Brittleness.
Cutting speeds used.
Tool life between
sharpenings.
Tool chipping with vibration.

Tool life and quality of a cutting tool material

Objectives: At the end of this lesson you shall be able to
- **explain tool life index equation**
- **state the quality of cutting tool material**
- **determine the maximum cutting speed for a given tool.**

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 1 indicates tool life index of various cutting tool.

Table 1

Tool life index

Material and conditions	Tool material	n
3 ½% nickel steel	Cemented carbide	0.2
3 ½% nickel steel (roughing)	Highspeed steel	0.14
3 ½% nickel steel (finishing)	Highspeed steel	0.125
High carbon, high chromium die steel	Cemented carbide	0.15
High carbon steel	High speed steel	0.2
Medium carbon steel	High speed steel	0.15
Mild steel	High speed steel	0.125
Cast iron	Cemented carbide	0.1

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.

The following example 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 life of 16 hours.

The value 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 (or) 8 hours

n = 0.125

Log C = Log V + n Log t1

= log 40 + (0.125 Log480)

= 1.6021 m/min + (0.125 x 2.681) min

= 1.6021 + 0.3351

= 1.9372

(ii) Determine Vmax for revised conditions

Vmax

Where t_2 = 960 min (or) 16 hours

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

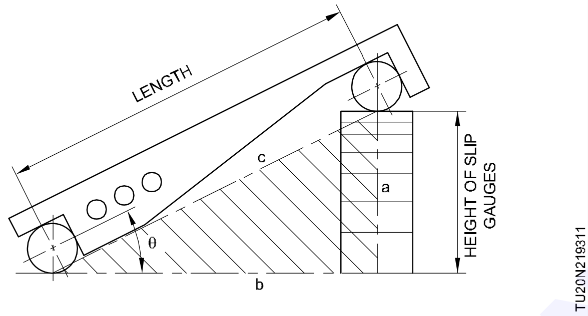
Checking of taper with Sine bar and roller

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

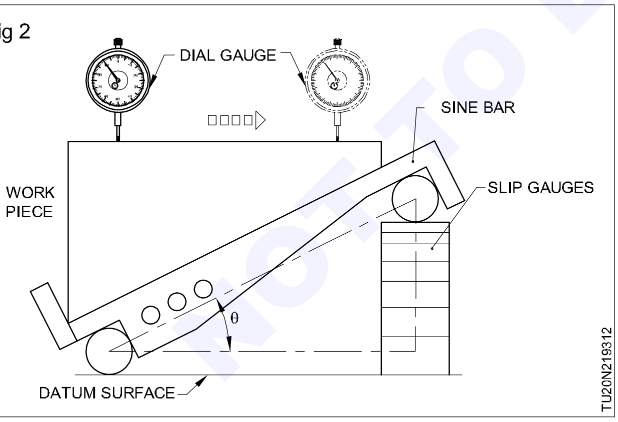


Checking the corrrectness of a known angle

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.

Examples

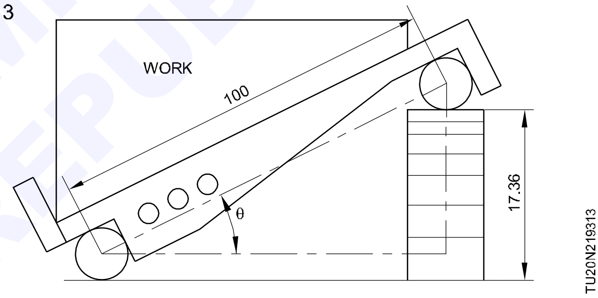
1 What will be the angle of the workpiece is the slip gauge pack height is 17.36 mm and the size of the sine bar used is 100 mm? (Fig 3)

$$\text{Sin }\theta\frac{a}{c} = \frac{17.36}{100}$$

$= 0.1736$

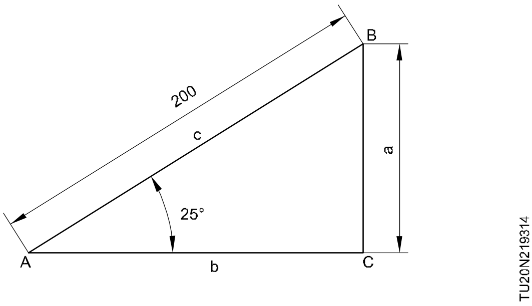
$\theta = 10°$

Fig 3



2 To determine the height of slip gauges for an angle of 25 ° using a sine bar of 200 mm long. (Fig 4)

Fig 4



$$\text{Sin }\theta\frac{a}{c}$$

$\theta = 25$

$a = \text{C sin e}\theta$

$= 200 \times 0.4226$

$= 84.52$ mm

The height of the slip required is 84.52 mm.

Note

The value of sine q 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.

3 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 5)

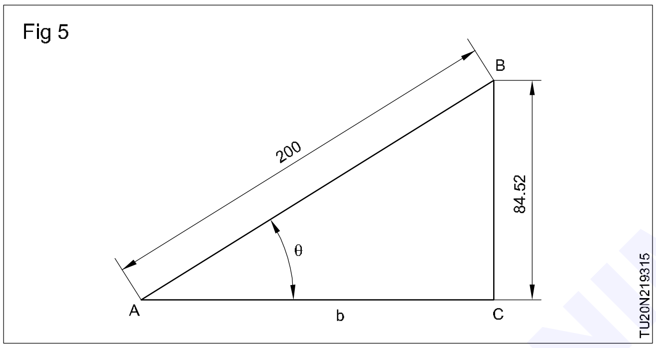
$$\text{Sin } \theta \frac{a}{c} = \frac{84.52}{200}$$

$$\text{Sin } \theta = 0.4226$$

The value of sine of the angle is 0.4226

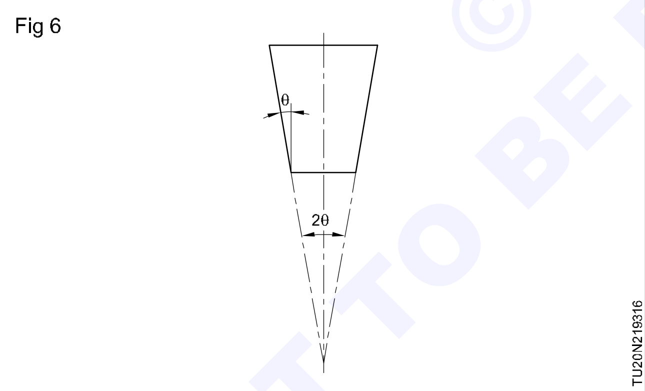
the angle = 25°

Fig 5



Angle of the taper Fig 6.

Fig 6



Small end diameter Fig 7.

Large end diameter Fig 7.

Check the included angle of the taper - 20

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 plate or the marking table. Two precision rollers are then placed at the smaller end resting on the datum surface and contacting the workpeice Fig 8.

Fig 7

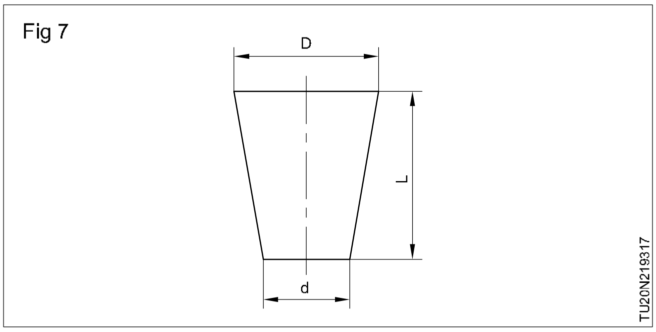
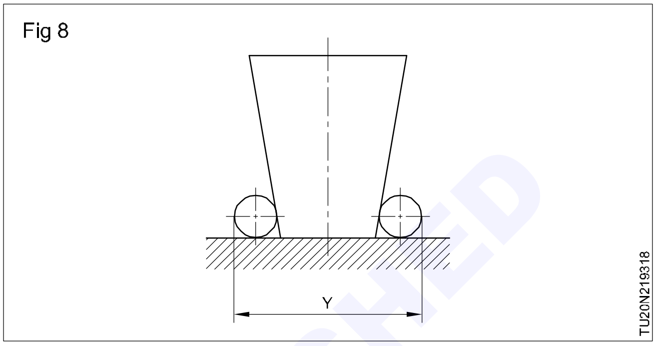


Fig 8

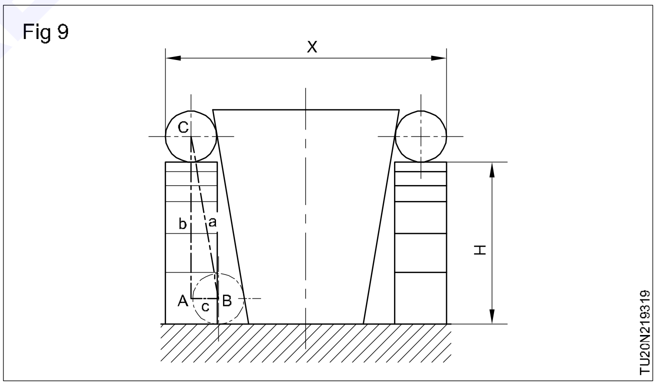


A method used for checking the dimensions of the tapered components is by using precision rollers or balls along with the slip gauges. Using this method the following elements of the tapers can be checked.

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 9.

Fig 9



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

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

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

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

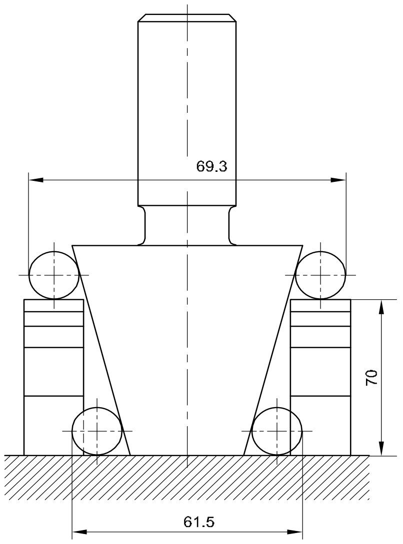
Where X is the measurement over the rollers placed on the slip gauge height, Y is the meaurement over the rollers at the smaller end and H is the slip gauge height.

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

Example

Calculate the included angle of the tapered component shown in Fig 10.

Fig 10



The measurement

X = 69.3 mm

Y = 61.5 mm

Height = 70mm

Tan θ = (69.3) - (61.5) / 2 x 70

= 7.8 / 2x70

$$= \frac{3.9}{70} = \frac{0.39}{7} = 0.0557$$

Referring to the log table under Natural Tangents we fine q = 3° 11’.

Hence included angle of the taper

2θ = 3°11’ x 2 = 6°22’

2θ = 6°22’

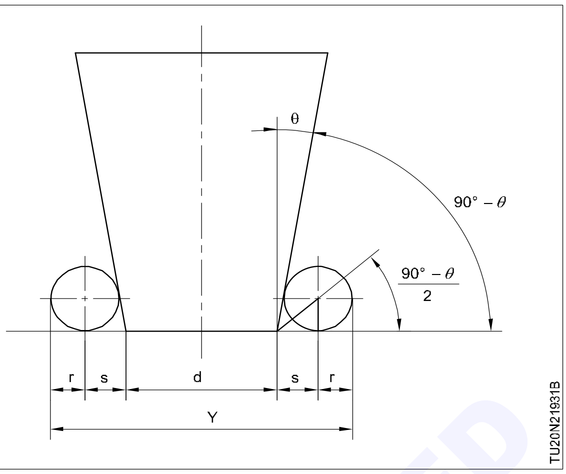
Diameters at any position of tapered components can be determined when the angle of taper is known.

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

Small end diameter d Fig 11.

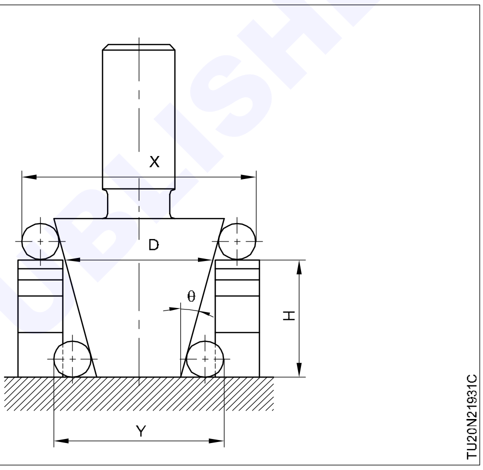
Large end diameter D Fig 11.

Fig 11



Determining small end diameter (Fig 12)

Fig 12



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

Calculating S Fig 13

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

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

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

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

Example (Fig 13

0 = 3° 11”

Y = 61.5 mm

r = (radius of roller) 6 mm

$$\text{Then d} = 61.5 - 12\left[\text{Cot}\left\{\frac{90-3°11"}{2}\right\}\right]+1$$

= 61.5 - 12 (1.0570+1)

= 61.5 - 12 x 2.0570

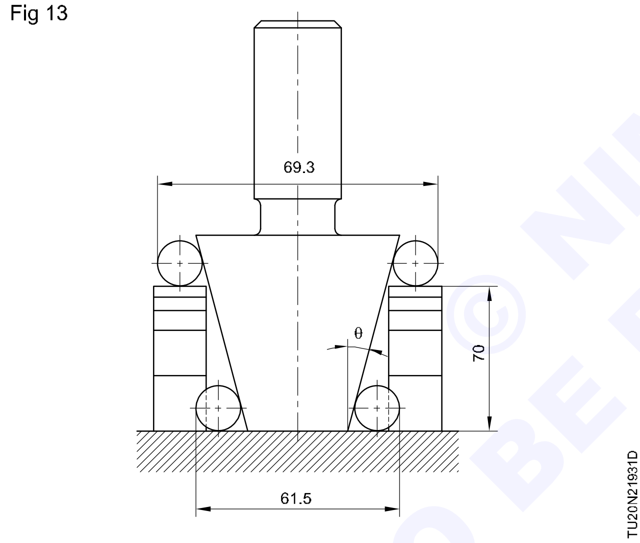
= 61.5 - 24.6840 = 36.3160 mm

Determining the large diameter of taper 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’.

H = X-2 (S+r)

Fig 13



Example Fig 14

q = 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)

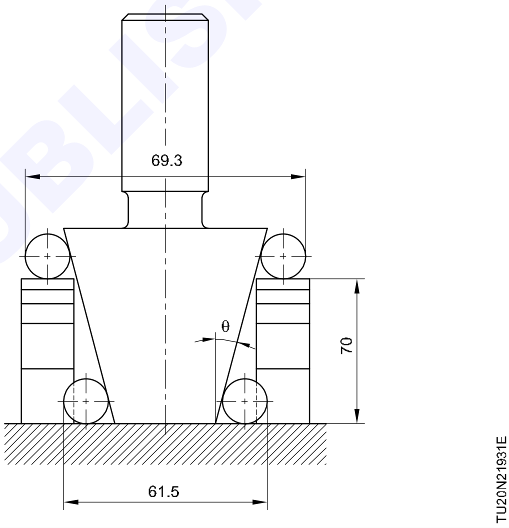
= 69.3 - 24.6840 = 44.6160 mm

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 x Tan q

Fig 14



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

Objectives: At the end of this lesson you shall be able to
- **explain, cutting speed and feed**
- **state the factors governing speed and feed**
- **read and select the recommended cutting speed for different materials from the chart.**

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

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

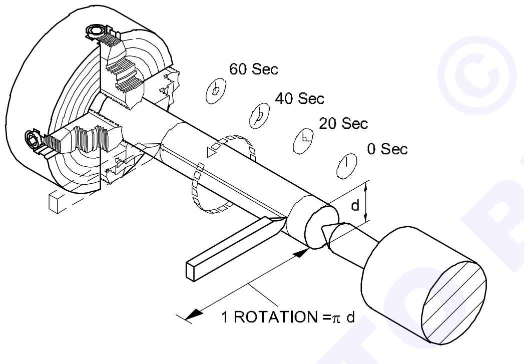
Where V = cutting speed in metre/min

π = 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)

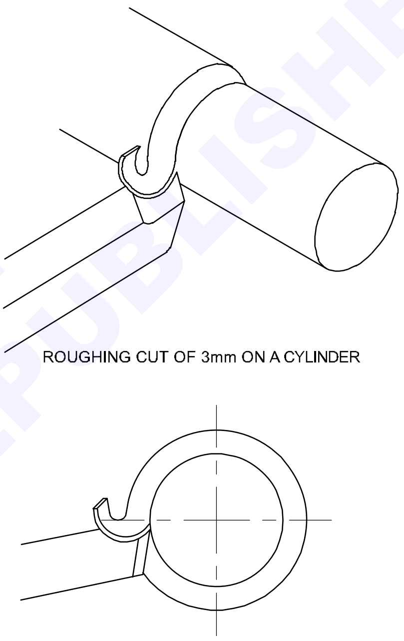
Example

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

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

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

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

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.

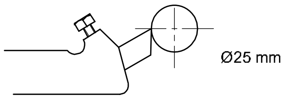
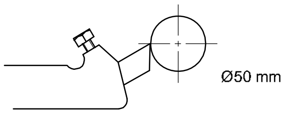
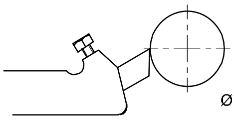
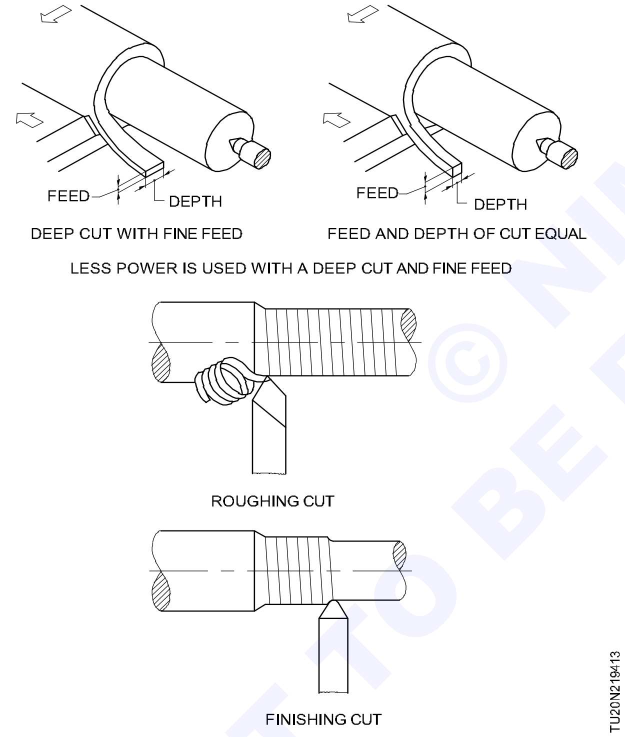
Cutting speed 120m/min	Length of metal passing cutting tool in I revolution	Calculated r.p.m. of spindle
	78.56 mm	1528
	157.12 mm	756
	235.68 mm	509.3

Fig 3



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

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

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

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

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

$$= 238.6 \text{ r.p.m.}$$

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

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

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

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

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

$$= 119.3 \text{ r.p.m.}$$

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 m/minute.

Ø of job = 40 mm

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

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

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

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

$$= 731.8 \text{ r.p.m.}$$

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.

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/rev

n = 1

N = 238.6 r.p.m.

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

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

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: $\ell \div s$.

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

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

Example 1

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.

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

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 minutes

Example 2

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_1 = Rpm of 120 m/min.

$N_1 = \frac{CS \times 100}{\pi \times D} = \frac{120 \times 1000}{3.14 \times 80}$ = 477.7 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 = for 2nd cut = $\frac{CS \times 1000}{\pi \times d}$

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

T_2 = Second rough cut machining time

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

= 11 minutes

N_3 = Rpm of 140m/min.

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

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

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

= 8.64 minutes

Total machining time = $T_1 + T_2 + T_3$

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

= 34.31 minutes.

Classification of tungsten carbide tipped tools

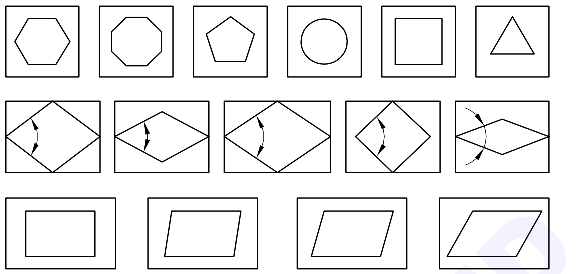
Objectives: At the end of this lesson you shall be able to
- **state what is tungsten carbide tools**
- **classify the tungsten carbide tools**
- **list the types of carbide tools and its specification.**

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

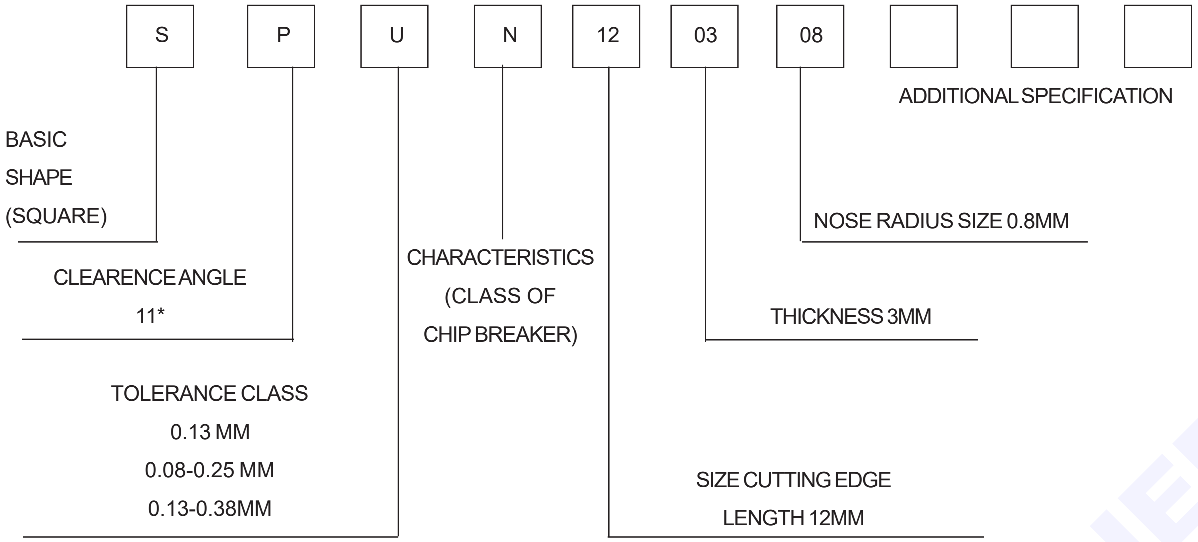
Tolerance class	Insert thickness ('s')	Inscribed circle ('d' dia).	Control dimension ('m')
A	0.025 mm	0.025 mm	0.005 mm
C	0.025 mm	0.025 mm	0.013 mm
E	0.025 mm	0.025 mm	0.025 mm
G	0.13 mm	0.025 mm	0.025 mm
H	0.025 mm	0.013 mm	0.013 mm
J	0.025 mm	*0.05–0.13mm	0.005 mm
K	0.025 mm	*0.05–0.13mm	0.013 mm
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° clearance 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 Left hand cutting N Right and 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?

$$V = \frac{d \times \pi \times n}{1000}, n = \frac{V \times 1000}{d \times \pi} = \frac{160 \times 1000}{80 \times 3.14} = 636.6 \text{r.p.m}$$

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

Refer to Table for the cutting speeds.

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

Table

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

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

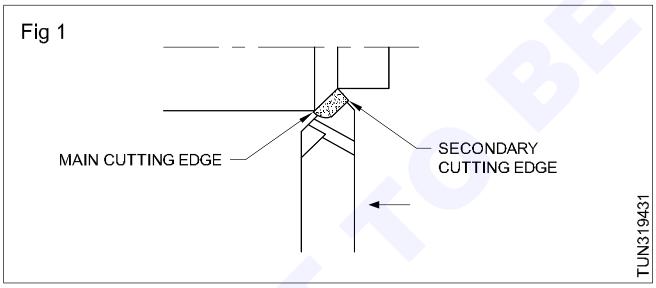
Cemented carbide tools are available as brazed tipped tools and throw away tips held in specially designed tool holders.

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

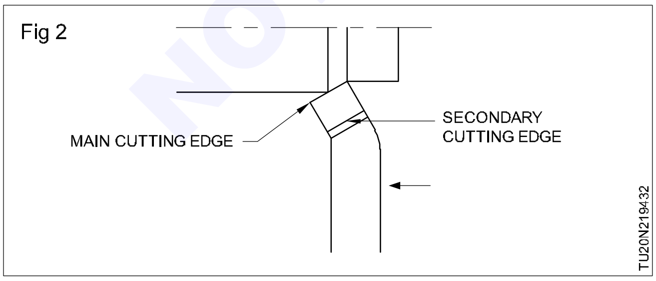
ISO 1 Straight turning tool

Fig 1



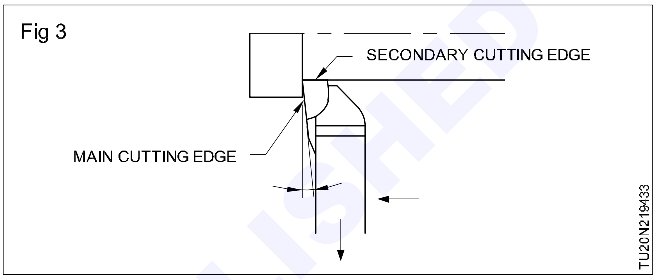
ISO 2 Cranked turning tool

Fig 2



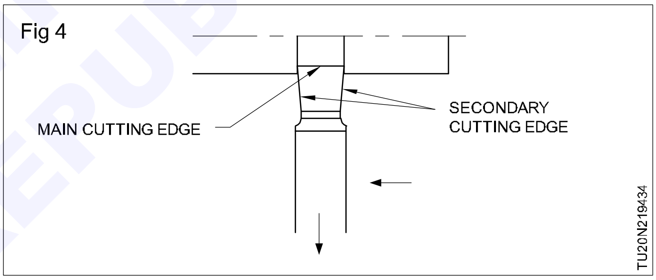
ISO 3 Offset facing tool

Fig 3



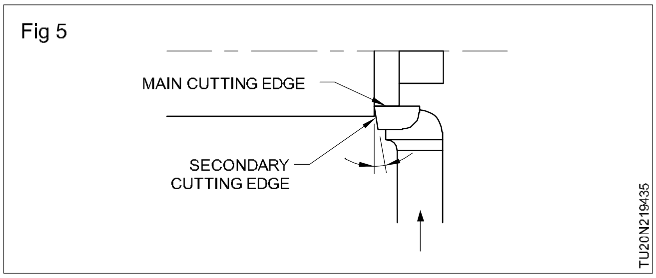
ISO 4 Wide nose square turning tool

Fig 4



ISO 5 Offset turning and facing tool

Fig 5



ISO 6 Offset side cutting tool (Offset knife tool)

Fig 6

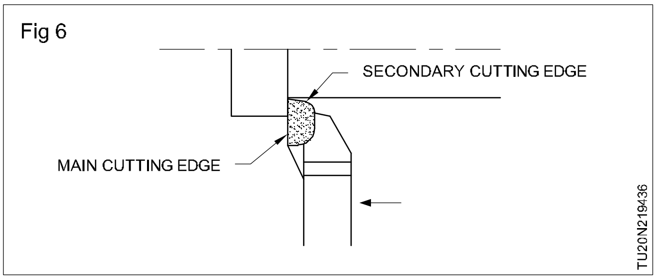
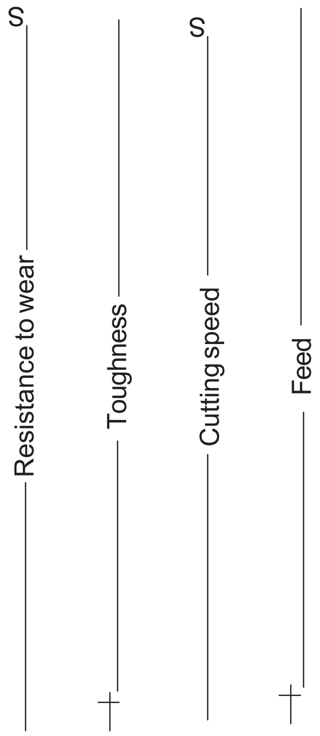


TABLE 1

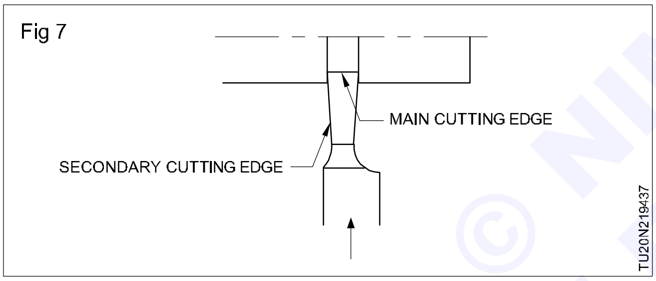
Classification of carbide tips according to their range of application. (IS: 2428 - 1964)

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		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.

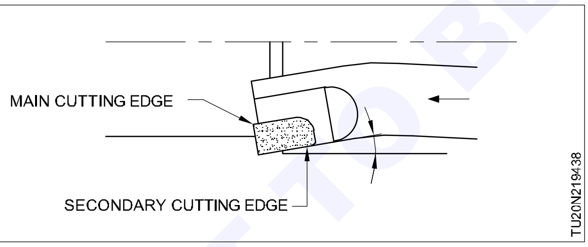
ISO 7 Recessing tool (parting tool)

Fig 7



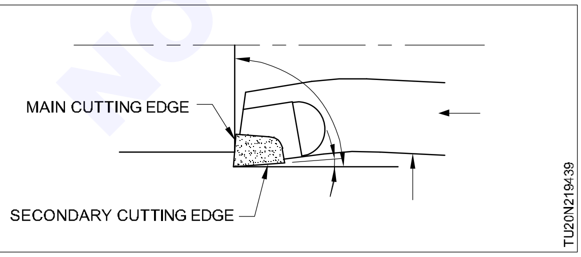
ISO 8 Boring tool

Fig 8



ISO 9 Corner boring tool (finishing)

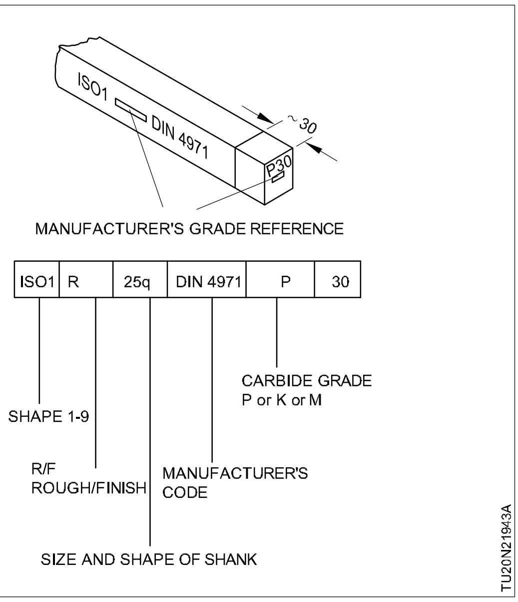
Fig 9



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



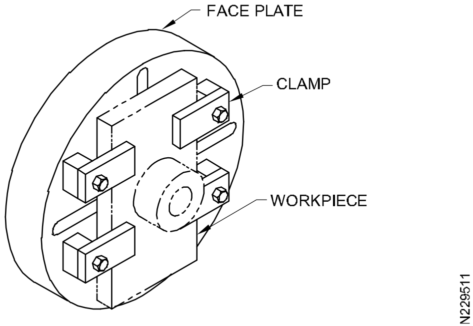
Face plate - accessories used on face plate, angle plate

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

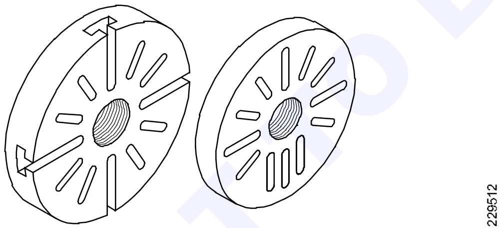


WORK CLAMPED DIRECTLY TO FACE-PLATE

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

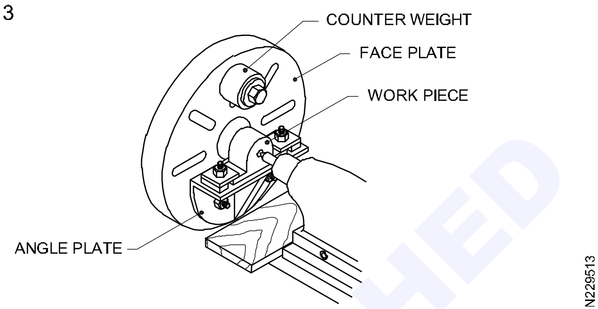
Since the type and shape of workpieces vary greatly, a large number of face-plate accessories such as bolts, clamps, parallels, step blocks and counterweights are used to set up and fasten the work to the face-plate.

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)

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

Fig 3



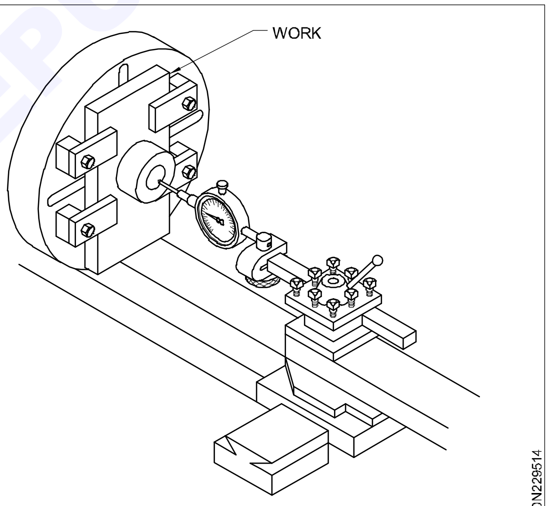
WORK HELD ON AN ANGLE PLATE

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.

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

Fig 4



TRUING A BORE WITH A DIAL INDICATOR

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.**

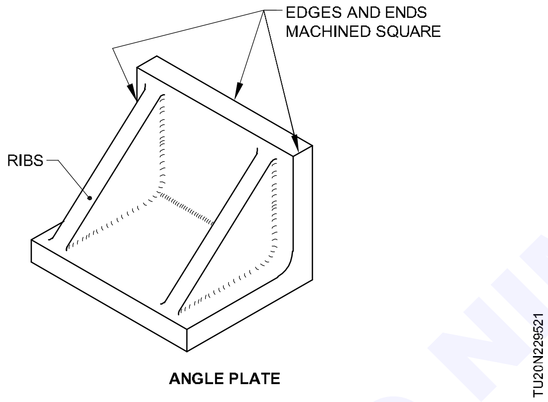
Constructional features

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

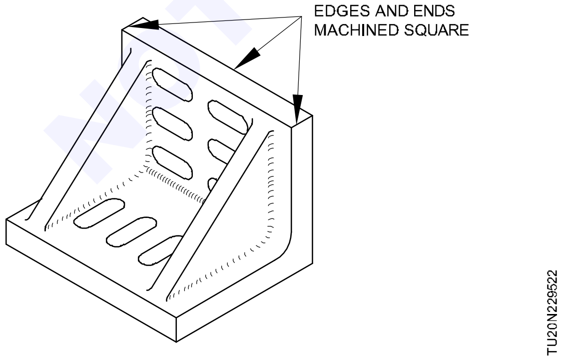


Among the three types of angle plates normally used. It plain solid angle plate is the most common. It has the two plane surfaces perfectly machined at 90° to each other such angle plates are suitable for supporting workpiece during layout work. They are comparatively smaller size.

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

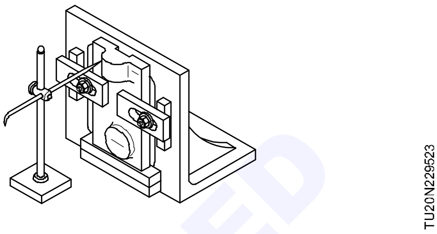
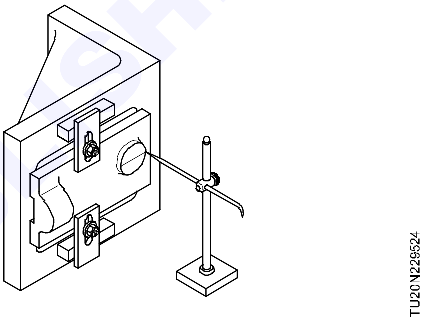


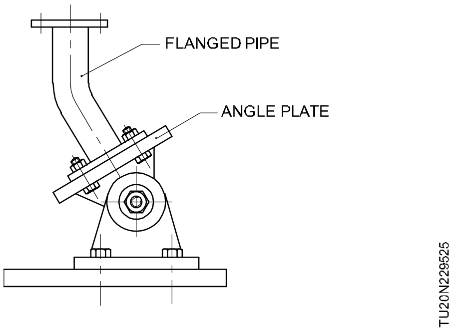
Fig 4



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.

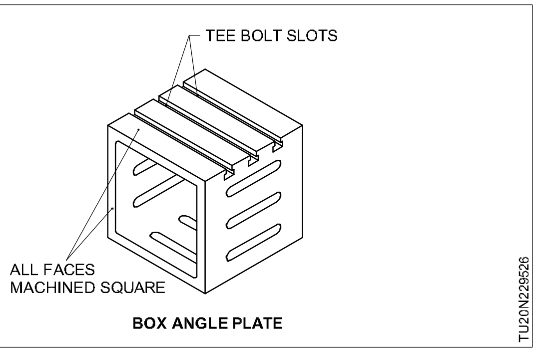
Fig 5



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



BOX ANGLE PLATE

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.

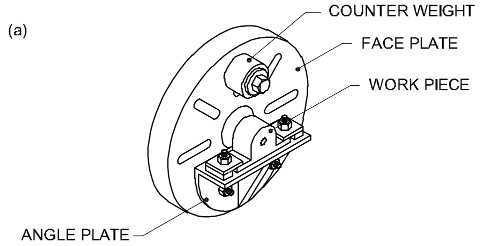
To check the balance, first disconnect the spindle drive. (Figs 1a and 1b)

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

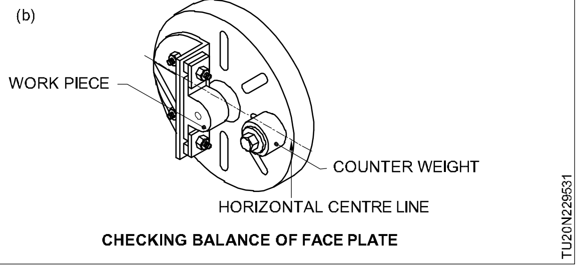
If the weight falls below the horizontal line, it is too heavy or too far out from the centre of the plate. If it rises, it is too light or too close to the centre. In either case an adjustment must be made until the balance weight remains in the horizontal position.

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 1



BALANCING WORK ON A FACE PLATE



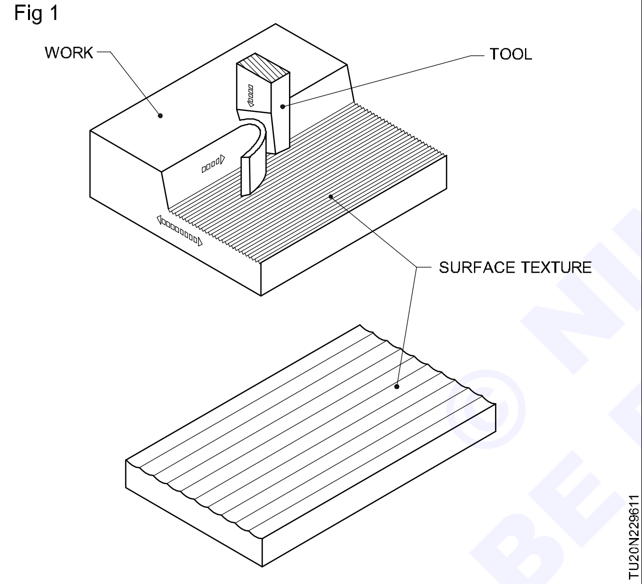
CHECKING BALANCE OF FACE PLATE

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.

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.

Fig 2

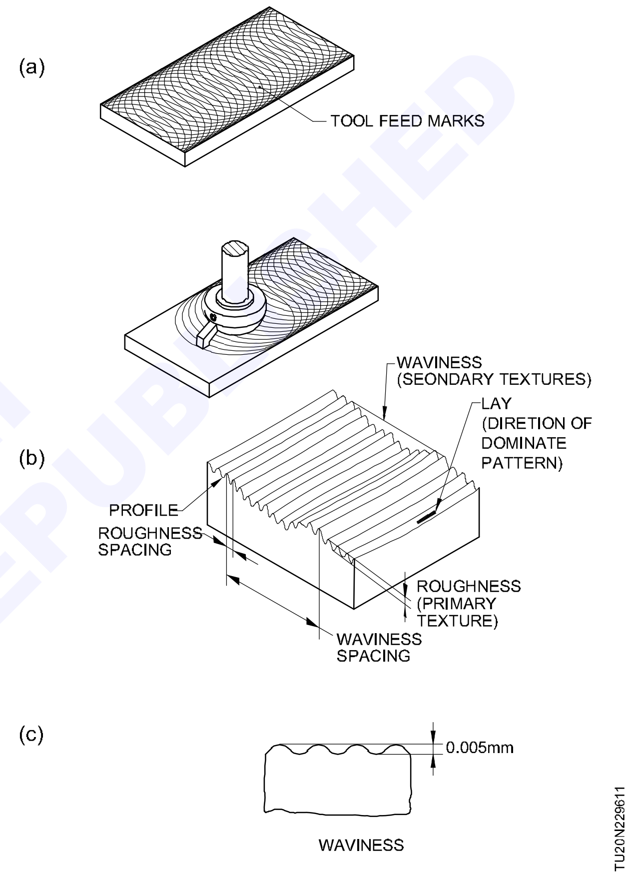
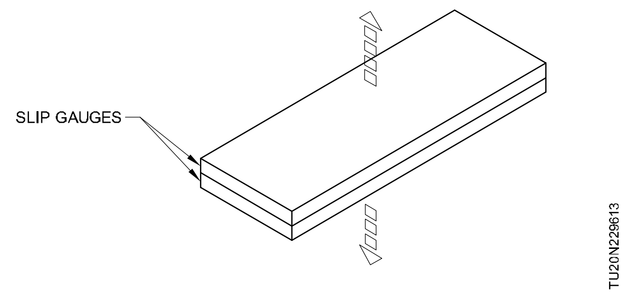


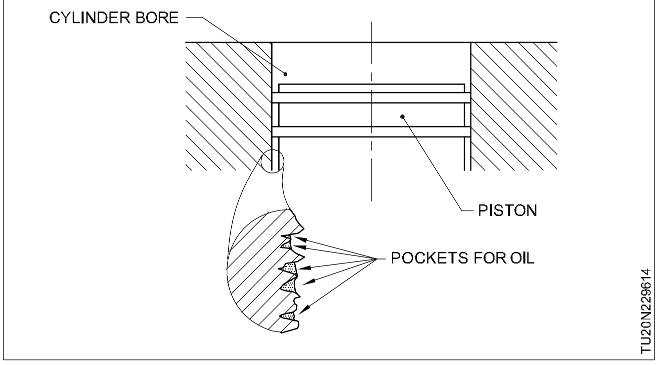
Fig 3



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.

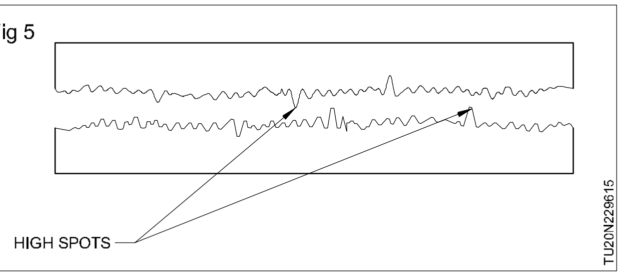
For sliding surfaces the quality of surface texture is very important.

Fig 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.

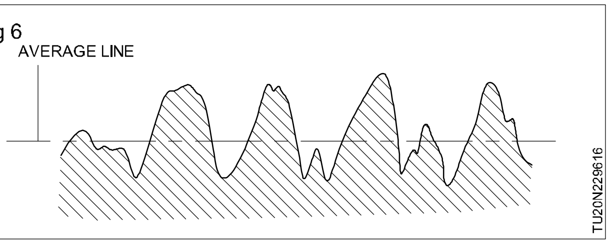
The surface texture quality can be expressed and assessed numerically.

'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.

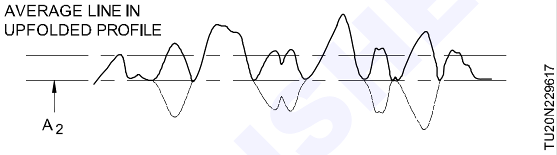
Fig 6



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



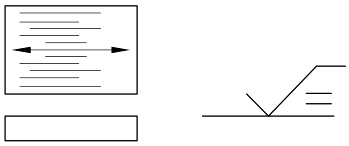
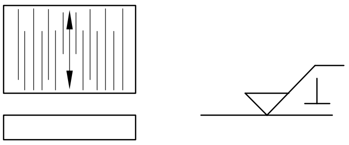
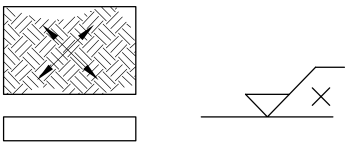
The distance between the two lines is the 'Ra' value of the surface.

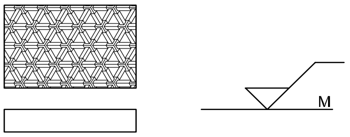
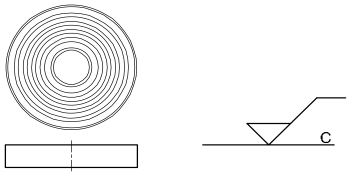
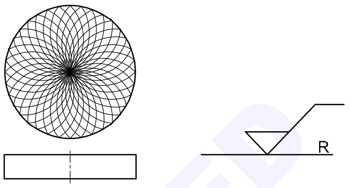
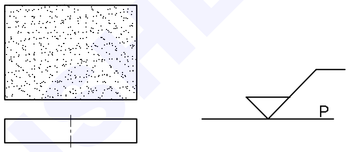
The 'Ra' value is expressed in terms of micrometre (0.000001) or (m); this also can be indicated in the corresponding roughness grade number, ranging from N_1 to N_{12}.

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.	

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.

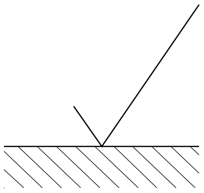
Table 1

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:

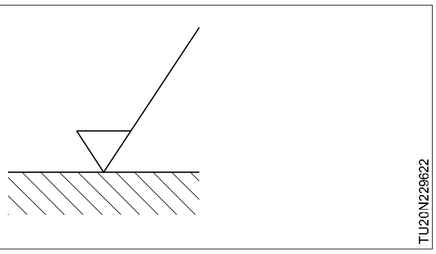
1 The basic symbol consists of two legs of unequal length inclined at approximately 60°. (Fig 1)

Fig 1



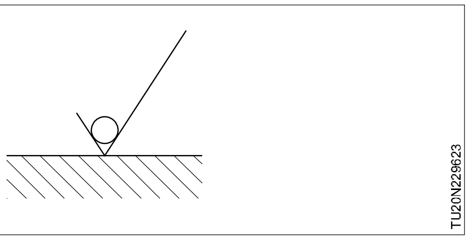
2 If the material removal by machining is required, a bar is added to the basic symbol. (Fig 2)

Fig 2



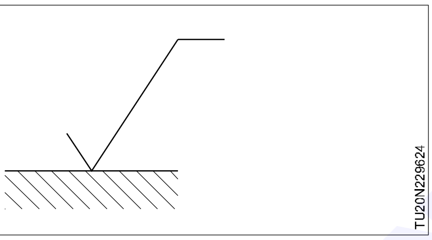
3 If the material removal is not permitted, a circle is added to the basic symbol. (Fig 3)

Fig 3



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

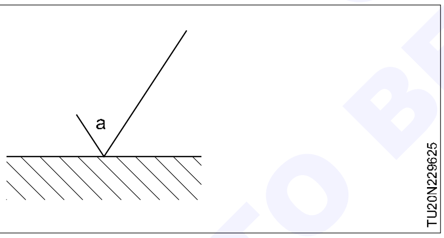
Fig 4



5 **Indication of surface roughness:**

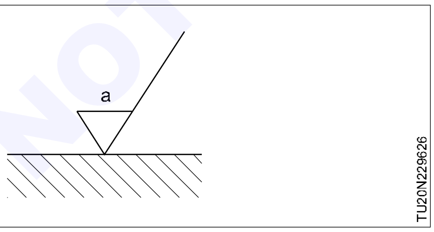
a Surface roughness obtained by any production method. (Fig 5)

Fig 5



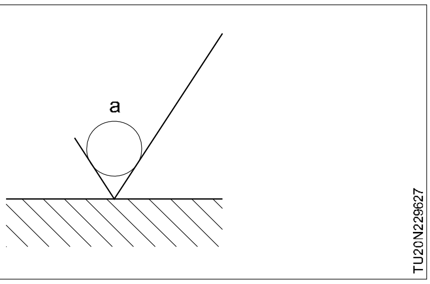
b Surface roughness obtained by removal of material by matching. (Fig 6)

Fig 6



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

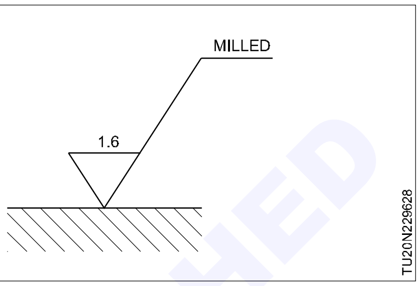
Fig 7



6 Indication of special surface roughness characteristics:

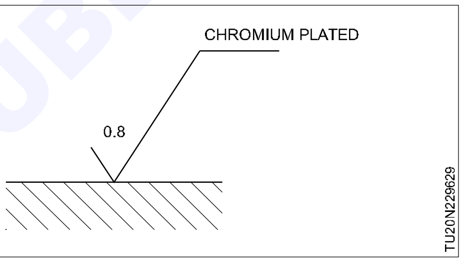
a Indicating the production method.(Fig 8)

Fig 8



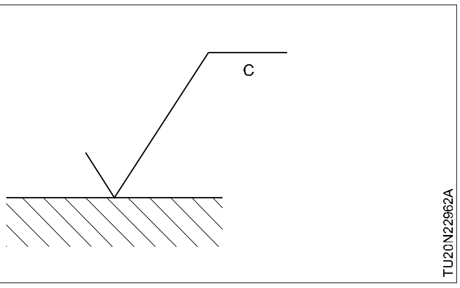
b Indicating the surface treatment or coating. Unless otherwise stated, the numerical value of the roughness, applies to the surface roughness after treatment of coating.(Fig 9)

Fig 9



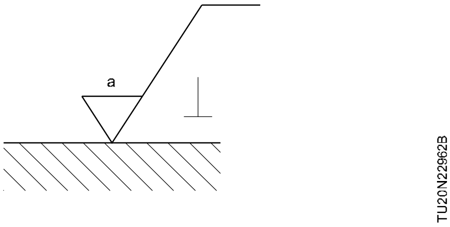
c Indicating the sampling length.(Fig 10)

Fig 10



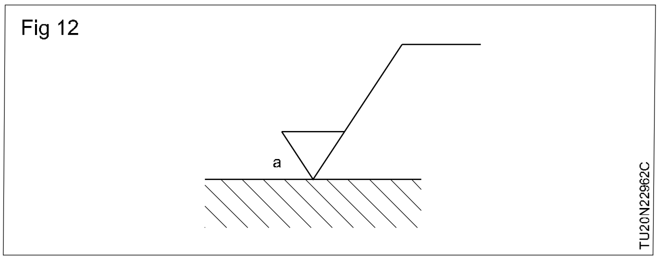
d Direction of lay, surface pattern by the production method employed.(Fig 11)

Fig 11



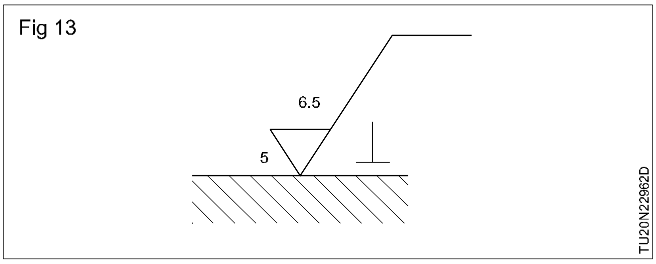
e Indication of allowance in mm.(Fig 12)

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

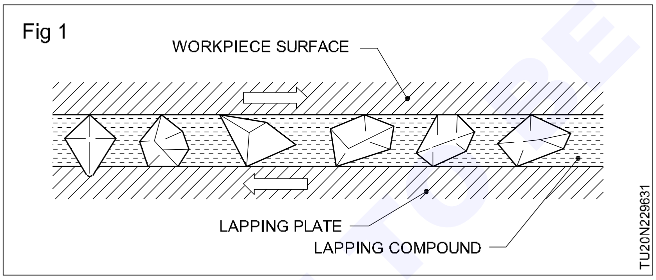
– refines surface finish

– 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.

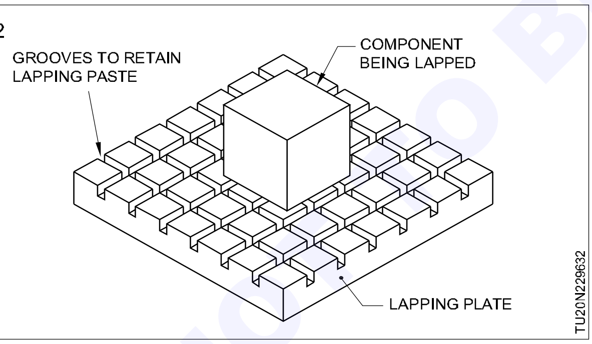
Abrasive of varying grain sizes from 50 to 800 are used for lapping, depending on the surface finish required on the component.

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.

Honing tool

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

Fig 3

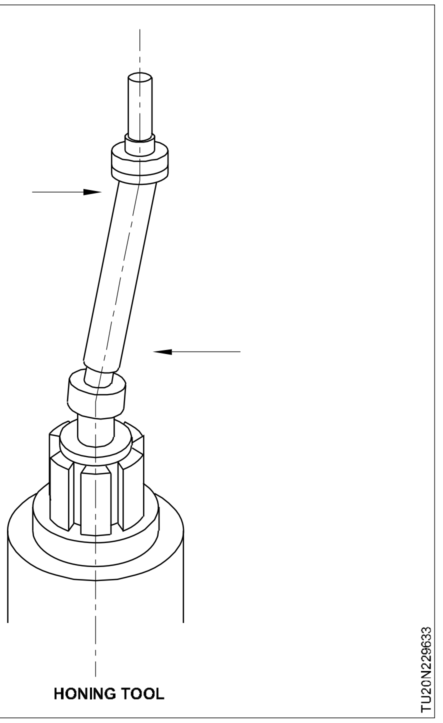
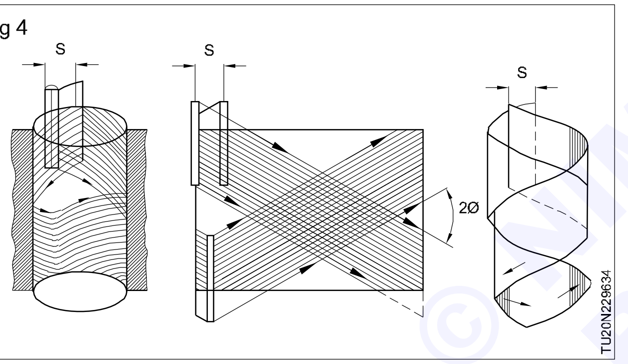


Fig 4



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.

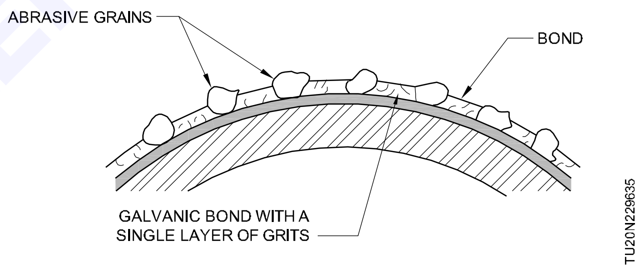
Super abrasive honing stick with monolayer configuration, where a layer of CBN grits are attached to stick by a galvanically deposited metal layer, is typically found in single stroke honing application.

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. Sample maintenance schedule is list in Annexure I.

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 II)

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.

Annexure - I

Preventive Maintenance Programme

S.No	Name of Machines	Monthly Maintenance					Quarterly Maintenance					Semi-Annual Maintenance			Annual Maintenance					
		Check for loose bolts, screws, and fasteners.	Inspect the electrical wiring	Lubricate all moving parts and ways.	Inspect and replace, the drive belts or chains.	Calibrate and verify machine tool alignment.	cleaning of the machine and its components.	Check and adjust the tension of drive belts or chains.	Inspect and clean electrical components	Inspect and replace,, seals, gaskets, and O-rings.	Calibrate and verify machine tool alignment.	Inspect the condition of the coolant system	Check and adjust the spindle bearings,	Inspect the tool holders and replace worn-out parts.	Perform a comprehensive inspection	Replace all filters in the coolant system.	Lubricate and grease all components	Inspect and replace the electrical wiring and connectors.	Inspect and replace, if necessary, the machine's way wipers	Calibrate and verify machine tool alignment.
1	Lathe machine																			
2	Milling machine																			
3	Shaping machine																			
4	Planning machine																			
5	Drilling machine																			
6	Grinding machine																			
7	Slotting machine																			
8	Gear cutter																			
9	Power press																			

Preventive Maintenance Programme

Annexure - II

Name of the Machine :
Machine Number :
Model No. & Make :
Inspected by

Location of the machine :

Annexure II

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			

Maintenance Records

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

Total productive maintenance

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 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 maintenance (TPM)?

It can be considered as the medical science of machines. Total productive maintenance (TPM) is a maintenance program which invloves a newly defined 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 program 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

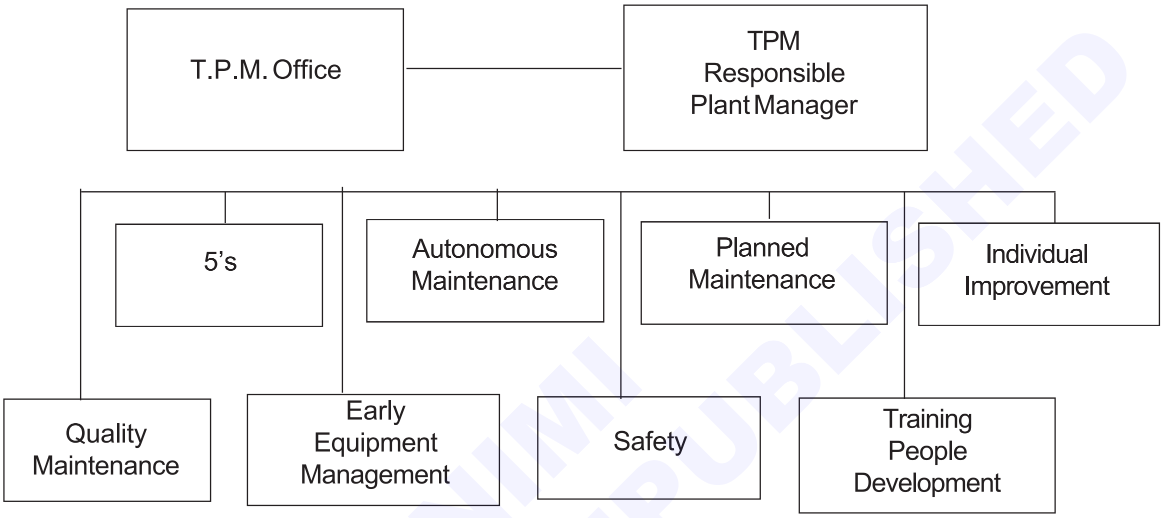
3 A long range outlook must be accepted as TPM may take a year or more to implement and is an on-going process. Changes in employee mind-set toward their job responsibilities must take place as well.

An Introduction to Total Productive Maintenance (TPM)

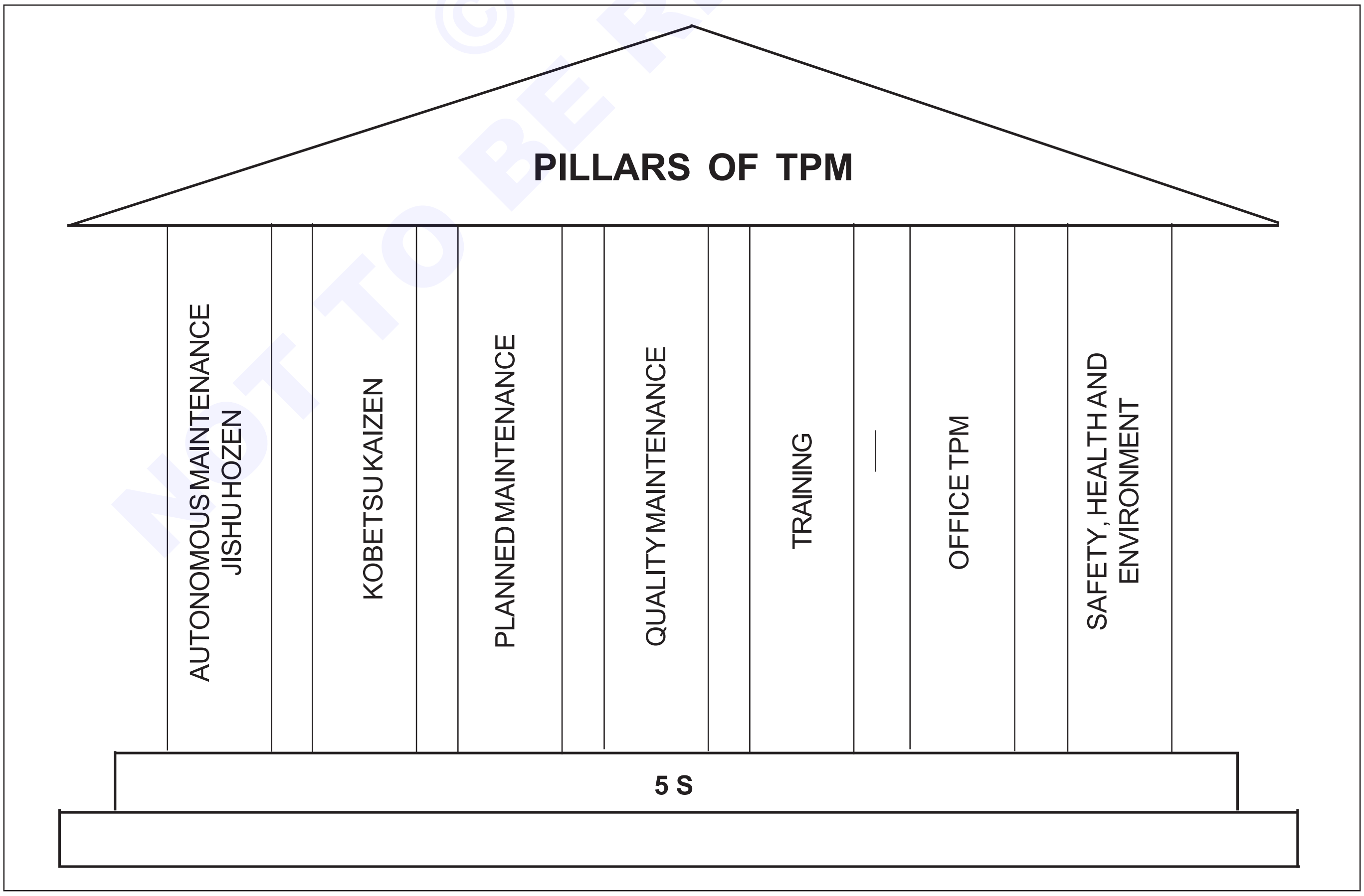
Motives of TPM	1 Adoptation of life cycle approach for imporving the overall performance of prodiction equipment. 2 Improving productivity by highly motivatd workers which is achieved by job enlargement. 3 The use of voluntary small group activities for identifying the cause of failure, possible plant 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 the manufacturing cost by 30%. 4 Satisfy the customers needs by 0% (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)



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 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.

Marking off and marking table

Objectives: At the end of this lesson you shall be able to
- **state what is marking off and witness mark**
- **brief the uses and construction of 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.

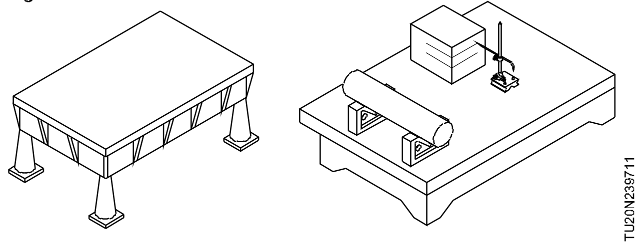
Witness marks

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 (Fig 1)

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

Fig 1



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.

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.

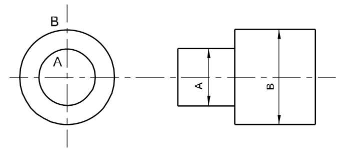
The surface of the marking table, made of cast iron, should be protected by applying a thin layer of oil.

Eccentric

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.**

Concentricity: When different diameters are turned in the same axis, it is said to be concentric turning. Fig 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)

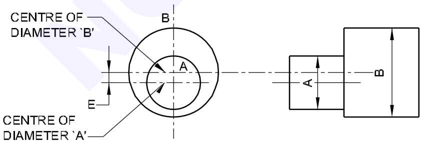
TU20N239721

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 2)

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)

TU20N239722

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.

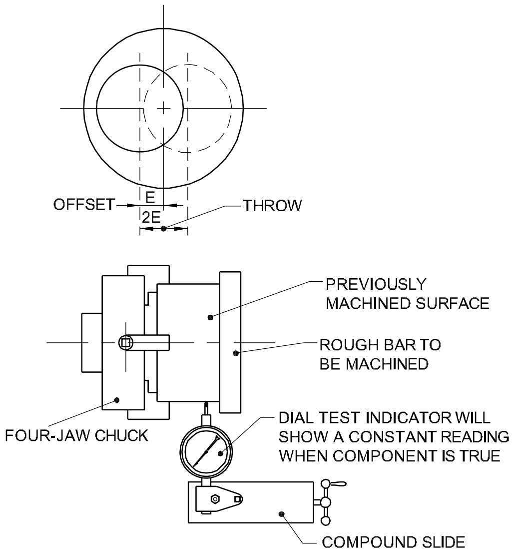
Method of identifying eccentricity: The eccentricity of a turned job is tested with the help of a dial test indicator. It is possible to test the offset of the turned job when the job is being held on a 4 jaw chuck.

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 4)

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.

Fig 3



TRUING-UP WITH THE DIAL TEST INDICATOR

TU20N239723

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 and its uses**
- **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

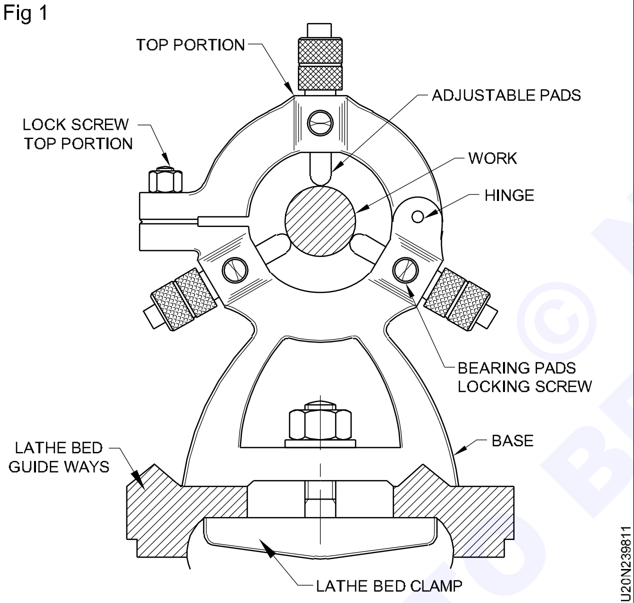
A steady rest is a lathe accessory used to give extra support for a long slender workpiece in addition to the centre support during turning.

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.

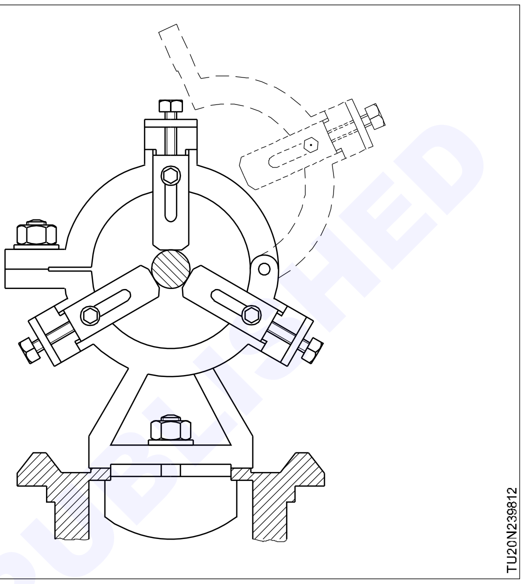
A fixed steady rest is fixed to the lathe bed and it is stationary. It gives support at one fixed place only.

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)

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.

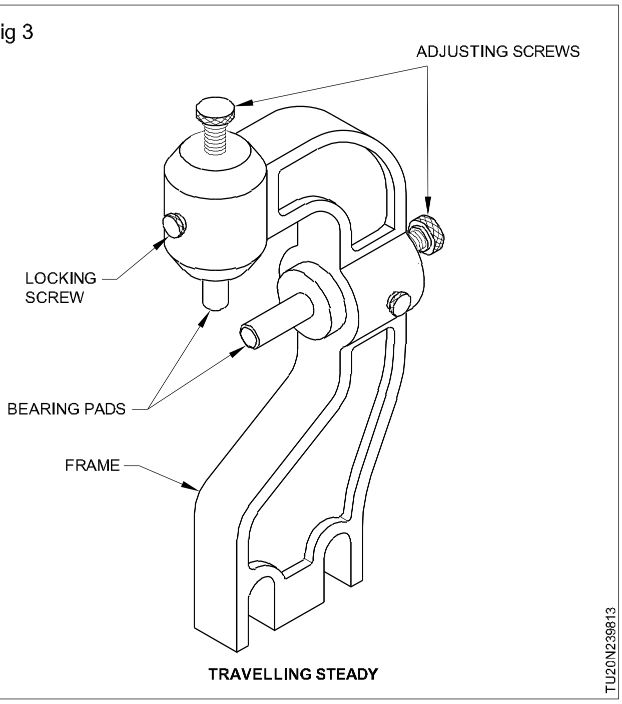
Fig 2



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.

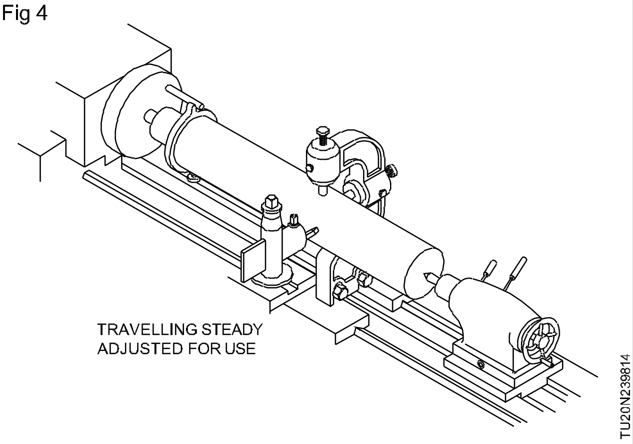
Fig 3



TRAVELLING STEADY

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

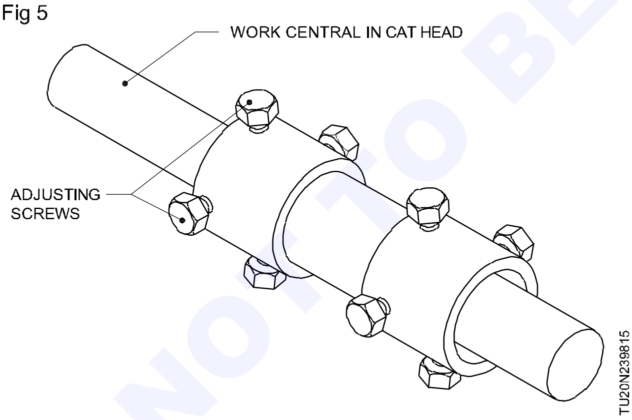
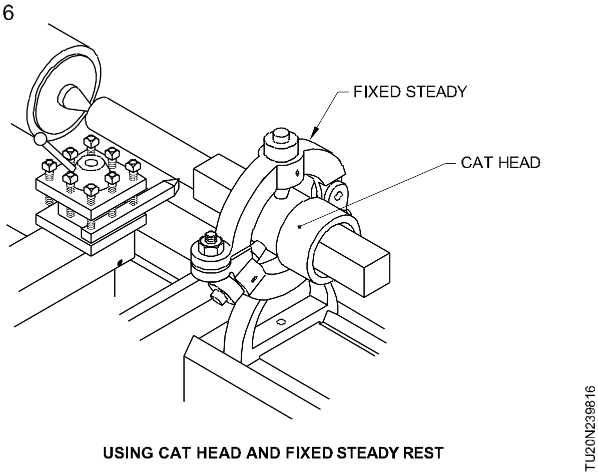
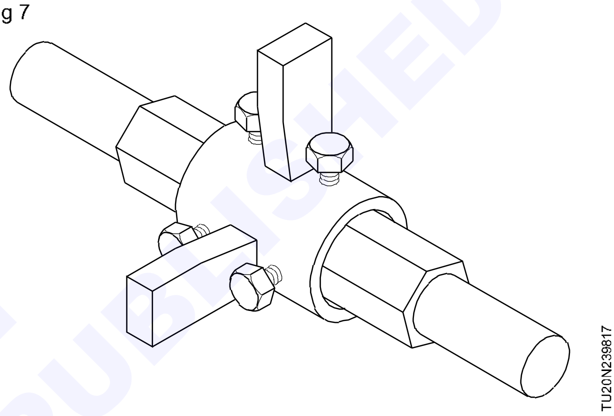


Fig 6



USING CAT HEAD AND FIXED STEADY REST

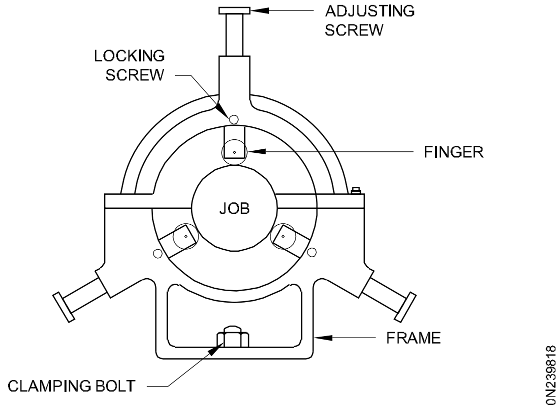
Fig 7



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

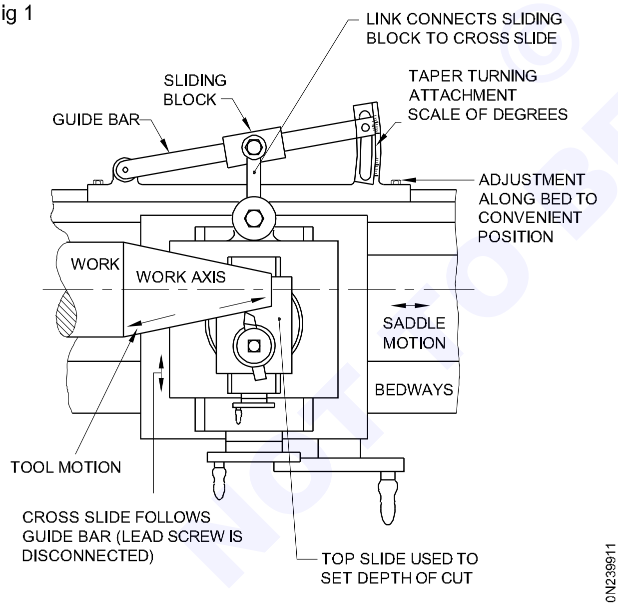
Objectives: At the end of this lesson you shall be able to
- **state the various operations performed in a centre lathe using different attachments**
- **state the features of the taper turning, milling and grinding attachments.**

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

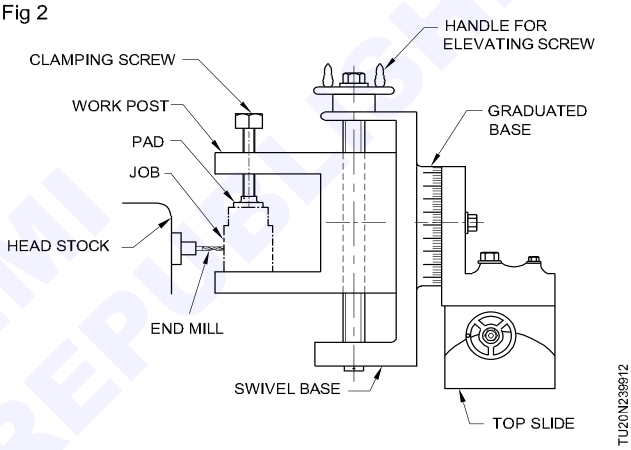
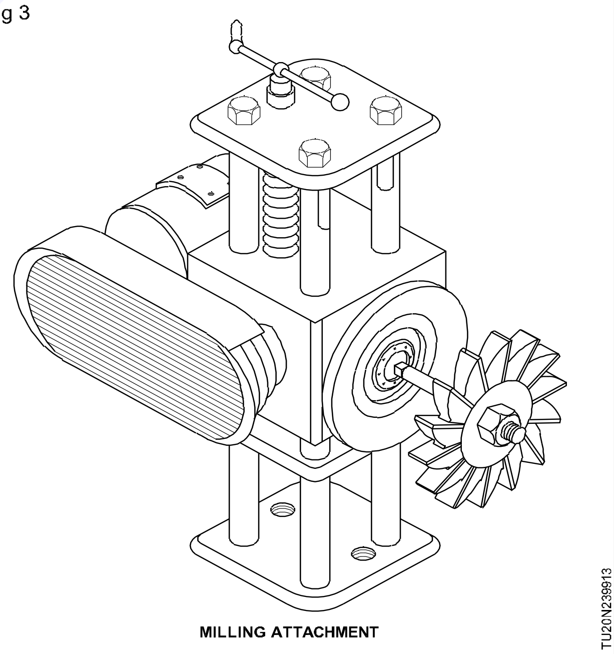


Fig 3



MILLING ATTACHMENT

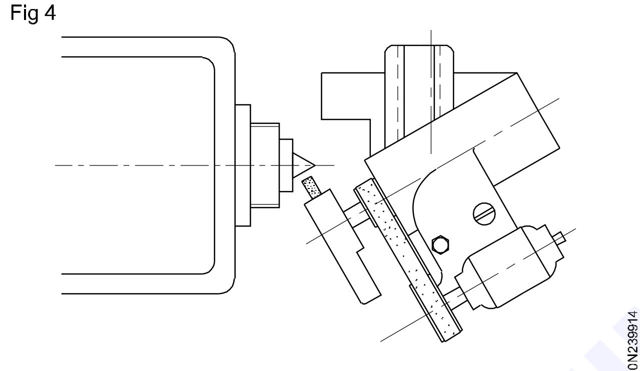
Grinding attachment

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

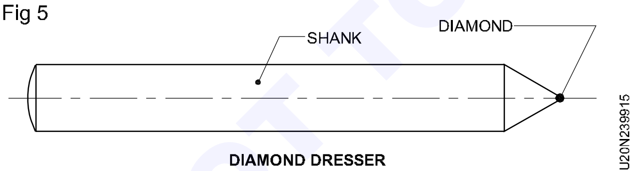


GRINDING A CENTER ON THE LATHE

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.

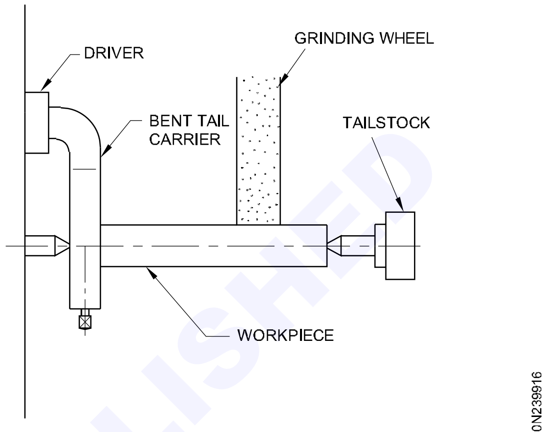
Grinding hardened steel parts

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)

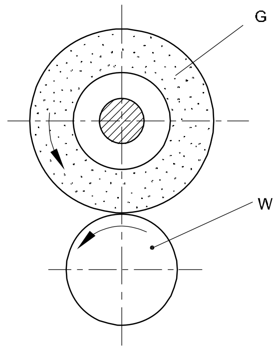
Fig 6



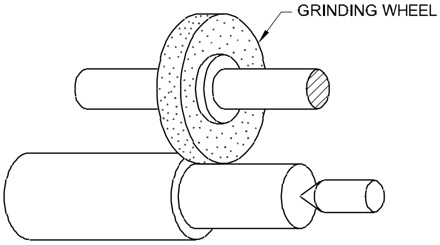
CLAMPING BETWEEN CENTERS

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

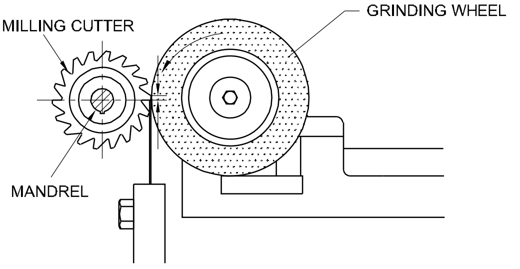
Selection of grinding wheel

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 gears 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)

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 1

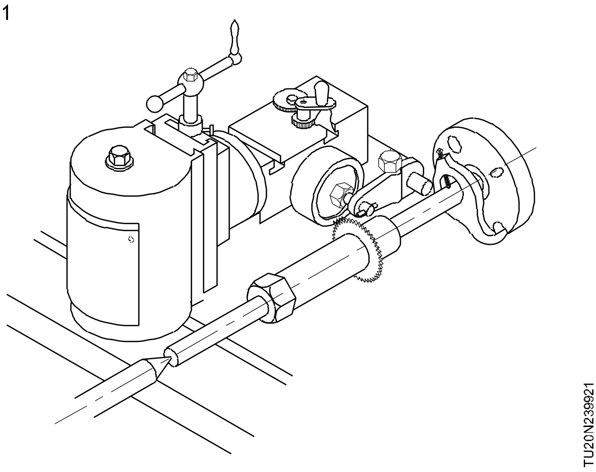
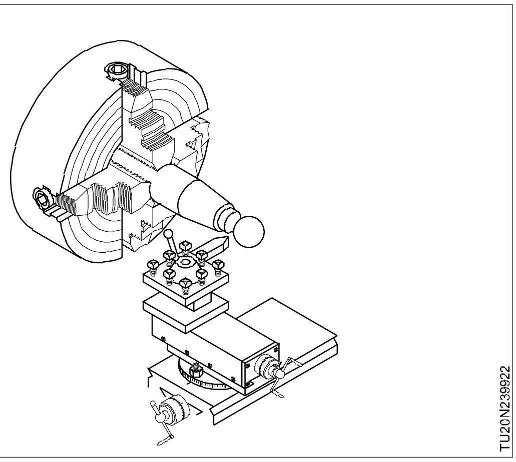


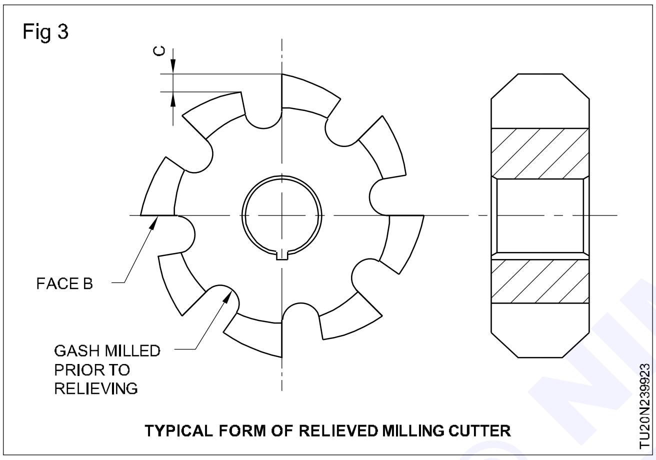
Fig 2



Relieving attachment

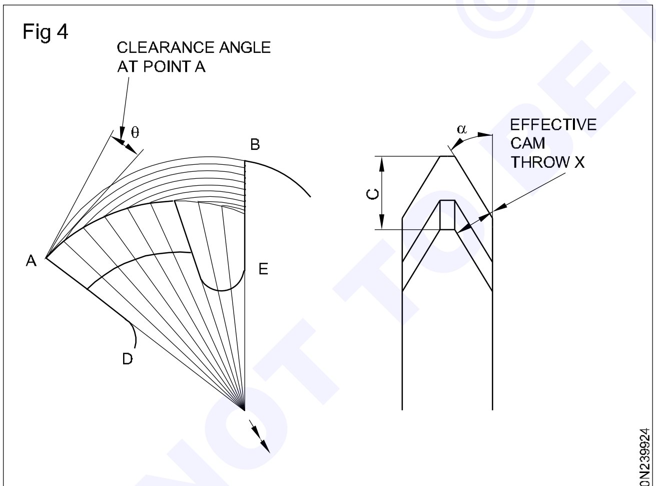
Figure 3 shows a form relieved milling cutter and Fig 4 exhibits the relieved portion in a cutter tooth.

Fig 3



TYPICAL FORM OF RELIEVED MILLING CUTTER

Fig 4



ENLARGED DETAIL OF A TOOTH

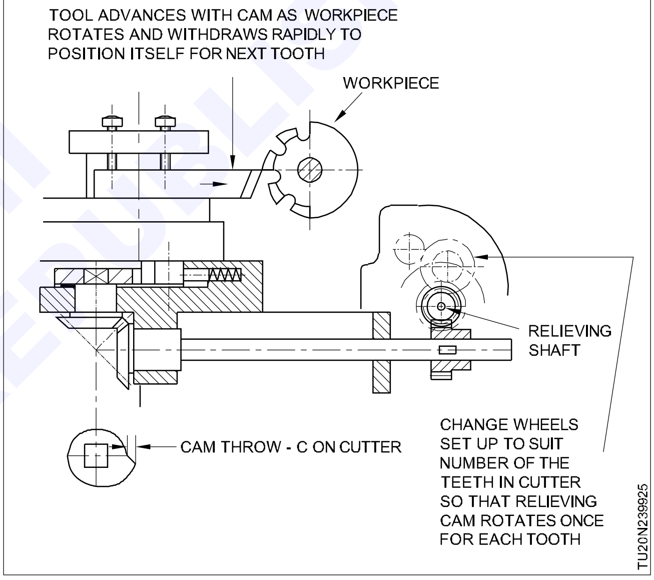
Relieving mechanism

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

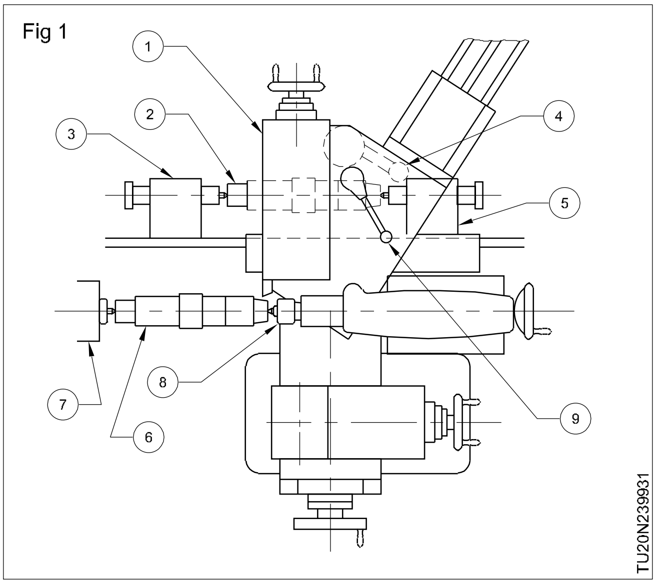


Copying attachment

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)

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

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.

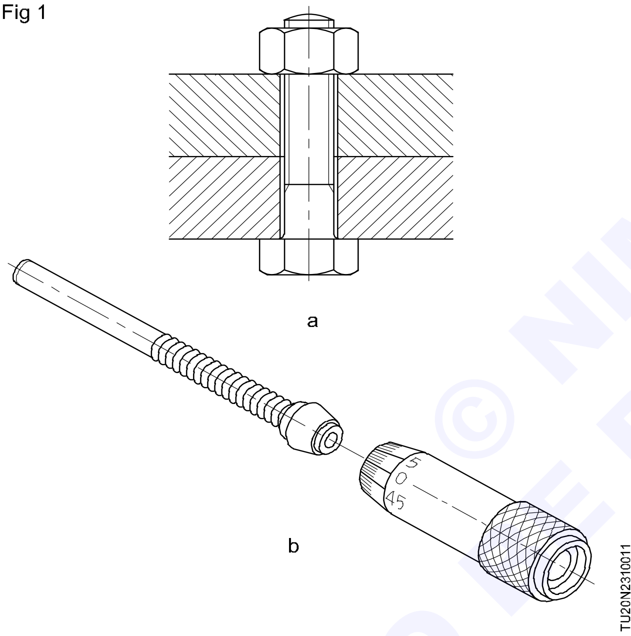
Various procedures of thread measurements

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.

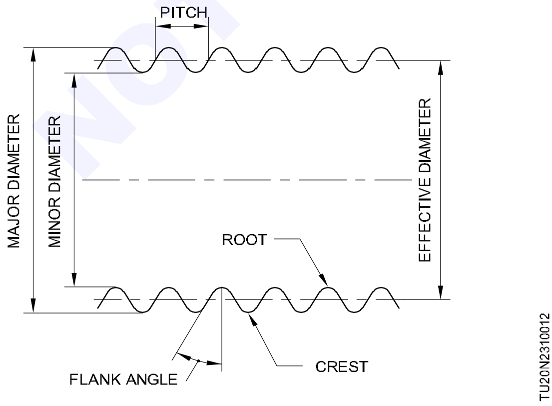
The accuracy requirement varies from a bolt used in structural work to threads of a fine measuring instrument. (Figs 1a and 1b)

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

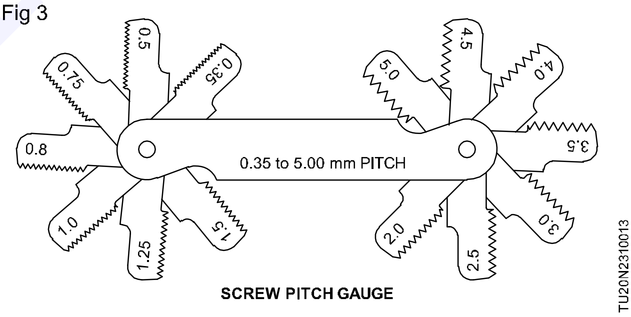
The above elements contribute to the strength and interchangeability of the threads.

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

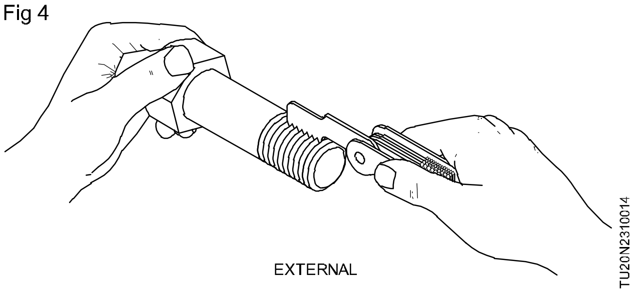
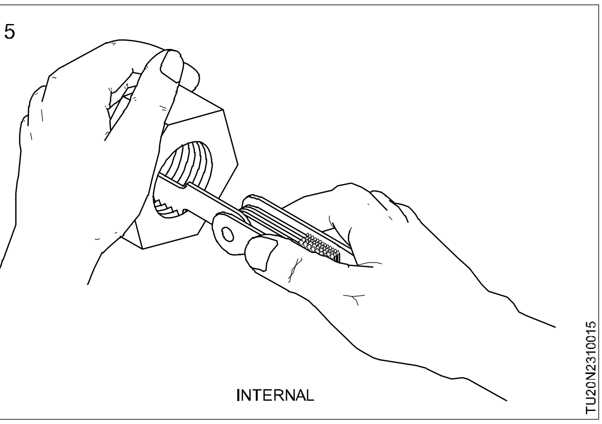


Fig 5

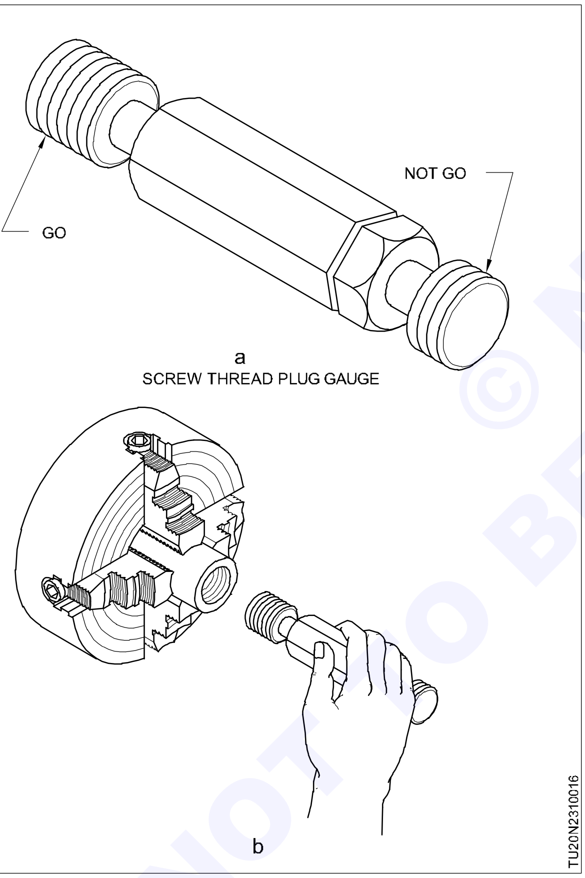


The decision about the correctness of the pitch is taken by comparing them by placing the appropriate blade on the screw. Each blade has an indication about the size of the screw and the pitch.

Example

Screw thread plug guage (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)

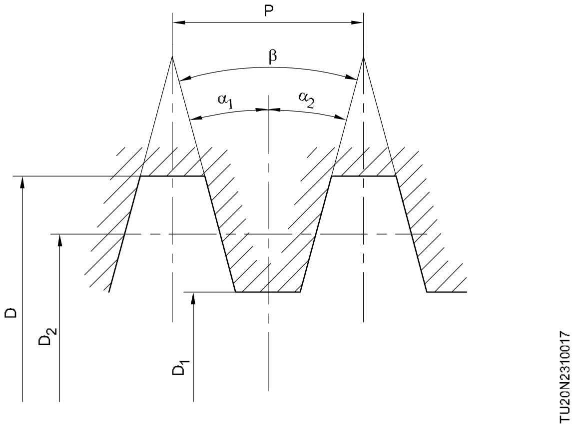
The pitch (P)

The major diameter (D)

The effective diameter(D_2)

The minor diameter (D_1)

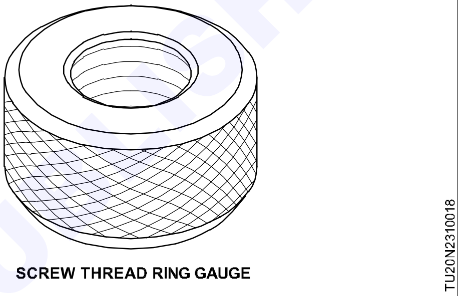
Fig 7



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.

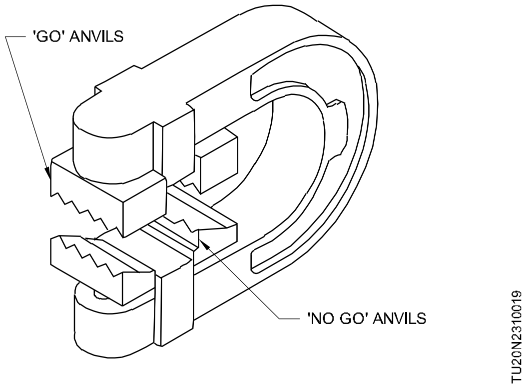
Fig 8



Screw thread 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

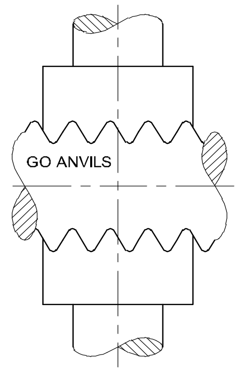
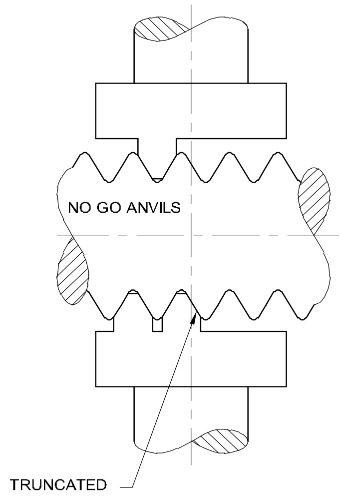


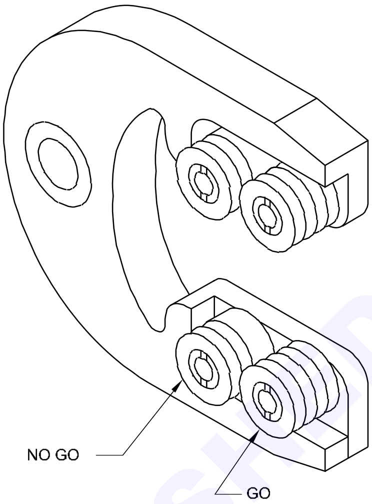
Fig 11



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.

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

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.

Fig 1

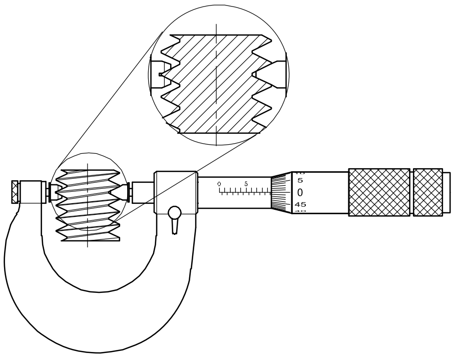
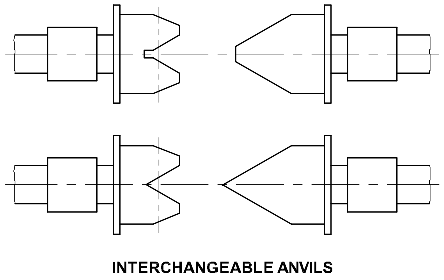
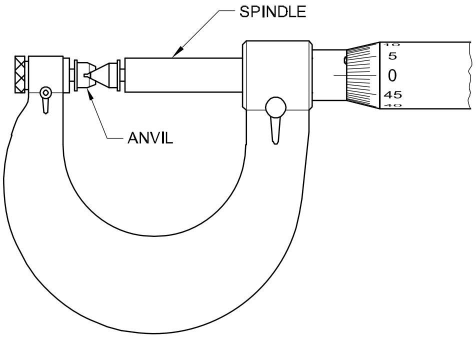


Fig 2



INTERCHANGEABLE ANVILS

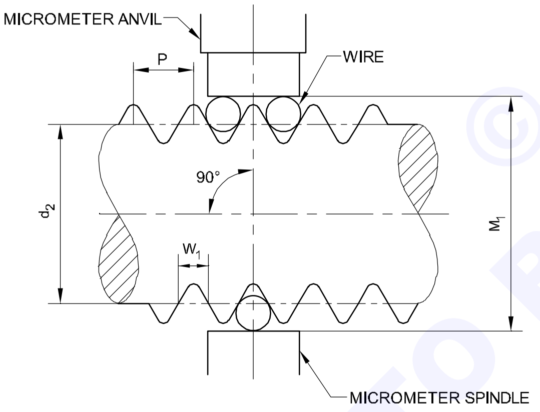
Fig 3



SPINDLE AND ANVIL OF MICROMETER AT ZERO SETTING

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)

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 5

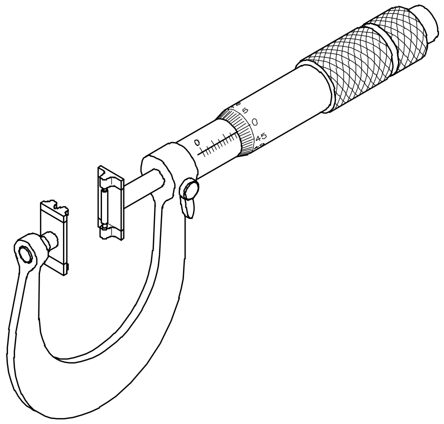
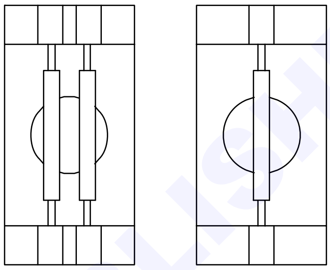


Fig 6

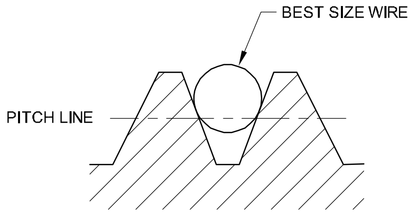


Selection of 'best wire' (Fig 7)

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 designation	Pitch P mm	Basic measurement 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 designation	Basic measurement 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.**

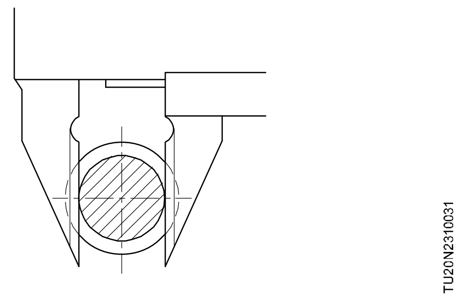
The measurement of the minor diameter and checking the form of the thread are important for producing accurate threads.

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

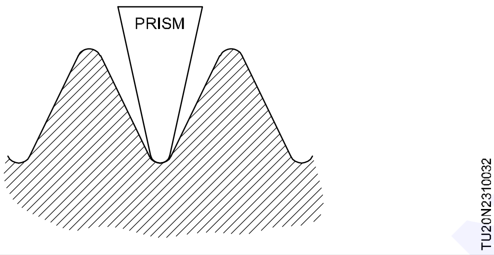
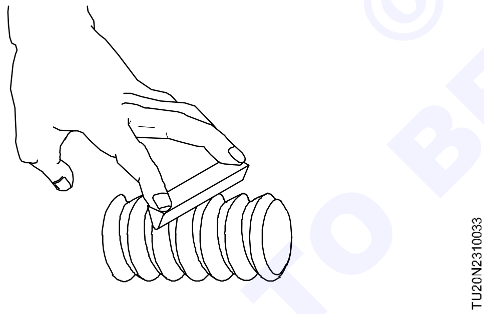


Fig 3



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.

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.

Fig 4

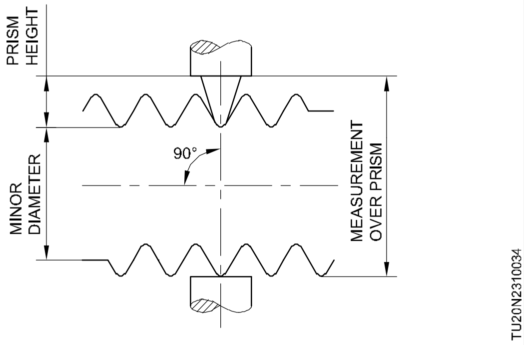


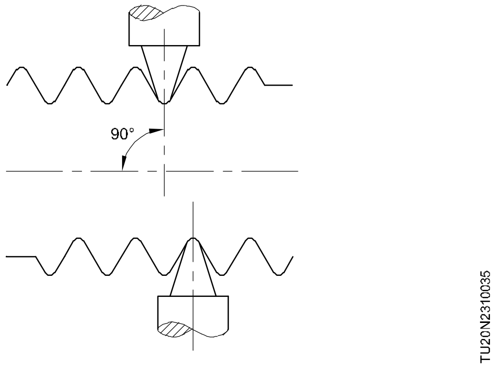
Table 1

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	

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

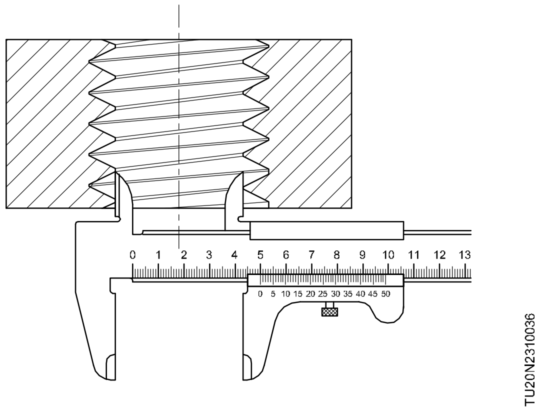


Measuring minor diameter of internal threads

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)

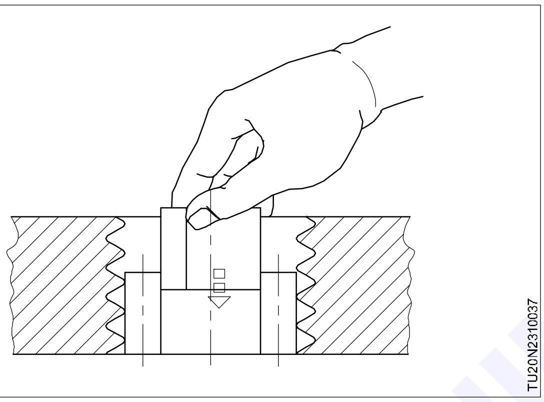
Direct accurate measurement of internal minor diameter is a difficult task.

Fig 6



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

Fig 7

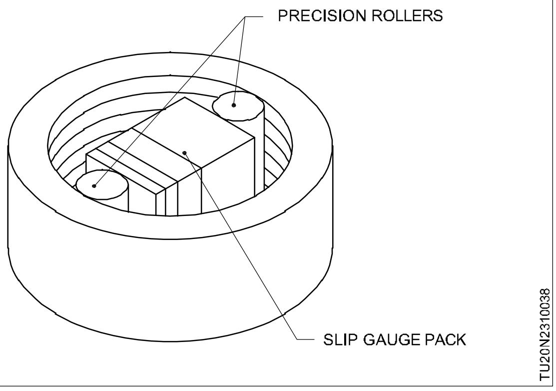


Using slip gauges and precision rollers

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

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

Fig 8

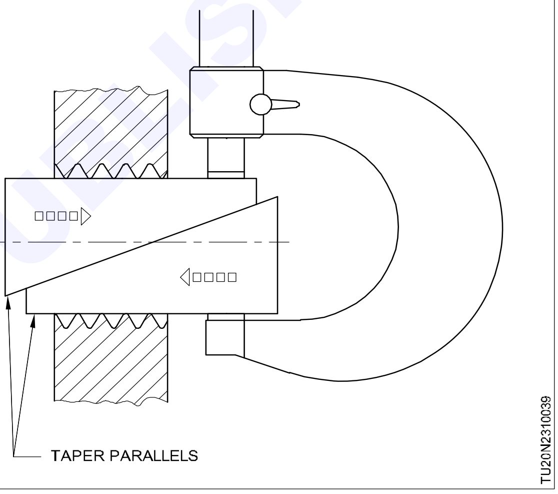


The sum of the size of the slip gauge pack and the diameters of the rollers is the minor diameter.

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.

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.

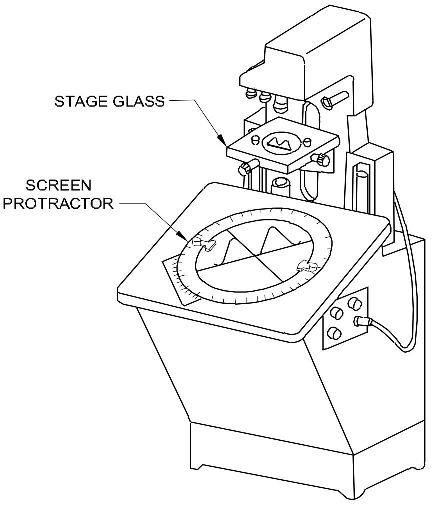
Dental wax or superfine plaster of paris can be used for preparing the cast mould.

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)

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

Fig 1



OPTICAL PROJECTOR

Fig 2

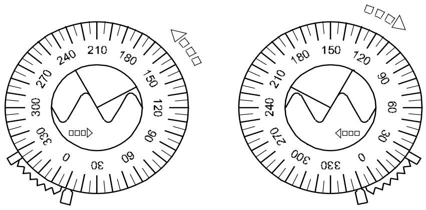


Fig 3

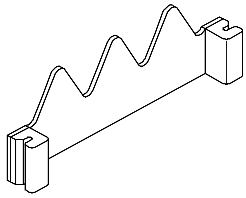


Fig 4

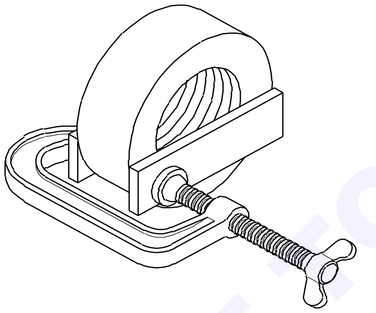
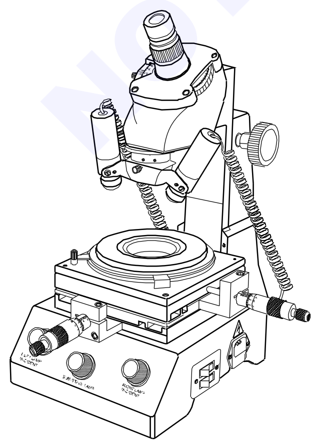


Fig 5



MICROSCOPE

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.

Fig 6

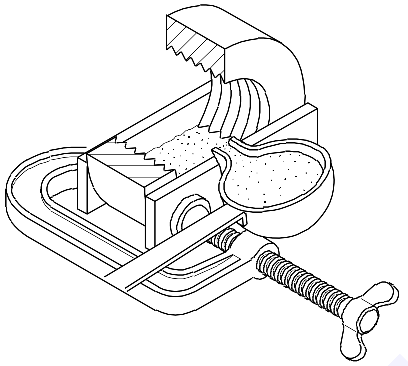
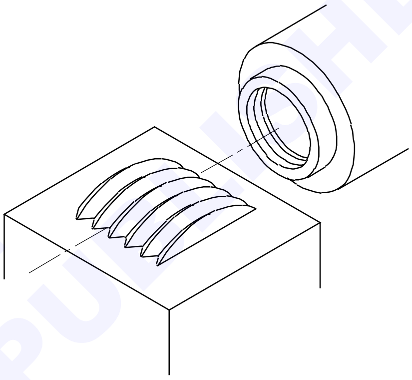


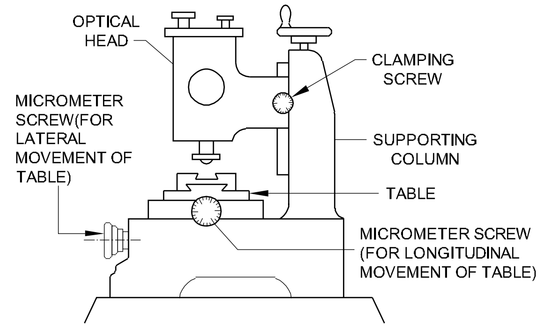
Fig 7



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 8 indicates the elements of microscope used for thread inspection.

Fig 8



TOOL MAKERS MICROSCOPE

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.**

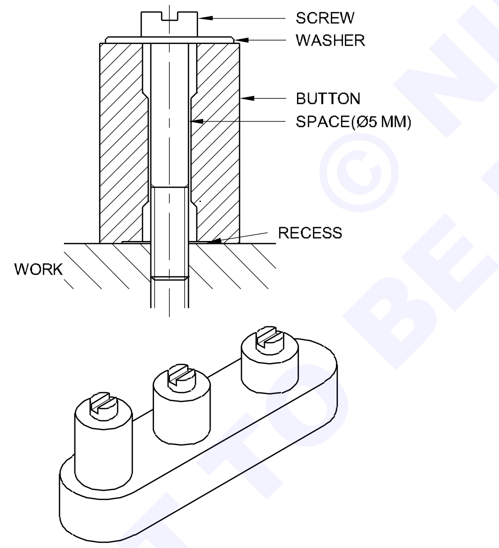
Tools marker’s button uses

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.

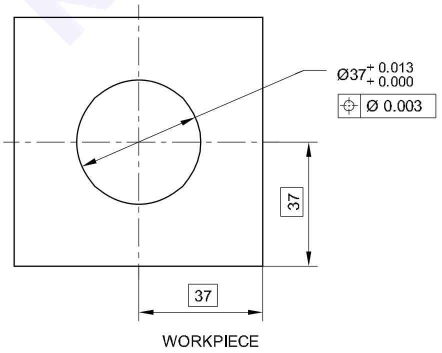
A toolmaker’s button (Fig 1) is a hardened and ground cylinder made of steel of 8, 10 or 12 mm diameter. The ends of the button are accurately square to its cylindrical axis. It has a 4 BA screw to hold the button in position on the workpiece, allowing sufficient clearance between the screw and the button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

Fig 1



Application (Fig 2)

Fig 2

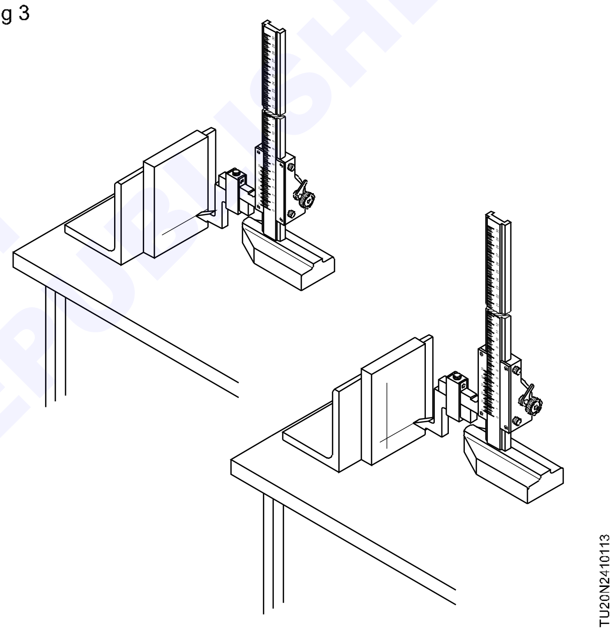


In the component shown here, a hole of Ø 37 mm is to be bored to a positional tolerance of 0.003 mm. The following procedure is to be followed.

Procedure

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

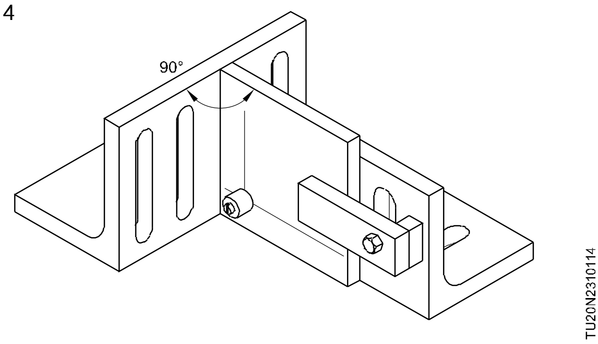
Fig 3



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

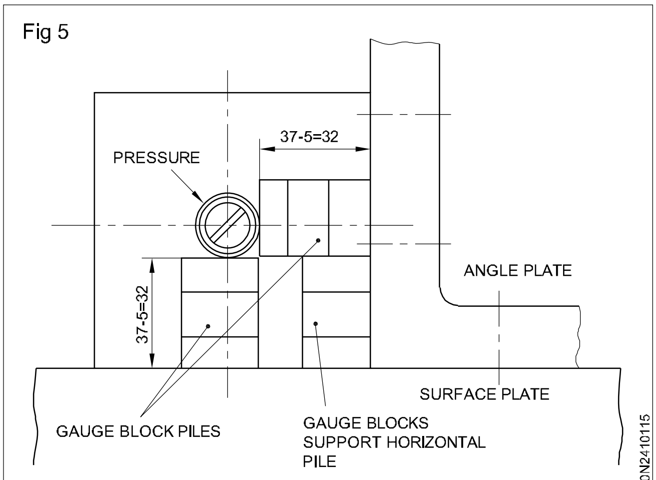
Fit a toolmaker’s button over this hole, but do not fully tighten the holding screw. (Fig 4)

Fig 4



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)

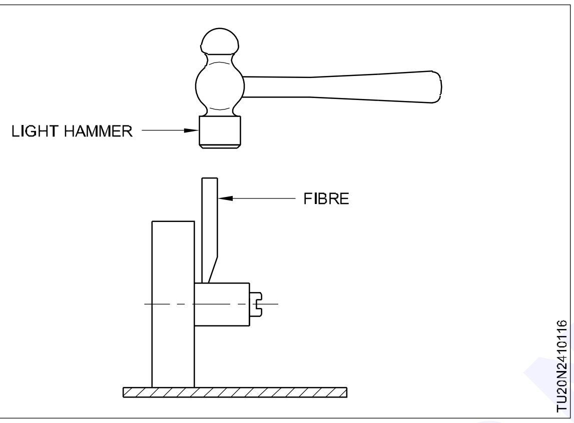
Fig 5



SETTING THE BUTTON

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

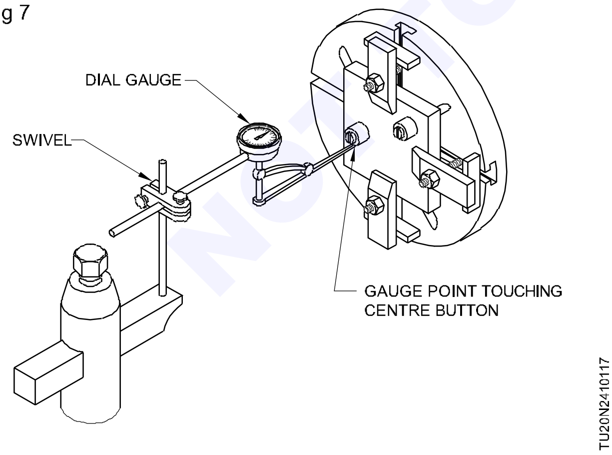
Fig 6



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)

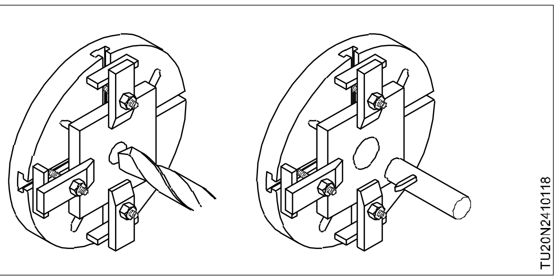
Fig 7



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

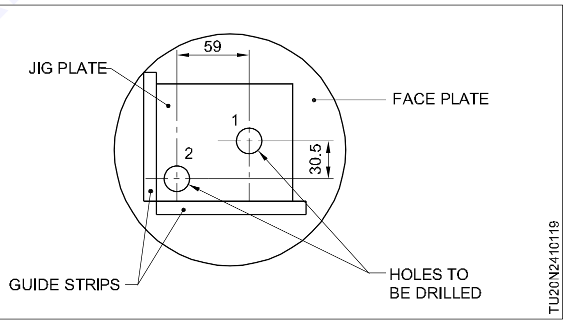
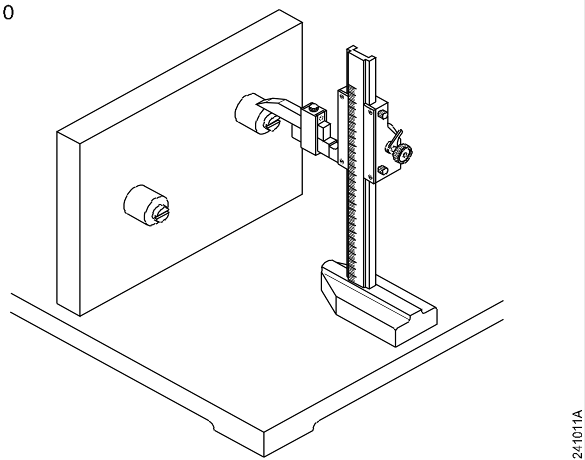


Fig 10



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

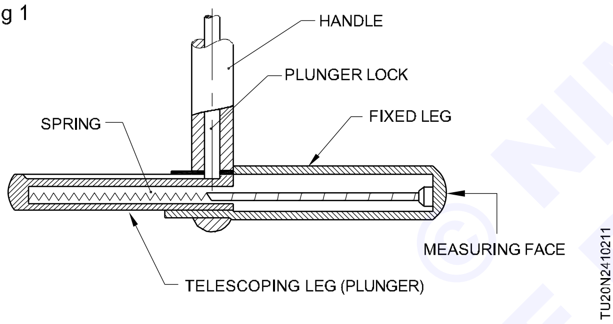
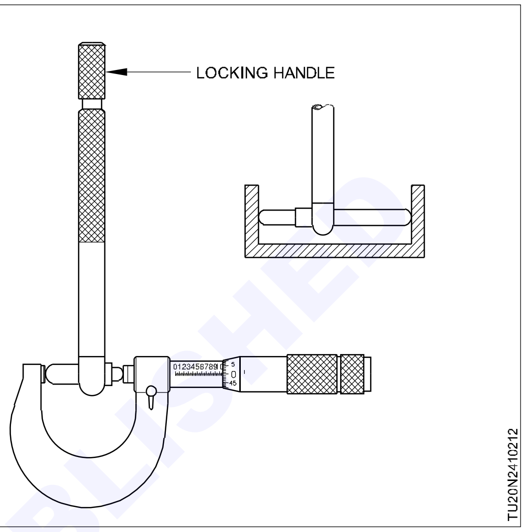
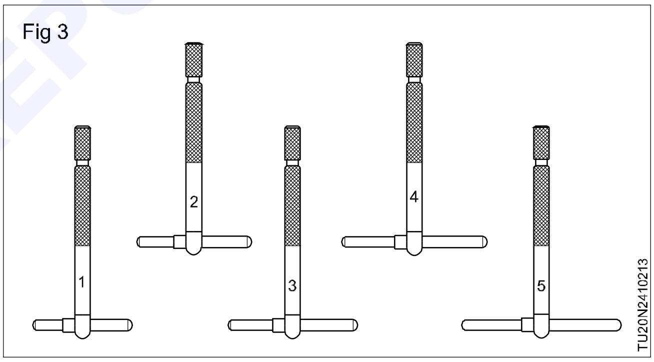


Fig 2



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 (Fig 1) but without the 'U' frame. 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

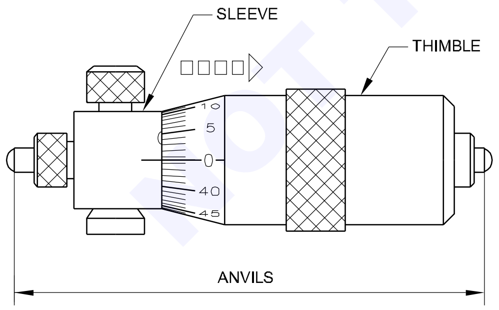
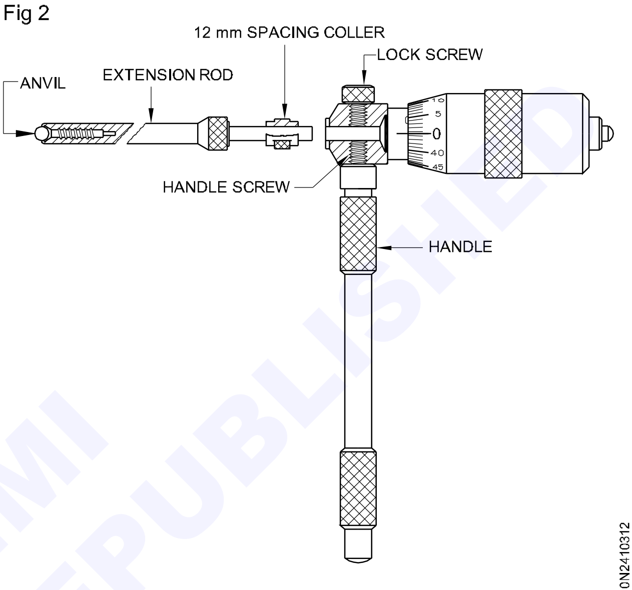


Fig 2

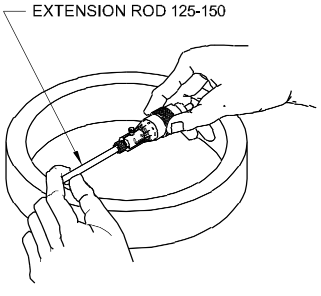


INSIDE MICROMETER

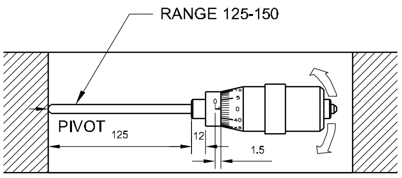
Determining the size of a bore or hole

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



MEASURING A LARGE BORE USING INSIDE MICROMETER WITH AN EXTENSION ROD FITTED



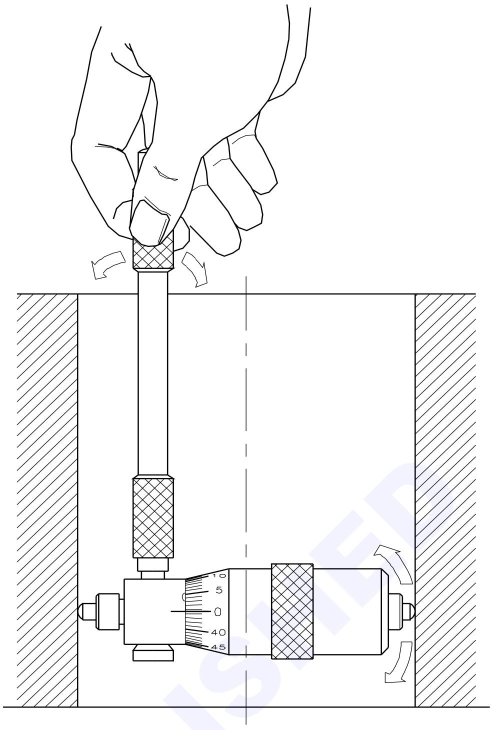
Determining the distance between internal parallel surfaces

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



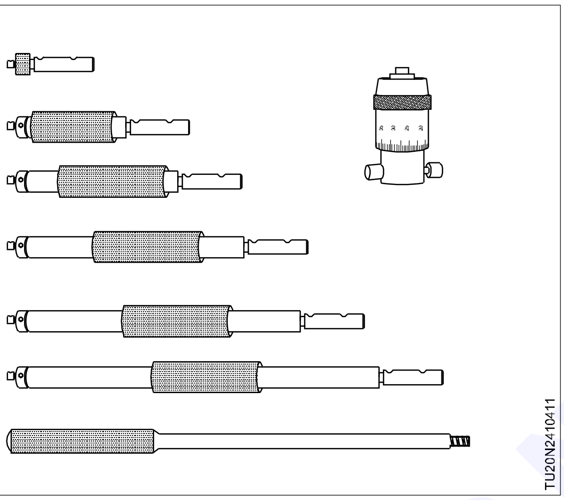
USING AN EXTENDED HANDLE WHEN MEASURING THE BORE OF A SLEEVE BEARING

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 least count of the instrument is 0.001”.

Fig 1



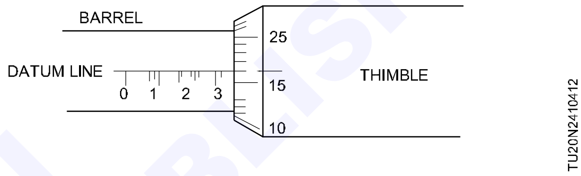
The inside micrometer inch is equiped with a 1/2” spacing coller an 4 extension rods for measuring holes of ranges 2” - 3”, 4” - 5” and 5” - 6”. The sleeve is marked with the main scale and the thimble with the thimble scale. The barrel has a limited adjustment 1/2”. When the inside micrometer is closed (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 read 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



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

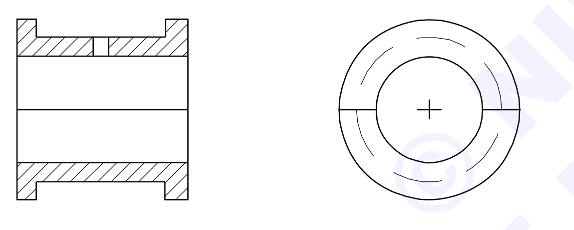
Split bearing are principally used where the replacement of spherical bearings would require costly additional work, involving the removal of gears or couplings, the dismounting of drives or the dismantling of shaft power trains. The use of split reduces the down time of machinery and plant.

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

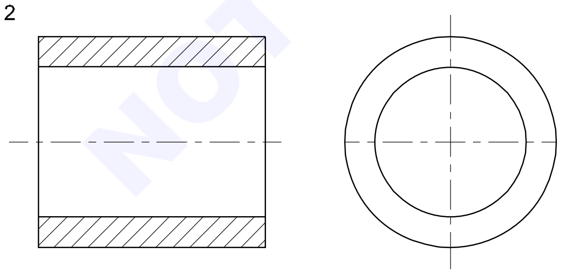


FLANGED PATTERN

- 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 shell 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.

Split bearings, can use 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

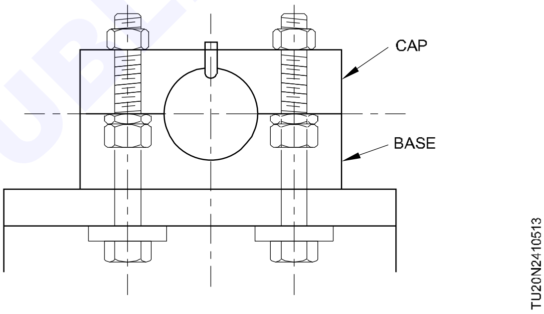
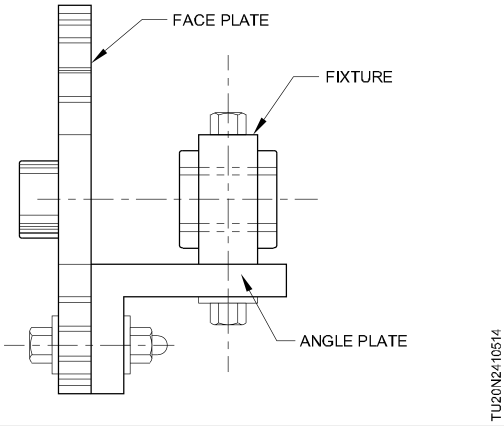


Fig 4



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.

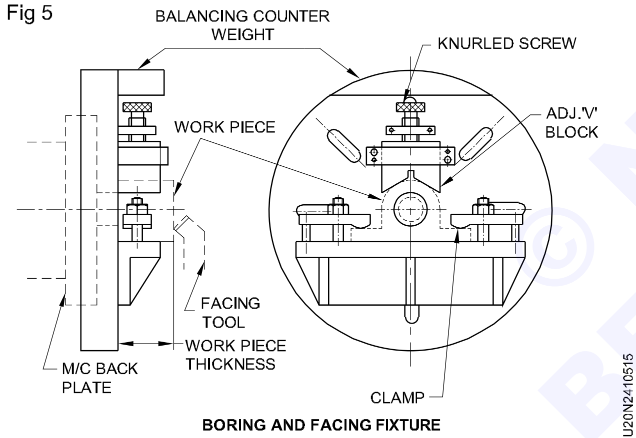
Lathe fixtures (Turning fixtures)

The standard work holding devices or fixures for lathe are:

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

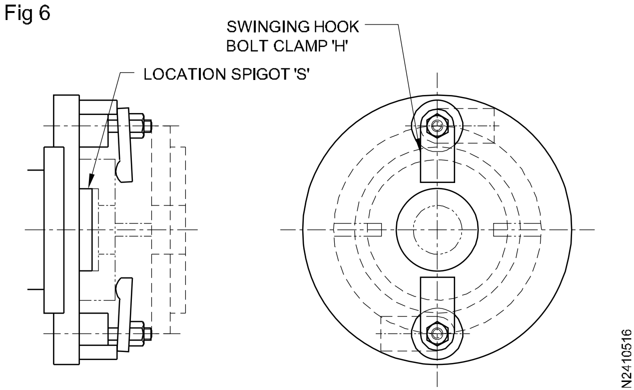
If the job can be held easily and quickly in the above mentioned standard devices, then there is no need for special work holding devices. However many jobs particularly casting and forging, because of their shapes, cannot be conveniently held by any of the standard devices. It then becomes necessary to build a special work holding devices for the job. Such a device is called lathe fixture. (Fig 5&6)

Fig 5



BORING AND FACING FIXTURE

Fig 6



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 Design 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:-

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

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

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

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

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

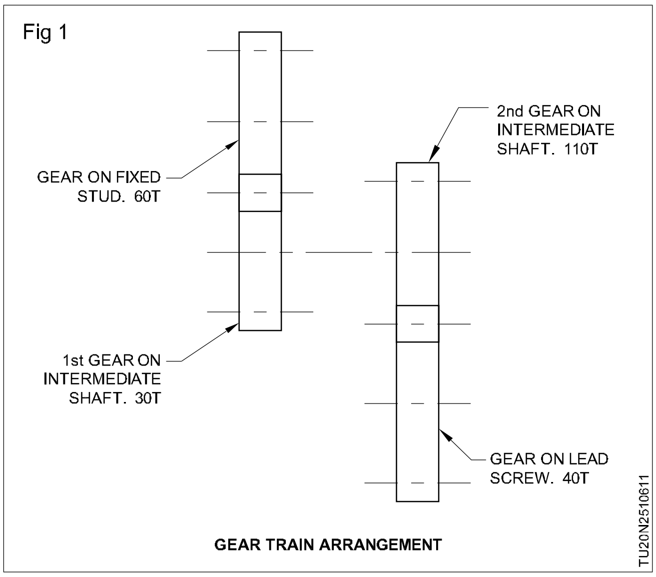
$$\text{Gear ratio} = \frac{11}{8}$$

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

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

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

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

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

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

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.

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

$$= 0.55 \times 6$$

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

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

$$\text{2nd fraction} = \frac{6 \times 20}{1 \times 20} = \frac{120}{220}$$

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

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

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

$$= 0.95 \times 6$$

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

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

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.

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

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

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 $\frac{5}{14}\times\frac{5}{14}$

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

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:

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

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

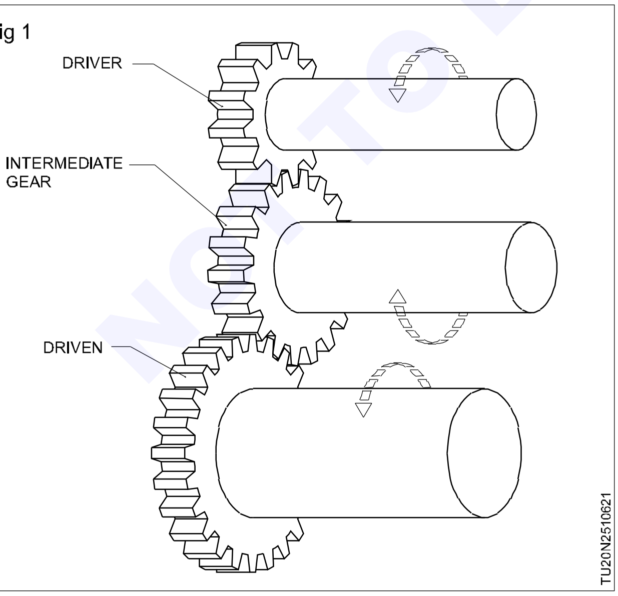
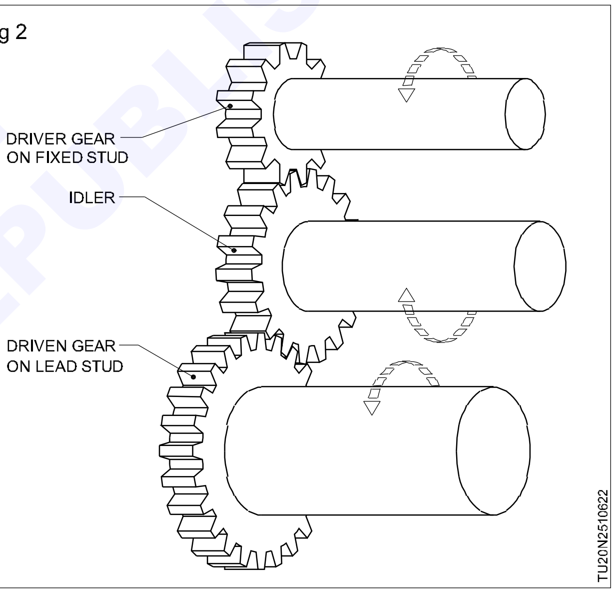


Fig 2 shows mountings of the driver and driven gears in a lathe.

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

Fig 2

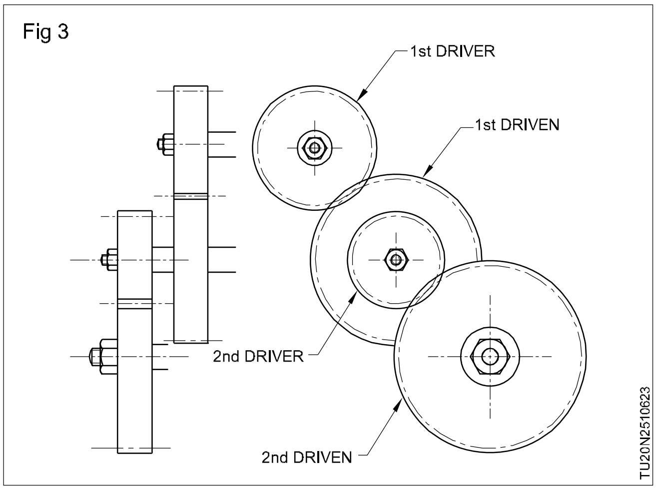


Compound gear train

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 = I”

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 = $\frac{315}{1000 \times 8}$

$= \frac{63}{1600}$

Gear ratio

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

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:

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

When lathe constant is given

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

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,

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

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

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

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 = $\frac{1}{6}$

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

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

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

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

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

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

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

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

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

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

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

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 functions and 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) .

$$\frac{DR}{DN} = \frac{1600}{63PN}$$

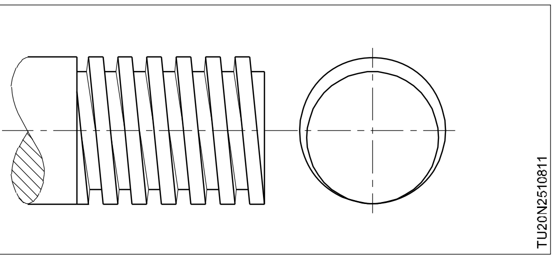
Multi-start thread and methods

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.

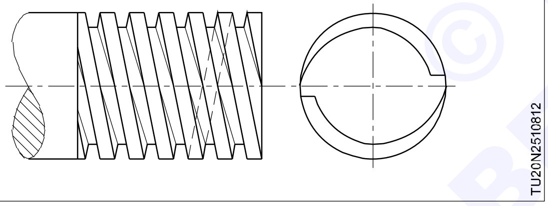
The lead on a single start thread is equal to the pitch. (Fig 1)

Fig 1



The lead on a double start thread is twice the pitch. (Fig 2)

Fig 2

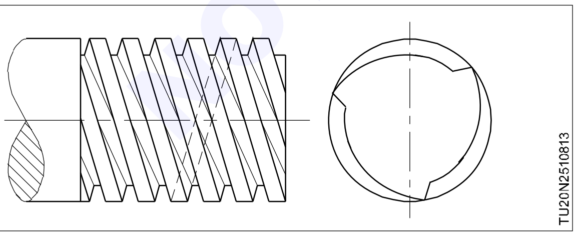


The lead on a triple start thread is three times the pitch. (Fig 3)

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

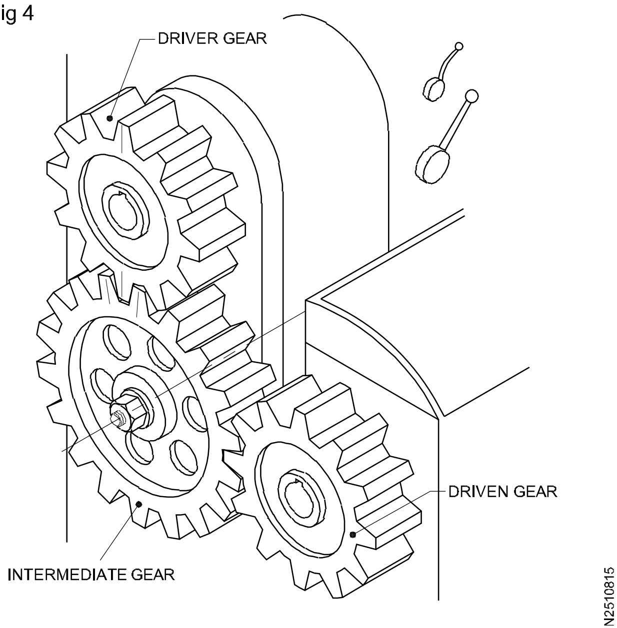
- Dividing the 1st driver of the change gear train
- Using slotted face plate
- Moving the top slide to a new position (compound slide)
- Using thread chasing dial

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.

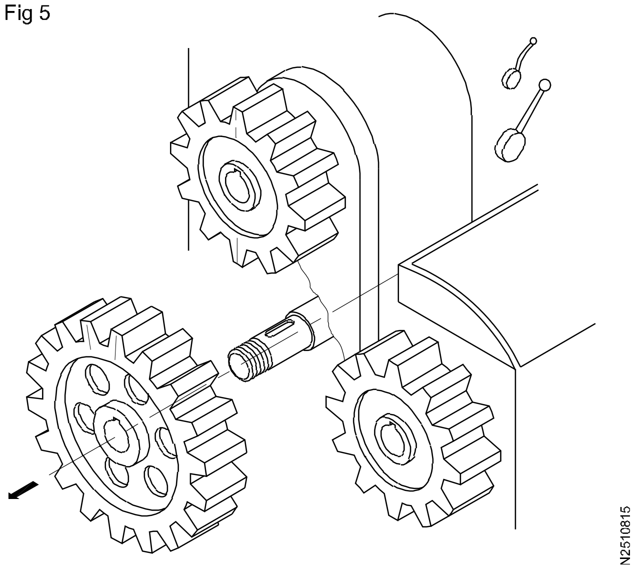
After finishing the first start, the lathe is stopped. One tooth of the 1st driver and the space of the first driven gear in which it is seating are marked. By counting the number of teeth from the marked tooth of the 1st driver, make another mark on the tooth which is exactly 180° away. Loosen the swing 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 5



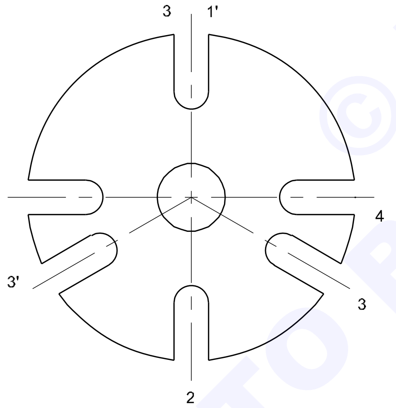
DISENGAGE THE INTERMEDIATE GEAR

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

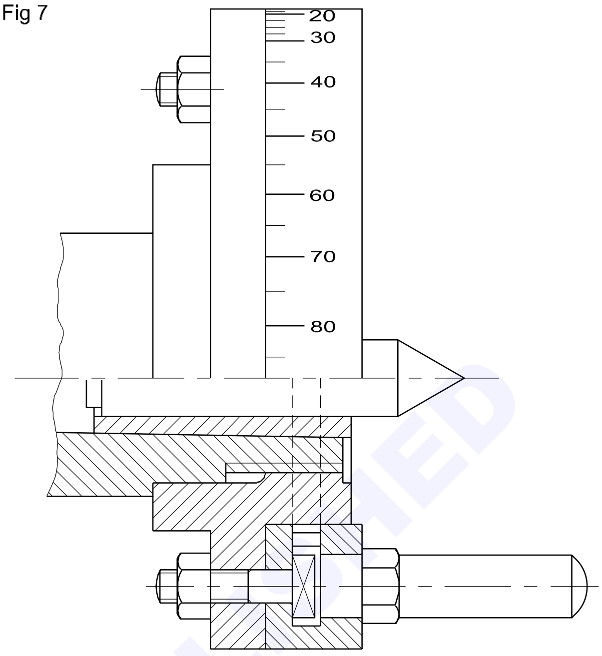
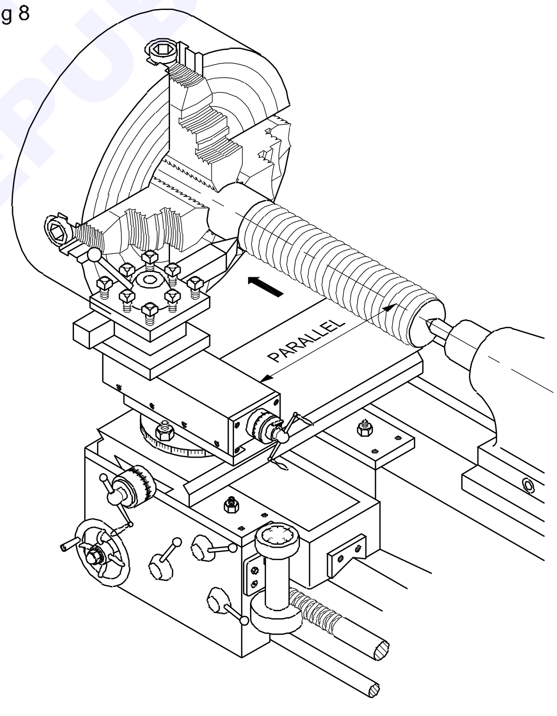


Fig 8



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

$$\text{Lead of the work} = \frac{1}{12} x4 = \frac{45}{60}$$

$$= \frac{4/12}{1/4} = \frac{4 \times 4}{12 \times 1} = \frac{16}{12}$$

$$\frac{16}{12} = \frac{4 \times 10}{3 \times 10} = \frac{40}{30} = \frac{Dr}{Dn}$$

$$\left.\begin{array}{l}\text{The number of threads}\\ \text{for which gears are}\\ \text{to be determined}\end{array}\right\} = 3.$$

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

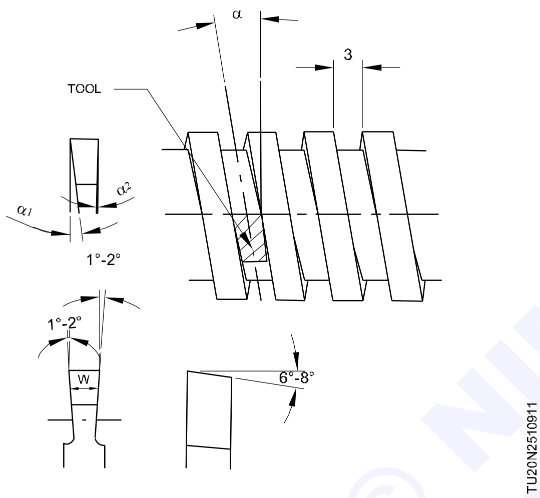
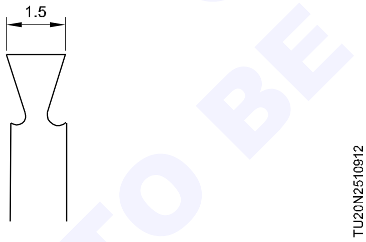


Fig 2



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 3

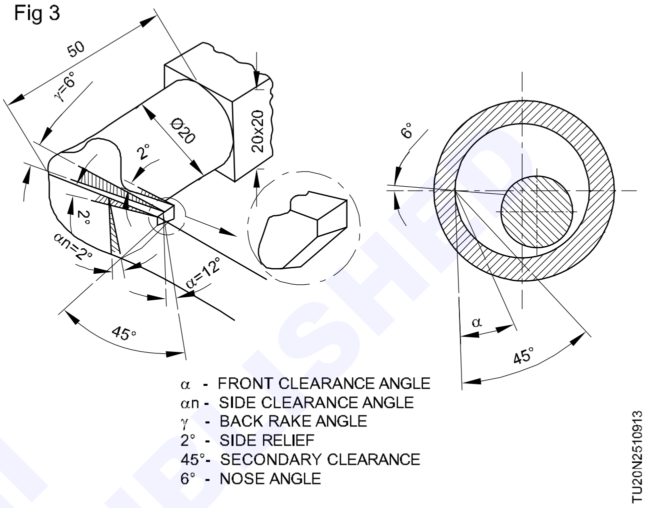


Fig 4

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

$$\frac{DR}{DN} = \frac{\text{lead of the thread}}{\text{lead of the lead screw}}$$

$$= \frac{6\times10}{5\times10} = \frac{60}{50}\text{ Ans}$$

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 x 2.4536

= 25.09 mm.

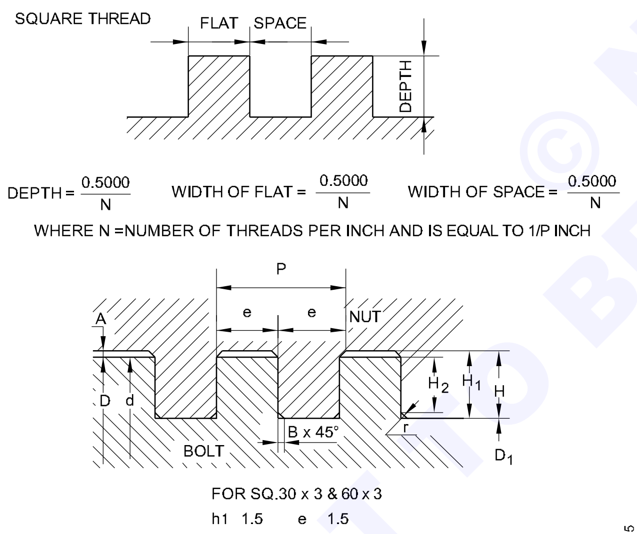
Square and trapezoidal threads

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 5.

Fig 5



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

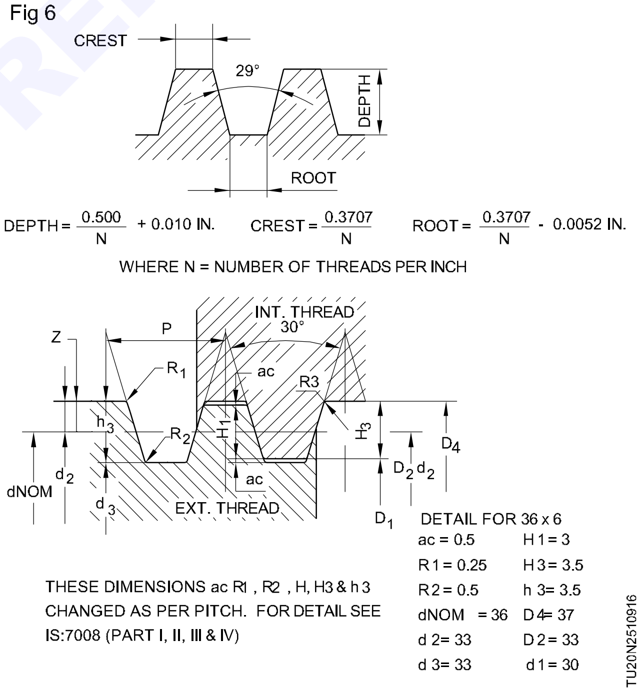
These threads have a profile which is neither square nor ‘V’ thread form and have a form of trapezoid. They are used to transmit motion or power. The different forms of trapezoidal threads are:

- acme thread
- buttress thread
- saw-tooth thread
- worm thread.

Acme thread (Fig 6)

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 6

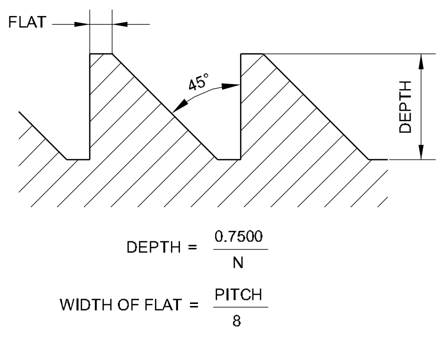


THESE DIMENSIONS ac R1 , R2 , H, H3 & h3 CHANGED AS PER PITCH. FOR DETAIL SEE IS:7008 (PART I, II, III & IV)

Buttress thread (Fig 7)

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. Fig 7 shows the various elements of a buttress thread. These threads are used in power press, carpentry vices, gun breeches, ratchets etc.

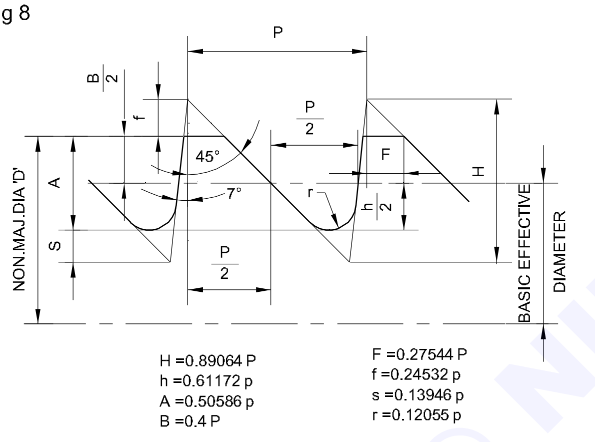
Fig 7



Buttress thread as per B.I.S. (Fig 8)

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.

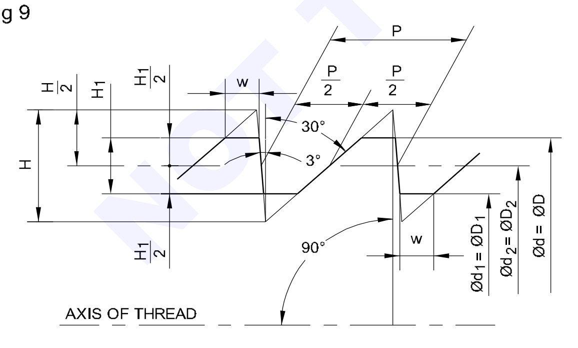
Fig 8



Saw-tooth thread as per B.I.S. 4696

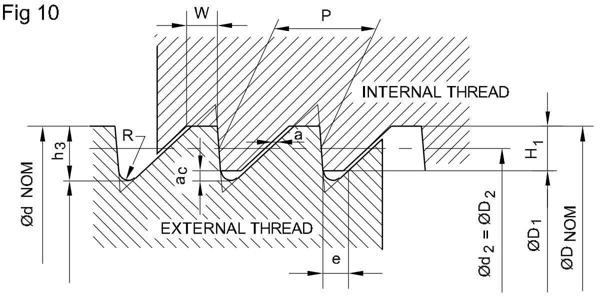
This is a modified form of buttress thread. In this thread, the flank taking the load is inclined at an angle of 3°, whereas the other flank is inclined at 30°. The basic profile of the thread illustrates this phenomenon. (Fig 9) The proportionate values of the dimensions with respect to the pitch are shown in Figs 10 and 11.

Fig 9



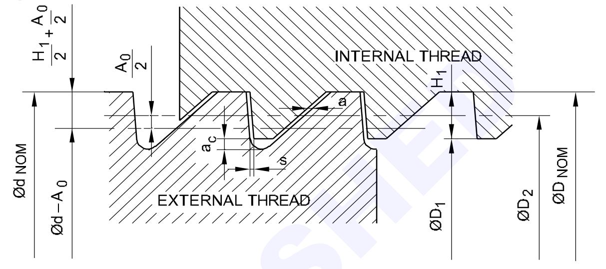
BASIC PROFILE FOR SAW TOOTH THREADS

Fig 10



DESIGN PROFILE FOR THREADS WITH CLEARANCE ON NON-BEARING FLANKS

Fig 11



DESIGN PROFILE FOR THREADS WITH CLEARANCE ON BEARING AND NON-BEARING FLANKS

The equations associated with the dimensions indicated in the two figures (Figs 6 and 7) are given below.

$H_1 = 0.75\ P$

$h_3 = H_1 + a_c = 0.867\ 77\ P$

$a = 0.1\quad P$ (axial play)

$a_c = 0.117\ 77\ P$

$W = 0.263\ 84\ P$

$e = 0.263\ 84\ P - 0.1\quad P = W - a$

$R = 0.124\ 27\ P$

$D_1 = d - 2\ H_1 = d - 1.5\ P$

$d_3 = d - 2\ h_3$

$d_2 = D_2 = d - 0.75\ P$

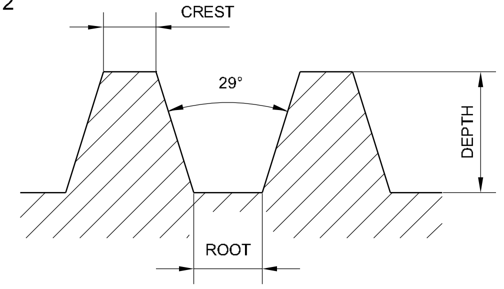
$S = 0.314\ 99\ A_o$, where A_o = basic deviation (= upper deviation) for external thread in the pitch diameter.

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. Fig 12 shows the elements of a worm thread.

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.

Fig 12



$\text{DEPTH} = \frac{0.6866}{N}$ $\text{ROOT} = \frac{0.310}{N}$ $\text{CREST} = \frac{0.335}{N}$

WORM THREAD

TU20N251091C

Module is the ratio between the pitch diameter of the gear and the number of teeth of the gear.

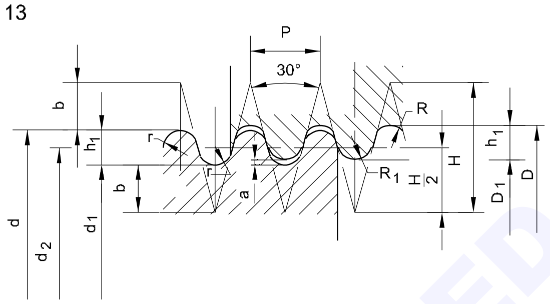
The linear pitch of the worm thread must be equal to the circular pitch of the worm gear. When the worm gear is of D.P. then the linear pitch of the worm thread in mesh is equal to p/DP. When the worm gear is of module teeth, then the linear pitch of the worm thread is equal to module x p.

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 13



BASIC PROFILE OF KNUCKLE THREAD

$b = 0.68301.P$

$h_1 = 0.5P$

$a = 0.05P$

DESIGNATION:- A KNUCKLE THREAD OF DIAMETER 60 mm & PITCH 4.233mm SHALL BE DESIGNATED AS k60x4.233 IS:4695

TU20N251091D

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.

$$\text{Tan } \theta = \frac{L}{\pi \text{ x d}}$$

Outside diameter

where q = Helix angle in degrees

L = Lead of the thread in millimetres (or) inches

π = Constant (3.14 or 22/7)

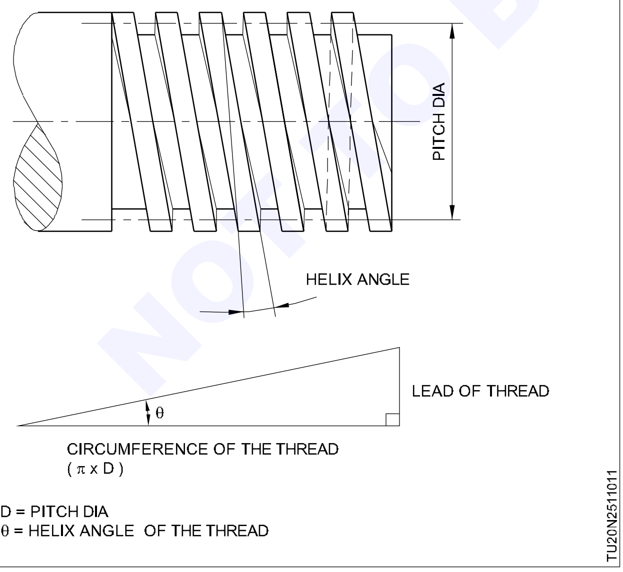
d = Pitch diameter of the thread in millimetres (or) inches.

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

$$\text{Pitchdiameter of the job} = 1\frac{1"}{2} - \frac{1"}{8}$$

$$= 1\frac{3"}{8}$$

$$\text{Tangent of helix angle} = \frac{L}{\pi \times d}$$

$$\frac{1"}{\frac{22}{7} \times 1\frac{3"}{8}}$$

$$= \frac{1}{\frac{22}{7} \times \frac{11}{8}} = \frac{7 \times 8}{22 \times 11}$$

$$\text{Tan } \theta = \frac{56}{242} = 0.2314.$$

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"

$$\text{Pitchdiameter of the job} = 2" - \frac{1"}{16} = 1\frac{15"}{16}$$

$$\text{Tan of helix angle} = \frac{L}{\pi \times d}$$

$$= \frac{\frac{3}{8}}{\frac{22}{7} \times 1\frac{15}{16}}$$

$$= {}^{3}\!/_{8} \times \frac{7}{22} \times \frac{16}{31} = \frac{3 \times 7}{11 \times 31}$$

$$= \frac{21}{341} = 0.0615.$$

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 consideration. 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.

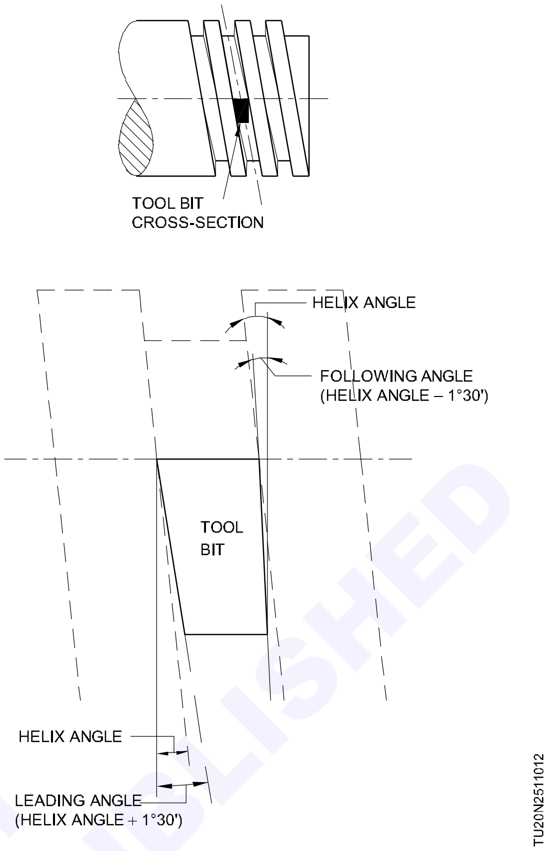
The side clearance on the leading side of the tool = the helix angle of the leading side of the thread + the normal side clearance that will be ground on the tool (1° approximately).

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.

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.)

Fig 2



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

$$= \frac{\text{Lead}}{\text{Major dia circumference}}$$

$$= \frac{0.250}{1250 \times \pi} = \frac{0.250}{3.927} = 0.0636$$

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 lathe 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)

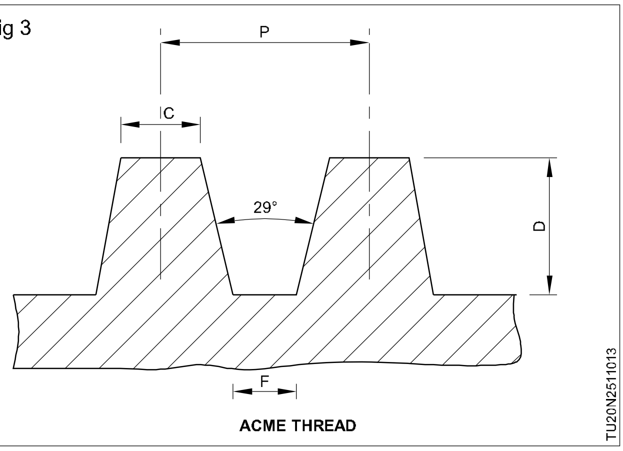
Angle= 29°

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



ACME THREAD

Buttress thread (Fig 4)

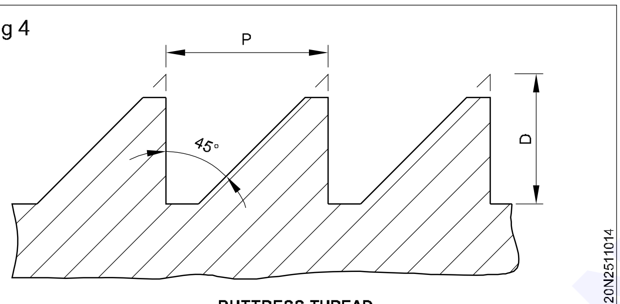
Angle = 45°

One side = 90°

Depth=0.75 x pitch

width of crest and root = 0.125x Pitch

Fig 4



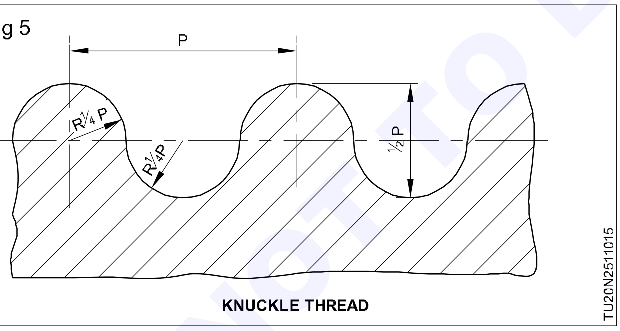
BUTTRESS THREAD

Knuckle thread (Fig 5)

$$R = \frac{1}{4} \times \text{pitch}$$

$$\text{Depth} = \text{pitch}$$

Fig 5



KNUCKLE THREAD

Worm thread or brown and sharp worm thread(Fig 6)

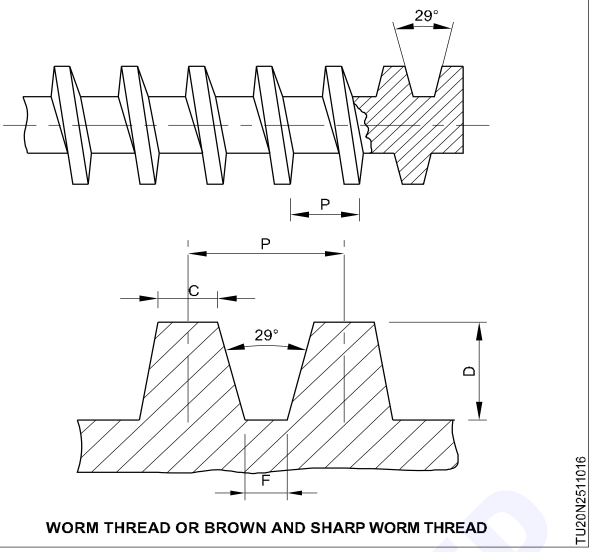
Angle = 29°

Depth = 0.6866x pitch

Plain portion of crest = 0.31 x pitch

Plain portion at depth = 0.335 x pitch

Fig 6



WORM THREAD OR BROWN AND SHARP WORM THREAD

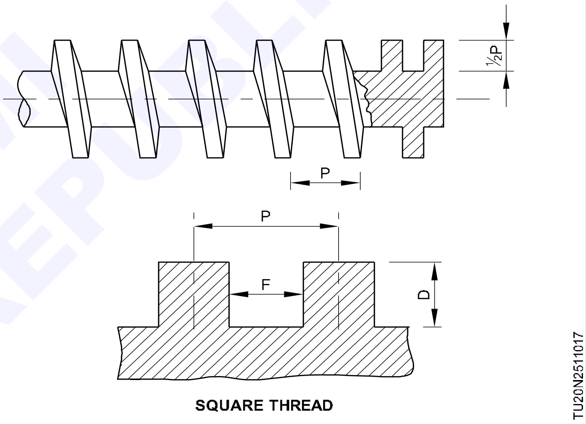
Square thread (Fig 7)

Angle = 90°

Width of tool = 0.5 pitch

Depth = 0.5 pitch

Fig 7



SQUARE THREAD

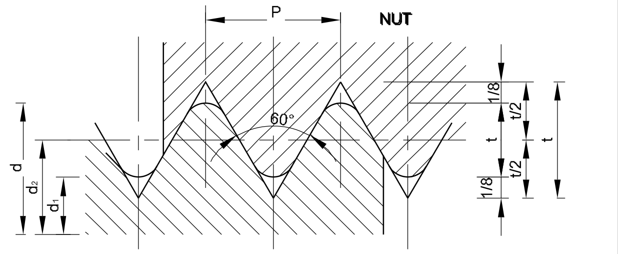
Metric 'V' thread (Fig 8)

Angle - 60°

Core dia

= Major dia - [2 x depth of thread]

Fig 8



HEIGHT OF FUNDAMENTAL TRIANGLE THREADS
(SHARPS V-THREAD)
$t = 0.8660 \times P$
$t_1 = 0.6495 \times P$
$r = 0.1082 \times P$

METRIC 'V' THREAD

CNC Technology basics

Objectives: At the end of this lesson you shall be able to
- **describe the basic of tehnology**
- **state the advantages of CNC turning**
- **state the advantages and disadvantages of CNC machine over conventional machine**
- **state the specification of CNC lathe.**

When the computer was invented, the inventor himself must not have dreamt of the use of computers in various fields of life which is drastically changing the entire scenario of the Universe. It is now an integral part of our day to day life. There is lot of research going on with the help of computers in the field of factory automation. The declining cost of computers coupled with the invention of Multi task high speed micro processors, really made an industrial revolution and there seems to be no end for this. A distinct trend can be observed in industries which include an increase in the use of Computer controlled Machine tools, the application of new manufacturing systems, such as laser beam machines and appearance of new generation of industrial robots in the production line, the manufacturing management through MRP I, MRP II & MRP III etc.(Material Resource Planning)

Evolution of automation

Automatically controlled factory is nothing more than the latest development in the industrial revolution that began in Europe two centuries ago and progressed through the following stages:

- Mechanisation started in 1870 at the beginning of industrial revolution with simple production machines.
- In 18th Century fixed automatic mechanism and transfer lines came into existence for faster output and shorter production time.
- Simple automatic control machines and copying machines were invented in the later part of the 18th century. After 1950 the industrial automation was started. In this second phase of the industrial automation/revolution, workers, instead of physically performing all the task are placed in the control of the machines.

Progressive change after 1950 is as follows

- The introduction of numerical control (NC) in 1952 opened a new era in automation.
- The extension of NC was Computerised Numerical Control (CNC) machine tools in which computer (Micro Processor) is included as an integral part of the control system.
- Commercial Industrial robot was manufactured in 1961 along with CNC systems. The use of these robots, are well utilised only after 1970's.
- The next logical extension is a fully automated factory which employs a flexible manufacturing system (FMS) and Computer Aided Design/Computer Aided Manufacturing (CAD/CAM) techniques.
- The latest of the above is Computer Integrated Manufacturing (CIM) which includes battery of CNC machines, with flexible modules for manufacturing tool head changers, automatic material handling system like AGV's (Automated Guided Vehicle) etc with minimum number of operating personnels.

Fundamentals of NC controls

NC equipment has been defined by Electronics Industries Association (EIA) as "A system in which actions are controlled by the direct insertion of numerical data at some point. The system must automatically interpret at some portion of the data".

In a typical NC system the part program is prepared on a punched tape. The part programme is arranged in blocks of information needed for processing a segment of work piece, the segment of length, speed, etc.

Advantage of NC machine are

- Complex shapes can be machined easily
- Accuracy and repeatability is achieved.
- High production rate
- Reduced component rejection
- Less operator skill and involvement

There are many disadvantages of NC system:

- If tape is spoiled the entire programme of manufacturing will be affected
- Editing of the program in tape is not very easy.
- Manual loading of tape is a laborious job.
- Instruction are read, block by block and carried out which is slow when compared to CNC machine tools:
- If the punch reader is not reading the program properly then the entire production is lost.

Computer numerical control

A dedicated micro processor or mini computer on the machine control makes the computer numerical control. CNC machines are very popular coupled with lots of other advantages:

Advantages

- Accuracy and repeatability is very high
- Reduced scrap and rework
- Reduced inspection time
- Ease of inter changeability of machined parts
- Reduced space
- Reduced material handling
- Less paper work
- Less lead time
- Less inventory
- Easy editing of programme
- Complicated shapes and contours are easily manufactured with quality assurance and better production management.
- Better utilisation of machines.
- Reduced tooling
- Reduced operator skill
- Jig not used but with minimum fixtures
- Reduced floor space
- Higher level of integration such as DNC, FMS, CAD/CAM, CIM etc.,

Disadvantages of CNC machines

- High cost of machine
- High cost of training needs
- High Maintenance cost

Major advantages of CNC machines are

- Higher production
- High quality production

These are achieved through:

Higher production

A. Keeping idle time as minimum as possible

B. Keeping machining time to a minimum

A Keeping idle time as minimum

1 Loading/unloading : Through quick work holding and work handling system like pallets, robots etc.

2 Tool Change time : Kept as minimum by ATC, quick change tool turret etc. (max.time is less than 8 sec.)

3 Movement of slide : Rapid movement is easily achieved through best servo feedback motors

4 Changing of cutting conditions : Step less Speed, feed etc are changed easily through programming instruction.

5 In process control : Self diagnosing and gauging through measuring probes the parts and tools are available as an in-process control.

B Keeping machining time to minimum

Higher metal removal rate through

- Proper cutting tools
- Rigid machine spindle
- Higher spindle power
- Higher feed power
- Rigid structure
- Multi spindle
- Multi turrets etc.

Higher quality is achieved

- Servo mechanism - For correction of feed through motors
- Curvic coupling - For quick indexing
- Linear motion guides - For heavy load movement of slides.
- Linear ball screw - For accurate friction free backlash free movement
- Encoders and tacho generator - For accurate positioning and velocity error correction etc.

Applications of CNC

In automobile, aircraft and general engineering industry, CNC machines are common sight now a days. CNC is used to control almost all types of machines and some of the commonly used machines are listed below:

- CNC lathes
- CNC Milling/drilling machine
- CNC turning centres.
- CNC Turn mill centre
- CNC Machine centre, Multi machining centre
- CNC Tool and cutter grinding.
- CNC Grinding machine, surface, cylindrical etc.
- CNC boring and jig boring machines etc.
- CNC EDM, Wire cut EDM etc.
- CNC Gear hobbing, gear shaping, gear grinding etc.
- CNC Electron beam welding
- CNC Laser/plasma/arc welding machine etc

- CNC Co-ordinate measuring machines
- CNC Nibbling press, press brakes, turret

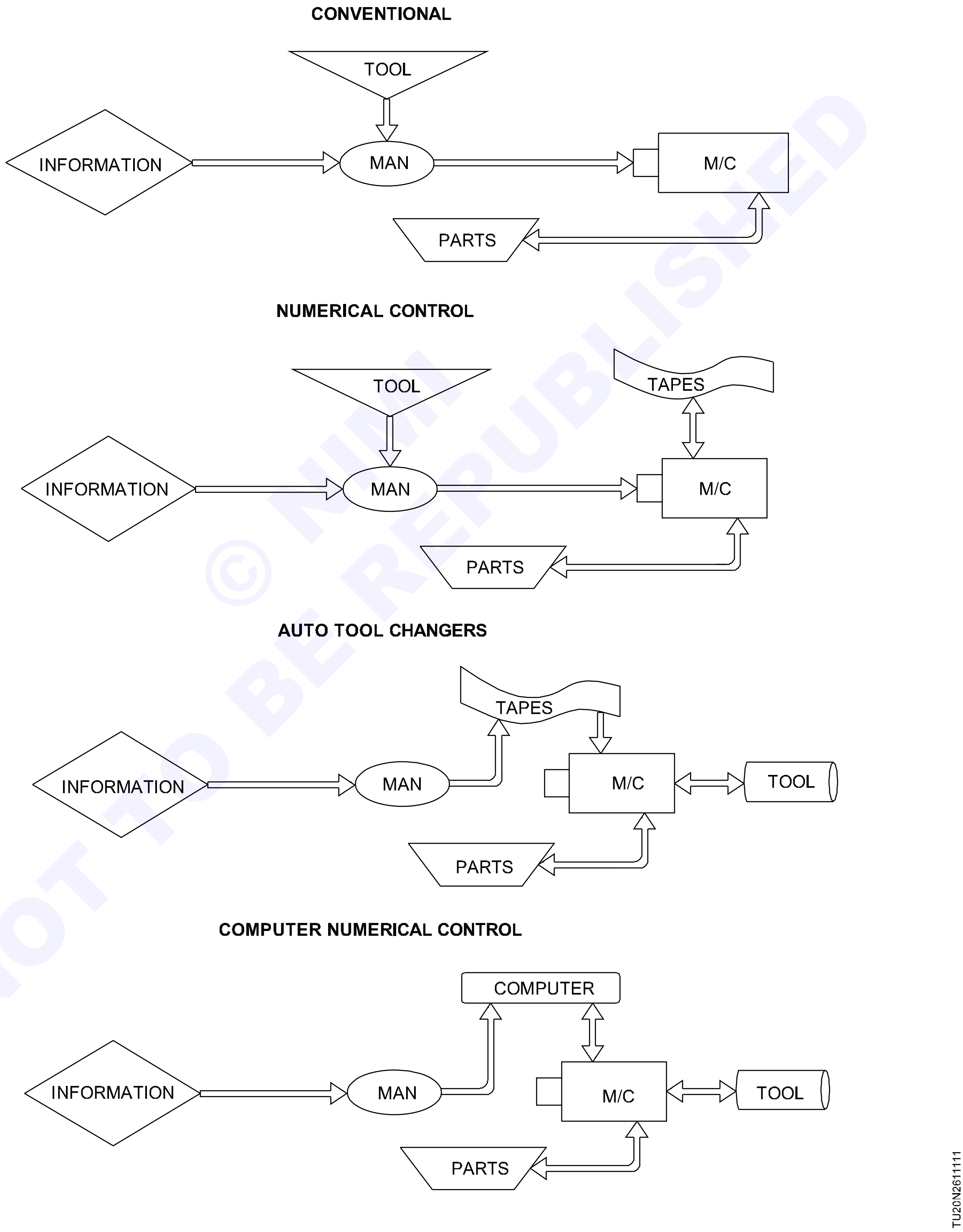
Present status CNC technology

Now a days, CNC controllers with system like Sinumeric, Fanuc, Friera, Allen Brandly, Mazak etc come with graphic display of tool, paths, along with other software's have considerably reduce the manual part programming of three dimensional jobs.

User defined parametric programming, Standard Cycles like stock removal, drilling milling pattern etc are now a day's standard component of the Systems.

The pictorial representation of conventional machine numerical control,. auto tool changer and computer numerical control are shown.

Fig 1



Some latest controls are having "DOS" front end with CAD/CAM facility in which one can design a component and get the computer assisted part programmes (CAPP) and proving the component on the machine control itself without wasting much time and money. Modern machine tools have multi spindle with a spindle speed of 75000 rpm; Cutting feed rate of 5000 mm/min, and rapid traverse of 20000 mm/min. Use of multi various sensing elements with adaptive controls, remote diagnostics system makes the machine more versatile and free from accidents.

The silent and salient use of computers in factory automation and in factory management will boost the quality and quantity in production, which in turn will definitely change the lifestyle of the people in future.

Difference between conventional and CNC lathes

Conventional lathe	CNC
1 Involves more manual work	Less manual work
2 Skilled labour needed	Basic Skill is enough
3 Less accuracy	More accuracy
4 Less flexible	More flexible
5 No part programming	Part programming required
6 Any alteration is difficult	Re-programming for dimensional changes made easier manually
7 For every component machining is done with great care.	Once the programme is done, the computer takes care
8 Simulation or trial run not possible	Simulation or trial run possible and correction may be done if required.
9 Less production rate	More production rate
10 Repeatability is not possible	Rate of repeatability is high
11 Individual operator required for each machine	One operator can operate more than one machine

Advantages of CNC machines over conventional lathes

- Less manual work.
- Semi skilled operator can operate the machine.
- Greater accuracy.
- More flexibility.
- Alteration in dimension is easier through programme.
- Simulation is possible with that we can verify the dimensions of the component.
- Production rate is more.
- Profitability is high.
- Repeatability is very high compared to conventional lathe.
- One operator can operate more than one machine.
- Lesser production cost.
- Reduced part inventory.
- Reduced floor space requirements.
- Improved manufacturing control.
- Complicated parts shape can be easily machined.
- More number of tools are made available

Disadvantages of CNC machines

- Higher investment cost.
- Higher maintenance cost.
- Training of CNC operator involves more cost.
- Semi skilled or unskilled operator cannot do programming in CNC.
- Cost of spare parts and tool cost are high.
- Suitable for mass production only.

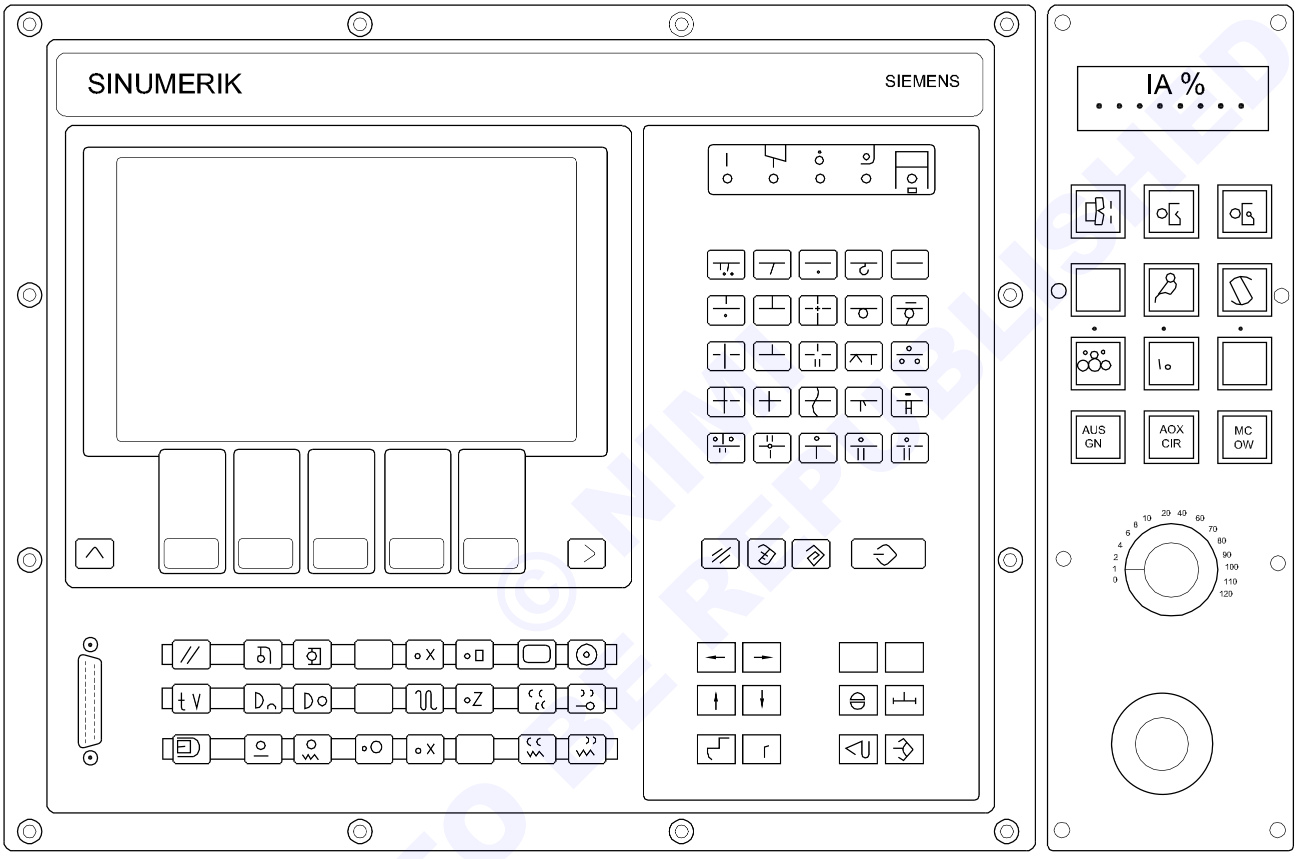
SPECIFICATIONS OF C.N.C LATHE

1	No. of controlled axis	2
2	Interpolation	Linear/circular/parabolic
3	Maximum swing over bed	320 mm
4	Maximum machining length	245 mm
5	Collet	ID = 56 mm OD = 48 mm
6	Spindle taper hole	∅52 mm
7	Maximum bar dia	∅38 mm
8	Spindle head type	A2 - 5
9	Spindle speed range	60 to 5000 R.P.M
10	Main motor	3.70 KW
11	Chuck size	∅200 mm
12	Chuck type	Hydraulic, solid
13	Rapid transfer speed on x axis	18 metre/min
14	Rapid transfer speed on z axis	18 metre/min
15	X axis travel	200 mm
16	Z axis travel	320 mm
17	Guideway type	Linear guideway
18	Turret type	Gang type
19	Turret tool	Boring bar size 20/20 mm ∅20 mm
20	Weight	1700 Kg
21	Dimensions	1600x1250x1650 mm

Control system and specification

Objectives : At the end of this lesson you shall be able to
- **describe the CNC machine model**
- **explain the control panel**
- **list the control keys and its function**
- **list the address keys and its function**
- **include machine model.**

Fig 1



EMCO CONTROL PANEL

Introduction

The CNC control system of a machine tool includes mainly control unit and the motion control system. The motion control system includes servo motor, drives, axis postioning devices. The control systems are classified

as contouring system, point to point system, closed loop/ open loop system and based on number of axis of the machine.The details of the control unit and the control systems are shown in Fig 1,2&3.

1 Power consumption main motor (relative display)

2 Chuck open/closed

3 Sleeve forwards

4 Sleeve backwards

5 Dummy (Not assigned)

6 Central lubrication manual with display

7 Tool turret, manual operation

8 Coolant On/Off with display

9 Single item for bar feed with display

10

11 Drives On

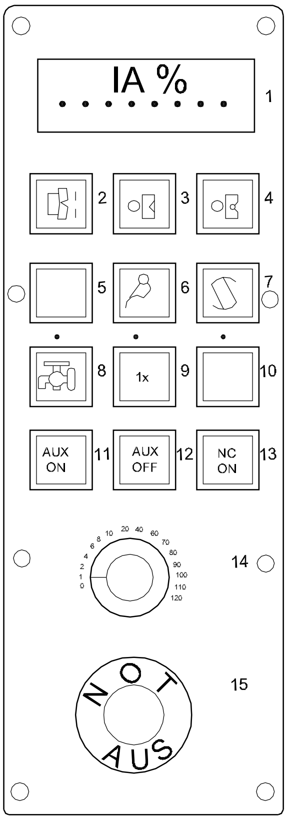
12 Drives off

13 Control On

14 Feed rate override switch 0-120%

15 Emergency-off button

Fig 2

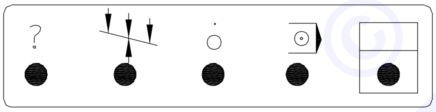


1 POWER CONSUMPTION MAIN MOTOR (RELATIVE DISPLAY)
2 CHUCK OPEN/CLOSED
3 SLEEVE FORWARDS
4 SLEEVE BACKWARDS
5
6 CENTRAL LUBRICATION MANUAL WITH DISPLAY
7 TOOL TURRET, MANUAL OPERATION
8 COOLANT ON/OFF WITH DISPLAY
9 SINGLE ITEM FOR BAR FEED WITH DISPLAY
10 DUMMY(NOT ASSIGNED)
11 DRIVES ON
12 DRIVES OFF
13 CONTROL ON
14 FEED RATE OVER RIDE SWITCH 0-120%
15 EMERGENCY-OFF BUTTON

EMCO CONTROL PANEL

TU20N2611212

Display field



Alarm (red)

The display is bright whenever an alarm occurs. The alarm number is displayed on the screen. The alarms are explained in the alarm list.

Position not yet reached (green)

The display is bright until the set position has been reached.

Feed stop (red)

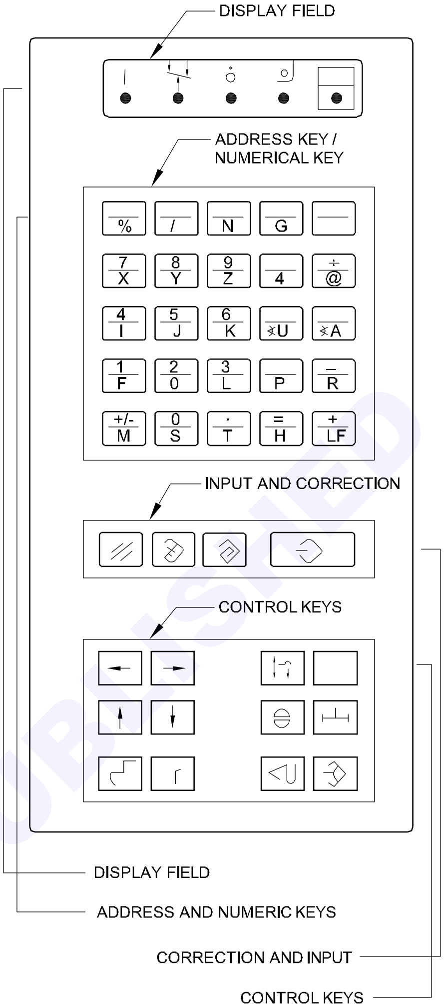
The display is bright if the feed is stopped while the program is running.

Program running (green)

The display is bright until the program has been completed. Even if the machine is not moving.

Keys and display field

Fig 3



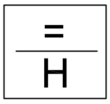
KEYS AND DISPLAY FIELD

TU20N2611213

Key assignment display (yellow)

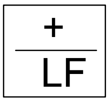
If the "key assignment display is bright, the lower-case function (the bottom function) of a dual-function key is displayed in the input line when the dual function key is depressed. If the "key assignment." Display is not bright, the upper function is entered. Switchover is executed automatically by the controller after a word has been entered in the parts cases, switchover can be executed by means of key in the address keypad.

	%	x)	Start of program
	/	x)	Skip block
	N	x)	Address blocknumber
	G	x)	Address path condition
		Alternate changeover to top and bottom functions. The display is then either bright or dark.	
7	X	Number 7	Address Position data
8	Y	Number 8	Address Position data
9	Z	Number 9	Address Position data
%	Q	x)	Address Position data
	4.	x)	Address Position data
÷	@	Number 4	Address Interpolation parameter
4	I	Number 6	Address Interpolation parameter
5	J	Number 5	Address interpolation parameter
6	K	Number 6	Address interpolation parameter
	B	x)	Address Radius B
	U	unassigned x)	Address Radius U
X	A	Multiplication	Address angle A
1	F	Number 1	Address Feed
2	D	Number 2	Address tool correction
3	L	Number 3	Address sub routine
	P	x)	Address No. of passes
–	R	Subtraction	Address additional function
+/-	M	Sign input line	Address additional function
0	S	Number 0	Address spindle speed
.	T	Decimal point	Address Tool number



Space character

Address Auxiliary function



Additional

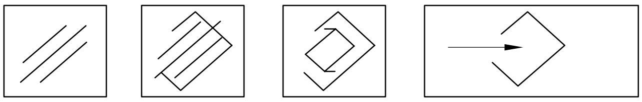
End of block

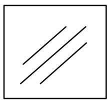
x)

In the input line characters appear which are not normally permitted for normal programming.

The characters a,b,c,d,e,f are required to enter or modify commands in CLB00 machine code.(@----).

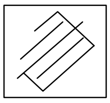
Input and correction





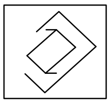
Delete input

The input line is deleted character-by-character. If the key remains depressed the characters entered are deleted in sequence until the input line is empty.



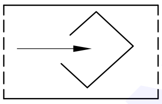
Delete word/block

The word identified by the correction pointer is deleted in the parts program memory if the respective address is in the input line. A complete block is deleted if the respective block number is in the input line.



Modifying a word

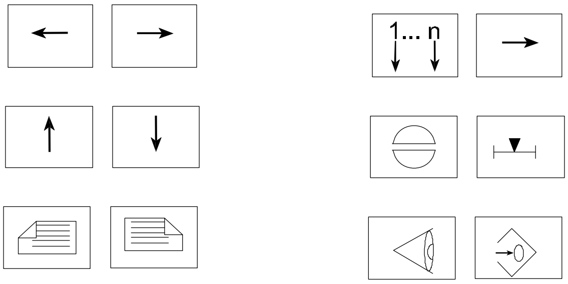
The word identified with the correction pointer is replaced by the word in the input line. The addresses of both words must be identical.

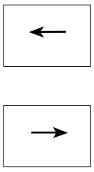


Entering a word

The word in the input line is transferred to the parts program memory, or into list displays or input forms.

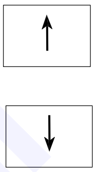
Control keys





Correction pointer left/right

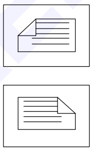
The correction pointer is moved to the left or right word-by-word. The correction pointer jumps from the beginning of the block to the end of the previous block or from the end of the block to the beginning of the following block. If the key remains depressed, the correction pointer goes to the beginning or end of the program.



Correction pointer backwards/forwards

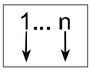
The correction pointer is moved backwards or forwards block-by-block.

If the key remains depressed, the correct pointer goes to the beginning or end of the program.



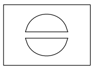
Paging backwards/forwards

The CRT display is paged up or down one page at a time.



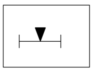
Channel selection switch

The CRT display is switched to the channel selected. Selection is by repeated depressing of the channel selection switch.



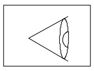
Acknowledging alarms

Alarm form alarm No. 3000 onwards, e.g. "General programming error" alarm can be acknowledged with this key. "Reset alarm is not acknowledged!



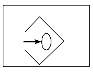
Actual value display selection with double-height characters

If the key is depressed again, the previous display reappears.



Diagnostics and start up

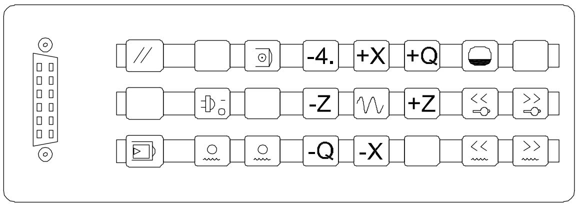
Selection of machine data and switching to start-up state after entering pass word.



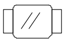
Search key

Searching for a particular address or word in the parts program memory. The expression appearing in the input line when the search key is depressed is looked for. After the search key has been depressed, the correction pointer marks the next expression found.

Integrated machine control panel

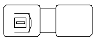


SINUMERIC 810 T



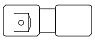
Reset

The running program is aborted Alarms are deleted (up to alarm No. 2999). The controller is brought to the basic state!



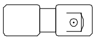
Single block

The parts program is processed block-by-block. Start is by means of "Program START".



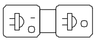
Program STOP

The running program is interrupted The program can be resumed with "Program START".

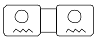


Program START

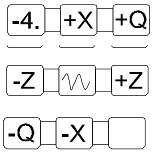
A parts program is started. The functions stored are transferred to the PLC in automatic mode.



Spindle STOP/START

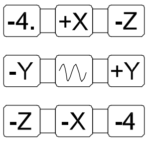


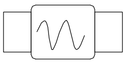
Feed STOP/START



Direction keys

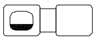
When the direction keys are actuated, the axes are traversed in jog mode.





Rapid traverse override key

If this key is depressed simultaneously with the direction keys the axes are traversed in rapid traverse mode.



Operating mode selection key

The following modes can be selected via a soft key:

PRESET

MDI AUTOMATIC

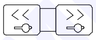
JOG INC 1 TO INC 10,000

(manual encoder)

REPOS

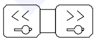
AUTOMATIC

REFPOINT



Spindle speed slower/faster

The programmed spindle speed can be modified in increments of 5 % in the range of 50 % to 120 % . If the key is depressed constantly, the final position is approached in increment of 5 %. The percentage value set is shown in the basic display.

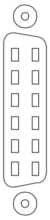


Feed slower/faster

The programmed feed is modified in the following increments from 0% to 120 % 0% 1% 2% 4% 6% 8% 10% 20% 30% 40% 50% 60% 70% 75% 80% 85% 90% 95% 100% 105% 110% 115% 120%.

If the key is depressed constantly the end position is approached. The percentage value set is displayed in the basic display.

In rapid traverse mode the 100% value is not exceed.

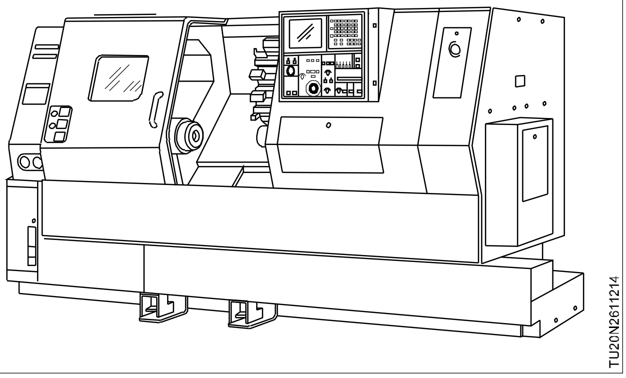


Universal interface port

Machine models

CNC machine are genrally grouped under 2 axes and 3 axis or multi axis system, Horizontal 2 axis (x and 2) are termed as CNC lathe Fig 4 and the vertical CNC machnes (Fig 5) are called as machining centre.

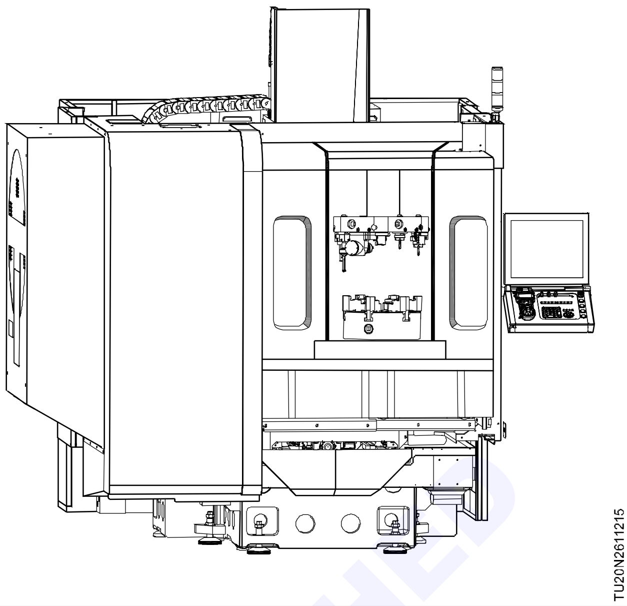
Fig 4



The machine models are also based on the spindle drives namely.

- Separately excited DC shunt motor.
- 3 phase AC induction motor.

Fig 5



The CNC machine models are also based feed drives are

- AC servo motor
- DC servo motor
- Brushless DC servo motor - stepper motor or liner motor.

Specification of CNC turning centre and control system

Objectives : At the end of this lesson you shall be able to
- **specify the CNC turning centre**
- **describe the CNC system system specification.**

Lathe and turning centers

Only basic data for a 2 axes CNC lathe are included

Typical specifications

While the traditional and standard workhorse two axis CNC lather has geen greatly improved by additional features, such as extra axes, sub-spindle, milling attachment, live tools, etc., the most important specifications still apply, regardless of the innovations.

	Metric	Imperial
Swing over bed	∅500mm	∅19.7 in
Swing over cross slides	∅245 mm	∅9.6 in
Normal turning diameter	∅210 mm	∅8.3 in
Maximum turning diameter	∅280 mm	∅11.0 in
Maximum turning length	485 mm	19.0 in
X-axis motion travel (stroke)	210 mm	8.3 in
Z-axis motions travel (stroke)	470 mm	18.5 in
X-axis rapid rate Z-axis rapid rate	30,000 mm/min	1181 in/min
Chuck size	∅200 mm	∅8.0 in
Bar capacity	∅50 min	∅2.0 in
Tool shank (external)	25.4 mm	1.0 in
Maximum cutting feed rate	1265 mm/min	50.0 in/min
Maximum spindle speed	4500 mm/min	
Spindle motor (maximum)	15.0 W	20 HP
Number of tool	12	
Tail stock travel	445 mm	17.5 in
Chip conveyor	Yes	

Fanuc system specifications

The following tables contain typical specifications of fanuc CNC system and are based on the ON.211 and 181 models. Keep in mind that control specifications generally show the maximum capabilities which are not always present for a particular machine tool.

Most of the specifications listed are common to both milling and turning controls, although each control type has features specific to the particular machine tool.

Use the tables as general reference only and consult machine documentaion for exact specifications.

Controlled axes

Programmable axes	3-X, Y, Z 4-X, Y, Z, B 2-X, Z (turning)
Simultaneously controlled axes	3 or 4
Least input increment	0.001 Mm 0.0001 inch 0.001 degree
Max. command value	±99999.999 mm ±9999.9999 in
Fine acceleration & deceleration control	Standard
Inch metric conversion	G20 (imperial) G21 (metric)
Interlock	All axes
Machine lock	All axes / Z-axis
Emergency stop	Standard
Over travel	Standard
Stored stroke check 1	Standard
Mirror image (setting)	Each axis
Programmable mirror image XY axes	M - functions
Backlash compensation	Standard
Pitch error compensation	Standard

Operation functions

Automatic operation	Memory
MDI operation	Standard
DNC operation	Reader -puncher interface required
DNC operation with memory card	PCMCIA card attachment required
Program number search	Standard
Sequence number search	Standard
Buffer register	Standard
Dry run	Standard
Single block	Standard
JOG feed	Standard
Manual reference position return	Standard
Manual handle feedrte	X1, X10, X100
Z axis neglect	standard

Interpolation function

Positioning	G00
Exact stop mode	G61
Exact stop	G09
Linear interpolation	G01
Circular interpolation	G02 (CW) - G03 (CCW) (multi - quadrant standard)
Dwell function	G04
Skip function	G31
Reference position return	G28
Reference position return check	G27
2nd reference position return	G30
Polar coordination interpolation	Optional
Cyclindrical interpolation	Optional
Helical interpolation	Optional
Thread cutting	G32, G90, G76 (turning only)

Feedrate functions

Rapid traverse rate	Standard
Rapid override	F0, 25%, 50%, 100%
Feedrate per minute	G94
Cutting feedrate clamp	Standard
Automatic acceleration and decelaration	Rapid traverse - linear cutting feed = exponential
Feedrate override	0-150% Or 0-200% (10% incr)
Jog override	0-100%
Feedhold	Standard

Spindle functions

Auxillary function lock	Standard
High speed MST interface	Standard
Spindle speed function	Standard
Spindle override	50-120%
Spindle orientation	M19
M-code function	3 digits
S-code fucntion	5 digits
T -code function	3 digits
Rapid tapping	Standar or optional

Tool functions

Tool offset pairs	16-digit 32 pairs
Tool length offset	G43, G44, G49
Cutter radius offset type	G40, G41, G42
Maximum number of tool offsets	999 sets (18i) 400 sets {0i and 21i)
Direct input of offset value	standard

Program input

EIA - ISO automatic recognition	Standard
Label skip and parity check	Standard
Control in / out (comments)	{ }
Optional block skip	/
Maximum dimensional input	±8digits
Program number	0 (4 digits)
Sequence number	N (5 digits)
Absolute programming incremental programming	G90 (XZ for turning) G91 (UW for turning)
Decimal point input	Standard
Calculator type input	Available
Plane selection	G17, G18, G19
Rotary axis designation	Standard
Coordinate system setting	Standard
Auto coordinates system setting	Standard
Work coordinate system	G52, G53, G54-G59
Manual absolute ON and OFF	Standard
Subprogram call	M98 - 4 nested levels

Circular interpolation with R	Standard
Program format	Fanuc
Program stop functions	M00 - M01
Program end fucntions	M02 M30
Reset	Standard
Programmable data input	G10, G11
Custom macro B	G65 (option)
Fixed cycles	G73, G74, G76, G80-G89 G98, G99
Multiple repetitive cycles (turning controls only)	G70, G71, G72, G73, G74, G75, G76
Corner rounding (turnng only)	Standard

Editing functions

Program storage capacity (max.)	640m (0i/21i) 1280 (18i)
Maximum number of registered programs	400 (0i/21i) 1000 (18i)
Program editing	Standard
Program protection	Standard
Background editing	standard

Setting and display fucntions

Status and current position	Standard
Program name	31 characters max.
Parameter setting and display	Standard
Self diagnosis function	Standard
Alarm and alarm history	Standard
Operation history display	Standard
Help function	Standard
Run time and parts count	Standard
Actual cutting feedrate display	Standard
Display of S and T code	Standard (all screens)
Servo setting screen	Standard
Multi -language display	English (default)
Spindle speed indicator	Standard
Spindle load indicator	Standard
Data protection key	Standard
Graphic function	Standard

Clock function	Standard
Dynamic graphic display	Standard
Display unit - color LCD	8.4" (0i) 10.4" (21i/18i)

Data input / output

Reader / puncher interfae	RS-232 interface
Memory card interface	Standard
External part number search	9999

Options

Only some sofware options are listed in the table

Additional work coordinate systems	48 sets or 300 sets
Extra custom macro common variables	Up to 999
Tool life management	
Additional tool life management sets	S12
Automatic corner override	
Automatic corner deceleration	
Coordinate system rotation	
Feedrate clamp by arc radius	
Hypothetical axis interpolation	
Custom macro interruption	
Clerk control	
Polar coordinate interpolation	
Program restart	
Increment system multiplie by 1/10	0.0001 mm 0.00001 inch
Scalling function	G50, G51
Small diameter pack drilling cycle	
Smooth interpoaltion	
Ciruclar threading	
3D coordinate coversion	
3D cutter radius offset	
Load monitoring	
Position compensation (for backward compatibility only)	G45-G48
Tool retract and recover	

Axes convention of CNC machines

Objectives : At the end of this lesson you shall be able to
- **list the different types of CNC lathe**
- **describe the axes designation CNC lathe**
- **explain the right hand thumb rule for CNC axes designation.**

Types of CNC machines

There are many different types of CNC machines are used in industry, the majority of them are CNC machining centres and CNC lathes. Here we will discuss about CNC lathe

Types of CNC Lathes

Basically, CNC lathes can be categorized by the type of design and by the number of axes. The two basic types are a vertical CNC lathe and a horizontal CNC lathe. Of the two, the horizontal type is by far the most common in manufacturing and machine shops. A vertical CNC lathe is for a large diameter work. For CNC programmer, there are no significant differences in programming approach between the two lathe types

A typical horizontal CNC lathe can further be described by the type of engineering design:

FRONT lathe ... an engine lathe type

REAR lathe ... a unique slant bed type

Slant bed type is very popular for general work, because its design allows cutting chips to fall away from the CNC operator and, in case of an accident, forces the part to fall down into a safe area, towards chip conveyer

Axes designation

A typical CNC lathe is designed with two standard axes - one axis is the X-axis; the other axis is the Z-axis. Both axes are perpendicular to each other and represent the typical two-axis lathe motions. X-axis also represents cross travel of the cutting tool; Z-axis represents its longitudinal motion. All varieties of cutting tools are mounted in a turret (a special tool magazine) and can be external or internal. Because of this design, a turret loaded with all cutting tools moves along both X and Z axes, which means all tools are in the work area at all times.

In CNC lathe work, the traditional axis orientation for a horizontal type of lathe is upwards and downwards motion for the X-axis, and left and right motion for the Z-axis, when looking from the machinist's position. This view is shown in the following three illustrations Figure 1,2 &3

Typical configuration of a two axis slant bed CNC lathe - rear type

Typical configuration of a CNC lathe with two turrets

Schematic representation of a vertical CNC lathe

Fig 1

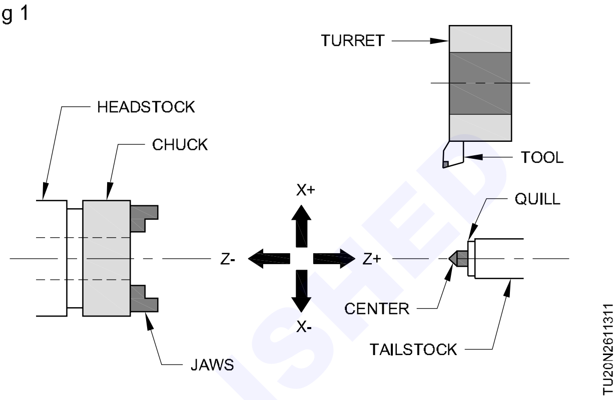


Fig 2

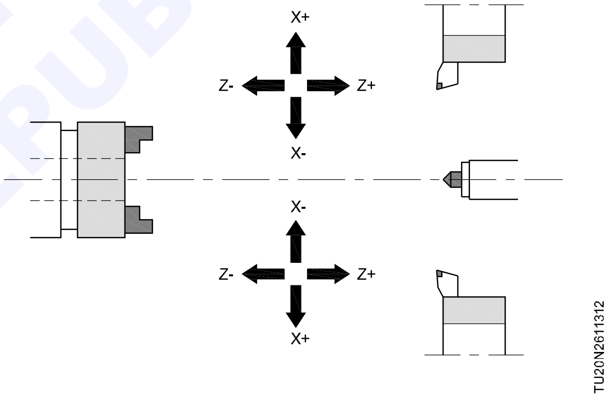
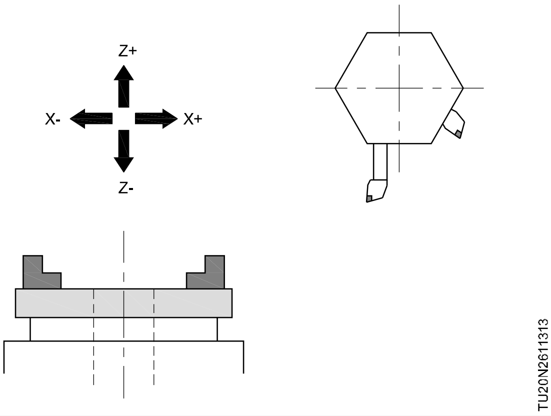


Fig 3



Two-axis Lathe

This is the most common type of CNC lathes. The work holding device, usually a chuck, is mounted on the left side of the machine (as viewed by the operator). Rear type, with slant bed, is the most popular design for general work. For some special work, for example in the petroleum industry (where turning tube ends is a common work), a flat bed is usually more suitable. Cutting tools are held in a specially designed indexing turret that can hold four, six, eight, ten, twelve and more tools. Many such lathes also have two turrets, one on each side of the spindle centre line

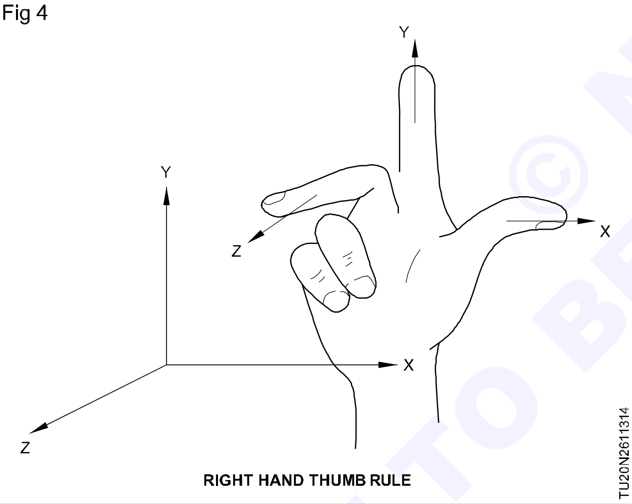
Three-axis Lathe

Three-axis lathe is essentially a two-axis lathe with an additional axis. This axis has its own designation, usually as a C-axis in absolute mode (H-axis in incremental mode), and is fully programmable. Normally, the third axis is used for cross-milling operations, slot cutting, bolt circle holes drilling, hex faces, side faces, helical slots, etc. This axis can re place some simple operations on a milling machine, reducing the setup time for the job. Some limitations do apply to many models, for example, the milling or drilling operations can take place only at positions projecting from the tool centre line to the spindle centre line (within a machining plane), although others offer off-centre adjustments

Axis - nomenclature

The basic designation of the axis (i.e.), in Fig 3 which is X, Y, Z, is decided by the right hand thumb rule and the main spindle axis. The thumb indicates X - axis, fore finger indicates Y - axis and the middle finger indicates Z - axis.

Fig 4



RIGHT HAND THUMB RULE

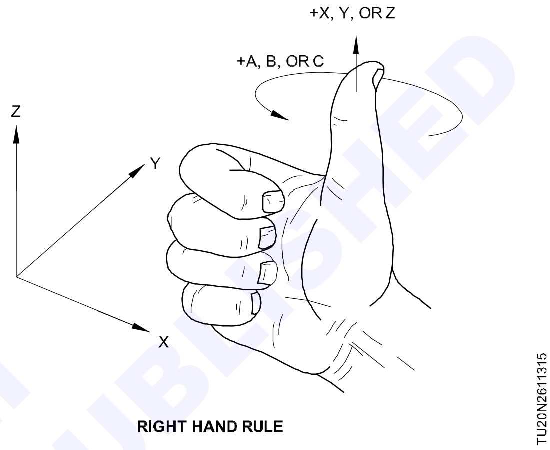
Auxiliary axes on NC machine

Apart from each side movement axes on the machine, some other auxiliary axes can exist. E.g. Rotary table. This rotary table axis is designed as A axis if it is parallel to X direction. Similarly, B and C axes for Y and Z respectively.

Right hand rule

The rotary movements about X, Y and Z are designated as A, B and C respectively. The right hand rule is used to define the positive direction of the coordinate axes as per the Fig 4.

Fig 5



RIGHT HAND RULE

Feedback control system and interpolations

Objectives : At the end of this lesson you shall be able to
- **explain about feed back system**
- **state the closed loop and open loop control**
- **brief and list about interpolation**
- **explain various interlocks used in CNC lathe.**

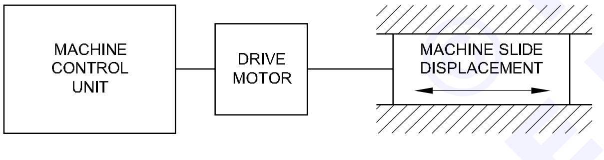
Feed back system

- The feedback system is also referred to as the measuring system.
- It uses position and speed transducers to continously and monitor the position at which the cutting tool is located at any particular instant.
- The MCU uses the difference between reference signals and feedback signals to generate the control signals for correcting position and speed errors. (Fig 1)

Open loop control system (Fig 1)

In an open loop control system (Fig 1) in which there is no arrangement for detecting or comparing the actual position of the cutting tool on the job with the commanded value.

Fig 1



Therefore, this system is not providing any check to see that the commanded position has actually been achieved. There is no feed back of information to the control also. These system are not good where extremely accurate positioning is required.

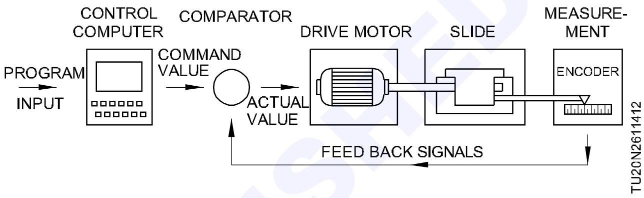
Closed loop control (Fig 2)

Closed loop control (Fig 2) is a term which is used very often when we talk about CNC machines. This term signifies, that the control system has provisions to ensure that the tool reaches the desired position, at the correct feed rate, even if some errors creep in due to unforeseen reasons.

For instance in the previous example 60,000 pulses sent in 2 minutes by the control should cause a tool travel of 60 mm at 30 mm/min, but even if the control sends these may pulses it cannot be ensured that the tool has really travelled exactly 60 mm.

A closed loop control has a device called encoder and this can continuously ascertain the distance actually travelled by the tool and then monitor the same, in the form of feedback signals to the control. The control studies this feedback information and takes corrective action in case any error is detected in the tool position/ feed rate.

Fig 2



Interpolation

As the co-ordinates of points on the profile of the job vary continuously, it is necessary to define the path of small segment.

This tedious work is done by the computer by means of "interpolator".

Definition

The methods by which control system calculate the intermediate points and the speed of the motor is known as interpolation.

The parameters supplied may be

1 Radius

2 Start and end point of a curve

3 Radius and centre of a circle

4 Gradient angle for a line.

Types of interpolations

Interpolations are classified as

1 Linear interpolation

2 Circular interpolation

3 Helical interpolation

4 Parabolic interpolation

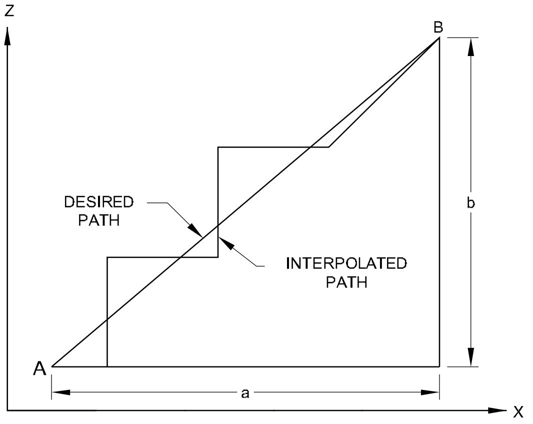
5 Logarithmic interpolation

6 Exponential interpolation of these the linear and circulate interpolators are commonly employed.

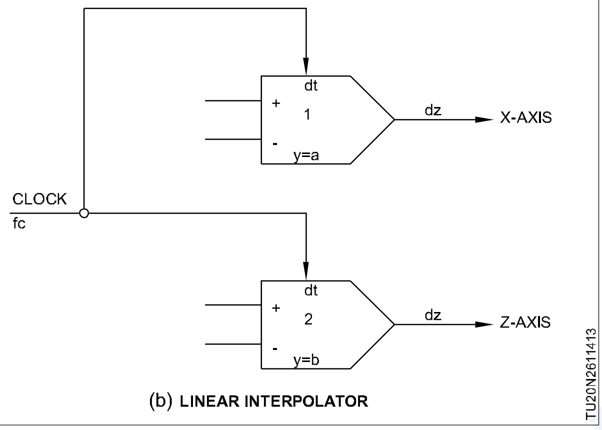
Linear interpolation

In this interpolation, the interpolated points lie on the straight line joining a pair of given points. (Fig 3)

Fig 3



(a) STRAIGHT LINE INTERPOLATION



(b) LINEAR INTERPOLATOR

This is done in two or three dimensions.

The fast of linear interpolator is to supply velocity commands to several axes simultaneously in pps (pulses per second) (Fig 3b)

By changing the frequency of the pulses, the feed can be controlled.

The linear interpolator consists of Digital Differential Analyser (DDA) integrators one for each axis of motion hence each integrator functions separately one for X-axis and the other for Y - axis. (Fig 4a)

Circular interpolation

In circular interpolation, the interpolated points lie on a specific circle between a pair of fixed points.

In most cases, the circular interpolation is limited to one quadrant in the machine tool system. (Fig 4b)

The input data should consists of the distances between the initial point and the centre of the circle.

Two Digital Differential Analysers (DDA) are required for circular interpolation.

Advantages of circular interpolation are

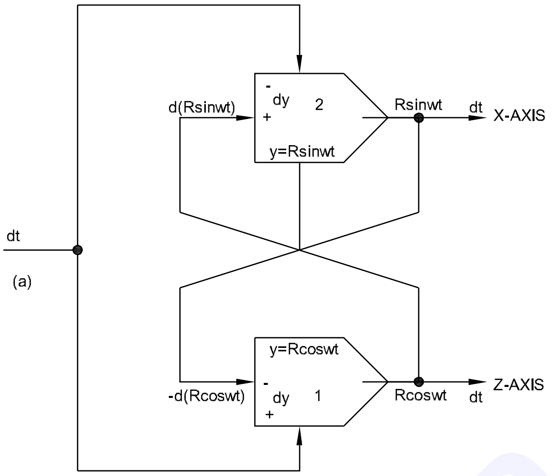
Better surface finish

Greater accuracy

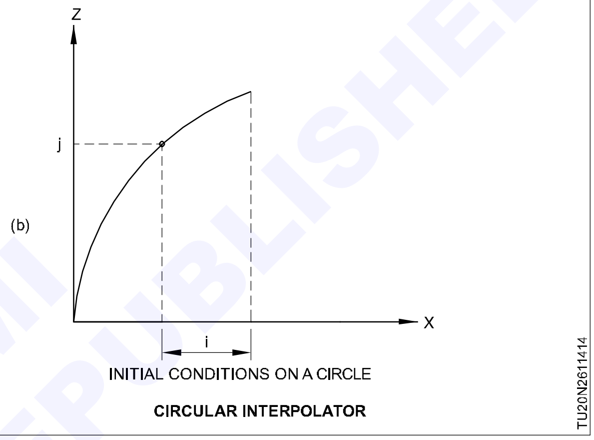
Less total machining time

Lower working costs.

Fig 4



LINEAR INTERPOLATOR



CIRCULAR INTERPOLATOR

Circular interpolation

The circular interpolation

Code G02 (clockwise)

Code G03 (anti - clockwise)

A circular interpolation permits the traversing of the tool with a defined speed along a circular path from the present Start-points to the programmed destination point.

Apart from the destination points co-ordinates, the control unit here also needs statements about the sense of rotation and the centre of the circle. The centre is entered with I, J and K with incremental dimensions with the centre points as origin.

The following assignment applies

I for the X - axis

K for the Z - axis

Circular Interpolation with mixed programming

Particularly the incremental statement of the centre of the circle usually represents some difficulties to the operator in practice, since it must often be evaluated using triangle calculations.

This is a prime example of where the mixed co-ordinate programming of the interpolation parameters in absolute dimensions comes in useful.

Concepts of Coordinate geometry

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

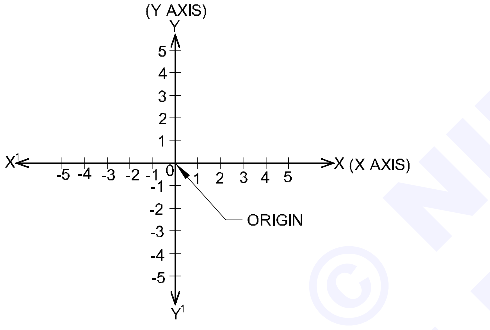
- **brief the concept of coordinate geometry**
- **describe the machine geometry**
- **state about Machine zero, Work zero and Reference point in CNC machine.**

Coordinate geometry is one of the important concept in mathematics. Coordinate Geometry is one of the new branch of mathematics to representation of a point on a plane with idea of two references.

The concepts of Co-ordinate Geometry were developed by Rene Descartes. He is a French mathematician and philosopher and he is established an association between algebraic equations and geometric curves and figures.

In Fig 1 we draw a vertical number line and horizontal number line meeting at a point perpendicular to each other. The intersection point is denoted as origin.

Fig 1



The horizontal number line XX^1 is known as X-axis and the vertical number line YY_1 is known as Y-axis.

The point where XX^1 & YY^1 intersecting each other is called the origin, and is denoted by 'O'.

In this plane the positive numbers lie on the directions OX is called the positive direction of the X-axis, similarly OY is the positive Y-axis respectively. Also the negative numbers lie on the directions OX^1 is called the negative directions of the X-axis, similarly OY^1 is the negative Y-axis respectively.

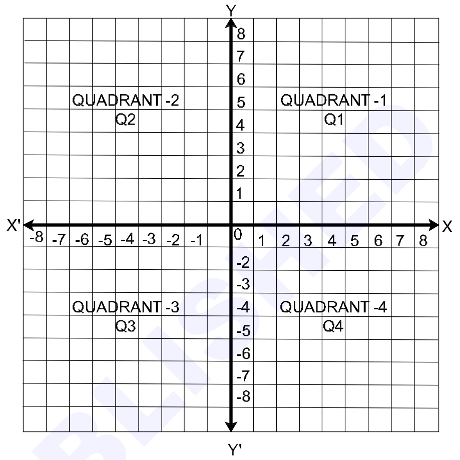
The plane here is known as the Cartesian plane or co-ordinate plane or XY-plane. The X & Y axes are known as coordinate axes.

The coordinate plane is divided into four parts by these coordinate axes. (fig 2) These four parts are called the quadrants and are denoted by Quadrant -1 (Q1), Quadrant -2(Q2), Quadrant -3(Q3) & Quadrant -4(Q4) in anti-clockwise direction.

Coordinates of a Point (or) Locating of coordinate points

A quadrant also defined as a part of a Cartesian or coordinate plane obtained when the two axes intersect each other.

Fig 2



Quadrant -1 (Q1) (+x, +y)

Quadrant -2 (Q2) (- x, +y)

Quadrant -3 (Q3) (-x, -y)

Quadrant -4 (Q4) (+x, - y)

X coordinate - The X-coordinate of a point is the distance from origin to foot of perpendicular on X-axis. The x-coordinate is also known as the abscissa.

Y coordinate - The Y-coordinate of a point is the distance from origin to foot of perpendicular on Y-axis. The y-coordinate is also known as the ordinate.

In the coordinate system, origin as a reference point to locate other points in a plane.

A coordinate is states the locate a point in two-dimensional space. The coordinates of a point are shown as (x, y). (Fig 3)

Coordinates of Origin: The coordinates of the origin "O" are denoted as a (0,0).

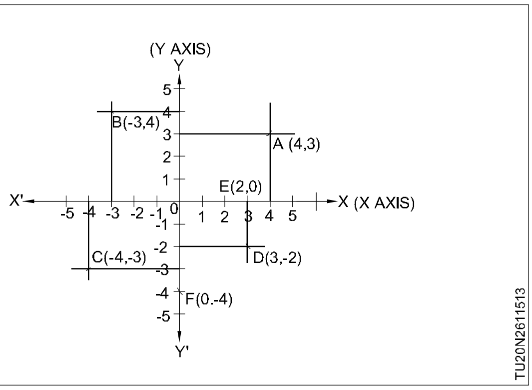
The point "A" is at a distance of 4 units measured along positive point of X-axis from origin. The same point is at a distance of 3 units measured along positive point of Y-axis from origin.

The x-coordinate (abscissa) of A is 4 & The y-coordinate (ordinate) of A is 3.

Hence the coordinates of A are (4,3)

The x-coordinate (abscissa) of B is -3 & The y-coordinate (ordinate) of B is 4.

Fig 3



Hence the coordinates of B are (-3,4)

The x-coordinate (abscissa) of C is -3 & The y-coordinate (ordinate) of C is -4.

Hence the coordinates of C are (-4,-3)

The x-coordinate (abscissa) of D is 3 & The y-coordinate (ordinate) of D is -2.

Hence the coordinates of D are (3, -2)

The point "E" is at a distance of +2 units from the Y-axis and at a distance zero from the X-axis. Therefore, the x-coordinate of "E" is 2 and y-coordinate is 0.

Hence the coordinates of "E" are (2,0).

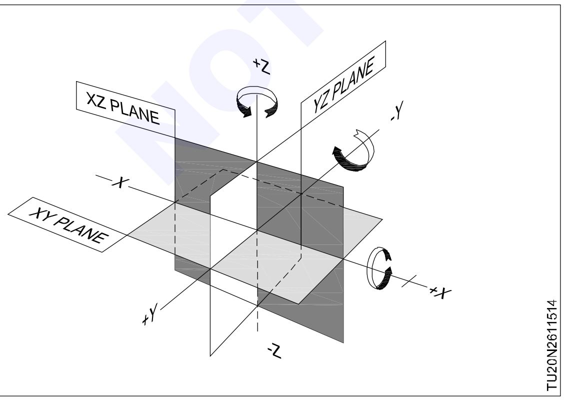
The point "F" is at a distance of -4 units from the X-axis and at a distance zero from the Y-axis. Therefore, the x-coordinate of "F" is 0 and y-coordinate is -4.

Hence the coordinates of "F" are (0, -4)

Machine geometry

Machine geometry defines the relationship of distances and dimensions between fixed point of the machine and selectable point of the part. Typical geometry of CNC machines uses the right hand coordinate system. Positive and negative axis direction is determined by an established viewing convention. The general rule for Z-axis is that it is always the axis along which a simple hole can be machined with a single point tool, such as a drill, reamer, wire, laser beam, etc. Fig 4 on the next page illustrates standard orientation of planes for XYZ type machine tools

Fig 4



Axis Orientation - Turning

Standard CNC lathes have two axes, X and Z. More axes are available, but they are not important at this point. Special additional axes, such as C-axis and Y-axis, are designed for milling operations (live tooling) and require unique version of a standard CNC lathe.

What is much more common for CNC lathes in industry, is the double orientation of XZ axes. CNC lathes are separated as front and rear lathes. An example of a front lathe is similar to the conventional engine lathe. All slant bed lathe types are of the rear kind. Identification of axes in industry have not always followed mathematical principles (Fig 5,6& 7)

Fig 5

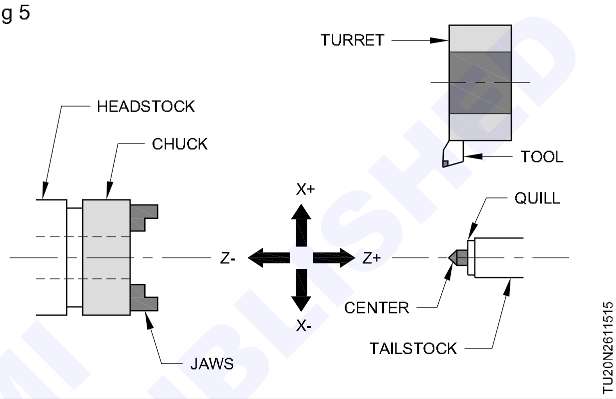


Fig 6

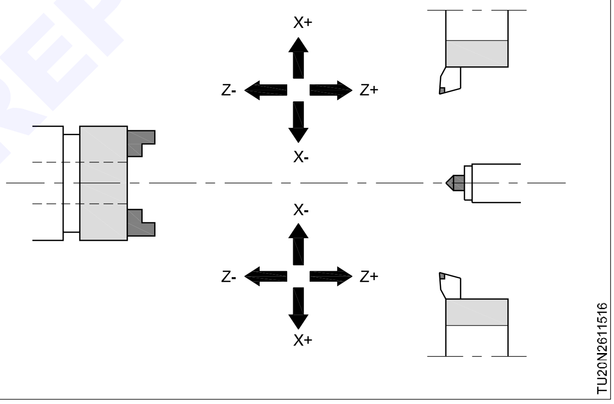
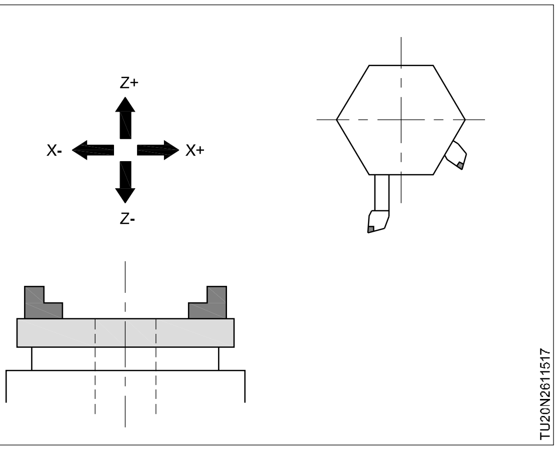
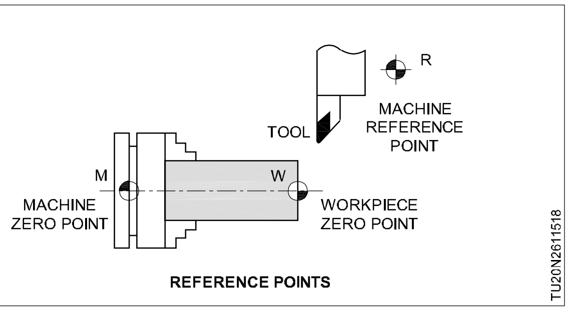


Fig 7



Cartesian coordinate system is used to describe the position of a 2D or 3D point in space. In Polar coordinate system a point is located by its distance to the point of origin and its angle to a specified axis. CNC Lathe Machine's coordinate system is used to ensure that machine is able to read the assigned coordinates correctly to indicate the position of the work piece. Key parts of the system are (Fig 8)

Fig 8



- Machine Zero point (M)
- Work piece Zero Point (W)
- Reference point (R)

Machine zero point

Machine Zero Point is the origin of the coordinate system which is defined by the manufacturer. They can't be changed. It is located in the centre of the work spindle nose for CNC lathes.

Work zero point

Work piece zero point is the origin of the work part based coordinate system. Its location is specified by the programmer.

Reference point

Reference point of any CNC machine has been selected at a specific fixed point during the initial machine design, by the machine design engineers. It is a fixed point, located within machine travel limits, and its actual position does not normally change. This point (position) is typically called the machine reference point, or simply - the home position.

Coordinate system on CNC machine

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes. When two program axes, the X-axis and Z-axis, are used, coordinates are specified as follows

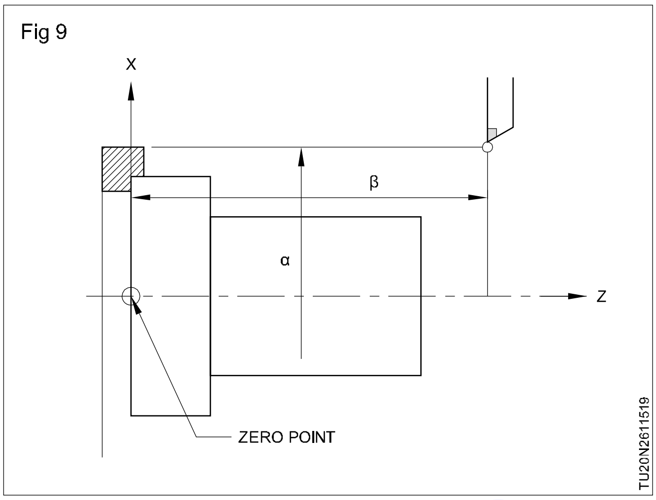
X_Z_

This command is referred to as a dimension word

Coordinates are specified in one of following three coordinate systems:

- Machine coordinate system
- Work piece coordinate system
- Local coordinate system

Fig 9



The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP_

Machine coordinate system

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine. A coordinate system with a machine zero-point set as its origin is referred to as a machine coordinate system. A machine coordinate system is set by performing manual reference position return after power-on. A machine coordinate system, once set, remains unchanged until the power is turned off.

Workpiece coordinate system

A coordinate system used for machining a work piece is referred to as a work piece coordinate system. A work piece coordinate system is to be set with the NC beforehand (setting a work piece coordinate system). A machining program sets a work piece coordinate system (selecting a work piece coordinate system). A set work piece coordinate system can be changed by shifting its origin (changing a work piece coordinate system)

Setting a Work piece Coordinate System

A work piece coordinate system can be set using one of three methods:

- Method using G code A work piece coordinate system is set by specifying a value after G code in the program.
- Automatic setting If bit 0 of parameter No. 1201 is set beforehand, a work piece coordinate system is automatically set when manual reference position return is performed.
- Input using the MDI panel Make settings on the MDI panel to pre-set six work piece coordinate systems

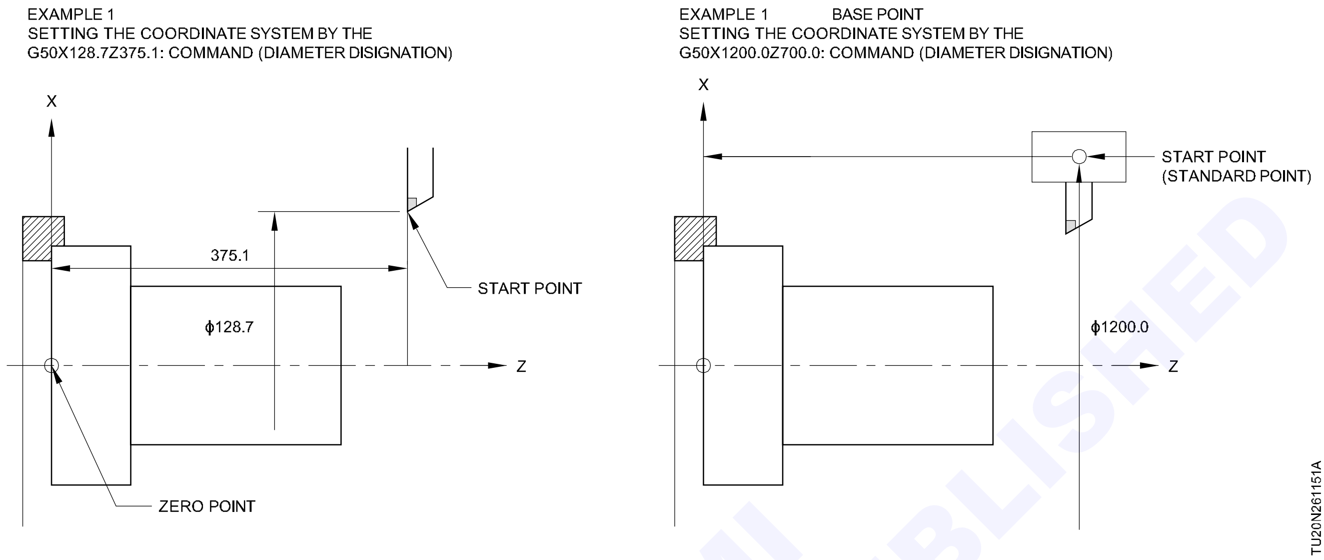
Then, use program commands G54 to G59 to select which work piece coordinate system to use. When an absolute command is used, a work piece coordinate system must be established in any of the ways described above.

Setting a work piece coordinate system by G50

G50 IP_

A work piece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates. If IP is an incremental command value, the work coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. If a coordinate system is set using G50 during offset, a coordinate system in which the position before offset matches the position specified in G50 is set.

Fig 10



Selecting a Work piece Coordinate System

The user can choose from set work piece coordinate systems as described below. (For information about the methods of setting,

1 G50 or automatic work piece coordinate system setting Once a work piece coordinate system is selected, absolute commands work with the work piece coordinate system.

2 Choosing from six work piece coordinate systems set using the MDI

By specifying a G code from G54 to G59, one of the work piece coordinate systems 1 to 6 can be selected

G54 Work piece coordinate system 1

G55 Work piece coordinate system 2

G56 Work piece coordinate system 3

G57 Work piece coordinate system 4

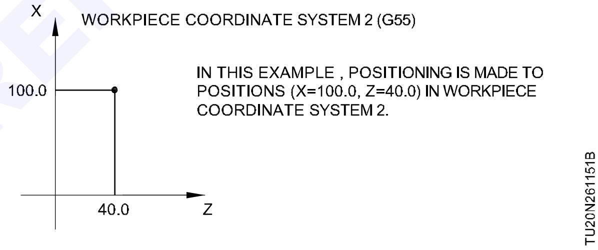
G58 Work piece coordinate system 5

G59 Work piece coordinate system 6

Work piece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected.

Examples

Fig 11



Programming, Zero Points

Zero Points can also be called Zero Shifts, Zero Offsets or Datum Points.

Zero Points allow parts to be programmed without regard for their position on the machine or to machine more than one part on the machine without re-coding all the points. If individual one batch parts are to be machined then the zero point may be set when the machine is set up, if parts are repeated, for example a quantity per month for a period of time then most controls allow zero points to be stored in the program (an example follows) which combined with fixtures dowel pinned to the machine bed allow for much faster set up times.

Programming sequence, ISO G codes and M codes

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

- **state about the CNC program**
- **list the three types of CNC program**
- **explain the ISO G and M codes**
- **describe the structure of the CNC program**
- **brief the absolute and incremental methods of programming.**

Program

A CNC program is a sequence of codes and data that tells the CNC machine what to do. The programmed codes, along with the right tooling in a CNC machine centre, allow for correct and repeatable part manufacturing.

Types of CNC programming

There are three basic CNC machine programming methods - manual, conversational, and CAM system programming

Manual programming, the operator inputs code manually, which can be time consuming and some what tedious. It also requires intimate knowledge of the programming language. However, manual programming is a valuable foundational skill set to have on hand, and it's an effective way to make quick modifications to CAM outputs and optimize code.

Conversational or "shop-floor" programming is an increasingly popular method that involves graphic and menu-driven functions. Operators input commands directly into the CNC machine, just like in manual programming, which alleviates potential issues that may arise from poor post proessors in CAM.

CAM system programming is similar to the conversational method, but more advanced. With a CAM system, the software provides an advanced GUI with tool path strategies to help the operator prepare and generate the program eliminating the need for any manual programming - and transfer it directly to the CNC machine.

All of these CNC machine programming methods involve G-code and M-code. G-code is the programming language that instructs the CNC machine what to do, facilitating accurate and repeatable parts, M -code controls all of the misscellaneous CNC machine functions, such as spindle rotation start and stop. M-codes are customizable and vary by machine, so operators and programmers must double check they're inputting the right functions before they start machining.

Preparation of part programming

1 Block numbers / sequence number (N words)

Each block of the program has a sequence number which is used to identify the sequence of a block of data in it which in ascending numerical order. When the part program is read from the tape, each sequence number is displayed on the panel of NC machine tool, as long as that block commands are performed. This enables the operators to know which sequence of block is being performed practically by the tool. It consists of a character 'N' followed by a three digit number raising from '0' to '999'.

2 Preparatory Function (G-words)

The preparatory function is used to initiate the control commands, typically involve a cutter motion i.e. It prepares the MCU to be ready to perform a specific operation and interpret, the data which follows the way of this function. It is represented by the character 'G' followed by a two digit number i.e.'00 to 99'.These codes are explained and listed separately.

3 Dimension words (X, Y & Z words)

These dimension word are also known as 'co-ordinates'. Which give the position of the tool motion .These words can be of two types:

a) Linear dimension words

- X, Y, Z for primary or main motion.
- U, V, W for secondary motion parallel to X, Y, Z axes respectively.
- p, q, r for another third type motion parallel to X,Y,Z axes respectively.

b) Angular Dimension Words

- a, b, c, for angular motion around X, Y, Z axes respectively.
- I, J, K in case of thread cutting is for position of arc centre; thread lead parallel to X, Y, Z axes.

These words are represented by an alphabet representing the axes followed by five or six digits depending upon the input resolution given. The following points may be noted while calculating the number:

- Decimal point should be not be allowed e.g. x = 7.875 will be represented as X07875 in a five system i.e. the last three digits are used for the decimal part of the number. Some machines allow omission of leading zeros, hence the same can be represented as X7875.

- It is recommended that dimensions should be expressed in mm.
- All angular dimensions should be expressed as a decimal fraction of a revolution.
- In absolute system, all dimensions should be positive.
- In incremental system the '+','-'sign represent the direction of motion.

4 Feed Rate Word (F - word)

It is used to program the proper feed rate, to be given in mm/min or mm/rev as determined by the prior 'G' code selection G94 and G95 respectively. This word is applicable to straight line or contouring machines, because in PTP systems a constant feed rate is used in moving from point to point.

It is represent by "F" followed by three digit number e.g.F100 represents a feed rate of 100 mm/min.

5 Spindle speed / cutting speed word (S - word)

It species the cutting speed of the process or the rpm of spindle. It is also represented by 'S' followed by the three digit number .If the speed is given in meter per min. then the speed is converted in rpm rounded to two digit accuracy, e.g. S-800 represents the 800 rpm of spindle.

6 Tool selection word (T - word)

It consists of "T" followed by max five digits in the coded number. Different numbers are used for each cutting tool. When the "t" numbers read from the tape ,the appropriate tool is automatically selected by ATC(Automatic tool changer).Hence this word is used only for machines with ATC or programmable tool turret .e.g. T01,T02,T03 ………….. represents the tool selection word. Also, sometimes T-word used for representing a tool offset number corresponding to X Y and Z directions. With the help of two additional digits, given after a decimal point .(In HMT T-70,9 pairs of tools offset can be stored).

7 Miscellaneous words (M-words)

It consists of character M followed by two digit number representing an auxiliary function such as spindle ON/OFF ,coolant ON/OFF or rewinding the tape. These functions do not relate two dimensional movement of the machine. This is more explained in next topic.

8 End of Block (EOB)

It identifies the end of instruction block.

G and M codes (G-codes)

This is the preparatory function word, consists of the address character G followed by a two digit code number, known as G-code. This comes after the sequence number word and a Tab Code. There are two types of G codes modal and non-modal. Modal codes remain active until cancelled by a contradictory and code of same class .e.g. G70 is a modal code which defines that the dimensional units are metric. It will remain active until cancelled by G-71,which tells that the dimensional units are in inches now. Non-modal g codes are active only in the block in which they are programmed.G04 is non-modal code.

List of G codes

Code	Group	Description
*G00	01	Rapid traverse
G01	01	Linear interpolation
G02	01	CW circular interpolation
G03	01	CCW circular interpolation
G04	00	Dwell time
G10	00	Offset setting by program
G20	06	Inch data input
G21	06	mm data input
G27	00	Reference point (Home) return check
G28	00	Reference point (Home) return
G30	00	Return to second reference point(Home)
G32	01	Thread cutting
G34	01	Variable lead thread cutting
*G40	07	Tool nose radius compensation cancel
G41	07	Tool nose radius compensation left
G42	07	Tool nose radius compensation right
G50	00	Work coordinate change / maximum spindle speed setting
G54-G59	14	Work piece coordinate system (G54 is default)
G70	00	Finishing cycle
G71	00	Multiple turning cycle (Stock removal in turning)
G72	00	Multiple facing cycle (Stock removal in facing)
G73	00	Pattern repeating cycle
G74	00	Peck drilling cycle
G75	00	Grooving cycle
G76	00	Multiple threading cycle
G90	01	Single turning cycle
G92	01	Single threading cycle
G94	01	Single facing cycle
G96	02	Constant surface speed
*G97	02	Constant RPM
G98	05	Feed per minute
*G99	05	Feed per revolution

List of M Codes	
Code	**Description**
M00	Program stop
M01	Optional stop
M02	End of program execution
M03	Spindle forward(CW , as viewed towards the tail-stock)
M04	Spindle reverse (CCW, as viewed towards the tail-stock)
M05	Spindle stop
M06	Auto tool change(not needed on recent controls)
M08	Coolant on
M09	Coolant off
M10	Chuck open (for machines with automatic chuck)
M11	Chuck close
M13	Spindle forward and coolant on / sub-spindle on
M14	Spindle reverse and coolant on/sub off
M19	Spindle orientate
M25	Quill extend
M26	Quill retract
M29	DNC mode
M30	Program reset and rewind
M38	Door open (for machines with automatic door)
M39	Door close
M40	Parts catcher extend
M41	Parts catcher retract
M43	Swarf conveyor forward
M44	Swarf conveyor reverse
M45	Swarf conveyor stop
M48	Lock feed and speed at 100%
M49	Cancel M48 (default)
M52	Threading pull out angle=90° (default)
M53	Cancel M52
M56	Internal chucking
M57	External chucking
M62	Auxiliary output-1 on
M63	Auxiliary output-2 on
M64	Auxiliary output-1 off
M65	Auxiliary output-2 off
M66	Wait for input -1
M67	Wait for input-2
M68	Turret indexing (tool changes) only at home position
M69	Turret indexing anywhere
M70	Mirror in X on
M76	Wait for -1 to go low
M77	Wait for input-2 to go low
M80	Mirror in X off
M98	Subprogram call
M99	Return to the calling program

List of G codes

G codes are instructions describing machine tool movement. A G code quite often requires other information such as feed rate or axes coordinates The FANUC standard has a large selection of G codes, all of which may not be available on all the machines. There are three G code system: A, B and C. System A is the most commonly used. Following is the list of some common G codes of system A:

When the power is turned ‘ON’ or ‘Reset button’ is pressed, the ‘G’ codes with * mark become active.

List of M codes

The list given below is a typical representative list .All of these may not be available on all the machines. On the other hand, some machine may use some extra code also. Note that most of the M codes, except a few such as M00, M01, M02, M03, M04, M05, M06, M08, M09, M19, M30, M98 and M99,are machine specific. Refer to the specific machine manual for the list of available M codes and their functions. M codes are defined and implemented by the machine tool builder. The control manufacture defines only G codes which are same on all the machines with the same control.

Part program

A set of commands given to the NC for machine motion is called a program. A program is composed of number of Blocks. Part program is use to specify the machining process for the cutting tools.

Example

O1203;

N1;

G28 U 0.0 W 0.0;

G50 S 1200 T 0300;

________________ ;
________________ ;
________________ ;

M01;

Part program

N2;

G28 U 0.0 W 0.0;

G50 S 1200 T 0200;

________________ ;
________________ ;
________________ ;
________________ ;

M01;

M30;

Part program

Decimal point input

Decimal point is used to input the units like Distance, Time, and Angle .

X 25.0 is use for input the distance value . X25.0 equal to 25mm or 25 inch.

G04 X1.0 is used to input the dwell time value.X1.0 is equal to one second.

A45 is used for input the angle value.A90 is equal to 45°

The following are the same meaning, in the case of decimal point.

X20.
X20.0 All are same meaning of
X20.00 movement of X 20 mm
X20.000

If the Decimal point is eliminated. The system read in microns.

X 50 = 0.05mm

X 500 = 0.5mm

X 5000 = 5.0mm

Decimal point can be inputted for the following addresses. X, Z, U, W, A, B, C, I, J, K, P, R, Q, F.

Note

1 micron=0.001mm

1 mm=1000 microns

1 inch=25.4mm

1 sec=1000 millisec

Structure or format of a part program

The complete part program for a given component consists of a beginning code of %.A part program consists of large number of blocks each representing an operation to be carried out in the machining of the part. The words in each block are usually given in the following order.

- Sequence number(N-word)
- Preparatory word(G-word)
- Coordinates (x-,Y-,Z- words for linear axes; A-, B-, C- words for rotational axes)
- Feed rate (F-word)
- Spindle speed(S-word)
- Tool selection(T-word)
- Miscellaneous command (M-word)
- End -of-block(EOB symbol)

The structure of part program used in Fanuc controller is given below.

```
          %              (Program start)
          O3642          (Program number)

Blocks    N010 ---------
          -----------------
          -----------------
          N100 M02;      (Program end)
```

Program number

Each of the program that is stored in the controller memory requires an identification. It is used while running and editing of the programs directly from the control console. This identification is specified in terms of a program number with 'O' word address. The number can be a maximum of four digits.

Sequence number (N-word)

Each block in a part program always starts with a block number, which is used as identification of the block. It is programmed with a 'N' word address.

Coordinate function

The coordinate values are specified using the word address such as X, Y, Z, U, V, W, I, J, K, etc. All these word address are normally signed along with decimal point depending upon the resolution available in the machine tool.

Comments

Parentheses are used to add comments in the program to clarify the individual function that are used to add comments in the program .When the controller encounters the opening parenthesis. It ignores all the information till it reaches the closing parenthesis.

Example

N010 G00 Z50 M05(Spindle stops and rapidly moves up)

Table common word addresses used in word address format

Address	Function
N	Sequence number to identify a block.
G	Preparatory word that prepares the controller for instruction given in the block.
X, Y, Z	Coordinate data for three linear axes.
U, V, W	Coordinate data for incremental moves in turning in the X,Y and Z directions respectively.
A, B, C	Coordinate data for three rotational axes X, Y and Z.
R	Radius of arc, used in circular interpolation.
I, J, K.	Coordinate values of arc centre, corresponding to X, Y and Z-axes respectively.
F	Feed rate per minute or revolution in either inches or millimeters.
S	Spindle rotation speed.
T	Tool selection, used for machine tools with automatic tool changer or turrets.

D	Tool diameter word used for offsetting the tool.
P	It is used to store cutter radius data in offset register. It defines first contour block number in canned cycles.
Q	It defines last contour block number in canned cycles.
M	Miscellaneous function.

M - Cods (Miscellaneous Function)

Address M and the follow numerals control ON/OFF of machine function, such as the spindle rotasion starator stop.

M00 - Program stop by inserting M00 in program the cutting cycle is stopped after the block containing M00 code the facility in useful if an inspection check is necessary during an operation Ex: M00;

M01 - Optional stop

This function is same as ‘M00’, But it will stop only when Optional stop button in the Machine panel is ‘ON’. Then cycle is started to continue by pressing Cycle Start Button.

M02 - Program end

The code is inserted at the end of the program. The machine stops permanently. Spindle rotation, Feed of axis and coolant discharge are stops. The system is reset by pressing Reset button in the machine panel and new cycle is started by pressing Cycle start.

M03 - Spindle on clockwise

By programming ‘M03’ the spindle is enabled to run in the clockwise direction.

M04 - Spindle on counter clockwise

By programming M04 the spindle is enabled to run in the counter clockwise direction.

M05 - Spindle stop

By programming ‘M05’ the spindle rotation is stopped.

M08 - Coolant on

By programming ‘M08’ coolant motor switches ‘ON’.

M09 - Coolant off

By programming ‘M09’ coolant motor switches ‘OFF’.

M30 - Program end & rewind

When CNC reads the code ‘M30’ the main program End and Rewind . That is the CNC control returns the cursor to the starting line of the program.

G - Codes (preparatory functions)

G codes take active part in part program execution and are programmed by letter G followed by two digits.

G codes once programmed, remains active until another. G code of the same group is programmed, after which the previous one gets cancelled, are said to be model.

G codes which remains active only in the block in which it is programmed, is said to be Block wise active (or) one shot g code.

G00 - Rapid traverse

The Tool moves at a rapid (fast) traverse rate with linear interpolation. The rapid traverse rate depends upon the machine type (for example maximum speed in a two wheeler is 80-120 Km/hr depends on type of make).

This can be used in air movement like positioning, relieving, non contact with work piece.

Format

1. G00 X ---------;
2. G00 Z-----------;
3. G00 X------ Z----;

G00 - code used for the following operations

1 Machining start

Making the tool approach the work piece.

2 During machining

Moving the tool to next command position when it is not in contact with the work piece.

3 Machining end

Separating the tool from the work piece.

G01 - Linear interpolation (straight cutting)

The cutter moves at specified feed rate. The feed rate is specified by address 'F' in the program.

Format

1 G01 X------- F-----;

Application

a Facing
b Grooving etc.

2 G01 Z-----F-------;

Application

a. Straight turning
b. Drilling etc

3 G01 X-----Z------F------;

Application

a. Taper turning
b. Chamfering

Where ‘F’ is the cutting feed rate specified in mm/rev.

Function F

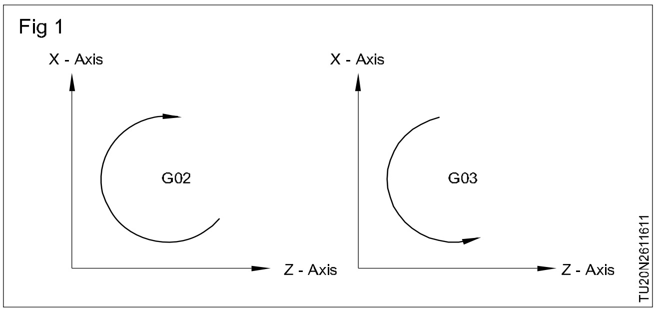
The feed rate is used to move the tool from one point to another point with constant feed rate. Feed is normally is given mm/rev. or mm/min. The rapid traverse rate and feed rate both are controlled by feed override switches in the machine panel.

Example

F	Four digits number following the address F
G01 X 50.0 Z -50.0 F0.1;	X -axis & Z - axis move with feed 0.1mm/Rev.

Circular interpolation (Fig 1)

Fig 1



G02-Circular interpolation clockwise direction

G03-Circular interpolation Anti clockwise direction

Format

$$\left.\begin{matrix}G02\\G03\end{matrix}\right\} X--Z--R--F--;$$

OR

$$\left.\begin{matrix}G02\\G03\end{matrix}\right\} X--Z--I--K--F$$

Where

X Z - End point of Arc

I K - Distance between start point of arc to center point of arc in X & Z axis

R - Radius of the arc

F - Feed

Command I and K specify the distance from the start point of arc to the center point of arc must be specified incrementally even under Absolute mode and sign (+) or (-) for Values I & K is determined by the direction.

Example

G02 X 40.0 Z-5.0 R 5.0 F 0.1

G03 X 40.0 Z-5.0 R5.0 F 0.1

Where, R=Radius

G04-Dwell

If a block with G04 is real during automatic operation, the feed is stopped for the time followed U, X, P, and then the next block will be executed.

Format

G04 (U, X, P) time

Example

G04 U 1.0 (Dwell of 1.0 second)

Note

Decimal point is not available in 'P'

Ex. Dwell of 2.5 seconds.

G04 U 2.5

G04 X 2.5

G04 P 2500

G28 - Zero Return (Home Position, First Reference value)

It is an inherent position on a machine axis. Automatic Reference Point Return is a function to return each axis to this inherent position automatically.

1 G28 U0

2 G28 W0

3 G28 U0 W0

G30 - Second reference return

It is same as G28.But is to settled before First Reference Value (G28). It is called Temporary Reference Value.

1 G30 U0

2 G30 W0

3 G30 U0 W0

G50 - Co-ordinate value setting & maximum spindle speed setting

1 G50 X---Z---;

Ex. G50 X 300.0 Z 150.0;

2 G50 X---Z---S---;

Ex. G50 X300.0 Z 150.0 S 3000

G96-Constant Surface Speed Control (Cutting Speed Specification)

The G96 is used with an "S"-Function.

The G96 is used when the cutting speed is specified.

When G96 command is used the spindle speed is changed automatically, as the cutting diameter is changed. That is for smaller work piece of its cutting diameter, the spindle speed becomes higher.

Calculation for cutting speed

$$V = \frac{\pi DN}{1000} = m/min$$

Where

V = cutting speed

D = Diameter of the work piece in 'millimeter'

N = spindle speed in rpm

G97-Constant Surface Speed Control Cancel (Spindle Speed Specification)

The G97 is used when the spindle rotating speed is specified.

Ex. G97 S300 M03.

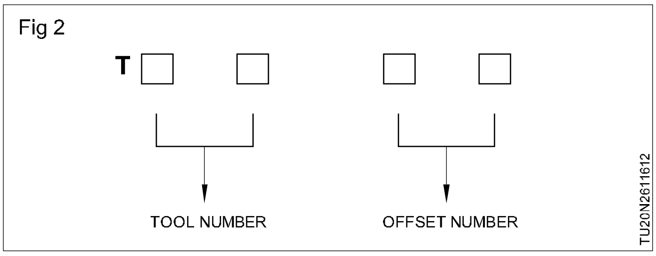
With this spindle rotates at 300 rpm.

For the following should use G97 always

- a. Threading
- b. Tapping
- c. Drilling etc

Tool function (Fig 2)

Fig 2



Address: T

A four digits number address T Specifies the tool number and tool offset number.

Format

Example : T01 01

Tool Number

The left most two digits specify the number of tool.

Offset Number

The right most two digits specify the number of tool offset.

Types of Offsets

There are two types offsets:

1 Wear offset
2 Geometrical offset

1 Wear offset

The tool is moved adding the wear amount to part program. Input the offset amount to the same number as the number on offset screen (WEAR)

2 Tool Geometry offset

The distance from top of the tool fixed on turret at machine zero point to the work piece zero point is input as tool geometry offset with this the CNC recognizes the position of work piece zero point. Input the offset amount to the same number as the number on offset screen (Geometry).

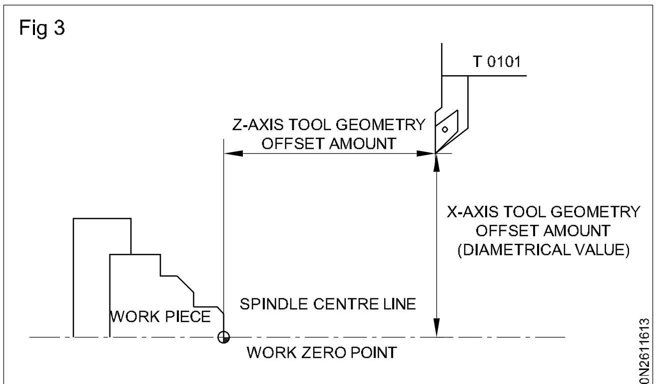
Tool geometry offset (Fig 3)

This offset amount is not need to be cancelled after every tool use because the next input of tool geometry offset cancels former offset automatically.

Tool wear offset

The tool wear offset is used to modify the finished work piece dimension in order to keep them within their tolerances. The programmed path is shifted by the offset amount parallel to X and Z axes. The offset amount is input to "TOOL OFFSET /WEAR".

Fig 3



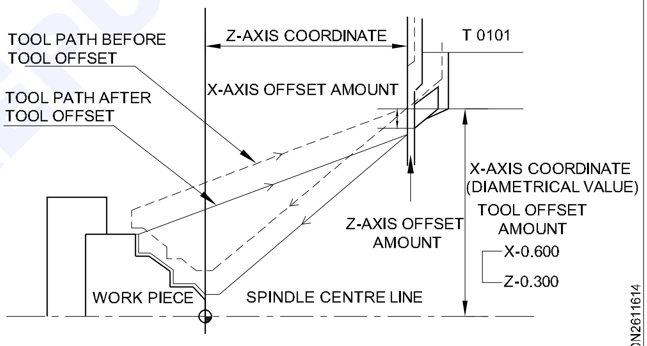
TOOL GEOMETRY OFFSET

When the control reads T0101 and executes, the tool is shifted by amount which is input in the tool wear offset number (X-0.600, Z0.300).

After the machining the tool is returned near the starting point and if T0100 (Tool wear offset cancel) is executed, it returns to the starting point before offset. The same movement is executed for other tools, only to assign tool wear offset numbers which are required on the programming the amount to be offset should be decided by the operator.

Procedure for setting work coordinate system (Fig 4)

Fig 4



WORK ZERO POINT

Step 1 Mark sure that the component is securely clamped.

Step 2 Now bring one of the tool near the face of the job.

Step 3 I.Select MDI Mode.
II. Press PROGRAM button.

Step 4 Enter S500 M03;

Step 5 Select handle/jog mode and select the appropriate feed.

Step 6 Rotate the spindle in CW or CCW depending on the type of the tool.

Step 7 Light facing out be taken up to the center.

Step 8 After the finish cut, move the tool back in x only. Don't disturb Z-axis.

Step 9 Now switch off the spindle.

Step 10	Press MENU offset. The wear geometrical and work shift are displayed on CRT.
Step 11	Now, press GEOM soft key and position the cursor using cursor movement buttons to be required offset number corresponding to the tool used.
Step 12	Press measure(m) key and press Z. Enter Zero(MZ0) press measure (m) key.
Step 13	Now rotate the spindle in appropriate direction and machine on OD
Step 14	Do not move X axis.
Step 15	Take Z away from the job.
Step 16	Stop the spindle.
Step 17	Press MENU OFFSET
Step 18	Press 'GEOM' soft key.
Step 19	Position the cursor to the required tool offset number.
Step 20	Press M….X….
Step 21	Input "The OD dimension measured. The X-offset for the said tool is set.
Step 22	Repeat the procedure for all tools.
Step 23	After taking offset, select MDI and enter S0.

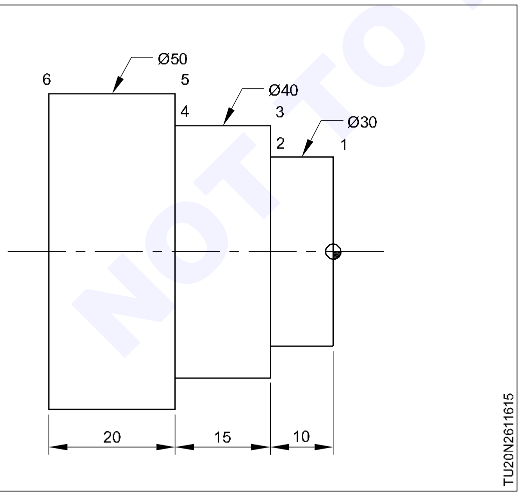
Programming method

In CNC for programming in Lathe, Absolute Command and Incremental Command are available.

Absolute method (Fig 5)

In absolute dimensions programming, all the points of the tool is coming from the datum point (or) zero point. In CNC Lathe machines "X" and "Z" is the absolute input. The "X" means diameter of work piece and the "Z" means distance from the finished end surface of work piece.

Fig 5



At the travel commands for tool are mean their coordinate value from the work piece zero point (X0, Z0).

Position	X	Z
1	30.0	0.0
2	30.0	- 10.0
3	40.0	- 10.0
4	40.0	- 25.0
5	50.0	- 25.0
6	50.0	- 45.0

In the above figure, points 1 to 6 can be specified as follows in absolute dimension programming.

Incremental method (Fig 6)

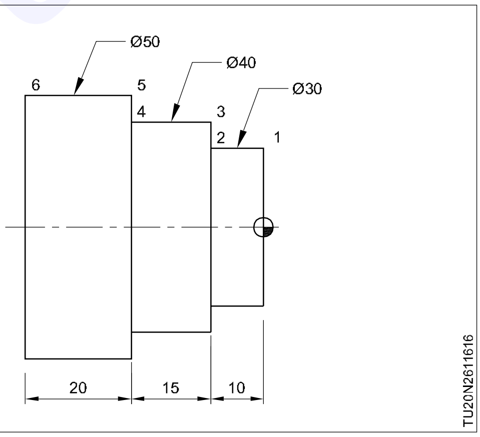
In this system, tool move from the previous point. In the incremental programming the address "U" (diametrical) for "X" axis and the address "W" for "Z" axis are used to distinguish incremental program from the absolute program.

The incremental command should have the direction (+) and distance from currently specified point to next command point. (-)

Example

In the Fig 6 the points, 1 to 6 can be specified as follows in incremental dimension programming.

Fig 6



Position	U	W
1	15.0	0.0
2	0.0	- 10.0
3	5.0	- 0.0
4	0.0	- 15.0
5	5.0	- 0.0
6	0.0	- 20

Stock removal in CNC turning external and internal

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

- **state which G code is used for stock removal in CNC program**
- **describe the format for the G71 code**
- **explain the external stock removal cycle program**
- **brief the internal stock removal program.**

Stock removal in CNC turning

G71 Cycle is a CNC code and used to stock removal in turning for CNC lathe machines. G71 canned cycle is most common roughing cycle for CNC lathe and turning machines. Its purpose is to remove stock by horizontal cutting, primarily along the Z-axis, typically from the right to the left. It is used for roughing out material out of a solid cylinder.

The G71 Cycle generally processes the profile to be processed with the tolerances you specified in the program, and then finish with the G70 Finishing Cycle. In addition, although many of operators use the G71 code to turning the outer diameter, boring or hole turning operations can also be performed with the G71 command.

G71 Cycle Format

Like most of CNC cycles, G71 canned cycle comes in two formats - a one-block (known as single line or type 1) and a double block format (known as also two line or type 2), depending on the control system, especially for Fanuc CNC controller. Even if it's known for Fanuc CNC controller, most of other controller also using same structure for G71 G code.

G71 Cycle for Fanuc 6T/1 0T/11 T/15T

The one-block (Single line or Type 1) format for the G71 turning cycle is:

G71 P ... Q ... I. .. K ... U ... W ... D ... F ... S ...

Parameters

P : First block number of the contour in program (N10, N20 ... etc.)

Q : Last block number of the contour in program (N80, N90 ... etc.)

I : Distance and direction of rough semi finishing in the X-axis - per side (Optional) K : Distance and direction of rough semi finishing in the Z-axis (Optional)

U : Amount left for finishing in the X-axis (in diameter)

W: Amount left for finishing in the Z-axis

D : Depth of roughing cut

F : Cutting feedrate (in/rev or mm/rev) overrides feedrates between P block and Q block

S : Spindle speed (ft/min or m/min) overrides spindle speeds between P block and Q block

Note: The I and K parameters are not available on all machines. They control the amount of cut for semi finishing, the last continuous cut before final roughing motions.

G71 Cycle for Fanuc 0T/16T/18T/20T/21 T

If the control requires a double block entry (Two line or Type 2) for the G71 turning cycle, the programming format is:

G71 U ... R .. .

G71 P ... Q ... U ... W ... F ... S ...

Parameters

First block:

U : Depth of roughing cut

R : Amount of retract from each cut

Second block:

P : First block number of the contour in program (N 10, N20 ... etc.)

Q: Last block number of the contour in program (N80, N90 ... etc.)

U : Amount left for finishing in the X-axis (in diameter)

W : Amount left for finishing in the Z-axis

F : Cutting feedrate (in/rev or mm/rev) overrides feedrates between P block and Q block

S : Spindle speed (ft/min or m/min) overrides spindle speeds between P block and Q block

Note: Do not confuse address U in the first block, depth of cut per side, and address U in the second block, stock left on diameter. The I and K parameters may be used only on some controls and the retract amount R is set by a system parameter

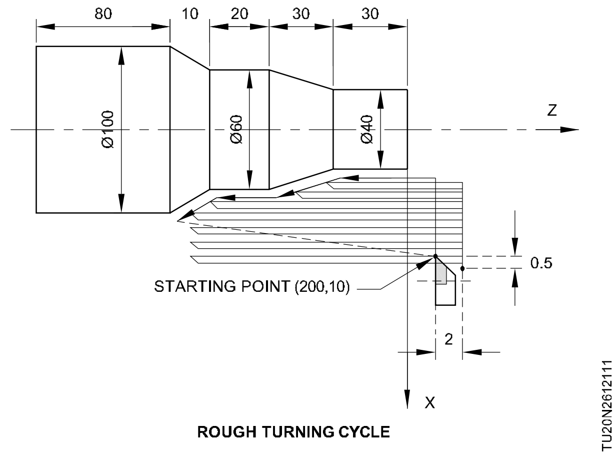
G70 Finishing Cycle

G70 Finishing Cycle is used for finish cutting operations (final cleaning cutting) in CNC lathes. G70 cycle is used to final cutting after any roughing cycles like G71 Turning Cycle, G72 Cycle or G73 Pattern Repeating Cycle. It's possible to proceed finish cutting with different tool, spindle speed or feedrate after roughing cycles, and use in same program. G70 finishing cycle follows same tool path and contour with G71 turning cycle but only once, not more.

It is not compulsory to use G70 after G71 turning but in general, CNC machine users perform rough cutting with G71, and finishing cut with G70. The amount of finishing passes to be left for G70 is specified with the U and W values in second row of G71 command.

Example program for external stock removal (Fig 1)

Fig 1



ROUGH TURNING CYCLE

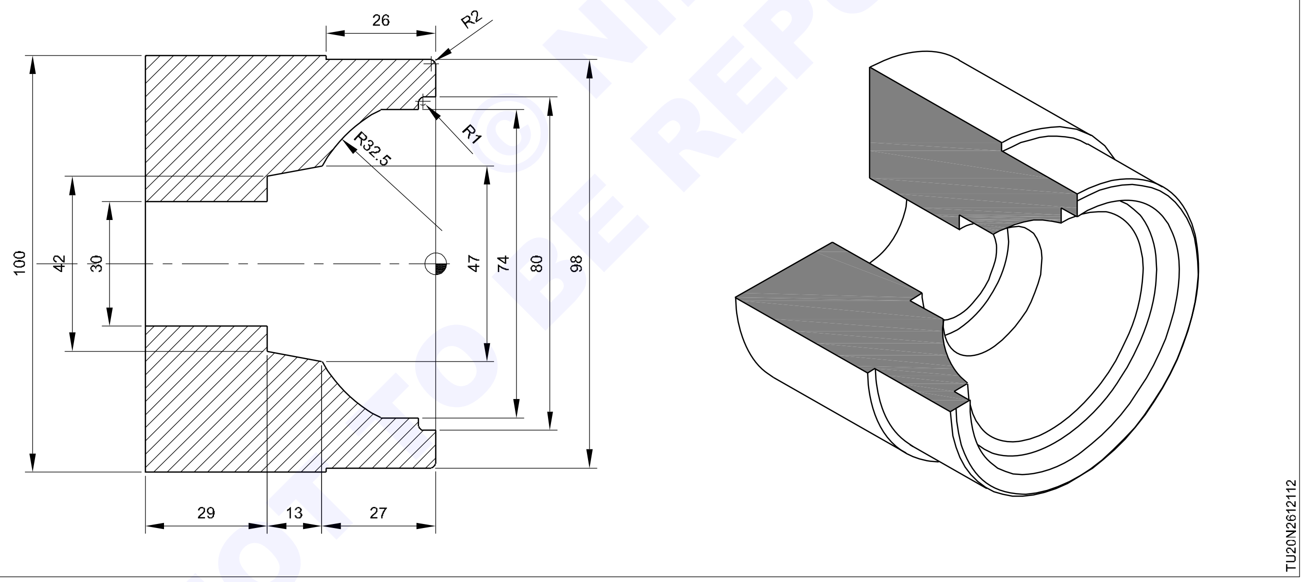
```
00004
G00 X200 Z10 M3 S800
G71 U2 R1 F 200
G71 P80 Q120 U0.5 W0.2
N80 G00 X40 S1200
G01 Z-30 F100
X60 W-30
W-20
N120 X100 W-10
G70 P80 Q120
M30
```

Example program for internal stock removal in CNC turning (Fig 2)

G71 Cycle Example

We will be machining the outer diameter and boring of the material given above with G71. considered that the middle of the material to be machined is already a 28 diameter hole - since no drilling is performed in the program example. So, in this program steps will be outer diameter machining with G71 and subsequently boring with G71 cycle.

Fig 2



```
03000;
N05 T0101;
N10 M4 S1800;
N15 G0 X102 Z0 M8;
N20 G71 U2 R1;
N25 G71 P30 Q45 U0.6 W0.2 F0.25;
N30 G1 X94;
N35 G3 X98 Z-2 R2;
N40 G1 Z-26;
N45 G1 X100;
N50 G70 P30 Q45;
N55 G0 X200 Z200 M9;
N60 T0808;
N65 M4 S1500;
N70 G0 X28 Z5 M8;
N75 G1 Z0 F0.15;
N80 G71 U1 .5 R1;
N85 G71 P90 Q130 U-0.4 W0.2 F0.15;
```

```
N90 G1 X80 F0.05;
N95 G1 Z-3;
N100 G3 X78 Z-4 R1;
N105 G1 X70;
N110 G1 Z-9;
N115 G3 X47 Z-27 R32.5;
N120 G1 X42 Z-40;
N125 G1 X30;
N130 G1 Z-69;
N135 G70 P95 Q130;
N140 G0 Z200 M9;
N145 G28 U0 W0;
N150 M30;
```

Things to remember

- Return motion to the start point is automatic, and must not be programmed.
- F cutting feed rates given after the G71 cycle lines is used in the G70 finishing turning cycle.
- G41 and G42 tool nose radius compensation cannot be used with the G71 cycle. If written in the program, the G70 is used during the finishing cycle.
- If the program is stopped during the G71 cycle and some manual axes movements are performed, it must be moved to the point where the program is stopped manually before starting the program again.
- P and Q lines defining the finish profile must be written on the same line as G71 code.
- The G71 canned cycle cannot be run under MDI mode.
- M98 and M99 commands are not used in lines where G71 cycle is written.
- Change of direction is allowed only for Type II G71 G code, and along one axis only (W0).
- For internal turning, finishing pass (U in second line) value must be given negative(-). (Such as G71 P20 Q50 U-0.3 W0 F0.12).

CNC lathe work holding - Thumb rules for quick decisions

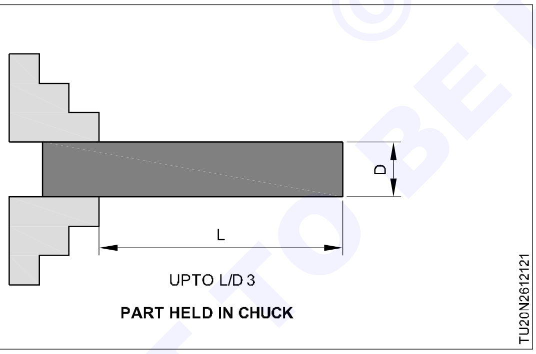
Objective : At the end of this lesson you shall be able to

- **state the length vs diameter ratio according to work holding methods.**

Part held in chuck (Fig 1)

The part held in the chuck is a cantilever, and the radial cutting force of the tool tends to bend the part.

Fig 1



UPTO L/D 3

PART HELD IN CHUCK

Part supported by tailstock (Fig 2)

You use a tailstock or steady rest to prevent the bending. Here's a CNC lathe work holding thumb rule that tells you when you can hold in a chuck, when to use a tailstock, and when to use a steady, based on the L/D (Length to Diameter) ratio of the part.

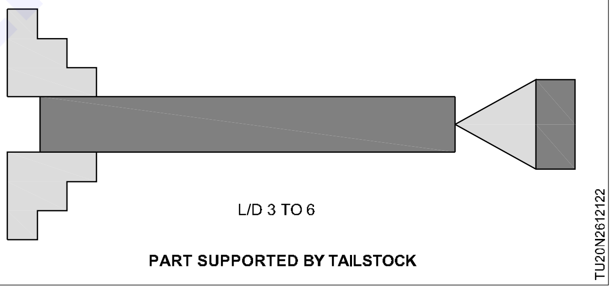
Part supported by tailstosk and steady, rest (Fig 3)

Chuck only: Use if L/D is less than 3. You can go up to L/D 5 with reduced cutting parameters, which reduce the cutting force.

Tailstock: Use if L/D is between 3 and 6. You can go up to L/D 10 with reduced cutting parameters.

Steady rest: Use if L/D is between 6 and 12. You can go up to L/D 20 with reduced cutting parameters.

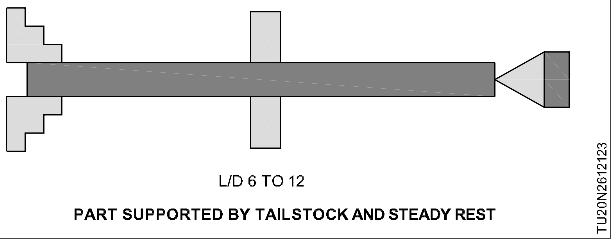
Fig 2



L/D 3 TO 6

PART SUPPORTED BY TAILSTOCK

Fig 3



L/D 6 TO 12

PART SUPPORTED BY TAILSTOCK AND STEADY REST

If you want to cut with the full recommended cutting parameters, every 3 L/D requires a new holding point.

E.g.,

A chuck is a single holding point

Chuck + tailstock is 2 holding points,

Chuck + tailstock + steady is 3 holding points, etc.

Formula to determine the number of holding points.

No. of holding points = (L/D)/3, rounded off to the next higher number.

Example: If L/D = 4.5 would require 2 holding points = (4.5/3) is 1.5, and this rounded off to the next higher number is 2.

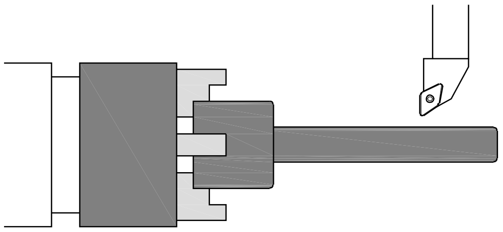
To cut with reduced cutting parameters, in this formula change the 3 to 5.

CNC turning - part bending and L/D ratio (Fig 4)

Part bending increases very rapidly with increase in L/D ratio

The L/D ratio makes a big difference to the quality of the turned part, so ensure that you work holding is good. For a long or a thin walled part, ensure that the cutting the depth of cut and feed rates are reduced as the L/D increases.

Fig 4



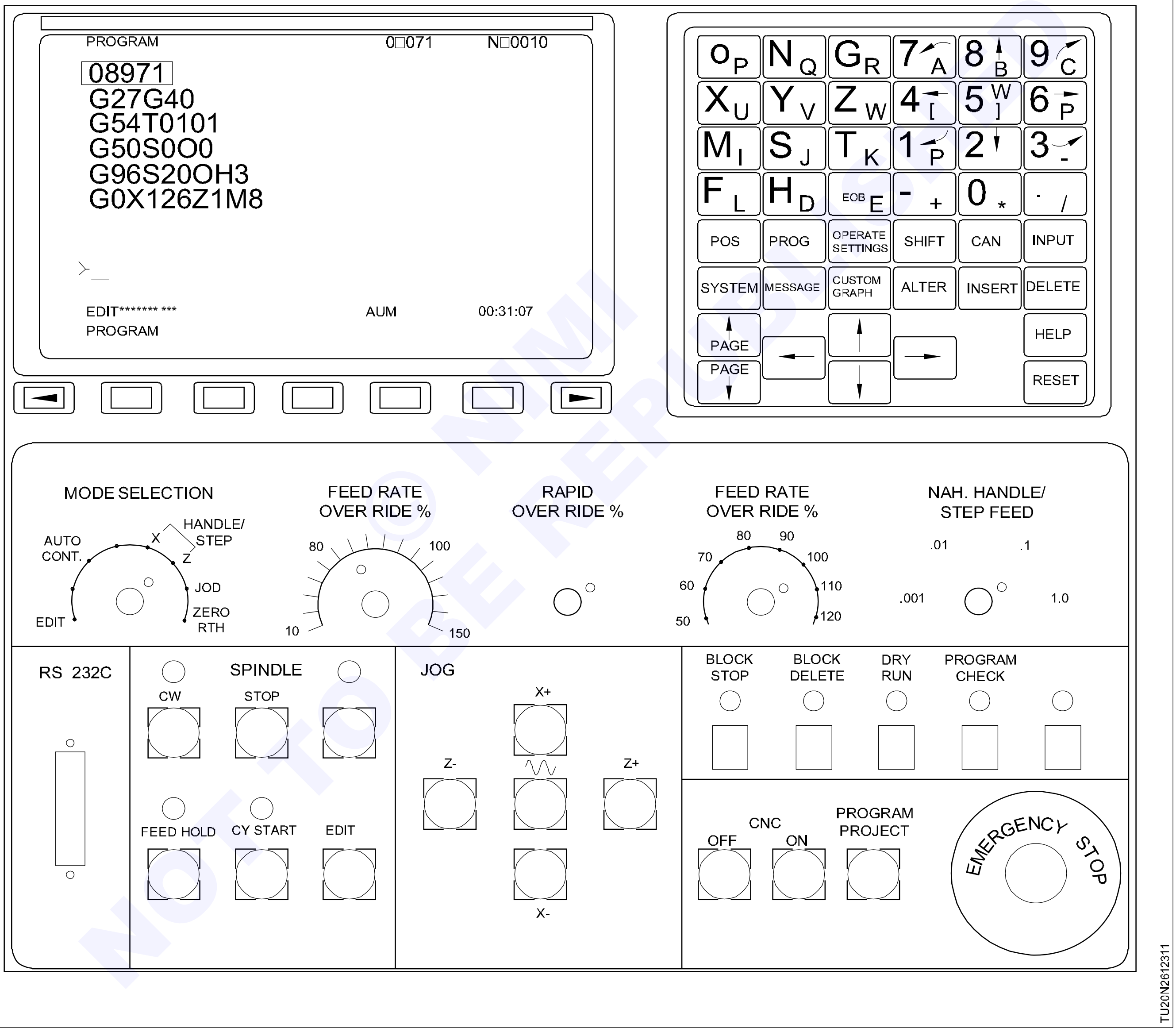
CNC TURNING - PART BENDING L/D RATIO

Machine operation modes - Jog - MPG - edit memory - Fanuc system

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

- **state the various operational modes**
- **state the work piece zero point**
- **state the tool zero point**
- **explain the machine reference point**
- **list the various types of offset**
- **prepare the part program for facing, plain turning, step turning, chamfering, radius turning and drilling.**

Fig 1



Operational modes

Jog mode

Jog mode is used for moving the turret in X and Z direction. After selecting jog mode if we press 'X+' axis button, the turret will move in 'X+' direction. In the same manner we can move in the ‘Z’ direction also.

Incremental jog mode

This mode is used to move the turret in micron level. By pressing the axis button, in this mode, we can move the button in 0.001, 0.01, 0.1, 1 mm range.

Edit mode

This mode is used to edit the program. In this mode edit key should be in 'ON' position, to input a program.

Manual mode: MDI mode

MDI mode means manual data input. In this mode, we can input the program command manually and execute the program.

Single block mode

This mode will function when the mode switch is set in AUTO mode only. If we switch on the single block switch and push the cycle start button, then the single block in the programme only will be executed. For the execution of the next block then again cycle start button should be pressed. If the single block switch is in OFF position, then the program will be executed continuously.

Auto mode

For this mode, the mode switch is (set in AUTO mode). In (this mode the program will be executed continuously) one block after another block.

In this mode if we (press the cycle start button), the current program in the CRT panel will be executed.

Zero points and reference points

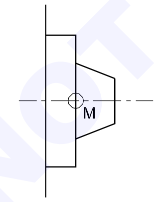
Zero point

In CNC machines, tool movements are controlled by coordinate systems. The origin of the co-ordinate system is considered as zero point. In some of the CNC machines, the zero point may be located at a fixed place and cannot be changed. This is known as fixed zero point. Some other machines, a zero point may be established by moving the slides so that the cutting tool is placed in the desired position in relation to the work pieces. This is known as floating zero point.

Machine zero point or machine datum (M)

It is a fixed point on a machine specified by the manufacturer. This point is the zero point for the coordinate system of the machine controller. In turning centre, the machine zero point is generally at the centre of the spindle nose face as shown in Fig 2. In machining centres, it is either fixed at centre of the table or a point along the edge of the traverse range.

Fig 2

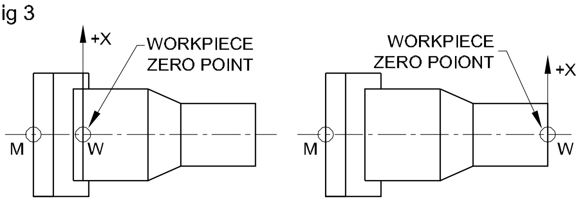


MACHINE ZERO PONT

Work piece zero point (W)

This point determines the work piece coordinate system in relation to the machine zero point as shown in Fig 3. This point is chosen by the part programmer and input to the machine controller. The position of this point may be choosen in such a way that the dimensions of the work piece drawing can be easily converted into coordinate values. For turned components, it is placed along the spindle axis in line with the right or left end face of the work piece. It is also known as program zero point.

Fig 3



WORK PIECE ZERO POINT

Tool zero point (T)

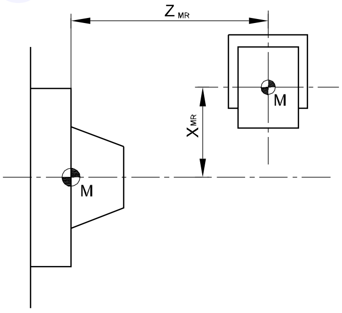
When machining a work piece, the tool must be controlled in precise relationship with the work piece along the machining path. This requires a point in the tool turret be taken as reference point, which is known as tool zero point.

At the tools in the tool turret have different shapes and sizes, the Offset distance between the tool zero point and work piece zero point is measured and entered in to the computer. This known as tool offset setting.

Machine reference point (R)

Machine reference point is also known as home position as shown in Fig 4. It is used for calibrating the measuring system of the sides and tool movements. It is determined by the manufactures.

Fig 4



MACHINE REFERENCE PONT($X_{MR}Z_{MR}$)

The value of the machine reference coordinates (XMR, ZMR) is fixed and cannot be changed by the user. The positioning of the reference point is accurately predetermined in every transverse axis by the trip dogs and limit switches.

Programming details (Fig 5)

- - - - - - - - - This line indicates the tool moves at rapid feed (G00)

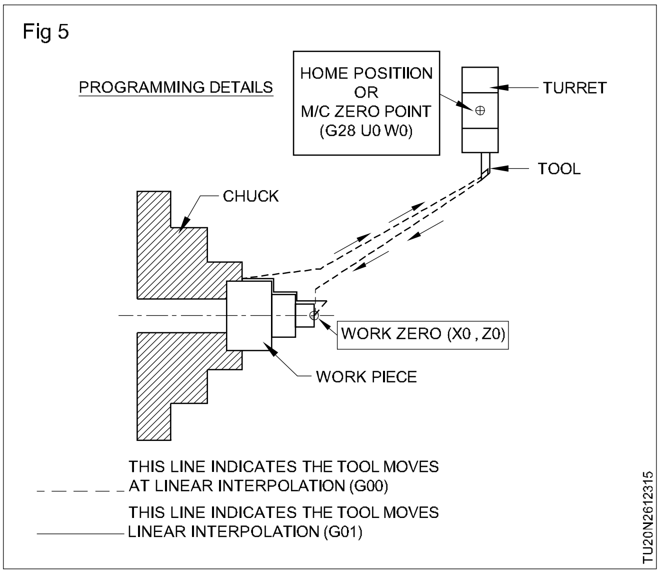
―――――― This line indicates the tool moves at linear interpolation (G01)

Operation devices

Edit alphanumeric keys

Used to edit the part program, tool offsets, work offsets etc..

Fig 5



Feed rate override switch

Enables manual overriding of the programmed feed rate during part machining. Can be varied between 0% and 120% of the programmed feed rate, in steps of 10%.

Rapid rate override switch

Enables manual overriding of the rapid traverse rate during rapid motions. Can be varied between 0% and 100% of the programmed feed rate, in steps if 25%.

Spindle speed override switch

Enables manual overriding of the spindle speed during part machining.

Feed hold

Stops the motion of all axes temporarily during machining.

Cycle start

Machine rest

Stops all functions being executed, like spindle rotation, axes motion, etc.

Emergency stop

Used when the machine is to be halted suddenly, like in case of tool breakage. Pressing it shuts down all systems of the machine except the console-axes drives, spindle drive, coolant pump, hydraulic power pack etc.

Single block ON/OFF

When OFF, execution of the part program is automatic and continuous. When ON, part program is executed block-wise. In block-wise execution the cycle start button must be pressed to executed each block.

Coolant ON/OFF Controls the coolant.

Data input/output

Used to transfer data between the machine and an external device like a PC. Data that can be transferred is part program, PLC data, tool offset and work offset.

Chip conveyor forward backward Moves the chip conveyor.

Dry run

Sets the Dry run mode ON or OFF. The Dry run mode is used to check the part program by executing it without actually cutting a part. During this mode commanded federate in the part program is not effective, and the axes moves at a fixed Dry run feed rate. Dry run feed rate is typically 1000 mm/min to 5000mm/min.

Machine lock and auxiliary function lock

Sets the Machine lock mode ON or OFF. The machine lock mode is used to check the part program by executing it without any axes motions and miscellaneous functions like tool change, spindle rotation, etc. The screen display appears as during normal execution.

Operational modes

- Auto mode
- Edit mode
- MDI mode (Manual data input)
- Jog mode
- MPG mode (manual pulse generation) data
- Input/output mode
- Zero return mode

I) Selection of tools, speed feed & depth of cut

D	=	work piece diameter	mm
V	=	cutting speed	m/min
S	=	Feed	mm/rev
N	=	RPM	rev/min
A	=	Depth of cut	mm
N	=	Efficiency	for example 0.75
Ks	=	Specific cutting force	N/mm^2
V	=	Metal removal rate	cm^3/min
P	=	power required	kW
R	=	nose radius	mm
K	=	Constant	for example 1.4
R_t	=	Profile depth	µm
R_a	=	Surface finish	µm

A Manual Pulse Generator (MPG) in a device for generating electrical pulses in electronic system under the control of human operation as opposed to the pulses automatically generated by software.

MPGS are used an computer numerically controlled (CNC) Machine Tools on some microscopes and on other devices that use precise component positioning. A typical MPG consists of a rotating knob that generated pulses that are sent to an equipment controller. The controller with then move the piece of equipment a pre determined distance for each pulse. The hand wheel of CNC control will move any of the slides of the m/c by one micron increment, such as 1 micrometer or 1 ten - thousand of an inch for each pulse the hand wheel will give one ratchet - like click to confirm the user that a single increment accured.

Several selector switches control the hand wheel is output one allows each of the m/c axes (xyz and 50m) to be selected in term.

Procedure for JOG feed

- Press the JOG mode switch.
- Select the axis to be moved.
- Keep the feed rate switch open.
- Keep pressing the direction switch until the tool reached the desired position.

The mode that allows for manual operation tool movement via the jog button - manual pulse generator (MPG)

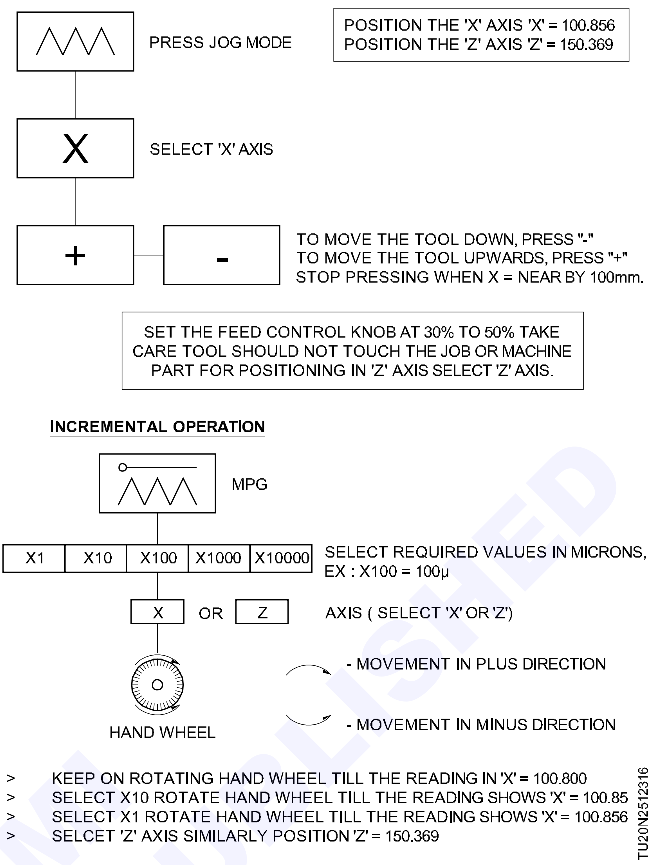
Procedure for INCREMENTAL feed (Fig 6)

- Select the INC mode.
- Keep the feed rate switch open.
- Select the distance to be moved in each step with the magnification dial
- Select the axis.
- Press the direction switch.
- Note the movement of the axis.

Procedure for MANUAL HANDLE feed

- Press the HANDLE switch.
- Select the axis.
- Select the incremental value.

Fig 6



- Move the tool along selected axis by rotating the handle 360 degrees moves the tool the distance equivalent to 100 graduation.

The instructor will demonstrate the various axis and models.

Canned cycles

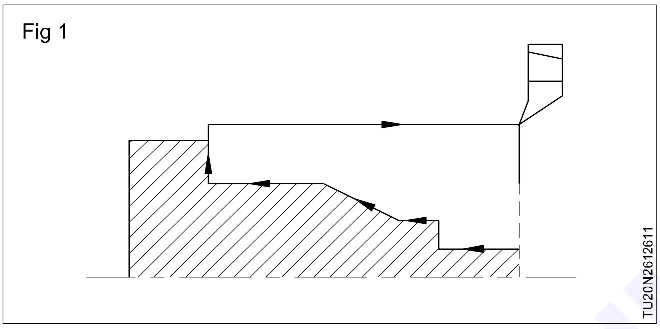
Objectives : At the end of this exercise you shall be able to
- **state the cycle used in CNC program**
- **learn to program for all canned cycles**
- **learn to program for threading OD/ID.**

Canned cycles

Canned cycle is used in stock removal operation in turning. In this cycle, the tool is positioned at the starting point. The finishing contour of the pocket is to be programmed like the normal programming using G code.

G70 - Finishing cycle (Fig 1)

Fig 1



Format:

G70 P_Q_F_;

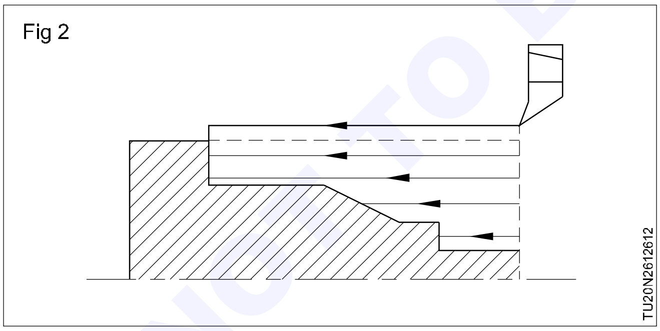
P : Starting Block Number

Q : Ending Block Number

F : Finishing Feed

G71 - Turning cycle (Fig 2)

Fig 2



Format

G71 U_R_;

G71 P_Q_U_W_F_;

U : Depth of cut per pass in X Axis (Radial value)

R : Relief Amount

P : Starting Block Number

Q : Ending Block Number

U : Finishing Allowance in X Axis

W : Finishing Allowance in Z Axis

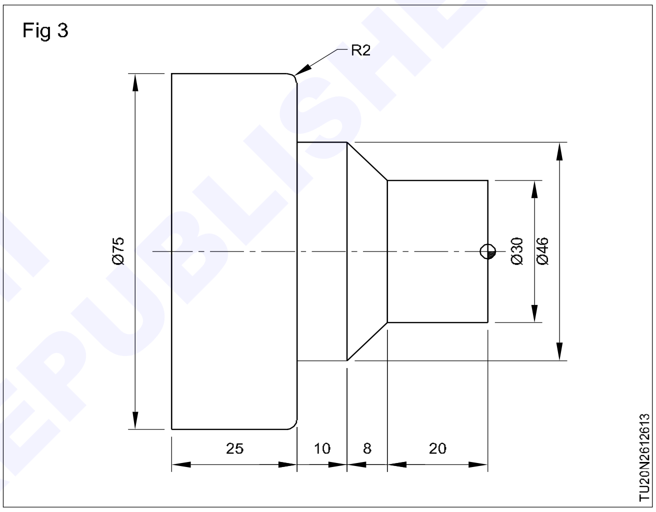
F : Feed

Example for G71 turning cycle

%

O0026 (Fig 3)

Fig 3



```
G28 U0 W0 ;
G50 S1200 T0300 ;
G96 S250 M03 ;
G00 X65.0 Z0.0 T0303 M08 ;
G01 X0.0 F0.1 ;
G00 X61.0 Z2.0 ;
G71 U1.0 R1.0 ;
G71 P10 Q20 U1.0 W0.5 F0.3 ;
N10 G00 X30.0 ;
G01 Z-20.0 F0.1 ;
X46.0 Z-28.0 ;
Z-38.0 ;
G1 X71 ;
G03 X75.0 Z-40 R2 ;
G01 Z-63 ;
N20 X80.0 ;
G00 X 80.0 Z2.0 M09 ;
G28 U0 W0 M05 ;
M01 ;
```

```
N2 ; (OD FINISHINING)
G28 U0 W0 ;
G50 S2600 T0700 ;
G96 S150 M03 ;
G00 X26.0 Z0.0 T0705 M08 ;
G01 X0.0 F0.1 ;
G00 X65.0 Z3.0 ;
G70 P10 Q20 F0.12 ;
G00 X80.0 Z2.0 M09 ;
G28 U0 W0 ;
M30 ;
```

G72 - Facing cycle

Format

G72 W_R_;

G72 P_Q_U_W_F_;

W : Depth of cut per pass in Z Axis

R : Relief Amount

P : Starting Block Number

Q : Ending Block Number

U : Finishing Allowance In X Axis

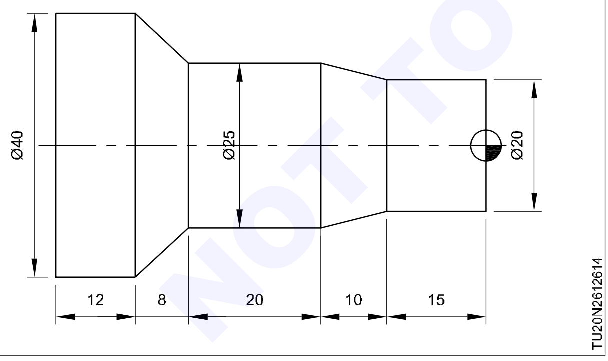
W : Finishing Allowance In Z Axis

F : Feed

Example for G72 Facing cycle

```
%
O0027 (Fig 4)
```

Fig 4



```
N1 ;
G28 U0 W0 ;
G50 S1300 T0200 ;
G96 S150 M03 ;
G00 X55.0 Z2.0 T0202 M08 ;
G72 W1.0 R2 ;
G72 P20 Q30 U1.0 W0.5 F0.3 ;
N20 G00 Z-65.0;
G01 X40 ;
Z-53.0 ;
X25.0 Z-45.0 ;
Z-25.0
X20.0 Z-15.0 ;
Z0.0 ;
N30 ;
G00 X55.0 Z2.0 M09 ;
G28 U0 W0 M05 ;
M01 ;

N2 ; (OD Finishing)
G28 U0 W0 ;
G50 S2600 T0700 ;
G96 S150 M03 ;
G00 X55.0 Z2.0 T0707 M08 ;
G70 P20 Q30 F0.12 ;
G00 X65.0 Z2.0 M09 ;
G28 U0 W0 M05 ;
M30 ;
```

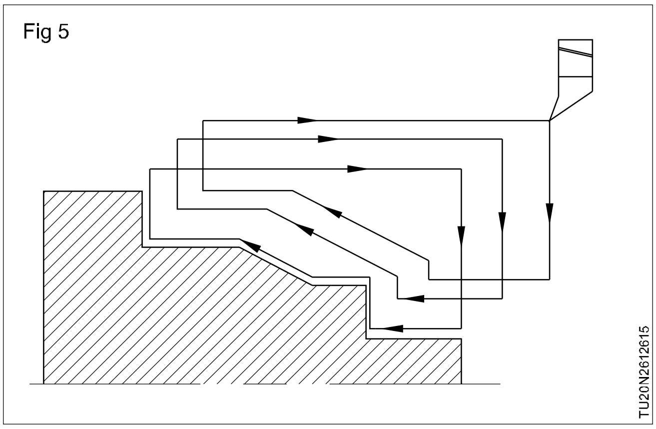
G73 - Pattern repeating cycle (Fig 5)

Format:

G73 U_W_R_;

G73 P_Q_U_W_F_;

Fig 5



U : Total amount of stock in X axis (radial value)

W : Total amount of stocks in Z axis

R : Number of passes

P : Starting block number

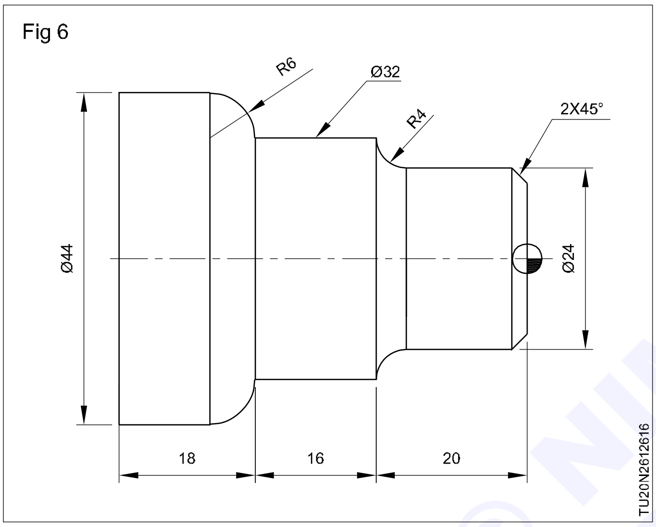
Q : Ending block number

U : Finishing allowance In X axis

W : Finishing allowance in Z axis

F : Feed

```
%
O0028 (Fig 6)
N1 ;
G28 U0 W0 ;
G50 S1200 T0300 ;
G96 S250 M03 ;
G00 X55.0 Z2.0 T0303 M08 ;
G73 U5.0 W3.0 R6 ;
G73 P40 Q50 U0.5 W0.1 F0.3 ;
N40 G00 X20.0 ;
G01 Z0.0 ;
```



```
X24.0 Z-2.0;
Z-16.0;
G02 X32.0 Z-20.0 R4.0;
G01 Z-36.0;
G03 X44.0 Z-42.0 R6.0;
N50 G01 Z-54.0;
G00 X55.0 Z2.0 M09;
G28 U0 W0 M05;
M01;
N2;
G28 U0 W0;
G50 S2600 T0700;
G96 S150 M03;
G00 X28.0 Z0.0 T0707 M08;
G01 X0.0 F0.1;
G00 X55.0 Z2.0;
G70 P40 Q50 F0.12;
G00 X65.0 Z2.0 M05;
M09;
G28 U0 W0;
M30;
```

G74 - Peck drilling cycle

Format

G74 R_;

G74 Z___Q___F___;

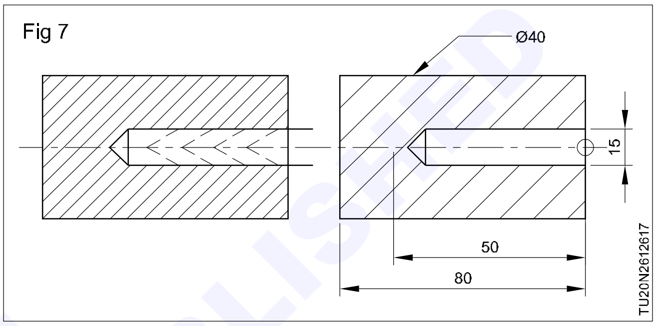
R : Retract value

Z : Depth of the hole

Q : Depth of cut per pass in Microns

F : Feed rate

```
%
O0029 (Fig 7)
```



```
N1; (C.D)
G28 U0 W0;
T0200;
G97 S1500 M04;
G00 X0.0 Z5.0 T0202 M08;
G01 Z-3.0 F0.12;
G00 Z5.0 M09;
G28 U0 W0 M05;
M01;
N2;
U0 W0;
T0700;
G97 SI500 M04;
G00 X0.0 Z5.0 T0707 M08;
G74 R4.0;
G74 Z-50.0 Q8000 F0.05;
G00 Z5.0 M09;
G28 U0 W0;
M05;
M30;
```

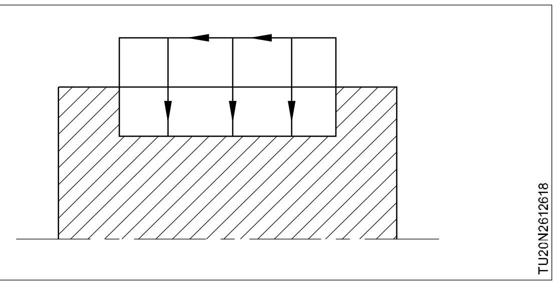
G75 - Grooving cycle (Fig 8)

Format:

G75 R___;

G75 X____Z___P____Q___R___F___;

Fig 8



R : Relieving the tool (mm)

X : Groove diameter (mm)

Z : Groove length (mm)

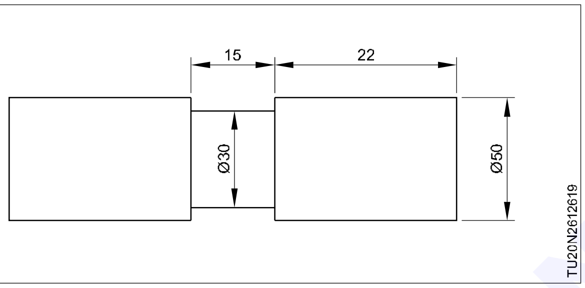
P : Depth of cut in X axis in microns (Radial value)

Q : Shift value in Z axis (microns)

F : Feed

```
%
O0030 (Fig 9)
```

Fig 9



```
G28 U0 W0 ;
T0500 ;
G97 S1200 M03 ;
G00 X52.0 Z5.0 T0505 M08 ;
G00 Z-26.0 ;
G75 R2.0 ;
G75 X30.0 Z-37.0 P500 Q3000 F0.05 ;
G00 X46.0 ;
Z5.0 M09 ;
G28 U0 W0 M05 ;
M30 ;
```

G76 - Multiple thread cutting cycle(Fig 10)

Format:

G76 P___Q____R___;

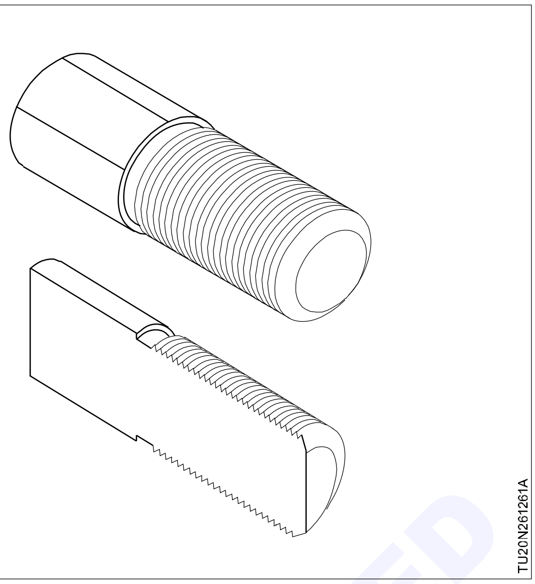
G76 X___Z___P___Q____F____;

Explanation for the cycle:

P : NCA

- N : Number of finishing passes
- C : Chamfer amount
- A : Included angle

Fig 10



Q : Minimum depth of cut in microns (radial value)

R : Finishing depth of cut in microns (radial value)

X : External threading (minor dia) internal threading(major dia)

Z : Thread length

P : Height of thread (microns)

Q : First depth of cut in microns (radial value)

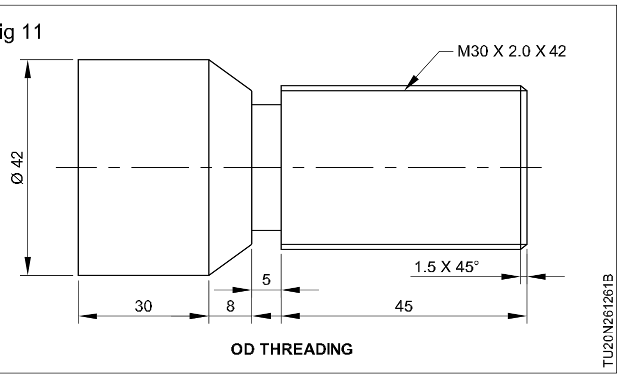
F : Feed (pitch of the thread)

Example

OD Threading

```
%
O0031 (Fig 11)
```

Fig 11



```
G28 U0 W0 ;
T0400 ;
G97 S600 M04 ;
G00 X32.0 Z5.0 T0404 M08 ;
G76 P030060 Q150 R20 ;
G76 X 27.54 Z-42.0 P1226 Q300 F2.0 ;
G00 X32.0 ;
Z5.0 M09 ;
G28 U0 W0 M05 ;
M30 ;
```

Threading calculation

Minor dia, d = D - (2h)

(h = 0.6134 p, for metric thread)

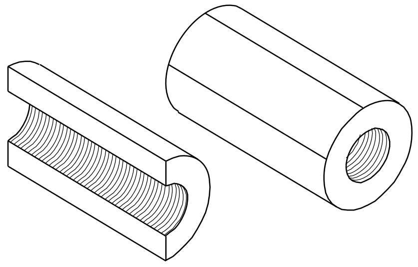
h = 0.6134   2.0=1.226 mm.

d = 30 - (2 x1.226)

= 27.54 mm.

ID Threading (Fig 12)

Fig 12

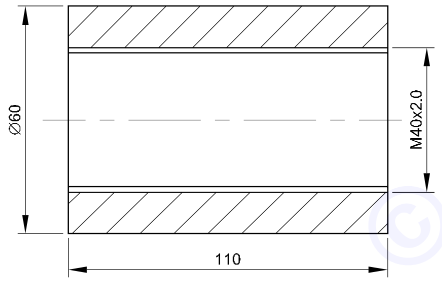


ID THREADING

%

O0032 (Fig 13)

Fig 13



```
G28 U0 W0 ;
T0500 ;
G97 S600 M04 ;
G00 X38.0 Z5.0 T0505 M08 ;
G76 P030060 Q150 R20 ;
G76 X40 Z-110.0 P1226 Q300 F2.0 ;
G00 X38.0 ;
Z5.0 M09 ;
G28 U0 W0 M05 ;
M30 ;
```

Threading calculation

Minor dia, d = D - (2h)

(h = 0.6134   P, for metric thread)

h = 0.6134 x 2 = 1.226 mm

d = 40 - (2 X 1.226)

=37.548 mm.

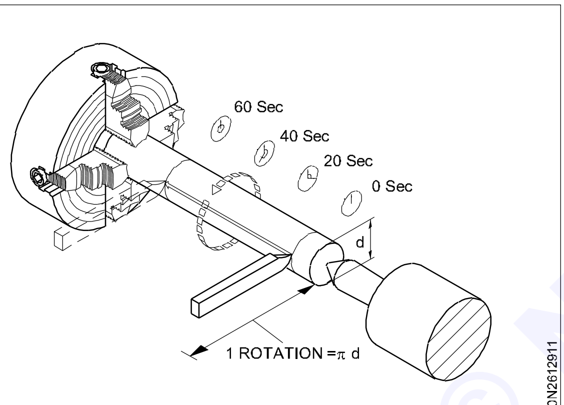
Cutting speed and feed

Objectives: At the end of this lesson you shall be able to
- **differentiate between cutting speed and feed**
- **state select the recommended cutting speed for different materials from the chart**
- **state the factors governing the cutting speed**
- **state the factors governing feed.**

Cutting speed (Fig 1)

Cutting speed is the speed at which the cutting edge passes over the material, and it is expressed in metres per minute. When a work of a diameter ‘d’ is turned in one revolution the length of the portion of work in contact with the work in contact with the tool is πxdxn. This is converted into metres and expressed in a formula form as

Fig 1



CUTTING SPEED FOR TURNING OPERATION

$$V = \frac{\pi \text{ x dn}}{1000} \text{ metre /min.}$$

Where V = cutting speed in m/min
π = 3.14
d = diameter of the work in mm
n = RPM

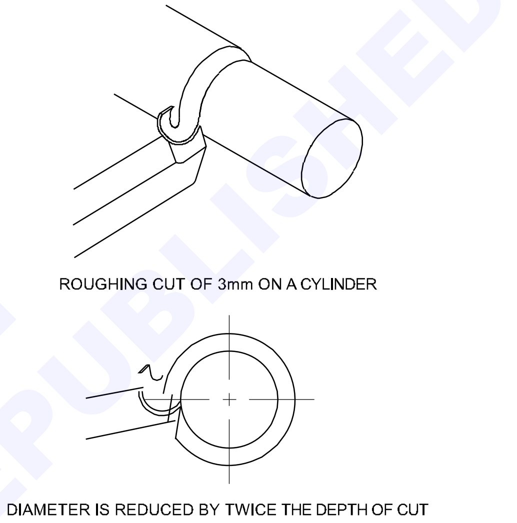
When more material is to be removed in lesser time, a higher cutting speed is needed. This makes the spindle to run faster but the life of the tool will be reduced due to more heat being developed. The recommended cutting speeds are given in a chart. As far as possible the recommended cutting speeds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

Example

Find out the rpm of a spindle of a 50mm bar to cut at 25m/min. $V = \frac{\pi \times DN}{1000}$; $n = \frac{1000\text{ V}}{\pi \times D}$

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

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

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

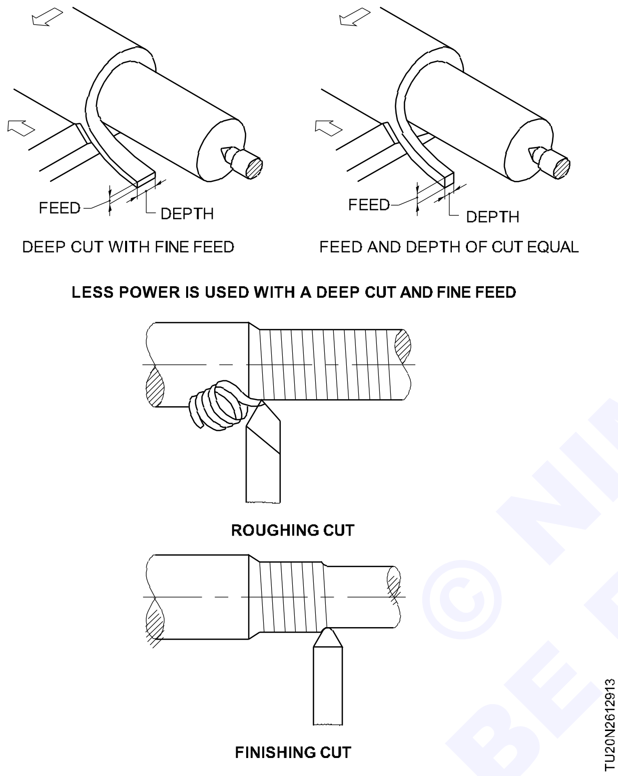
surface finish required on the work

rigidity of the tool

coolant used

Cutting speed 120m/min	length of metal passing cutting tool in 1 revolution	Calcualted r.p.m. of spindle
	____ 78.56mm	1528
	____ 157.12mm	756
	____ 235.68mm	509.3

Fig 3



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.

For super HSS tools the feeds would remain the same, but cutting speeds could be interested 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

Cutting speeds and feeds for H.S.S. tools

Depth of cut

The depth of cut is the difference between machined and un machined surface.

If D1 = initial diameter

D2 = Final diameter

$$\text{Depth of cut} = \frac{D1-D2}{2}$$

TABLE 1

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 steel)	0.2-1.00	35-80
(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

The relationship between limiting spindle speed and constant cutting speed

When cutting in the constant cutting speed mode, as the tool moves towards the axis, the spindle speed increases.

The spindle speed N in RPM is calculated using this equation , where V is the cutting speed and D is the diameter at which the tool is cutting.

$$V= \frac{\pi \times D \times N}{1000} \quad \text{hence } N= \frac{1000 \times V}{\pi \times D}$$

At a cutting speed of 250, at 30 mm. Diameter thee RPM would be 2652. At 20 dia. The RPM would be 3978. At 1 mm. dia. The RPM would be 79,577.

At a certain diameter the spindle speed goes beyond the machine's capability. At the axis of the part, in fact, the RPM would theoretically be infinity (D is zero). The machine however has a certain maximum spindle RPM, so in the CNC program we need to specify what this maximum is. This is specified as Limiting spindle speed. When the spindle speed reaches this value, the controller clamps it at this speed and the rest of the motion is done at a constant spindle speed equal to the limiting speed. E.g., if we want to cut at a constant cutting speed of 250 m/min and limit the RPM to 3000, for Fanuc we would write this

```
G96 S250
G92 S3000
```

So what should I program as the limiting spindle speed? If the part is held rigidly in the chuck and is circular, just set the limiting spindle speed to the machine's maximum spindle RPM. If the part is non-circular or is held in a fixture that is not balanced, centrifugal forces might cause the part to fly off or damage the fixture.

Operational modes

- Auto mode
- Edit mode
- MDI mode (Manual data input)
- Jog mode
- MPG mode (manual pulse generation) data
- Input/output mode
- Zero return mode

I Selection of tools, speed feed & depth of cut

D = work piece diameter mm

V = cutting speed m/min

S = Feed mm/r

N = RPM r/min

A = Depth of cut mm

N = Efficiency for example 0.75

Ks = Specific cutting force N/mm^2

V = Metal removal rate cm^3/min

P = power required kW

R = nose radius mm

K = Constant for example 1.4

R_t = Profile depth µm

R_a = Surface finish µm

II Approximate value for power required

The above formula for power is exact but the specific cutting force ks is included. The ks-value is hard to set because it is dependent on many factors. Such as work piece materials, chip breaker, cutting rake, feed setting angle chip thickness.

RPM	$n = \frac{v.1000}{\pi D}$ =r/min
Cutting speed	$v = \frac{\pi.D.n}{1000}$ m/min
Metal removal rate	V = v.s.A cm^3/mir
Power required	$P = \frac{v.s.A.ks}{6000\eta}$ kW

A simplified formula for approximate power required is shown below, Based on the most common type of application-medium rough to rough turning of normal steel, with a light-cutting edge-a specific cutting force. Ks=1800 N/mm² and a machine efficiency factor=0.75 are used.

$$P = \frac{V.S.A}{25} KW$$

III Approximate value for surface finish

$$\text{Profile depth } R_t = k\frac{s^2 \ 1000}{8r}$$

The constant k is depends on two factors, the work piece material and how well the cutting edge profile is reproduced on the work piece. In normal machine steel k=1.4, Surface finish Ra = R_t/3.5.

Surface Ra = s^2.50/R

IV Effect of nose radius and feed rate on the surface finish requirements

The table below gives the recommended maximum values of feed rate for finishing normal steels, when turning materials which give rise to edge build-up, the cutting speed must be sufficiently, high to avoid such tendencies, if possible When turning highly abrasive materials, the feed rates should be reduced by about 20% .To convert Ra to CLA multiply by 40.

	Nose radius, mm					
	0.2	**0.4**	**0.8**	**1.2**	**1.6**	**2.4**
Ra value	**Feed rate, mm/rev**					
0.6	0.05	0.07	0.10	0.12	0.14	0.17
1.6	0.08	0.12	0.16	0.20	0.23	0.29
3.2	0.12	0.26	0.23	0.29	0.33	0.40
6.3		0.23	0.33	0.40	0.47	0.57
8.0			0.40	0.49	0.57	0.69

A large nose radius will usually result in a better surface finish, provided that the cutting edge is sufficiently sharp and that the larger nose radius does not give rise to vibrations. It is recommended that the depth of cut for finishing should be more than the nose radius of the chosen insert. Filets, etc on the component often restrict the choice of nose radius on finishing.

V Cutting speed-wear life

Providing the machining conditions are good i.e. stability of the work piece and tool, it is possible to increase the wear life of the insert.

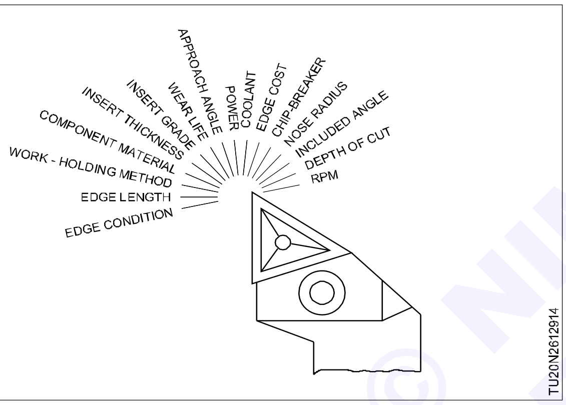
To achieve longer wear life, the cutting speed must be reduced. Multiply the recommended cutting speeds by the following factors.

The cutting speeds given in this guide are for 30 min.wear life. If higher surface speeds are required that wear life will decrease.

Approx. wear life Mm	Factor
* 15	1.25 x V
* 30	1.0 x V
45	0.89 x V

VI Edge condition factors (Fig 4)

Fig 4



- Fixed conditions
- Material specification
- Amount of material to be removed
- Component dimension
- Component shape
- Hardness
- Surface condition
- Operation
- Finish requirement
- Type of machine
- Condition of machine
- Power available
- Chucking or clamping method

Once the fixed conditions have been considered, the tooling and data parameters can be variable conditions

- Select carbide grade
- Select radius
- Select insert shape
- Select insert size
- Select insert rake
- Select tool size
- Select tool-holder shank size
- Select tool-holder style

Now the cutting speed, depth of cut and the feed over revolution can be selected.

Right - Hand tool vs Left-Hand tools for turning

Objective: At the end of this lesson you shall be able to
- **differentiable between right hand vs left hand turning in terms of speed.**

The turning operation, carried out on lathe, cutting tools (called single point turning tool) is mounted on tool post and the entire tool post moves by various mechanical arrangements to provide necessary feed motion. Since the tool carriage can move both-left and right directions, so it is necessary to pre-set the cutting tool on the tool post based on desired direction of movement.

For two different directions, two different tools are required, and accordingly single point turning tool can be classified into two-categories -right handed cutting tool and left handed cutting tool.

Almost all current model slant bed turning centres allow machinists to use either right - or left-nand tooling.

If holding tools designed to machine in a direction toward the word holding device (Z minus), right-hand tools require an M03 (spindle forward) direction, and left-hand tools required an M04 (spindle reverse) direction. Since most cutting tools are more readily availbale in right-hand versions (right-hand tools are commonly less expensive and are purchased off the shelf), and because some operations must be performed with right-hand tools (tapping and right-hand threading, for example), most CNC users prefer using right-hand tools.

Keep in mind, however, that your turning centre's rigidity and strength will commonly be better with left-hand-tooling. It is notice that wehen left-hand tools are used, the force of the machining operation is thrown into the machine's bed and machining will be very stable. On the other hand, notice that when right-hand tools are used, the force of the machining operation tends to pull the turret away from the machine bed, and machining is less stable.

For light duty machining, it truly doesn't matter which type of tooling you use. But as machining operations become more powerful (as is the case when performing heavy rough turning and boring operations), left-hand tools are better. You would be best off using them and scarificing the cycle time it takes for spindle reversals. And by the way, if you experience problems with inconsistent sizing (even for finishing operations) on tight tolerances, using left-hand tools will improve the stability of machining.

Selection of cutting parameters from a tool manufacturer's catalogue for various operations

Objective: At the end of this lesson you shall be able to
- **select the cutting tool depending of the operations and cutting conditions.**

Guide lines for the selection of cutting tools

- First you have to study the drawing thoroughly
- Notice the material and its properties type of material, hardness and machinability
- Draw the outer profile separately
- Dimension all grooves positive arcs, negative arcs and champers
- Select roughing and finishing cutting speeds, feeds and depth of cuts
- Select OD rough turning tool, grooving, threading such tools
- Select ID rough boring tool, grooving, threading such tools
- List out the selected tools
- Decide which make tool you have to be used. Ex.

  Widia,

  Sandvik,

  Scar….
- Take the respective catalogue and find the listed tools and their respective inserts depend on the hardness and nature of material.

Example 1 for the selection of cutting tool from the manufactures catalogue.

First step

Define material and type of operation

Define material according to ISO 'P', 'M' and 'K' and identify the operation from the table of contents

Second step

Define application and machining conditions

Locate first choise of insert geometry and grade by application.

F - Finishing

M - Medium

R - Rough

Conditions

○ - Good

◑ - Normal

● - Difficult

Third step

Chose insert with recommended cutting data

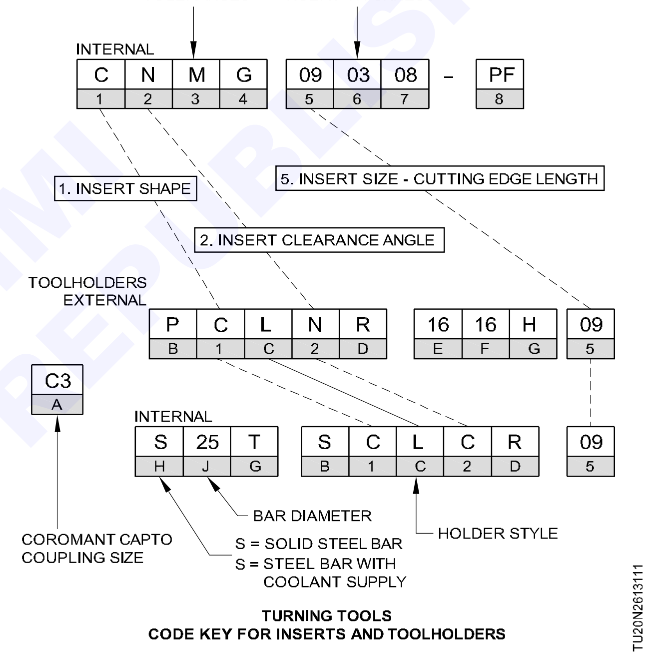
Select the insert from the ordering data and note down the speed, feed and depth of cut recommended.

Fourth step

Choose tool holder

Select the tool holder using the insert shape and size

Fig 1



TURNING TOOLS
CODE KEY FOR INSERTS AND TOOLHOLDERS

External machining

CoroTurn® RC

External machining, from roughing to finishing

CoroTurn® TR

First choice for external profiling

CoroTurn® 107

External machining of small, long and slender components

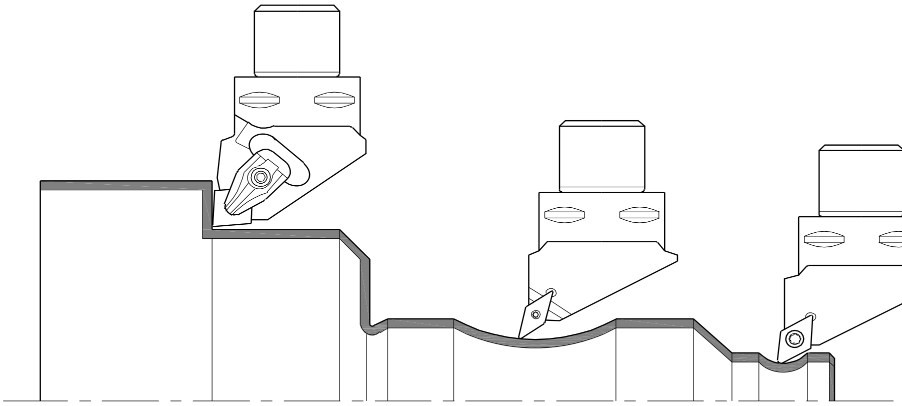
General points to consider

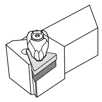
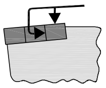
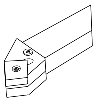
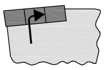
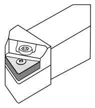
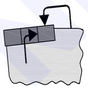
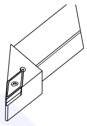
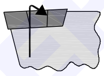
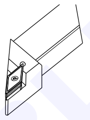
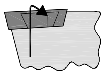
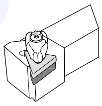
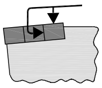
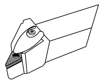
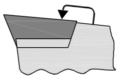
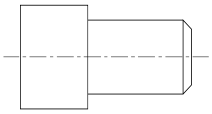
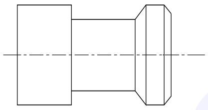
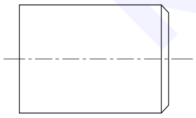
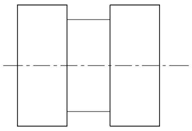
1 Use an entering angle less than 90° (lead angle larger than 0°), if possible, to reduce the impact and the forces.

2 First recommendation is to use Coromant Capto® cutting units.

3 When using conventional tools, use the largest tool holder shank possible, for maximum stability.

Fig 2



Tooling system Coromant Capto® Shank holder cutting units	Negative basic-shape inserts			Positive basic-shape inserts		Ceramic and CBN inserts	
	CoroTurn® RC A115 A137 -	T-Max P A124 A152 I12 -	 A134 A159 -	CoroTurn® 107 A166 A174	CoroTurn TR A193 A195 I14	CoroTurn® RC A200 A208 -	T-Max® A207SL A218 -
	 Rigid clamp design 	 Lever design 	 Wedge clamp design 	 Screw clamp design 	 Screw clamp design 	 Rigid clamp design 	 Top clamp design 
Longitudinal turning/ facing 	• •	•	•	•	•	• •	•
Profiling 	• •	•	•	• •	• •	• •	•
Facing 	• •	•	•	•		• •	•
Plunging 		•		• •			• •

• • Recommended tooling system • Alternative tooling system

Internal machining

CoroTurn® XS: Internal machining of extra small hole diameters, starting at 0.3 mm (.012 inch) diameters (Small part machining)

CoroCut® MB: Internal machining of small holes diameters, starting at 10 mm (.394 inch) diameter)

T-Max P: Internal turning of holes from 20 mm (.750 inch) in diameter with short tool overhangs and stable conditions.

CoroTurn® 111: For optimization of internal turning operations requring small cutting forces when machining with long tool overhangs.

CoroTurn® 107

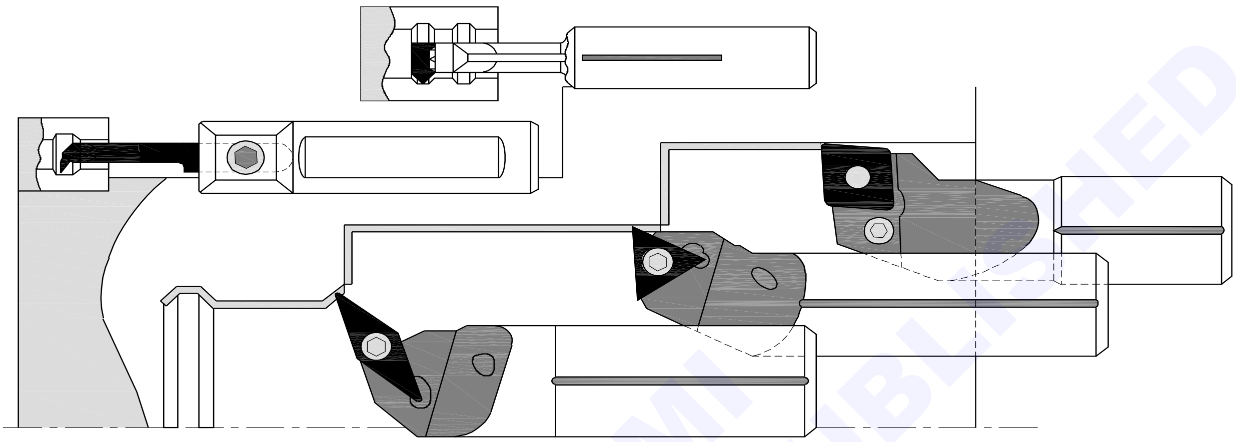
First choice for internal machining of small and medium holes from 6 mm (.236 inch) diameter.

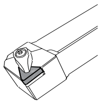
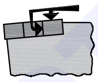
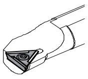
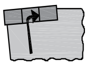
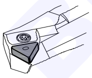
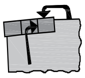
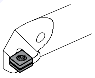
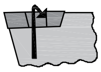
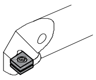
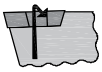
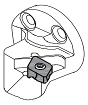
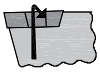
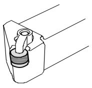
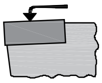
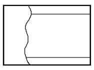
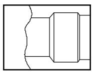
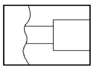
General points to consider

1 Use an entering angle close to 90° (lead angle 0°) but never less than 75° (never more than lead angle 15°), to reduce bar deflection and vibration.

2 Use the largest bar size and smallest possible bar overhang, to provide maximum stability.

Fig 3



Tooling system External machining	Negative basic-shape inserts			Positive basic-shape inserts		Ceramic and CBN inserts	
Generant cable cutting units for multi	CoroTurn® RC	T-Max P		CoroTurn® 107	CoroTurn TR 111	CoroTurn® TR	T-Max®
task machining	A261	A263	A266	A280	-	-	-
shank holder	A269	A273	A275	A286	A309	-	A319
SL cutting units	I21	I16	-	I27	I32	I18	-
	 Rigid clamp design 	 Lever design 	 Wedge clamp design 	 Screw clamp design 	 Screw clamp design 	 Screw clamp design 	 Top clamp design 
Longitudinal turning/ facing 	• •	• •	•	• •	• •	•	•
Profiling 	•	•		• •	• •		
Facing 	•	•		• •	•	•	•

• • Recommended insert shape • Alternative insert shape

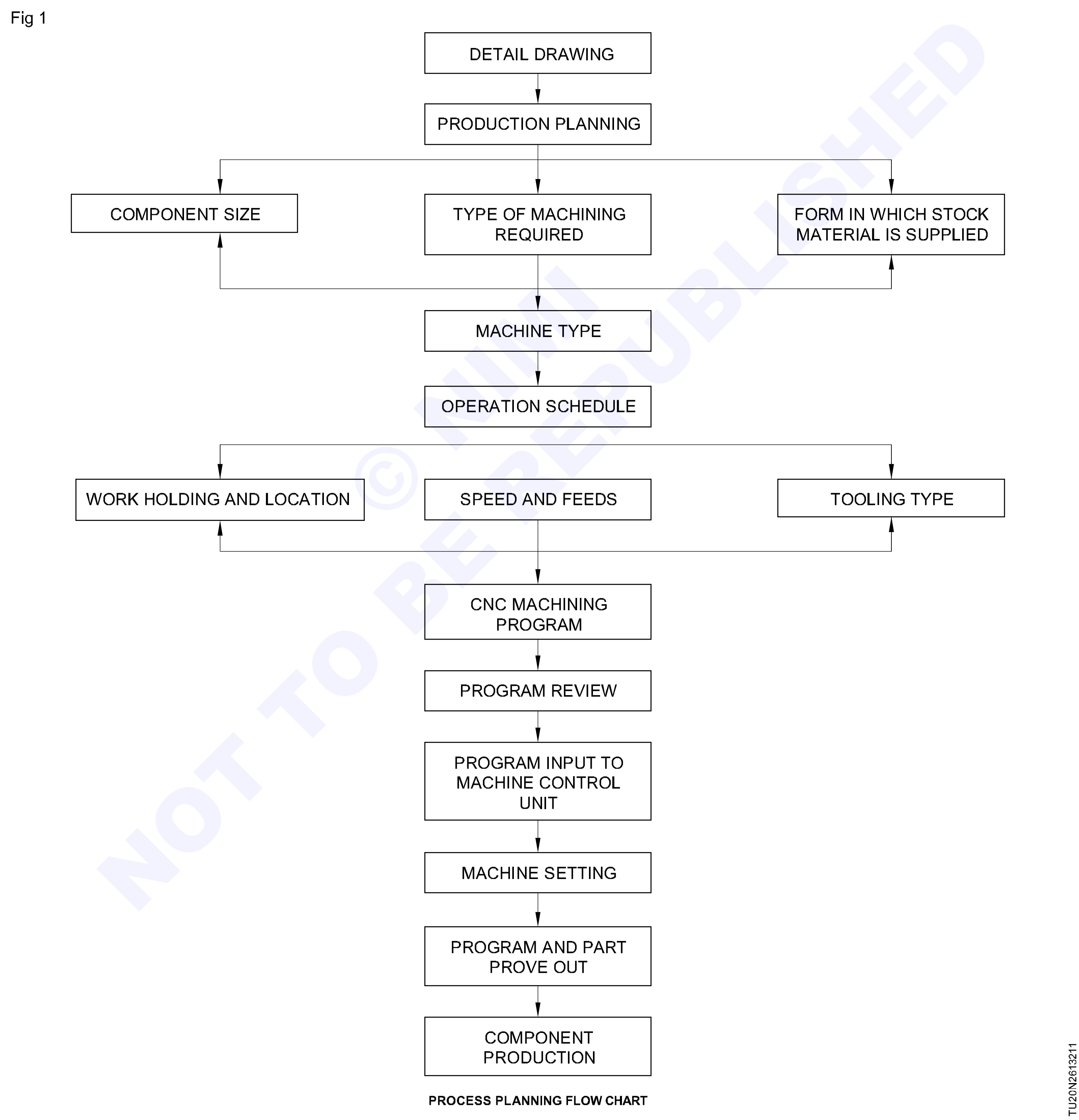
Process planning, sequencing tool layout & selection

Objectives : At the end of this lesson you shall be able to
- **state the process planning**
- **name the types of process planning**
- **state about the machining sequence**
- **state about the tool layout.**

Process planning

Process planning flow chart

Fig 1



PROCESS PLANNING FLOW CHART

Process planning is a preparatory step before manufacturing, which determines the sequence of operations or processes needed to produce a part or an assembly. This step is more important in job shops, where one - of - a kind products are made or the same product is made infrequently.

The manufacturing process begins with the process planning and ends with actual product. Process planning is considered the back bone of manufacturing, since it attempts to determine the most efficient sequence to produce the product quickly and inexpensively as possible.

A process planner must be aware of the various aspects of manufacturing to plan properly. The planner works typically with blue prints and may have to communicate with the design department of the company to clarify or request changes in the final design to fit manufacturing requirements. The outcome of process planning is a production plan, tools procurement, material order and machine programming. Other special manufacturing needs such as design of jigs and fixtures are planned.

Numerical control is concerned with controlling the operation of a single machine, but process planning considers the sequence of production steps needed to make a part from start to finish, generally using successive operations on several machines. The planning describes the routine of the work piece through the shop floor and its state at each work station.

Flow diagrams and other information such as part specifications, tooling requirements and machining conditions can be used to develop a production sequence for fabricating the part in the fastest, most economical manner.

Once the process planning phase is completed, the actual production of the product begins. The produced parts are inspected and usually must pass certain standard quality control (assurance) requirements. Parts that survive inspection are assembled, packaged, labeled and shipped to customer.

An important part of process planning is a concept called "group technology" (GT). This is a manufacturing philosophy, that takes advantage of the similarities among parts and processes. Instead of treating each part as unique, group technology organizes the parts in to families according to either similar shape or common manufacturing operation.

Types of process planning

The Types are

1 Manual process planning

2 Automated process planning

3 Generative process planning.

Machining sequence

Machining sequence defines the order of machining operations. Technical skill and machine shop experience does help in program planning, but some common sense approach is equally important. The sequence of machining must have a logical order - for example, drilling must be programmed before tapping, roughing operations before finishing, first operation, before second etc. With in this logical order, further specification of the order of individual tool motions is required for a particular tool. For example in turning, a face cut may be programmed on the part first,and then roughing all material on diameters will take place. Other example is, to program a roughing pass for the diameter, then face and continue with the remainder of the diameter roughing after words. In drilling, a center drill before drilling may be useful for some applications, but in another program a spot drill may be a better choice. There is no fixed rule, on which method is better - each CNC programming assignment has to be considered individually, based on the criteria of safety and efficiency.

The basic approach for determining the machining sequence is the evaluation of all related operations. In general, program should be planned in such a way that the cutting tool, once selected, will do as much work as possible, before a tool change. On most CNC machines, less time is needed for positioning the tool than for a tool change. Another consideration is in benefits gained by programming all heavy operations first, then the lighter semi finishing or finishing operations. It may mean an extra tool change or two, but this method minimizes any shift of the material in the holding fixture while machining. Another important factor is the current position of a tool when a certain operation is completed. For example, when drilling a pattern of holes in the order of 1-2-3-4, the next tool (such as a boring bar, reamer or tap) should be programmed in the order of 4-3-2-1 to minimize unnecesary tool motions.

T01- Spot drill	T02- drill	T03- Tap
Hole - 1	Hole - 4	Hole - 1
Hole - 2	Hole - 3	Hole - 2
Hole - 3	Hole - 2	Hole - 3
Hole - 4	Hole - 1	Hole - 4

Typical machining - sequence (Spot drill, drill and tap shown as an example).

Typical turning - sequence (facing, rough turining, finish turning, grooving threading etc.

This machining sequence may have to be changed after the final selection of tools and the set up method. The reverse sequence may not be practical in sub programs.

Tool layout

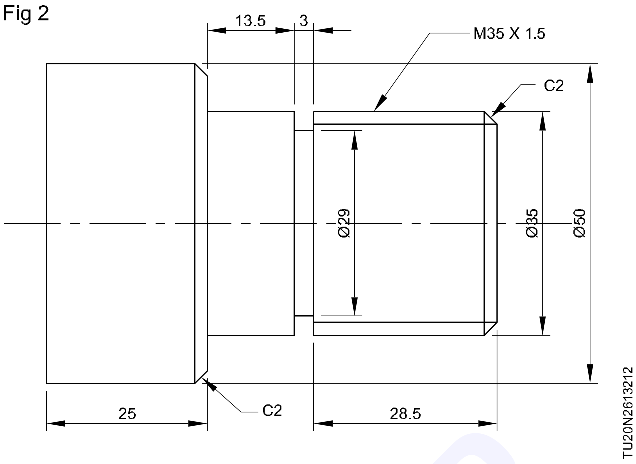
The tool layout for a job constitutes the predetermined plan for machining operation of a particular component the layout is dependent upon the number of pieces to be manufactured in lot size.

As a general rule, standard tools should be used as much as possible and also for small batches of work, the layout should be simple for large quantities and long run special tools should be used. The accuracy and cost of component largely depends upon the tool layout.

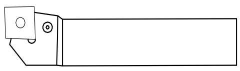
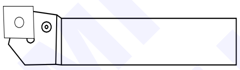
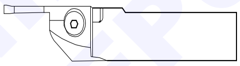
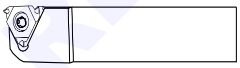
For preparation of the tool layout, it is necessary to have the finished drawing of the part to be machined and if is a forging or casting will determine how much machining has to be done

Example of tool layout

Fig 2



Tool layout

Sl.No	Operation	Tool	Tool geometry name	Cutting speed	Depth of cut	Feed
1	OD Rough Turning	PCLNL 2525 M12 CNMG 120408 - Insert		180	2	0.2
2	OD Finish Turning	PCLNL 2525 M12 CNMG 120404 - Insert		180	1	0.2
3	OD Grooving	Groove tool holder LH		150	3	0.15
4	OD Threading	LH thread holder 25x25x150mm lenth DEG., DEPTH 3.0, LH		100	0.2	Pitch 1.5

Cutting parameters

Objectives : At the end of this lesson you shall be able to
- **cutting speed (surface speed)**
- **feed rate and**
- **depth of cut.**

Cutting speed

Cutting speed is the peripheral linear speed of the part passing the tool, in meters/min. It is also called the surface speed. (Fig 1)

$$V = \frac{\pi DN}{1000}$$

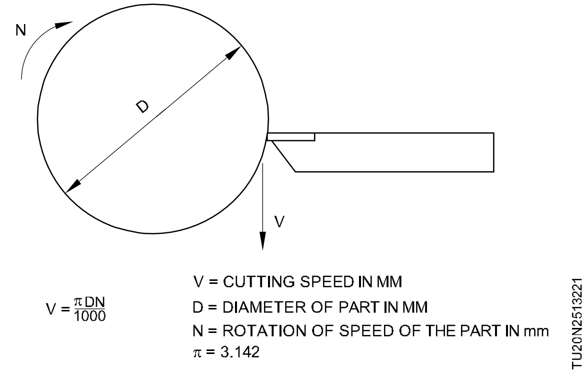
V = Cutting speed in m/min

D = Diameter of part in mm.

N = rotation speed of the part in rpm

= 3.142

Fig 1



Spindle speed remaining constant, the cutting speed changes as the part diameter at which the tool is positioned changes. In case (Fig 2) below,the cutting speed remains constant as the tool moves from point. A to point B, since the diameter is the same. In case 2 (Fig 3) the cutting speed increases and in case 3 (Fig 4) it decreases.

Fig 2

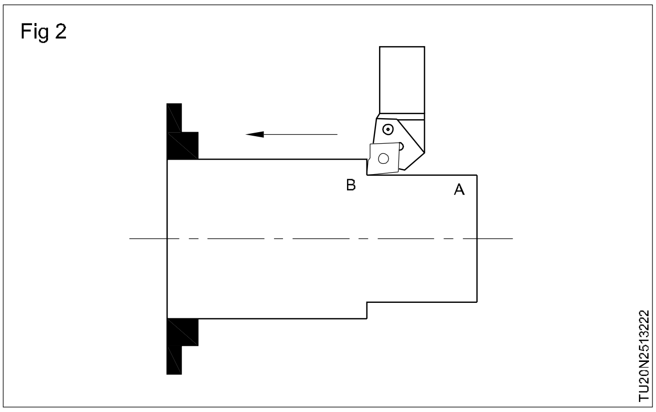


Fig 3

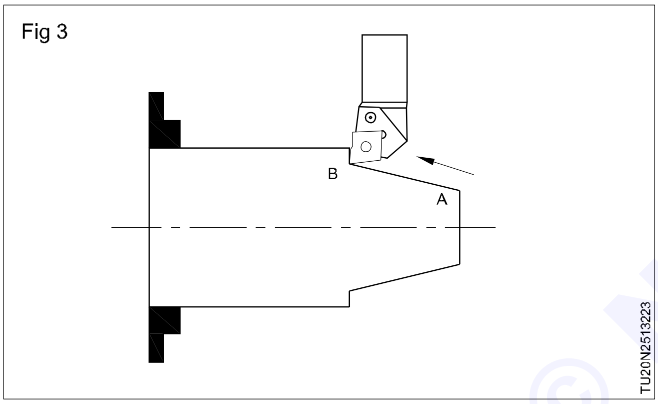
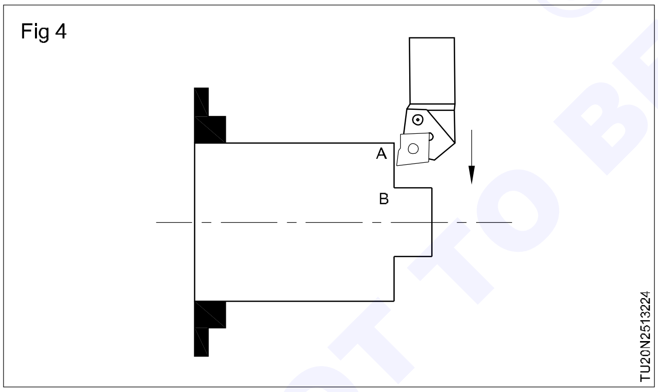


Fig 4



Spindle speed

The spindle speed is the rotary speed of the part in revolutions per minute (RPM). It can be calculated from the cutting speed using the following equation:

$$N = \frac{1000 \text{ x } Vs}{\pi D}$$

Feed Rate

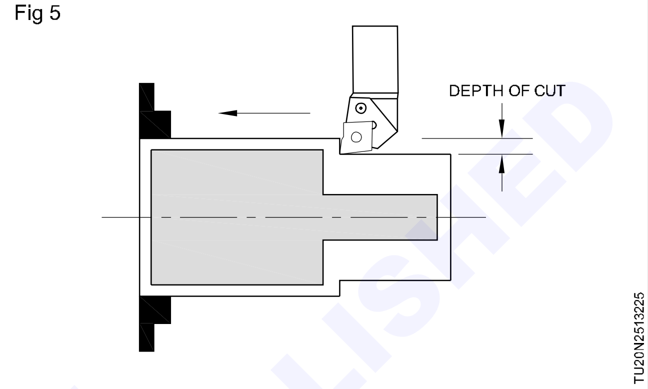
Feed rate is the rate of linear travel of the tool. It is programmed as the distance travelled per minute or per revolution. On CNC lathes it is usually programmed as the mm/rev. The relationship between feed rate per minute and feed rate per revolution is:

Feed per minute= Spindle speed X Feed per revolution

Depth of cut

Depth of cut is the distance from the current tool position to the previous material.(Fig 5)

Fig 5



Tool life

As a tool cuts material, its surface gradually gets eroded The tool wear causes a reduction in dimensional accuracy and surface finish quality, and increase in cutting forces. The tool wear increases with increase in the cutting parameters, but all cutting parameters do not affect it to the same extent. The parameters in decreasing order of importance are:

Surface speed

Feed rate

Depth of cut

Too high a cutting speed and feed rate will result in early tool failure, poor dimensional accuracy and poor surface finish, and higher tool cost. Too low cutting speed and feed rate will lead to increased machining time and greater cost per part. The cutting parameters must therefore be selected with care.

Cutting parameters for CNC turning applications

Speed / feed /depth of cut for general turning

	Part Material	Cutting speed Vc m/min	Feedrate /mm/rev			Depth of cut mm	
			Rough	Finish	Rough	Finish	
						0.4 R	0.8R
Steel	Steel 130 - 180 BHN	240 - 320	0.2 - 0.35	0.1 - 0.2	1 - 3	0.25 - 0.4 Radial	0.3 - 0.5 Radial
	Steel 180 - 250 BHN	160 - 240	0.2 - 0.35	0.1 - 0.2	1 - 3	0.25 - 0.4 Radial	0.3 - 0.5 Radial
	Cast Steel 180 - 250 BHN	140 - 200	0.2 - 0.35	0.1 - 0.2	1 - 3	0.25 - 0.4 Radial	0.3 - 0.5 Radial
SS	Stainless steel Bar/ Forged 200 BHN	100 - 140	0.2 - 0.3	0.1 - 0.2	1 - 3	0.2 - 0.4 Radial	0.3 - 0.5 Radial
	Stainless steel Casting 200 - 330 BHN	75 - 140	0.2 - 0.3	0.1 - 0.2	1 - 3	0.2 - 0.4 Radial	0.3 - 0.5 Radial
C I	Grey CI 180 - 260 BHN	180 - 250	0.2 - 0.3	0.15 - 0.2	1 - 3	0.2 - 0.4 Radial	0.3 - 0.5 Radial
	Nodular CI 250 BHN	160 - 220	0.18 - 0.25	0.15 - 0.2	1 - 3	0.2 - 0.4 Radial	0.3 - 0.5 Radial
AL	Aluminium 60 - 100 BHN	500 - 1000	0.25 - 0.5	0.1 - 0.2	2 - 5	0.25 - 0.6 Radial	0.4 - 1.0 Radial
	Aluminium Cast 75 - 130 BHN	400 - 800	0.2 - 0.4	0.1 - 0.2	2 - 5	0.25 - 0.6 Radial	0.4 - 0.8 Radial

Speed / feed / width of cut for grooving

	Part Material Vc m/min	Cutting speed	Feedrate /mm/rev		Width of cut for plunge type Rough %
			Rough	Finish	
Steel	Steel 130 - 180 BHN	120 - 180	0.08 - 0.2	0.05 - 0.1	70 - 80% of tool width
	Steel 180 - 250 BHN	100 - 150	0.08 - 0.2	0.05 - 0.1	
	Cast Steel 180 - 250 BHN	80 - 120	0.08 - 0.2	0.05 - 0.1	
SS	Stainless steel Bar/ Forged 200 BHN	70 - 120	0.08 - 0.2	0.05 - 0.1	
	Stainless steel Casting 200 - 330 BHN	60 - 110	0.06 - 0.15	0.05 - 0.1	
CI	Grey CI 180 - 260 BHN	80 - 150	0.08 - 0.2	0.05 - 0.15	
	Nodular CI 250 BHN	60 - 110	0.06 - 0.15	0.05 - 0.15	
AL	Aluminium 60 - 100 BHN	250 - 400	0.15 - 0.3	0.05 - 0.15	
	Aluminium Cast 75 - 130 BHN	200 - 350	0.15 - 0.3	0.05 - 0.15	

Threading

	Part material	Cutting speed Vc m / min	No cutting passes pitch						Thread depth	Ist DOC Pitch			
			0.75	1	1.5	2	2.5	3		1	1.5	2	3
Steel	Steel 130 - 180 BHN	60 - 120	5	6	8	10	13	15	0.65 X Pitch	0.16	0.2	0.22	0.25
	Steel 180 - 250 BHN	50 - 90	6	6	9	11	14	17					
	Cast steel 180 - 250 BHN	40 - 70	7	7	10	12	15	19					
SS	Stainless steel Bar / Forged 200 BHN	30 - 60	7	8	10	12	16	19		0.16	0.2	0.22	0.25
	Stainless steel Casting 200 - 330 BHN		7	8	11	14	18	20					
C I	Grey CI 180 - 260 BHN	60 - 110	5	6	8	10	13	15					
	Nodular CI 250 BHN	50 - 90	6	7	10	12	14	16		0.18	0.22	0.25	0.28
	Aluminium 60 - 100 BHN	150 - 220	5	6	8	10	12	15					
AL	Aluminium cast 75 - 130 BHN	120 - 200	5	6	8	11	14	16		0.18	0.22	0.25	0.3

Note: Feed rate mm/ min should not cross 2500 - 3000 mm/min on machine

To check: Use this formula: calculated RPM X (Pitch X No. of starts)= mm/min on machine

Speed / feed for drilling

Cutting speed m/min

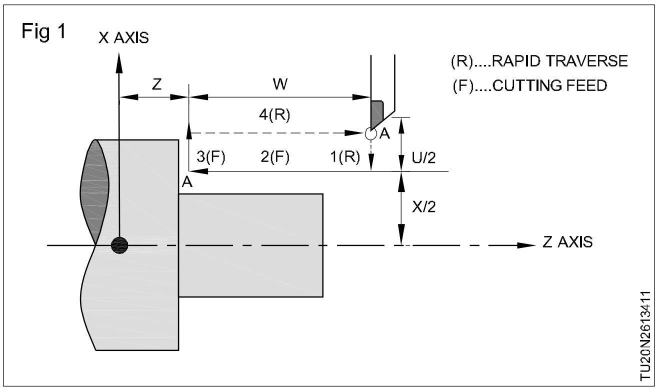
	Para material	HSS	F /rev		Solid carbide	F/rev		Insert type	
		Vc	Dia <10	Dia >10	Vc	Dia <10	Dia >10	Vc	F/rev
Steel	Steel 130 - 180 BHN	25-35	0.05-0.12	0.12-0.25	50 - 80	0.05 - 0.12	0.12 - 0.25	120 - 180	0.08 - 0.15
	Steel 180 - 250 BHN	20 - 30	0.05-0.12	0.12-0.25	40 - 70	0.05 - 0.12	0.12 - 0.25	100 - 150	0.08 - 0.15
	Cast steel 180 - 250 BHN	20 - 30	0.05-0.12	0.12-0.25	40 - 65	0.05 - 0.12	0.12 - 0.25	80 - 120	0.06 - 0.12
SS	Stainless steel bar/Forged 200 BHN	15 - 25	0.05-0.12	0.12-0.20	35 - 50	0.05 - 0.12	0.12 - 0.2	80 - 100	0.06 - 0.12
	Stainless steel casting 200 - 300 BHN	15 - 20	0.05-0.12	0.12-0.20	30 - 50	0.05 - 0.12	0.12 - 0.2	70 - 100	0.06 - 0.12
C I	Grey CI 180 - 260 BHN	25 - 40	0.05-0.12	0.12-0.30	60 - 90	0.05 - 0.12	0.12 - 0.3	180 - 250	0.1 - 0.2
	Nodular CI 250 BHN	25 - 35	0.05-0.12	0.12-0.25	50 - 80	0.05 - 0.12	0.12 - 0.25	150 - 220	0.1 - 0.2
AL	Aluminium 60 - 100 BHN	50 - 80	0.08-0.15	0.15-0.30	150 - 250	0.08 - 0.15	0.15 - 0.3	250 - 350	0.12 - 0.2
	Aluminium cast 75 - 130 BHN	50 - 80	0.08-0.15	0.15-0.30	150 - 250	0.08 - 0.15	0.15 - 0.3	250 - 350	0.12 - 0.2

Machining operation and tool path

Objectives : At the end of this lesson you shall be able to
- **describe the tool path in straight turning cycle**
- **brief the tool path in stock removal in turning operation**
- **explain the tool path in stock removal in facing operation**
- **enumerate the tool paths in grooving, threading and drilling operation in lathe.**

Straight cutting cycle (Fig 1)

Fig 1



Operations A straight cutting cycle performs four operations:

1 Operation 1 moves the tool from the start point (A) to the specified coordinate of the second axis on the plane (specified X-coordinate for the ZX plane) in rapid traverse.

2 Operation 2 moves the tool to the specified coordinate of the first axis on the plane (specified Z-coordinate for the ZX plane) in cutting feed. (The tool is moved to the cutting end point (A') in the direction of the length.)

3 Operation 3 moves the tool to the start coordinate of the second axis on the plane (start X-coordinate for the ZX plane) in cutting feed.

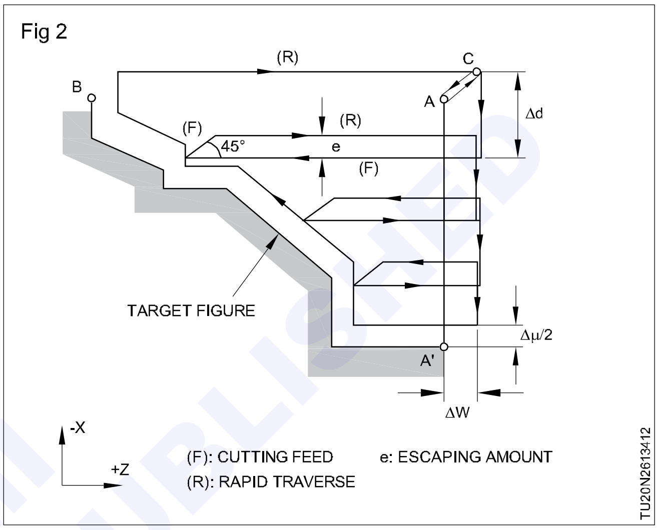
4 Operation 4 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).)

Stock Removal in Turning (Fig 2)

Operations

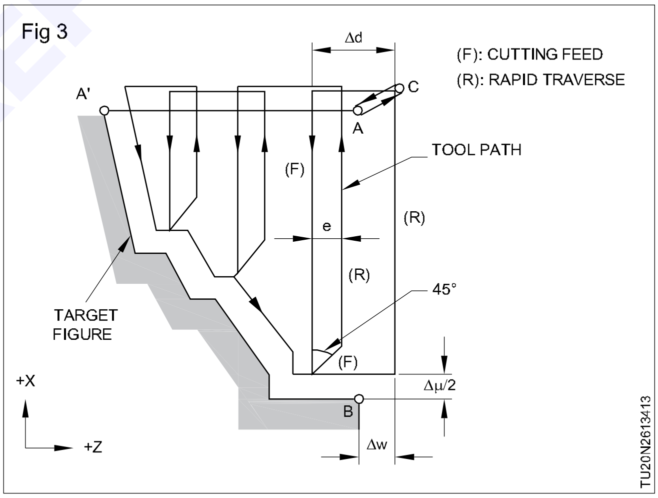
When a target figure passing through A, A', and B in this order is given by a program, the specified area is removed by Δd (depth of cut), with the finishing allowance specified by Δμ/2 and Δw left. After the last cutting is performed in the direction of the second axis on the plane (X-axis for the ZX plane), rough cutting is performed as finishing along the target figure. After rough cutting as finishing, the block next to the sequence block specified at Q is executed.

Fig 2



Stock Removal in Facing (Fig 3)

Fig 3



Cutting path in stock removal in facing

Operations

When a target figure passing through A, A', and B in this order is given by a program, the specified area is removed by Δd (depth of cut), with the finishing allowance specified by Δμ/2 and Δw left.

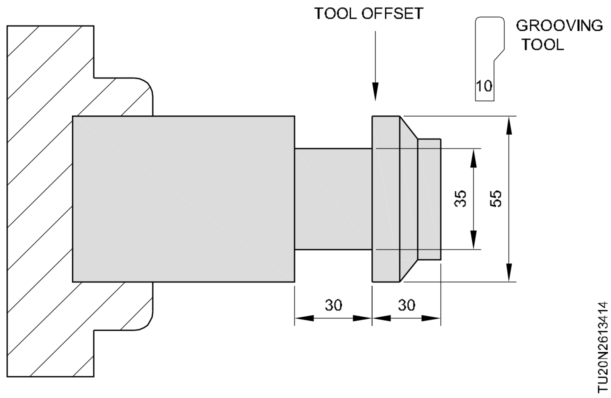
Grooving tool path (Fig 4)

Example program

N70 G75 R1 ;

N80 G75 X35 Z-60 P3000 Q9000 ;

Fig 4



N80- Grooving cycle command , grooving depth on x-axis is 35 , last groove position in z-axis is 60 , Peck increment in x-axis 3000 micron = 3 mm , stepping in z-axis is 9000 micron = 9 mm (next groove by moving 9mm in z-axis)

Multiple Threading Cycle (Fig 5&6)

Fig 5

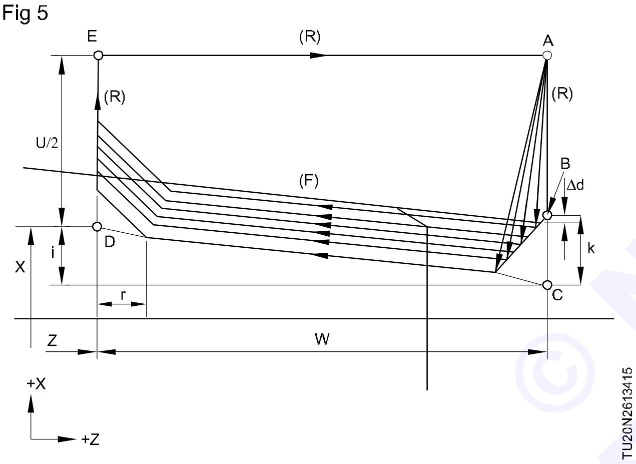
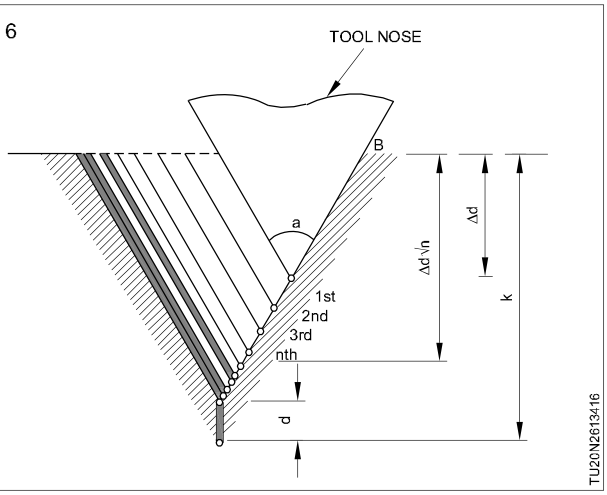


Fig 6



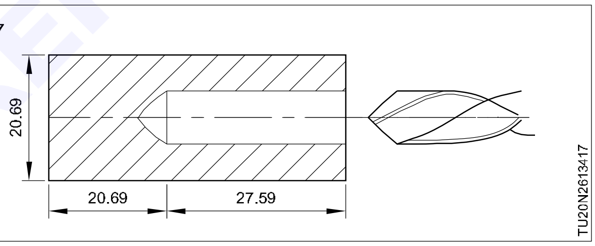
Operations

This cycle performs threading so that the length of the lead only between C and D is made as specified in the F code. In other sections, the tool moves in rapid traverse. The time constant for acceleration/deceleration after interpolation and FL feed rate for thread chamfering and the feed rate for retraction after chamfering are the same as for thread chamfering with G92 (canned cycle).

Face drilling (Fig 7)

Simply position the drill to a safe starting point and then call the drilling cycle. The drill then drills to each incremental peck depth and then retracts to clear the chips.

Fig 7



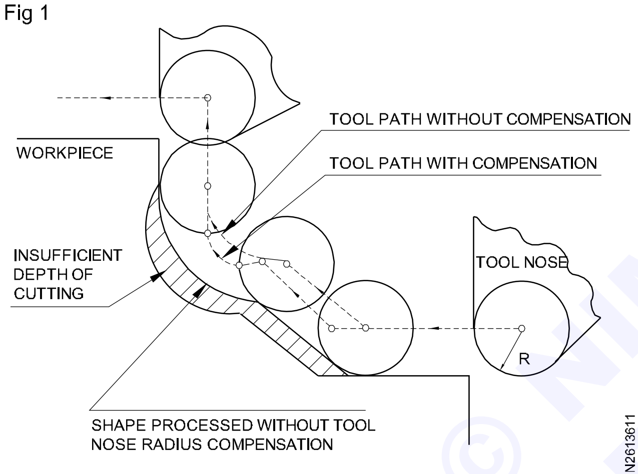
Tool nose radius compensation (TNRS)

Objectives : At the end of this lesson you shall be able to
- **state the purpose of the tool nose radius compensation**
- **list the imaginary code for nose radius for various operations**
- **describe the G codes for tool nose compensation**
- **explain the method to entering tool nose radius in geometry page.**

Overview of tool nose radius compensation (Fig 1)

It is difficult to produce the compensation necessary to form accurate parts when using only the tool offset function due to tool nose roundness in taper cutting or circular cutting. The tool nose radius compensation function compensates automatically for the above errors.

Fig 1

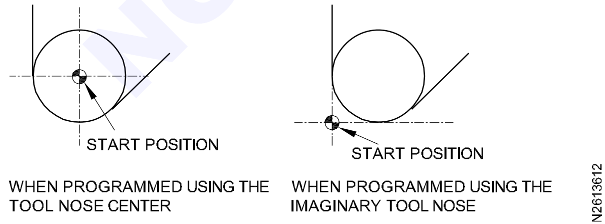


TOOL PATH OF TOOL NOSE RADIUS COMPENSATION

Imaginary Tool Nose (Fig 2)

The tool nose at position A in following figure does not actually exist. The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius centre to the start position than the imaginary tool nose (Note). Also when imaginary tool nose is used, the tool nose radius need not be considered in programming. The position relationship when the tool is set to the start position is shown in the following figure.

Fig 2

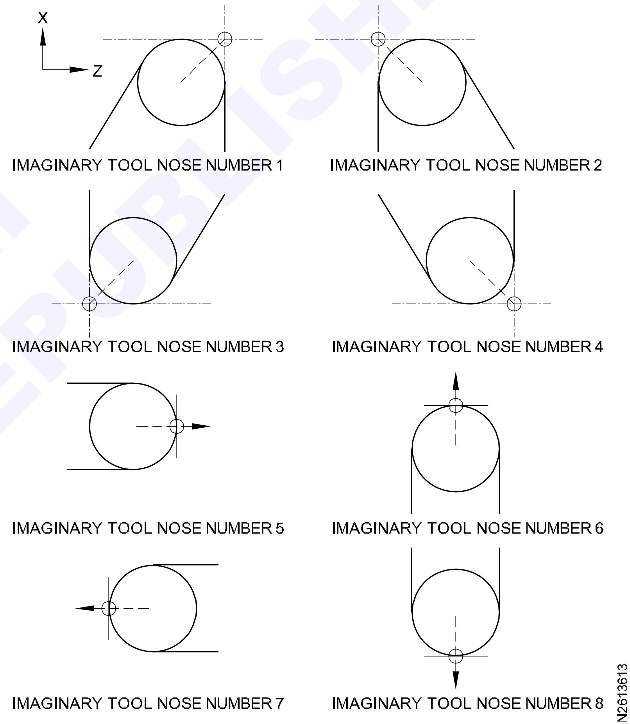


TOOL NOSE RADIUS CENTER AND IMAGINARY TOOL NOSE

Direction of Imaginary Tool Nose

The direction of the imaginary tool nose viewed from the tool nose centre is determined by the direction of the tool during cutting, so it must be set in advance as well as offset values. The direction of the imaginary tool nose can be selected from the eight specifications shown in the Fig 3 together with their corresponding codes. This Fig 3 illustrates the relation between the tool and the start position. The following apply when the tool geometry offset and tool wear offset option are selected.

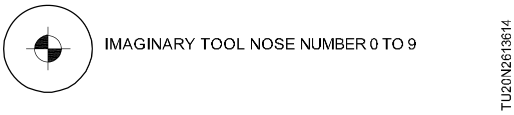
Fig 3



DIRECTION OF TOOL NOSE

Imaginary tool nose numbers 0 and 9 are used when the tool nose centre coincides with the start position. (Fig 4)

Fig 4



Offset Number and Offset Value are entered in tool geometry offset page (Fig 5&6).

Fig 5

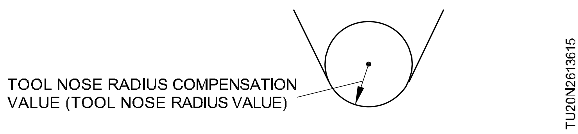
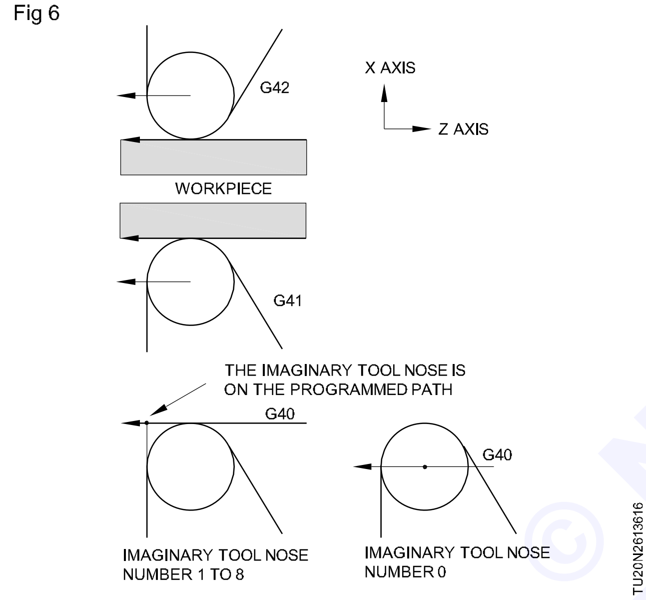


Table 1 - Tool geometry offset

Geometry offset number	OFGX (X-axis geometry offset	OFGZ (Z-axis geometry offset	OFGR (Tool nose radius geometry offset	OFT (Imaginary tool nose direction)	OFGY (Y-axis geometry offset amount)
G01	10.040	50.020	0	1	70.020
G02	20.060	30.030	0	2	90.030
G03	0	0	0.20	6	0
G04	:	:	:	:	:
G05	:	:	:	:	:
:	:	:	:	:	:

Fig 6



Position the cursor at appropriate place and input the values

The tool nose radius compensation value during execution is the sum of the geometry offset and the wear offset.

Work Position and Move Command

In tool nose radius compensation, the position of the work piece with respect to the tool must be specified (Fig 7& 8)

Fig 7

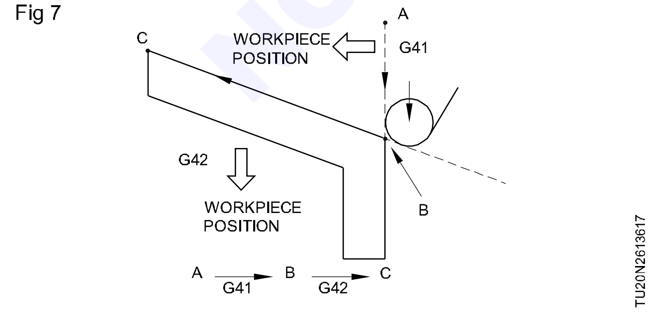
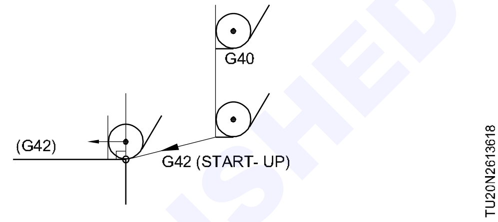


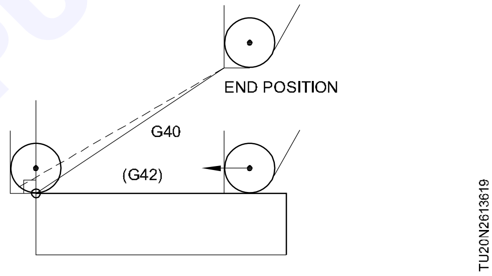
Fig 8



Tool movement when the work piece position changes

The work piece position against the tool changes at the corner of the programmed path as shown in Fig 9.

Fig 9



Although the work piece does not exist on the right side of the programmed path in the above case. the existence of the work piece is assumed in the movement from A to B. The work piece position must not be changed in the block next to the start-up block. In the above example, if the block specifying motion from A to B were the start-up block, the tool path would not be the same as the one shown.

Start-up

The block in which the mode changes to

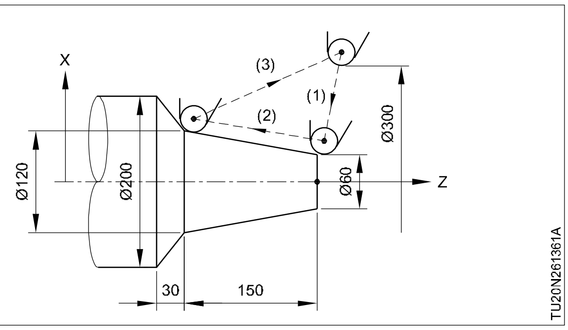
G41 or G42 from G40 is called the start-up block.

G40 _ ;

G41 _ ; (Start-up block)

Transient tool movements for offset are performed in the start-up block. In the block after the start-up block, the tool nose centre is positioned Vertically to the programmed path of that block at the start position (Fig 10)

Fig 10



Offset cancel

The block in which the mode changes to G40 from G41 or G42 is called the offset cancel block. G41 _ ; G40 _ ; (Offset cancel block) The tool nose centre moves to a position vertical to the programmed path in the block before the cancel block. The tool is positioned at the end position in the offset cancel block (G40)

Examples

(G40 mode)

1 G42 G00 X60.0 ;

2 G01 X120.0 W-150.0 F10 ;

3 G40 G00 X300.0 W150.0 I40.0 K-30.0 ;

Cutting tool material for CNC machining

Objectives : At the end of this lesson you shall be able to
- **list the types of carbide tipped tools**
- **explain the CDC and PVD**
- **brief the other cutting tool materials**
- **describe the various shapes of inserts**
- **explain the ISO system of designation of lath tool holders.**

Inserted carbide tooling is the preferred tooling for many CNC applications. For the full utilization of CNC machines it is essential to pay due attention to the selection and usage of tooling,

Carbide Tool

Carbide tools, especially indexable carbide tools, are the leading products of CNC machining tools. The varieties of integral and indexable carbide tools or inserts have been extended to various cutting tool fields, among which indexable carbide tools have been expanded from simple turning tools and face milling cutters to various precision, complex and forming tools.

Types of carbide tools:- they are

- CVD coated carbide tipped tools
- PVD coated carbide tipped tool
- Cemented carbide tipped tools

CVD coated tipped tools

CVD stands for Chemical Vapour Deposition. The CVD coating is generated by chemical reactions at temperatures of 700-1050°C.

CVD coatings have high wear resistance and excellent adhesion to cemented carbide.

The first CVD coated cemented carbide was the single layer titanium carbide coating (TiC).

Alumina coatings (Al2O3) and titanium nitride (TiN) coatings were introduced later.

More recently, the modern titanium carbonitride coatings (MT-Ti(C,N) or MT-TiCN, also called MT-CVD) were developed to improve grade properties through their ability to keep the cemented carbide interface intact.

Applications

CVD coated grades are the first choice in a wide range of applications where wear resistance is important. Such applications are found in general turning and boring of steel, with crater wear resistance offered by the thick CVD coatings; general turning of stainless steels and for milling grades in ISO P, ISO M, ISO K. For drilling, CVD grades are usually used in the peripheral insert.

PVD coated tipped tools

Physical Vapour Deposition (PVD) coatings are formed at relatively low temperatures (400-600°C). The process involves the evaporation of a metal which reacts with, for example, nitrogen to form a hard nitride coating on the cutting tool surface.

The main PVD-coating constituents are described below. Modern coatings are combinations of these constituents in sequenced layers and/or lamellar coatings. Lamellar coatings have numerous thin layers, in the nanometer range, which make the coating even harder.

PVD-TiN - Titanium nitride was the first PVD coating. It has all-round properties and a golden colour.

PVD-Ti(C,N) - Titanium carbonitride is harder than TiN and adds flank wear resistance.

PVD-(Ti,Al)N - Titanium aluminium nitride has high hardness in combination with oxidation resistance, which improves overall wear resistance.

PVD-oxide - Is used for its chemical inertness and enhanced crater wear resistance.

Applications

PVD coated grades are recommended for tough, yet sharp, cutting edges, as well as in smearing materials. Such applications are widespread and include all solid end mills and drills, and a majority of grades for grooving, threading and milling. PVD-coated grades are also extensively used for finishing applications.

Uncoated cemented carbide cutting tool material

Uncoated cemented carbide grades represent a very small proportion of the total cutting tool assortment. These grades are either straight WC/Co or have a high volume of cubic carbonitrides.

Applications

Typical applications of this cutting tool material are machining of HRSA (heat resistant super alloys) or titanium alloys and turning hardened materials at low speed.

The wear rate of uncoated cemented carbide grades is rapid yet controlled, with a self-sharpening action.

Cermet cutting tool material

Cermet is a cemented carbide with titanium-based hard particles. The name cermet combines the words ceramic and metal. Originally, cermet was a composite of TiC and nickel. Modern cermets are nickel-free and have a designed structure of titanium carbonitride Ti(C,N) core particles, a second hard phase of (Ti,Nb,W)(C,N) and a W-rich cobalt binder.

Ti(C,N) adds wear resistance to the grade, the second hard phase increases the plastic deformation resistance, and the amount of cobalt controls the toughness.

Applications

Cermet grades are used in smearing applications where built-up edge is a problem. Its self-sharpening wear pattern keeps cutting forces low even after long periods in cut. In finishing operations, this enables a long tool life and close tolerances, and results in shiny surfaces.

Typical applications are finishing in stainless steels, nodular cast irons, low carbon steels and ferritic steels.

Ceramic cutting tool material

All ceramic cutting tools have excellent wear resistance at high cutting speeds.

There are a range of ceramic grades available for a variety of applications.

Sialon (SiAlON) grades combine the strength of a self-reinforced silicon nitride network with enhanced chemical stability. Sialon grades are ideal for machining heat resistant super alloys (HRSA).

Applications

Ceramic grades can be applied in a broad range of applications and materials, most often in high speed turning operations but also in grooving and milling operations. The specific properties of each ceramic grade enable high productivity when applied correctly. Knowledge of when and how to use ceramic grades is important for success.

Polycrystalline cubic boron nitride cutting tool material

Polycrystalline cubic boron nitride, CBN, is a cutting tool material with excellent hot hardness that can be used at very high cutting speeds. It also exhibits good toughness and thermal shock resistance.

Modern CBN grades are ceramic composites with a CBN content of 40-65%. The ceramic binder adds wear resistance to the CBN, which is otherwise prone to chemical wear. Another group of grades are the high content CBN grades, with 85% to almost 100% CBN. These grades may have a metallic binder to improve their toughness.

Applications

CBN grades are largely used for finish turning of hardened steels, with a hardness over 45 HRc. Above 55 HRc, CBN is the only cutting tool which can replace traditionally used grinding methods. Softer steels, below 45 HRc, contain a higher amount of ferrite, which has a negative effect on the wear resistance of CBN.

CBN can also be used for high speed roughing of grey cast irons in both turning and milling operations.

Polycrystalline diamond cutting tool material

PCD is a composite of diamond particles sintered together with a metallic binder. Diamond is the hardest, and therefore the most abrasion resistant, of all materials. As a cutting tool material, it has good wear resistance but it lacks chemical stability at high temperatures and dissolves easily in iron.

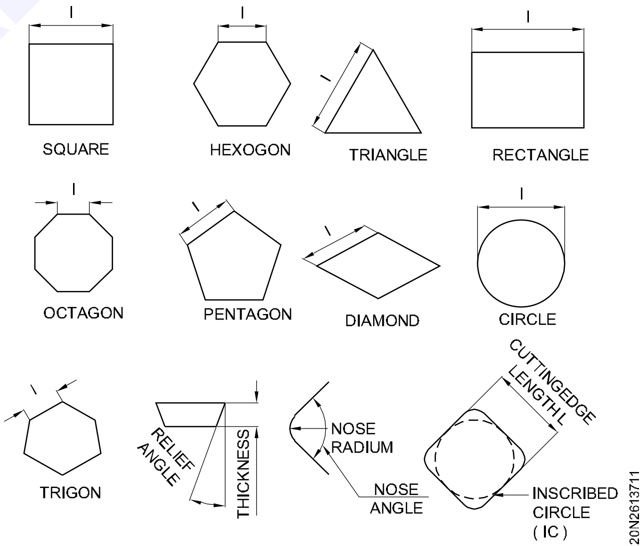
Applications

PCD tools are limited to non-ferrous materials, such as high-silicon aluminium, metal matrix composites (MMC) and carbon fibre reinforced plastics (CFRP). PCD with flood coolant can also be used in titanium super-finishing applications.

Tool inserts shapes

Inserts are available in various shapes such as triangle, square, rectangle, pentagon, hexagon, octagon, diamond shaped and circle. They cannot be resharpened, but they have a number of cutting edges. (Fig 1)

Fig 1



INDEXABLE TOOL INSERTS

Inserts are produced in various sizes and thicknesses. Smallest possible size is chosen to produce the desired depth of cut. Thickness of an insert affects its strength. Hence, for a large depth of cut and feed, a thicker insert is chosen.

ISO standard is commonly followed for specifying inserts. An example is CNMG120408. The first letter, C in this case, indicates the shape of the insert. The common types are:

Symbol	Shape
S	Square
T	triangular
H	hexagonal
O	octagonal
P	pentagonal
L	rectangular
R	round
A, B, K	parallelogram (nose angles 85°, 82° and 55° respectively)
C, D, E, F, M, V	Diamond shaped or rhombic (nose angles 80°, 55°, 75°, 50°, 86°, 35 ° respectively)
W	Trigon (nose angle 80°)

The second letter specifies the relief angles

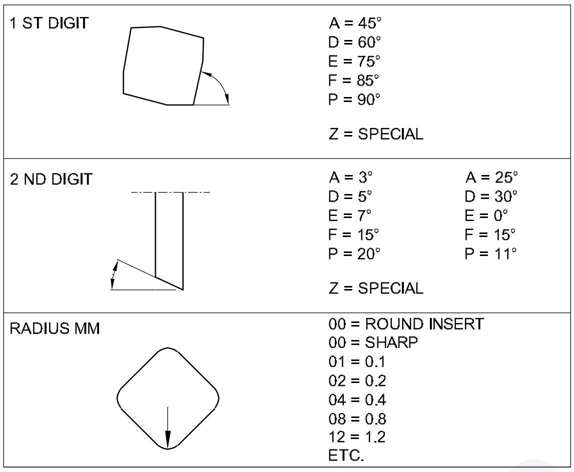
Symbol	Relief Angle
N	0°
A	3°
B	5°
C	7°
P	11°
D	15°
E	20°
F	25°
G	30°

The third letter specifies tolerances on various dimensions (Fig 2) (e.g., thickness) of the insert. The different tolerance classes are A, F, C, H, E, G (absolute values) and J, K, L, M, N, U (tolerance values depend on the diameter of the inscribed circle of the insert).

Significance of the fourth letter

The fourth letter describes the overall geometrical features of the insert (refer table). For example, an insert may or may not have a hole at the centre. The hole may be cylindrical or cylindrical with single or double countersink. The insert may or may not have a chip-breaker. The chip-breaker may be single-sided or double-sided.

Fig 2



INSERT WITH WIPER EDGE / RAADIUS

TU20N2613712

- **Cutting edge condition** (Fig 3)

  F for sharp,

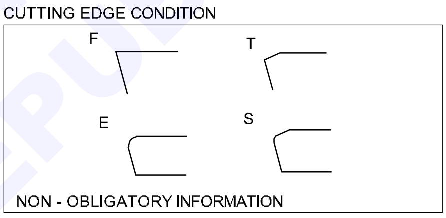
  T for chamfered,

  E for honed and

  S for chamfered and honed.

  This information , however, is non-obligatory.

Fig 3



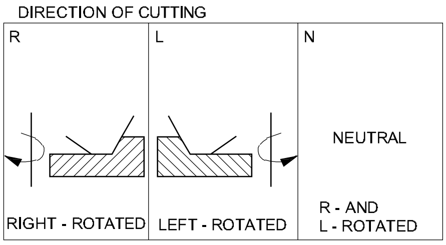
TU20N2613713

- **Cutting direction** (Fig 4)

  L for machining with left - rotated (CCW) spindle (M04),

  R for machining with right - rotated (CW) spindle (M03) and N for both left-and right - rotated.

Fig 4



CUTTING EDGE CONDITION AND DIRECTION OF ROTATION OF MILLING INSERTS

TU20N2613714

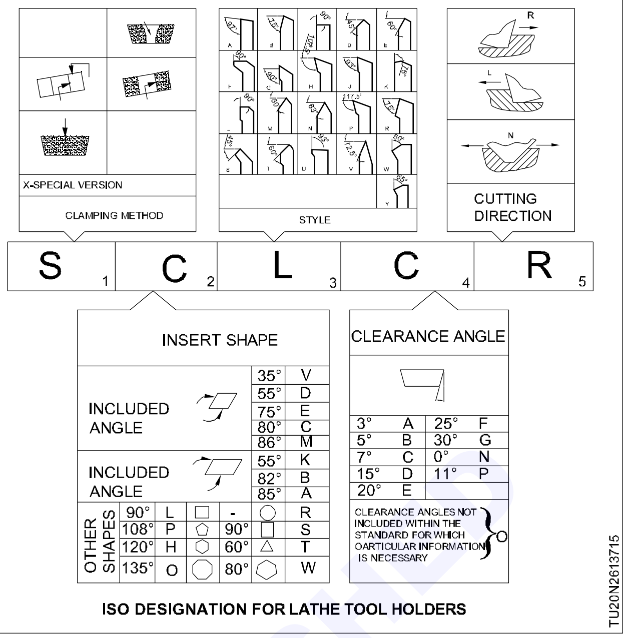
Symbol	Hole	Shape of Hole	Chip-Breaker
N	without	no hole	without
R			single-sided
F			double -sided
A	with	cylindrical	without
M			single-sided
G			double -sided
W		cylindrical with 40°-60° countersink	without
T			single sided
Q		cylindrical with 40°-60° double countersink	without
			double-sided
B		cylindrical with 70°-90° countersink	without
H			single-sided
C		cylindrical with 70°-90° double countersink	without
J			double-sided
X		special shape	

The appropriate character designations are appended to the right, after the radius specification.

Tool holders for lathe: There is an ISO designation system for tool holders also, to suit various types of inserts. The first five characters describe insert clamping method, compatible insert shape, insert holding style of the tool holder (side cutting edge angle/end cutting edge angle and straight shank/offset shank), clearance angle and cutting direction, respectively. Next four digits specify shank height and shank width in mm (two digits for each). Tool length is specified next, by a character code. The next and the last two digits specify cutting edge length in mm. Manufacturers, specific codes may be appended in the end. (Figs 5&6)

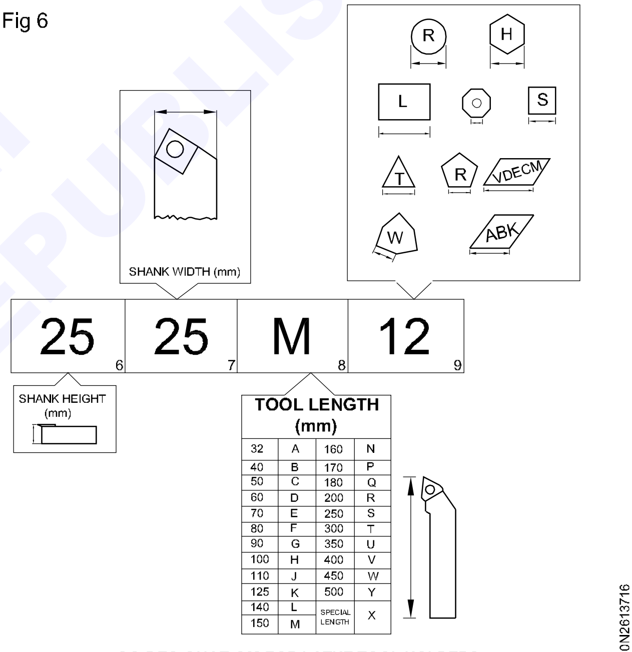
Insert clamping method (S)	Retained via central screw
Compatible insert shape (C)	80° diamond
Style of the tool holder body (L)	-5° side cutting edge angle, 5° end cutting edge angle and offset shank
Clearance angle (C)	7°
Cutting direction (R)	Right-hand
Shank height (25)	25 mm
Shank width (25)	25 mm
Tool length (M)	150 mm
Cutting edge length (12)	12 mm
Manufacture specific information	None

Fig 5



ISO DESIGNATION FOR LATHE TOOL HOLDERS

Fig 6



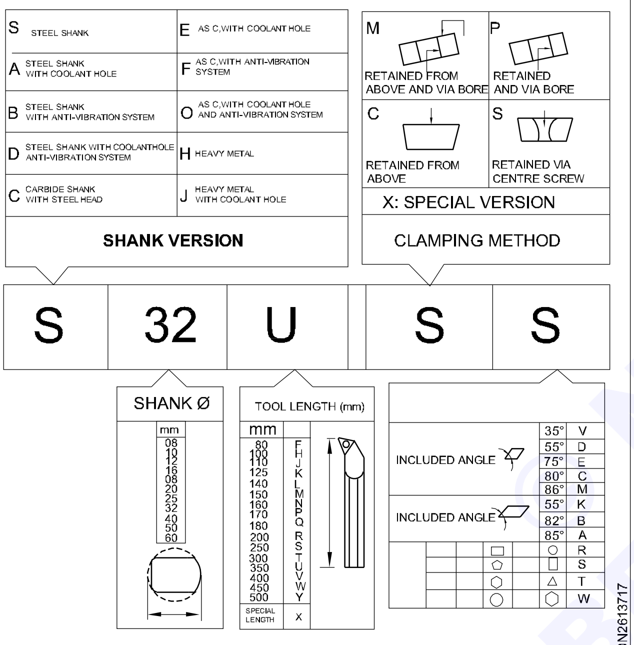
ISO DESIGNATIOM FOR LATHE TOOL HOLDERS

Boring bars for lathe (Figs 7&8)

There is a similar ISO designation system for the internal tool hodlders also (which are called boring bars). See Fig 9 and Fig 10 for details. Refer to Table 1 also for shapes not shown in these figures. In fact, tool length, clamping method, compatible insert shape, body style, clearance angle and cutting edge length have the **same representation** for tool holders as well as boring bars. For a boring bar, the cutting. If cutting is possible with clockwise rotation (MO3) of the spindle. The example, considered in the figure is S32U SSKCR12 type boring bar. Manufacturer-specific information may be appended in the end, after a gap. Separating dash (1) is also used in place of gaps. Referring to Table 1 and Figs 8&9, the description of S32U SKKCR12 is as follows:

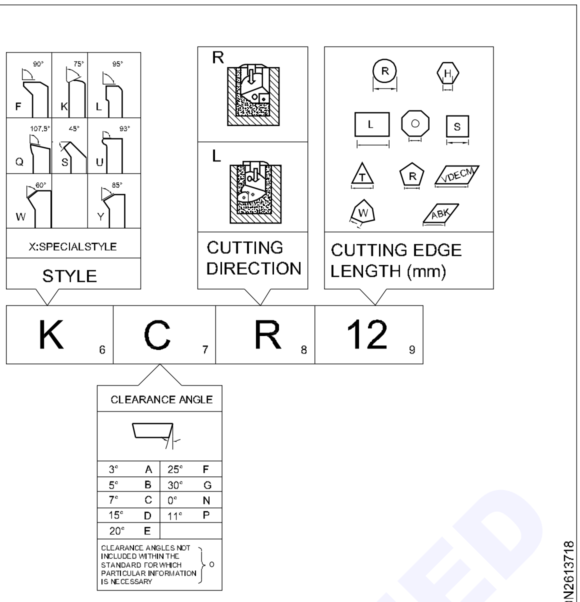
Shank type (S)	Steel shank
Shank diameter (32)	∅ 32 mm
Tool length (U)	350 mm
Insert clamping method	Retained via central screw
Compatible insert shape (S)	Square
Style of the boring bar body (R)	-15° end cutting edge angle. offset shank.
Clearance angle (C)	7°
Cutting direction (R)	Right-rotated (CW spindle)
Cutting edge length (12)	12 mm
Manufacturer information	- specific None

Fig 7



ISO DESIGNATION FOR LATHE BORING BARS

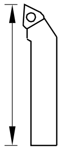
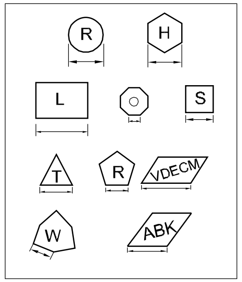
Fig 8



ISO DESIGNATION FOR LATHE BORING BAR

TABLE 1

ISO designation for lathe tool holders (Contd.)

1st char	2nd char	3rd char	4th char	5th char	6th char	7th char	8th char	9th char
Insert holding method	**Insert shape**	**Tool holder style**	**Insert relief (clearance)**	**Hand**				
M=top clamp and lock pin via the bore	A-85° parallelogram	A = 0° side cutting straight shank	N = 0°	R=right - hand	Shank height	Shang width	Tool holder length 	Insert cutting edge length 
P = lock pin via the bore only	B = 82° parallelogram	B = 15° side cutting cutting straight shank	A = 3°	L = left- hand				
C=top clamp only	C=80° diamond	C = 0° end cutting shank	B = 5° straight	N - neutral				
S=centre screw lock only	D=55° diamond	D = 45° side cutting, 45° end cutting, straight shank	C = 7°					
X = other methods	E = 75° diamond	E = 30° side cutting, straight shank	P = 11°					
	H = hexagon	F = 0° end cutting, offset shank	D = 15°					
	K = 55° parallelogram	G = 0° side cutting, offset shank	E = 20°					
	L = rectangle	H = -17.5° side cutting, offset shank	F = 25°					
	M=86° diamond	J = -3° side cutting,offset shank	G = 30°					
	O = octagon	K = 15° end cutting, offset shank						
	P = pentagon	L = 5° side cutting, 5°end cutting, offset shank						
	R = round	M = 40° side cutting, 50° end cutting, straight shank						
	S = square	N = 27° side cutting, straight shank						

TOOL LENGTH (mm)			
32	A	160	N
40	B	170	P
50	C	180	Q
60	D	200	R
70	E	250	S
80	F	300	T
90	G	350	U
100	H	400	V
110	J	450	W
125	K	500	Y
140	L	SPECIAL LENGTH	X
150	M		

TABLE 1

ISO designation for lathe tool holders (Contd.)

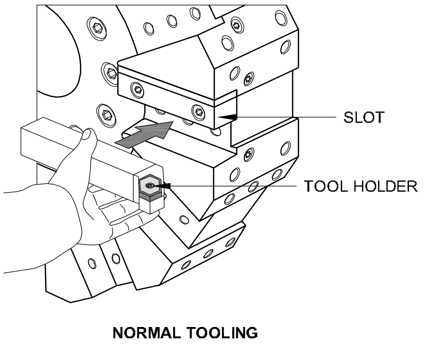
1st char	2nd char	3rd char	4th char	5th char	6th char	7th char	8th char	9th char
Insert holding method	**Insert shape**	**Tool holder style**	**Insert relief (clearance)**	**Hand**				
	T = triangle	P = -27.5° side cutting, offset shank						
	V = 35°	R = 15° side diamond cutting, offset shank						
	W = 80° trigon	S = 45° side cutting, offset shank						
		T = 30° side cutting, offset shank						
		U = 3° end cutting, offset shank						
		V = 17.5° side cutting, straight shank						
		W = 30° end cutting, offset, shank						
		Y = -5° end cutting, offset shank						

Tooling systems for CNC turning centres

Objective : At the end of this lesson you shall be able to
- **state the tool setting in CNC turning centres.**

A normal tool is lamped by inserting it in a slot in the turret and tightening screws to hold it firmly (Fig 1). While doing this, care must be taken to seat the holder properly against available butting surfaces. Finally, the tool offsets must be taken by touching the part or taking skin cuts.

Fig 1



Rectagular or square type shank tools (Fig 2) are fitted in the tool holders (Fig 3) and the tool holders are fitted in turret.

Fig 2

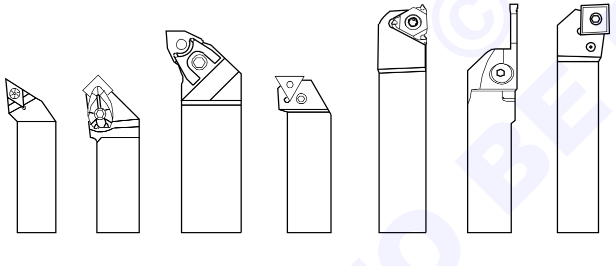
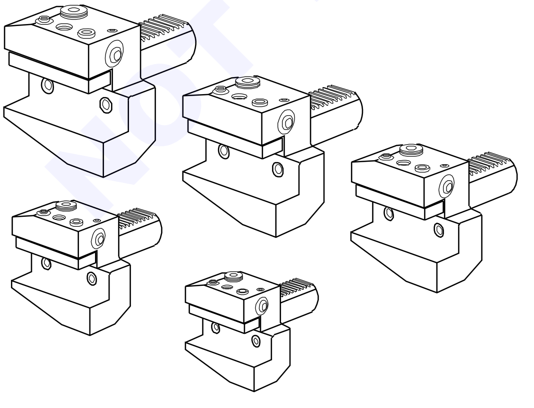


Fig 3



Whereas the round shank tools are fitted in round shank tool holders as shown in Fig 4 and the tool holders along with the tools are fitted in turret as shown in Fig 5.

Fig 4

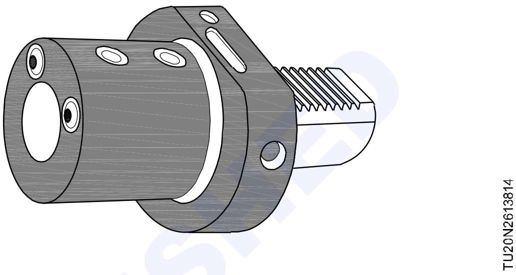
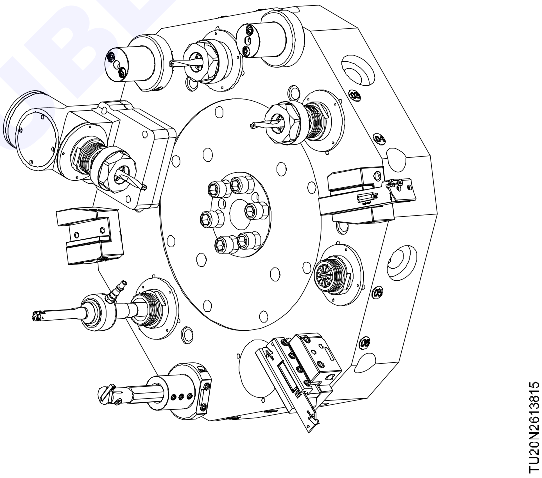


Fig 5



Quick change tool holder systems on CNC lathes (Fig 6&7)

Fig 6

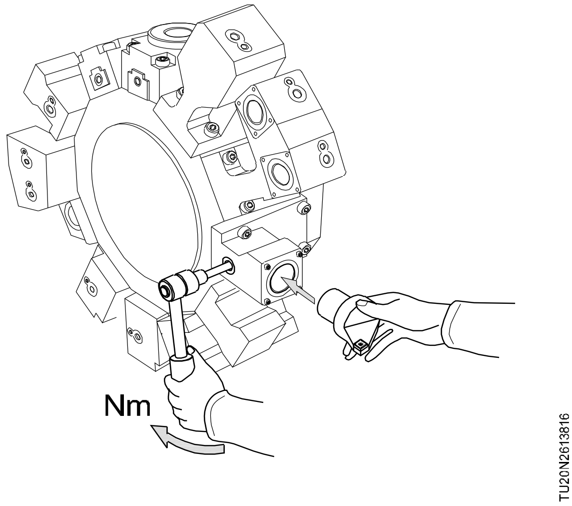
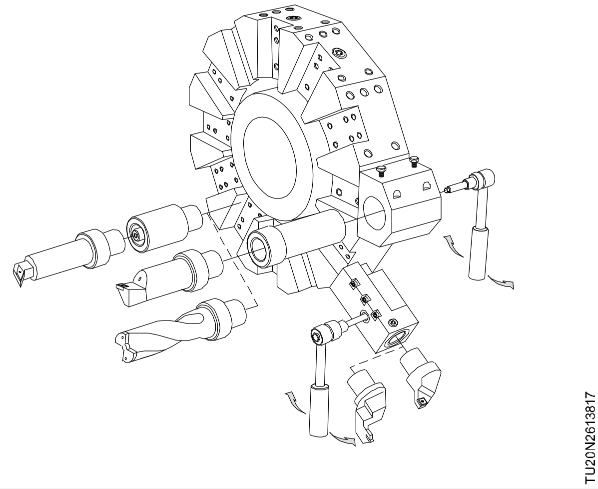


Fig 7



A quick change tool holder, or quick change tooling, is a modular system that enables tools to be changed very fast on the machine during.

The system reduces machine downtime for tool change during setup changes, and for changing worn out inserts during machining.

With a quick change tool holder, you do not clamp the tool directly in the turret. The tool is replaced by an interchangeable cutting unit. You insert the cutting unit into a standardized locking unit. There are different locking units for internal and external tools, and the locking units remain on the turret forever - there is no need to keep removing them for every part. The interface between the units is such that the cutting unit will only sit in a particular position in the locking unit, and will sit accurately every time.

When doing the setup for a new part, you just quickly insert and clamp a new set of cutting units into the appropriate locking units. The tool offsets are predetermined for each cutting unit, and remain the same every time that you put in the unit.

Writing part program as per drawing and verifying programming using simulator

Objectives : At the end of this lesson you shall be able to
- **list the steps in planning and part programming for cnc turning**
- **write the manual cnc part program as per drawing dimension**
- **verify the program in cnc simulator.**

Manual CNC programming is a traditional and the most tedious approach which requires the programmer to be aware of the machine's responses by anticipating the program's outcome.

However, if programmer follow the steps given below it will be easy to write the cnc part program

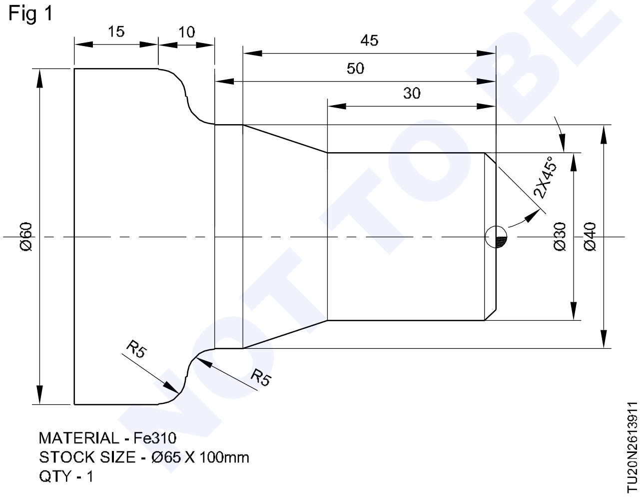
Steps in planning and programming

- Read the drawing and determining the machining process and its control
- List out the operations required
- Selecting the appropriate tool for machining
- Determining the cutting parameter
- Deciding the work holding method
- Fixing the programming zero on the part
- Writing the part program using G codes and M codes
- Verify the program in a cnc simulator

Example

Part drawing (Fig 1)

Fig 1



Material:- Fe310 stock size:- Ø 65 X 100 mm

Planning

Select the turning centre with Fanuc control

Operation required rough profile turning with canned cycle

Work holding method: - 3 jaw self centring chuck

Tool required external rough turning carbide tool turning

Cutting parameter: -

Programming zero fixed at the left side face centre

Write the part program using G71 outer diameter roughing cycle

Verify the program in cnc simulator

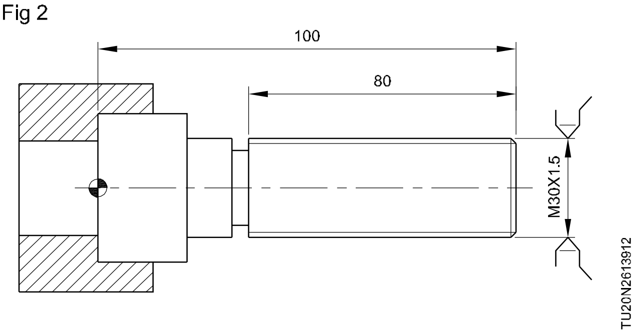
Programming

```
O1234; (PROFILE TURNING  PROGRAMMING)
N1 G90 G21 G80 G40;
N2 T0303;
N3 S750 M04;
N4 G00 X100.0 Z20.0;
N5 G71 P6 Q14 U4.0 W2.0 D4.0 F0.3 S550;
N6 G01 X26.0 Z0.0 F0.15;
N7 G01 X30.0 Z-2.0;
N8 G01 X30.0 Z30.0;
N9 G01 X40.0 Z-45.0;
N10 G01 X40.0 Z-50.0;
N11G02 X50 Z-55.0 I5.0 K0.0;
N12 G03 X60.0 Z-60.0 I0.0 K-5.0;
N13 G01 X60.0 Z-75.0;
N14 G01 65.0 Z-75.0;
N15 G00 X100.0 Z100.0 M05;
N16 G28 U0.0 W0.0 T0300;
N17 T0606 M04 S800;
N18 G00 X100.0 Z5.0;
N19 G70 P13 Q14 F0.1;
N20 G00 X100.0 Z100.0 M05;
N21 G28 U0.0 W0.0 T0600;
N22 M30;
```

Theading program (Fig 2)

Example given that F = 1.5mm, =1.5mm, =1mm, cutting for four times and each cutting depth is separetely 0.8mm, 0.6mm, 0.4mm, 0.16mm. it is diameter programming.

Fig 2



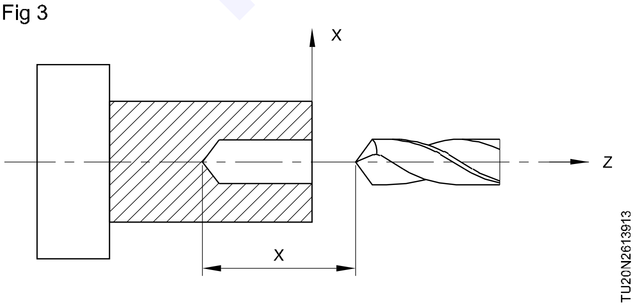
```
%3316
N1 T0101
N2      G00     X50     Z120
N3      M03     S300
N4      G00     X29.2   Z101.5
N5      G32     Z19     F1.5
N6      G00     X40
N7      Z101.5
N8      X28.6
N9      G32     Z19     F1.5
N10     G00     X40
N11     Z101.5
N12     X28.2
N13     G32     Z19     F1.5
N14     G00     X40
N15     Z101.5
N16     U-11.96
N17     G32     W-82.5 F1.5
N18     G00     X40
N19     X50     Z120
N20     M05
N21     M30
```

Tapping (G34) programming (Fig 3)

G34 K_F_P_Explanation of the parameters

K The distance from the starting point to the bottom of the hole

Fig 3

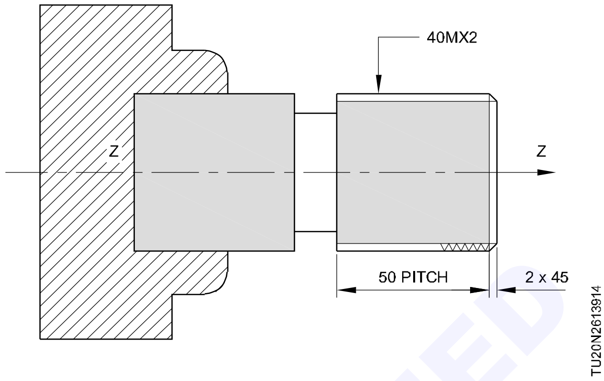


F Thread lead

P Dwell time at the bottom of a hole

G76 fanuc threading cycle CNC program with description (Metric thread) (Fig 4)

Fig 4



```
N10     M06     T0101;
N20     M04     G97     S1000;
N30     G00     X45     Z5;
N40     G76     P020060         Q100    R50
N50     G76     X38.7   Z-50    P1227   Q100    F2;
N60     G00     X45     Z5
N70     M05     M09     M30
```

Description of main program

N10 - Tool change command, select tool no 1

N20 - Spindle ON anti clockwise, constant spindle speed command, speed is 1000 rpm.

N30 - Rapid action command where X45 and Z5

(P02 =No. of finished path

00 = Chamfer amount at end

60 = Angle of tool tip)

Q100 = Each cut is 0.1 mm

R20 = Finishing allowance 0.02 mm

N50 - Threading cycle command, minor dia x axis, threading along Z-axis up to -50 threading depth, depth of finish cut 0.1 mm, pitch is 2.

M40X2

Major diameter is 40

Pitch is 2

Thread depth calculation = Pitch x 0.61363

= 1.227 mm in the micron is 1227

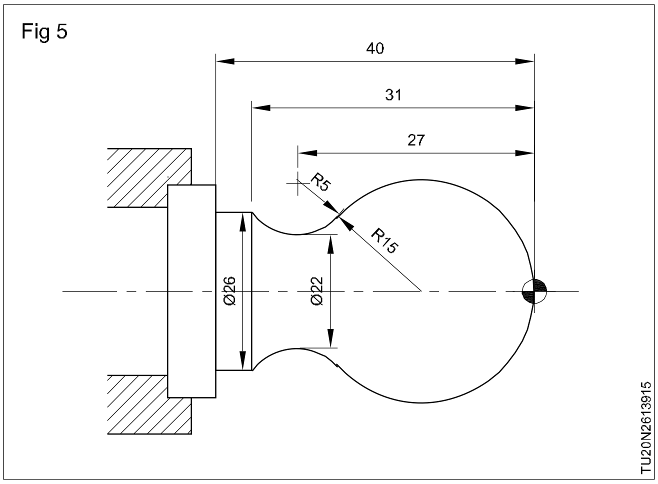
Minor diameter = 40-1.23 = 38.7 mm

N60 - Rapid action command where X45 and Z5

N70 - Spindle off, coolant off, main program end.

Programming with constant surface control (Fig 5)

Fig 5

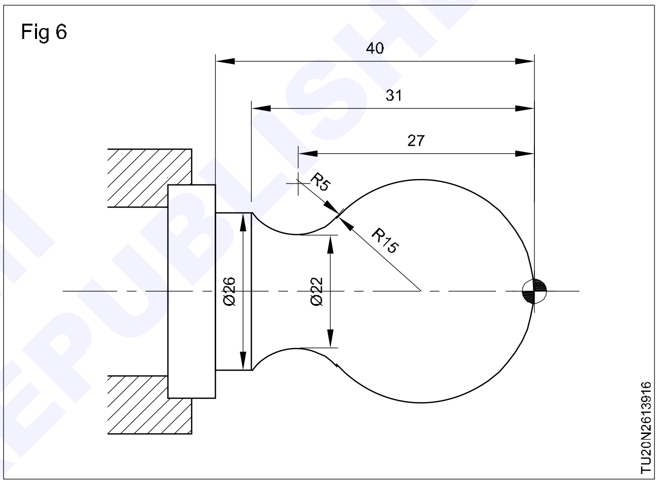


```
%3318
N1    T0101
N2    G00   X40    Z5
N3    M03   S460
N4    G96   S80
N5    G46   X400   P900
N6    G00   X0
N7    G01   Z0     F60
N8    G03   U24    W-24   R15
N9    G02   X26    Z-31   R5
N10   G01   Z-40
N11   X40   Z5
N12   G97   S300
N13   M30
```

Program with tool nose radius compensation (Fig 6)

```
%3323
N1    T0101
N2    M03   S400
N3    G00   X40    Z5
N4    G00   X0
N5    G01   G42    Z0     F60
N6    G03   U24    W-24   R15
N7    G02   X26    Z-31   R5
N8    G01   Z-40
N9    G00   X30
N10   G40   X40    Z5
N11   M30
```

Fig 6



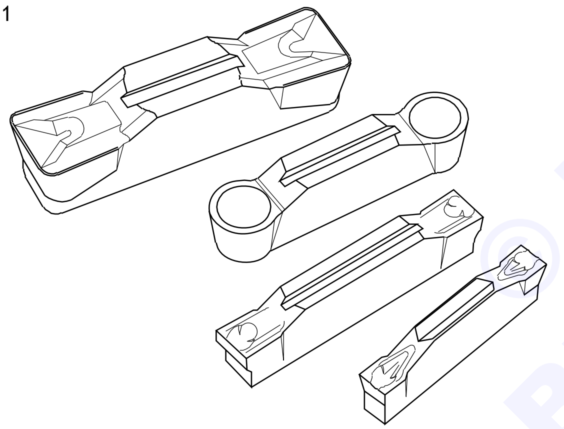
Tool selection related to grooving, drilling, boring and threading operations

Objectives : At the end of this lesson you shall be able to
- **list points to considered for the selection of grooving tools**
- **explain the what is u drill and grade of inserts used**
- **brief the index able boring tool selection**
- **state the index able threading insert and its grades.**

Grooving Inserts

Grooving inserts have either a square or round shape on the end. The shape typically corresponds to the shape of the groove being cut, although round inserts are the strongest inserts available and are preferred for roughing in demanding situations. When the depth of the groove is greater than the width, a multiple grooving operation is preferred. Multiple grooving consists of multiple plunge moves into the groove. The width of the tool is smaller than the width of the groove to allow for multiple side-by-side cutting passes. Figure 1 shows the different shapes of grooving inserts. (Fig 1)

Fig 1



GROOVING INSERTS

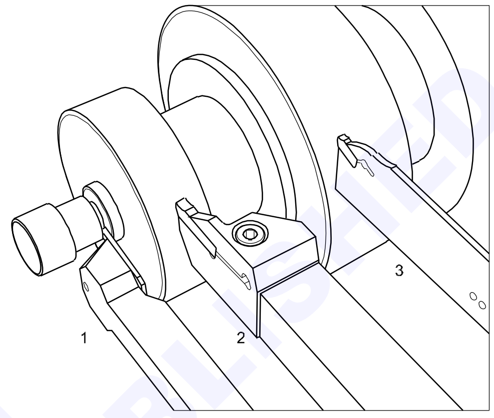
When the width of the groove is greater than the depth, a plunge turning operation is a better choice. Plunge turning combines plunging and turning motion, first plunging the tool to a shallow depth of cut, then moving laterally to the opposite end of the groove before plunging again to the next depth. For the tool holder, the tool overhang should be minimized to improve the stability of the tool during plunge moves. The width and depth of the groove on this part are about the same size, so the grooving operation can consist of multiple grooving. The width of the insert should be between 3 mm and 5mm to allow for multiple plunges across the 8mm wide groove.

External grooving tools shown in Fig 2

Internal grooving tool with tool holders shown in Fig 3

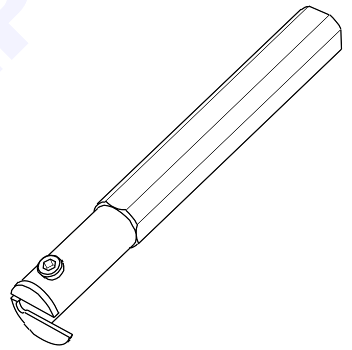
Drilling Tools: Index able drills/Hole Making Tools Also Known as U drills. U drills are drilling tool with carbide inserts which can change easily and cost economical a normal u drill look like this. (Fig 4)

Fig 2



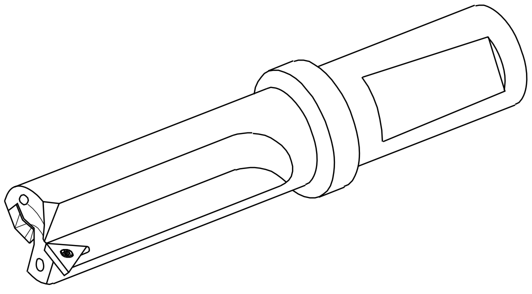
EXTERNAL GROOVING

Fig 3



INTERNAL TOOL WITH TOOL HOLDER

Fig 4



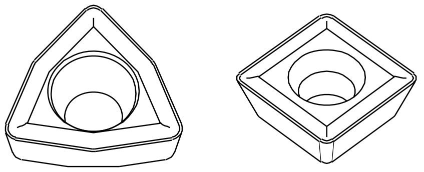
DRILLING TOOLS

U drill are those cutting tools which are used to make drill in any work piece weather its wood, aluminium, brass, steel, cast iron, titanium etc.

U drills are most commonly used because of economic cost and easy usage and can be used on both turning and milling machine.

They are SPMG and WCMX type U drills comes in Square and Trigon Shape inserts from Drilling range LXD 2 to LXD5 , 12 mm to 60 mm now 8 mm to 12 mm small drills are also becoming common if they are successful they can be very cost effective for these diameters drills. (Fig 5)

Fig 5



SQUARE AND TRIGON SHAPE INSERTS

No need of Centre Mark (Centre Hole)

As u drills can plunge directly into work piece so you don't need centre drill mark it saves a turret space in lathe and extra time in both the machines.

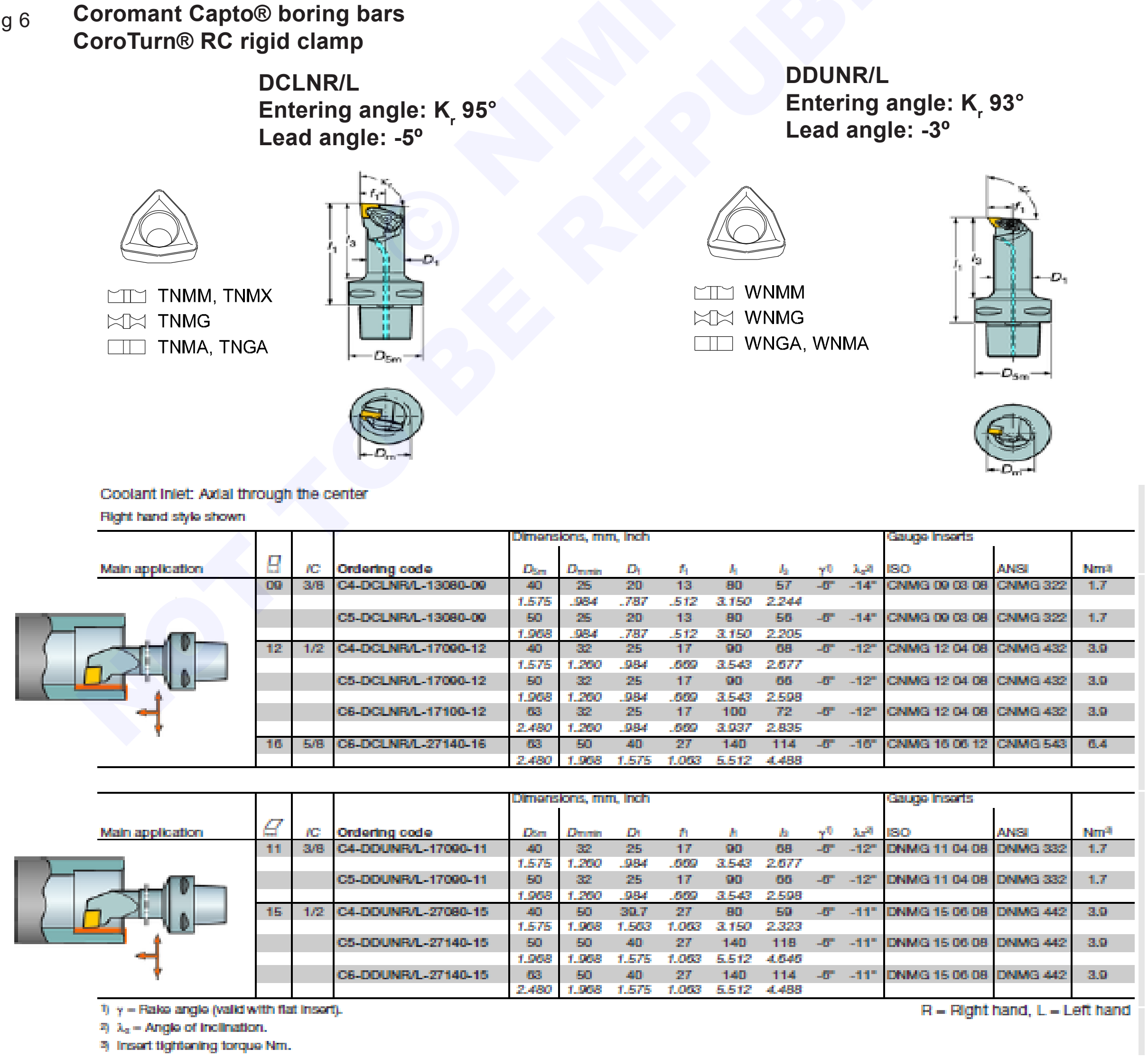
Tolerance: Drill has +- tolerance the more LXD has more the tolerance if you choose a LXD5 U drill there are chances of .2 size of drill size the smaller the diameter the less is tolerance so its not a good choice to take a LXD5 drill when you require LXD3.

Through Coolant: U drills are through coolant as external coolant cannot reach in depth so it's always better to choose drill with coolant holes only as coolant can reach inside and save it from burning and help better chip evacuation

Boring Tools: Index able boring bars are single-point cutting tools that use replaceable inserts to size, straighten, and finish the inside of drilled or cast holes. They mount to a lathe or turning machine and remain stationary while the machine rotates the work piece against the boring bar's cutting tip. The inserts are seated at the cutting tip and can be swapped out for new ones of the same style or a different style compatible with the boring bar without removing the boring bar from the machine. Each insert will have multiple cutting edges, so it can be rotated (indexed) to expose a fresh cutting edge when the old one dulls. Index able boring bars require fewer tool changes than solid tools in high-volume metalworking and fabrication applications with high speeds, high feeds, and difficult-to-machine materials.

Fig 6 show the details of the boring tool

Fig 6



Coromant Capto® boring bars
CoroTurn® RC rigid clamp

DCLNR/L
Entering angle: K_r 95°
Lead angle: -5º

DDUNR/L
Entering angle: K_r 93°
Lead angle: -3º

Coolant inlet: Axial through the center
Right hand style shown

Main application		IC	Ordering code	Dimensions, mm, inch D_{5m}	$D_{m\,min}$	D_1	f_1	l_1	l_3	γ[1]	λ_s[2]	Gauge inserts ISO	ANSI	Nm[3]
	09	3/8	C4-DCLNR/L-13080-09	40	25	20	13	80	57	-6°	-14°	CNMG 09 03 08	CNMG 322	1.7
				1.575	.984	.787	.512	3.150	2.244					
			C5-DCLNR/L-13080-09	50	25	20	13	80	56	-6°	-14°	CNMG 09 03 08	CNMG 322	1.7
				1.968	.984	.787	.512	3.150	2.205					
	12	1/2	C4-DCLNR/L-17090-12	40	32	25	17	90	68	-6°	-12°	CNMG 12 04 08	CNMG 432	3.9
				1.575	1.260	.984	.669	3.543	2.677					
			C5-DCLNR/L-17090-12	50	32	25	17	90	66	-6°	-12°	CNMG 12 04 08	CNMG 432	3.9
				1.968	1.260	.984	.669	3.543	2.598					
			C6-DCLNR/L-17100-12	63	32	25	17	100	72	-6°	-12°	CNMG 12 04 08	CNMG 432	3.9
				2.480	1.260	.984	.669	3.937	2.835					
	16	5/8	C6-DCLNR/L-27140-16	63	50	40	27	140	114	-6°	-16°	CNMG 16 06 12	CNMG 543	6.4
				2.480	1.968	1.575	1.063	5.512	4.488					

Main application		IC	Ordering code	Dimensions, mm, inch D_{5m}	$D_{m\,min}$	D_1	f_1	l_1	l_3	γ[1]	λ_s[2]	Gauge inserts ISO	ANSI	Nm[3]
	11	3/8	C4-DDUNR/L-17090-11	40	32	25	17	90	68	-6°	-12°	DNMG 11 04 08	DNMG 332	1.7
				1.575	1.260	.984	.669	3.543	2.677					
			C5-DDUNR/L-17090-11	50	32	25	17	90	66	-6°	-12°	DNMG 11 04 08	DNMG 332	1.7
				1.968	1.260	.984	.669	3.543	2.598					
	15	1/2	C4-DDUNR/L-27080-15	40	50	39.7	27	80	59	-6°	-11°	DNMG 15 06 08	DNMG 442	3.9
				1.575	1.968	1.563	1.063	3.150	2.323					
			C5-DDUNR/L-27140-15	50	50	40	27	140	118	-6°	-11°	DNMG 15 06 08	DNMG 442	3.9
				1.968	1.968	1.575	1.063	5.512	4.646					
			C6-DDUNR/L-27140-15	63	50	40	27	140	114	-6°	-11°	DNMG 15 06 08	DNMG 442	3.9
				2.480	1.968	1.575	1.063	5.512	4.488					

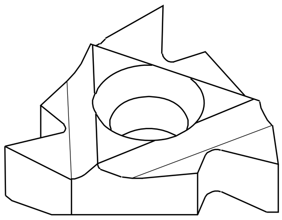
[1] γ = Rake angle (valid with flat insert).
[2] λ_s = Angle of inclination.
[3] Insert tightening torque Nm.

R = Right hand, L = Left hand

Threading Inserts (Fig 7)

There are three types of threading inserts: Full Profile, Partial Profile, and Multi-Tooth.

Fig 7



THREADING INSERT

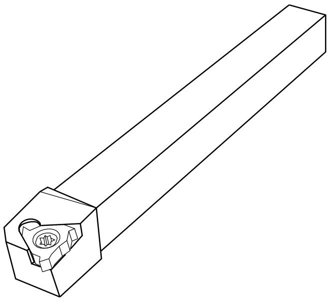
- Full profile inserts cut the full shape of the thread groove, from bottom to top. These are useful for high productivity in threading.
- Partial profile inserts cut the bottom of the thread groove but leave clearance at the top. These are useful for cutting a range of thread sizes with a small inventory of inserts.
- Multi-Tooth inserts feature multiple teeth in a series to reduce the number of passes required to cut a thread. These are useful for mass production in threading.

GC1125 is the first choice grade for ISO P, M, K and N materials in Coro Thread 266. This PVD grade combines the superior wear resistance of a coated grade with the edge sharpness and toughness of an uncoated grade. Optimized for steel threading and for medium to high speeds.

Indexable threading insert with holder shown in Fig 8

Searching for tools in an online database

Fig 8



INDEX ABLE THREADING WITH HOLDER

Machining Cloud is an independent provider of CNC cutting tool product data. A single source of access to the most current product data from a variety of suppliers, in digital format, available from your desktop. Machining Cloud provides the most up-to-date information, directly obtained from the manufacturers. The data is formatted very closely to each manufacturer's catalogue system so it is familiar to users.

Download the Machining Cloud App

Install the Machining Cloud App

Search for cutting tools in the Machining Cloud online databases

The Machining Cloud has partnered with several cutting tool manufacturers. This lesson will show you how to select a tool manufacturer and search for tools in their online database. You are not restricted to one manufacturer in the Machining Cloud. You will see how to select cutting tools from different tool manufacturers and add them to your personal tool list.

Search for an O.D. roughing tool

Select the holder with the 1" x 1" x 6" long shank and click Add to Tool Assembly.

Programming on CNC Tapping

Objectives : At the end of this lesson you shall be able to
- **state the G84 code in CNC**
- **brief tapping cycle**
- **list the differefce between rigid tapping & long form tapping.**

G84 code in CNC progrmming refers to tapping cycle in CNC. This is carried out with tapping heads, and Tension/ Compression tap holders.

G84 code is commonly used program for tapping, used on threaded holes. There are two ways of tapping program.

- One using tapping cycle with rigid tapping capabilities.
- Long Form (no canned cycle needed) programming when a tapping head is used that do not have rigid tapping.

To use rigid tapping, the CNC machine should support the synchronnisation of feed motion with the spindle speed.

G84 code is tapping RH threads with M3 spindle rotation.

G74 code is tapping LH threads with M4

Rigid motion mode is used in Fanuc control using M29 code.

Advantages

- A tapping head may have a gear ratio that allows it to retract faster. You will want to change feed rate when retracting.
- A Dwell at the bottom of the hole may be helpful as the spindle reverse to even out the amount of spring adjustment being used.
- It is possible to drag the feed rate a bit.

Tapping comes under canned cycle with intermittent feed, with Dwell spinlde CW with feed retraction in Z direction as shown in table below.

Example: Tap 1/4-20 thread 0.500” deep at 0,

Here is the ‘G’ code

```
M03;
M8 (Speed & feed rate)
S400    F20 (Tapping)
Z1.0;
G00     X0.0    Y0.0;
G01     M29;
G84     Z-0.5   R02;
```

M03 spindle in right direction

M8 collant ON

S400 Spindle speed 400rpm & feed rate 20

Next we move to save Z&XY.

Switch G01 & M29 to turn rigid tapping lastly G84 is run with Z indicating coordinate & R retracting coordinate.

If we had more holes to be tapped we can list their XY coordinate, viz

```
G84     Z-0.5   R0.2;
X0.0    Y1.0 etc  etc;
```

CNC programme for grooving, threading external and internal

Objectives : At the end of this lesson you shall be able to
- **brief the grooving program G75**
- **explain the external threading program G76**
- **enumerate the internal program G7.**

Grooving is a turning operation. In CNC G75 grooving cycle is used, the format of fanuc G74 is shown below

N10 G75 R

N20 G75 X Z P Q R

G75 First CNC programming block

R - Return amount

G75 Second CNC programming block

X - Groove depth (Groove end position in x-axis)

Z - Last groove position in Z-axis (Eng position in z-axis)

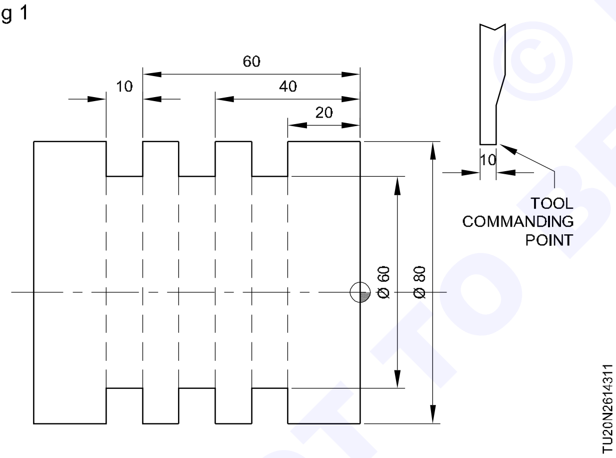
P - Peck increment in x-axis (depth of each cut in x-axis)

Q - Stepping in z-axis

R = Relief amount at end of the cut

G75 Canned cycle grooving CNC programming example (Fig 1)

Fig 1



N20 G97 S400 M03 T0101

N30 G00 X90.0 Z1.0

N40 X82.0 Z-60.0

N50 G75 R1.0

N60 G75 X60.0 Z-60.0 P3000 Q20000 F0.1

N70 G00 X90.0

N80 X200.0 Z200.0 T0100

N90 M30

External threading

The important thing in threading operation is to fix up the thread start and thread end positions. The end position is easier in Z axis, as we know the length of threaded component. For thread cutting in CNC lathe.

The start position is interesting as we start threading somewhere from outside the threads. You need to leave some allowance in Z to give CNC machine to synchronize the feed rate with spindle rotational position, cutting thread puts more stress on tool. Thread height, taper amount, thread pitch or lead, thread infeed angle (TN angle) are important in making the programme.

Format of G76 is threading cycle is shown below

Fanuc double line G76 threading cycle

G76 P(m) (r) (a) (d min) R(d)

G76 X (U) Z(W) R(i) P(K) Q(d) F(L)

P word: The P-word has 6 digits consisting of three 2-digit clusters for m, r, and a.

M = Repetitive finishing count (1 to 99) - spring passes.

R: chamfering amount (1 to 99)

A: Angle of Tool nose, select 80,60,55,30,29 or 0 degrees.

Q Word: d min is the minimum cutting depth. If the depth of either a roughing or finish pass is less than this, it is clamped to be at least this much.

R Word: d is the finish allowance

X/Z/U/W words (2nd line): Specify the coordinates of the end point. X, Z use the current mode (absolute or relative) whle U, W can be used to specify a relative position.

R word (2nd line): i is the taper amount when cutting tapered threads.

P word (2nd line): k is the thread height expressed as a radius (not diameter) value.

Q word (2nd line): d is the depth of the first cut

F Word (2nd line): L is the lead of the thread. Example: Fanuc 2 line

G76 cutting a tapered pipe thread: Fanuc single line G76 threading cycle

Fanuc single line G76 threading cycle

G76 X.. Z..I..K..D..F..A..P..

X = Diameter of last thread end

Z = Position of the thread end

I = Taper over total lengthK = Single depth of the thread - positive

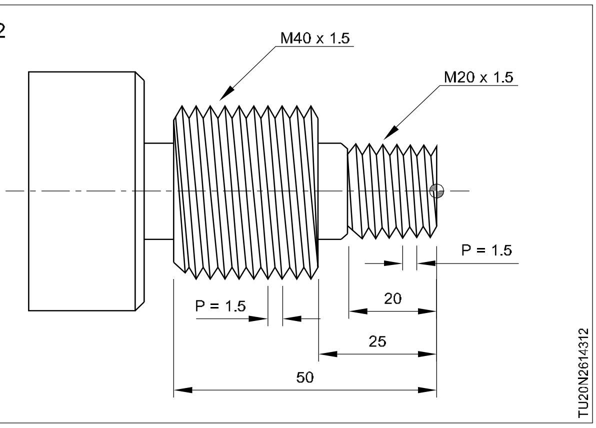
D = Depth of first threading pass - positive

A = Included angle of the insert - positive

P = Infeed method (one of 4)

CNC programming with G76 threading cycle (Fig 2)

Fig 2



N10 T0103

N20 G97 S800 M03

N30 G00 X30 Z5 T0303

N40 G76 P021060 Q100 R100

N50 G76 X18.2 Z-20 P900 Q200 FI.5

N60 G00 X50 Z-20

N70 G76 P021060 Q100 R100

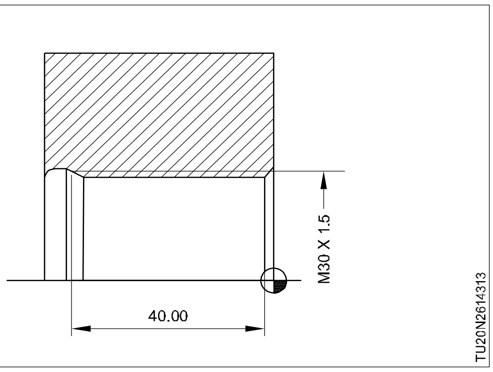
N80 G76 X38.2 Z-52 P900 Q200 FI.5

N90 G00 X200 Z200

N100 M30

CNC program of internal threading with G76 threading cycle (Fig 3)

Fig 3



N17 T0101

N18 G54

N19 G97 S800 M3

N20 G0 X25 Z6 M8

N21 G76 P0100060 Q100 R0.02

N22 G76 X30 Z-40 P919 Q250 F1.5

N23 G0 X150 Z100

Trouble shooting in CNC machines

Objectives : At the end of this lesson you shall be able to
- **list the trouble shooting areas in CNC machine**
- **state the general failure in electrical and electronic devices**
- **brief the trouble shooting symton and remedies in CNC machine elements.**

Introduction: A CNC Machine operates on electrical,electronics mechanical, pneumatic and hydraulic system invovled in its automation. If one of the above elements doesnot function or function partially the whole machine will come to stand still position. In CNC machine trouble shooting arises due to electrical,servo and mechanical problems.

The trouble shooting can be split in to:

- PLC trouble shooting
- Low voltage problem
- Power up problem
- Zero return problem
- Door inter lock circuit problems
- Relay board repair and problems
- Hydraulic system trouble shooting

General causes of failure in electrical and electronic devices

- Poor earthing
- Loose connection
- Dust particles or coolant entry
- Improper ventillation
- Blocked air circulation
- Poor supply voltage condition
- Sag for duration less than 2.5 seconds
- Surge for duration less than 2.5 seconds
- Spike (very high voltage lasting for few micro seconds)
- Blockout/Brown out greater than 2.5 seconds
- EMI (electromagnetic interference) affect switching drill, leads to data corruption.

Tips

1 Earthing between servo amplifier and CNC machine is a wrong practice. CNC mchine and electrical cabinet should be earthed directly individually.

2 Loose connections leads to failure of CRT monitors, input/output cards, measuring circuit, interfaces.

3 Dust particles results in short circuit,corrosion of tracts, PCBs and fuse getting blown. To avoid this dust protective guards should be fitted.

4 Improper ventillation results in malfunctioning of electronic circuits.

5 Poor supply voltage with poor voltage regulation affects the sensitive electronic device. Suggested not to operate welding machines near CNC machine.

6 Electrostatic charges transfer from human body may damage electronic components,leads to lose of data and hence, store PCBs in anti-static covers.

Poor switching devices like contactors,chokes. Transformers are to be grouped & separated from CNC system. The AC cables should not run parallel for long distance.

Safety Interlocks in CNC machines:

Power chucks should be damped befroe execution in MDA, Auto mode or single mode.

During cycle, in case of pressure failure, both spindle and axis should stop.

If slide lubrication system fail, cycle may continue till M30/ M02 of programme. The fault has to be rectified for cycle restart.

Jogging of axis is not permitted when the cycle is in progress. Spindle & collant stops automatically with M30/ M02 command.

If an unspecified battery is used (For memory, pulse coder) it may explode. Replace the battery only with the specified one (A02B-0177-K106). Dispose used batteries in accordance with applicable laws.

Trouble shooting of Alarm 960 due to abnormal 24V input power.

Shaft jammed, Defective motor, PLC malfunction, Fuse on spindle drive etc are other causes.

Effects :

Spindle not rotating

The cause and effect diagram shown, can be further developed by adding some more causes.

By carrying out the corrective action the problem can be sorted out.

Trouble shooting on specific elements

Elements	Sympton	Remedy
1 Motor Surface	Cutting fluid on motor	Clean the motor
2 Fan motor	Not rotating remove dirt etc	Tighten the loose connection,
3 Motor shaft bearing	Unusual sound	Replace Bearing
4 Abnormal vibration	Noise	Check all Bearing, belts,etc
5 Cooling air path	Clogged with dust	Clean air path
6 Interlock safety circuit	control cabinet	Check all interlock keys fully engaged at rest. Check interlock module in control cabinet, check contacters
7 Chuck/Actuator	Hyd, pump not running	Check ladder diagram Power ON and test, set the pressure higher Check input signal of foot switch
8 Soft over travel machine alarm occurs on zero returning	Brake does not stop axis	Reset after completing zero return
9 CNC is crashed (some slips during machine switch)	CNC control gets goofed up	Power down & then backup or reset
Backup battery	Needs replacement	Keep encoder position when power is shut off
Lubrication pump problem	Noise in axis Axis motor overload alarms	Double the oil Check for leaks Check the distribution mainfold Clean the filter if clogged, ensure correct oil is fed.

Remedies For Chart 1

a Any or all of the following: Replace dirty filters. When strainers in solvent compatible with system fluid. Clean clogged inlet line. Clean reservoir breather vent. Change system fluid. Change to proper pump drive motor speed. Overhaul or replace super-charge pump. Fluid may be too cold.

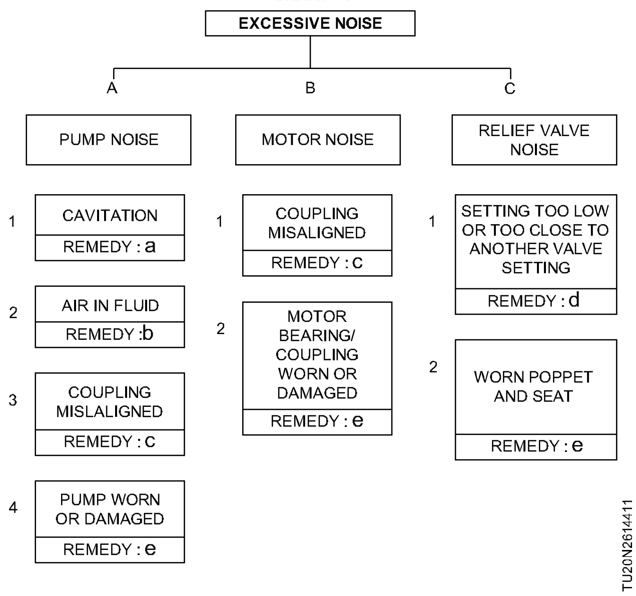
b Any or all of the following: Tighten leaky inlet connection. Fill reservoir to proper level (with rate exception all return lines should be below fluid level in reservoir). Bleed air from system. Replace pump shaft seal (and shaft if worn at seal journal).

c Align Unit and check condition of seals, bearings and coupling.

d Install pressure gauge and adjust to correct pressure.

e Overhaul or replace.

Fig 1



Remedies For Chart 2

a Any or all following: Replace dirty filters.clean clogged inlet line. Clean reservoir breather vent. Change system fluid. Change to proper pump drive motor speed. Overhaul or replace supercharge pump.

b Any or all the following: Tighten leaky inlet connections. Fill reservoir to proper level (with rare exception all return lines should be below fluid level in reservoir). Bleed air from system. Replace pump shaft seal (and shaft if worn at seal journal).

c A link unit and check condition of seals and bearings. Locate and correct mechanical binding. Check for work load in excess of circuit design.

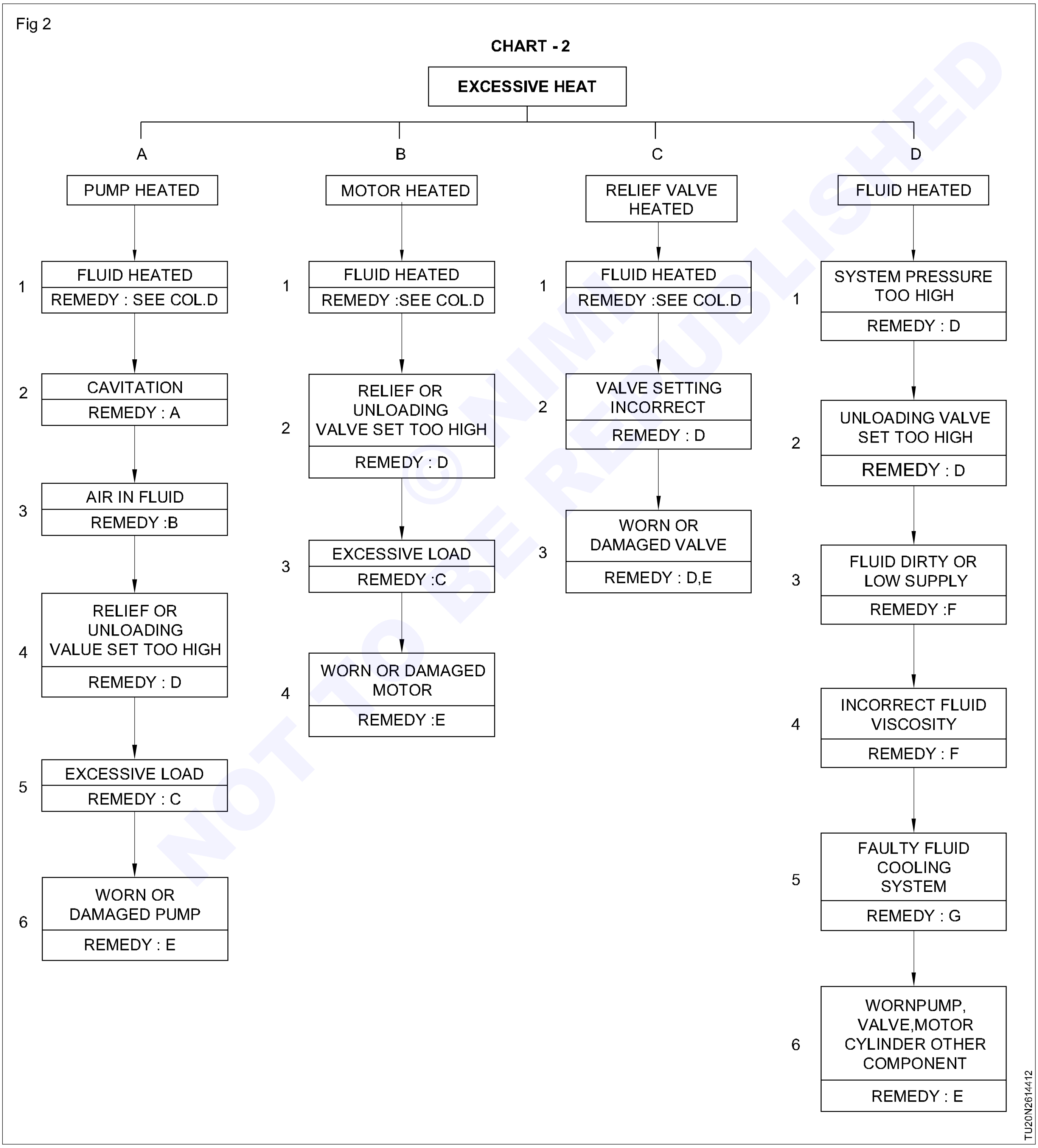
d Install pressure gauge and adjust to correct pressure (keep atleast 125 psi difference between valve settings)

e Overhaul or replace.

f Change filters and also system fluid if improper viscosity. Fill reservoir to proper level.

g Clean cooler and/or cooler strainer. Replace cooler control valve. Repair or replace cooler.

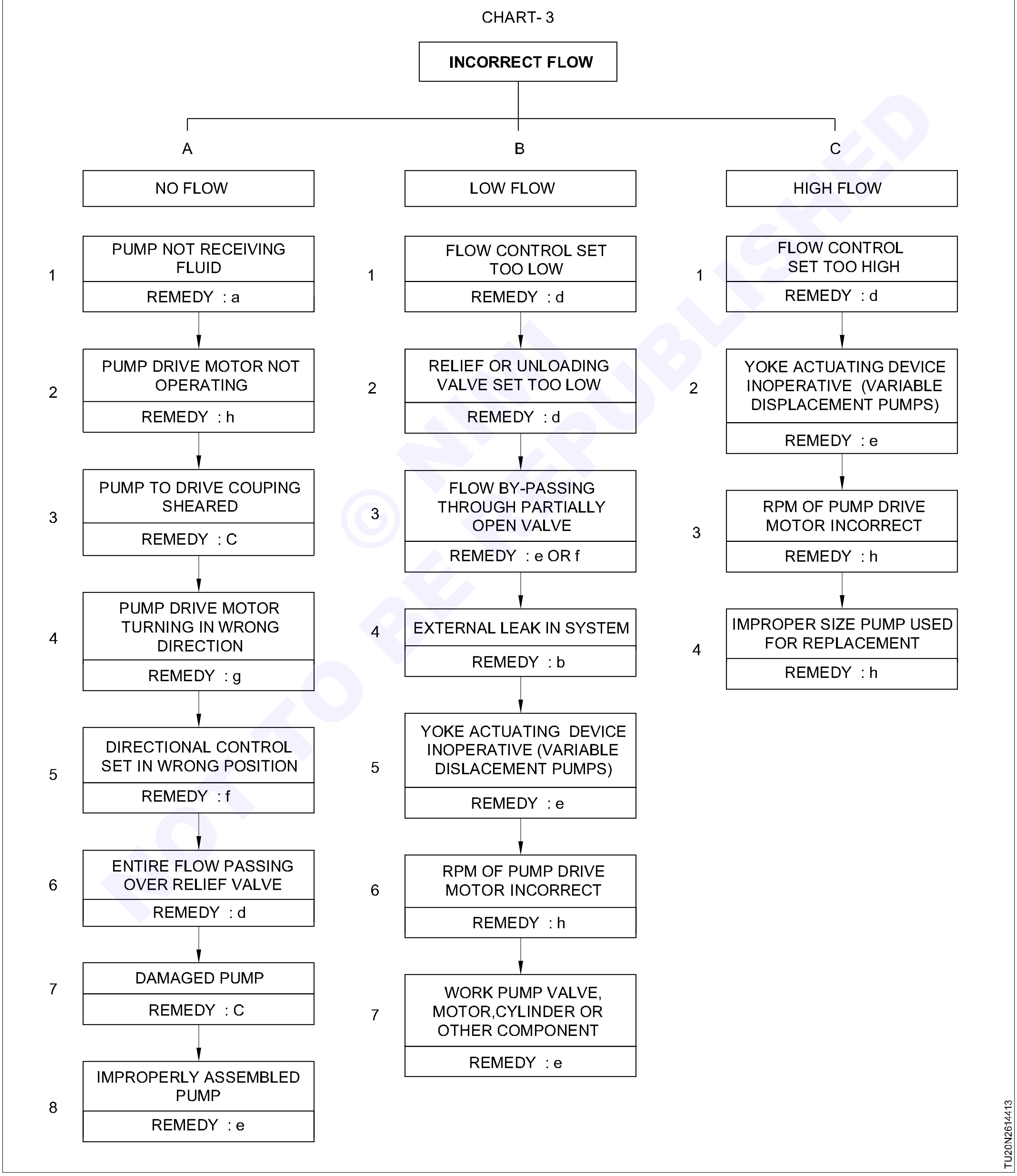
Fig 2



Remedies For Chart 3

- a Any or all of the following : Replace dirty filters, clogged inlet line. Clean reservoir breather vent. Fill reservoir to proper level. Overhaul or replace supercharger pump.
- b Tighten leaky connection. Bleed air from system.
- c Check for damaged pump or pump drive. Replace and align coupling.
- d Adjust
- e Overhaul or replace.
- f Check-position of manually operated controls. Check electrical circuit on solenoid operated controls. Repair or replace pressure pump.
- g Reverse for rotation.
- h Replace with correct unit.

Fig 3



Remedies For Chart - 4

a Replace dirty filters and system fluid.

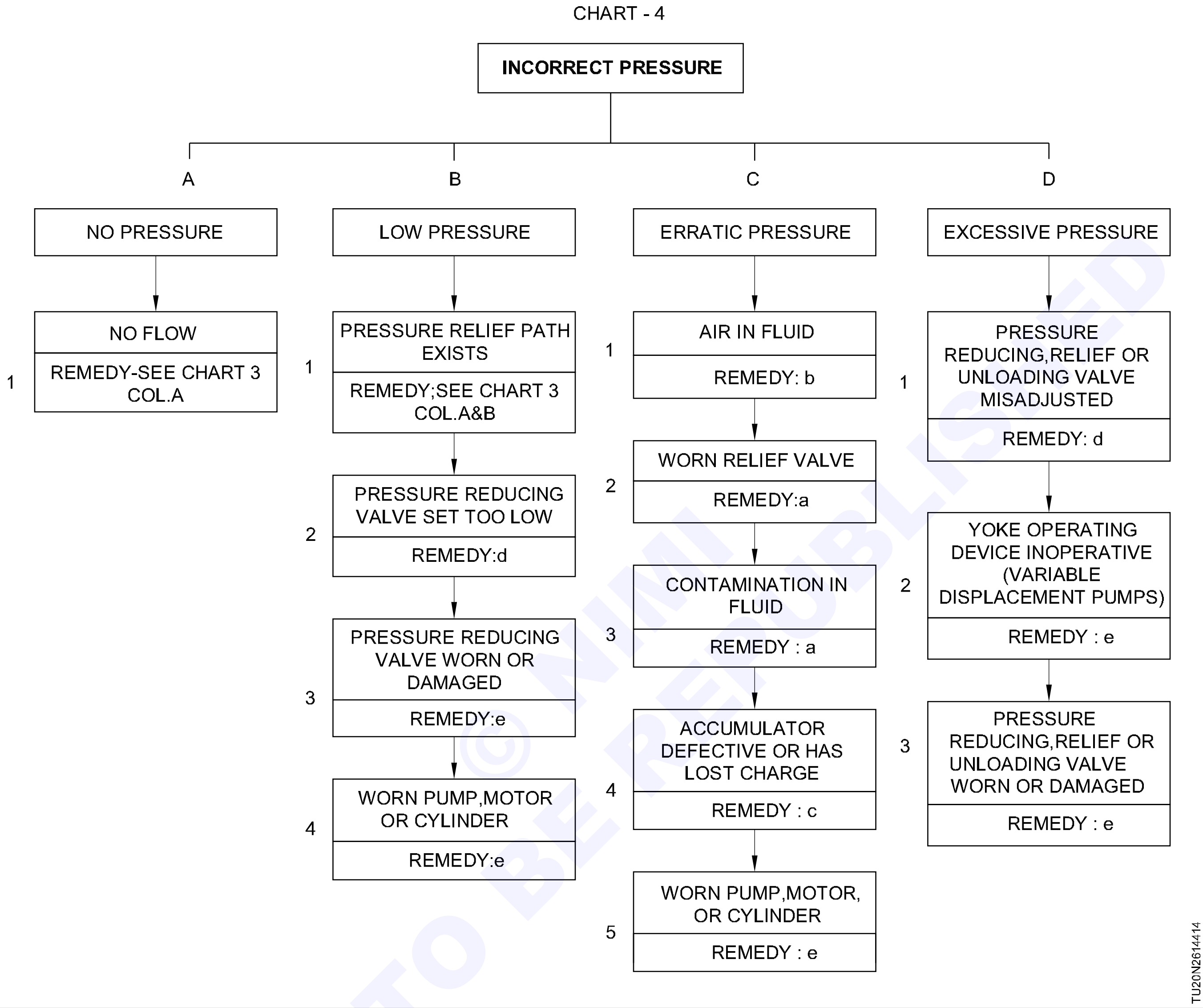
b Tighten leak connecitons (fill reservoir to proper level and bleed air from system).

c Check gas valve for leakage. Charge to correct pressure. Overhaul of defectives.

d Adjust.

e Overhaul or replace.

Fig 4



Remedies For Chart - 5

a Fluid may be too-cold or should be changed to clean fluid of correct viscosity.

b Locate bind and repair.

c Adjust, repair or replace.

d Clean and adjust or replace. Check conditons of system fluid and filters.

e Overhaul or replace.

f Repair command console or interconnecting wires.

g Lubricate.

h Adjust, repair or replace counterbalance valve.

Fig 5

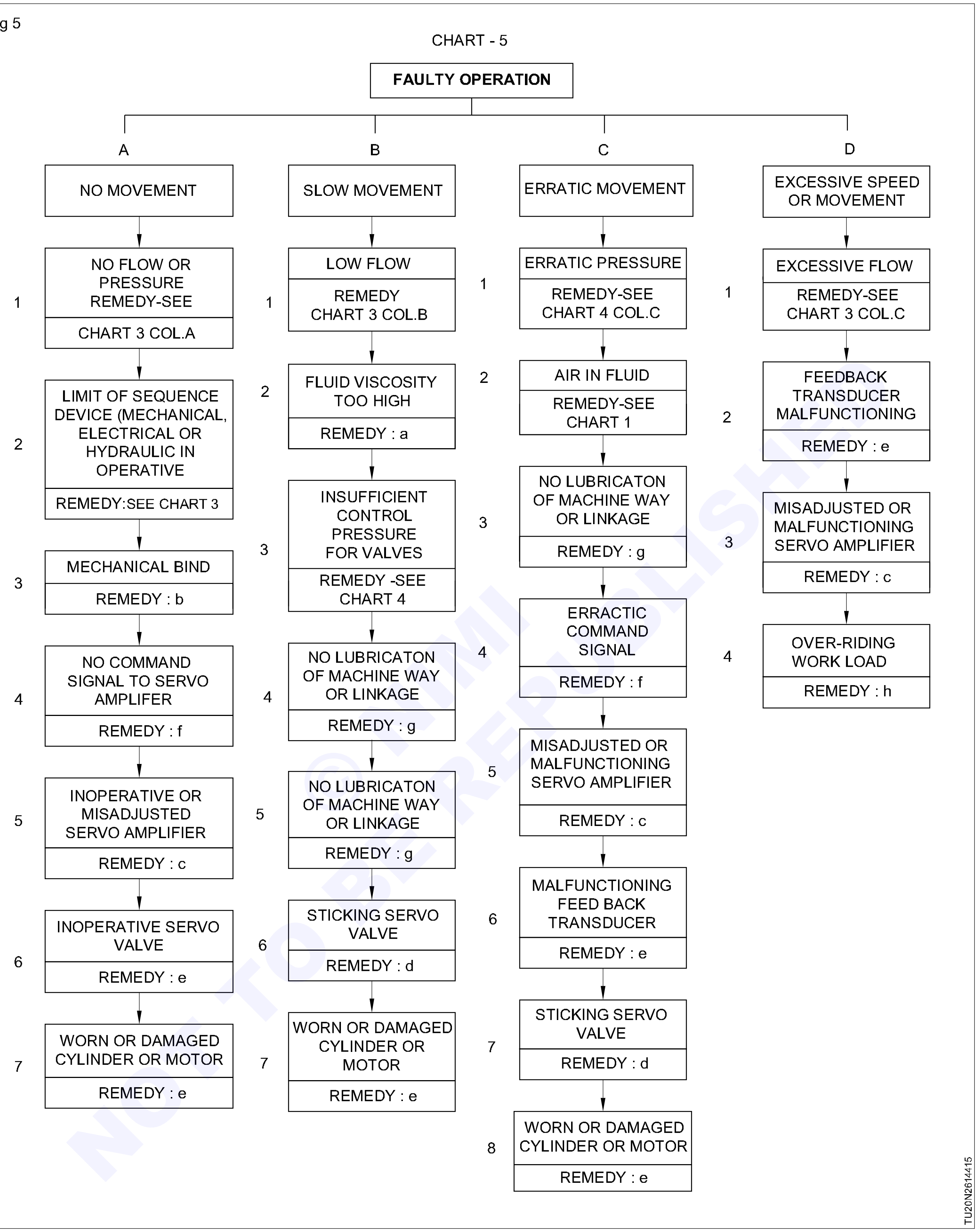


CHART - 5
FAULTY OPERATION
A
NO MOVEMENT
1
NO FLOW OR PRESSURE REMEDY-SEE
CHART 3 COL.A
2
LIMIT OF SEQUENCE DEVICE (MECHANICAL, ELECTRICAL OR HYDRAULIC IN OPERATIVE
REMEDY:SEE CHART 3
3
MECHANICAL BIND
REMEDY : b
4
NO COMMAND SIGNAL TO SERVO AMPLIFER
REMEDY : f
5
INOPERATIVE OR MISADJUSTED SERVO AMPLIFIER
REMEDY : c
6
INOPERATIVE SERVO VALVE
REMEDY : e
7
WORN OR DAMAGED CYLINDER OR MOTOR
REMEDY : e
B
SLOW MOVEMENT
1
LOW FLOW
REMEDY CHART 3 COL.B
2
FLUID VISCOSITY TOO HIGH
REMEDY : a
3
INSUFFICIENT CONTROL PRESSURE FOR VALVES
REMEDY -SEE CHART 4
4
NO LUBRICATON OF MACHINE WAY OR LINKAGE
REMEDY : g
5
NO LUBRICATON OF MACHINE WAY OR LINKAGE
REMEDY : g
6
STICKING SERVO VALVE
REMEDY : d
7
WORN OR DAMAGED CYLINDER OR MOTOR
REMEDY : e
C
ERRATIC MOVEMENT
1
ERRATIC PRESSURE
REMEDY-SEE CHART 4 COL.C
2
AIR IN FLUID
REMEDY-SEE CHART 1
3
NO LUBRICATON OF MACHINE WAY OR LINKAGE
REMEDY : g
4
ERRACTIC COMMAND SIGNAL
REMEDY : f
5
MISADJUSTED OR MALFUNCTIONING SERVO AMPLIFIER
REMEDY : c
6
MALFUNCTIONING FEED BACK TRANSDUCER
REMEDY : e
7
STICKING SERVO VALVE
REMEDY : d
8
WORN OR DAMAGED CYLINDER OR MOTOR
REMEDY : e
D
EXCESSIVE SPEED OR MOVEMENT
1
EXCESSIVE FLOW
REMEDY-SEE CHART 3 COL.C
2
FEEDBACK TRANSDUCER MALFUNCTIONING
REMEDY : e
3
MISADJUSTED OR MALFUNCTIONING SERVO AMPLIFIER
REMEDY : c
4
OVER-RIDING WORK LOAD
REMEDY : h
TU20N2614415

Factors affecting Quality & Productivity

Objectives : At the end of this lesson you shall be able to
- **list the factors affecting turned parts.**

Factors affecting turned part/production

The quality and characteristics of a turned part produced on a lathe can be influenced by various factors. These factors can affect the precision, surface finish, and overall performance of the turned part. Some of the key factors include:

Machine Setup and Condition

The condition of the lathe machine, including its alignment, rigidity, and overall maintenance, can significantly impact the turned part's quality.

Tooling Selection

The choice of cutting tools, tool geometry, and tool material is crucial for achieving the desired surface finish and dimensional accuracy.

Cutting Speed (Surface Speed)

The rotational speed of the workpiece and tool, often referred to as cutting speed or surface speed, must be optimized for the material being turned to prevent overheating, tool wear, and achieve the desired finish.

Feed Rate

The rate at which the tool advances along the workpiece (feed rate) affects chip formation, tool life, and surface finish.

Depth of Cut

The depth of cut refers to how much material is removed in a single pass. It can influence cutting forces, tool wear, and surface finish.

Material Properties

The type of material being turned, its hardness, ductility, and other properties can affect tool wear, chip formation, and surface finish.

Coolant and Lubrication

Proper coolant or lubrication is essential to control heat generation, reduce tool wear, and improve chip evacuation.

Tool Wear and Tool Life

As cutting tools wear, the quality of the turned part can deteriorate. Monitoring and replacing tools at the right time is crucial.

Vibration and Chatter

Excessive machine vibrations or chatter can lead to poor surface finish and dimensional inaccuracies. Damping and minimizing vibrations are important.

Workpiece Holding and Fixturing

The stability and precision of the workpiece holding system or fixtures can affect part quality. Proper workpiece clamping is necessary to prevent distortion.

Cutting Inserts and Inserts Wear

In the case of indexable inserts, the condition and wear of the inserts can significantly impact the quality of the turned part.

Part Geometry and Tolerances

The desired part geometry and tolerances must be well-defined in the machining process to achieve the required precision.

Operator Skill and Experience

The operator's skill, experience, and knowledge of the machining process play a critical role in producing high-quality turned parts.

Environmental Conditions

Factors such as temperature and humidity in the machining environment can influence part quality, particularly in materials sensitive to environmental changes.

Chip Evacuation

Efficient chip evacuation is important to prevent chip recutting, which can negatively impact surface finish.

Parting of opeation in a CNC turning

Objectives : At the end of this lesson you shall be able to
- **list the tips of parting off operation**
- **explain the insets grade of parting of operation depending of the metal to be part off**
- **brief G75 cycles.**

Tips for parting off minimize overhand (OH)

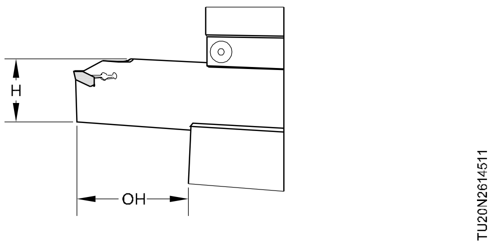
At long OH:

- Use a light cutting geometry e.g. CM

OH less that 1.5 x H: (Fig 1)

- Use recommeded feed for the geometry

Fig 1



OH exceeds 1.5 x H:

- Reduce feed rate to the lower end of recommeded feed for the geometry.
- Shorter overhang decreases bending down in cubic

Centre height (Fig 2&3)

- Center height ±0.1 mm (±0.1 mm (±0.004 inch)
- At long overhangs, set cutting edge 0.1 mm (0.004 inch) above centre to compensate for bending down.

Fig 2

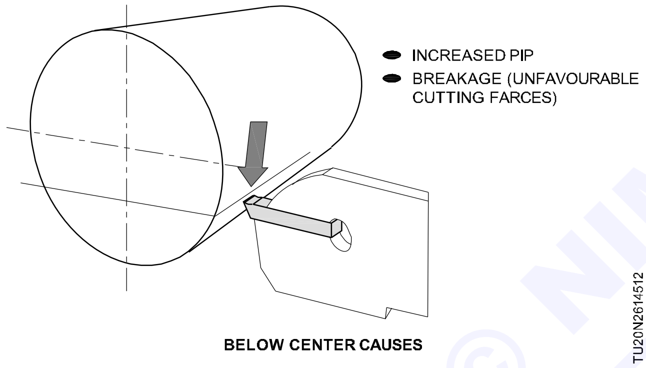
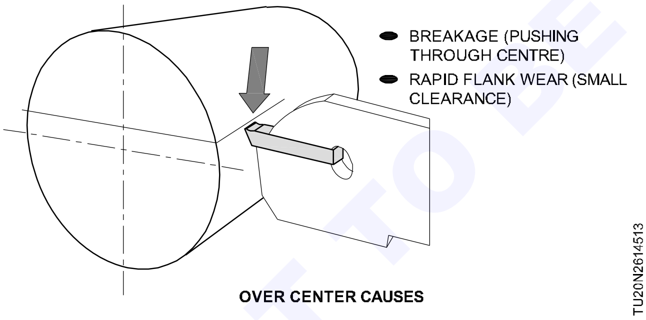


Fig 3



Always reduce feed before centre

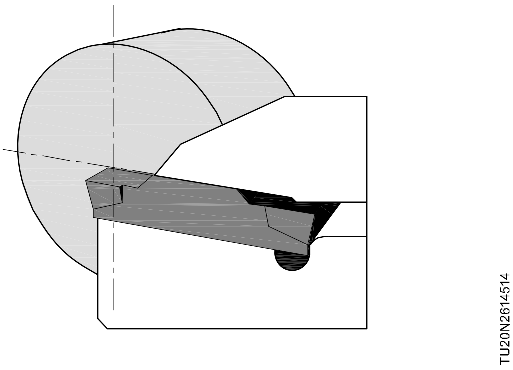
Breakages in parting off bars generally occur at centre. Always reduce feed by -75% from 2mm (0.08 inch) before centre:

- Lower feed at centre reduces the forces and increases tool life.
- Higher feed in periphery improves productivity and tool life.
- Feed reduction at centre drastically increases tool life.

Always stop feed before reaching centre (Fig 4&5)

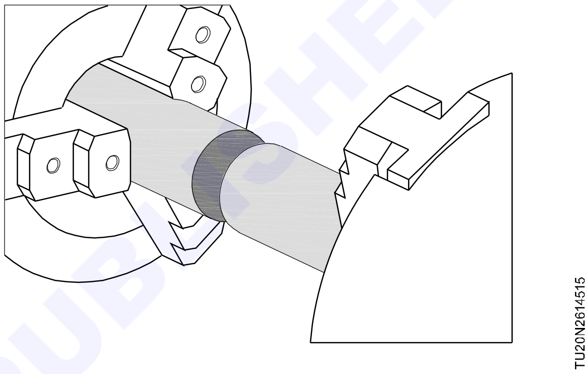
- Stop feed 0.5 mm (0.02 inch) before centre
- The component will fall off by the centrifugal force.
- Feeding through centre causes breakage (Fig 4)

Fig 4



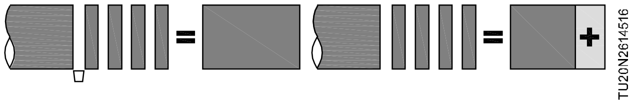
- A sub chuck can be used to pull the component (Fig 5)

Fig 5



- Leave a pip with ∅1mm (0.04 inch) to be pulled off.
- Reduce insert width to save material. (Fig 6)

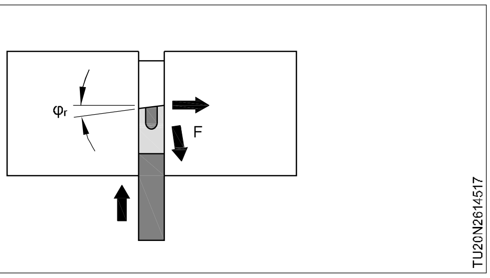
Fig 6



Pip free parting (Fig 7)

- Front angle reduces pip and burr on one side
- Use front angle inserts only at small overhangs
- Front angle reduces tool life and increase bending
- For longer overhangs use neutral inserts

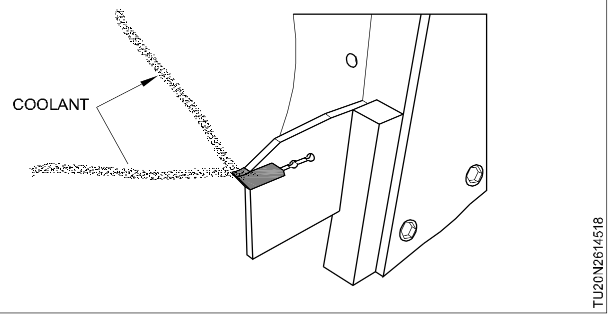
Fig 7



High precision coolant (HP) (Fig 8)

- Accesses cutting edge even in deep grooves
- Tools with HP is first choice for parting and grooving
- Improves chip control and surface finish

Fig 8



- Internal coolant decreases temperature
- Largest gains at long time in cut and materials with low conductivity (HRSA, stainless steel)
- Effective coolant allows usage of tougher grades with maintained or increased tool life
- Increase cutting speed with 30-50% when HP is used
- Shut off coolant at the diameter while the machine reach its rpm limit to avoid build-up edge.

Program for parting off

Many of the machine controls have a peck option on G75 grooving code, which is used for parting off. In conventional CNC mode, there is a provision for “Parting off with peck” option Fanuc’s G75 is newer control uses a two line format:

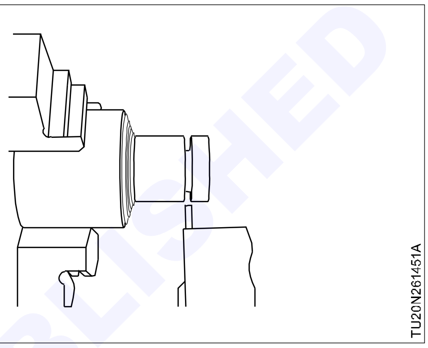
G75 R0.002

G75 X1.0 P0.125 F10 (Fig 10)

Where R is the amount of retraction after each peck (X1.0, Z-10.0) is the lower left corner of the grooving to its reference point and the grooving is being done from right to left. P is the depth of cut for each peck at a feed rate of F. So each peck cuts a distance of P retracts a distance R, Then re-engages the material and does another peck of distance p and cycle goes on till the bottom is reached.

If ‘Q’ value is specified with a Z value, then it means the groove is wider than tool widths.

Fig 10



Bar Feeding System through Bar Feeder

Objectives : At the end of this lesson you shall be able to
- **learn about the bar feeding operation**
- **learn the advantages of bar feeding mechanism**
- **learn the limitations of the system.**

The Bar feeding mechanism is a metal cutting machine tool designed to feed the metal. This machine is exclusively intended for mass production and they represent faster and more efficient way to feed a stock. Automatic Bar Feeding mechanism (ABF) consists of three major blocks. They are:

- Metal feed mechanism
- Bar clamping mechanism
- IR sensor unit

The IR unit determines the dimension for cutting the bar clamping is through pneumatic system, which is highly reliable and effective.

Need for Automation of Bar feeding

- To achieve mass production in manufacturing process.
- To reduce man power employed.
- To increase the efficiency of the machine.
- To reduce the workload & production cost.
- To reduce material handling & production time.
- To reduce the fatique of workers.
- To achieve good product quality.
- To reduce maintenance cost.

The motor will be rotated, so that the bar is moving from initial position to the determined position.

IR sensor unit is used to determine the bar dimension to be cut.

The compressed air from the compressor reaches the solenoid valve. The solenoid valve changes the direction of flow according to the signal from the sensor unit.

The pneumatic cylinder is used to clamp the work piece automatically with the help of IR sensor unit.

Advantage

1 Simple in construction than mechanical hacksaw.

2 It is a compact one

3 Less maintenance

4 Fast production

Limitation

- Additional cost is required to do the automation.
- Leakage of air affects the working of the unit.

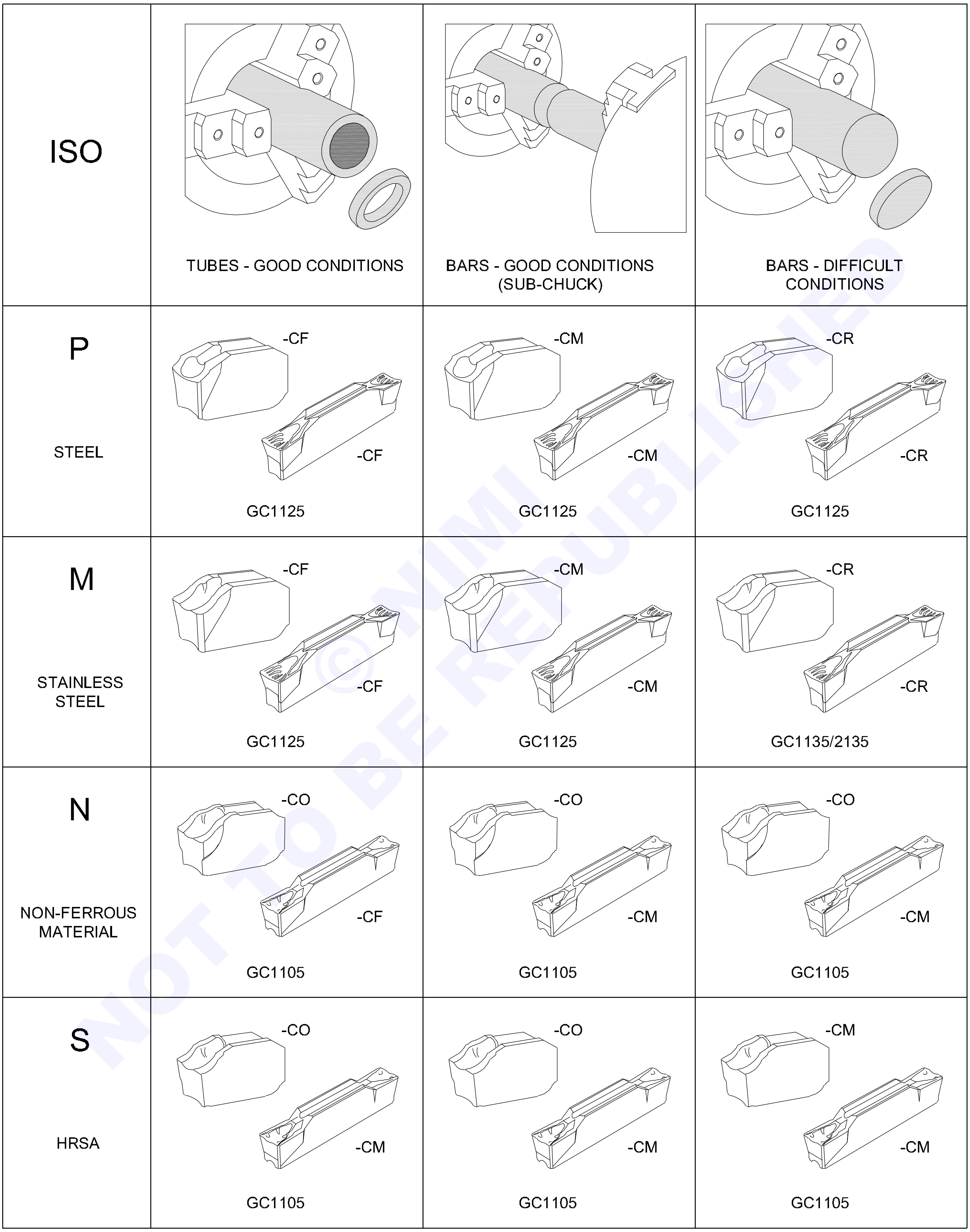
Application

- Small and medium scale industries Application.
- Metal Cutting Industries and Work Shops.

Geometry and grade

Table 1

Fig 9



ISO	TUBES - GOOD CONDITIONS	BARS - GOOD CONDITIONS (SUB-CHUCK)	BARS - DIFFICULT CONDITIONS
P STEEL	-CF -CF GC1125	-CM -CM GC1125	-CR -CR GC1125
M STAINLESS STEEL	-CF -CF GC1125	-CM -CM GC1125	-CR -CR GC1135/2135
N NON-FERROUS MATERIAL	-CO -CF GC1105	-CO -CM GC1105	-CO -CM GC1105
S HRSA	-CO -CM GC1105	-CO -CM GC1105	-CM -CM GC1105

GEOMETRY AND GRADE

Data input and output

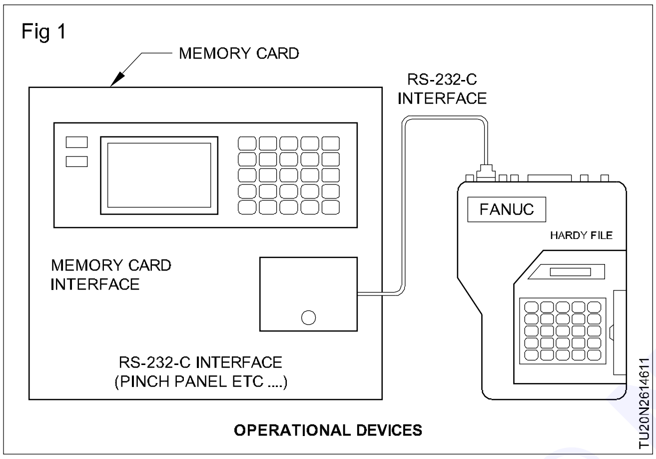
Objectives : At the end of this lesson you shall be able to
- **list the type of data can be input and output**
- **brief the steps in inputting the program explain the steps in outputting the program**
- **state the steps in putting and outputting offset datas.**

Data input /output

NC data is transferred between the NC an external input/output devices such as the handy file.

Information can be read into the CNC from a memory card and written from the CNC to the memory card, using the memory card interface at the left of the indicator. (Fig 1)

Fig 1



The following types of data can be entered and output.

1 Program

2 Offset data

3 Parameter

4 Pitch error compensation data

5 Custom macro common variable

Before an input/output device can be used, the input/output related parameters must be set.

Procedure for inputting a program

- Make sure the input device is ready for reading
- Press the edit switch on the machine operator's panel
- When using a floppy, search for the required file according to the procedure in section III -8.2.
- Press function key PROG, then the program contents display screen or program directory screen appears.
- Press soft key [(OPRT)]
- Press the rightmost soft key (next - menu key)
- After entering address O, specify a program number to be assigned to the program. When no program number is specified here, the program number used on the floppy or NC tape is assigned.

-- Press soft keys [READ] and [EXEC]

The program is input and the program number specified in step 7 is assigned to the program.

Procedure for outputting a program

- Make sure the output device is ready for output.
- To output to and NC tape, specify the punch code system (ISO or EIA) using a parameter.
- Press the EDIT switch on the machine operator's panel.
- Press function key PROG , then the program contents display screen or program directory screen appears.
- Press soft key [(OPRT)]
- Press the rightmost softkey (next-menu key).
- Enter address O.
- Enter a program numner. If -9999 is entered, all programs stored in memory are output.
- To output multiple programs at one time, enter a range as follows:
- Programs No. to No are output
- The prpgram library screen displays program numbers in ascending order when bit 4 (SOR) of parameter NO. 3107 is set to 1.
- Press soft keys [PUNCH] and [EXEC]

The specified program or programs are output.

Offset data input and output

Inputting offset data

Offset data is loaded into the memory of the CNC from a floppy or NC tape. The input format is the same as for offset value output. When an offset value is loaded which has the same offset number as an offset number already registered in the memory, the loaded offset data replaces existing data.

Procedure for inputting offset data

- Make sure the input device is ready for reading
- Press the EDIT switch on the machine operator's panel
- When using a floppy, search for the required file according to the procedure in section III -8.2.

- Press function key OFFSET SETTING to display tool offset screen.
- Press soft keys [(OPRT)], then the tool compensation screen appears.
- Press rightmost softkey (continuous menu key)
- Press soft keys [READ] and [EXEC]
- The input offset data will displayed on the screen after completion of input operation.

Outputting offsetting data

All offset data is output in a output format from the memory of the CNC to a floppy or NC tape

Procedure for outputting offset data

- Make sure the output device is ready for output.
- Specify the punch code system (ISO or EIA) using a parameter.
- Press the EDIT switch on the machine operator's panel
- Press function key OFFSET SETTING to display tool offset screen.
- Press soft key [(OPRT)]
- Press the rightmost soft key (continuous menu key)
- Press soft keys [PUNCH] and [EXEC].
- Offset data is output in the output format described below.

DNC system

Objectives : At the end of this lesson you shall be able to
- **state the DNC system**
- **list the functions of DNC system**
- **brief the advantages of DNC system.**

DNC System

It is a manufacturing system in which number of machines are controlled by a computer through direct-connection and in realtime. It is also a system connecting to a set of NC. Machines to common memory for a part programme or machine programme storage with provision of on-demand distribution of data to machines.

The main components of a DNC are;

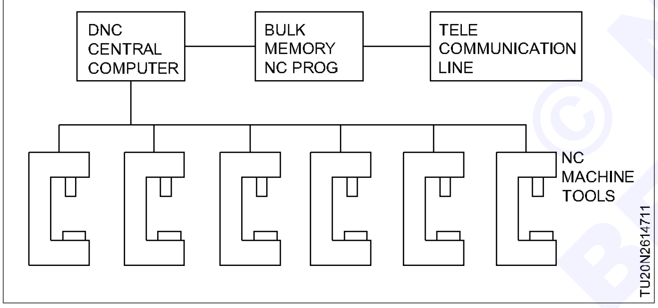
1 Central computer.

2 Bulk memory with storage of part programme.

3 Telecommunication lines and set of NC machine tools.

A schematic sketch indicating a set of NC machine connected to a DNC is shown below:

Fig 1



There are two types of DNC machines namely 1. Behind the Tape Reader System (BTR), 2. Special Machine Control Unit (MCU). In BTR system computer is linked directly to NC controler unit the operation of the system is very similar to conventional NC. The controler unit, uses two temporery storage buffer to receive instruction from DNC computer and convert them in to machine action. One buffer receives Data the other is providing controlling instruction to machine tools. The cost of BTR system is very less. The special machine control unit replace a regular controler, with the control unit capable of facilating communication between machine tool and computer. This system achives very high accuracy of interpolation and fast metal removal rate.

Functions of DNC

The function of DNC system is to perform.

1 NC with out punched tape.

2 NC part programme storage.

3 Data collection, processing and reporting.

4 Communication.

A communication Network accomplishes the previous functios of DNC. This links DNC with central computer and machine tools, Central computer and NC part programmer terminal, central computer and Bulk Memory.

Advantages of DNC system

It eliminates punched tape and tape reader.

It is convenient to store are NC part programme in computer files.

It has greater computational capability and flexibility.

It can report the shop performance at the click of a button.

DNC's are very convenient in editing and dignoistic feature.

DNC is a first step in the development of production plant automated with computer.

Use of CAM Programme

Objectives : At the end of this lesson you shall be able to
- **list the important of CAM in manufacturing process**
- **brief the output format of CAM programe**
- **explain terms like CAD/CAE/DNC etc.**

Introduction

Computer-aided manufacturing (CAM) is the use of a software to control machines and the related devices & equipment in the manufacture of components. CAM software also assist in all operations such as process planning, production management, product transportation within the plant, storage of semi finished/finished product.

The purpose of CAM is to increase productivity and improve tooling, to enhance greater consistency of quality, minimising raw material wastage. CAM software is also useful for training & academic educational purposes.

CAM process has come out of CAD and sometimes CAE computer aided engineering, as a model generated by CAD, verfied by CAE and put into CAM software, which controls the CNC machine tool

CAM is a numerical control (NC) programming tool, for 2D/3D models generated in CAD. CAM does not eliminate the need for skilled professionals, but assists manufacturing personnel in building skills through visualisation, simulation and optimization tools. Integration of CAD with other components of CAD/CAM/CAE, Product Life Cycle (PLC) management environment requires an effective CAD data exchange, such as IGES/STL/ Parasolid formats. The output from CAM is usually a text file of G-code/M-code, add thousands of commands, which can be transfered to a DNC programme.

CAM is specifically effective in typical areas like.

- High speed machining, streamlining tool paths.
- Multifunction machining.
- 5-axis machining & Automatic machining.

A finished components has to undergo different machining process in its conversion from raw material i.e,

a Rough machining: From raw stock, raw casting passes through zig-zag clearing, plunge roughing, rest roughing, trochodial milling aiming at maximum material removal in the least amount of time, maintaining dimensional accuracy.

b Semi-finishing: In this process, small amount of material only removed, to enable the tool to cut accurately such as Raster passes, constant step-over process, pencil milling etc.

c Finishing machining: It involve very light material removal to produce polished finish product. The feed is increased to target SFM with high speed refered as highspeed machining (HSM) CAM is mainly useful for contour milling on four or more axis.

Usefulness of CAM

Good CAM software is equally as important to manufacturers as the powerful machines and tool they use to cut desired parts. Machine shops of all sizes and budgets are reaping benefits of good CAM software beyond efficiency programming their machining jobs. Users can structure their job, set their Toolpath, then use the simulation function to make sure their plan goes accordingly. Part gouges or collisions can potentially ruin a very crucial and expensive CNC machine, threatening to affect that shop's profitablilty or ability to take on additional projects. It is much easier to read just a toolpath in the CAM software than it is to fix or repalce a CNC machine!

Structuring your job with CAM Software

Let's take s sprocket for example. You created your sprocket using CAD software, now how will you make it? CAM software looks at what designed& determines how to machine it out of materials. For starters, most CAM software products provide a standard "Job Tree" for machining strategy organization. Then you will need to set up and save the features of the machine from your shop within the CAM software. This is important for developing programs that are specific to your machine, allowing you to easily machine future projects, editing machine setting as needed. Next, you will identify your stock, allowing you to set initial work cooridinates, material type and the tools to be used during machining. Lastly, you set your cutting conditions, tool patterns, tool crib and tool holder for error-free CNC programming. One of the greatest benefits of CAM software is the ability to save the information you put in the system, making future projects much more easily programmed.

Setting your Toolpaths within the CAM software

Once you have your starting points clearly identified (machine, tools and stock), you can move into the next phase of developing your machining operations. Start with your stock. Decide the toolpaths for your roughing and finishing cycles that will ultimately determine your desired part design. Your Job Tree comes into play now, keeping your machining operations organized and correctly sequenced. CAD-CAM software, like that from BobCAD-CAM, used wizard guides that act as a series of dialog boxes that walk you through the process step-by-step until your Toolpath is properly created. Wizards are sort of a fool-proof way to make sure all your information is entered in correctly, reducing programming time significantly for users.

After you created a Toolpath for each of your operations, you can move into most critical aspect of CAM software; simulation. This allows you to see your machine create your part in a digital environment. There are three major things that are being accomplished during the time:

1 Part deviation analysis- the CAM software identified what's not cut by the tool or where the tool went too deep using multiple colors to represent different levels of deviation.

2 Costly gauges or collisions are digitally detected prior to engaging your shop's machine.

3 Cycle time can be easily calculated.

After you structure your job, creat your Toolpath and run simulations to satisfaction, you are ready for post processing. This is the final step before your part is machined. G-code is created by the CAM software, which is the NC file that is read by the CNC machine. Think of the G-code as the GPS in your car; it basically tells the machine where to go. Each machine's G-code can vary, so it's important you have the proper post processor added to your CAM software.

Setting of tool for taper threads calculation of taper setting and thread depth

Objectives : At the end of this lesson you shall be able to
- **state the application of taper thread**
- **state the methods of producing taper threads**
- **describe methods of turning tapers on a lathe with its merits and demerits.**

Taper thread

Taper threads are used on fasteners and pipe. A common example of a fastener with a taper thread is wood screw. The threaded pipes used in some plumbing installations for the delivery of fluids under pressure have a threaded section that is slightly conical. Examples are NPT and BSP series. The seal provided by a threaded pipe joint is created when a tapered externally threaded into an end with internal threads.

Inside tapered threads

- Insert the pipe into the chuck of the lathe and tighten down the chucks with an Allen wrench. Install the tap in the opposite chuck of the lathe. Move the ram forward until the tapping tool reaches the inside of the pipe. Center the tapping tool to the interior of the pipe.
- Start the ram and slowly close the tapping tool. Lubricate the tapping tool periodically with metal cutting fluid as it machines the tapered threads. Reverse the ram of the lathe and slowly back the tapping tool out of the interior of the pipe. Check the tapered threads with a thread gauge.
- Set the steel plate, with holes drill holes, on the base plate of the drill press and secure it in place. Insert a tapping tool in the chuck of the drill press and secure with a chuck tool. Fill the drilled hole with metal cutting fluid and slowly lower the tapping tool. Align the tapping tool before machining the tapered threads.

Outside tapered threads

- Insert the pipe in the lathe's chuck and secure it in place, similar to the way you secured the pipe when machining inside tapered threads. Open the mouth of the opposite chuck and insert a threading die, instead of attaching a tapping tool to the opposite chuck. Tighten down the chuck. Align the threading die with the outside of the pipe.
- Cover both the end of the pipe and the interior of the die with metal cutting fluid .Start the lathe and close the ram slowly until it starts machining the tapered threads. Pour more cutting fluid on the end of the pipe as you continue to close the ram of the lathe. Reverse the lathe ram after reaching the desired thread depth. Checks the threads with the thread gauge.
- Insert the screw, bold or rod in the chuck of the lathe. Install a threading die in the opposite chuck of the lathe and machine the tapers threads, similar to the way you cut the threads on the outside of the pipe.

Cutting taper thread

Whereas the axial moment required for pitch feed is provided by the machine, the radial moment required in addition for producing a tapered thread is obtained by the tool slide feed. Do not clamp the slide by tightening locking screw .During thread cutting both the machine feed and the feed of the head must be in continuous engagement.

Same as per taper turning ,the sensitiveness to release of retaining pin must be reduced to the minimum by means of regulating screw stop rod must not be displaced by changing over from clockwise to anti-clockwise rotation from the position held.

Depending on the pitch of thread, taper angles will vary with varying cross feeds. The tool mounting and depth adjustment are same as taper turning attachment method.

The method of operation is the same as for straight thread cutting.

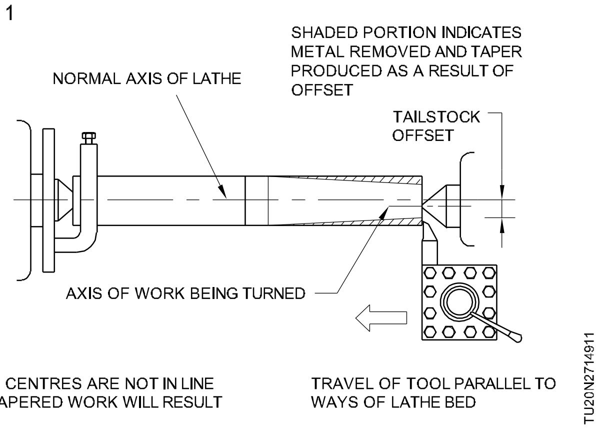
The taper for the taper thread is prepared by using normal taper turning attachment methods.

Tapers may be produced by any conventional methods according to the requirement and die sets are also used for producing taper threads.

Tailstock offset method (Fig 1)

In this method the job is held at an angle and the tool moves parallel to the axis. The body of the tailstock is shifted on its base to an amount corresponding to the angle of taper.

Fig 1

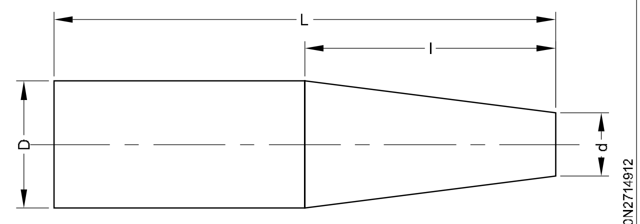


Tail stock set over (Fig 2)

Taper

Tapers may be produced by any conventional methods according to the requirement and die sets are also used for producing taper threads.

Fig 2



COMPRESSOR CROSS-SECTION VIEW

The taper can be turned between centres only and this method is not suitable for producing steep tapers.The amount of offset is found by the formula. (Fig 3)

$$\text{Offset} = \frac{(D-d)XL}{2l}$$

where D = big dia. of taper

d = small dia. of taper

l = taper length

L = total length of job

Advantages

Power feed can be given

good surface finish can be obtained

maximum length of the taper can be produced

external thread on taper portion can be produced

duplicate tapers can be produced

Disadvantages

only external taper can be turned

accurate setting of the offset is difficult

taper turning is possible when work is held between centres only.

damages the centre drilled holes of the work.

the alignment of the lathe centres will be disturbed

steep tapers cannot be turned

Calculation of Thread depth formula.

BSN - 0.6403 x Pitch

Metric - 0.6134 x Pitch

Square - 0.5 x Pitch

Acme - 0.5 x Pitch

B.A - 0.6134 x Pitch

Buttress - 0.75 x Pitch

Worm thread - 0.68 x Pitch

Taper turning by attachment (Fig 3)

This attachment is provided on a few modern lathes. Here the job is held parallel to the axis and the tool moves at an angle. The movement of the tool is guided by the attachment.

Taper angle is found by formula (Fig 3)

$$\tan\theta = \frac{D-d}{2l}$$

Advantages

Both internal and external tapers can be produced.

Threads on both internal and external taper portions can be cut.

Power feed can be given.

lengthy taper can be produced.

Good surface finish is obtained

The alignment of the lathe centres is not disturbed.

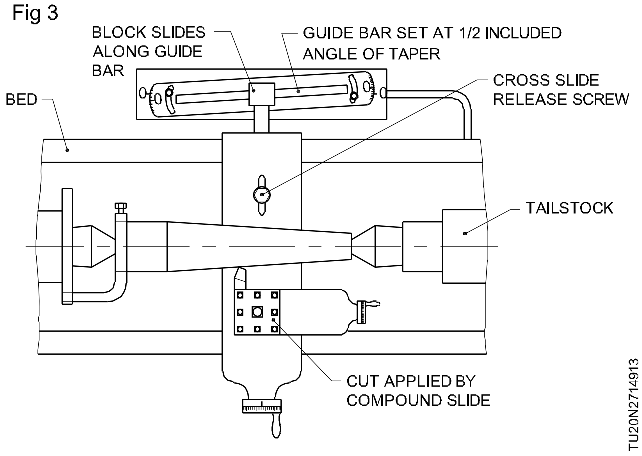
It is most suitable for producing duplicate tapers because the change in length of the job does not affect the taper.

The job can be held either in chuck or in between centres

Disadvantage

only limited taper angles can be turned.

Fig 3



Heat treatment of plain carbon steels

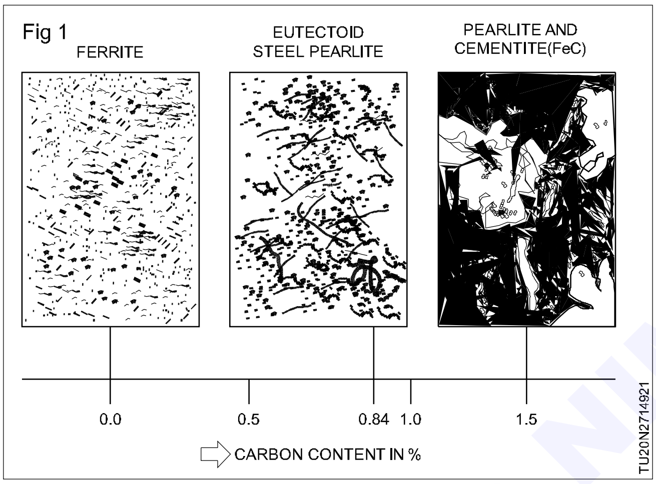
Objectives : At the end of this lesson you shall be able to
- **state the purpose of heat treatment of steel**
- **state the types of structure, constituents and properties of plain carbon steels.**

Heat treatment and its purpose

The properties of steel depend upon its composition and its structure. These properties can be changed to a considerable extent, by changing either its composition or its structure. The structure of steel can be changed by heating it to a particular temperature, and then, allowing it to cool at a definite rate. The process of changing the structure and thus changing the properties of steel, by heating and cooling, is called 'heat treatment of steel'.

Types of structure of steel (Fig 1)

Fig 1



The structure of steel becomes visible when a piece of the metal is broken. The exact grain size and structure can be seen through a microscope. Steel is classified according to its structure.

Steel is an alloy **of** iron and carbon. But the carbon content in steel does not exceed 1.7%.

Ferrite

Pig iron or steel with 0.08% carbon is FERRITE which is relatively soft and ductile but comparatively weak.

Cementite

When carbon exists in steel as a chemical compound of iron and carbon it is called 'iron carbide' or CEMENTITE.This alloy is very hard and brittle but it is not strong.

Eutectoid/Pearlite steel

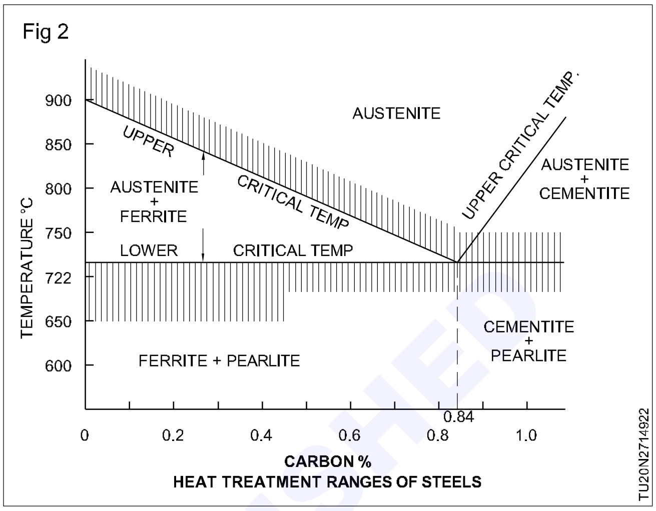
A 0.84% carbon steel or eutectoid steel is known as PEARLITE steel. This is much stronger than ferrite or cementite.

Hypereutectoid steel

More than 0.84% carbon steel or hypereutectoid steel is pearlite and cementite.

Structure of steel when heated (Fig 2)

Fig 2



HEAT TREATMENT RANGES OF STEELS

If steel is heated, a change in its structure commences from 723° C. The new structure formed is called 'AUSTENITE'. Austenite is non-magnetic. If the hot steel is cooled slowly, the old structure is retained and it will have fine grains which makes it easily machinable.

If the hot steel is cooled rapidly the austenite changes into a new structure called "MARTENSITE'. This structure is very fine grained, very hard and magnetic. It is extremely wear-resistant and can cut other metals.

Heat treatment processes and purpose

Because steel undergoes changes in structure on heating and cooling, its properties may be greatly altered by suitable heat treatment.

The following are the various heat treatments and their purposes.

Hardening	To add cutting ability To increase wear resistance
Tempering:	To remove extreme brittleness caused by hardening to an extent. To induce toughness and shock resistance
Annealing:	To relieve strain and stress To eliminate strain/hardness To improve machinability To soften the steel
Normalising:	To refine the grain structure of the steel

Hardening of carbon steel

Objectives : At the end of this lesson you shall be able to
- **state the hardening of steel**
- **state the purpose of hardening steel**
- **state the process of hardening.**

What is hardening?

Hardening is a heart-treatment process in which steel is heated to 30-50° C above the critical range, smoking time is allowed to enable the steel to obtain a uniform temperature throughout its cross-section. Then the steel is rapidly cooled through a cooling medium.

Purpose of hardening

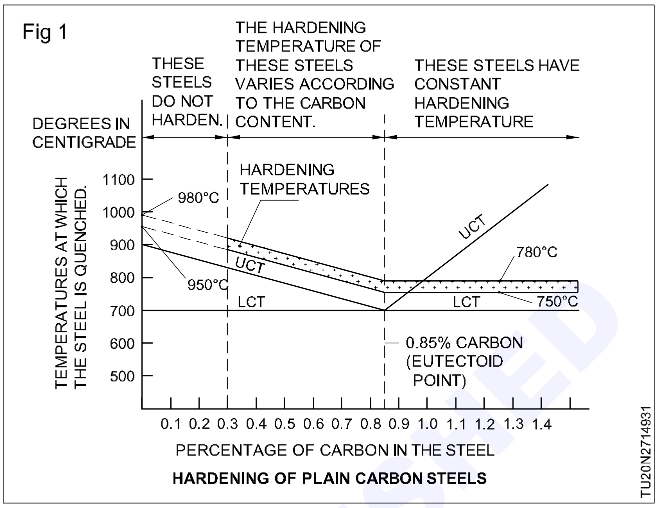
To develop high hardness and wear-resistance properties.

Hardening affects the mechanical properties of steel-like strength, toughness, ductility etc.

Hardening adds cutting ability.

Process of hardening

Steel with a carbon content above 0.4% is heated to 30-50°C above the upper critical temperature. (Fig 1) A soaking time of 5mts./10 mm thickness of steel is allowed. (Fig 1)



Then the steel is cooled rapidly in a suitable medium. Water, oil, brine or air is used as a cooling medium, depending upon the composition of the steel and the hardness required.

Tempering the hardened steel

Objectives : At the end of this lesson you shall be able to
- **state the tempering**
- **state the purpose of tempering**
- **relate the tempering colours and temperatures with the tools to be tempered**
- **state the process of tempering of steels.**

What is tempering?

Tempering is a heat-treatment process consisting of reheating the hardened steel to a temperature below 400°C, followed by cooling.

Purpose of tempering the steel

Steel in its hardened condition is generally too brittle to be used for certain functions. Therefore, it is tempered.

The aims of tempering are:

- to relieve the internal stresses
- to regulate the hardness and toughness
- to decrease the brittleness
- to restore some ductility
- to induce shock resistance

Process of tempering the steel

The tempering process consists of heating the hardened steel to the appropriate tempering temperature and soaking at this temperature, for a definite period.

The period is determined from the experience that the full effect of the tempering process can be ensured only, if the tempering period is kept sufficiently long. Table 1 shows the tempering temperature and the colour for different tools.

TABLE 1

Tools or articles	Temperature in degrees (ºC)	Colour
Turning tools	230	Pale straw
Drills and milling cutters	240	Dark straw
Taps and shear blades	250	Brown
Punches, reamers, twist drills	260	Reddish brown
Rivets,snaps	270	Brown purple
Press tools, cold chisels	280	Dark purple
Cold set for cutting steels	290	Light blue
Springs, screw drivers	300	Dark blue
	320	Very dark blue
	340	Greyish blue
For toughening without undue hardness	450-700	No colour

Annealing of steel

Objectives : At the end of this lesson you shall be able to
- **state the purpose of annealing**
- **state the process of annealing.**

The annealing process is carried out by heating the steel above the critical range, soaking it for sufficient time to allow the necessary changes to occur, and cooling at a predetermined rate, usually very slowly, within the furnace.

Purpose

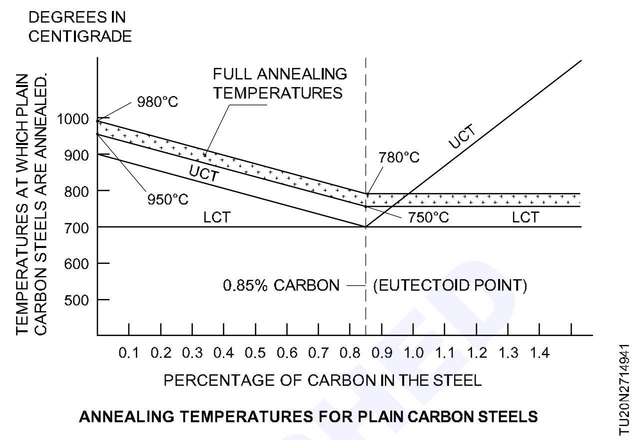
- To soften the steel
- To improve the machinability
- To increase the ductility
- To relieve the internal stresses
- To refine the grain size and to prepare the steel for subsequent heat treatment process

Annealing process

Annealing consists of heating of hypoeutectoid steels to 30 to 50°C above the upper critical temperature and 50°C above the lower critical temperature for hypereutectoid steels. (Fig 1)

Soaking is holding at the heating temperature for 5 minutes/ 10mm of thickness for carbon steels.

Fig 1



ANNEALING TEMPERATURES FOR PLAIN CARBON STEELS

The cooling rate for carbon steel is 100 to 150°C/hr

Steel, heated for annealing, is either cooled in the furnace itselfby switching off the furnance or its covered with dry sand, dry lime or dry ash.

Normalising of steel

Objectives : At the end of this lesson you shall be able to
- **state the meaning of normalising steel and its purpose**
- **state the process of normalising steel**
- **state the precaution to be taken while normalising steel.**

The process of removing the internal defects or to refine the structure of steel component is called normalising.

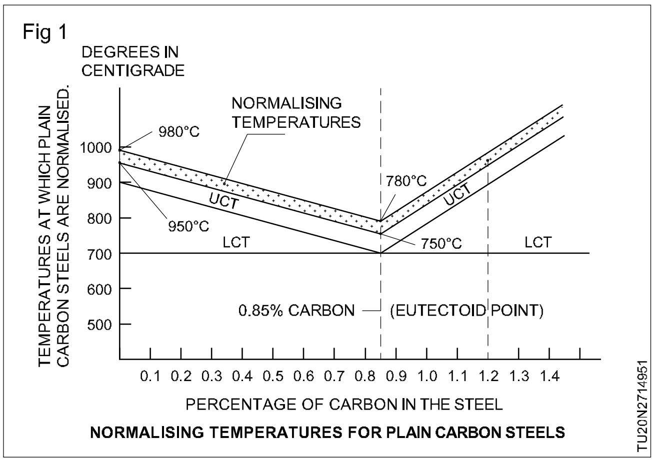
Purpose

- To produce fine grain size in the metal
- To remove stresses and strains formed in the internal structure due to repeated heating and uneven cooling or hammering
- To reduce ductility
- To prevent warping

Process

To get the best results from normalising, the parts should be heated uniformly to a temperature of 30 to 40°C above the upper critical temperature (Fig1), followed by cooling in still air, free from drought, to room temperature. Normalizing should be done in all forgings, castings and work-hardened pieces.

Fig 1



NORMALISING TEMPERATURES FOR PLAIN CARBON STEELS

Precautions

Avoid placing the component in a wet place or wet air, thereby restricting the natural circulation of air around the component. Avoid placing the component on a surface that will chill it.

Heating / Quenching steel for heat treatment

Objectives : At the end of this lesson you shall be able to
- **differentiate between the lower critical and the upper critical temperatures**
- **state the three stages in the heat treatment process**
- **determine the upper critical temperature for different carbon steels from the diagram.**

Critical Temperatures

Lower critical temperature

The temperature, at which the change of structure to austenite starts-723°C, is called the lower critical temperature for all plain carbon steels.

Upper critical temperature

The temperature at which the structure of steel completely changes to AUSTENITE is called the upper critical temperature. This varies depending on the percentage of carbon in the steel. (Fig 1)

Example

0.57% and 1.15% carbon steel: In these cases the lower critical temperature is 723°C and the upper critical temperature is 800°C.

For 0.84% carbon steel, both LCT and UCT are 723°C. This steels is called eutectoid steel.

Three stages of heat treatment

- Heating
- Soaking
- Quenching

When the steel on being heated reaches the required temperature, it is held in the same temperature for a period of time. This allows the heating to take place throughout the section uniformly. This process is called soaking.

Soaking time

This depends upon the cross-section of the steel, its chemical composition, the volume of the change in the furnace and the arrangement of the charger in the furnace. A good general guide for soaking time in normal conditions is five minutes per 10mm of thickness for carbon and low alloy steels, and 10 minutes per 10 mm of thickness for high alloy steels.

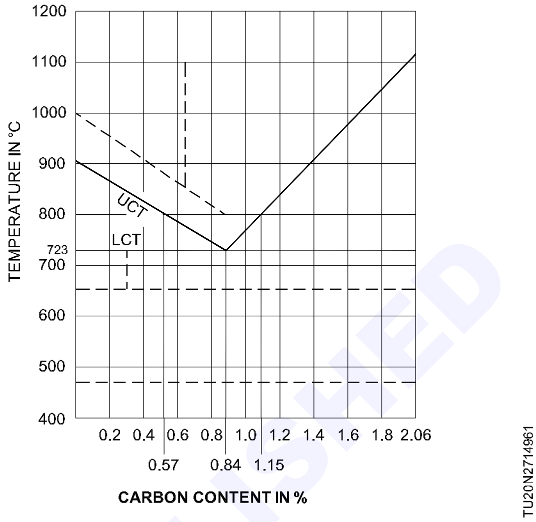
Heating steel

This depends on the selection of the furnace, the fuel used for heating, the time interval and the regulation in bringing the part up to the required temperature. The heating rate and the heating time also depend on the composition of the steel, its structure, the shape and size of the part to be heat-treated etc.

Preheating

Steel should be preheated at low temperature up to 600°C as slowly as possible.

Fig 1



Quenching

Depending on the severity of the cooling required, different quenching media are used.

The most widely used quenching media are:

- brine solution
- water
- oil
- air

Brine solution gives a faster rate of cooling while air cooling has the slowest rate of cooling.

Brine solution (Sodium chloride) gives severe quenching because it has a higher boiling point than pure water, and the salt content removes the scales formed on the metal surfaces due to heating. This provides a better contact with the quenching medium and the metal being heat-treated.

Water is very commonly used for plain carbon steels. While using water as a quenching medium, the work should be agitated. This can increase the rate of cooling.

The quenching oil used should be of a low viscosity. Ordinary lubricating oils should not be used for this purpose. Special quenching oils, which can give rapid and uniform cooling with less fuming and reduced fire risks, are commercially available. Oil is widely used for alloy steels where the cooling rate is slower than plain carbon steels.

Cold air is used for hardening some special alloy steels.

Induction hardening

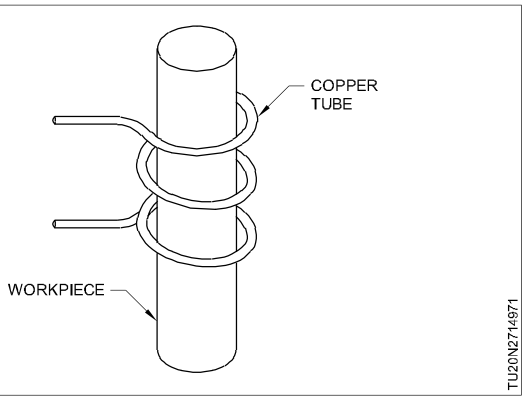
Objectives : At the end of this lesson you shall be able to

- **state the process of the induction hardening method**
- **state the advantages of the induction hardening process.**

Induction hardening

This is a CG & M method of surface-hardening in which the part to be surface-hardened is placed within an induction coil through which a high frequency current is passed. (Fig 1) The depth of penetration of the heating becomes less, as the frequency increases.

Fig 1



The depth of hardening for high frequency current to 1.0mm. The depth of hardening for medium frequency current is 1.5 to 2.0 mm. Special steels and unalloyed steels with a carbon content of 0.35 to 0.7% are used.

After induction-hardening of the workpieces, stress relieving is necessary.

The following are the advantages of this type of hardening

- The depth of hardening, distribution in width and the temperature are easily controllable.
- The time required and distortion due to hardening are very small.
- The surface remains free from scales.
- This type of hardening can easily be incorporated in mass CG & M.

Carburising of steel

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

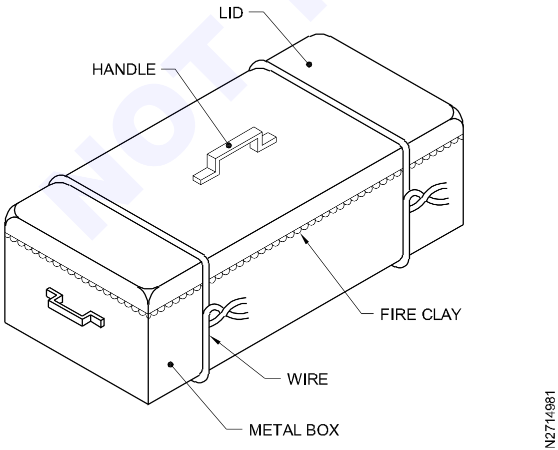
- **state the process of pack carburising**
- **state the process of liquid carbursing**
- **state the process of gas carburising.**

Pack carburising (Fig 1)

The parts are packed in a suitable metal box in which they are surrounded by the carburising medium, such as wood, bone, leather or charcoal, with barium carbonate as an energiser.

The lid is fitted to the box and sealed with fireclay and tied with a piece of wire so that no carbon gas can escape and no air can enter the box to cause de-carburisation.

Fig 1



PACK CARBURISING

Liquid carburising

Carburising can be done in a heated salt bath. (Sodium carbonatte,sodium cyanide and barium chloride are typical carburising salts). For a constant time and temperature of carburising, the depth of the case depends on the cyanide content.

This is suitable for a thin case, about 0.25 mm deep. Its advantage is that heating is rapid and distortion is minimised, and it is suitable for batch CG & M.

Gas carburising

The work is placed in a gas-tight container which can be heated in a suitable furnace, or the furnace itself may be the container.

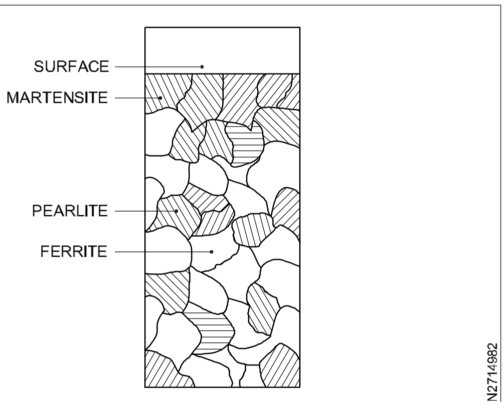
The carburising gas 'methane or propane's is admitted to the container, and the exit gas is vented.

Fig 2 illustrates the appearance of the structure across its section produced by carburising.

Heat treatment

After carburising has been done, the case will contain about 0.9% carbon, and the core will still contain 0.15% carbon. There will be a gradual transition carbon content between the case and the core. (Fig 2)

Fig 2



THE MICROSTRUCTURE OF CARBURISED STEEL

Owing to the prolonged heating, the core will be coarse and in order to produce a reasonable toughness, it must be refined.

To refine the core, the carburised steel is reheat about 870° C and held at that temperature long enough to produce a uniformitly of structure, and is then rapidly to prevent grain growth during cooling.

The temperature of this heating is much higher the suitable for the case, therefore, an extremly brittle martensite will be produced.

The case and the outer layers of the core must be refined.

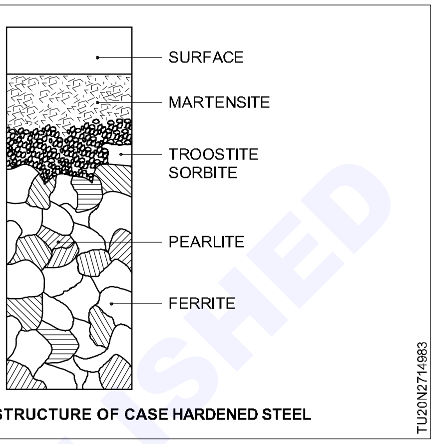
The refining is done by reheating the steel about 760º to suit the case, and quenching it.

Tempering

Finally the case is tempered at about 200° C to relive quenching stresses.

Fig 3 illustrates the apperance of the structure across its section produced by case hardening.

Fig 3



THE MICROSTRUCTURE OF CASE HARDENED STEEL

Nitriding

Objectives: At the end of this lesson you shall be able to
- **state the process of case hardening by gas nitriding**
- **state the process of case hardening by nitriding in a salt bath.**

In the nitriding process, the surface is enriched not with carbon, but with nitrogen. There are two systems in common use, gas nitriding and salt bath nitriding.

Gas nitriding

The gas nitriding process consists of heating the parts at 500°C in a constant circualtion of ammonia gas for up to 100 hours and cooling them in air.

Nitriding in salt bath

Special nitriding baths are used for salt-bath nitriding. This process is suitable for all alloyed and unalloyed types of steel, annealed or not annealed, and also for cast iron.

Process

The completely stress-relieved workpieces are preheated (about 400°C) before being put in the salt bath (about 520-570°C). A layer 0.01to 0.02mm thick is formed on the surface which consists of a carbon and nitrogen compound. The duration of nitriding depends on the cross-section of the workpieces (half an hour to three hours). It is much shorter than for gas nitriding. After being taken out of the bath, the workpieces are quenched and washed water and dried.

Advantages

The parts can be finish-machined before nitriding because no quenching is done after nitriding, and, therefore, they will not suffer from quenching distortion.

In this process, the parts are not heated above the critical temperature, and, hence warping or distortion does not occur.

The hardness and wear-resistance are exceptional. There is a slight improvement in corrosion resistance as well.

Since the alloy steels, used are inherently strong when properly heat-treated, remarkable combinations of strength and wear-resistance are obtained.

Interchangeability meaning, procedure for adoption,quality control procedure for quality production

Objectives : At the end of this lesson you shall be able to
- **state the advantages and disadvantages of mass CG & M**
- **outline the meaning of the term, interchangeability**
- **state the necessity for the limit system.**

Mass CG & M

Mass CG & M means CG & M of a unit, component or part in large numbers.

Advantages of mass CG & M

Time for the manufacture of components is reduced

The cost of a piece is reduced

Spare parts can be made available quickly

Gauges are used to check the components

Even unskilled workers can be employed for checking

Measuring time is saved

Disadvantages of mass CG & M

Special purpose machines are necessary

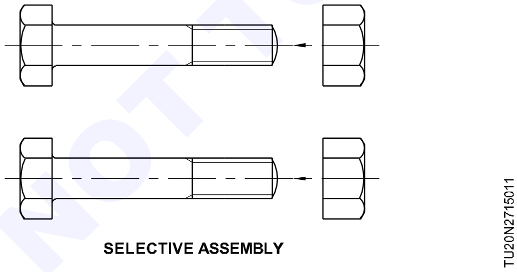
Jigs and fixtures are needed

Gauges are to be used, hence the initial expenditure will be high

Selective assembly

Figures 1&2 illustrate the difference between a selective assembly and a non-selective assembly. It will be seen in Fig 1 that each nut fits only one bolt. Such an assembly is slow and costly, and maintenance is difficult because spares must be individually manufactured. spares must be individually manufactured.

Fig 1

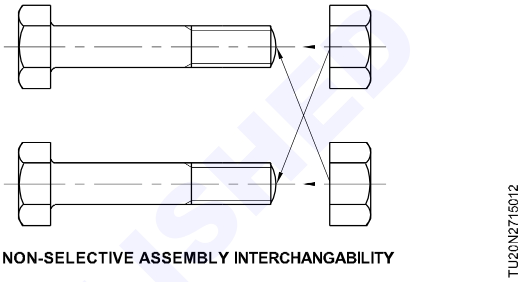


SELECTIVE ASSEMBLY

Non-selective assembly

Any nut fits any bolt of the same size and thread type. Such an assembly is rapid, and costs are reduced. Maintenance is simpler because spares are easily available. (Fig 2)

Fig 2



NON-SELECTIVE ASSEMBLY INTERCHANGABILITY

Non-selective assembly provides interchangeability between the components.

In modern engineering CG & M, i.e. mass CG & M, there is no room for selective assembly. However, insome special circumstances, selective assembly is still justified.

Interchangeability

When components are mass-produced, unless they are interchangeable, the purpose of mass CG & M is not fulfilled. By interchangeability, we mean that identical components, manufactured by different personnel under different environments, can be assembled and replaced without any rectification during the assembly stage and without affecting the functioning of the component when assembled.

Necessity of the limit system

If components are to be interchangeable, they need to be manufactured to the same identical size which is not possible, when they are mass-produced. Hence, it becomes necessary to permit the operator to deviate by a small margin from the exact size which he is not able to maintain for all the components. At the same time, the deviated size should not affect the quality of the assembly. This sort of dimensioning is known as limit dimensioning.

A system of limits is to be followed as a standard for the limit dimensioning of components.

Quality control

Objectives : At the end of this lesson you shall be able to
- **state the features of quality control**
- **state the meaning of quality control**
- **sate the different factors of quality control**
- **state the recent trends in quality control**
- **state the different levels of inspection.**

Quality control

The quality of a product is generally defined as its usefulness towards the purpose for which it is manufactured. In other words 'Fitness for use'.

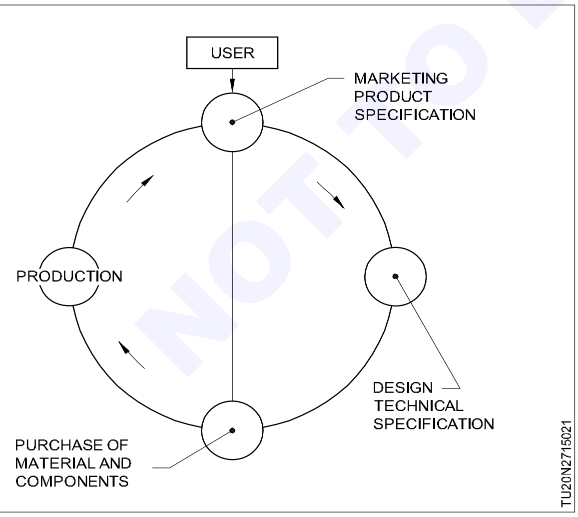
In olden days, the manufacture of a product was in batches ie., in smaller quantities, not exceeding say 10 or 100. In this case, the components were made to fit each other through an inspection process, where the good ones were segregated from the bad.

With the advent of mass CG & M, this method of inspection caused a lot of rejections and the manufacturers started to use preventive inspection where in measures were taken to reduce the rejection to almost zero.

Thus, one started hearing about the concept of `zero defects'. The aim of the inspection is to prevent rejection. This new approach towards producing products fit for use, without the rejection losses and re–work called for the imposition of controls at different stages of the manufacture of a component. These controls ensured the required quality, and thus quality control became an important activity in the industries.

It is thoroughly understood today that quality starts from the customer and ends with the customer. Thus, the customer has become a very important person in the minds of the manufacturers of consumables and other products. So the quality of a product has to be built into the product at the following stages. (Fig 1)

Fig 1



Product specification

The details regarding the product in question are collected from the probable consumers (users) by the staff of the marketing and sales departments.

These details should truly reflect the requirements and aspirations of the users. Thus, the conformance of the real use with the product specification is the first step of quality control.

Technical specification

The product specifications are scrutinized by various experts from the CG & M and product service departments. Based on the recommendations of these experts the technical specifications are prepared to very minute details.

The correspondence between the product specification and the technical specification is the second quality step.

Manufacturing the product, including packing and forwarding to the customer

For the manufacture of the product the right type of raw material, machinery and manpower should be involved so that the end product conforms to the technical specifications. Moreover, the product thus produced should reach the consumer without any damage.

This forms the third quality step.

Thus, quality control is the culmination of all these activities required for defining, quantifying, effecting and measuring the quality of the product. The main objective of quality control is to achieve fitness for use at the lowest cost.

This calls for an understanding of the following points.

- There is always an optimum quality level.
- All personnel in an enterprise are responsible for the quality of a product.
- The quality must be controlled and built in at the various stages planned. This requires well organised cooperation between all the departments in the enterprise.
- There is one quality level that strikes the optimum level between the utility and cost of product, as shown in Fig 2.

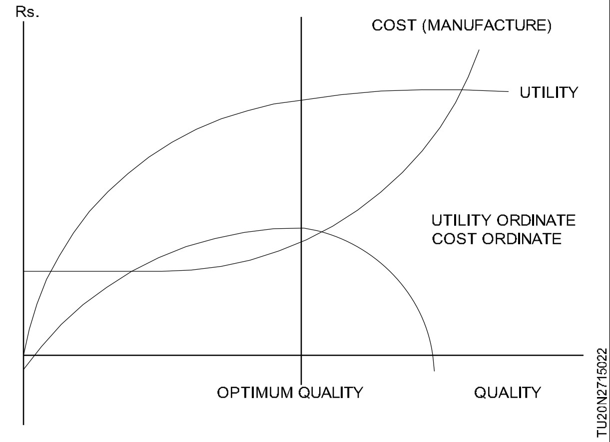
The two parameters, utility and cost in the graph, conform to the following basic laws.

Utility increases with increase in quality but the trend is decreasing beyond the optimum quality level. This is the law of the decreasing marginal utility.

The cost does also increase with increasing quality. The increase is progressive.

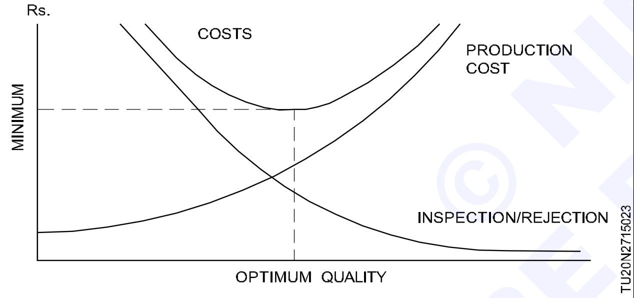
This is the law of increasing marginal cost.

Fig 2



When the design is finalised and CG & M starts, the cost of the product quality can be illustrated as in the graph. (Fig.3) A well controlled and efficient CG & M requires less inspection effort and vice versa. It is the total cost of the CG & M and the inspection/rejection that has to be kept at a minimum. It is hence important to avoid sub-optimising ie., keeping one or the other parameter at a minimum. It is hence important to avoid this. From the graph it is also evident that too high or too low a quality is uneconomical.

Fig 3



The optimum quality should be estimated from the total cost.

Total costs cannot be precisely computed, due to practical limitations in estimating and accounting, but it is obvious that the cost should include more than that of the quality and inspection function. In fact, it should include all the functions.

If all the functions are influencing the optimum quality, they are also responsible for the quality.

The old concept of quality control was that the inspection or quality department was responsible for the quality. This concept still persists to a considerable extent.

This concept is wrong and is eliminating from the secondary duty of inspection. Namely, to sort out faulty component or products and ensure an economical quality level after the inspection. This is however a postmortem activity and the primary duty of the inspection is to help other functions to prevent mistakes.

This is done by monitoring the quality in the different steps and at different stages, and feed back this information to the concerned functions.

This monitoring should be based on costs, and penetrate down to processes and operations so that detailed information on quality costs are obtained.

Quality costs are mostly classified as follows.

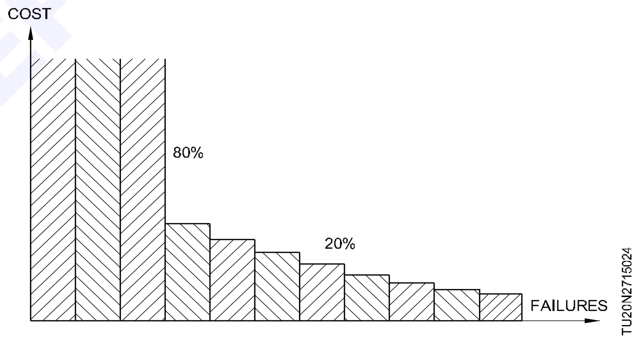
Quality costs

Inspection costs		**Rejection costs**	
Labour, equipment and overhead for		Internal	External
Preventive Inspection	Segregation	Rejections Re-work Delay Consequential costs, etc.	Guarantee Service Distance goodwill loss, etc.

Quality costs are often quite substantial and surveys of the European and American markets have found that aspect.

The quality costs are collected and presented to the concerned parties. This is the ground for improvement and control. (Fig 4)

Fig 4



It must be possible to apply control of quality in all phases of market use, design and CG & M and a good quality cost reporting system will indicate where the costs are higher.

We may find in the near future that quality control has given way to the new concept namely Quality Engineering wherein, the emphasis would be on the usefulness of a product, not only for the individuals but also for the entire humanity.

The occurance of non–florification trading on nuclear weapons is an apt example of such an understanding with regard to the quality of a product.

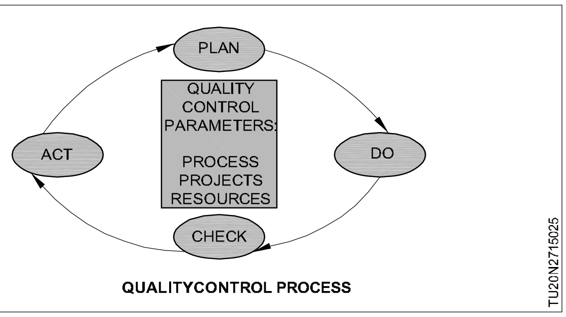
Quality control

What is quality control?

Quality control is a set of methods used by organizations to achieve quality parameters or quality goals and continually improve the organization 's ability to ensure that a software product will meet quality goals.

Quality control process (Fig 5)

Fig 5



QUALITYCONTROL PROCESS

The three class parameters that control software quality are:

- Products
- Processes
- Resources

The total quality control process consists of :

- Plan - It is the stage where the quality control processes are planned.
- Do- Use a defined parameter to develop the quality.
- Check- Stage to verify of the quality of the parameters are met.
- Act- Take corrective action if needed and repeat the work.

Quality control characteristics

- Process adopted to deliver a quality product to the clients at best cost.
- Goal is to learn from other organizations so that quality would be better each time.
- To avoid making errors by proper planning and execution with correct review process.

Quality control basic concepts

In this article I explain three fundamental concepts that every buyer should be familiar with when it comes to quality inspections:

1 Inspection levels

2 The AQL (Acceptance quality limit)

3 When to inspect?

After 10 minutes, you will be able to understand the reliability of an inspector's findings and take more informed decisions based on an inspection report.

If you have not started doing professional quality control, you will need to understand these 3 concepts to make sure the inspection plan meets your needs.

1 Inspection levels

Why use random sampling?

Shipments often represent thousands of products. Checking 100% of the quality would be long and expensive. A solution is to select samples at random and inspect them, instead of checking the whole lot.

But how many samples to select? On the one hand. Checking only a few pieces might prevent the inspector from noticing quality issues; on the other hand, the objective is to keep the inspection short by reducing the number of samples to check.

The relevant standards purpose a standard severity, called "normal level", which is designed to balance these two imperatives in the most efficient manner.

Within this normal severity, there are three general levels: I, II, and III. Level II is used for more than 90% of inspections. For example, for an order of 8.000 products, only 200 samples are checked.

Military Standard 105 was created by the US Department of Defence to control their procurements more efficiently. In 1994 they decided to relay on non-governmental organizations to maintain this type of standard. The ANSI, ISO, and other institutes have created their own standard, but in essence they are similar. The major third -party QC firms use the same standards and the same statistical tables.

When to adopt a different level

Suppose you source a product from a factory that often ships substandard quality. You know that the risk is higher than average. How to increase the Discriminating power of the inspection? You can opt for level III, and more samples will be checked.

Similarly, if a supplier has consistently delivered acceptable products in the past and keeps its organization unchanged, you can choose level I. As fewer samples have to be checked, the inspection might take less time and be cheaper.

The relevant standards give no indication about when to switch inspection levels, so most importers rely on their "gut feeling".

The "special levels"

Inspectors frequently have to perform some special tests on the products they are checking. In some cases the tests can only be performed on very few samples, for two reasons:

1 They might take a long time (e.g. doing a full function test as per claims on the retail box).

2 They end up in product destruction. (E.g. unstitching a jacket to check the lining fabric).

For these situations only, the inspector can choose a "special level".

So we have three "general" inspection levels, and four "special levels". For a given order quantity, each level gives a different number of samples to check. Let's see how it plays in two examples.

2 The AQL (acceptance quality limit)

In part 1, we explained the different inspection levels that can be used. Another basic concept rings familiar to many importers, but is often not clearly understood: the AQL (Acceptance Quality Limit).

There is no such thing as zero defect

First, as a buyer, you have to know what proportion of defects is tolerated on your market. If you are in the aviation business, any defective part might cause a disaster, so your tolerance will be very, very low. But you will have to accept a higher percentage of defects if you source consumer products that are assembled by hand in china or in India.

An objective limit is necessary

So, how many defects are too many? It is up to you, as a buyer, to make this decision. There are two reasons why you should not leave this to the inspector's judgment:

1 When it comes to giving instruction to an inspector, you should never leave gray areas-as they might open the door to corruption.

2 Your supplier should have clear criteria for acceptability, or they will see rejections as unfair.

The AQL is the proportion of defects allowed by the buyer. It should be communicated to the supplier in advance.

The three categories of defects

Some defects are much worse than others. Three categories are typically distinguished:

- Critical defects might harm a user or cause a whole shipment to be blocked by the customs.
- Major defects are not accepted by most consumers, who decide not to buy the product.
- Minor defects also represent a departure from specifications, but some consumers would still buy the product.

For most consumer products, critical defects are not allowed, and the AQL for major defects and minor defects are 2.5% and 4.0% respectively.

Some important remarks

- A professional inspector will notice defects and evaluate their category by himself. But it is better if the buyer himself describes the most frequent defects and assigns categories to each one.
- Defects can be on the product itself, on the labeling or on the packaging.
- If one sample presents several defects, only the most severe one is counted.

How to read the AQL tables

The master tables included in the relevant standards are commonly called AQL tables. Let's take an example.

You buy 8.000 widgets from a factory, and you choose inspection level II. In the table below (which is only valid for single sampling plans), you see that the corresponding letter is L.

Now let's turn to the next table (which is only appropriate for normal-severity inspections). The letter L gives you the number of samples to draw at random: 200 pcs.

And what about the AQL? Let's say you follow the usual practice of tolerating 0% of critical defects, 2.5% of major defects, and 4.0% of minor defects. The maximum acceptable number of defects is 10 major and 14 minor. In other words, the inspection is failed if you find at least 1 critical defect and /or at least 11 major defects and /or at least 15 minor defects.

3 When to inspect?

The first two parts focused on the different inspection levels and on the AQL tables. So you Know how to set the number of samples to check and how many defects have to be accepted. With these setting and your detailed product specifications, a QC inspector can check your products and reach a conclusion (passed or failed).

But importers face one more question when should the products be inspected? This is an extremely important issue for buyers willing to secure their supply chain. Spending a few hundreds of dollars to check and fix issues early can be an excellent investment; if might save you weeks of delay, shipments by air, and /or lower quality products that you have to accept and deliver to your own customers.

Four types of inspections

Let's picture the simplified model where one factory turns raw materials into finished products. (If you also have to manage the quality of sub = suppliers' products, the same model can be applied to them)

Pre-CG & M inspection

This type of inspection is necessary if you want to check the raw materials or components that will be used in CG & M. Buying cheaper materials can increase a factory's margin considerably, so you should keep an eye on this risk. It can also be used to monitor the processes followed by the operators.

During CG & M inspection

This inspection allows you to get a good idea of average product quality, and to ask for corrections if problems are found. It can take place as soon as the first finished products get off the line, but these samples might not be representative of the whole. So usually an inspection during CG & M is done after 10-30% of the products are finished.

Final (pre-shipment) inspection

Inspecting the goods after they are made and packed is the standard QC solution of most importers. The inspector can really check every detail, including counting the total quantity and confirming the packaging. Final inspections are usually performed in a hurry, just before shipment. To avoid creating delays, inspectors can usually start after all products are finished and 80% + of the shipment quantity is packed.

Loading supervision

In some cases, a buyer wants to make sure the factory ships the right products, in the right quantity, and with the right loading plan.

Records maintained in shop floor by a worker

Job card: Each and every job is accompanied by job card in a industry. The job card is very important document to be maintained by an operator. It include the following details.

1 Document No
2 Date
3 Customer Name
4 Work order No
5 Operator Name
6 Time Record
7 Operation Name / Number
8 Quantity etc

Inspection record

The record is used for inspecting the workpiece by the operator or quality inspector.

1 Date of Inspection
2 Name of the component
3 Inspected by (Operator/ Inspector)
4 Quantity accepted/Rejected
5 Rework details if any

Accepted / Rejected / Rework tag : After inspecting the work piece, accepted tag is enclosed with sign and date if the work pieced are accepted and rejected tag is enclosed is the pieces are rejected. If there is any rework, rework tag is enclosed. The rework tag has the details of rework to be done.

INSPECTION RECORD

DATE	BY	ACCEPTED	REJECTED

ACCEPTED

Customer ____________________
W.O No ____________________
No PCB ____________________
Part No. ____________________
Inspector ____________________
Commented ____________________

INSPECTED

READY TO USE

SIGNED BY ____________________
DATE ____________________

Importance of Technical term used in Industry

Objectives : At the end of this lesson you shall be able to
- **state the meaning of different terms used in industry**
- **calculate the machine time in turning.**

Engineering terminology

Broach (v) to finish the inside of a hole to a shape other than round, as in a keyway (n) The tool for the process, which has serrated edges and is pushed or pulled through the hole to produce the required shape.

Bushing (n) a smooth walled bearing (AKA a plain bearing). Also a tool guide in a jig or fixture.

Cam (n) A mechanical device consisting of an eccentric or multiply curved wheel mounted on a rotating shaft, used to produce variable or reciprocating motion in another engaged for contacted part (Cam follower). Also Camshaft

Casting (n) any object made by pouring molten metal into a mold.

Chamfer (n) a flat surface made by cutting off the edge or corner of a object (bevel) (v) the process of creating a chamfer.

Clevis (n) A U-shaped piece with holes into which a link is inserted and through which a pin or bolt is run. It is used as a fastening device which allows rotational motion.

Collar (n) A Cylindrical feature on a part fitted on a shaft used to prevent sliding (axial) movement.

Collet (n) a cone-shaped sleeve used for holding circular or rod like pieces in a lathe or other machine.

Core (v) to form the hollow part of a casting, using a solid form placed in the mould (n) the solid form used in the coring process, often made of wood, sand, or metal. Mould with core is final cast manifold.

Counter bore (n) a cylindrical flat-bottomed hole, which enlarges the diameter of an existing pilot hole. (v) The process used to create that feature.

Countersink (n) a conical depression added to an existing hole to accommodate and the conic head of a fastener recessing it below the surface of a face. (v) The process used to create that feature.

Coupling (n) A device used to connect two shafts together at their ends for the purpose of transmitting power. May be used to account for minor misalignment or for mitigating shock loads.

Die (n) one of a pair of hardened metal plates or impressing or forming desired shape. Also, a tool for cutting external threads.

Face (v) to machine a flat surface perpendicular to the axis of rotation of a piece.

Fillet (n) A rounded surface filling the internal angle between two intersection surfaces. Also Rounds

Fit (n) The class of contact between two machined surfaces, based upon their respective specified size tolerances (Clearances, transitional, interference)

Fixture (n) A device used to hold a work piece while manufacturing operations are performed upon that work piece.

Flange (See bushing example) (n) a projecting rim or edge for fastening, stiffening or positioning.

Gauge (n) A device used for determining the accuracy of specified manufactured parts by direct comparison.

Gear Hobbing (v) A special form of manufacturing that cuts gear tooth geometrices. It is the major industrial process for cutting involute form spur gears.

Idler (n) A mechanism used to regulate the tension in belt or chain. Or, a gear used between a driver and follower gear to maintain the direction of rotation.

Jig (n) A special device used to guide a cutting tool (drill jig) or to hold material in the correct position for cutting or fitting together (as in welding or brazing)

Journal (n) The part of a shaft that rotates within a bearing key (woodruff key shown) (n) A small block or wedge inserted between a shaft and hub to prevent circumferential movement.

Keyseat (n) A slot or groove cut in a shaft to fit a key. A key rests in a keyseat.

Keyway (n) A slot cut into a hub to fit a key. A key slides in a keyway.

Knurl (v) To roughen a turned surface, as in a handle or a knob.

Pinion (n) A plain gear, often the smallest gear in a gearset, often the driving gear, May be used in conjuction with a gear rack (rack and pinion, see below)

Planetary Gears (n) A gear set characterized by one or more planet gear (s) rotating around a sun gear. Epicyclic gearing systems include an outer ring gear (known as an planetary system)

Rack (w/pinion gear) (n)A toothed bar acting on (or acted upon) by a gear (pinion)

Ratchet (n) A space mechanical device used to permit motion in one direction only.

shims (n) a thin strip of metal inserted between two surfaces to adjust for fit. (v) The process of inserted shims

Spline (n) A cylindrical pattern of keyways. May be external (L) or Internal (R)

Tap (v) To cut internal machine threads in a hole, (n) the tool used to create that feature.

Undercut (n) A cut having inward sloping sides, (v) to cut leaving an overhanging edge.

Definitions of technical terms- CNC SYSTEM

Absolute co - ordinates: The distance (or dimensions of the current position from the origin (or zero point) of a coordinate system (or measuring system) measured parallel to each axis of the system.

Address: A name (or label, or number) identifying a storage area in a control system or computer memory.

Analog: The use of physical quantity (like voltage) whose amplitude represents that of another quantity (like distance)

APT: Automatically programmed tools.

ASCII: American standard code for information interchange.

Auxiliary function: Another name for miscellaneous function.

Axis: When associated with machine tools, an axis is a direction in which a machine tool table or head can move.

Backlash: The maximum movement at one end of a mechanical system (such as geartrain) which does not cause the other end to move.

Base number: A base number (or radix) is an implied number used when expressing a value numerically in the normal decimal system, the base is 10 and 87 is a short way to representing $8x10^1+7x10^o=87$ in binary notation 1011 represents $1x2^3+0x2^2+1x2^1+1x2^0$ = 8+2+1 = 11 in decimals

Binary coded decimal number (BCD): The representation of a number by groups of four binary digits for each decimal digit in a number.

Binary digits: The characters 0 or 1 used in the binary system to express any value (see Base number)

Bit: A binary digit or its representation.

Block: A collection of words in some agreed form. Usually on a control tape, and separated from succeeding blocks by an End of -Block code.

BIS - British Standards institution.

Buffer: A temporary storage area where information is held until it is moved into an operating area.

CCW: Counter clock wise.

Characters: The set of letters, decimal digits, signs (such as + - :% etc)

Circular interpolation: A contour control system with circular interpolation cuts an arc of a circle from one block on a control tape.

Co -ordinate system: A series of intersecting planes, or planes and cylinders (usually three) which form a reference system.

CPU: Central processing unit. The controlling unit in a digital computer.

Cutter diameter compensation: Provision in the control system to modify the cutter offset by entering a numerical correction to the cutting tool diameter.

Cutter offset: Position of reference point on the tool.

Cutting speed: The velocity of the cutting edge of the tool relative to the work piece.

Cycle: A sequence of operations which frequently repeated.

Depth of cut: The amount of metal (in mm or inch) removed perpendicularly to the direction of feed in one pass of the cutter over the work piece.

Digit: a character used in a numbering system.

Downtime: Time during which equipment is out of action because of faults.

Dwell: A pause of programmed duration, usually to ensure that a cutting action has time to be completed.

End-of-block mode: An agreed code which indicates the completion of a block of input information on punched tape.

Feed: The movement of a cutting tool into a work piece.

Feed rate: The rate, in mm/min or in/min, at which the cutting tool is advanced into the work piece.

Format: An agreed order in which the various types of words will appear within a block,

Hardware: Equipment e.g., Machine tool, or Control, or computer.

Incremental co-ordinates: The distance of current position from the preceding position, measured in terms of axial movements in the co-ordinate system.

Interpolation: The process of supplying the positions of a set of more closely spaced points between more widely spaced points such as change points.

IPM: Inches per minute (in/min).

IPR: Inches per revolution (in/rev) of a cutter or work piece.

ISO: International Organization for Standardization.

Machine language or machine instruction: The instructions and data for computers are based on patterns of bits.

Machine zero: The machine zero point is at the origin of the coordinate measuring system of the machine.

Manual data input (MDI): A means, on the control panel, of inserting numerical information into the control system.

Manuscript: Handwritten program.

Miscellaneous function: A control tape term for codes such as M03 used to control machine tool functions such as 'rotate spindle clockwise'.

NC: Numerical control.

Preparatory function word: A word near the beginning of a control tape block which calls for a change in mode.

Preset tools: The setting of tools in special holders away from the machine tool.

Program: A systematic arrangement of instructions or information to suit a piece of equipment.

RPM: Revolutions per minute (rev/min), a measure of spindle speed.

Sequence number: The number allocated to a block or group of blocks to identify them. Commonly takes a form like N 278.

Servomechanism: A closed-loop positioning-control system.

Software: Programs, sequences of instructions, etc. Not equipment (see Hardware).

Spindle speed: the rotational speed in RPM of the spindle or shaft which supplies the cutting power.

Subroutine: A sequence of computer-programming statements or instructions which perform an operation frequently required.

Word: An agreed arrangement of characters and digits (usually less than 10) which conveys one instruction, piece of information, or idea.

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.**

Work organisation

Work organisation is to arrange and distribute the work between the work teams in such a way that the best use is made of the available labour, materials, tools and equipment.

- to order the operations and activities of the work should follow each other.
- to decide the various work teams.
- to decide the various work teams.
- to instruct and communicate correctly in order to avoid misunderstandings.

Organisation of work

The organisation of work includes many aspects, such as

- Pace of work (speed of an assembly line, quotas).
- Work load.
- Number of people performing a job (staffing levels).
- Hours and days on the job.
- Length and number of rest breaks and days away from work.
- Layout of the work.
- Skill mix of those workers on the job.
- Assignment of tasks and responsibilities and
- Training for the tasks being performed.

Some common terms technically used in industry are as follows:

- Lean CG & M
- Continuous improvement
- Just-in-time CG & M
- Work teams
- Total productive maintenance
- Total quality management
- Outsourcing/ contracting out

Lean CG & M

An overall approach to work organisation that focuses on elimination of any "waste" in the CG & M / service delivery process. It often includes the following elements: "continuous improvement", "just-in-time CG & M" and work teams.

Continuous improvement

A process for continually increasing productivity and efficiency, often relying on information provided by employee involvement groups or teams. Generally involves standardizing the work process and eliminating micro-breaks or any "wasted" time spent not producing/ serving.

Just-in-time CG & M

Limiting or eliminating inventories, including work-in-progress inventories, using single piece CG & M techniques often linked with efforts to eliminate "waste" in the CG & M process, including any activity that does not add value to the product.

Work teams

Work teams operate within a CG & M or service delivery process, taking responsibility for completing whole segments of work product. Another type of team meets separately from the CG & M process to "harvest" the knowledge of the workforce and generate develop and implement ideas on how to improve quality, CG & M and efficiency.

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 CG & M. The quality concept of assuming the best quality from inception to implementation throughout the CG & M process.

Outsourcing/ Contracting out

Transfer of work formerly done by employees to outside organisations. In many workplaces undergoing restructuring, worker knowledge about the productive/ service process is gathered through "employee involvement" and then used by management to "lean out" and standardize the work process, thereby reducing reliance on worker skill and creativity. This restructuring has resulted in job loss for some underperformed, while increasing the work load and work pace for those who remain on the job. The result of these changes in work organisation is that it is no longer just machines that are wearing out-it is the workers themselves.

Different types of documentation as per industrial needs

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

- **state the purpose of documentation**
- **list the different types of documentation**
- **explain the documents format - batch processing, BOM, cycle time, productivity report, manufacturing inspection report.**

Documentation

Documentation and records are used throughout the manufacturing process as well as supporting processes (quality control) must meet the basic requirements. Documentation is a set of documents provided on paper, or online, or on digital or analog media, such as audio tape or CDs. Examples are user guides, white papers, online help, quick reference guides.

The stages of recording the documents is to

- prepare, review, update and approve documents.
- identify changes and current revision status of documents.
- use of applicable documents available at points of use with the control documents of external origin
- identify and distribute relevant versions to be identifiable and remain legible.
- prevent unintended use of obsolete documents and archiving.

The different types of documentation as per industrial needs includes

- Processing charts
- Bill of materials (BOM)
- CG & M cycle time format
- Productivity reports
- Manufacturing stage inspection report
- Job cards format
- Work activity log
- Batch CG & M record format
- Estimation of work
- Maintenace log format

Process chart

A process chart is a graphical representation of the activities performed during manufacturing or servicing jobs. Graphical representation of the sequence of operations (workflow) constituting a process, from raw materials to finished product.

Process charts are used for examining the process in detail to identify areas of possible improvements.

The different types of process charts they are

- Operation process chart
- Flow process chart (man/ material/ equipment type)
- Operator chart (also called two handed process chart)
- Multiple activity chart
- Simo chart

Batch record forms

The documents used and prepared by the manufacturing department provide step-by-step instructions for CG & M-related tasks and activities, besides including areas on the batch record itself for documenting such tasks.

Batch CG & M record is prepared for each batch should include information on the CG & M and control of each batch. The batch CG & M record should confirm that it is the correct with standard operating procedure.

These records should be numbered with a unique batch or identification number and dated and signed when issued.

The batch number should be immediately recorded in data processing system. The record should include date of allocation, product identity and size of batch.

Documentation of completion of each significant step in the batch CG & M records (batch produciton and control records) should include :

- Dates and when apporiate time.
- Major equipment used machinery and specified batch numbers of raw material reprocessed materials used during manufacturing.
- Critical process parameters records.
- Trial products or sample (if reqiured).
- Signatures of staff for sequence of operation.
- Laboratory test result and line inspection notes.
- Achieved CG & M aganist target.
- Packaging and label (if any) details.

Batch processing record: (Sample format - 1)

The format 1 used in documentation of batch processing record has the description of the job, necessarily mentioned with part number and name of the part.

BATCH PROCESSING RECORD - FORMAT - 1

Batch Processing Record

Description of job	Batch no. :
Part no. :	Batch quantity :
Name of part :	Batch record no. :
	Purchase order no. :
Description of process :	

Manufacturing Organisation :

Period of manufacture (Year - Qtr): Start date of manufacture: End date of manufacture:

Number of pages according to batch: Inserted pages: Manufacturing facilities:

Total number of pages

1. Operator / Technician
Date Name and signature

2. CG & M in-charge:
Date Name and signature

3. Section manager
Date Name and signature

4. Plant in-charge:
Date Name and signature

5. CG & M in-charge:
Date Name and signature

Remarks (if any)

A predetermined batch quantity with batch number alloted and identified with batch record number is documentation. The product reference is made with purchase order number.

The CG & M process is descriptively written about the sequence of operation to be carried out on the product.

The manufacturer organization name, period of manufacture preferably the year with starting date of manufacture and end date of manufacturer and number of pages of document according to batch quantity processed, and total number of pages of document, inclusive of inserted pages and manufacturing facilities is provided with.

The batch processing record is signed with date mentioning name of person responsible and their designation. The remarks if any on the process should be also mentioned then and there.

Bill of materials (BOM) format - 2

The list of parts involved in manufacturing of an assembly arranged in order is given in this format.

The format shown is as per bureau of Indian Standards IS:11666-1985 as example for Engineering Component drawings.

The BOM in the form of tabular columns has the component marked with item number, and its name is given under description and number of is mentioned under quantity, with reference drawing ie., sub assembly/part drawing number.

The material designation as per code of practice or standards is mentioned, and any other specific notes are given under remarks column.

The BOM is placed on the manufacturing drawing containing with assembly and parts in standard sheet sizes of engineering drawing.

BILL OF MATERIAL (BOM) - FORMAT - 2

as per IS: 11666-1985

S.No	Item No.	Description	Quantity	Reference Drawing no.	Material as per standard	Remark

Cycle time

Cycle time is the total time from the beginning to the end of the process. Cycle time includes process time, during which a raw material worked with to bring it closer to required form output, and delay time, during which the workpiece waiting for next operation.

The time taken to perform one operation repeatedly measured from “Start to Start” the starting point of one product’s processing in a specified machine or operationuntil the start of another similar product’s processing in the same machine or process. Cycle time is commonly categorized into same machine/process.

Machine cycle time

The processing time of the machine working on a part.

Auto cycle time

The time a machine runs un-aided (automatically) without manual intervention.

Overall cycle time

The complete time it takes to produce a single unit. This term is generally used when speaking of a single machine or process.

Total cycle time

This includes all machines, processes, and classes of cycle time through which a product must pass to become a finished product. This is not lead time, but it does help in determining it.

CG & M cycle time (Format - 3)

This format 3 should contain mentioning the organization name department / section name. The process which is being observed for analysing the cycle time is mentioned with line in charge name and the date/time of the operations, with operator name is indicated.

The time observation on each operation, sequence noted in the column, and lowest repeatable is also mentioned for each operation. The times observation for machine cycle time is also noted, with any notes be recorded in respective operations in sequence.

CG & M CYCLE TIME - FORMAT 3

Organisation Name: **Department / Section :**			**Process:**				**Line Incharge:**				**Date/Time:**
Operator :										**Machine Cycle Time**	**Notes**
Operator Sequence	**Observed Times**								**Lowest Repeatable**		

Productivity report

Productivity report to measure and review the efficiency of a person, machine, factory, system, etc., in converting inputs into useful outputs. Productivity report is computed by dividing average output per period by the total costs incurred or resources (capital, energy, material, personnel) consumed in that period.

The base document daily CG & M report which reveals the actual output against the target plan and on investment cost incurred as mentioned above decides the cost efficiency.

Daily CG & M report (Format 4): The output of CG & M is shown in the format, referring the job order no quantity, material and size, every process involved, to produce a component, quality control, packing should contain the details of planned quantity and produced quantity is recorded in the document. This is the base details for arriving the productivity report.The incurred cost is worked out considering infrastructure, raw materials and facilities.

DAILY CG & M REPORT - FORMAT- 4

	Daily CG & M Report						**Organisation Name:**					
Date:	**Department:** **Section:**											
	Process - I		Process-II		Process-III		Process-IV		Quality Control		Packing	
	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												

Signature of section Incharge

Manufacturing stage inspection report (Format 5)

The format 5 is to monitor the CG & M in various stages for which manufacturing stage inspection conducted for documentation to review the productivity. The format gives the details of product being inspected showing the details of customer reference by purchase order (PO) number and date, job order number and date, process involved in manufacture of product, the quality submitted for inspection. The accepted and rejected quality recorded with inspection record review date and the inspection person signature who conducted the stage inspection is recorded date wise for mentioned /specified period with start and end dates.

MANUFACTURING STAGE INSPECTION REPORT - FORMAT - 5

Organisation Name :

Status: From Date .../.../..... To Date .../.../.....

Date	Product ID/ Code	Customer	P.O No. & Date	Job Order No.	J.O Date	Process	Qty	Accepted	Rejected	Inspection Record No.	Inspection conducted by

Documentations - 2

Objectives : At the end of this lesson you shall be able to
- **state the purpose of job card and its format details**
- **explain work activity log format details**
- **state the details of batch CG & M format.**

Job card

A job card is a document showing the details of a job to be performed in a CG & M shop. It is used to authorize and instruct the work team to take up the CG & M work.

Job card format - 1

Job card has the details of commencing the job, customer name, work order no, document number, reference number and date.

The details which have to be recorded about the product line description showing the operations each into recording of start time and total time of operation. The location time recorded is to track if any delay/ reasons and necessary actions if taken with remarks.

If the product has to be completed with any of the further operations in sequence, this card will travel along with job for next workstations for further operations if any to complete the requirement of job, and recorded till finishing of the job.

JOB CARD - FORMAT-1

<table>
<tr><td colspan="5" rowspan="3">**Job Card**</td><td>Doc No.</td><td colspan="2"></td></tr>
<tr><td>Rev No.</td><td colspan="2"></td></tr>
<tr><td>Date</td><td colspan="2"></td></tr>
<tr><td colspan="3">Order Starting Date</td><td colspan="5"></td></tr>
<tr><td colspan="3">Customer</td><td colspan="5"></td></tr>
<tr><td colspan="3">Work Order No.</td><td colspan="5"></td></tr>
<tr><td colspan="8">Details</td></tr>
<tr><td rowspan="2">S.No.</td><td rowspan="2">Date</td><td rowspan="2">CG & M Line Description</td><td colspan="2">Time (Minutes)</td><td>Location</td><td rowspan="2">Time</td><td rowspan="2">Remarks</td></tr>
<tr><td>Start Time</td><td>End Time</td><td>Total Time</td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
</table>

Work activity log format - 2

This document is to record the activity/operations performed by the operator from time to time (format) shows time duration as one hour (For whole day shift). The operator has to record every hour, activity description, equipment/ machinery/instrument used to perform the job.

Any remarks may noted by the operator to complete this record.

WORK ACTIVITY LOG - FORMAT-2

Organisation Name:

Department:

Section:

Employee Name:

Supervisor Name:

Date:

Start / Stop	Operations performed	Equipment / Machinery/ Instruments used	Remarks
8.00 to 9.00 a.m.			
9.00 to 10.00 a.m.			
10.00 to 11.00 a.m.			
11.00 to 12.00 noon			
12.00 to 1.00 p.m.			
1.00 to 2.00 p.m.			
2.00 to 3.00 p.m.			
3.00 to 4.00 p.m.			
4.00 to 5.00 p.m.			
5.00 to 6.00 p.m.			

Batch CG & M record format - 3

This document is for recording the details of CG & M covering the processing steps with documented page number with deviation against each in short description.

This document is to be prepared under heading description of job part number, batch number, name of the part. The processing steps number serially for each process with sequential operations in logical order with documented page number. The description of deviation are noted against each operations in sequence gives the detail of batch CG & M record for every part.

BATCH CG & M RECORD - FORMAT-3

Batch CG & M Record in accordance with batch processing record

Manufacturing Organisation Name: ______________________

Description of job: ______________________

Name of part: __________________________

Batch No.: ____________________________

The following deviations have appeared (continued)

No. process step	**Name of processing step**	**Documented page no.**	**Short description of deviation**
1	Raw material preparation: Operation 1: Descaling Operation 2: Degreasing Operation 3: Wire brushing		1. _____________ 2. _____________ 3. _____________ 4. _____________
2	Sizing of material: Operation 1: Shearing Operation 2: Deburring		1. _____________ 2. _____________ 3. _____________

Estimation and maintenance records

Objectives : At the end of this lesson you shall be able to
- **state the purpose of estimation**
- **explain the details of formats for estimation sheet**
- **explain the details of formats for maintenance log, history sheet of machinery and equipment and checklist for preventive maintenance.**

Estimation is the method of calculating the various quantities and the expenditure to be incurred on a particular job or process.

In case the funds available are less than the estimated cost the work is done in part or by reducing it or specifications are altered,

The following essential details are required for preparing an estimate.

Drawings like plan, elevation and sections of important parts.

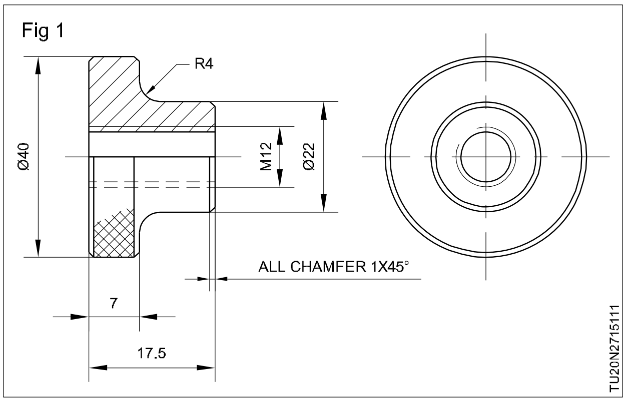
Detailed specifications about workmanship & properties of materials, etc.

Standard schedule of rates of the current year.

Estimating is the process of preparing an approximation of quantities which is a value used as input data and it is derived from the best information available.

An estimate that turns out to be incorrect will be an overestimate if the estimate exceeded the actual result, and an underestimate if the estimate fell short of the actual result.

A cost estimate contains approximate cost of a product process or operation. The cost estimate has a single total value and it is inclusive of identifiable component values.

Part Name: Thumb Screw **Assembly: Dial gauge holder**	**Part No:3** **Material:Fe 3 10** **Stock S 12:45x30**	 Fig 1

Operation No	Operation description	Machine	Estimated time	Rate / piece per hr	Instrument	
01	Job setting on four jaw chuck	Lathe	10 min		Steel rule	
02	Facing	Lathe	5 min			
03	Centre drilling	Lathe	15 min			
04	Turning 39.7 mm	Lathe	5 min		Vernier caliper	
05	Step turning Ø22 mm	Lathe	5 min		Vernier caliper	
06	Form radius of 4mm	Lathe	10 min		Radius caliper	
07	Space b/w,'div' and 'a' hole	Lathe	10 min			
08	Cut a internal thread use machine tap	Lathe	20 min		threaded a plug gauge	
09	1 x 45^0 Champer	Lathe	5 min			
10	Reverse the hold the job true it	Lathe	7 min			
11	Facing	Lathe	10 min		Vernier caliper	
12	turn 39.7mm and revolving centre support	Lathe	5 min 10 min		Vernier caliper	
13	clamp the knurling tool and knurl the surface	Lathe	20 min			
14	1 x 45^0 chamfer of knurl surface	Lathe	15 min			

Maintenance log - Format 5

This format is made with details of maintenance activities performed machinewise,

MAINTENANCE LOG - FORMAT - 5

Organisation Name :

Department:

Section:

Name of the machine:

S.No	Date	Nature of fault	Details of rectification done	Signature of in-charge

History sheet of machinery equipment - Format 6

The document recorded with historical data about the machinery and equipment, contains all details about supplier address, order no., date of receipt, installed and placed, Date of commissioning and machine dimensions, weight, cost, particulars of drive motor, spare parts details, belt specification, lubrication details, major repair/ overhauls done with dates recorded then and there for analysing the functional and frequency of breakdown etc.,

MACHINERY AND EQUIPMENT RECORD FORMAT - 6

Organisation Name:	
Department:	
Section:	
History sheet of machinery & Equipement	
Description of equipment	
Manufacturer's address	
Supplier's address	
Order No. and date	
Date on which received	
Date on which installed and place	
Size : Length x Width x Height	
Weight	
Cost	
Motor particular	Watts/ H.P./ r.p.m: phase: Volts:
Bearings/ spares/ record	
Belt specification	
Lubrication details	
Major repairs and overhauls carried out with dates	

Checklist for preventive maintenance inspection - Format 7

The very essential document required to observe, the functional aspects of each parts, defects and the remedial measures taken is recorded.

This format enables to program the frequency of maintenance schedules so as to minimise frequent breakdown of machinery/equipments.

PREVENTIVE MAINTENANCE RECORD - FORMAT 7

Organisation Name :

Department :

Section :

Name of the Machine :

Machine Number :

Model No & Make :

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/chain and its tension				
Bearing condition (Look, feel, Listen noise)				
Driving clutch and brake				
Exposed gears				
Working in all the speeds				
Working in all feeds				
Lubrication and its system				
Carriage & its travel				
Cross-slide & its movement				
Compound slide & its travel				
Tailstock's parallel movement				
Electrical controls				
Safety guards				

Inspected by :

Signature :

Name :

Date : Signature of in-charge

Terms used in part Drawings and Geometrical tolerances, and symbols

Objective: At the end of this lesson you shall be able to
- **state the meanings of various drawing terminology.**

Glossary of drawing terminology

Abstraction: The reduction of an image or object to an essential aspect of its form or concept.

Acute angle: An angle lesser than 90 degrees.

Ascending planes: Flat surfaces not parallel to the ground or floor planes whose vanishing points appear above the horizontal line.

Background: The most distance zone of space in a three-dimensional illusion.

Concave: A shape that is hollow and curved, like the inside portion of a bowl.

Convex: A shape that curved outward, like the surface of a sphere.

Content: The subject matter of a work of art, including its emotional, intellectual, symbolic, the matic , and narrative connotations, which together give the work its total meaning.

Descending planes: Flat surfaces not parallel to the ground or floor planes whose vanishing points fall below the horizon line.

Descriptive drawing: Highly detailed renderings intended to articulately define what is seen.

Design: A working plan or an arrangement of parts, form, color etc.

Drawing: The act by which an artist records what he or she sees or feels; The fine art of making a descriptive visual expression by dragging a tool across a receptive surface-usually a piece of fine-quality, toothed paper.

Full range: A complete gradation of tonal values from the lightest to the darkest values whether black and white or in a color context.

Geometric shape: Shape created by mathematical laws and measurements, such as a circle or a square.

Ground plane: The flat, horizontal plane on which the view stands that extends to the horizon.

Horizon line: A line corresponding to the eye level of the viewer in linear perspective; Contains the vanishing points in a one -point or two-point perspective drawing.

Horizontal: Parallel to the plane of the horizon; flat and even, level.

Idealized drawing: A rendering which achieves a satisfactory resemblance to the subject while simultaneously eliminating imperfections.

Kneaded eraser: A soft, malleable eraser that is used to lighter values or as an aid in the creation of gradations.

Layout: The placement of an image within a two - dimensional format.

Life drawing: Drawing from live forms in order to gain visual understanding of the movements, gestures, and physical capabilities of live bodies as aesthetically pleasing art forms.

Line: The pathway of a moving point as in the trail of a scratch, a brush stroke, or the engraved deposition of drawing material resulting from dragging a tool over a receptive surface; visible by contrasts in value.

Line quality: The visual traits and /or expressive character of lines.

Modeling: The change from light to dark across a surface; a technique for creating the illusion of form and/or space.

Multiple perspectives: Difference eye levels and perspectives used in the same drawing.

Oblique: A slanting line or plane that is not parallel to the edges of the picture plane.

Obtuse angle: An angle greater than 90 degrees.

Orthogonal line: In linear perspective, an oblique line that is drawn using a vanishing point.

Parallel: Extending in the same direction at the same distance a part, so as never to meet.

Pattern: A repeated compositional element or regular repetition of a design or single shape.

Perpendicular: At right angles to a given plane or line exactly upright or vertical.

Perspective: A mathematical method of representing forms as they recede in space and diminish in size using converging lines that meet a central vanishing point on the horizon line.

Pictorial: Refers to a picture, not only its actual two-dimensional space but also its potential for three dimensional illusion.

Picture plane: the actual flat surface, or opaque plane, on which drawing is produced. It also refers to the imaginary, transparent "windows of nature" that represents the format of a drawing mentally super imposed over real-world subject matter.

Planer analysis: A structural description of a form in which its complex curves are generalized into major plane zones.

Plane: A perfectly flat surface as a geometric plane, or floor plane;

Profile: The shape of a head (or) face (i.e.) seen or drawn from the side.

Rectilinear shapes: Two-dimensional shapes that are characterized by having four sides that meet one another at right angles; shapes based upon rectangle and squares.

Render: To draw with an unusually high degree of detailed representation.

Scale: Size and weight relationships between forms.

Sketch: A drawing, or an act of drawing done quickly with minimal or no elaboration, but effectively captures the essential form in space, with basic light and shadow.

Tangent: Touching a curved surface at one point but not intersecting it.

Taper: To decrease gradually in width or thickness.

Texture: The surface character or tactile qualities experienced either through the sense of touch or through the imagination.

Three-dimensional: Having height, width and depth; synonymous with form.

Three-dimensional space: the illusion of volume or volumetric space.

Three-point perspective: A system for depicting three-dimensional depth on a two dimensional surface. In addition to lines relending to two points on the horizon, lines parallel and vertical to the ground appear to converge to a third vanishing point located either above or below the horizon line.

Two -dimensional: Having height and width; synonymous with shape.

Two dimensional space: Space that has height and width with little or no illusion of depth.

Two-point perspective: A form of linear perspective which the line receding into space converge at two vanishing points on the horizon line(eye level), one to the left of the object being drawn and one to the right.

Unit of measure: A portion of a complex shape or three-dimensional form that is used to measure all of the order elements within the shape or form-the head serves as a good unit of measure for determining proportional relationships within human figure.

Working drawings: The studies artists make in preparation for a final work of art.

Geometrical tolerancing

Objectives: At the end of this lesson you shall be able to
- **define geometrical tolerance**
- **state the necessity of using geometrical tolerances**
- **list the recommended symbols for tolerancing under the three groups of form, attitude and location.**

Form i.e. straightness, flatness, roundness, cylindricity and profile of a line and a surface

Attitude i.e. parallelism, squareness and angularity

Location i.e. position, concentricity and symmetry

Definition of geometrical tolerance

Geometrical tolerance is the maximum permissible overall variation of form or position of a feature.

Reason for using geometrical tolerance

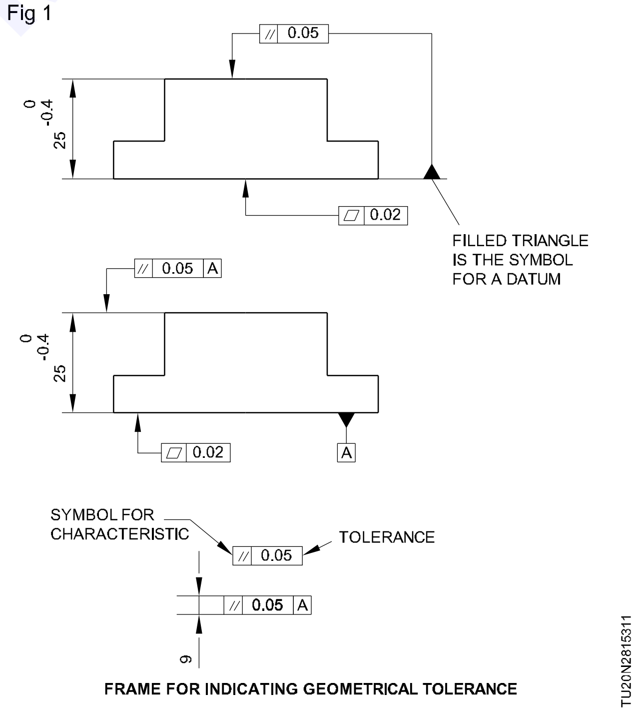
This will help the operator to produce the components, particularly those parts which must fit together precisely.

To have an international system which will overcome the usual language barrier. This is achieved by the use of symbols which represent geometrical characteristics.

General principles of geometrical tolerances

The geometrical tolerance consists of a frame which contains a symbol, representing the geometrical tolerance zone, in this instance 0.05 for the characteristic of parallelism. The symbol for flatness is shown accompanied by the tolerance zone figure of 0.02 in the lower frame. (Fig 1)

Fig 1



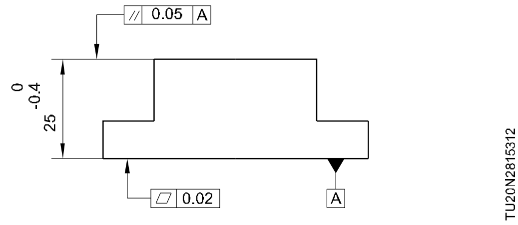
FRAME FOR INDICATING GEOMETRICAL TOLERANCE

Notice that from each of the frames a leader is drawn so that it is normal, i.e. at 90° to the relevant face and ending with an arrow-head against the face.

Notice also that from the ‘parallelism’ frame, another leader is drawn terminating in a blacked-in equilateral triangle on a projection drawn out from the base line. The blacked-in triangle (about 4.5 mm high from base to apex) is the symbol used to represent a datum face or line.

An alternate method of arranging the frames and symbols is shown in Fig 2 where the datum is given a letter and a frame of its own and an independent leader line ending in the blacked-in triangle, inverted and drawn against the actual component base line. The datum letter ‘A’ is then added as an extra component in the geometrical tolerance frame.

Fig 2



Recommended symbols for geometrical tolerancing

Geometrical tolerances are arranged into three groups. They are tolerances of form, of attitude and of location.

Form

Tolerances of FORM are identified by the use of symbols for the following characteristics.

Characteristic	Symbol
Straightness	—
Flatness	▱
Roundness	○
Cylindricity	⌭
Profile of a line	⌒
Profile of a surface	⌓

The application of symbols is indicated in fig 3, where (3a), (3b), (3c) & (3d) show the use of geometrical tolerances controlling the straightness of a circular

Section part. In (3a) and (3b) the leader lines from the tolerance frame end in an arrow-head against the axis applies to the full length of the part. The interpretation at (3a) shows that for functional acceptance, the entire main axis must lie between two parallel straight lines 0,1 apart in that plane. At (3b) the symbol for the diameter ® precedes the tolerance. This means that the entire main axis must lie within a cylindrical tolerance zone 0.1 mm diameter.

Figures (3c) and (3d) show the same geometrical tolerance, applied this time to the diameter dimension of the smaller diameter of the part.

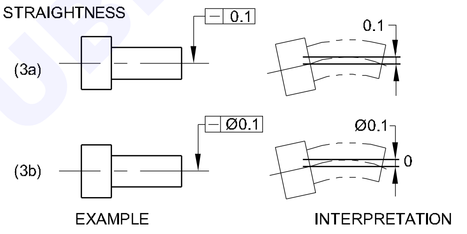
This means that the geometrical tolerance applies over the length of the dimensional feature only.

Figure (3e) and figure (3f) deal with the geometrical tolerance for flatness of a surface, where the symbol for flatness is followed by the tolerance figure of 0.05. This figure indicates that the actual surface (as previously shown in Figure 1) must be between two parallel planes 0.05 apart. If a particular form of direction is prohibited, then this is stated in a note form against the tolerance frame. Eg. ‘Not concave’.

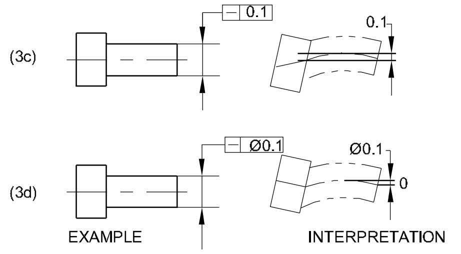
The geometrical tolerance controlling roundness of a part is shown in figures (3g), (3h) and (3j). The interpretation for (3g) and (3h) is that the true form of the periphery of the part at any cross-section perpendicular to the axis must lie between two concentric circles whose radial distance apart is 0.02 for (3g) and 0.03 for (3h).

For the sphere shown in (3j) the geometrical tolerance applies to concentric circles with the radial distance apart 0.04 at the periphery at any section of maximum diameter.

Fig 3



STRAIGHTNESS TOLERANCE FOR ENTIRE AXIS OF COMPONENT



STRAIGHTNESS TOLERANCE FOR AXIS OVER LENGTH OF DIMENSIONED FEATURE ONLY

Fig 4

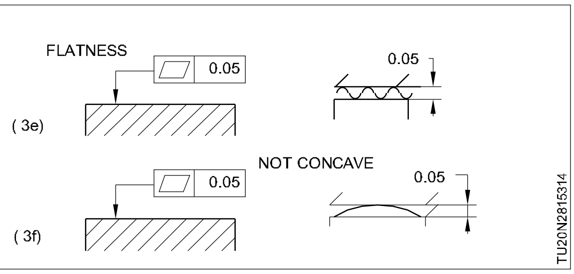
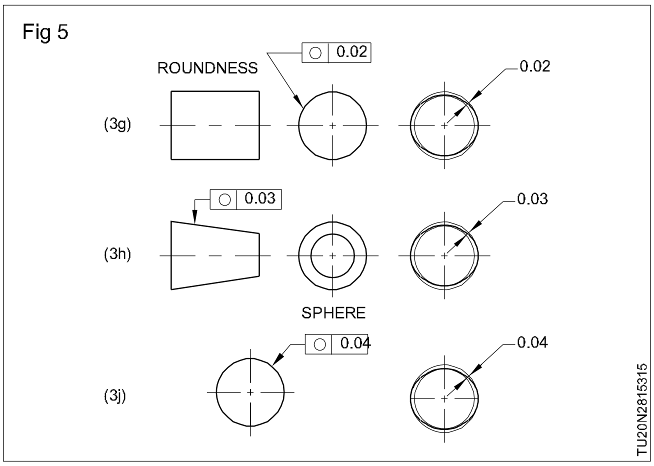


Fig 5



The sphere controlling cylindricity is shown in Fig 4. Here the interpretation shows, that for acceptance, the surface of the part must be within two coaxial cylinders, whose radial distance apart is 0.05.

Fig 6

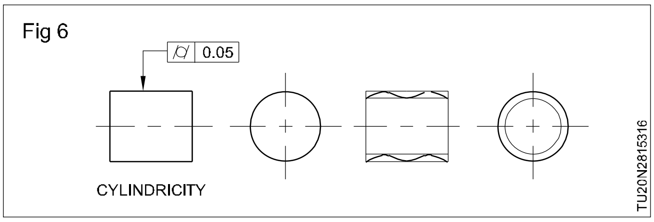
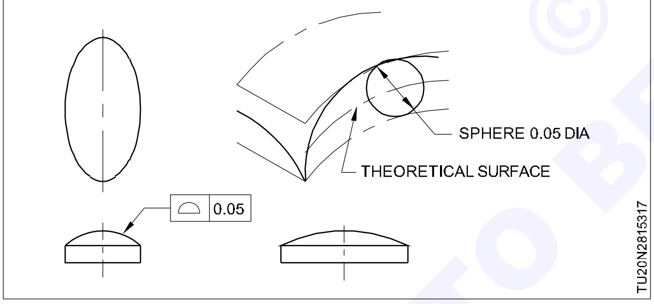


Fig.5 shows the method of applying a geometrical tolerance to a curved surface. The symbol is followed by the tolerance 0.05 which means that the actual surface must lie between two surfaces enveloping a succession of sphere 0.05 diameter whose centre lies on the theoretical surface.

Fig 7

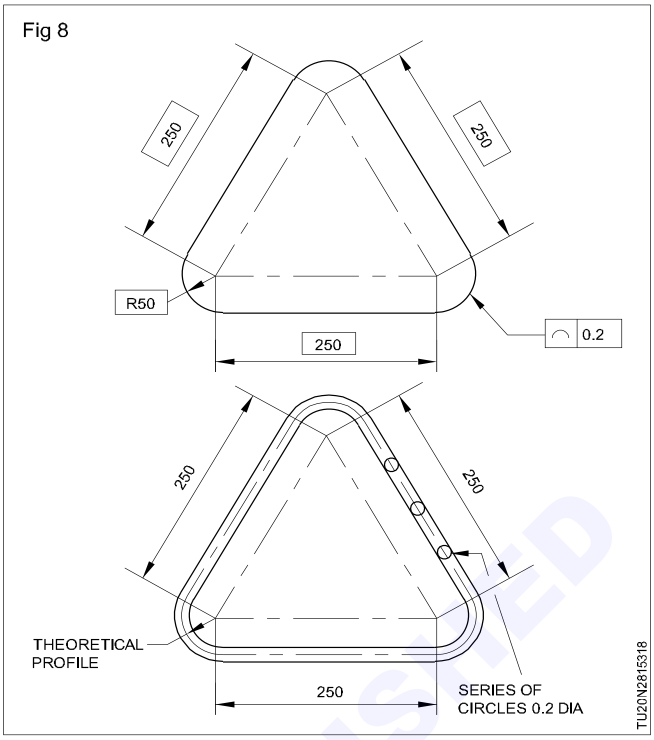


In Fig.6 the geometrical tolerance is applied to linear dimensions controlling the profile. The rectangular 'boxes' around the 250 centre dimension and the 50 radius is the method used to indicate theoretical dimensions i.e. the dimensions relevant to perfect form.

The interpretation of the geometrical tolerance is that the actual profile must be between two lines which touch a succession of circles 0.2 dia. whose centre lies on the theoretical profile.

Characteristic	**Symbol**
Parallelism	//
Squareness	⊥
Angularity	∠

Fig 8



Attitude

Tolerances of attitude are identified and indicated by the use of symbols for the following.

A typical application of tolerances for these three characteristics is shown in figures 9 to 18. Fig 9, 10 and 11 show the application of tolerancing to control 'parallelism'. Fig 9 shows that the axis of the upper hole must lie between the two lines 0.08 apart which are parallel with the datum axis, i.e., the axis of the lower hole, as indicated by the leader ending in the blacked-in triangle. In Fig10 the method uses a separate datum letter 'A' which is added to the frame after the tolerance of 0.05 diameter. (Note the symbol Ø.) The requirement is that the upper hole axis must lie within a cylindrical zone 0.05 diameter with its axis parallel with the axis of the datum hole 'A'. Fig 11 shows a component whose upper surface must be between two parallel planes 0.05 apart, parallel with the bottom datum surface. While the overall tolerance zone is 0.05 as shown in the upper section of the frame, the figures in the lower section of the frame stipulate that over any length of 100 the parallelism tolerance is reduced to 0.02.

Fig 9

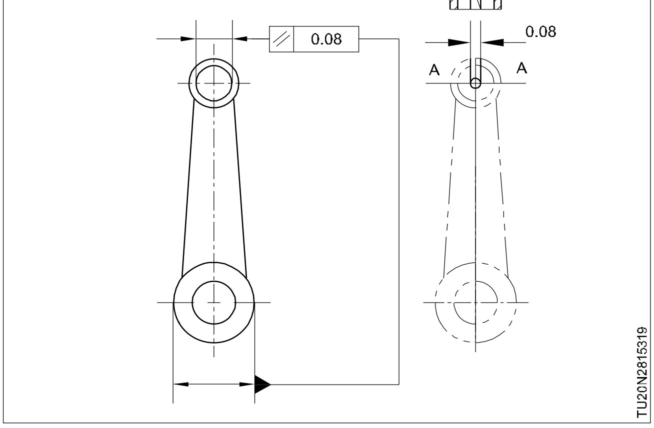


Fig 10

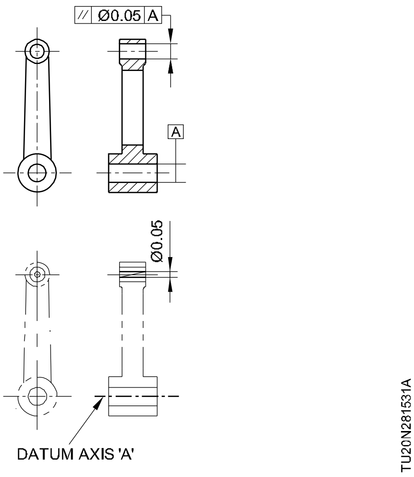
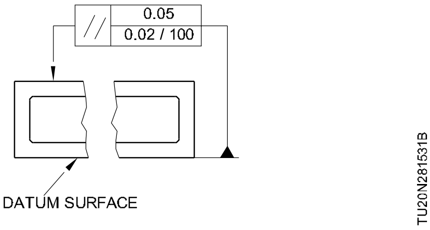


Fig 11

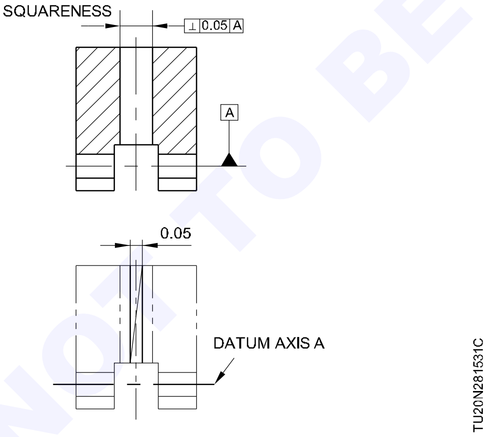


Examples of the application of the geometrical tolerance for 'squareness' are shown in Figs 12, 13,14 with 12, 13 and (14) using the separate box method for indicating the datum.

The interpretation is as follows.

The axis of the vertical hole must lie between two parallel lines, 0.05 apart, which are perpendicular to the common datum axis 'A' of the two horizontal wholes. (Fig 12)

Fig 12



The axis of the upper cylindrical portion must lie within a cylindrical tolerance zone of 0.01 diameter, the axis of which is perpendicular to the datum surface 'A'. (Fig 13)

This shows that the right hand end face must lie between two parallel planes 0.05 apart, which are perpendicular to the datum axis. (Fig 14)

Here the datum surface is indicated by a leader from the frame. The requirement is that the right hand face must lie within the two parallel planes, 0.05 apart, which are perpendicular to the datum surface. (Fig 15)

Fig 13

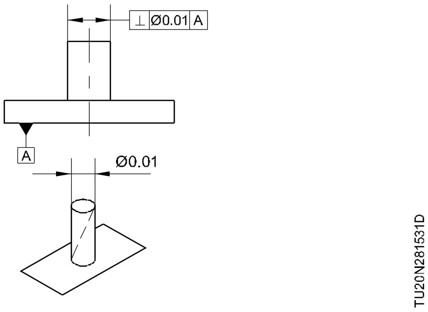


Fig 14

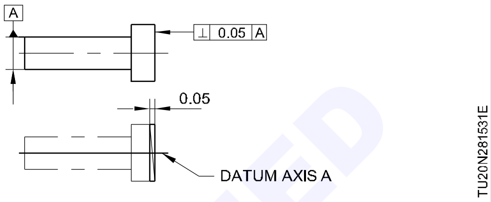
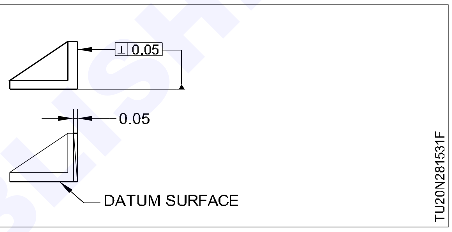


Fig 15



Geometrical tolerances for the control of ANGULARITY are shown in Fig 16.

The Fig 16 shows that the requirement is the axis of the hole must lie within the cylindrical tolerance zone 0.1 diameter, the axis of which must be included at the theoretical angle of 60° to the datum surface A.

In Fig 17 the requirement is that the right hand end face must lie within the two parallel planes 0.08 apart which are inclined at the theoretical angle of 75° to the datum axis A of the through hole.

Fig 16

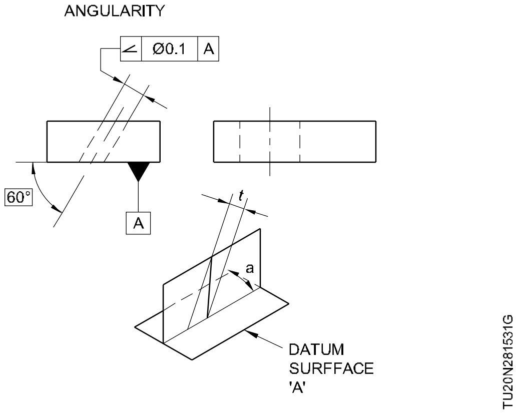


Fig 18 shows a component whose upper angle face, must lie between the two parallel planes 0.05 apart which are inclined at the theoretical angle of 35° to the base, the datum surface. Notice that the theoretical angle in each example is boxed.

Fig 17

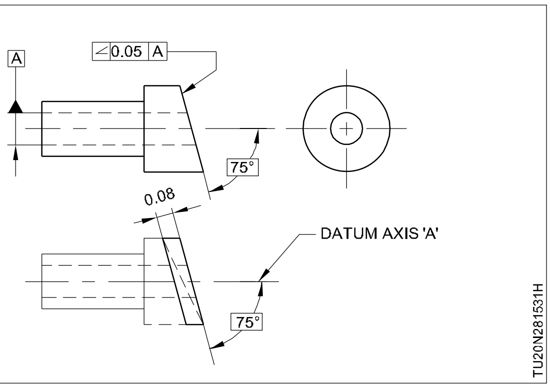
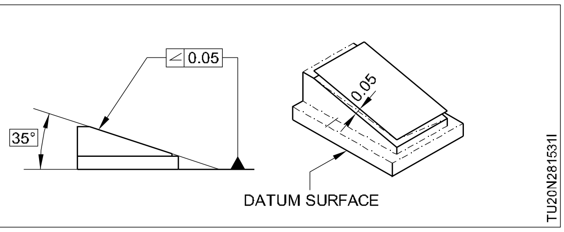


Fig 18



Location

Tolerances of location are identified and indicated by the use of symbols for the following characteristics.

Characteristic	**Symbol**
Position	⌖
Concentricity	◎
Symmetry	⌯

Figs 19, 22, 25 show typical examples of these characteristics and symbols. In Figure (10a) the hole centre dimensions of 25 and 30 are boxed to show that these are the theoretical dimensions. The geometrical tolerance requires that the hole centre must lie within a cylindrical zone 0.05 diameter. The use of theoretical positions, also known as 'true positions', implies that the axis of the cylinder is square with the plane of the drawing.

Fig 19

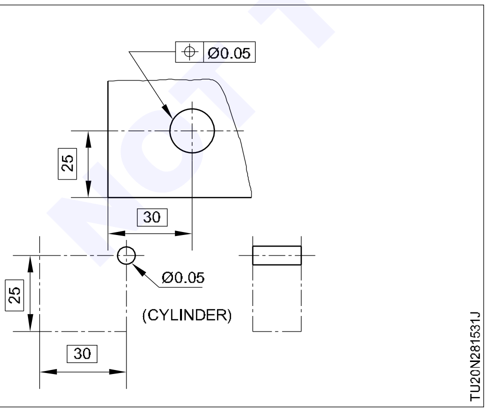
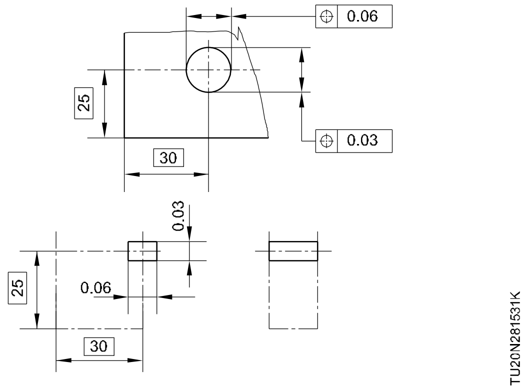


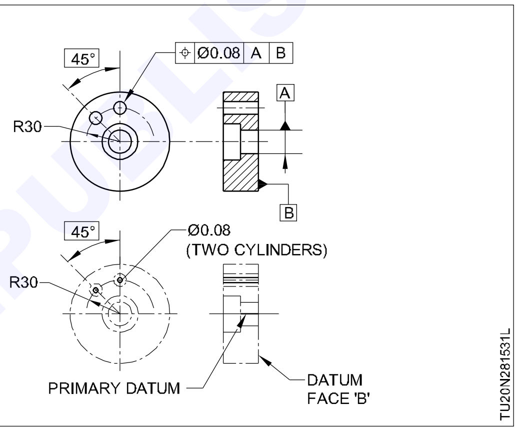
Fig 20 shows the hole with the same true positions, but with the geometrical tolerances arranged to give greater tolerance along the horizontal axis. The resulting requirement is that the axis of the hole must lie within a rectangular box whose sides are 0.03 and 0.06, and length equal to the width of the component.

Fig 20



In Fig 21 the two holes are shown with their true position spaced at 45° on a 30mm pitch circle radius.The geometrical tolerance shows that each actual hole centre must lie within a cylindrical zone 0.08 diameter whose axis lies at the true centre position. The tolerance cylinders are disposed relative to the two datum features, namely the axis of the smaller bore and the right hand end face. The datum letters are included in the tolerance frame.

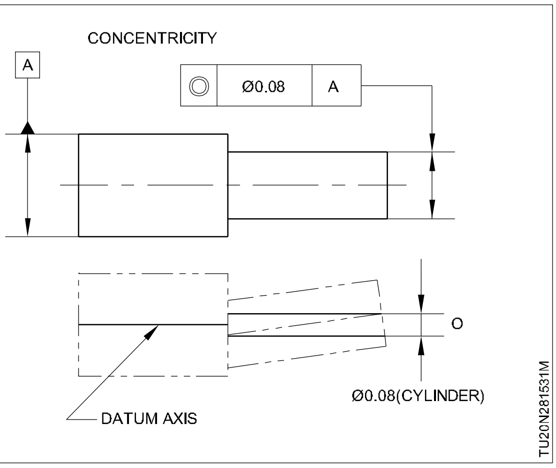
Fig 21



Examples of geometrical tolerance for 'CONCENTRICITY' are given in figures. (22), (23) and (24). The interpretations are as follows.

In Fig 22 the axis of the smaller diameter must lie within the cylindrical zone 0.08 diameter which must be coaxial with the datum axis i.e. the axis of datum diameter 'A'.

Fig 22



In Fig 23 the axis of the two end portions must lie within a cylindrical tolerance zone 0.08 dia.

Fig 23

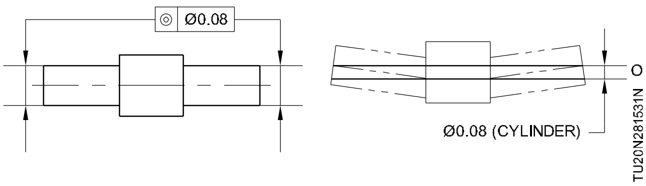
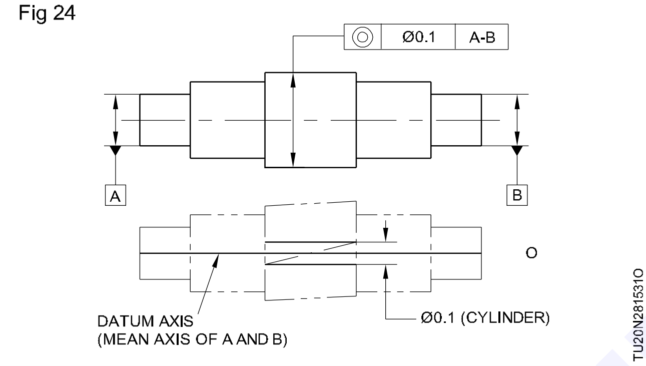


Fig 24 The axis of the large central portion must lie within a cylindrical zone 0.1 diameter which is coaxial with the mean axis of the datum diameters 'A' and 'B'. (Notice that to indicate the requirement of the mean axis the datum letters are separated by a hyphen and enclosed in the same compartment of the tolerance frame)

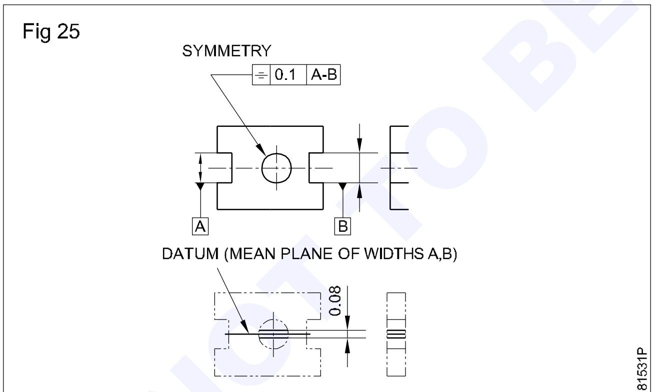
Fig 24



The geometrical tolerance of 'SYMMETRY' follows in Fig 25,26 and 27 where the interpretations are:

- Fig 25 the axis of the hole must lie between two parallel planes, 0.08 apart which are symmetrically disposed about the mean axial plane of datum width 'A' and 'B'

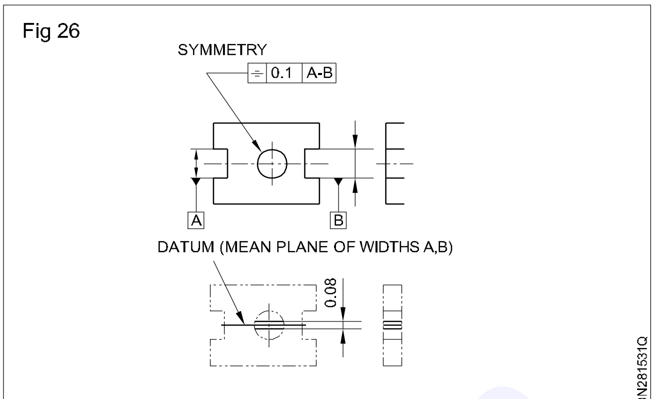
Fig 25



TWO PLANES 0.08mm APART SYMMETRICALLY
DISPOSED ABOUT THE DATUM

- Fig 26 the mean plane of the slot must be between two parallel planes, 0.05 apart symmetrically disposed about the mean plane of the datum width 'W'

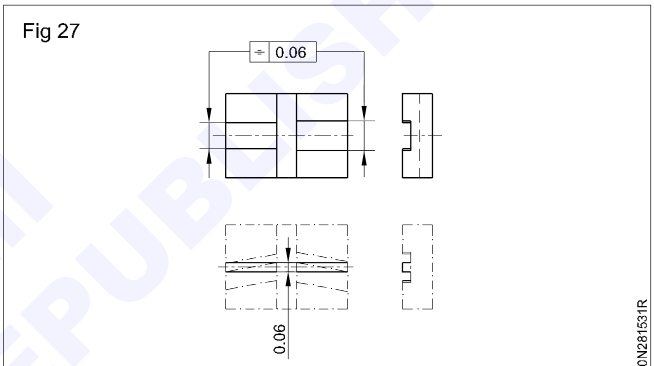
Fig 26



TWO PLANES 0.08mm APART SYMMETRICALLY DISPOSED ABOUT THE DATUM

- Fig 27 the median planes of the two end slots must be between two parallel planes, 0.06 apart.

Fig 27



TWO PLANES 0.06 mm APART

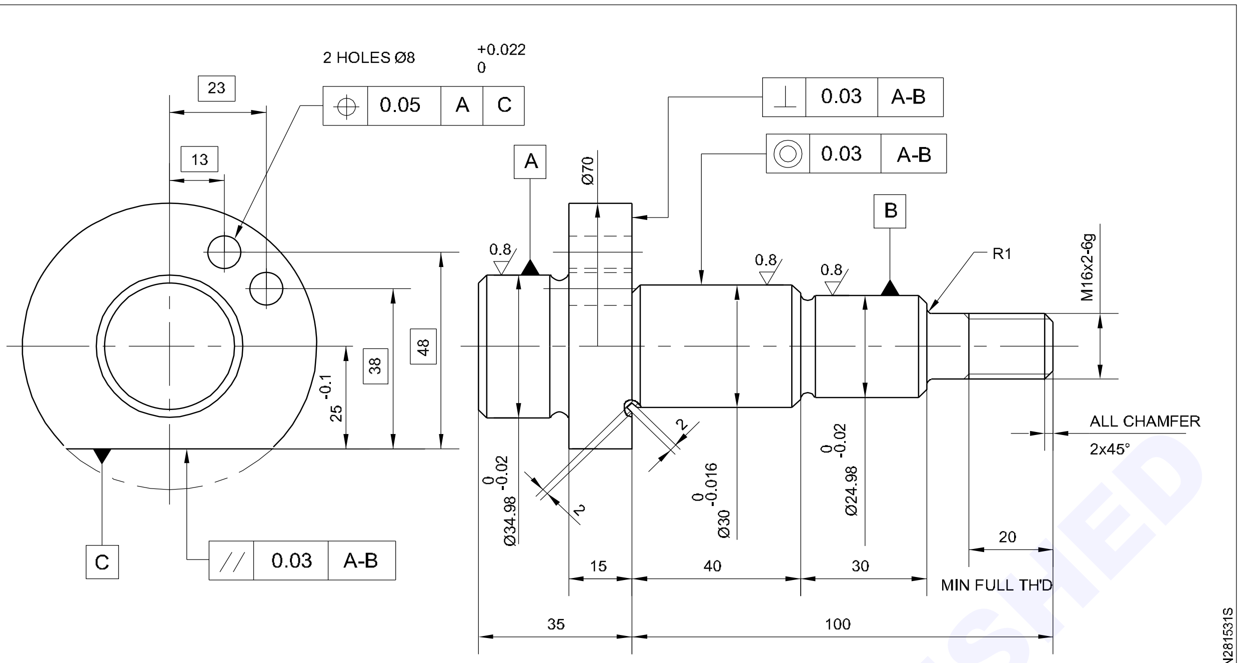
Geometric tolerance classes and types

A geometrical tolerance is the maximum permissible overall variation of from or position from that shown in the drawing.

Geometrical tolerances are applied apart from dimensional tolerances, where it is necessary to control more precisely the form or shape of some feature of the part because of its functional requirements.

A detailed drawing which indicates geometrical tolerances. (Fig 28)

Fig 28



A DETAILED DRAWING WHICH INDICATES GEOMETRICAL TOLERANCES

Class	Type	Symbol	Reference entry
Form	Straightnes	—	Surface of revolution, axis, straight edge
	Flatness	▱	Plane surface (not datum plane)
	Circularity	○	Cylinder, cone, sphere
	Cylindricity	⌭	Cyclinder surface
Profile	Line	⌒	Edge
	Surface	⌓	Surface (not datum plane)
Orientation	Angularity	∠	Plane, surface, axis
	Parallelism	//	Cyclinder, surface, axis
	Perpendicularity	⊥	Planar surface

Location	Position	⌖	Any
	Concentricity	◎	Axis, surface of revolution
	Symmetry	⌯	Any
Run out	Circular	↗	Cone, cylinder, sphere, plane
	Total	⌰	Cone, cylinder, sphere, plane

Automatic lathe - types - parts, toolholders, theory of calculation

Objectives: At the end of this exercise you shall be able to
- **differentiate between a centre lathe and a CG & M lathe**
- **list and name the parts of a capstan lathe**
- **specify a capstan lathe**
- **state the constructional features of a capstan lathe.**

Automatic lathes

Lathes, which are capable of automatically performing a preset cycle of operations by presenting the work to the cutting tools and of ejecting the finished workpiece, are known as automatic lathes.

The various types of automatic lathes are:

- automatic turret lathes
- single spindle automatic lathes
- multi–spindle automatic lathes.

These machines are usually similar to their standard types with the addition of automatic features. The degree to which a lathe is made automatic is determined by the purpose for which it is to be used.

Modern automatic lathes are available with the following features.

- Load the workpiece (manual).
- Start the machine (manual).
- Circulate the coolant (automatic).
- Perform all cutting operations (automatic).
- Change the speeds, feeds, and tools (automatic).
- Gauge and inspect the finished workpiece (manual).
- Unload the workpiece (automatic).
- Repeat the cycle without attention from the operator (automatic).

Automatic turret lathes

The automatic turret lathe is sometimes called an automatic chucking machine. It is similar in principle to the standard turret lathe, and additionally, it is automatic in operation except for the insertion and removal of the work.

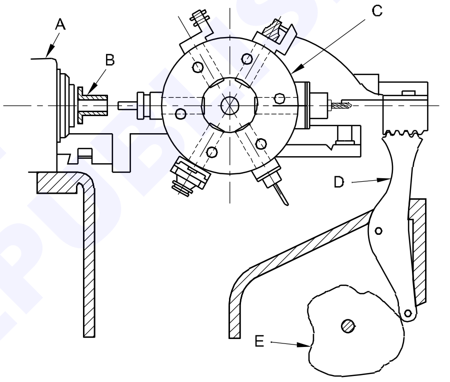
These lathes vary in size and range from heavy duty lathes for rapid and accurate CG & M of components to lathes used to manufacture small items from barstock.

The feeding of tools, indexing of turret, movement of turret slides and cross-slides and change of speeds and feeds are automatically controlled by cams or a plug board system.

Single spindle automatic lathes

There are many types of single spindle automatic lathes in use. (Fig 1) The automatic screw machine is a popular type. In a single spindle automatic machine, only one component at a time is machined and this machine is used for a wide range of operations. These operations vary from cutting small screw threads to machining castings and forgings.

Fig 1



A- HEAD STOCK : B - JOB
C - TURRET : D - CAM LEVER
E - TURRET CAM : F - CROSS SLIDE

SINGLE SPINDLE AUTIMATIC LATHE

TU20N2815411

The control of feeds and speeds and the indexing of tools may be done by cams, plug boards or the numerical control system.

Automatic screw machines

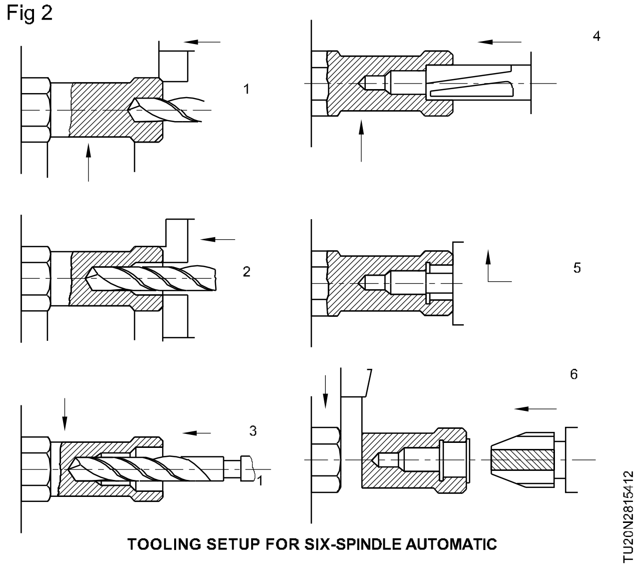
These machines are used to produce screws, bolts and pins from bar stock. The cross slide, turret slide, and bar–feed mechanism are all operated automatically.

Multi-spindle automatic lathes

Multi-spindle automatic lathes represent an extremely fast method of producing work. The work may be bar–fed or held in a chuck. The bars are fed into the spindles which are located in a spindle carrier. The spindles are positioned in a circular pattern in the carrier. The carrier which is being operated by an indexing mechanism may accommodate 4, 6 or 8 spindles. The spindle rotates in the carrier and the carrier is indexed from one tool station to the next tool station, according to the sequence.

The tooling set up for a six–spindle automatic lathe is shown in Fig 2.

Fig 2



TOOLING SETUP FOR SIX-SPINDLE AUTOMATIC

The tools are mounted in a cam or gear-operated tool slides, which operate at right angles and parallel to the spindle.

The difference in operation between multi-spindle and other automatic lathes is that in a multispindle machine the rotating work is indexed from tool station to tool station whilst in the others the tools themselves are indexed.

The spindle–carrier assembly is a strong and precisely made unit. It consists of a spindle carrier, work spindle, spindle stop and start, and the indexing mechanisms.

The nearly radial position of the cross slide directs all cutting thrusts towards the centre of the carrier. This arrangement permits heavy feeds, and provides extra room for tooling without holder interference.

The Index error is minimized by the tangential cutting action.

Tools and equipments

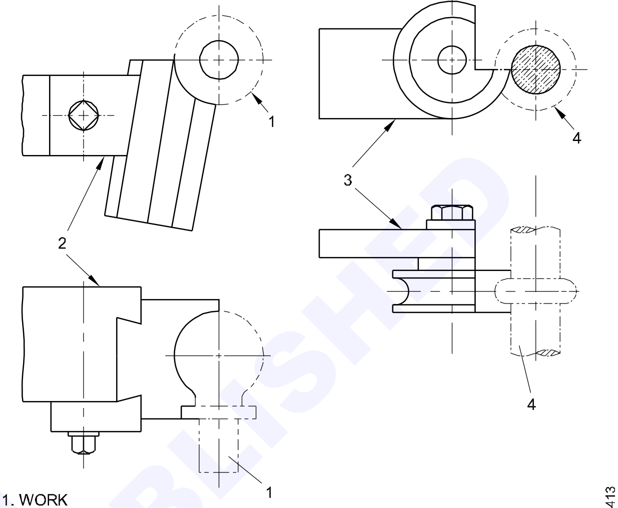
A wide range of tools are available for use on automats. The tool-holders used are interchangeable between machines of the same series and frequently between stations on the same machine.

Standard tool-holders and attachments are designed to cover most requirements, and when necessary, special holders are available to handle the less common CG & M processes.

Tools may be preset in their respective tool-holders or presetting fixtures which are usually situated away from the machine. Change over from the manufacture of one type of component to another type is confined to change of tape and replacement of the preset tools, collets and the raw material for the new component. This is done when the machine is numerically controlled.

Circular form tools (Fig 3)

Fig 3



1. WORK
2. STRAIGHT FORMING TOOL
3. CIRCULAR FORMING TOOL
4. WORK

In any type of automatic lathes, turning concave, convex or any irregular shape on the component is achieved using form tools. These form tools are of two types, straight and circular. The straight types are used for wider surfaces and the circular types are used for narrower surfaces. Fig 3 illustrates forming operations performed by straight and circular form tools.

By using circular form tools many operations are combined and performed, all in one stroke, by shaping the circular form tools accordingly.

While using circular form tools, special attention is given for selecting the correct speeds and feeds. This is due to the presence of a wider surface area coming in contact with the circular form tools during the machining.

The cross feed ranges from 0.01 to 0.08mm per revolution and the cutting speed is slightly less than that for longitudinal turning.

Use of cams in automatic lathes

Objectives: At the end of this exercise you shall be able to
- **classify the cams used on automatic lathes**
- **state use of cams in automatic lathes.**

Fully automatic or semi-automatic lathe machines are often controlled with the help of drum cams and cam plates. The lathe tools necessary for CG & M are mounted in one or more supports ie. drum turret or horizontal turret head. The drum cam or cam plate, which rotates at constant speed, controls the movements of the tools. The turret head is indexed first and then the tool carriage and/or tool post is moved towards the workpiece at the correct feed rate.

Cams are used to actuate various slides in the automatic lathes.

Cams may be classified according to their shapes and mechanical designs into several distinct groups with the subdivisions numbering a dozen or more.

For practical purposes, all cams can be divided into three classes; radial or plate (Fig 1), cylindrical or barrel (Fig 2) and pivot beam type (Fig 3) which is found useful in fixture designs.

Fig 1

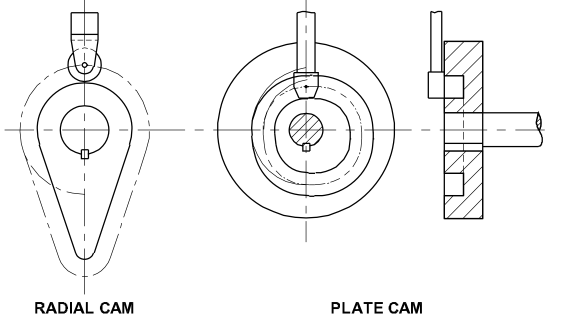
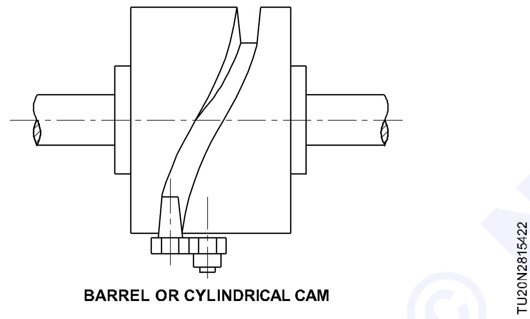


Fig 2



In fully automatic lathes (generally used for machining bar stock) the workpiece feeding, clamping and cutting off are also governed by the cam control system.

The various features considered at the time of preparing cam design worksheets are sequence of operation, speeds, feeds, depth of cuts, index of turret and other slides (front, rear and top).

Fig 3

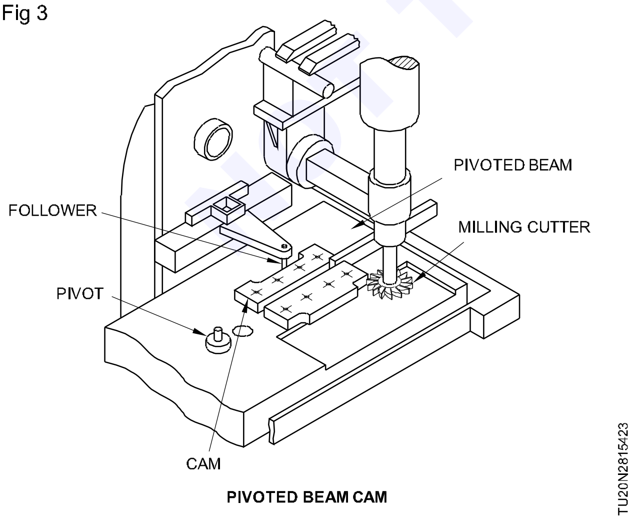


Fig 5 is the tool layout for the job shown in the worksheet of Fig 4.

Fig 4

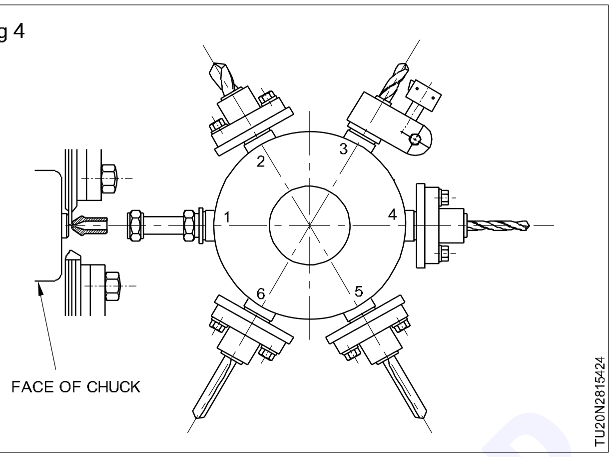
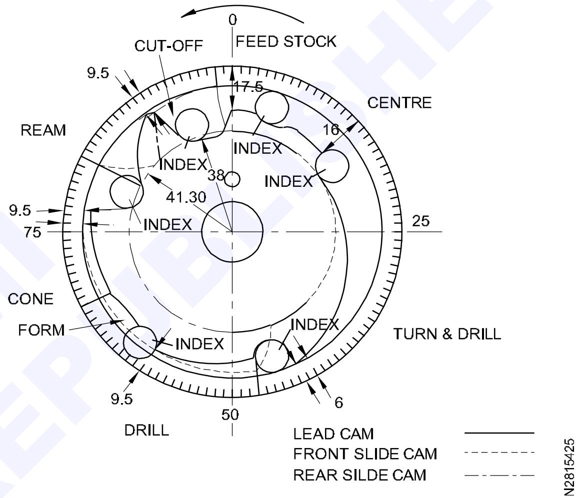


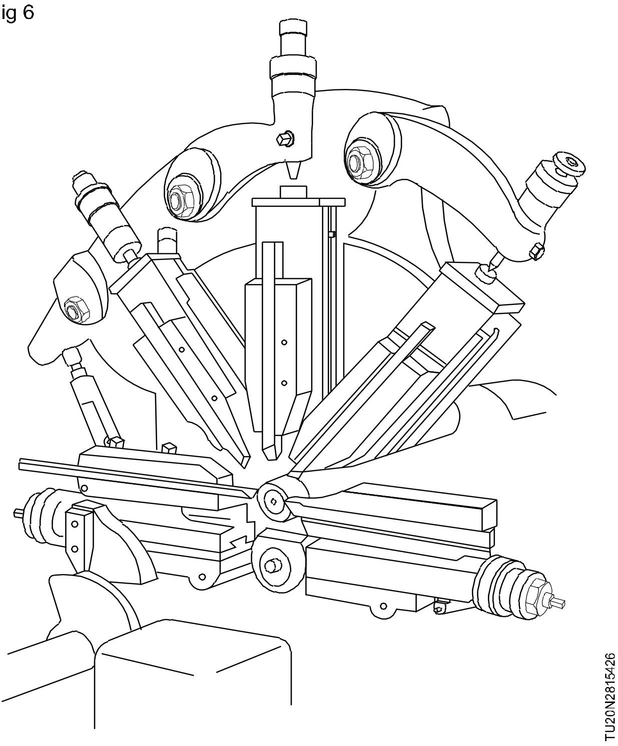
Fig 5

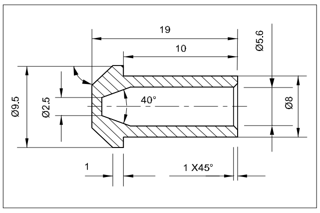


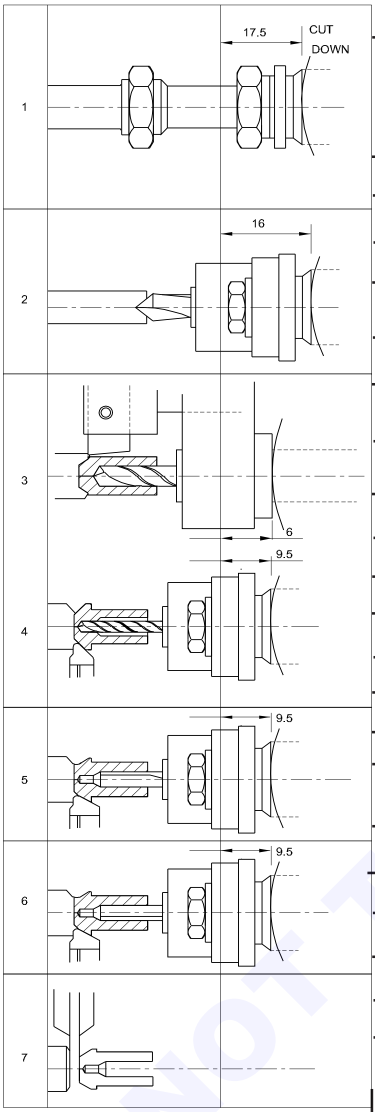
CAM LAYOUT FOR JOB SHOWN IN FIG 4 (ALL DIMENSIONS IN mm)

Fig 6 is the cam layout for the job shown in Fig 4.

Fig 6



Important SHEET	**Automatic screw machine**			**Short stroke**	
	**Work spindle speeds rev/min.**			**Cutting speed m/min.**	
	Fast	**R.H** **R.H**	**4995**	**Turning** **Drilling 5.25 dia.**	**150** **82**
				Drilling 2.5 dia.	**46**

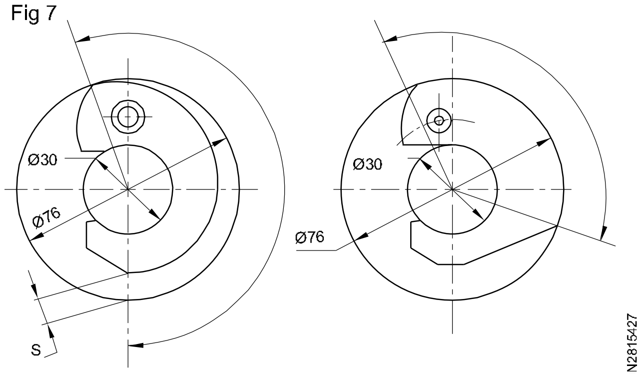
Distance between turret and chuck		Sequence of operation	Throw in mm	Feed mm per rev.	Revs.of work spindle			Hund-reths
Minimum 46	Travel 35				for each opera-tion	over lapped open	Correc ted	
		**Feed stock**	**Dwell**		**26**		**26**	**4 1/2**
		Index turret (one station)			26		26	4 1/2
		Centre End of bar	3	0.15	20		29	5
		Dwell To clean up					6	1
		Index turret (one station)			26		26	4 1/2
		Turn Φ8 and drill Φ 5.25	14.25	0.125	114		117	20
		Dwell To clean up					6	1
		Index turret (one station)			26		26	4 1/2
		Drill Φ 2.5 hole	6.5	0.07	93		93	16
		Drop back To clear					9	1 1/2
		Form (over lapped)	2.4	0.012		200	(204)	(35)
		Dwell To clean up					6	1
				0.005				
		Index turret (one station)			26		26	4 1/2
		Cone bottom of Φ 5.25	3.2	0.07	46		50	8 1/2
		Dwell To Clean up					6	(1)
		Index turret (one station)			26		26	4 1/2
		Ream ₵ 5.6 and cone	12.75	0.25	51		53	9
		Dwell To clean up					6	1
		Index turret (over lapped)			26		26	(4 1/2)
		Cut off Cross-slide(R)	1.5	0.05	30		32	5 1/2
		Cut off Cross-slide(R)	0.5	0.07	7		3	1 1/2
		Clear cut off tool					12	2
		Totals before correction	92 1/2%		517			7 1/2
		Totals after correction	92 1/2%		539		583	100

In an another type of single spindle automatic lathe the slides are operated by individual cams. Fig 7 show the various tools slides.

The cams used on such a machine are shown in figures 8 and 9.

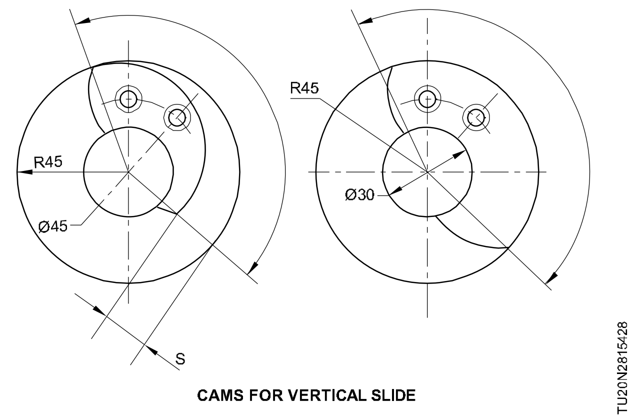
A single cam to control the various slide movements is a complicated one for designing and manufacturing. Whereas, when one cam is made to actuate one slide, the design part of it becomes easier. Only that particular cam may be replaced in case a defect is noted in the component. Moreover a set of cams designed for a particular component may be useful for the manufacture of components with the same shape and design, but with slight dimensional variations.

Fig 7



CAMS FOR CROSS SLIDE

Fig 8



CAMS FOR VERTICAL SLIDE

Differences between centre lathe and CG & M lathe

	Centre lathe	**CG & M lathe** (Capstan and turret lathe)
1	Single-way tool post holds only one tool.	Four-way tool post accommodates different types of tools in four ways.
2	At the end of every operation, the tool needs to be changed. It requires more operation time.	Pre-set tool can be indexed. It avoids tool setting time.
3	Not suitable for CG & M jobs.	Suitable for CG & M jobs
4	Most suitable for tool room jobs.	Most suitable for mass CG & M jobs.
5	High skill is necessary to operate a centre lathe	A semi-skilled person can operate, pre-set turrent of a capstan or turret lathe
6	Combination cuts are employed with difficulty.	Combination cuts can be given both on carriage and turret head simultaneously.
7	By using a single point tool, various sizes of thread can be cut	Tap and dies are used to form threads; sometimes short lead screw is provided to cut threads with chasers
8	Tailstock is provided to hold cutting tools like drills and support the job with centres.	Turret is provided to hold six or more tools
9	There is no separate bed over the main bed for the tool movement	There is an auxiliary bed provided for CG & M (capstan) for the auxiliary slide.
10	Job cannot be fed continuously after completion of operation.	Job can be fed by a bar -feed attachment
11	Time taken is more for producing a workpiece.	Time taken is less for producing a workpiece
12	The cost of machine is low.	The cost of machine is high.
13	Trips and stops are not provided for stopping the tool slides.	Trips and stops are provided for rapid stopping and starting
14	Generally chuck work and in between centre work can be carried out.	Generally bar works can be carried out.
15	The manufacturing cost of components is more.	The manufacturing cost of components is comparatively lower.

Tool-holding devices

The wide variety of work performed in a capstan lathe in mass CG & M necessitated designing of different types of tools and tool-holders for holding the tools for typical operations. The tool-holders may be mounted on turret faces or on crossslide tool post, and may be used for holding tools for bar and chuck work. Certain tool- holders are used for holding tools for both bar and chuck work while box tools are particularly adopted in bar-work. In the capstan or turret lathe practice, the whole assembly of the holder and its tool is designated according to the type of the holder. Thus a slide tool holder with the tool mounted in it is called a 'Slide tool' and a knee tool-holder with the tool fitted into it is called a 'Knee tool'. Special tool-holders are also sometimes designed for special purposes. The most important and widely used tool-holders are listed below.

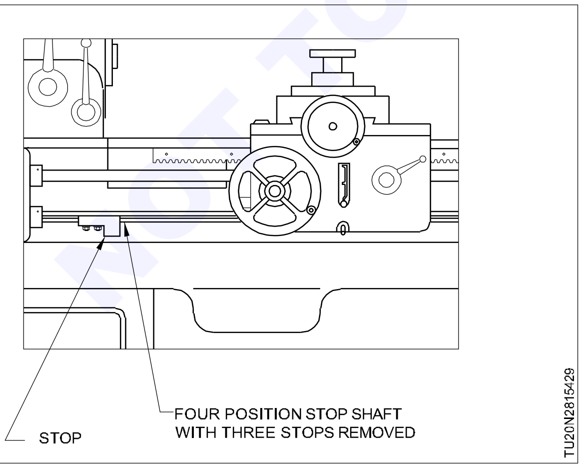
- Straight cutter-holder
- Plain or adjustable angle cutter-holder
- Roller steady turning tool-holder.
- Recessing tool-holder.
- Knee turning tool-holder.
- Stop and centre drill-holder.
- Combination tool-holder.

Stops and trips

One of the features enabling the capstan and turret lathes to produce quantities of similar parts is the system of stops and trips for controlling the movements of the cutting tools.

The stops for controlling the travel of the saddle, and of the turret slide of a turret lathe, are carried on a shaft suspended on brackets from the front of the machine bed. An illustration of this, as applied to the saddle on a small capstan lathe, is shown in Fig 9, where the stop shaft is at the bottom of the illustration.

Fig 9



Provision is made for four stops round the shaft so that the travel of four tools may be controlled at any particular time. It is arranged that the stop first trips out the feed and then serves as a dead stop for the small handoperated movement necessary to complete the travel. For bringing each stop in position, when all the four are being used, the shaft is rotated by hand. On some of the more elaborate turret lathes the stop shaft for the turret faces has six sides with stops, and is automatically rotated to synchronize with the turret.

A similar principle is adopted in respect of the cross - slide stops on most machines. A rotating bar is mounted on the side of the cross-slide and carries four adjustable stops at each end. Hand rotation of the shaft is used to bring the stop at each end and to position for any particular tool. To prevent swarfs from getting in the unit, it is generally provided with a cover.

Cutting speed

Similar to a centre lathe, the cutting speed in a capstan or turret lathe is the rate at which any point on the work passes over the tool, and this is expressed in metres per minute.

Feed

It is the amount the tool moves per revolution of the work. This is expressed in mm per revolution.

Depth of cut

It is the perpendicular distance measured between a machined and an un-machined surface.

Bar-feeding mechanism

The capstan and turret lathes while working on bar work require some mechanism for bar-feeding. The long bars which protrude out of the headstock spindle require to be fed through the spindle up to the bar-stop after the first piece is completed and the collet chuck is opened. In simple cases, the bar may be pushed by hand. But this process unnecessarily increases the total operational time, because the spindle and the long

Principal parts of capstan and turret lathes

The capstan lathe has essentially the same parts as the engine lathe except that the turret complex mechanism incorporated in it makes it suitable for mass CG & M work. The different parts of a capstan lathe are shown in Figs 10,11 and 12.

1 Headstock
2 Cross-slide tool post
3 Hexagonal turret
4 Saddle for auxiliary slide
5 Auxiliary slide
6 Lathe bed
7 Feed rod
8 Saddle for crossslide
9 Feed stop rod for turret
10 Hand wheel for turret slide.

Fig 10

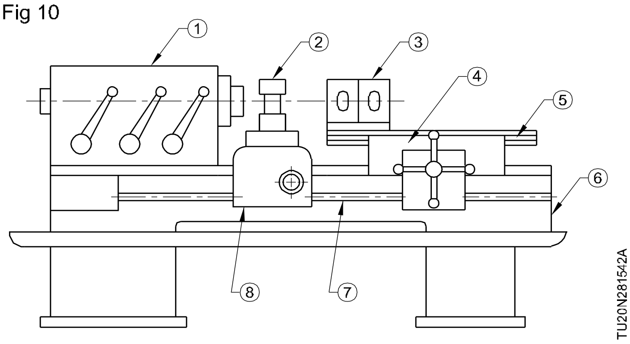


Fig 11

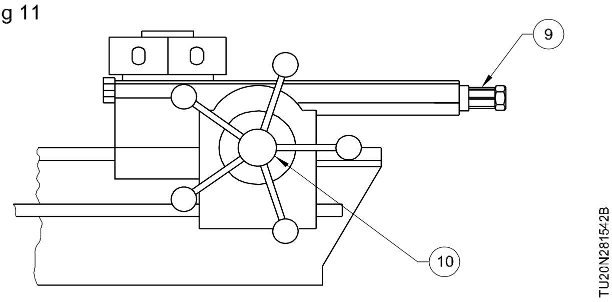
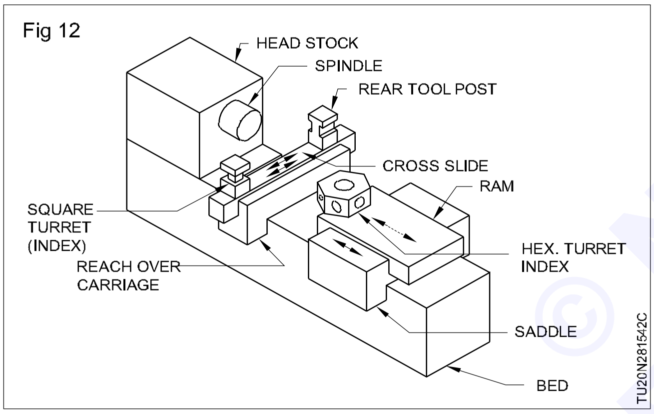


Fig 12



A capstan lathe is specified by providing the following features

- The maximum diameter of rod that can be passed through the headstock spindle.
- The swing diameter of the work that can be rotated over the lathe bedways.
- The minimum and maximum spindle speeds.
- Number of feeds available to the carriage and turret saddle.
- The capacity of the motor. (H.P. of the machine)

The capstan lathe: construction

A capstan lathe consists of a lathe bed, headstock, turret head or slide, saddle, carriage, cross-slide, tool posts and feed rod.

The lathe bed of a capstan lathe is quite similar in construction to that of a centre lathe, the only difference is in the ways of the bed. The bed ways are provided flat instead of in a combination (Vee and Flat) to receive heavy loads during multiple cutting.

Generally an all-geared headstock is used for rapid change of the spindle speeds. This should be selected prior to an operation by changing the lever or pressing the speed-changing button. The spindle is hollow and the bar-stock should be passed through it.

On the right side of the capstan lathe there is a hexagonal turret head mounted on a vertical spindle. The turret face has a bored hole on the centre which receives lengthy tools like drills, reamers, holders for tools and split bushes.

A star wheel is provided in front of the machine which operates manually for the to and fro motion of the turret head assembly. Power feed is also provided for the turret head with suitable feed rate. The turret head is fully retracted, and at the end of its travel a turret indexing mechanism is automatically operated for the next position of the tool. Each movement of the tool will be controlled by adjustable stops. Each stop is set for correct travel of the tool movement, which will stop further movement of the tool.

The type of the cross-slide depends upon the size of the machine. The slide is positioned in between the headstock and turret head over the saddle. If it has two tool posts, one is at the front and the other is at the rear end of the slide. The front tool post is moving inward to bring the cutting tool into operation. The rear tool post is moving inward when the front tool post is moving outward. The rear tool post actually does the ending operation like parting off, with or without chamfering. All the lateral movements of the saddle and the cross - slide will be controlled by trips and stoppers for correct movement of the tools in operation.

The feed-rod controls the motion for the carriage in the same way as in an engine lathe.

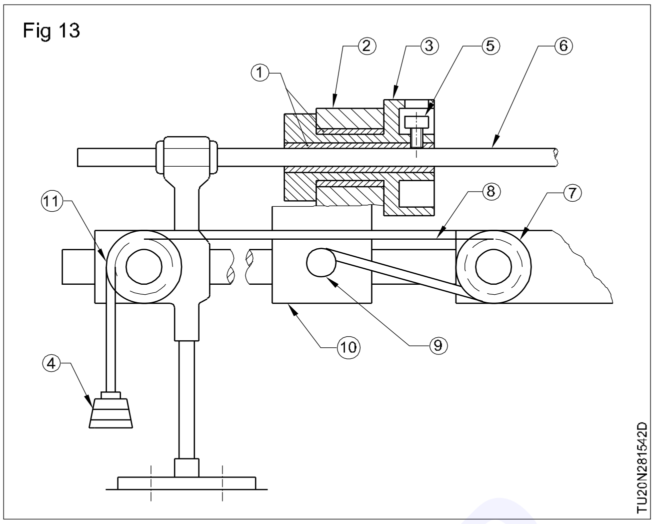
The stop-rod is mounted below the bed in between the gearbox housing and feed-rod housing. It has four splines over the periphery to accommodate the stoppers in measuring positions with respect to the related turning operations for the saddle movements.

bar must come to a dead stop before any adjustment can be made. Thus in each case unnecessary long time is wasted in stopping, setting and starting the machines. Various types of bar-feeding mechanism have, therefore, been designed which push the bar forward immediately after the collet releases the work without stopping the machine, and thereby enable the setting time to be reduced to the minimum.

Working principle of the bar feeding mechanism (Fig 13)

The bar 6 is passed through the bar-chuck 3, through the spindle of the machine and then through the collect chuck. The bar-chuck 3 rotates in the sliding bracket body 2 which is mounted on a long slide bar. The bar chuck 3 grips the bar centrally by two set screws 5 and rotates with the bar in the sliding bracket body 2. One end of the chain 8 is connected to the pin 9 fitted on the sliding bracket 10 and the other end supports the weight 4, the chain running over to fixed pullys 7 and 11 mounted on the slide bar. The weight 4 constantly exerts an end thrust on the bar-chuck while it revolves on the sliding bracket and forces the bar through the spindle, the moment the collet chuck is released. Thus bar-feeding may be accomplished without stopping the machine.

Fig 13



Turret heads

Objectives : At the end of this lesson you shall be able to
- **state the turret head**
- **list the types of turrets**
- **state the constructional and functional features of a turret head.**

The turret head

The turret head is also known as a tool head. The turret is a six sided (hexagonal) block mostly and it is carried on the bed of the machine for accommodating and bringing forward the tools. Each of the faces of the hexagon is accurately machined, square to the axis of the lathe and has four tapped holes to accommodate the clamping bolts of the tool-holders or attachments. At the centre of each face is a through hole into which the shanks of the tool-holders may be accommodated and clamped. A pad bolt is often employed for the purpose of clamping. (Fig 1)

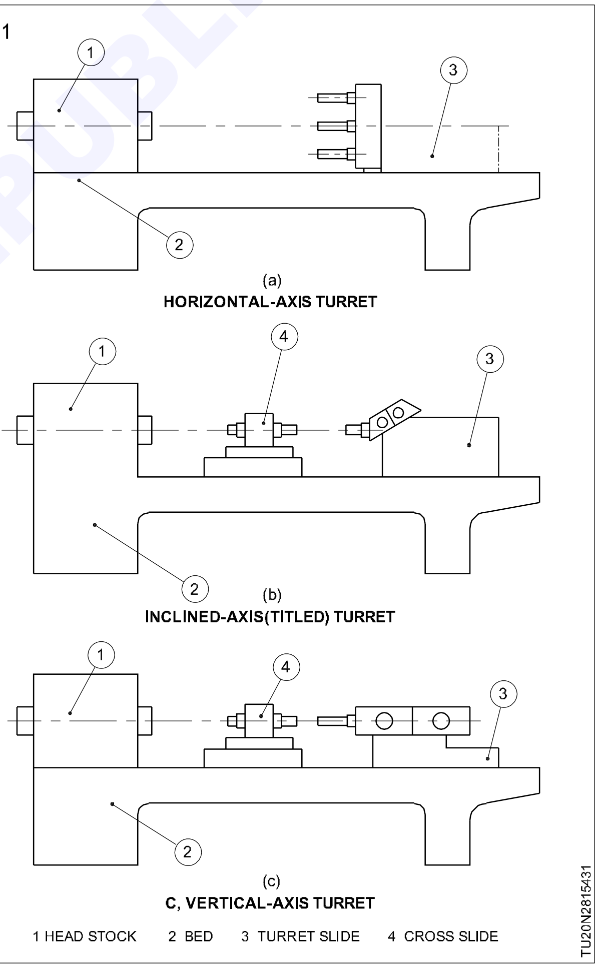
Smaller types of capstan lathes have the turrets not hexagonal but circular. They have six holes for accommodating the tools. During the cycle of operation it is necessary to bring each tool into position for its work by indexing and locating the turret to the successive position.

On the turret lathe, the turret head is supported on a free bearing. The turret can be pulled round easily by hand when the clamping arrangement has been released.

The turret is located in each of the six correct positions by some hand-operated plunger. The machine operator releases the clamp and locates the plunger before indexing the turret manually.

On the capstan lathe, means are provided whereby the turret is automatically indexed to its next position, when it reaches the extreme end of its withdrawal movement from the previous operation.

Fig 1



(a)
HORIZONTAL-AXIS TURRET

(b)
INCLINED-AXIS(TITLED) TURRET

(c)
C, VERTICAL-AXIS TURRET

1 HEAD STOCK 2 BED 3 TURRET SLIDE 4 CROSS SLIDE

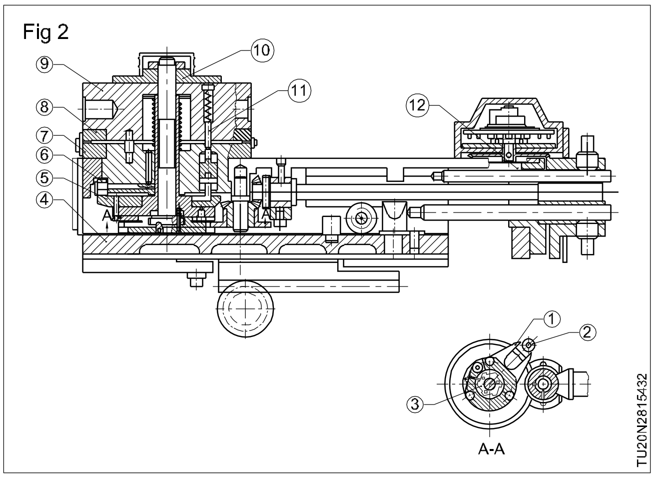
Types of turret heads

Turret heads are classified according to the axis of rotation which may be horizontal, vertical or inclined.

Turret lathes with turret heads of vertical and inclined axis are provided with cross-slides whereas those with horizontal axis turrets have no cross-slides since the cross feed of the tool is effected by the turret slide itself.

Constructional features of a turret head with vertical axis (Fig 2)

Fig 2



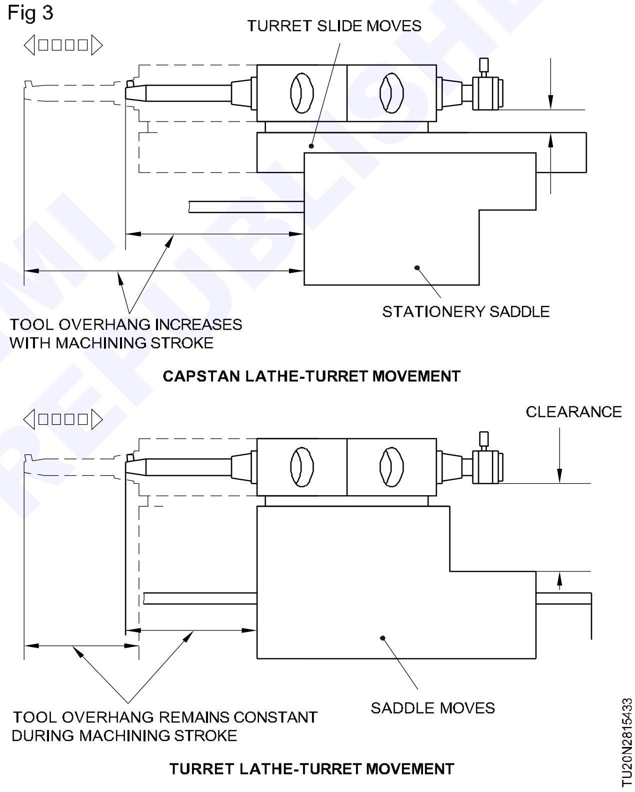
This is movable along the bed-ways. The turret head can be indexed by hand or automatically on the withdrawal of the turret slide. As this takes place, the fork (1) along with the cam (5) turns relative to the fixed stop (2). The cam lifts the turret (9) through rollers (6). The flanges (7 and 8) of the claw clutch, which serve to lock the turret, come out of engagement.

As the fork turns further, the ratchet gearing (3) indexes the turret into the next position.The lock pins (11) prevent the turret from overrun, performing a preliminary locking function. When the slide is advanced the fork (1) turns in the opposite direction, the cam (5) lowers the rollers (6) with turret (9) and flanges (7) and (8) of the claw clutch come back into engagement, finally locking the turret in position. The locking force is adjusted with the nut (10). The stopper roller (12) with adjustable stops, whose number corresponds to the number of the turrets' tool position is indexed through a gear transmission simultaneously with the turret. The stops limit the travel of the turret relative to the saddle (A). The stops are adjusted when the machine is set up for operation.

In the capstan lathe turret head stops are fitted in the saddle. The overhang increases as the turret moves towards the workpiece held in the headstock. This overhang limits the strength and consequently the capacity of the machine. In the turret lathe the overhang is constant, permitting a heavier work load. Slide stops in this type are fitted in the lathe bed. (Fig 3)

In each machine the sliding movement of the turret head is made by operating a handwheel. This turns the pinion which mashes with the rack in the bed. Movement of the turret head back from the headstock automatically indexes the turret.

Fig 3



CAPSTAN LATHE-TURRET MOVEMENT

TURRET LATHE-TURRET MOVEMENT

Roller box turning tools

Objectives : At the end of this lesson you shall be able to
- **state the purpose of roller box turning tool**
- **list the types of roller box turning tools**
- **state the constructional features of each type**
- **state the functional features of the each type.**

Roller box turning tools

Plain turning operations may be performed on capstan and turret lathes by feeding the tool

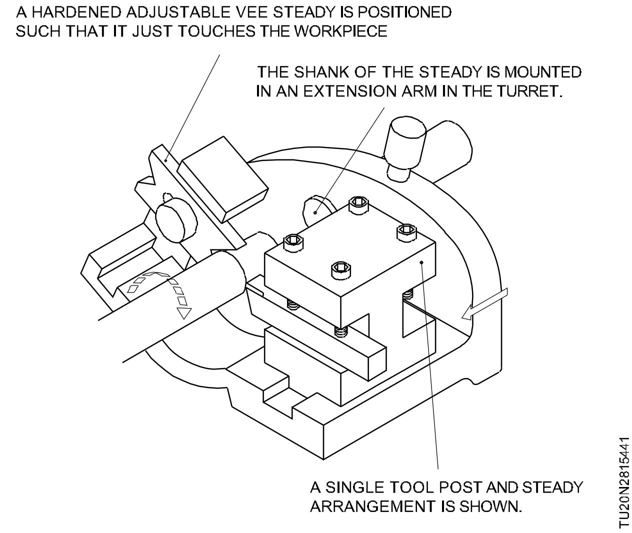
- from the hexagonal turret
- by the indexing tool post on crossslide
- by feeding tools from both.

Turning operations carried out from the hexagon turret provide rigid support to the tool for making heavy cuts. Tools used in the turret are of special design and are referred to as box turning tools. Box turning tools are of two types.

- `V' steady turning tools
- Roller steady turning tools

`V' Steady tool–holder (Fig 1)

Fig 1



These holders are for turning brass and for making light cuts on other free cutting materials. A short length of the workpiece is hand–machined to the required diameter and the steady is adjusted to this surface.

The cutting tool is set slightly ahead of the Vee steady. These tools may have single or multiple tool blocks.

A hardened adjustable Vee steady is positioned such that it just touches the workpiece.

The shank of the steady is mounted in an extension arm in the turret.

Where good surface finish is needed on the turned portion, it is preferable to have the cutting tool in advance to the Vee steady, and where concentricity is of prime importance, the cutting tool may be set behind the Vee steady.

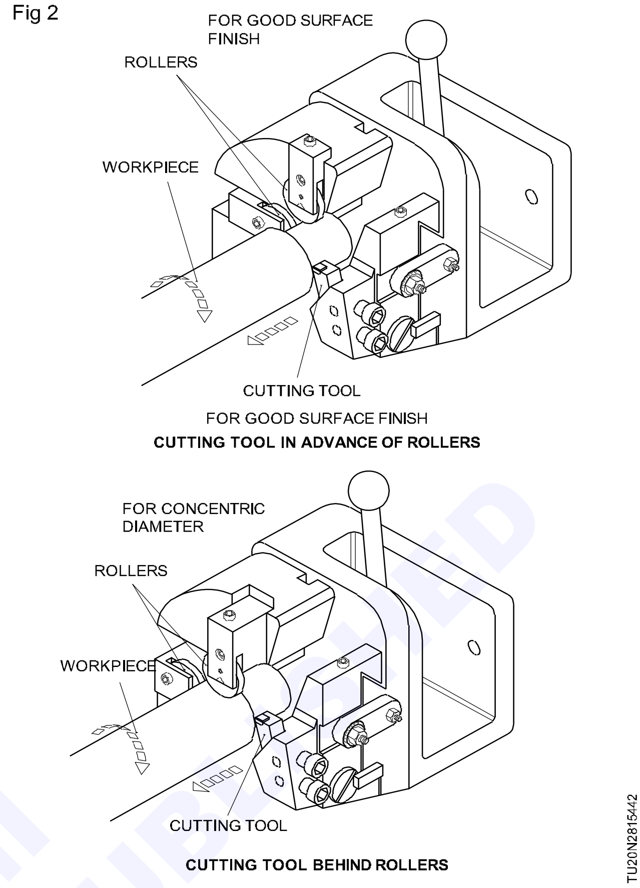
The roller steady tool-holders (Fig 2)

These holders are used for most turning operations. Each holder has a limited size capacity. A short length of the work is machined to the correct diameter. The rollers and tool are set on this diameter. The cutting action forces the workpiece against the rollers, producing a high burnished finish on the work. The cutting tool is set in advance of the rollers for most turning operations. This set up produces a good surface finish. Some steadies are available with rollers set ahead of the cutting tool.

The roller steady tool-holders are of two types.

- The tangential type (Fig 3)
- The ending type (Fig 4)

Fig 2



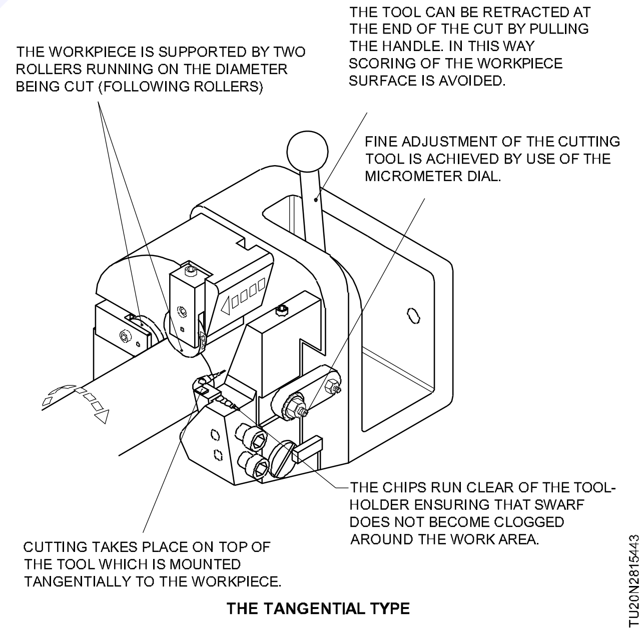
FOR GOOD SURFACE FINISH

CUTTING TOOL IN ADVANCE OF ROLLERS

CUTTING TOOL BEHIND ROLLERS

In the tangential type the workpiece is supported by two rollers running on the diameter being cut (following rollers). The tool can be retracted at the end of the cut by pulling the handle. By this, scoring of the workpiece surface turned is avoided. Fine adjustment of the cutting tool is achieved by the use of the micrometer dial. The chips flow clear off the tool-holder ensuring that swarf does not become clogged around the work area. (Fig 3)

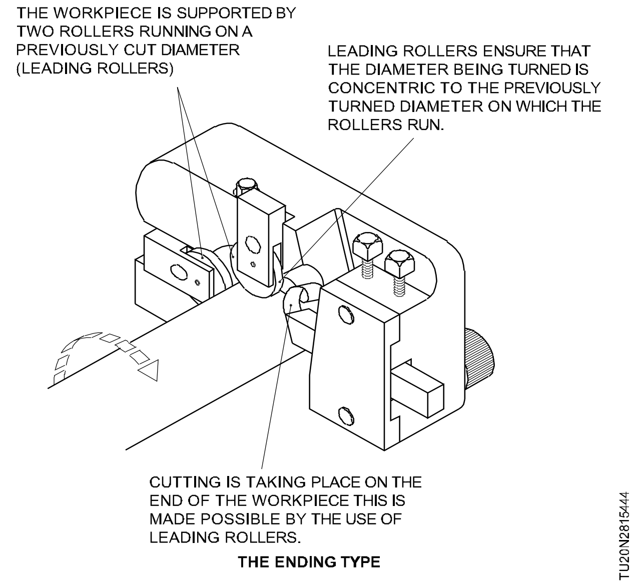
Fig 3



THE TANGENTIAL TYPE

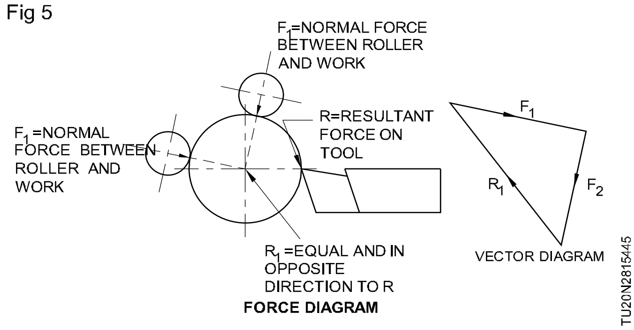
In the ending type shown in figure 4 with leading or forward rollers, the workpiece is supported by two rollers on a previously cut diameter. Leading rollers ensure that the diameter being turned is concentric to the previously turned diameter on which the rollers run.

Fig 4



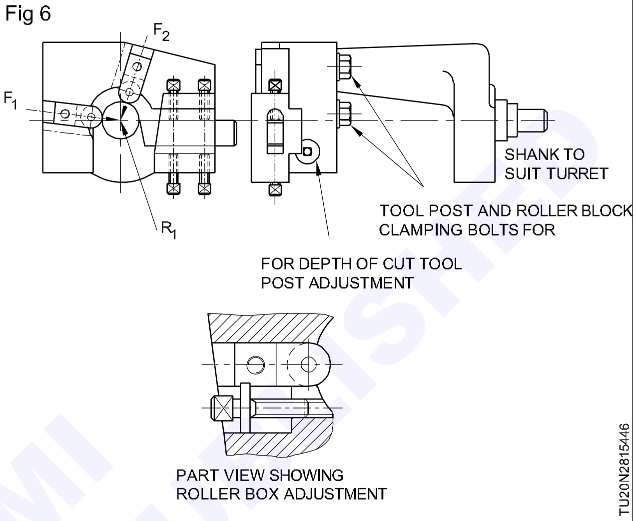
THE ENDING TYPE

Fig 5



FORCE DIAGRAM

Fig 6



The leading rollers may not produce the same standard of surface finish.

When using the ending type of roller steady tool-holder, the cutting is taking place on the end of the workpiece. This is made possible by the use of the leading rollers. (Fig 4)

Figure 5 shows the geometry of a roller box showing forces on the tool and the rollers.

Figure 6 shows the roller steady tool-holder adjustments. The forces F1, F2 and R1 are also illustrated.

Turret lathes

Objectives: At the end of this lesson you shall be able to
- **specify a turret lathe**
- **name the various attachments & accessories used on turret lathes**
- **distinguish between horizontal and vertical types of turret lathes.**

Specification of a turret lathe

A turret lathe is specified by giving the

- maximum diameter of a rod that can be held
- maximum spindle hole diameter
- maximum swing over the bed
- number of spindle speeds
- number of feeds (carriage) cross-slide and longitudinal
- number of feeds (turret carriage)
- power required for machine motor and coolant motors.

Accessories / attachments

Work-holding accessories

Self centering chuck, independent chuck, combination chuck, air operated chuck, push-out type collet chuck, draw in type collet chuck, dead length type collet chuck.

Turret : Pentagon, hexagon or octagon type.

Tool-holders

Straight cutter holder

Plain or adjustable angle cutter holder

Multiple cutter holder

Offset cutter holder

Combination tool-holder or multiple turning head

Slide tool-holder

Knee tool-holder

Drill holder

Boring bar holder or extension holder

Reamer holder

Knurling tool-holder

Recessing tool-holder

Form tool-holder
- straight
- circular

Tap-holder

Die-holder

Balanced tool-holder (box tool)

- V-steady box tool-holder
- roller box tool-holder

Bar ending tool-holder

Attachments

Taper turning attachment

Thread cutting attachment

Horizontal turret lathes are classified as:

- ram type turret lathes
- saddle type turret lathes
- chucking machine.

Saddle type turret lathes (Fig 1a)

The hexagonal turret is mounted directly on a saddle and the whole unit moves back and forth on the bed ways to apply feed. This type of turret lathe is heavier in construction and is particularly adopted for larger diameter bar works and chuck works. The machine can accommodate longer workpieces.

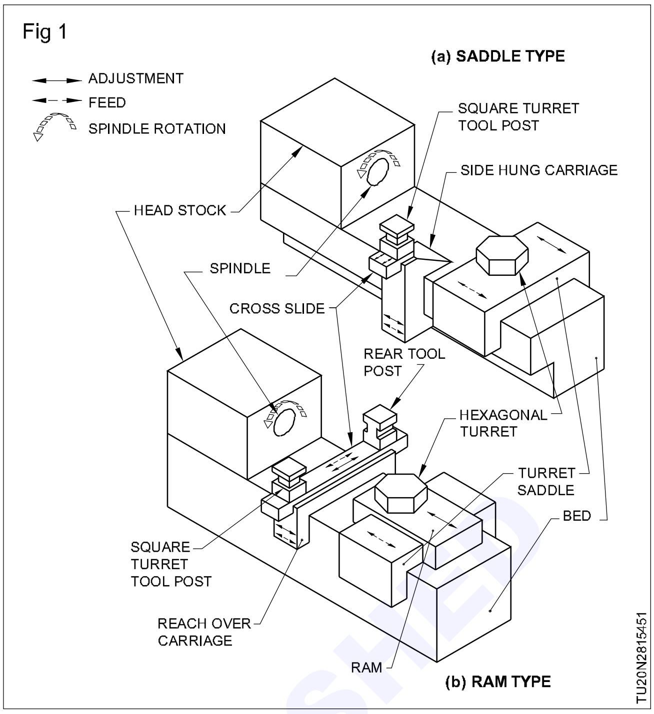
Ram type turret lathes (Fig 1b)

The ram type turret lathe carries the hexagonal turret on a ram or a short slide. The ram slides longitudinally on a saddle positioned and clamped on lathe bedways. This type of machine is lighter in construction and is suitable for machining bars of smaller diameters. The tools are mounted on the square turret and the six faces of the hexagonal turret. The feeding movement is obtained when the ram moves from the left to the right. When the ram is moved backward the turret indexes automatically and the tool mounted on the next face comes into operation.

Chucking machine

The chucking machines are used to machine castings and forgings that must be held in chucks or fixtures. This type of machine handles a large variety of work which is basically the same as that done on standard turret lathes equipped for chuck work.

Fig 1



Vertical turret lathes

A vertical turret lathe is a machine with a vertical turret head and a horizontal work table. Most of the operations performed on a regular turret lathe are also done on this type of machine. It is preferred for large and heavy work.

The principal parts of a vertical turret lathe are the

- base
- column
- rail
- table
- saddle and turret head
- side head

Many attachments are used on vertical turret lathes.

Dead stops and trips on capstan and turret lathes

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

- **state the necessity of stops and trips**
- **list the types of stops and trips**
- **state the constructional and functional features of stops and trips**
- **types of stop and trip arrangements.**

Stop and trips

Stops and trips are meant for controlling the movement of the cutting tool on the crossslide and on the hexagonal turret. They serve the purpose of providing dimensional accuracy on identical components produced in large quantities. They minimize delays for measuring and gauging. The stops for controlling the travel of the saddle and of the turret of a turret lathe are carried on a shaft suspended on brackets from the front of the machine bed. It is arranged that the stop first trips out the feed, and then serves as dead stop for the small handoperated movement necessary to complete the travel. For bringing each stop into position, when several of them are being used, the shaft is rotated by a hand wheel provided. On some of the most elaborate turret lathes the stops for saddle movement, which are four in number are provided on a shaft below the headstock. (Fig 1) The stop shaft for the turret has six stops which are automatically rotated to synchronize with the turret. (Fig 2)

Fig 1

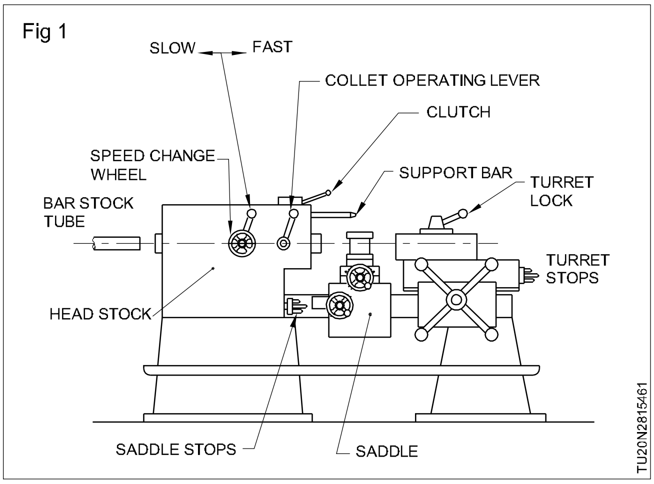
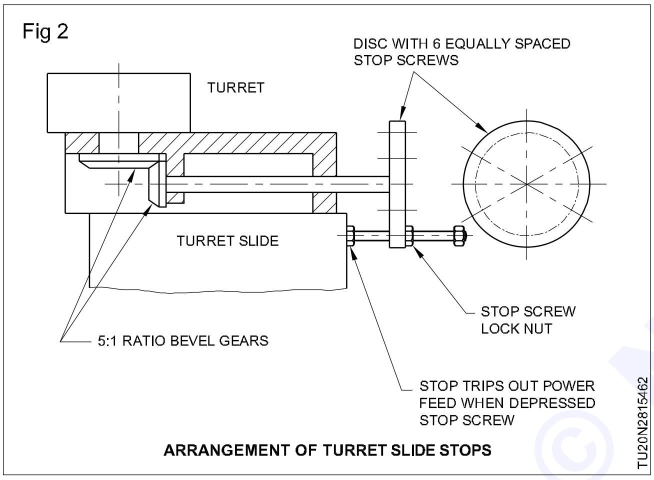


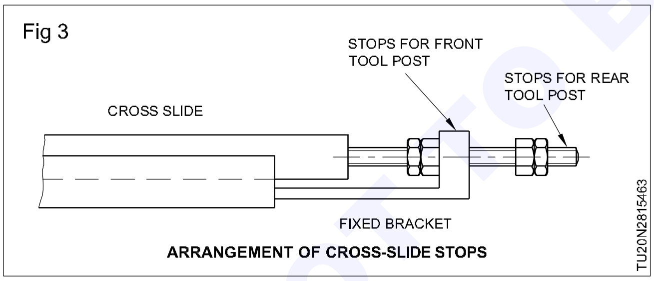
Fig 2



ARRANGEMENT OF TURRET SLIDE STOPS

The cross-slide is also provided with a shaft with stops to control the depth of cuts of each tool on the cross-slide tool post. (Fig 3)

Fig 3



ARRANGEMENT OF CROSS-SLIDE STOPS

Finger stops

A finger stop is positioned radially on the hand wheel and is aligned with a mark on the graduated scales when the relevant dimension on the workpiece is down to size. There may be three or four finger stops, each aligned to a different mark on the graduated scale. (Fig 4)

Feed trips

A feed trip switches off the power feed to the tool when the workpiece is almost to size. When the trip device touches the stop, it trips the power feed. A further manual movement of the slide, through a previously determined distance will bring the slide to the dead stop as shown. (Fig 5)

Fig 4

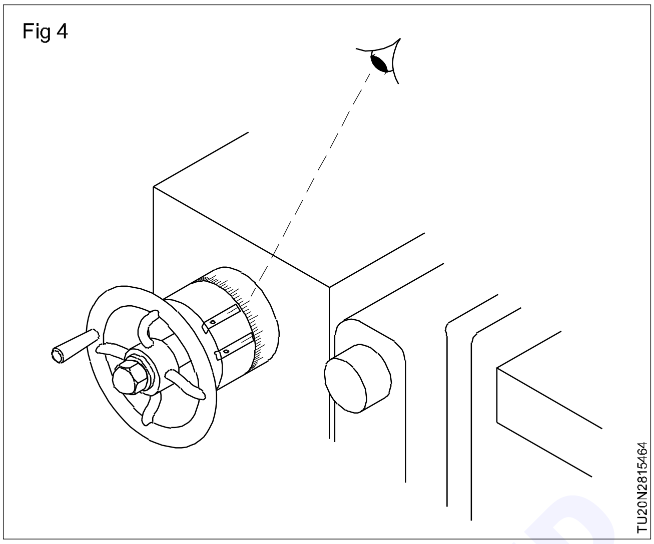
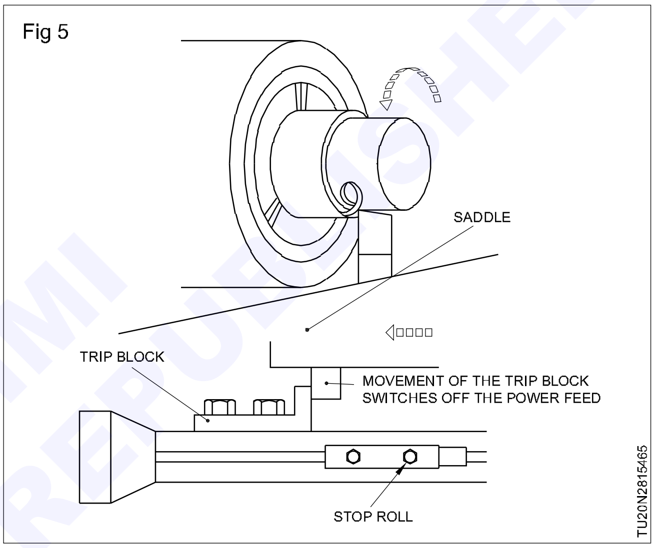


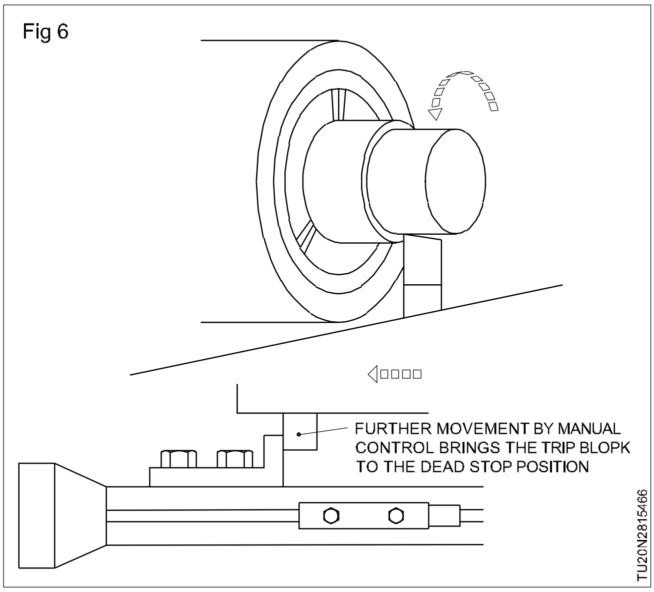
Fig 5



Dead stops

A dead stop terminates the tool travel at a point where a workpiece diameter or depth is correct to the drawing specification. (Fig 6)

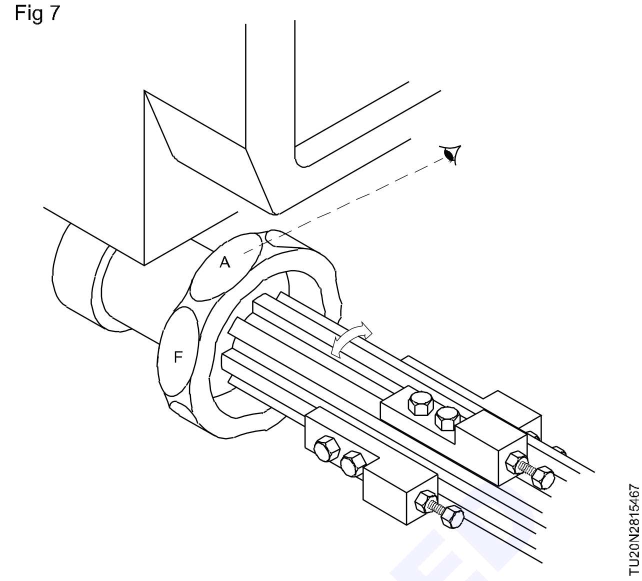
Fig 6



Indexing a stop-bar or roll

A `stop-bar' is indexed to the position in conjunction with the relevant tool. This may be carried out manually by indexing the bar to present a letter, a record being made of which letter goes with each tool. In the case of a turret, the stop-bar will be indexed automatically with the turret and, therefore, each stop is always presented for the same turret station. (Fig 7)

Fig 7



Collets used on capstan and turret lathes

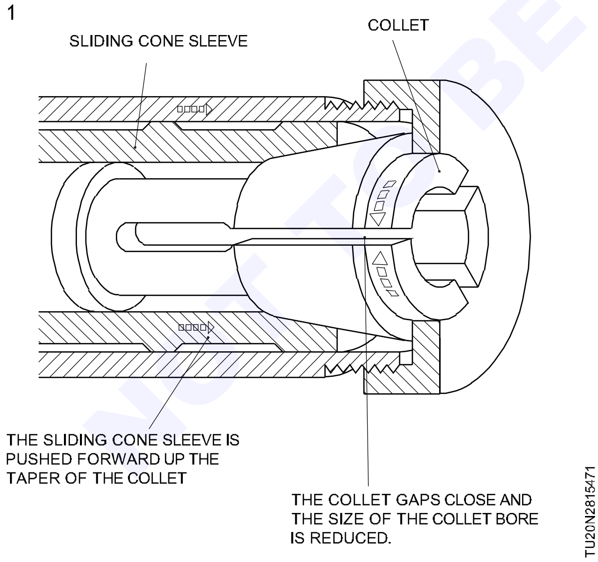
Objectives : At the end of this lesson you shall be able to
- **list the collets used for bar work on the capstan and turret lathes**
- **state the constructional and functional features of each type of collet.**

The collet chuck

Collets provide a means of holding cold-drawn bar materials or previously machined parts to undergo a second operation. They maintain an accurate alignment of the component to be machined with the rotating spindle axis of the lathe.

The working principle of the collet chuck is illustrated in Fig 1.

Fig 1

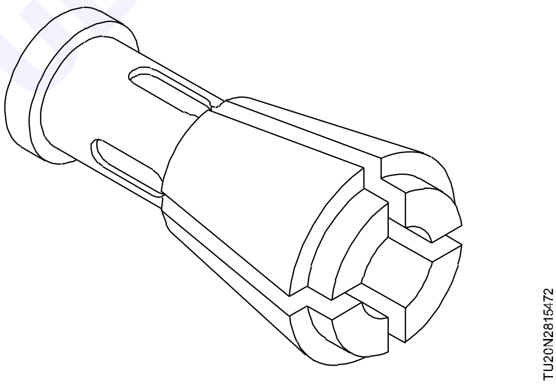


The common constructional features of collets have already been dealt with in the previous exercises.

Types of collets

Dead length type (Fig 2)

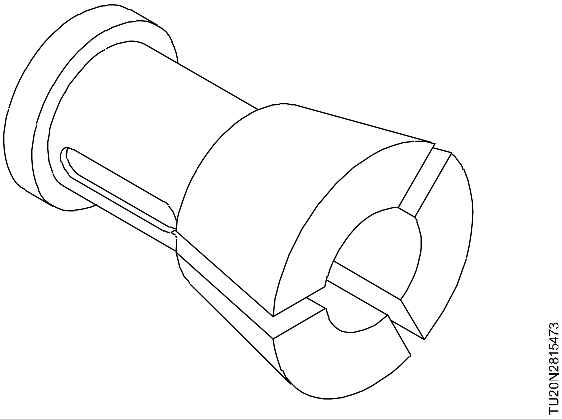
Fig 2



The workpiece does not move when this type of collet is operated; and, therefore, an accurate length of work-piece can be maintained.

Draw back type (Fig 3)

Fig 3

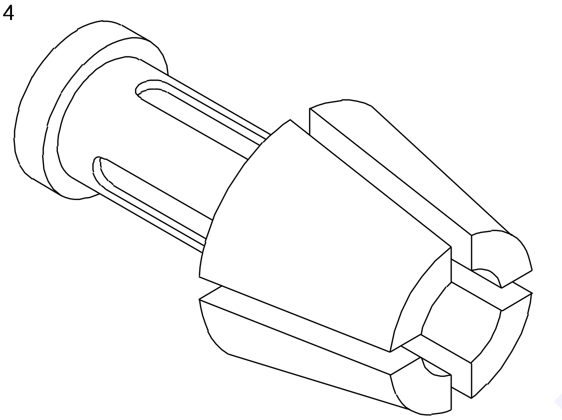


This is also referred to as draw in collet. This collet is usually screwed to a draw bar or tube which draws the collet into the sleeve causing it to grip the bar. A keyway located in the body of the collet accommodates the key in the sleeve preventing rotation of the collet, which ultimately prevents the collet getting unscrewed from the draw bar during operation. This type of collet is not suitable for accurate length workpiece as it tends to pull the workpiece away from the bar-stop when closing.

Pushout type (Fig 4)

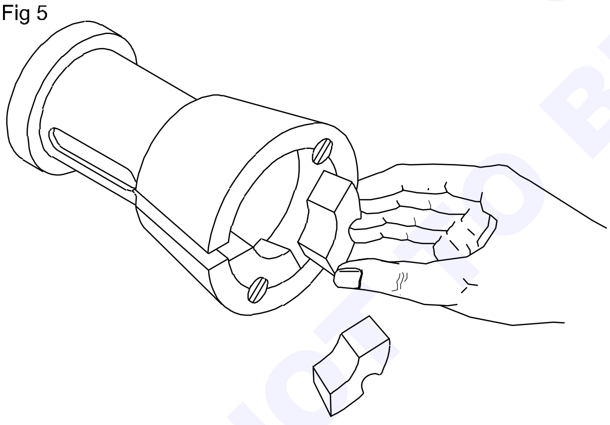
When this type of collet is closed by pushing against the taper of the accommodating sleeve, it tends the bar also to be pushed along with the collet against the bar-stop. Unless the turret hand wheel is held by pressing in the anticlockwise direction, the component length may show variations.

Fig 4



Master collet with interchangeable liners (Fig 5)

Fig 5

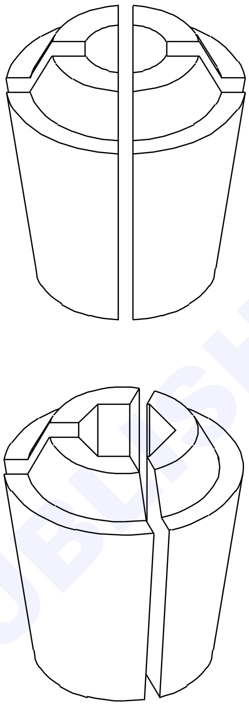


The master piece is of collet shape. This has a tapered bore into which liners or pads, mostly three in number, can be fitted. When the pads are fitted to the internal taper of the master piece, the inner bore that results by the assembly of the pads accommodates the bar. The masterpiece controls the grip. The pads are assembled to the master piece by means of stop screws.

The collet pads are used in most of the larger sizes of bar work.

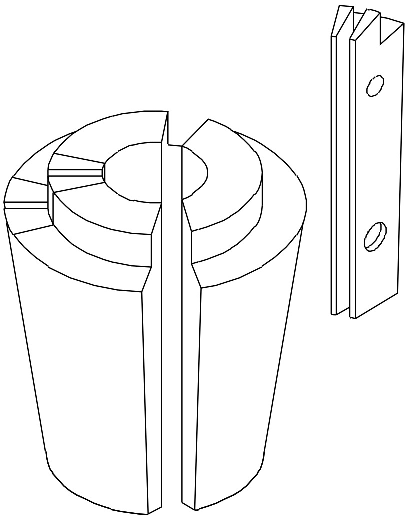
Their range of adjustment is much greater than with solid collets, usually about 1.5 mm. The collet pads are usually not suitable for gripping short workpieces. These pads are available in sets of three or four pieces. The pads are available to accommodate hexagonal bars also. When hexagonal bars are held, the bar must be rotated after inserting so as to get the corners properly engaged. (Fig 6)

Fig 6



Sometimes the collet pads are provided with spring spacers to keep the collet pads apart when the chuck is opened making the workpiece loading easier. (Fig 7)

Fig 7



Points to be remembered during selecting a solid collet

The above type of collets accommodates only one size of bar for which they are manufactured. If the bar diameter differs by even 0.1 mm from the nominal size, the operation of the collet will not be proper and the accuracy of the collet will be lost.

Clean and deburr the workpiece and form a good chamfer on the end face of the bar which enters the collet.

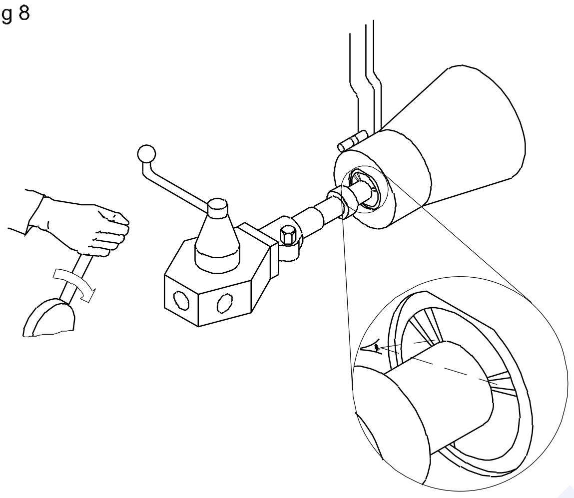
Check that the collet just fits on the workpiece without forcing.

A round bar having a scaly unmachined outer surface must never be held in a collet chuck.

Checking the collet tension

Index the bar-stop into position and apply force to the bar using the star wheel of the turret slide. (Fig 8)

Fig 8



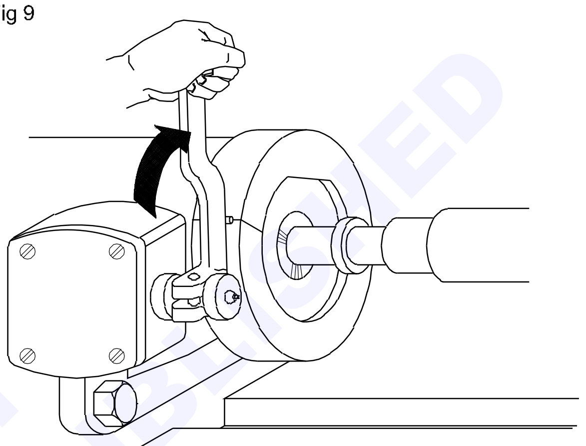
If the bar moves under pressure, open the chuck and readjust the knurled cap.

Feed for correct adjustment on a manually operated chuck, by feeling for resistance when closing the chuck. (Fig 9)

If long heavy bars are to be held, put the machine in motion and check that the bar does not move in the collet when accelerated up to the cutting speed.

When soft materials and thin walled tubes are to be held, check that no excessive damage has been caused by the grip of the collet to the surface of the soft bar or the diameter of the thin tube.

Fig 9



Turret lathes

Comparison between turret and capstan lathes

Turret lathe	Capstan lathe
1 The turret is mounted on the saddle which slides directly on the bed.	The turret is mounted on an auxiliary slide which slides on the saddle.
2 The construction provides utmost rigidity to the tool support as the entire cutting load is taken up by the lathe bed directly.	The auxiliary bed feeds the tools into the work. The overhanging of the auxiliary bed from the stationary saddle presents a non-rigid construction which is subjected to bending, deflection or vibration under heavy cutting loads.
3 The turret lathe can operate under severe cutting conditions, accommodating heavier workpieces with high cutting speeds, feeds and depth of cuts.	The capstan lathe can be operated under light cutting conditions, accommodating workpieces with high cutting speeds, feeds and less depth of cuts.
4 The turret lathes are capable of turning bars up to 200 mm in diameter.	The capstan lathes are capable of turning bars up to a maximum of 60 mm diameter only.
5 Larger and heavier works can be done on turret lathes.	Smaller and lighter bar work can be done on a capstan lathe.
6 In the turret lathe, the hand feeding is a laborious process due to the movement of the entire saddle unit.	On a capstan lathe, the hexagonal turret can be moved back and forth rapidly without having to move the entire saddle unit.

The constructional features of a turret lathe

A turret lathe consists of a bed, an all-geared headstock, and a cross slide on which a four-way tool post is mounted to hold four different, single point tools. A tool post fitted at the rear of the cross-slide holds a parting tool in an inverted position. The tool post mounted on the cross-slide is indexed by hand. In a turret lathe there is no tailstock but in its place a hexagonal turret is mounted on a saddle which is sliding on the bed. The six faces of the turret can hold six different tools. The turret may be indexed automatically or manually and each tool may be brought in line with the lathe axis in a regular sequence. The workpieces are held in collets or on chucks. The longitudinal movements of the turret and cross-feed movement of the crossslide are controlled by adjustable stops. These stops enable different tools set at different positions to move by a predetermined amount for performing different operations on repetitive workpieces without measuring the length (or) diameter of the machined surfaces in each case.

These special characteristics of the turret lathe enable it to perform a series of operations such as turning, drilling, boring, reaming, thread cutting, necking, chamfering, cutting off and many other operations in a regular sequence to produce a large number of identical pieces in a minimum time.

Types of turret lathes

Turret lathes can be classified into two main groups

- Horizontal type turret lathe.
- Vertical type turret lathe.

Horizontal type turret lathes

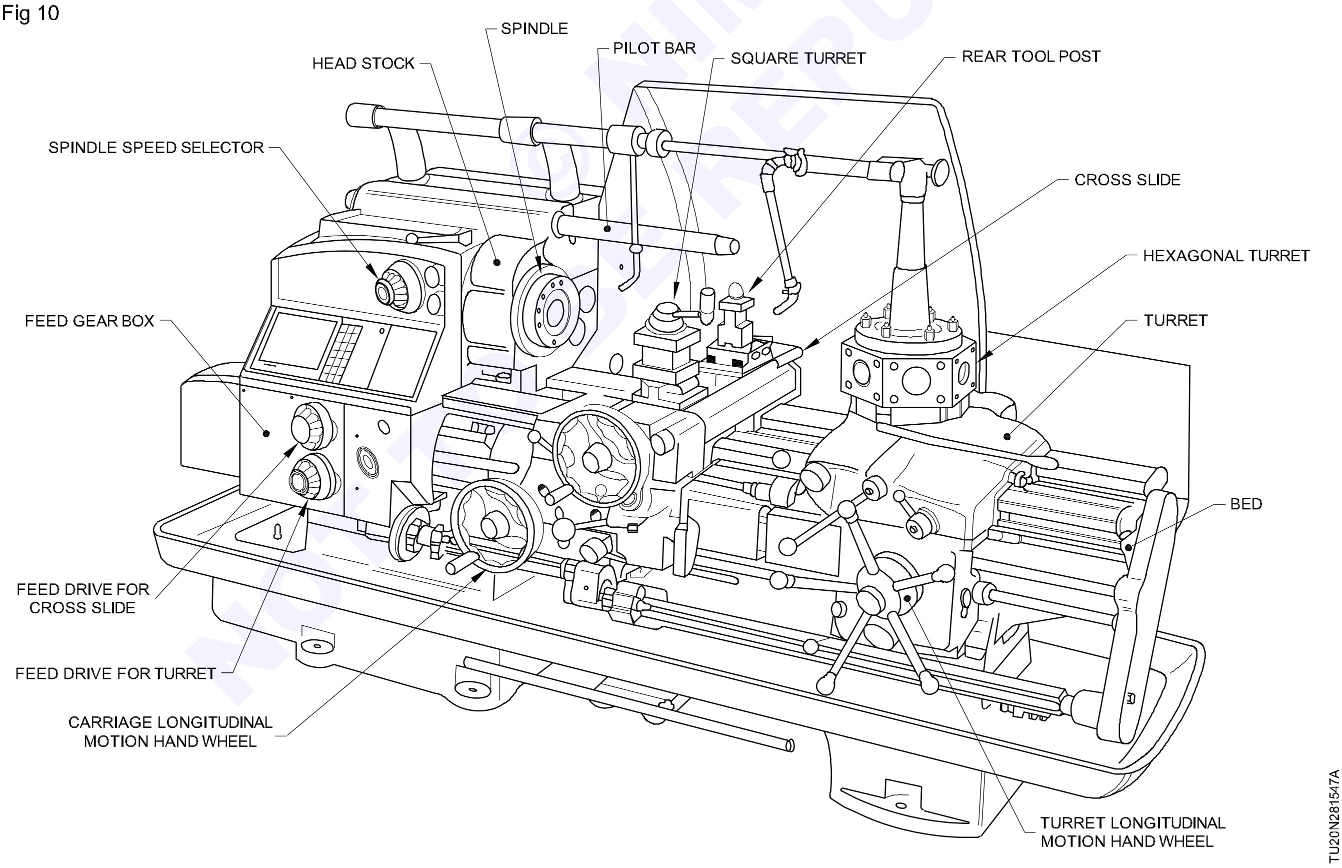
They are lathes which perform all the operations in a horizontal position with the help of a cross-slide tool post and turret head. These operations include turning, drilling, boring, reaming, thread cutting, necking, chamfering and cutting off.

Vertical turret lathes

These are lathes with a vertical turret head and a horizontal work table. All the operations performed on a regular turret lathe can be done on this type of machine also. It is preferred for large and heavy workpieces.

Parts of a horizontal turret lathe. (Fig 10)

Fig 10



Types of tool - holders in capstan and turret

Objective: At the end of this lesson you shall be able to
- **state the different types of tool-holders and their uses.**

The following are the different types of tool-holders.

Single cutter turner tool-holder

Roller steady turning tool-holder

Multiple cutter turner tool-holder

Combination end face and turner tool-holder

Quickacting slide tool-holder

Centre drilling tool-holder and bar-stop

Adjustable knee tool

Die head

Clutch type tap and die-holder

Floating tool-holder

Combination facing and spotting drill-holder

Taper shankdrill socket-holder

Drill chuck (drill-holder)

Combination stock stop and centre-holder

Flanged tool-holder

Knee turning tool-holder

Recessing and boring tool-holder for boring

Recessing tool-holder.

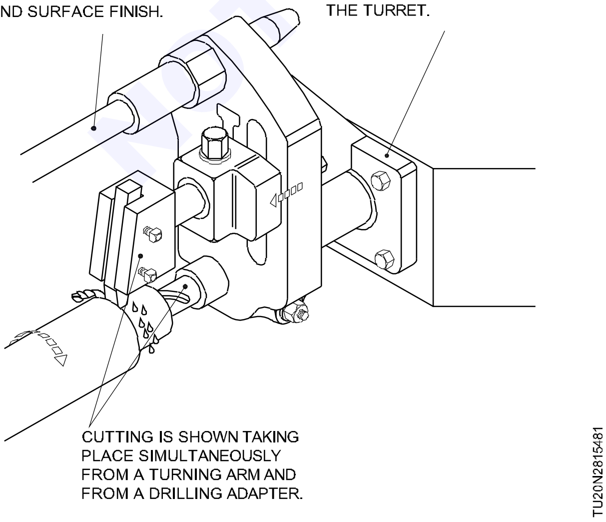
Uses of the common tool holders used on turret lathes

Knee turning tool-holder (Fig 1): A knee turning tool is used while the job is held rigidly in a chuck or fixture. It is used for machining components by combined operations. The rigidity of this tool-holder can be increased by using the overhead support bar protruding from the headstock.

Fig 1

THE ADJUSTABLE PILOT BAR RUNS IN THE PILOT OF THE KNEE TURNING TOOL HOLDER. THIS MAKES THE ARRANGEMENT MORE RIGID RESULTING IN IMPROVED ACCURACY AND SURFACE FINISH.

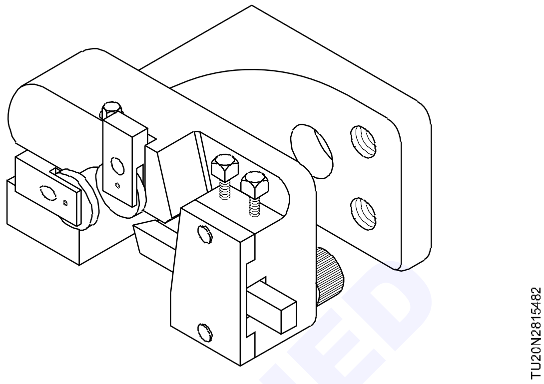
THE KNEE TURNING TOOL HOLDER IS LOCATED AND SECURED TO THE TURRET.



CUTTING IS SHOWN TAKING PLACE SIMULTANEOUSLY FROM A TURNING ARM AND FROM A DRILLING ADAPTER.

Roller steady turning tool - holder (Fig 2)

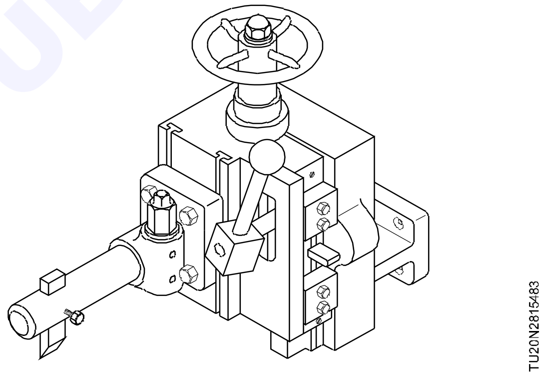
Fig 2



Roller steady turning tool-holder is used mainly for turning external diameters while a fairly good finish is needed on the component. The roller steady supports the job when a lengthier job is to be turned.

Recessing and boring tool-holders (Fig 3)

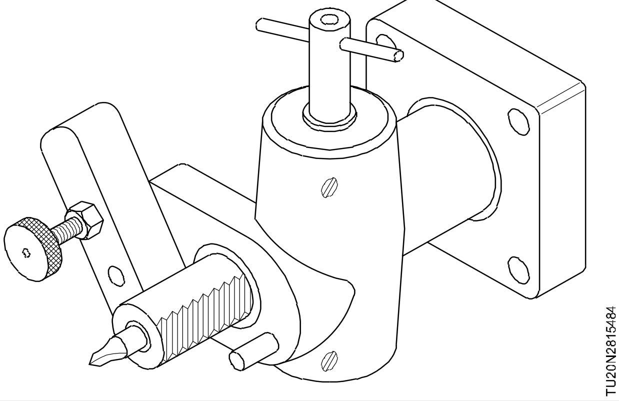
Fig 3



Recessing and boring tool-holders are used in combination for internal recessing and boring.

Bar-stop and centre drill tool-holder (Fig 4)

Fig 4

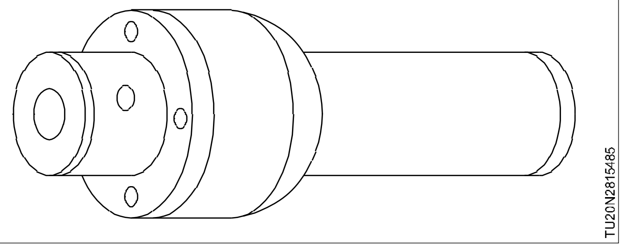


A bar-stop is used for setting the bar stock protruding to the required length to a preset position from the chuck. A centre drill tool-holder is used for centre drilling where drilling is to be done in a job.

Floating reamer-holder (Fig 5)

Where reaming is to be done in a turret lathe, a floating reamer-holder is used to ream a drilled hole. The floating holder facilitates by aligning itself with the axis of the drilled hole without causing much load on the reamer.

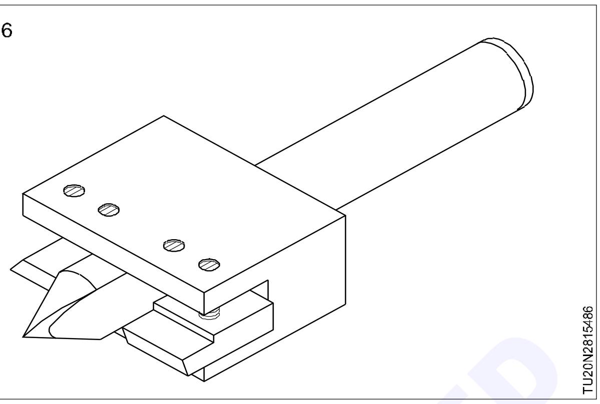
Fig 5



Combination facing and start drilling holder (Fig 6)

Combination facing and start drilling holder is used to face the end of the job and to provide a small hole for short depth ready for a drilling operation.

Fig 6

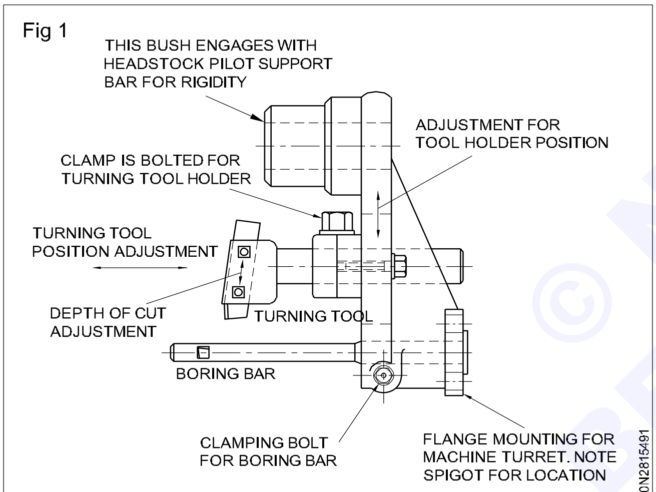


Knee turning - holder and combination tool - holder

Objective : At the end of this lesson you shall be able to
- **state the constructional features of a knee turning tool-holder and a combination tool-holder.**

Knee turning tool-holder (Figs 1&2)

Fig 1

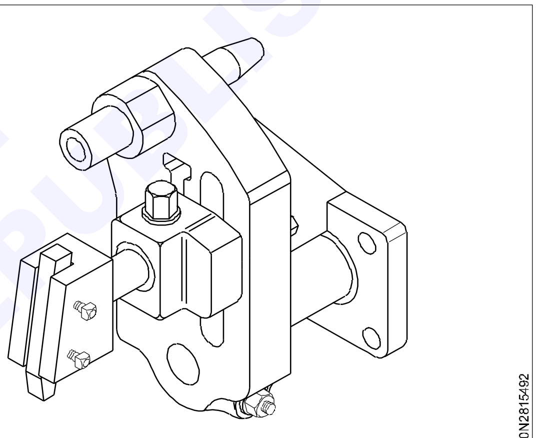


KNEE TOOL HOLDER

Work which is held in a chuck or in a turning fixture is fairly rigid, and needs no additional support while machining takes place. If the tool-holder mounted on the turret head is meant for combined operations, like turning and forming, the tool-holder must also be rigid and well supported. It is also essential that some arrangement should be provided to take up the heavy load which comes on the turret head. Such combination of operations is usually carried out with a knee tool-holder. The rigidity of this type of tool-holder can be increased by using the overhead support bar protruding from the headstock. This engages the guide bush on the tool-holder and maintains the alignment of the turret and the spindle axis, preventing deflection of the turret by the cutting force applied.

A boring bar can also be fitted so that a hole can be machined at the same time as the external surface is being machined. The chuck or fixture should be fitted with a guide bush to pilot the end of the boring bar, and again maintain the geometry of the tool by reducing tool deflections.

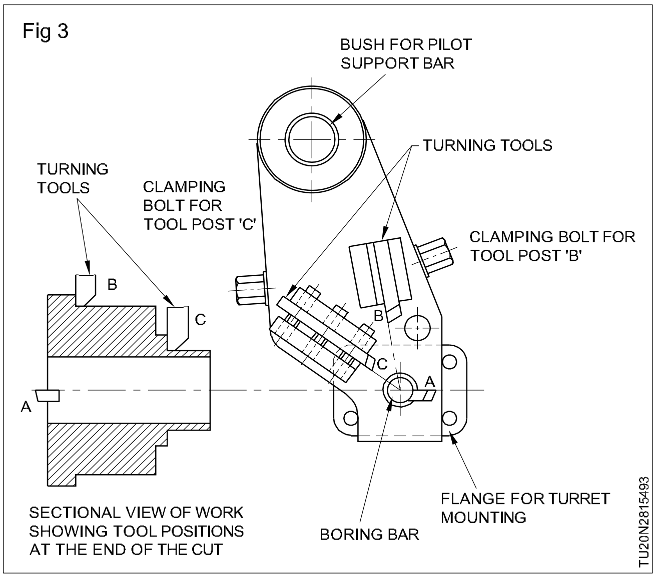
Fig 2



KNEE -TURNING TOOL HOLDER

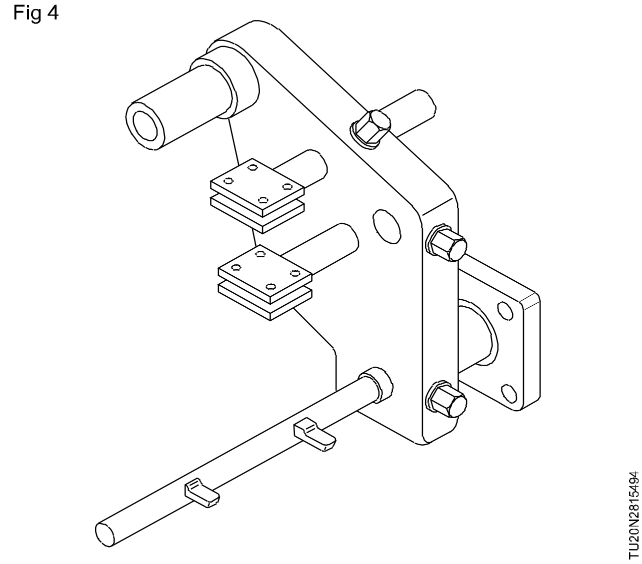
Combination tool-holder (Figs 3)

Fig 3



SECTIONAL VIEW OF WORK SHOWING TOOL POSITIONS AT THE END OF THE CUT

Fig 4

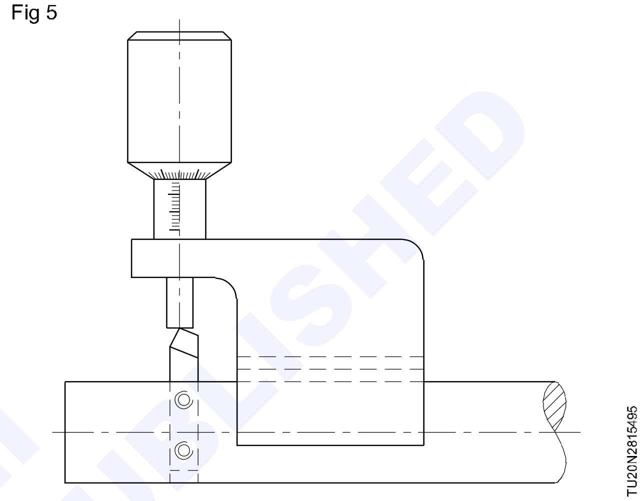


A further development of the knee tool-holder is the combination tool-holder which can be fitted with two turning tools and a boring bar. Thus simultaneous machining operations are possible. By increasing the number of tools in the boring bar, multi diameter bores can be machined at the same time.

Such set ups involve more complex and time consuming tool setting and are only economic where the time saved in machining outweigh the setting time. The machine down time for setting can be reduced by the use of pre-set tooling, ie. by two combination tool-holders, one in the machine and the other out for tool maintenance. When the tools have been reground they are preset in position and the complete tool-holder is changed. Presetting of boring tools is facilitated by the use of a boring bar cutter micrometer. (Fig 3)

The knee turning tool-holder has provision for only one turning tool, and it has a greater range of adjustment. But the adjustment for positioning the turning tool head in the combination type tool-holder is merely that of a tool overhang in the tool post.

Fig 5



Methods of producing external threads on capstan lathes

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

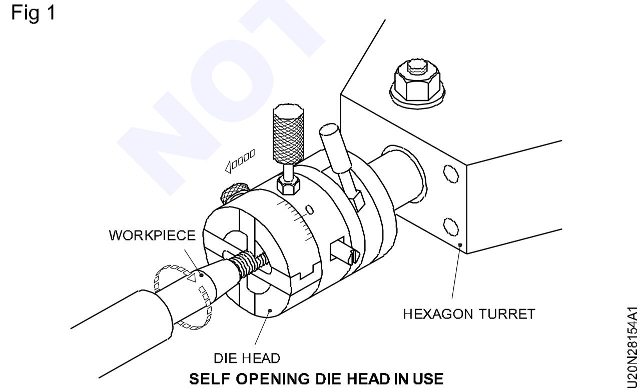
- **state various methods of producing external threads on capstan lathes**
- **state the construction and function of a self-opening die head**
- **state the types of self-opening die heads**
- **list and name the parts of a die insert.**

External threads are generally manufactured

- by using a single point tool with self opening die heads
- by thread rolling heads.

Self-opening die heads (Fig 1)

Fig 1



SELF OPENING DIE HEAD IN USE

Self-opening die heads are most commonly used in the thread manufacturing process. They are made in a wide range of sizes for producing threads up to about 100 mm diameter.

In addition to dies/chasers for standard threads, many dies/chasers for special threads are also available.

The die head is fed to the work by the operator who then allows it to feed itself along the work, and follows up with the turret. The turret stop is set slightly short to the thread length. When the die head movement is stopped by the turret stop, the front portion of the die head continues to feed forward under the self-feeding action until it is pulled clear of the detent pin.

Long accurate threads need a positive method of feeding the die head over the work. On capstan lathes this is achieved with a hexagon turret, leadon attachment. If the machine is equipped with a cross slide and the thread chasing attachment, the cros sslide is linked to the hexagon turret to provide positive lead.

Advantages when using a self-opening die head

The direction of workpiece rotation does not have to be reversed.

A roughing and finishing arrangement enables two cuts to be made.

An adjusting screw permits variations to be made from the standard size, or enables several cuts to be made on coarse threads.

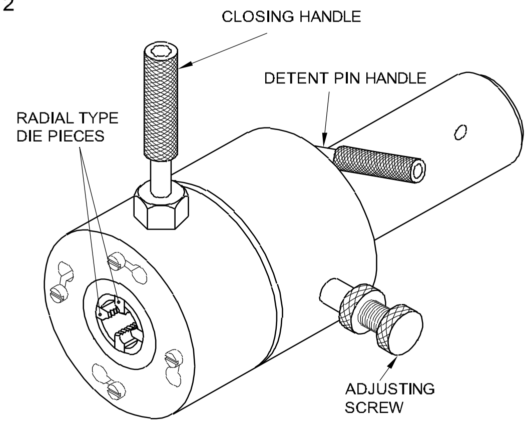
The cutting stresses maintain the die in the correct position, preventing errors caused by die tilt or movement.

The two types of die heads in use are the radial die heads and the tangential die heads.

Radial die head (Fig 2)

This die head uses radial cutting chasers. Adjustment can be made to the thread cutting size and also for roughing and finishing cuts to be made on the workpiece.

Fig 2

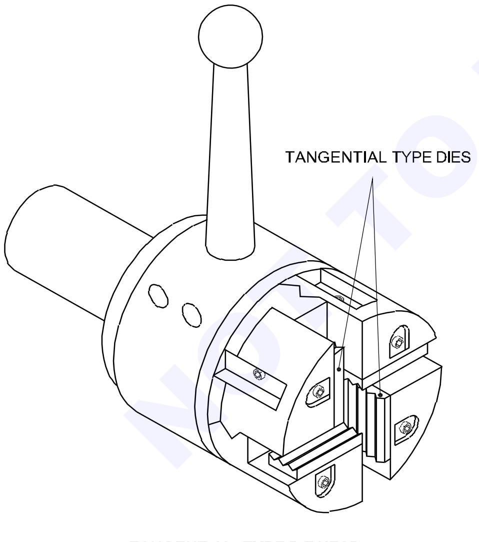


RADIAL DIE HEAD

Tangential die head (Fig 3)

These die heads use chasers of tangential type. These chasers can be sharpened quite often. This gives them a long life.

Fig 3

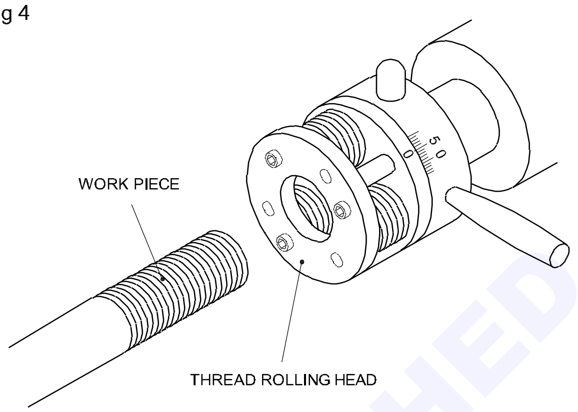


TANGENTIAL TYPE DIE HEAD

Thread rolling heads (Fig 4)

Thread rolling is a modern method of producing threads on turret lathes and automatic machines. They produce threads with excellent grain flow, and the pitch, the form and the accuracy of the threads are made to close tolerances.

Fig 4



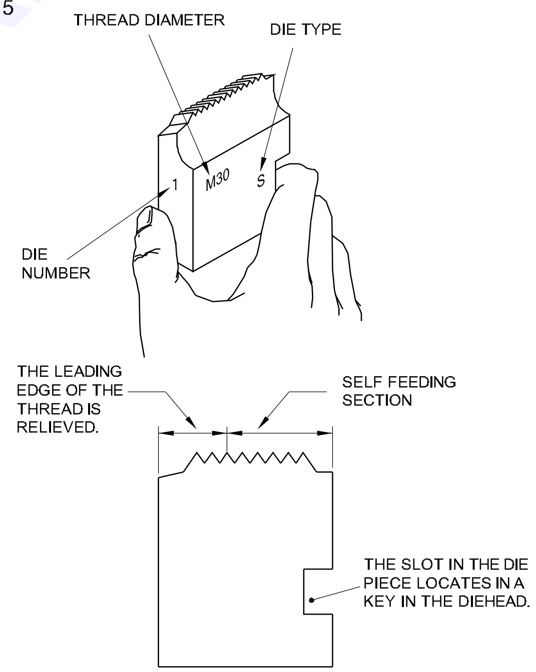
THREAD ROLLING

The dies (Fig 5)

The markings on the die piece indicate the

- die type
- thread diameter
- die number
- thread relieved on the leading side
- slots to locate the die in the die head.

Fig 5

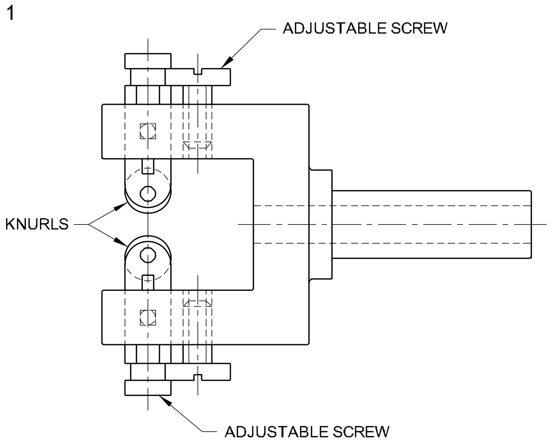


Accessories for capstan and turret lathe

Objectives: At the end of this lesson you shall be able to
- **state the use of the knurling tool-holder**
- **state the use of the multiple-cutter holder**
- **state the use of the drill-holder**
- **state the use of the boring bar-holder**
- **state the use of the reamer-holder.**

Knurling tool-holder (Fig 1)

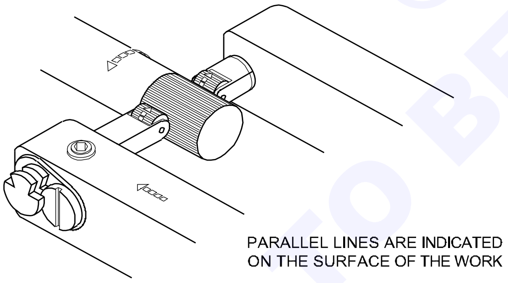
Fig 1



KNURLING TOOL HOLDER

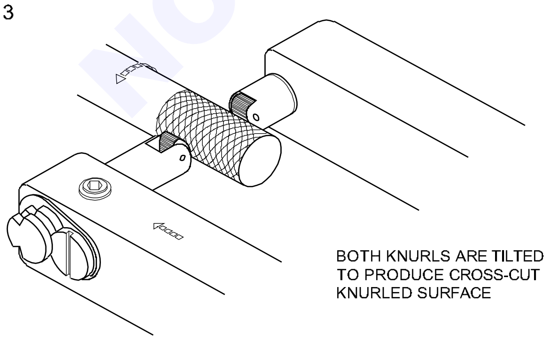
The knurling can also be performed by the tooling from the turret head. The knurling tool-holder shown in Fig 1 is mounted on the turret head. The position of the knurls can be adjusted with the screw to accommodate different diameters of the work. Different patterns of knurled surfaces are obtained by tilting the knurls.(Figs 2 and 3)

Fig 2



STRAIGHT KNURL

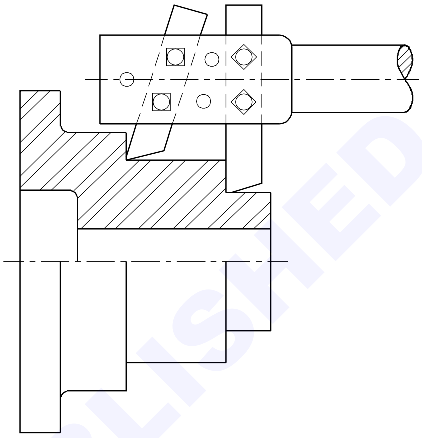
Fig 3



CROSS KNURL

Multiple-cutter holder (Fig 4)

Fig 4

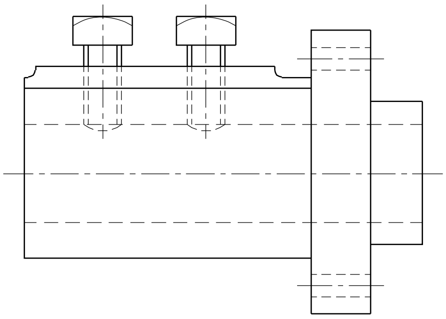


MULTIPLE-CUTTER HOLDER

The multiple-cutter holder accommodates more than one tool. The one shown in Fig 4 enables turning of two different diameters simultaneously. Turning and boring tools or turning and facing tools can also be set in the holder to perform multiple operations simultaneously.

Drill holder (Fig 5)

Fig 5



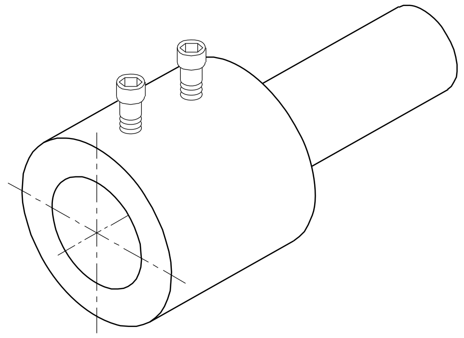
FLANGE TOOL HOLDER

The twist drills having morse taper shanks are usually held in a socket which is parallel outside and tapered inside. These sockets are inserted in the bracket of a flange tool-holder and clamped to it by set screws. Straight shank drills are mounted on Jacob's drill chuck, which in turn is held by the flanged tool-holder and socket.

Boring bar-holder (Fig 6)

This holder is also called an extension holder or a flanged tool-holder. These holders are intended for holding drills, reamers, boring bars, etc.

Fig 6



EXTENSION HOLDER

Reamer-holder

The standard practice of holding reamers in a capstan/turret lathe is in some form of floating holder which permits some amount of end movement of the reamer to align itself with the work.

Fault analysis production turning

Objectives: At the end of this lesson you shall be able to
- **list the common faults and state their causes and effects**
- **state the remedies for avoiding repetition of the faults.**

Common faults

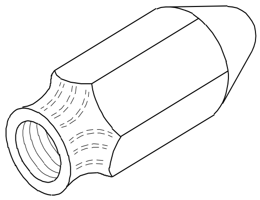
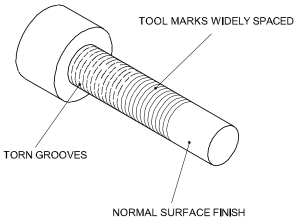
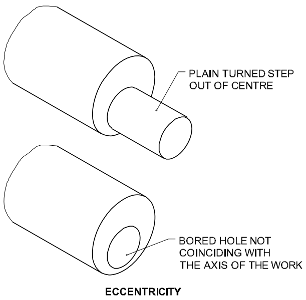
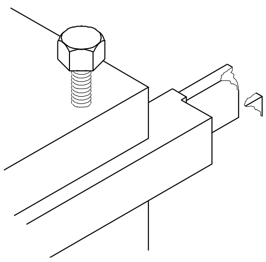
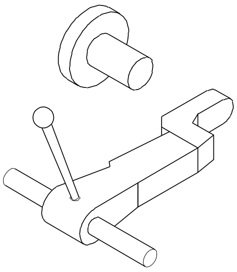
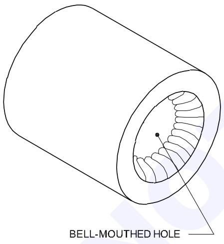
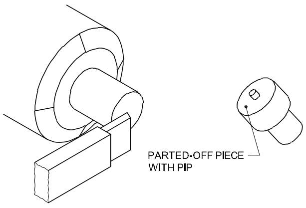
Figure	Cause	Remedy
Chatter (Fig 1) Lateral grooves and ridges formed on the workpiece giving a moulted appearance. 	Collet not gripping the bar	Ensure collet tension is correct
	The workpiece insecurely supported	Hold the workpiece rigidly
	Play in the spindle bearings or tool slideways	Inform the supervisor, maintenance section
	Spindle speed too high	Reduce the spindle speed
	Incorrect feed rate	Adjust the feed rate suitably
	Incorrect tool height	Check the centre height of the tool and set to correct height.
	Incorrect tool geometry or a worn out tool bit.	Regrind the tool to the specification
	Play in the tool slideways	Inform the supervisor
	Incorrect or insufficient flow of cutting oil	Change or adjust the flow of cutting oil
	Overhang of the tool in the tool-holder too great	Set the tool back in the tool-holder with a minimum overhang.
	Tool loose in the tool-holder	Clamp the tool rigidly
	Play in the spindle bearings	Inform the supervisor, maintenance section.
Poor surface texture (Fig.2) The cutting tool marks widely spaced and / or torn surface of the workpiece. 	Incorrect tool geometry or worn out tool bit	Regrind the tool bit to specification.
	Insufficient coolant.	Increase the coolant supply.
	Tool point too keen.	Hone small radius on to the tool point unless a sharp corner is specified.
	Incorrect spindle speed, feed rate, material or tool material.	Check each against the specification and adjust according to the requirement.

Figure	Cause	Remedy
Eccentricity (Fig.3) 	Dirty collect and collet sleeve Worn out collect.	Remove, clean and lubricate the collect sleeve. Replace the collect.
Breakage of tool (Fig 4) 	Tool overhang too great. Tool set below centre. The part-off too is too far from the collet. Incorrect spindle speed and feed rate.	Reset the tool with a minimum overhang. Reset the tool to the centre height. Reset the tool as near as possible to the collet. Set the speeds and feed to those recommended.
Bar feed defective (Fig 5) 	Loose or dirty collet. Insufficient feed cord weight. Jammed feed cord.	Reset the collet tension and clean, if necessary. Increase the feed cord weight. Examine the feed cord line and the pulleys.
Bell-mouthed hold (Fig.6) 	Chipped centering tool. The centering tool or the drill misaligned.	Regrind or replace the centering tool. Reset the centering tool or drill centrally.
Pip leftout after parting off (Fig 7) 	Worn out part-off tool, or incorrect tool geometry. Centre height of tool too high or too low.	Regrind the part-off tool to the specification. Reset the centre height of the tool.

Methods of producing internal threads on capstan / turret lathes

Objectives: At the end of this lesson you shall be able to
- **state various methods of producing internal threads**
- **state construction details of a collapsible tap**
- **state the functional features of a collapsible tap.**

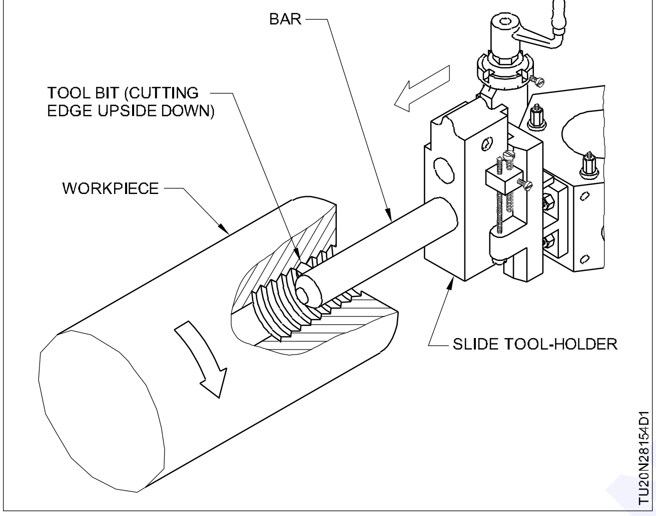
Internal threads are cut using 2 types of tools

– Single point thread cutting tool

– Solid tap

Internal threading by using a single point thread cutting tool (Fig 1)

Fig 1



Threading with a single point tool is usually carried out on large workpieces or when special threads are required.

The tool may be mounted either on the hexagon turret or on the square turret fitted to the cross-slide. A threading drive accessory is fitted to the lathe, which enables the tool to be fed along the work at the appropriate rate for the desired pitch.Several cuts are normally made, each slightly deeper than the previous cut, until the thread depth appropriate to the selected pitch is obtained. The threading tool is normally held in a bar mounted in a slide tool-holder.

Solid taps

Solid taps are used for small diameter threads. They are usually spiral fluted.

The tap is fitted to the hexagon turret in a special tap-holder. The holder is designed to release the tap automatically at the end of the cut, permitting the tap to rotate along with the workpiece. (Fig 2)

The procedure for cutting the thread is as follows

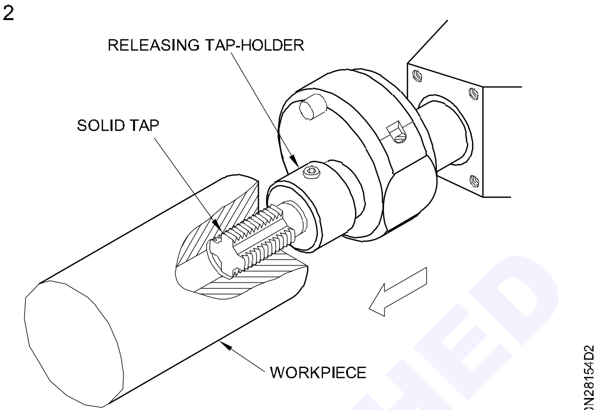
Move the turret to the workpiece.

Start the tap in the hole by exerting pressure on the turret drive hand wheel.

Keep slight forward pressure on the turret drive to prevent the tap from pulling the turret along as the thread cutting operation progresses.

This precaution will prevent distortion of the thread.

Fig 2



THREAD CUTTING WITH A SOLID TAP

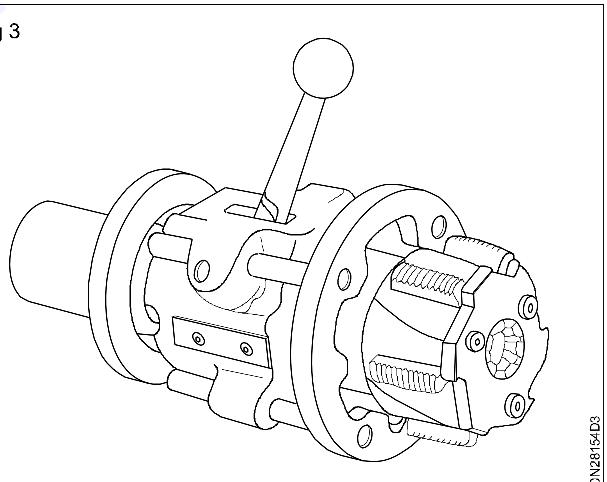
Excessive forward drive pressure will also distort the thread.

Forward motion of the tap should be stopped first before it reaches the end of the hole being threaded. When using the hexagon turret this distance is set with the turret stop. When the stop is reached by the advancing turret, the automatic release operates, releasing the tap and allowing it to revolve with the workpiece.

Reverse the headstock spindle rotation to drive the tap back out of the threaded hole.

Collapsible tap (Fig 3)

Fig 3



COLLAPSIBLE TAP

The collapsible tap is a mechanism in which thread cutting chaser inserts, normally six in number, are positioned in the slots provided in the head. These inserts are brought to the cutting position by pulling the lever provided. At the end of the threading operation, the tap collapses. The tap may then be drawn directly out of the work without reversing the rotation of the headstock spindle. This reduces the CG & M time. This facility is applicable only for cutting threads in holes of larger diameter due to the presence of collapsible mechanism. For cutting threads in successive pieces, the lever is again brought to the engaging position so that the thread cutting inserts are brought back from the collapsed or released state to the cutting position.

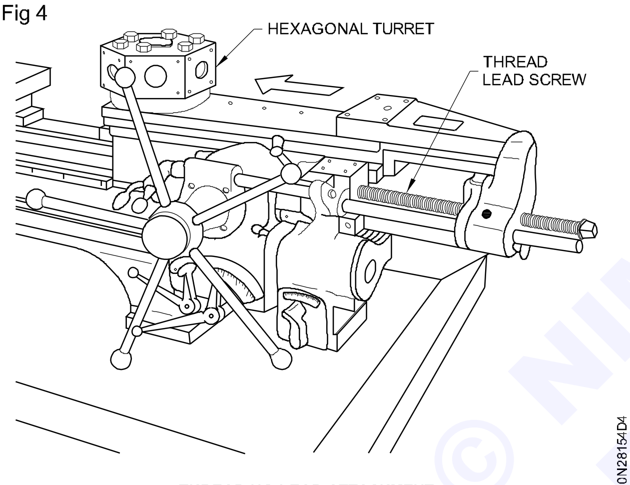
Threading attachment

To produce accurately pitched threads, when using a single point thread cutting tool, the tool must be advanced into the work at a rate determined by the thread pitch. The technique for producing correct rate differs in the following two set ups anyone of which may be provided in the capstan or turret lathe.

A thread lead attachment is used most commonly to drive the turret of the capstan lathe and a thread chasing attachment is used to drive the saddle of the turret lathe.

Threading lead attachment (Fig 4)

Fig 4



THREADING LEAD ATTACHMENT

This attachment is used on capstan lathes.

A thread lead (lead screw) of the required pitch is fitted to the turret drive. The turret is moved by the action of the half nut engaging the lead screw. (Fig 4)

The thread lead attachment includes a release mechanism which is operated by the turret stops. The length of the thread cut may be set automatically by adjusting the position of the turret stops.

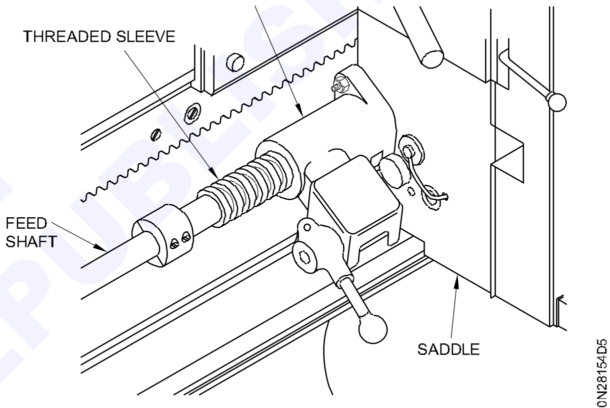
Thread chasing attachment (Fig 5)

This attachment is used on the turret lathes to drive the saddle at the required rate for the threading operation.

A threaded sleeve is fitted on the feed shaft and a meshing threaded collar is fixed to the saddle. The collar draws the saddle along the threaded sleeve.

The pitch of the thread on the sleeve determines the pitch of the thread cut on the workpiece.

Fig 5



THREAD CHASING ATTACHMENT FITTED TO SADDLE